MIA Welcome Note

IWA Modelling and Integrated Assessment Specialist Group

Dr. Elena Torfs (Chair of MIA SG)

Dr. Saba Daneshgar (Vice-chair of MIA SG)

inspiring change

UNIVERSITÉ LAVAL

GHENT UNIVERSITY

MIA SPECIALIST GROUP

“This group targets people from research, consulting companies, institutions and operators to think along ***the use of models and computing tools to support the understanding, management and optimization of water systems****.”*

PRIORITIES

- Stimulate interaction and knowledge transfer in the domain of modelling and integrated assessment
- Interact with other IWA SGs and other academic/professional organisations

How to find us

Website: http://iwa-mia.org/

https://iwaconnectplus.org

MIA SG: ACTIVITIES

Task Groups (TGs)

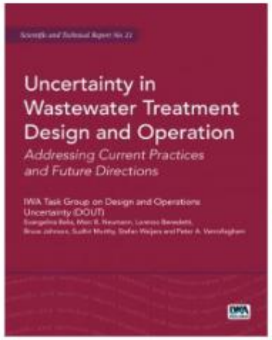
- Benchmarking of Control Strategies for WWTPs (BSM) AND Good Modelling Practice (GMP) AND Design and Operations Uncertainty (DOUT) AND Generalised Physicochemical Modelling (PCM) AND Use of Modelling for Minimizing GHG Emissions from Wastewater Systems (GHG) (All finished)
- Membrane Bioreactor Modelling and Control (MBR)
- Good Modelling Practice in Water Resource Recovery Systems (GMP2)

Working Groups (WGs)

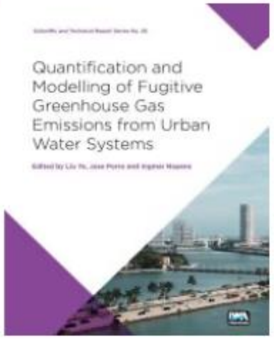
- Integrated Urban Water Systems (IUWS)
- Computational Fluid Dynamics (CFD)
- Good Modelling Practice (GMP)
- Hybrid Modelling (HM) (New)

Conferences / Events

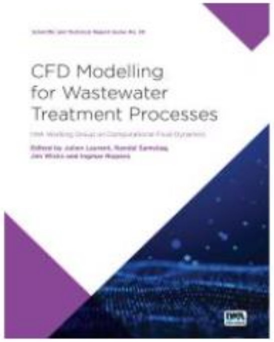
- WRRmod
- Watermatex



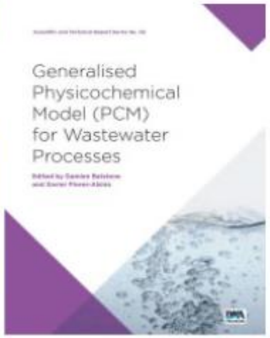
STR
(2021, OA)



STR
(2022, OA)



STR
(2022)



STR
(2022, OA)

MIA SG: RECENT & UPCOMING CONFERENCES

11th Symposium on Modelling and Integrated Assessment (Watermatex2023)

- Location: Québec City, Canada, 23-27 Sept. 2023
- Chair/vice-chair: Prof. Peter Vanrolleghem (Univ. Laval)/Dr. Elena Torfs (Univ. Ghent)

9th Water Resource Recovery Modelling seminar (WRRmod2024)

- Location: Notre Dame, Indiana, USA, 6-10 April 2024
- Organisers: Adrienne Menniti, Leon Downing, Tom Johnson, Rob Nerenberg

- **WRRmod2026** (To be announced soon)

FIND MIA SG ON SOCIAL MEDIA

Follow the Modelling and Integrated Assessment Specialist Group on:

– https://iwaconnectplus.org

– https://www.linkedin.com/company/iwa-mia-specialist-group-on-modelling-and-integrated-assessment

– https://twitter.com/iwa_mia_sg

– **MIA SG open web site** http://iwa-mia.org

to get informed about our latest events, publications and news!

AGENDA AND HOUSEKEEPING

Presentation 1
Ed Wicklein (Carollo Engineers)

Presentation 2
María Elena Valle Medina (ENGEES/UNISTRA), Julien Laurent (ENGEES/UNISTRA), Alonso Griborio (Hazen)

Presentation 3
Christopher DeGroot (Western University, Canada)

Presentation 4
Giacomo Bellandi (AM-Team)

Presentation 5
Stephen Saunders (Ibis Group), and Giacomo Bellandi (AM-Team)

Q&A Session Moderator: *María Elena Valle de Medina (ENGEES/UNISTRA)*

- This session is being recorded;
- Keep microphones and cameras turned off during the presentations.
- The normal chat function is disabled;
- Please put any questions and comments you may have in the Q&A and we will do our best to answer them during the session (in writing or orally).
- During Q&A session you can activate the camera and microphone if you pose a question.

AGENDA AND HOUSEKEEPING

Presentation 1
Ed Wicklein (Carollo Engineers)

Presentation 2
María Elena Valle Medina (ENGEES/UNISTRA), Julien Laurent (ENGEES/UNISTRA), Alonso Griborio (Hazen)

Presentation 3
Christopher DeGroot (Western University, Canada)

Presentation 4
Giacomo Bellandi (AM-Team)

Presentation 5
Stephen Saunders (Ibis Group), and Giacomo Bellandi (AM-Team)

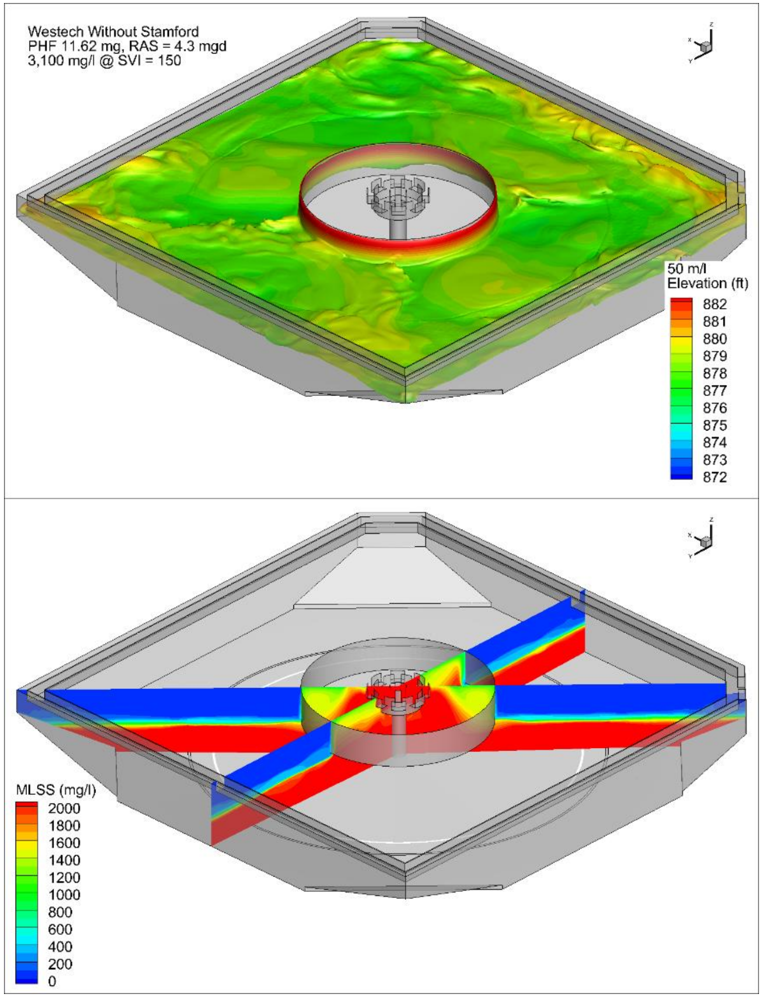
Q&A Session Moderator: *María Elena Valle Medina (ENGEES/UNISTRA)*

CFD MODELING OVERVIEW: BACKGROUND TO CURRENT PRACTICE ENHANCING WATER AND WASTEWATER TREATMENT

Ed Wicklein
Carollo Engineers

CFD MODELING BACKGROUND AND CURRENT MODELING PRACTICE

- What is CFD?
- Modeling tools
- CFD modeling workflow
- Building quality models



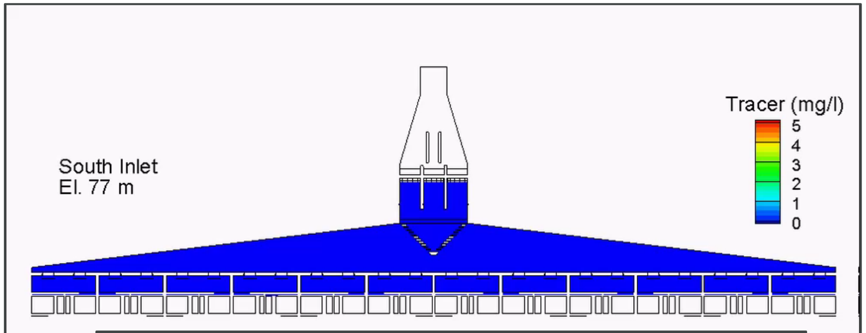
WHAT IS CFD (COMPUTATIONAL FLUID DYNAMICS)?

“ . . . flows and related phenomena can be described by partial differential equations, which cannot be solved analytically except in special cases. To obtain an approximate solution numerically, we have to use a discretization method which approximates the differential equations by a system of algebraic equations, which can then be solved on the computer.” - Ferziger and Peric (2002) *Computational Techniques for Fluid Dynamics*, 3rd Edition, Springer.

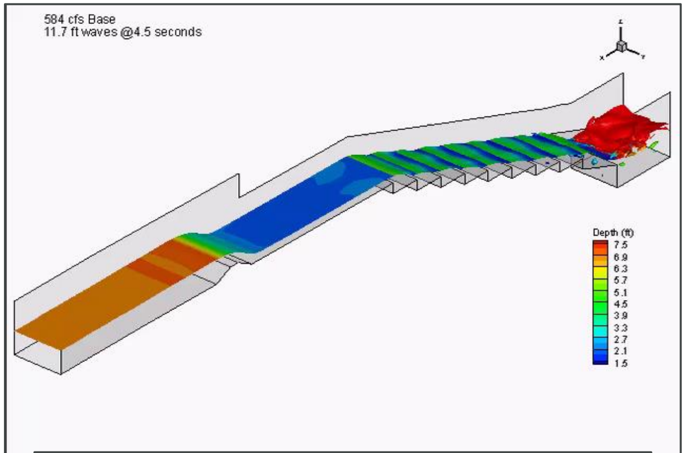
“Computational fluid dynamics, then, is a separate discipline, distinct from and supplementing both experimental and theoretical fluid dynamics, with its own techniques, its own difficulties, and its own realm of utility, offering new perspectives in the study of physical processes.” - Roach, P.J. (1982) *Computational Fluid Dynamics*, Hermosa Publishers.

COMPUTATIONAL FLUID DYNAMIC MODELING (CFD) IS AN ENGINEERING TOOL FOR EVALUATING FLUID FLOWS

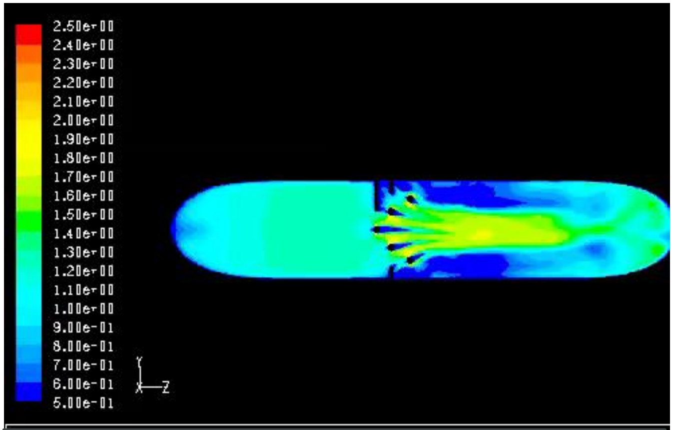
- Captures:
  - Three dimensions in space
  - Turbulent fluid motion
  - Multiphase physics
  - Energy
  - Reactions
  - Time
- Solution visualization with high resolution graphics
- 30+ year history in our field



Chemical Mixing



Transient Free Surface Conditions



Velocity Fluctuation and Eddy Shedding

CFD SOLVES THE NAVIER-STOKES EQUATIONS BY NUMERICAL SCHEMES

$$\frac{\delta\rho}{\delta t}+\frac{\rho\delta U_i}{\delta X_i}=0$$

Continuity

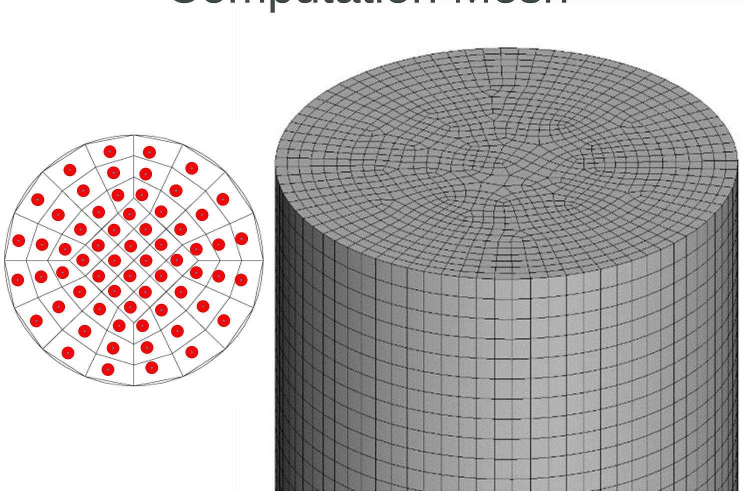
$$\frac{\delta U_i}{\delta t}+U_j\frac{\delta U_i}{\delta X_j}=-\frac{1}{\rho}\frac{\delta P}{\delta X_i}+\nu_t\frac{\delta^2 U_i}{\delta X_j\delta X_j}+F_i$$

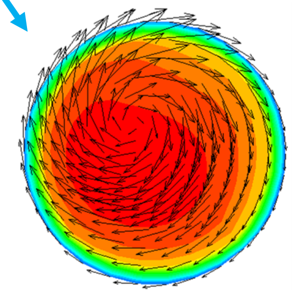
Momentum

$$\nu_t=C_\mu\frac{k^2}{\varepsilon}$$

Turbulence

Computation Mesh





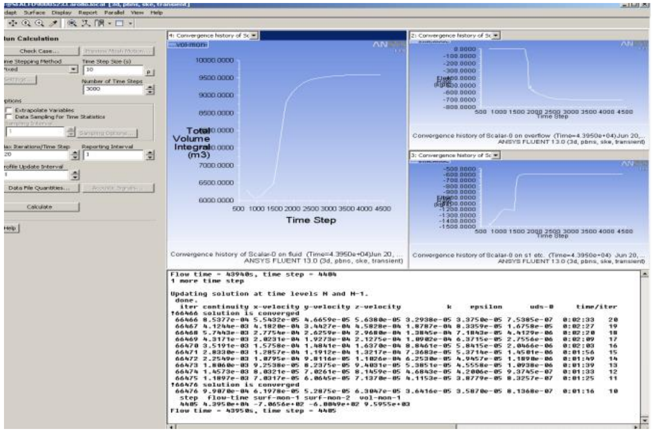
Flow Solution

SOFTWARE TOOLS USED CAN BE INTEGRATED OR DISCRETE

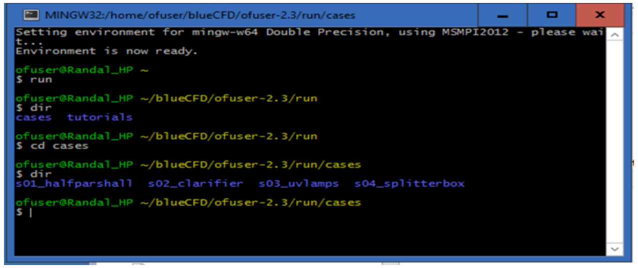
- Meshing – many choices
- Solver must be a well validated package
  - Fluent[1]
  - CFX[1]
  - CCM+[1]
  - Flow-3D[1]
  - OpenFOAM[2]
- Visualization Software – many choices

1) Commercial examples commonly reported
2) Open Source

Commercial software has well developed interface

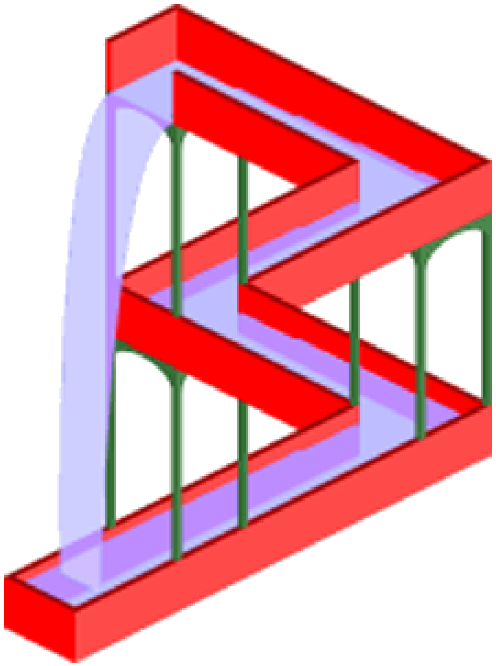


Open source software more likely run from command line



CHALLENGES IN CFD MODELING:
HOW DO WE ENSURE THE SOLUTION IS RIGHT?

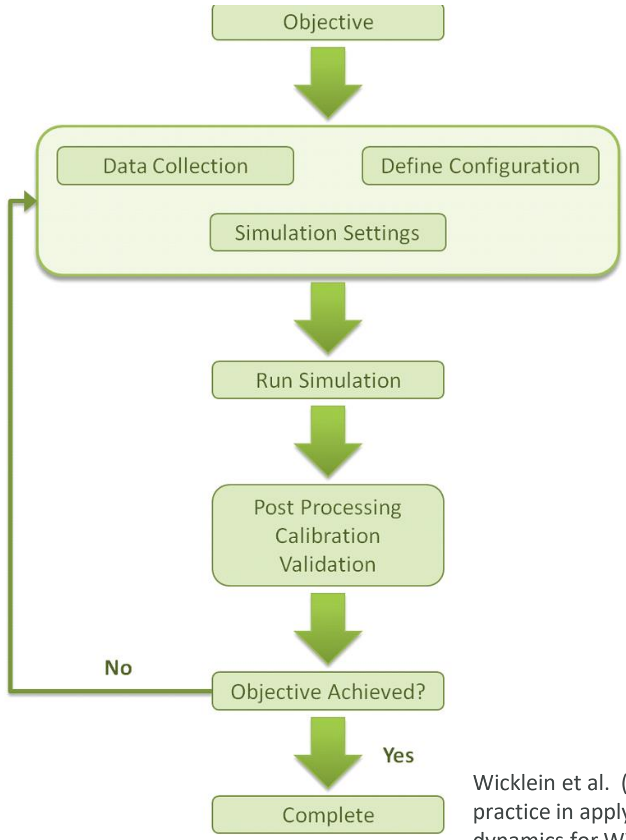
- Sufficient domain
- Quality mesh
- Appropriate boundary conditions
- Correct numerical methods
- Verified and validated code
- Adequate Convergence



http://www.illusionspoint.com/illusions/visual-optical-illusions/page/10/

BASIC CFD WORKFLOW FOR CFD MODELING

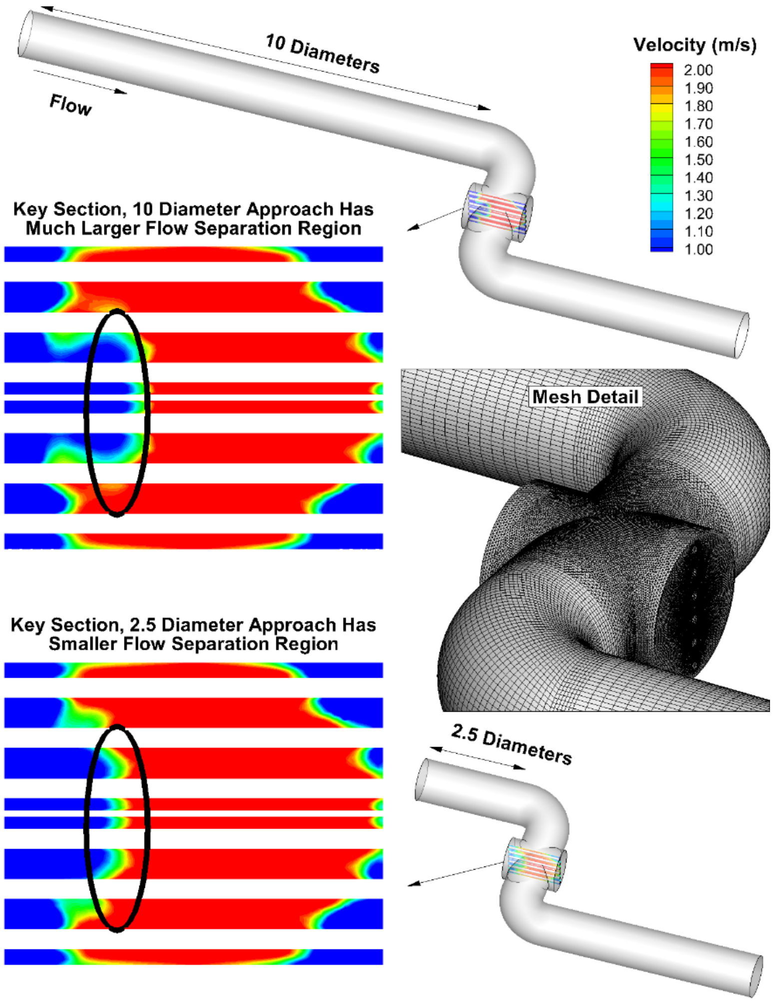
- Define Objective
- Gather key information
  - Geometry
  - Flow properties
  - Numerical modes required
  - Modeling scenarios
- Create model
  - Develop computational mesh
  - Setup simulation
- Solve
- Verification loops required for accurate solution
  - Mesh quality checks:
    - are cell shapes and relationship sufficient to calculate good solution?
  - Mesh independence check:
    - is the computational cell size sufficiently refined to capture problem?



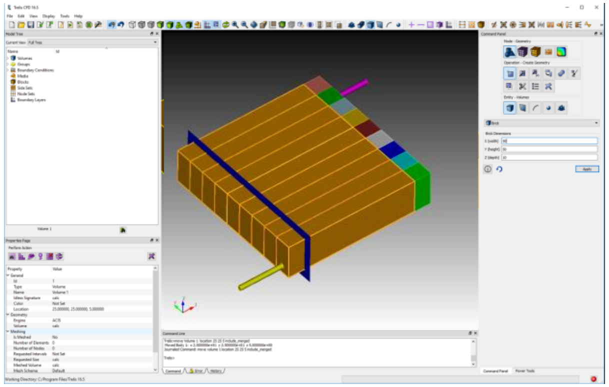
Wicklein et al. (2016) Good modelling practice in applying computational fluid dynamics for WWTP modelling, *WST*, 73 (5) 969-982.

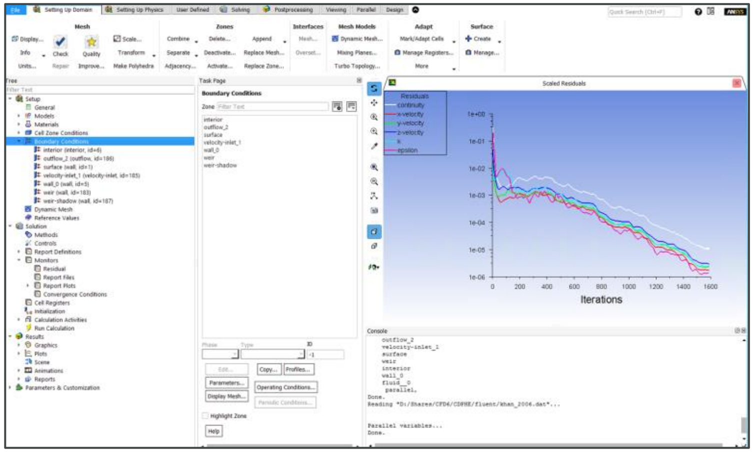
GEOMETRY AND MODEL BOUNDARY DRIVE PROBLEM SOLUTION

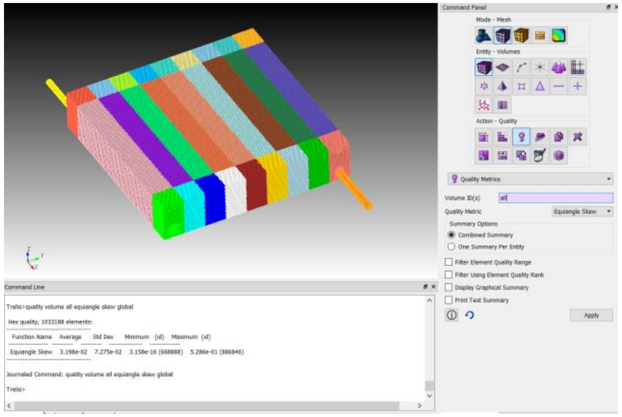
- Accurate geometry is required, new design or built facility?
  - 2d drawings
  - 3d CAD
  - Surface mapping
- Determine water level approach
- Boundary's need to be sufficient distance from area of interest
- Appropriate Type
  - Inflow
  - Outflow
  - Pressure
  - Porous
  - Walls

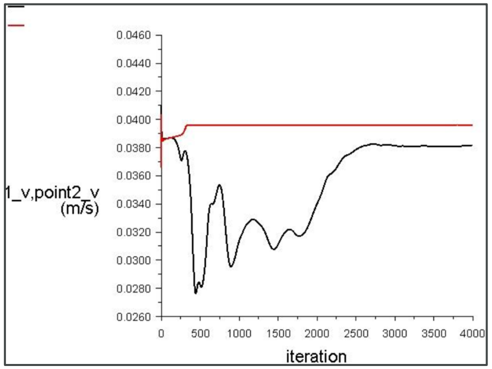


BUILD OR CLEAN GEOMETRY, DEVELOP MESH, SETUP AND SIMULATE FLUIDS AND PHYSICS

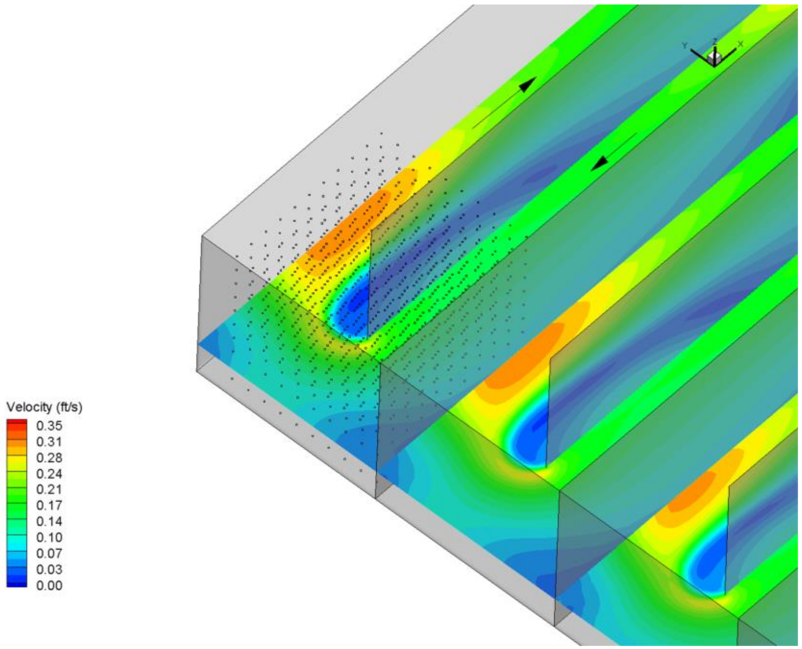


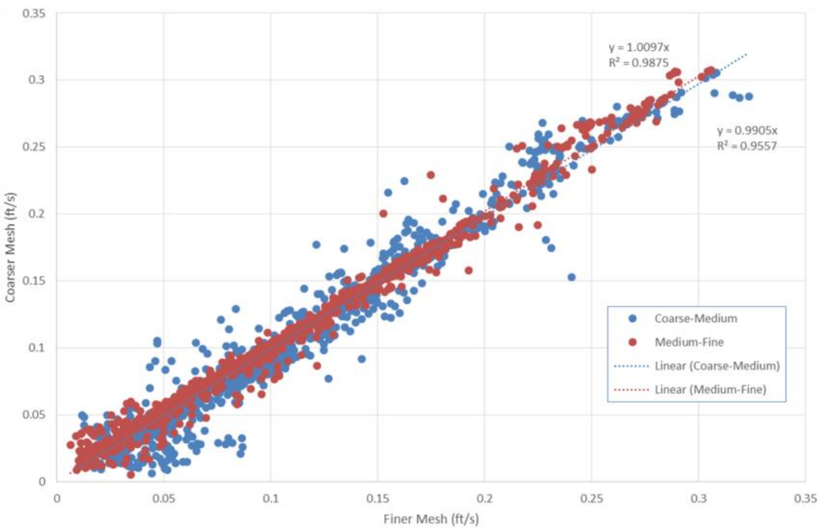






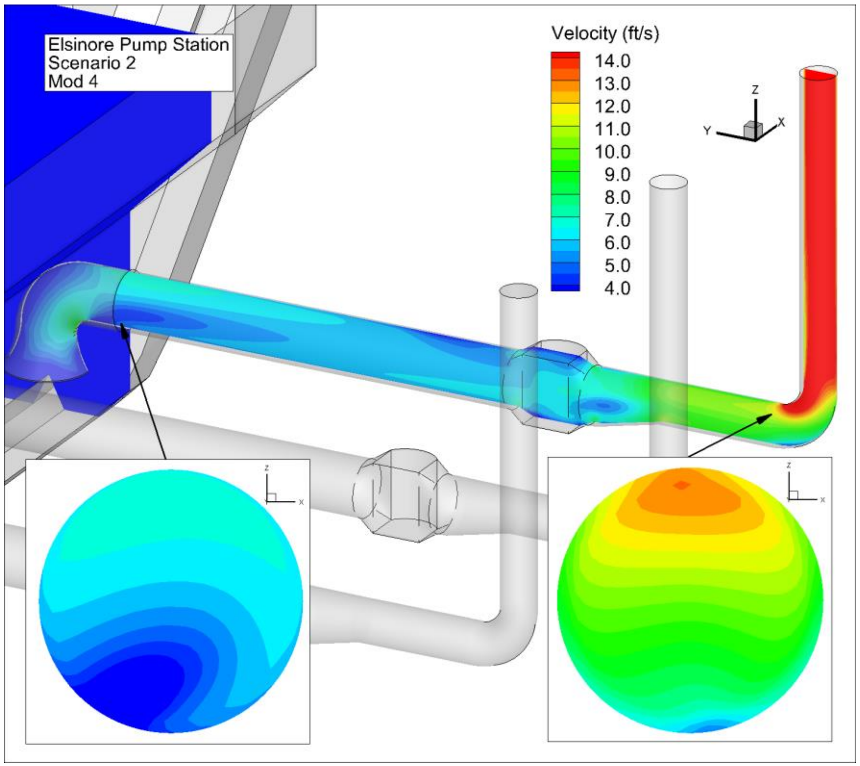
MESH RESOLUTION TESTING MAY BE REQUIRED TO VERIFY GRID INDEPENDENCE



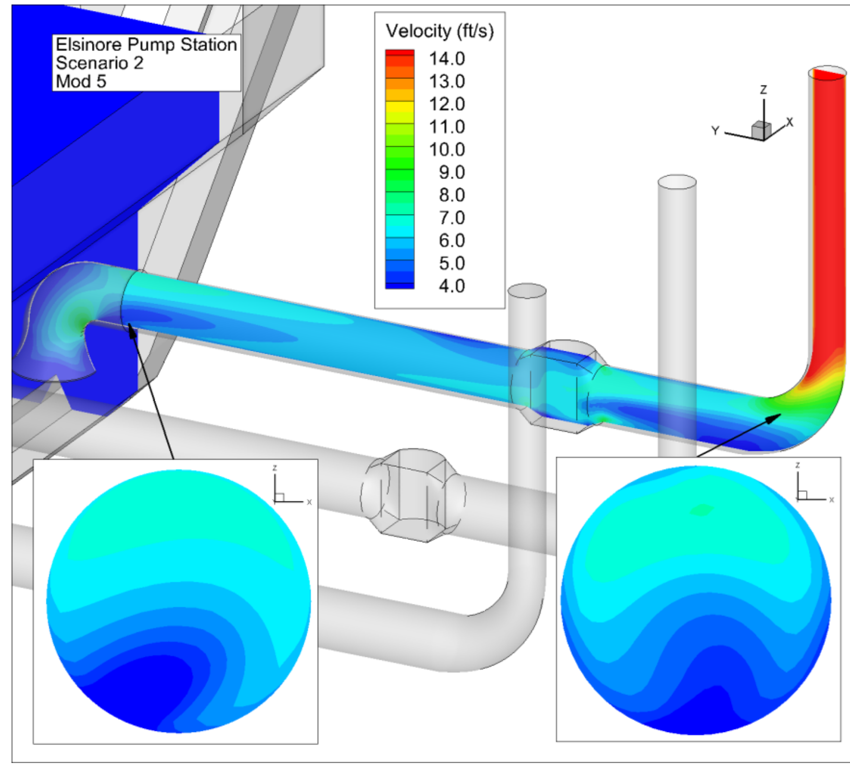


Mesh	Cell Size Range (in)	Number of Cells	Percent Refinement
Coarse	1.6 to 7	648,830	
Medium	1.6 to 4.8	1,320,723	103
Fine	1.2 to 3.6	3,289,125	149

AFTER MESH INDEPENDENCE VERIFICATION AND OTHER CHECKS: SOLVE THE PROBLEM AND DELIVER RESULTS



Before: Eccentric reducer and short radius elbow led to high flow separation into pump suction, leading to potential performance problems.



After: Elimination of eccentric reducer and use long radius reducing elbow significantly improved hydraulics at pump suction.

AGENDA AND HOUSEKEEPING

Presentation 1
Ed Wicklein (Carollo Engineers)

Presentation 2
María Elena Valle de Medina (ENGEES/UNISTRA), Julien Laurent (ENGEES/UNISTRA), Alonso Griborio (Hazen)

Presentation 3
Christopher DeGroot (Western University, Canada)

Presentation 4
Giacomo Bellandi (AM-Team)

Presentation 5
Stephen Saunders (Ibis Group), and Giacomo Bellandi (AM-Team)

Q&A Session Moderator: *María Elena Valle de Medina (ENGEES/UNISTRA)*

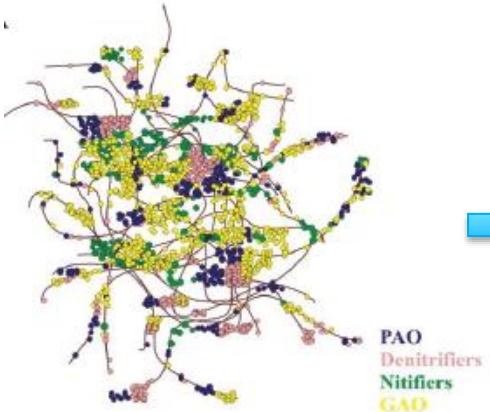
SST PERFORMANCE ASSESSMENT USING CFD : IMPACT OF DENSIFIED SLUDGE CHARACTERISTICS

Alonso Griborio
Hazen and Sawyer

M. Elena Valle
Unistra / ENGEES

Julien Laurent
Unistra / ENGEES

DENSIFIED ACTIVATED SLUDGE (DAS) IS AN ENHANCEMENT TO CONVENTIONAL BNR THAT FACILITATES RAPID SETTLING



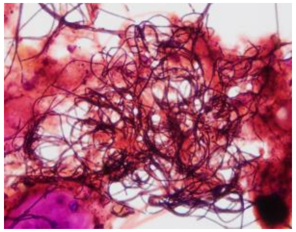
Underlying benefit is due to <u>improved settleability</u>

WHAT IS DENSIFIED ACTIVATED SLUDGE (DAS)?

Densified AS

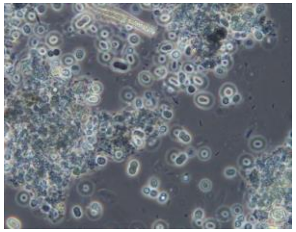
Flocculant AS

bulking



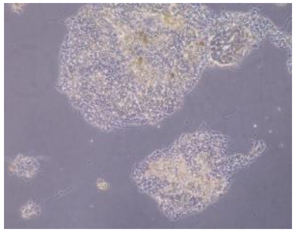
SVI$_{30}$ > 150 mL/g

dispersed/non-bulking



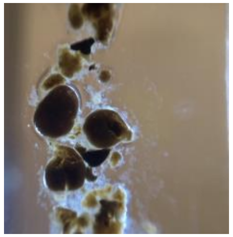
SVI$_{30}$ ~ 120 to 150 mL/g

compact



SVI$_{30}$ 80 to 120 mL/g

granular

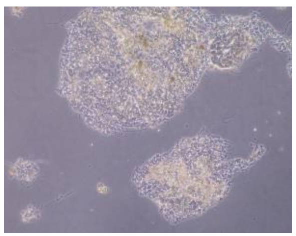


SVI$_{30}$ < 50 mL/g

Continuum of Densification

HOW DOES DAS COMPARE WITH CONVENTIONAL AND GRANULAR SLUDGE?

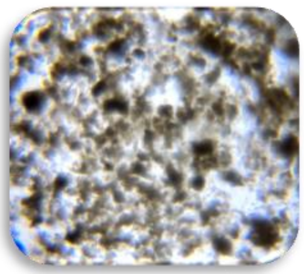
Compact AS (non-granular)



SVI_{30} 80 to 120 mL/g

non-patented AS
Conventional
Selector Design

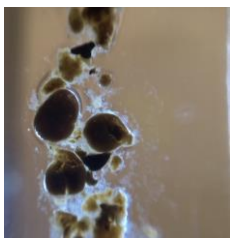
Densified AS (non-granular)



SVI_{30} < 80 mL/g

non-patented AS*

Densified AS (granular)



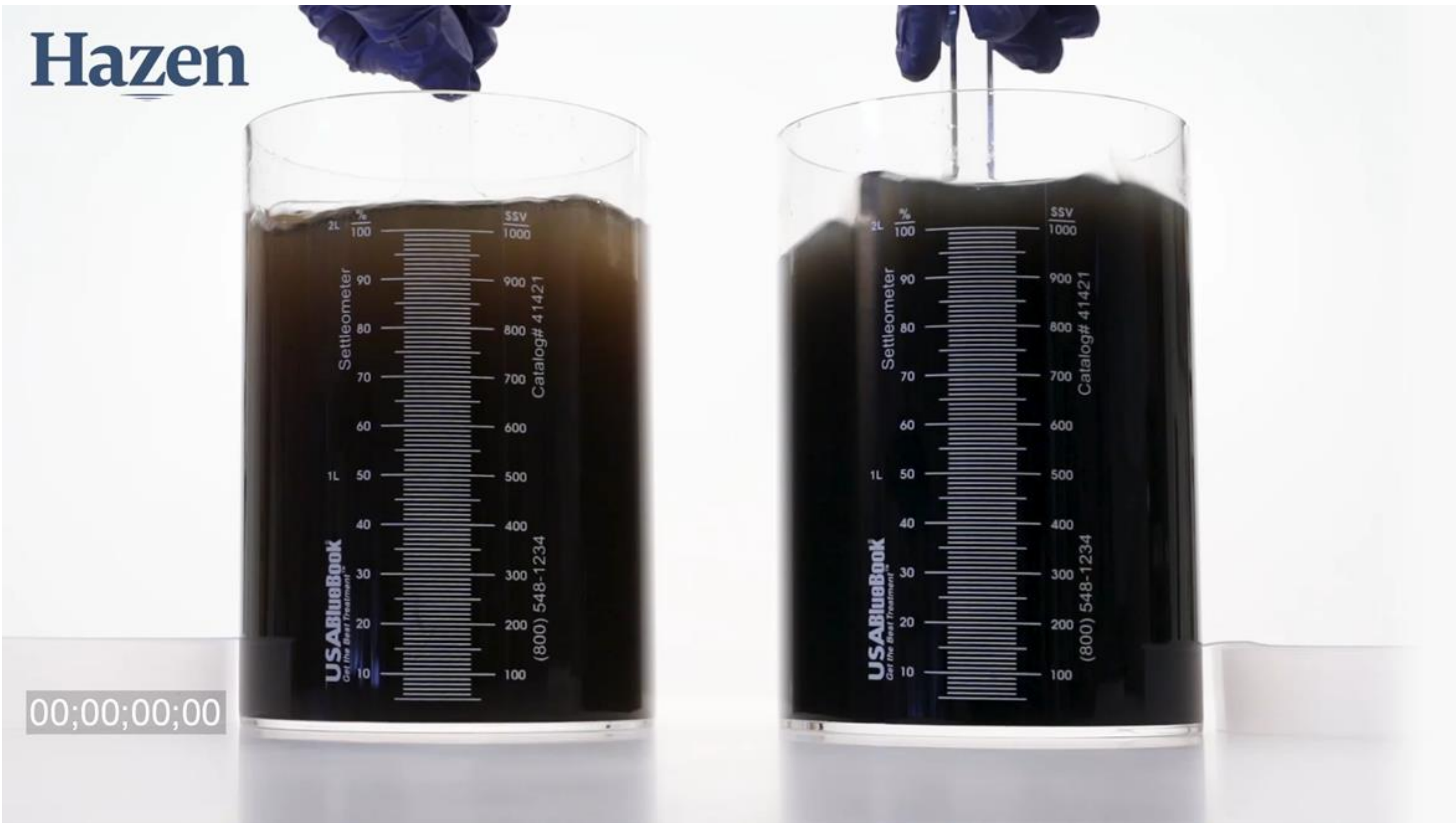
SVI_{30} < 50 mL/g

AquaNEREDA™
inDENSE™

How do we Bridge the Gap?

ENHANCED SETTLING CAN UNLOCK CAPACITY AND REDUCE INFRASTRUCTURE

Densified Activated Sludge



ENHANCED SETTLING CAN UNLOCK CAPACITY AND REDUCE INFRASTRUCTURE

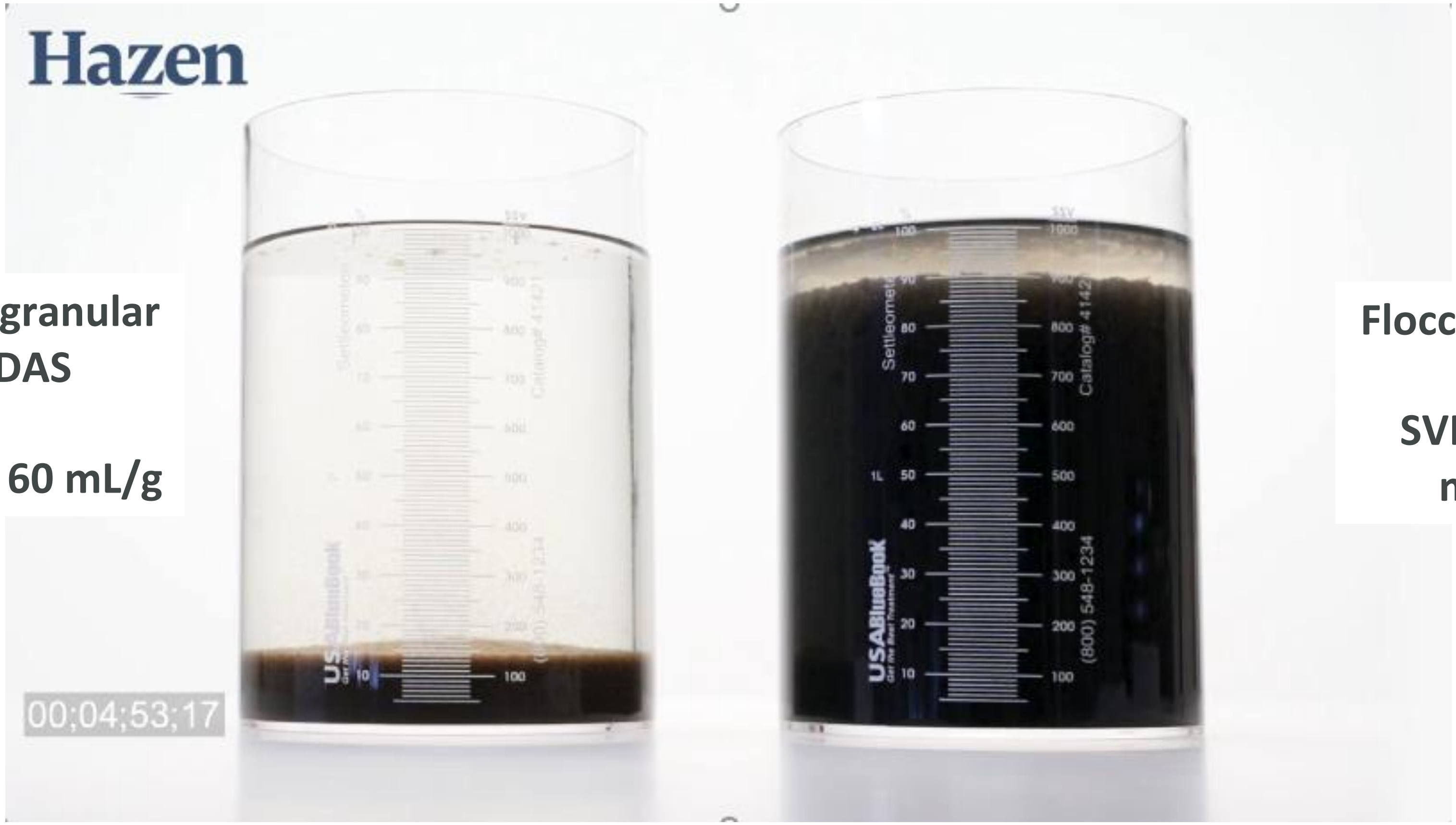
Densified Activated Sludge

ENHANCED SETTLING CAN UNLOCK CAPACITY AND REDUCE INFRASTRUCTURE

Densified Activated Sludge

Non-granular DAS

SVI ~ 60 mL/g



Flocculant AS

SVI ~ 150 mL/g

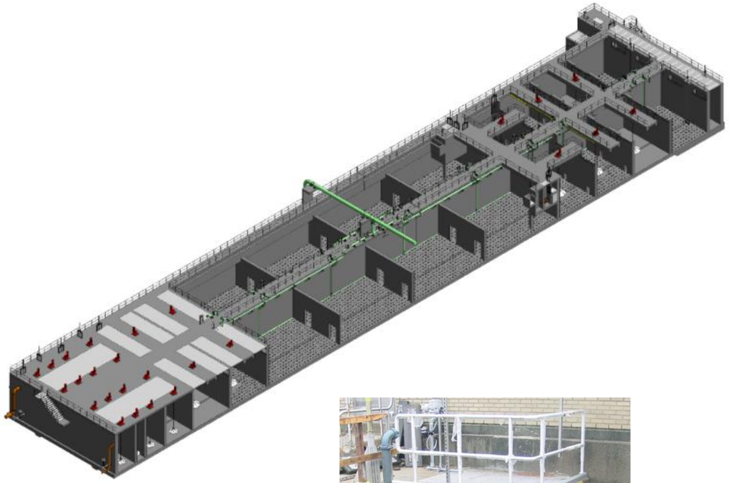
WHAT ARE THE INGREDIENTS TO ACHIEVE DENSIFIED ACTIVATED SLUDGE (DAS)?

Promote Strong Feast Famine

Robust DO

Kinetic and Metabolic Selection

e.g., Staged Anaerobic selector,
Plug Flow Aerobic



Washout of Slow Settling Biomass

External (Physical) Selection

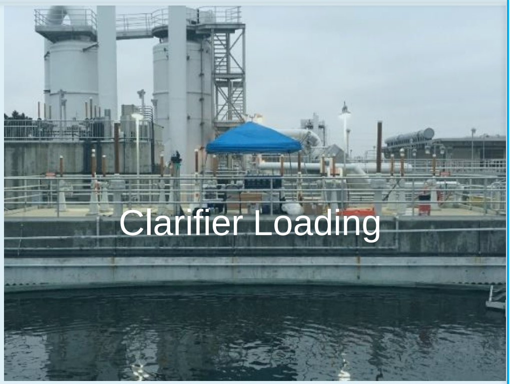
e.g., Surface wasting, hydrocyclones, screens

CASE STUDY 1:
METRO WATER RECOVERY (DENVER, CO)

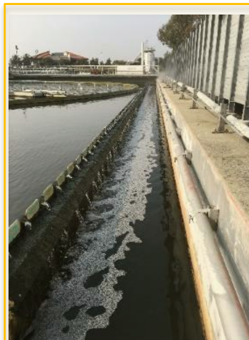
- Robert W. Hite Treatment Facility (220 mgd / 833000 m^3/d)
- Very stringent future limits
  - TN < 2 mg/L
  - TP < 0.1 mg/L
- Denver Metro conducted a long term-large scale pilot (3 – 8 MGD / 11360 – 30280 m^3/d)
- Clarifier CFD models developed for DAS and conventional clarifiers

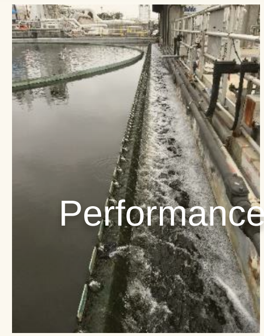
CFD Modeling Demonstrated Significant Saving Opportunity

EXTENSIVE CLARIFIER TESTING CONDUCTED FOR CFD MODEL DEVELOPMENT AND CALIBRATION

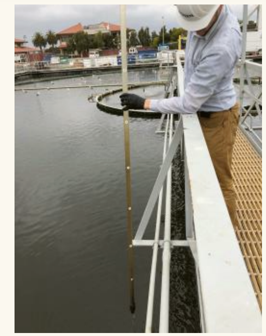


Influent and effluent flows and MLSS





Clarifier effluent TSS / Turbidity



Sludge blanket depth
and profiles

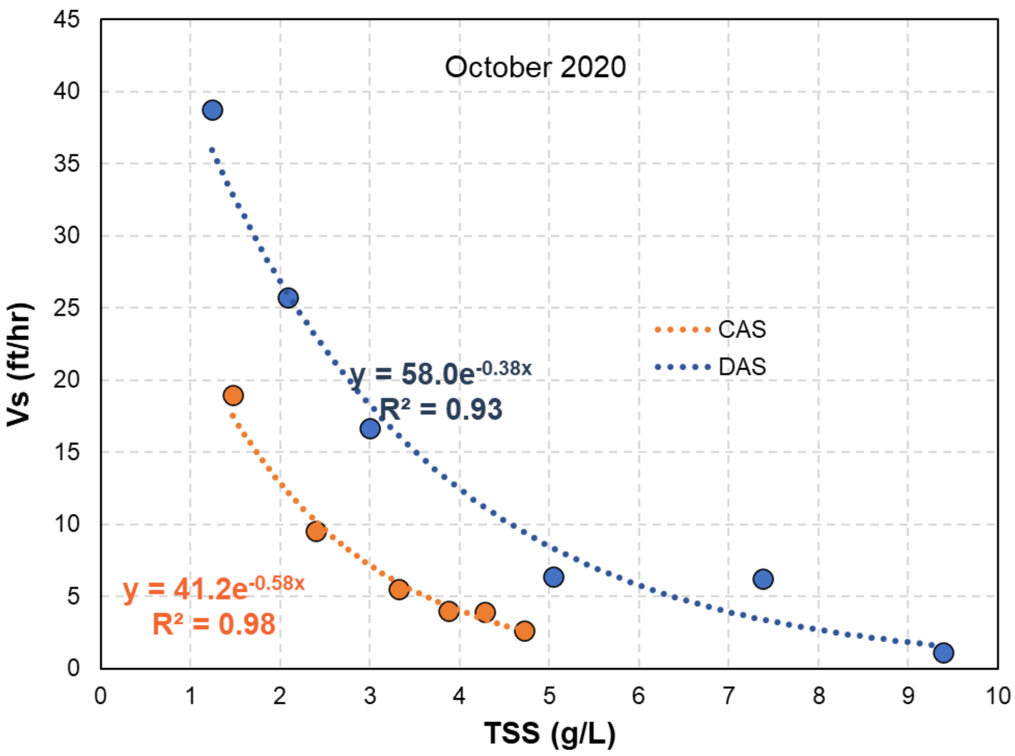
RAS flow and RAS TSS

Settling properties
Settling column/SVIs

Discrete settling

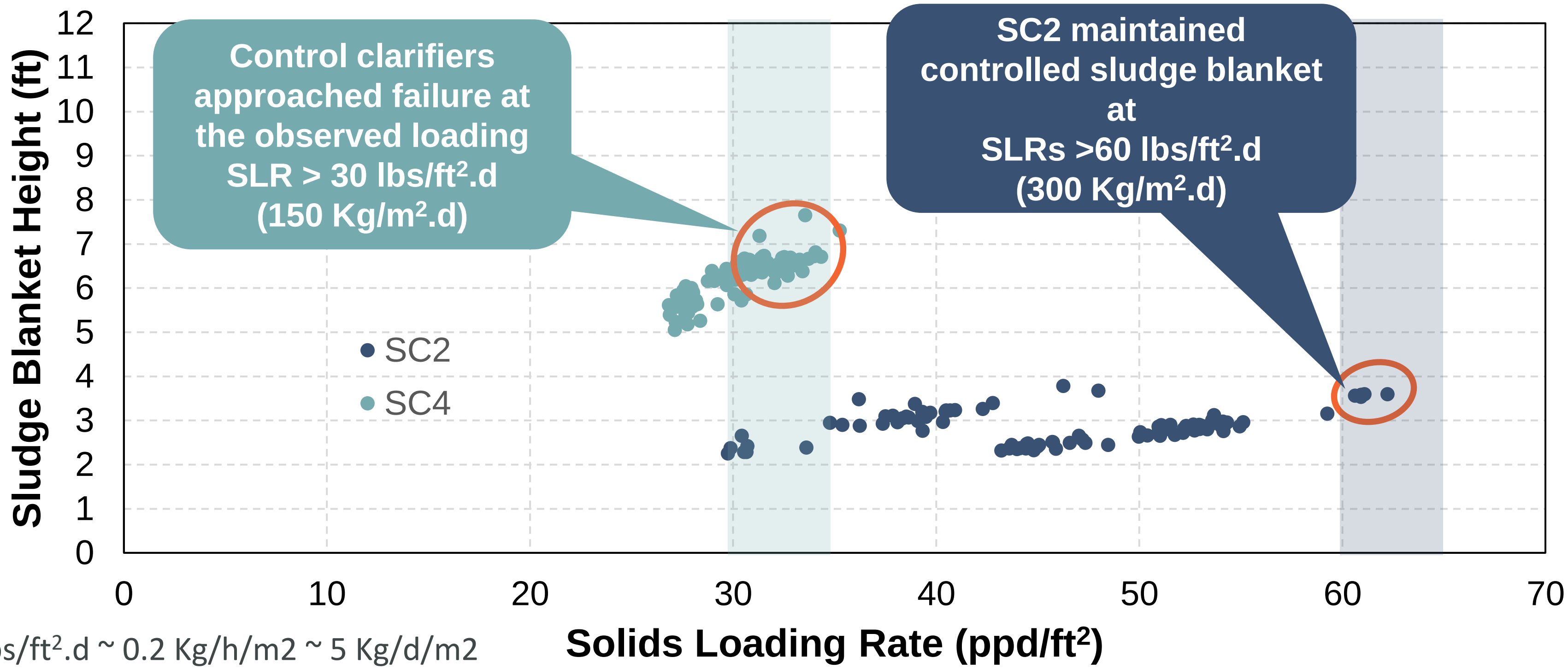
Flocculation parameters
FSS/DD

SETTLING COLUMN TEST DATA



	Vo ft/h (m/h) How quickly settling occurs (bigger is better)	K L/g How well sludge compacts (smaller is better)
DAS (Typical)	50 to 70 (15.2 to 21.3)	0.3 to 0.5
Conventional Activated Sludge (Typical)	25 to 45 (7.5 to 13.7)	0.3 to 0.6
Denver DAS Train	58.0 (17.7)	0.38
Denver Conventional Train	41.2 (12.6)	0.58

STRESS TESTING CONFIRMED SIGNIFICANT PERFORMANCE IMPROVEMENT FROM DENSIFICATION



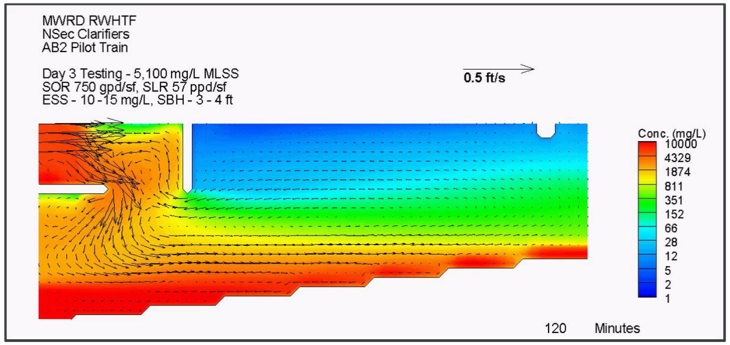
1 lbs/ft^2.d ~ 0.2 Kg/h/m2 ~ 5 Kg/d/m2

Surface overflow rate during testing ~ 700 – 900 gpd/ft^2 (1.2 to 1.5 m^3/m^2.h)

2D AND 3D MODELS DEVELOPED AND CALIBRATED AND APPLIED TO CAPACITY ANALYSIS

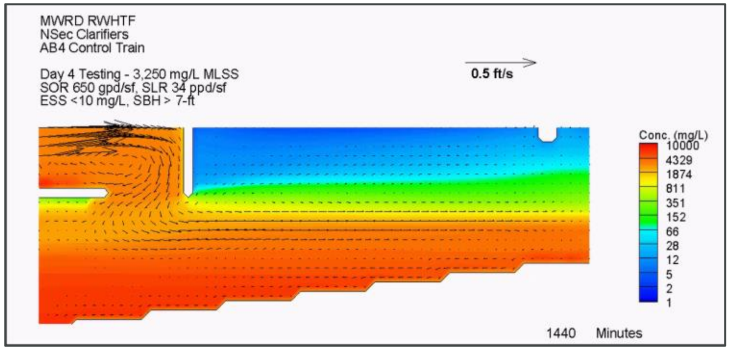
DAS Pilot Train

Clarifier 2	Average Flow (m^3/d)	Average MLSS (mg/L)	Sludge Blanket Height (ft)	
			Field Data	Model
Day 2	25360	5650	3	3
Day 3	34450	5180	4	4
Day 4	27260	5220	3	3



Control (Conventional) Train

Clarifier 4	Average Flow (m^3/d)	Average MLSS (mg/L)	Sludge Blanket Height (ft)	
			Field Data	Model
Day 1	26120	3240	6	6
Day 2	24230	3450	7	6
Day 4	28010	3800	7	7

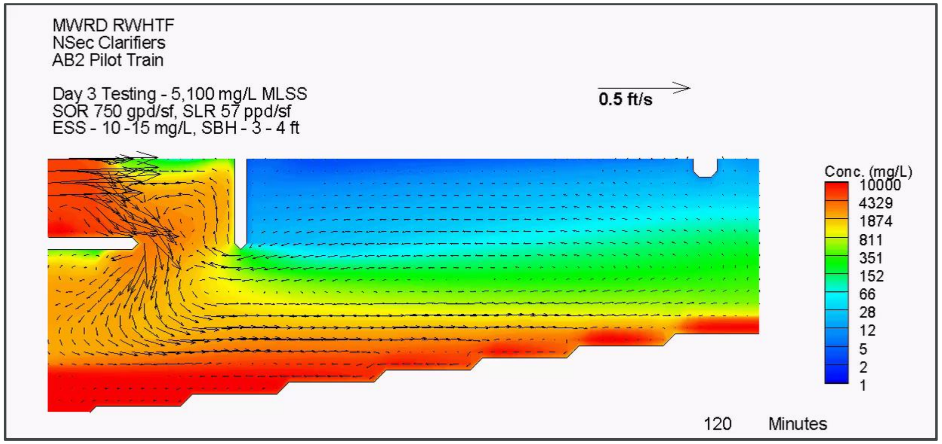


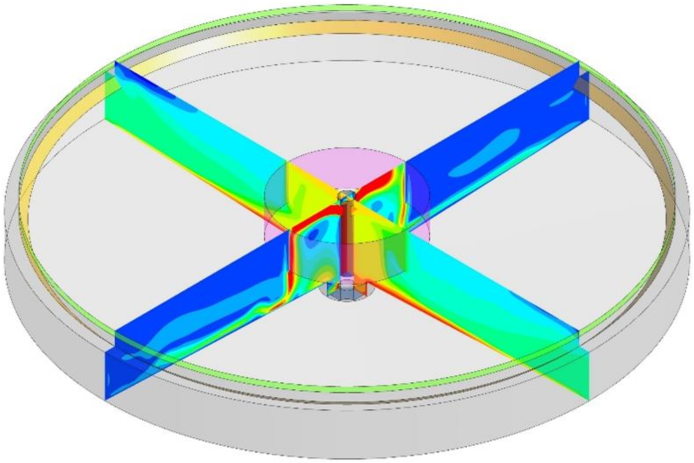
33

UNLOCKING CAPACITY WITH DAS AND CFD MODELING

- CFD modeling demonstrated that, with DAS, the clarifiers can treat the 2040 flows and loads at meet the stringent limits
  - Leverage existing infrastructure to avoid new ABs & SCs ($100M+ capital savings)

		SVI 80 mL/g
Aeration Basin		
Flow	mgd (m^3/d)	107 (405000)
MLSS	mg/L	**5600**
Aer SRT	days	8.0
Clarifiers		
SLR	lbs/ft^2.d (Kg/m^2.d)	65 (320)





CFD STUDY AT DIJON WRRF

- Context
  - WRRF with winter filamentous bulking events
  - Retrofit of existing clarifiers with densification technology
  - Comparison of conventional (SVI) and densified (SVI)

- Clarifier dimensions:
  - Diameter 52 m
  - Total surface of 1933 m2

- Analysis of different scenarios:
  - MLSS increased concentration at clarifier's inlet
    - From 3.5 g/L to 6 g/L at standard dry-weather operation conditions Q_{in} = 300 m^3/h (1.90 mgd) Q_{RAS} = 280 m^3/h (1.78 mgd).
  - Simulation of high surface overflow rates and low surface underflow rates at MLSS of 3.5 g/L:
    - Clarifier inflow Q_{in} (2000 to 4000 m3/h) (12.68 to 25.36 mgd)
    - Recirculated activated sludge (280 to 1290 m3/h) (1.77 mgd to 8.18 mgd)

CFD STUDY AT DIJON WRRF

MODEL CONSTRUCTION

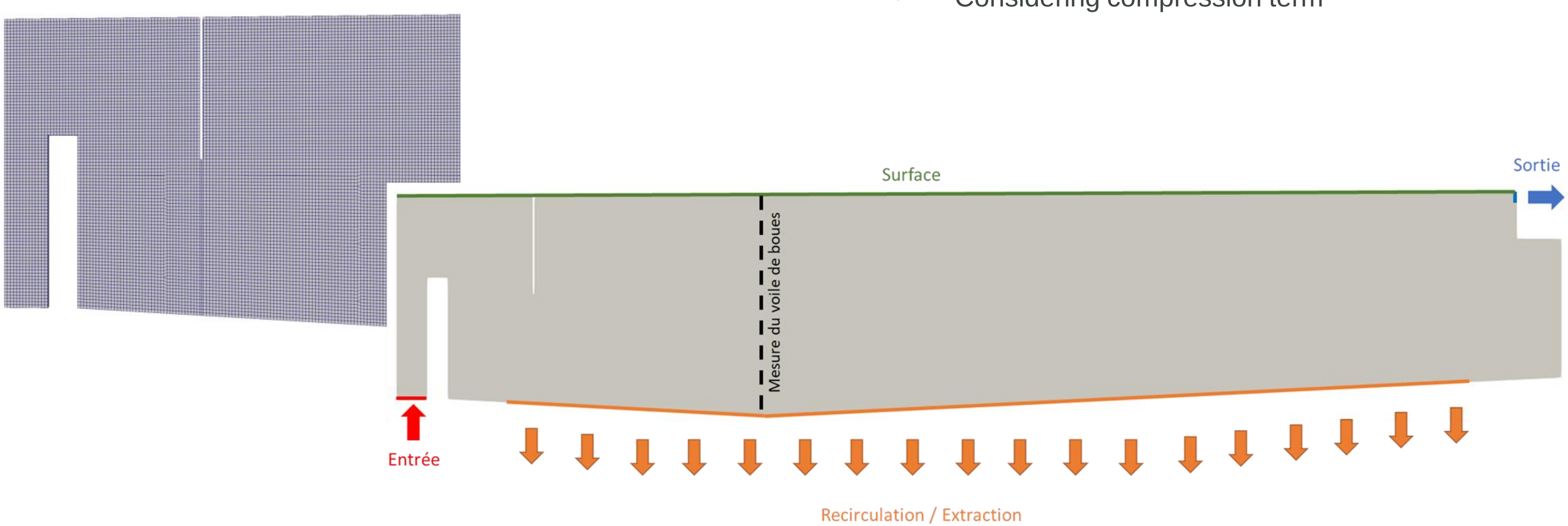
Simulations at steady state

Axisymmetric hexahedral mesh

46 000 cells (size 5 cm)

Hypothesis:

- Mixture model
- Turbulence : k-ε (buoyancy term)
- Non-newtonian behavior (Bingham plastic)
- Settling velocity parameters deriving from batch tests, (*ROCHE et al., 2022a, 2023*)
  - Vesilind exponential model
  - Considering compression term



CFD STUDY AT DIJON WRRF

RESULTS

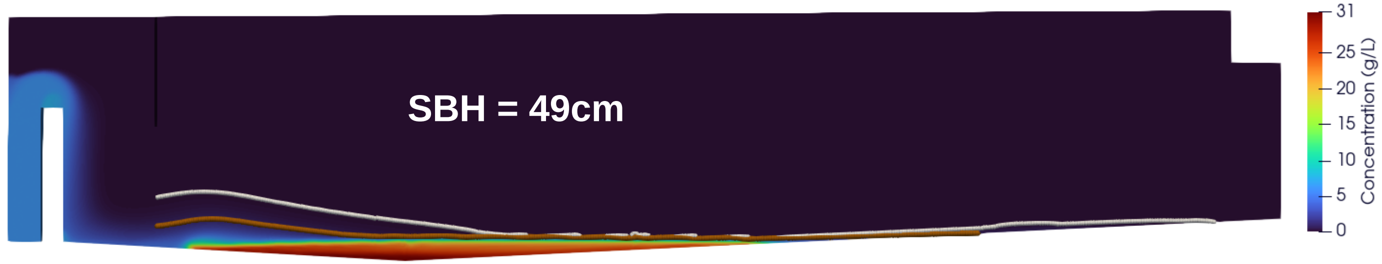
Densified sludge

Q_{in} = 300 m³/h

Q_{RAS} = 280 m³/h

MLSS = 6 g/L

SRL = 43 Kg/m².d



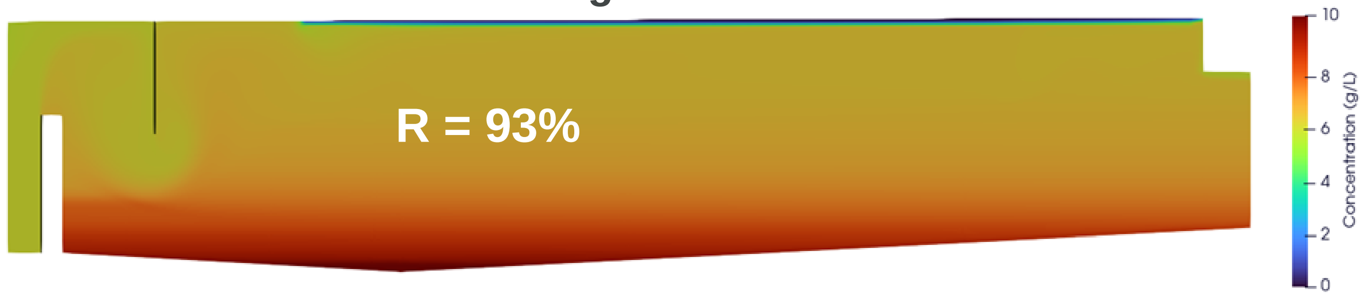
Conventional activated sludge

Q_{in} = 300 m³/h

Q_{RAS} = 280 m³/h

MLSS = 6 g/L

SRL = 43 Kg/m².d

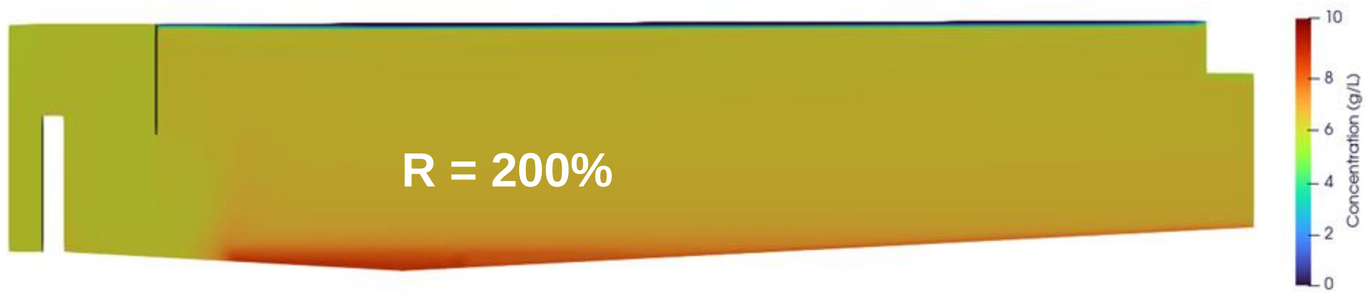


Q_{in} = 300 m³/h

Q_{RAS} = 600 m³/h

MLSS = 6 g/L

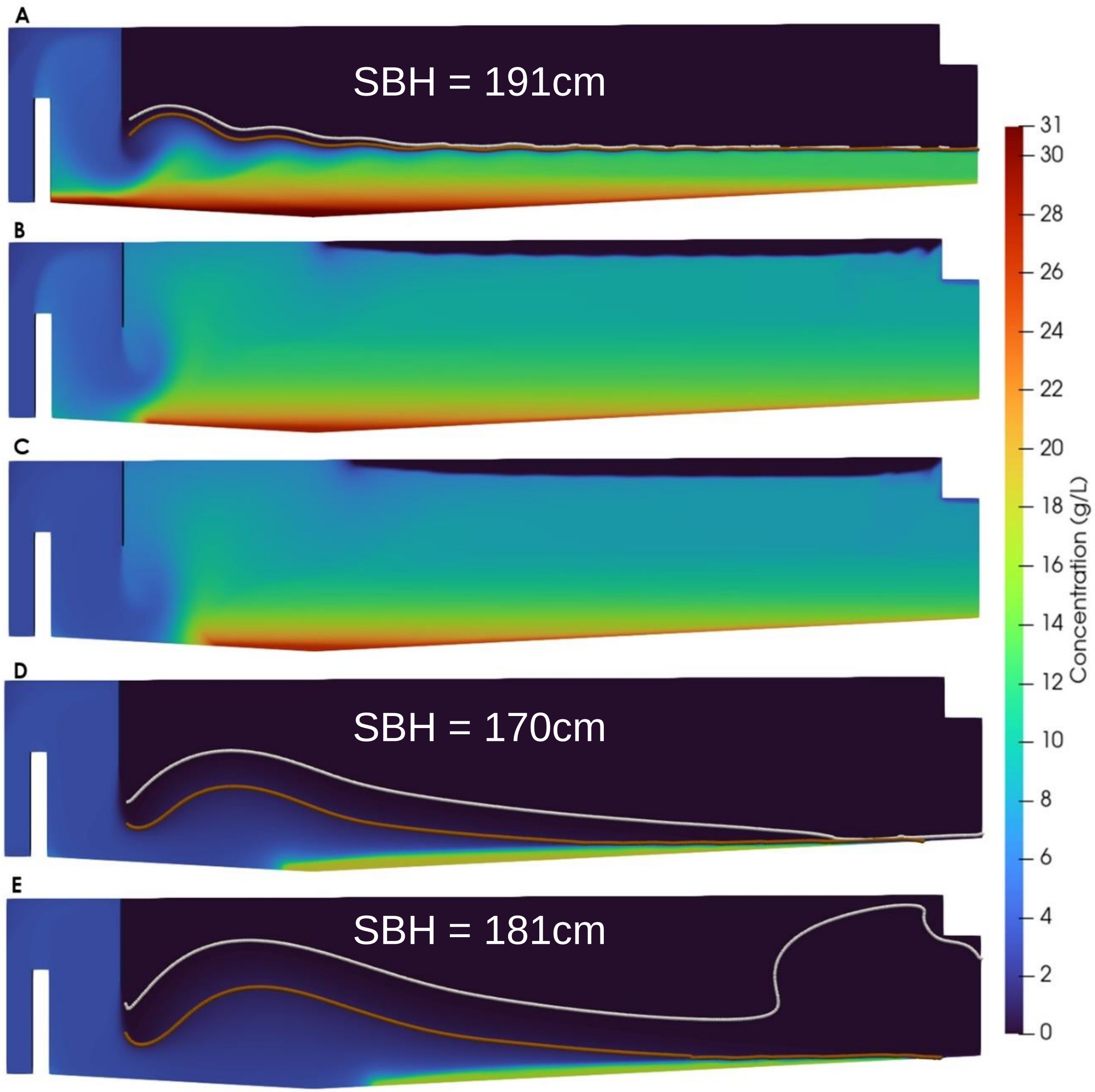
SRL = 67 Kg/m².d



CFD STUDY AT DIJON WRRF

RESULTS

Densified MLSS 3.5 g/L



	Q_{in} m³/h (mgd)	Q_{RAS} m³/h (mgd)	Applied Peak SOR / Design SOR
A	2000 (12.68)	280 (1.78)	155%
B	3000 (19.02)	450 (2.85)	233%
C	4000 (25.36)	600 (3.80)	310%
D	3000 (19.02)	900 (5.71)	233%
E	4000 (25.36)	1290 (8.18)	310%

CFD STUDY AT DIJON WRRF

CONCLUSIONS

- The simulations were conducted employing a mesh that ensured reliable convergence and minimized computational time using OpenFOAM®.
- Even within an axisymmetric approach, the simulations exhibited a precise depiction of the full-scale clarifier. Field observations confirmed the absence of significant sludge blanket in dry-weather conditions with DAS
- Parameters of the settling model were calibrated before in batch settling test. A compression term was considered to obtain a better estimation of the sludge blanket height.
- It is recommended to conduct non-steady state simulations to validate the model against experimental data acquired on-site.
- Further research is needed to analyze the effects of the viscosity of the high concentrated densified sludge

AGENDA AND HOUSEKEEPING

Presentation 1
Ed Wicklein (Carollo Engineers)

Presentation 2
María Elena Valle de Medina (ENGEES/UNISTRA), Julien Laurent (ENGEES/UNISTRA), Alonso Griborio (Hazen)

Presentation 3
Christopher DeGroot (Western University, Canada)

Presentation 4
Giacomo Bellandi (AM-Team)

Presentation 5
Stephen Saunders (Ibis Group), and Giacomo Bellandi (AM-Team)

Q&A Session Moderator: *María Elena Valle de Medina (ENGEES/UNISTRA)*

PREDICTING OXYGEN MASS TRANSFER USING A CFD POPULATION BALANCE MODEL

Christopher DeGroot
Western University, Canada

PREDICTING OXYGEN MASS TRANSFER USING A CFD POPULATION BALANCE MODEL

- Aeration is typically the largest consumer of power in WWTP
- Depending on power generation characteristics, GHG emissions associated with electricity use can be significant
- Nitrous oxide (N_2O) emissions are typically the most significant direct GHG emissions from a WWTP
- Therefore, modelling aeration is an area of significant interest for reducing operating expenses and GHG emissions

GAS-LIQUID MASS TRANSFER

- Understanding gas-liquid mass transfer is fundamental to modelling and optimizing activated sludge processes
- The rate of mass transfer per unit volume is given as:

$$\frac{dC}{dt} = K_L a(C_s - C)$$

- In conventional process modelling $K_L a$ is considered as a single lumped parameter that is homogeneous within each CSTR
  - Can add α factor to correct from clean water to process water
- Using CFD modelling, K_L and a can be taken separately and can be spatially varying depending on local fluid dynamics

GAS-LIQUID MASS TRANSFER

- The mass transfer coefficient is given by Higbie's model:

$$K_L = \sqrt{\frac{D_L|U_r|}{\pi d_S}}$$

- The interfacial area density is given as:

$$a = \frac{6\alpha_g}{d_s}$$

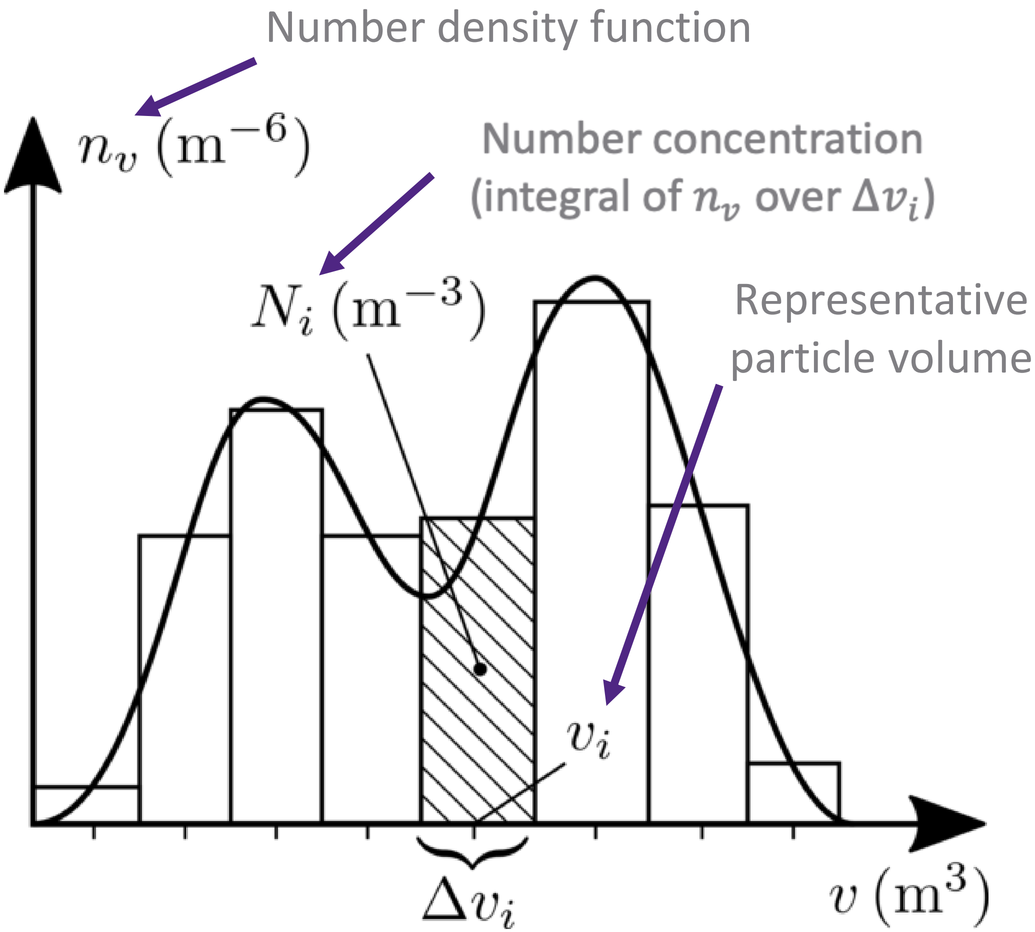
- In many CFD studies the Sauter diameter d_S is assumed to be constant throughout the domain
  – Is this accurate?

POPULATION BALANCE MODEL

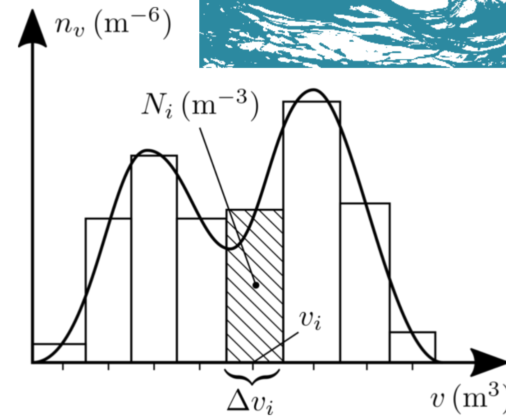
- The bubble size distribution (BSD) can be modelled with a population balance model (PBM) within a CFD simulation
- Removes assumption of constant bubble size, but adds complexity
- PBM divides BSD into finite number of discrete size bins
- Fluid volume fractions are related by:

$$N_i v_i = \alpha_i$$

$$\Sigma \alpha_i = \alpha_g$$



POPULATION BALANCE MODEL



- Number concentrations are converted to size group fractions:

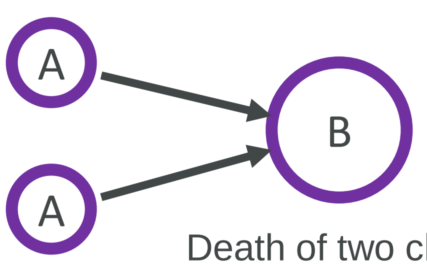
$$f_i = \frac{\alpha_i}{\alpha_g}$$

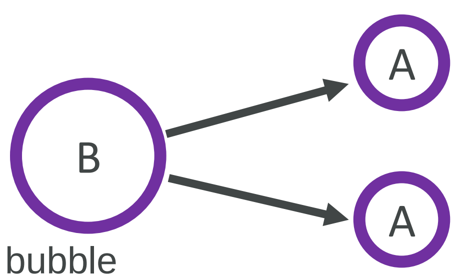
$$\Sigma f_i = 1$$

- A conservation equation for each fraction is solved in the domain:

$$\frac{\partial(\alpha_g f_i)}{\partial t} = \nabla \cdot \left(\alpha_g \boldsymbol{U}_g f_i\right) = v_i\left(B_{f\,i,C} - D_{f\,i,C} + B_{f\,i,B} - D_{f\,i,B}\right)$$

- Birth and death rates require closure models





Death of two class A bubbles

Birth of one class B bubble

Death of one class B bubble

Birth of two class A bubble

CFD-PBM MODEL

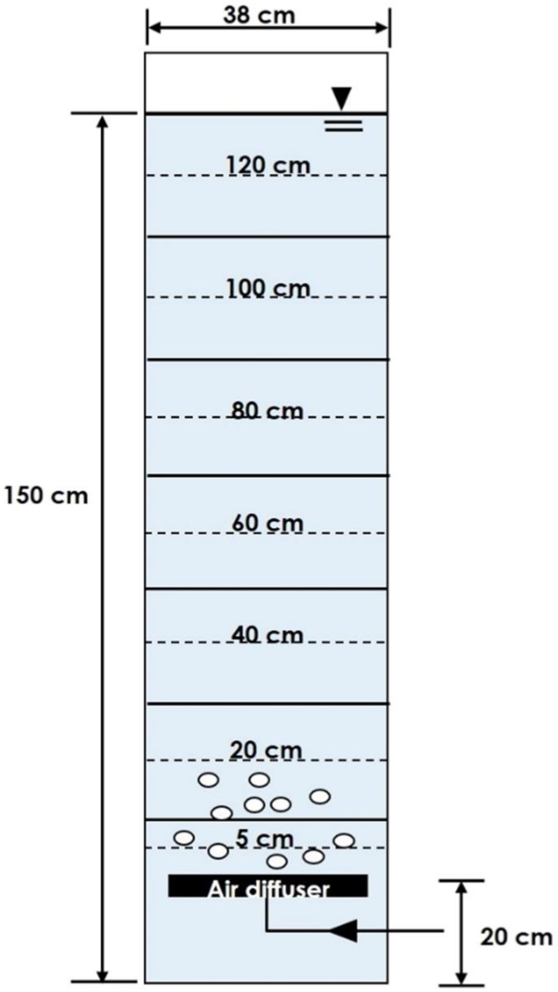
- To test the effect of constant vs. variable BSD, we conducted CFD simulations for a bubble column where experimental data existed in the literature
- The multiphase CFD model is based on an Euler-Euler two-fluid model implemented in OpenFOAM where the fluid volume fraction is solved by the PBM
- Governing equations are derived by conditional averaging of the single-fluid governing equations
- The momentum equations include an additional force term:

$$\Sigma \boldsymbol{F}_k = \boldsymbol{F}_D + \boldsymbol{F}_{VM} + \boldsymbol{F}_L$$

- Each force term requires a closure model

BUBBLE COLUMN STUDY

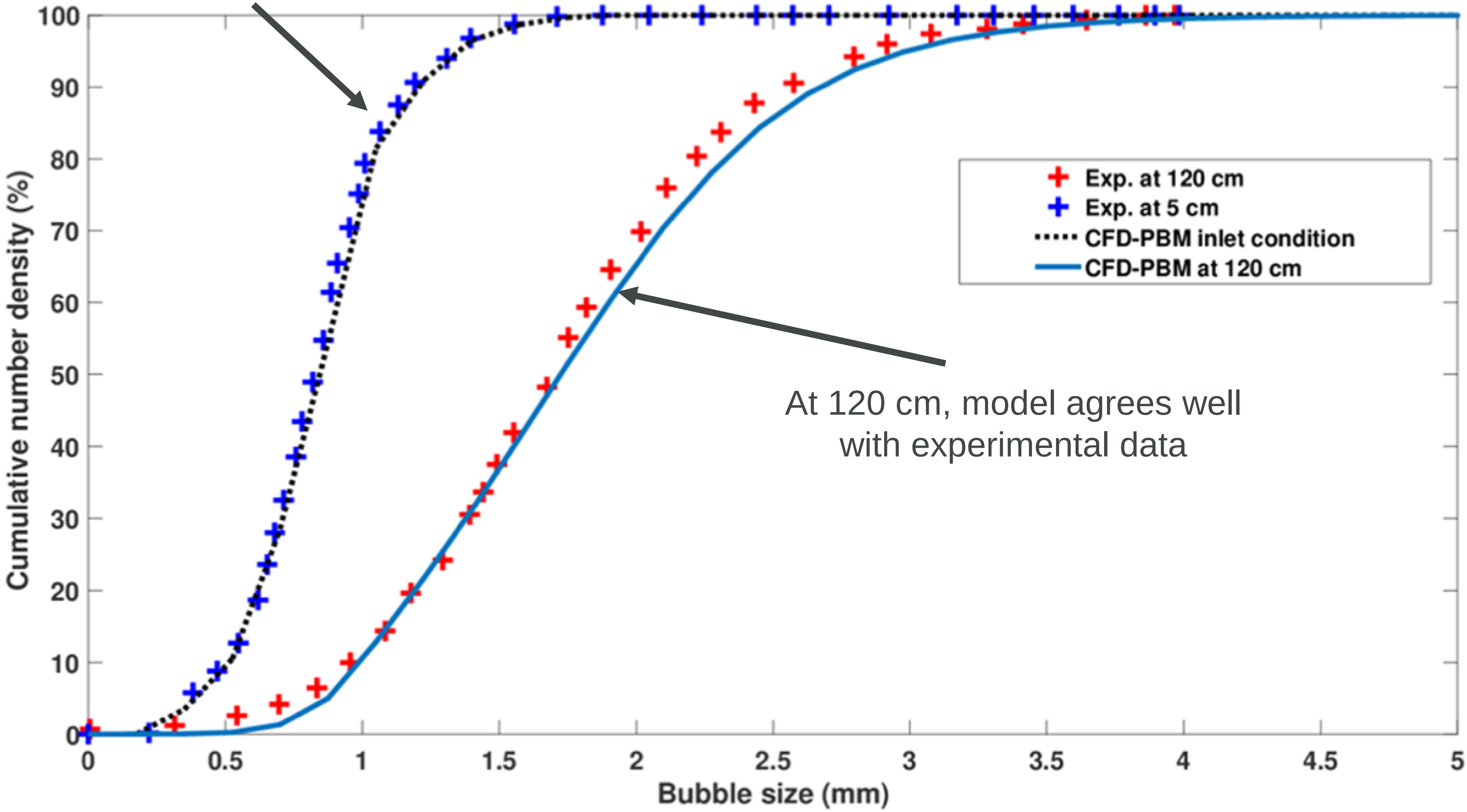
- To test the effect of constant vs. variable BSD, we conducted CFD simulations for a bubble column where good experimental data existed in the literature
- Tested sensitivity of model parameters and best results with:
  - Drag force model: Schiller-Naumann
  - Lift force model: Tomiyama
  - Virtual mass force coefficient: 0.5
  - Number of BSD size classes: 32
  - Coalescence model: Prince and Blanch
  - Breakup model: Lehr
  - Air volume fraction at inlet: 0.9



Amaral, A., Bellandi, G., Rehman, U., Neves, R., Amerlinck, Y., & Nopens, I. (2018). Water Research, 131, 346-355.

BUBBLE COLUMN STUDY – RESULTS

Shows that BSD does not change from diffuser to 5 cm plane; can use this data as boundary condition

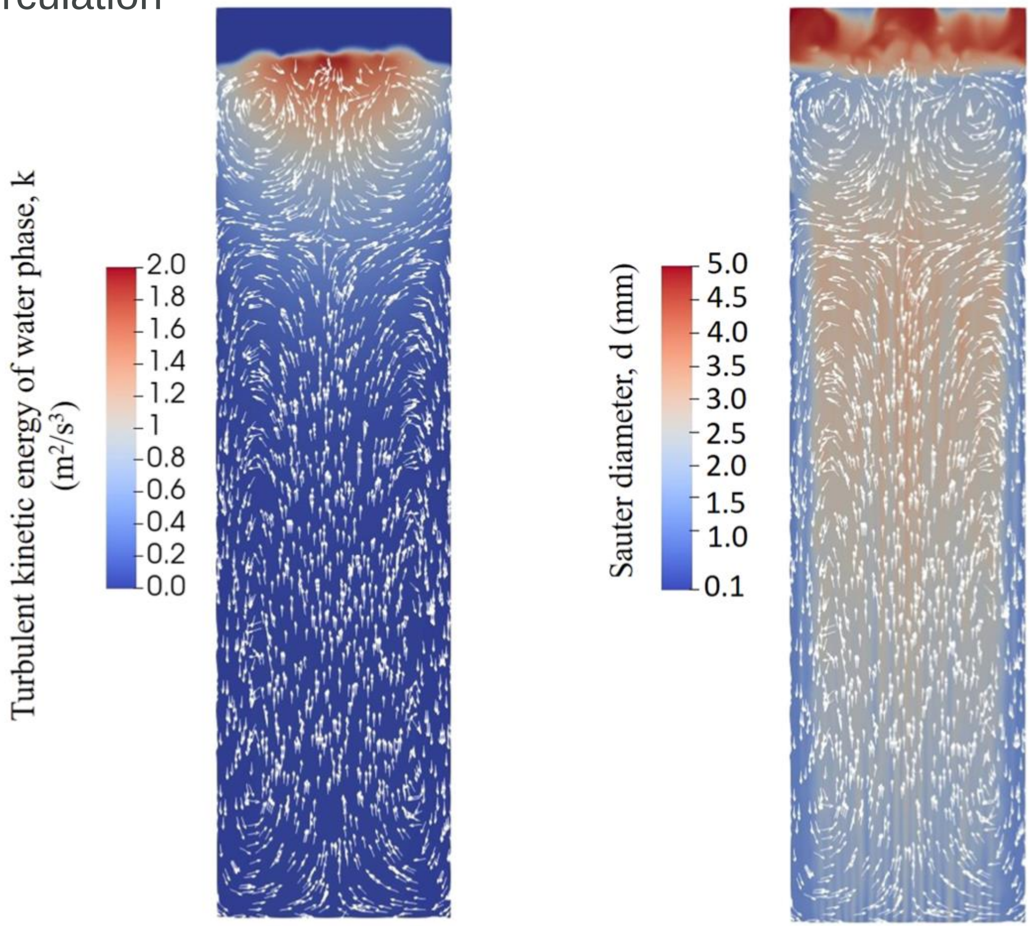


At 120 cm, model agrees well with experimental data

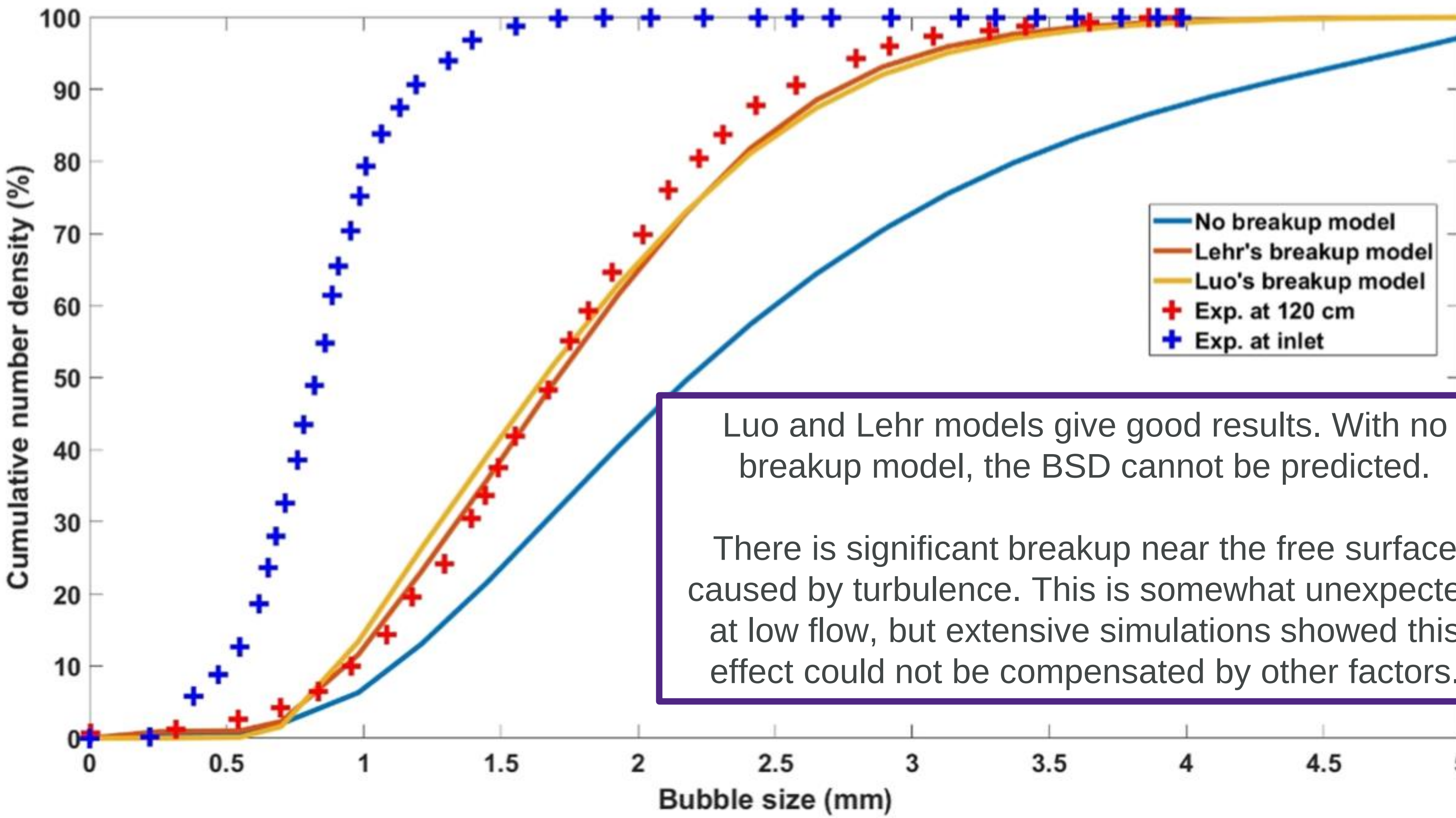
BUBBLE COLUMN STUDY – RESULTS

High TKE near free surface in recirculation zone

TKE causes bubble breakup



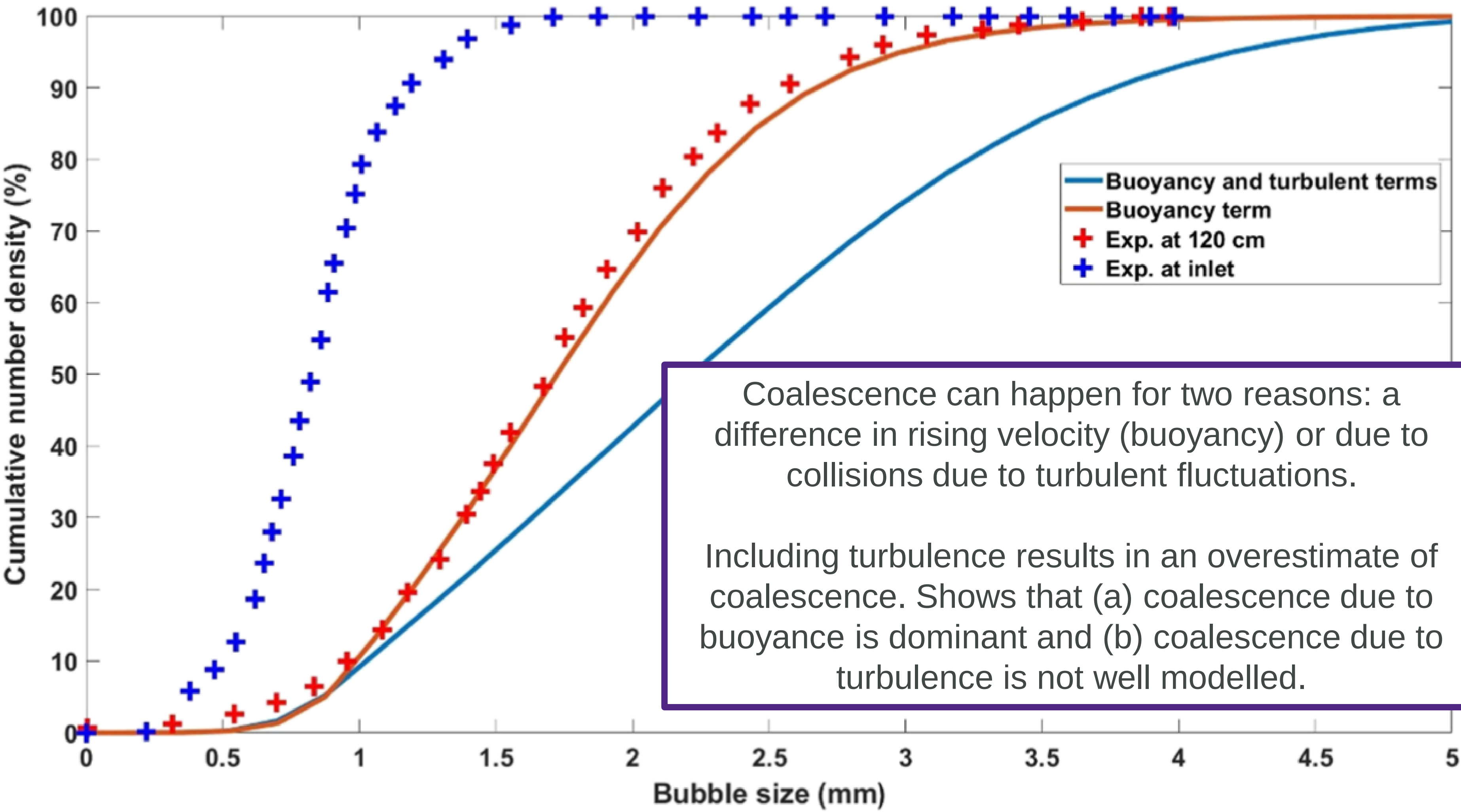
BUBBLE COLUMN STUDY – EFFECT OF BREAKUP MODEL



Luo and Lehr models give good results. With no breakup model, the BSD cannot be predicted.

There is significant breakup near the free surface caused by turbulence. This is somewhat unexpected at low flow, but extensive simulations showed this effect could not be compensated by other factors.

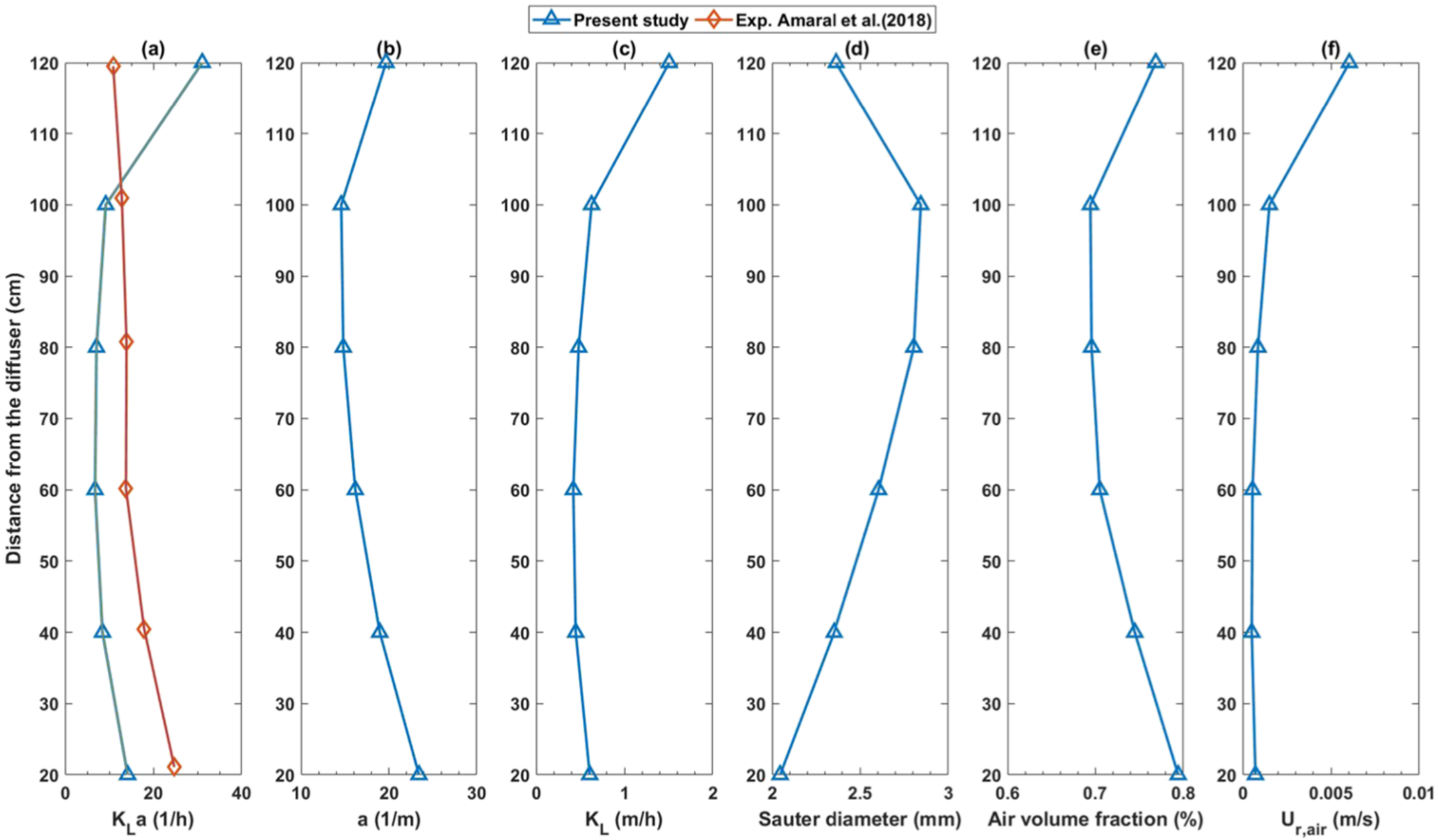
BUBBLE COLUMN STUDY – EFFECT OF COALESCENCE MODEL



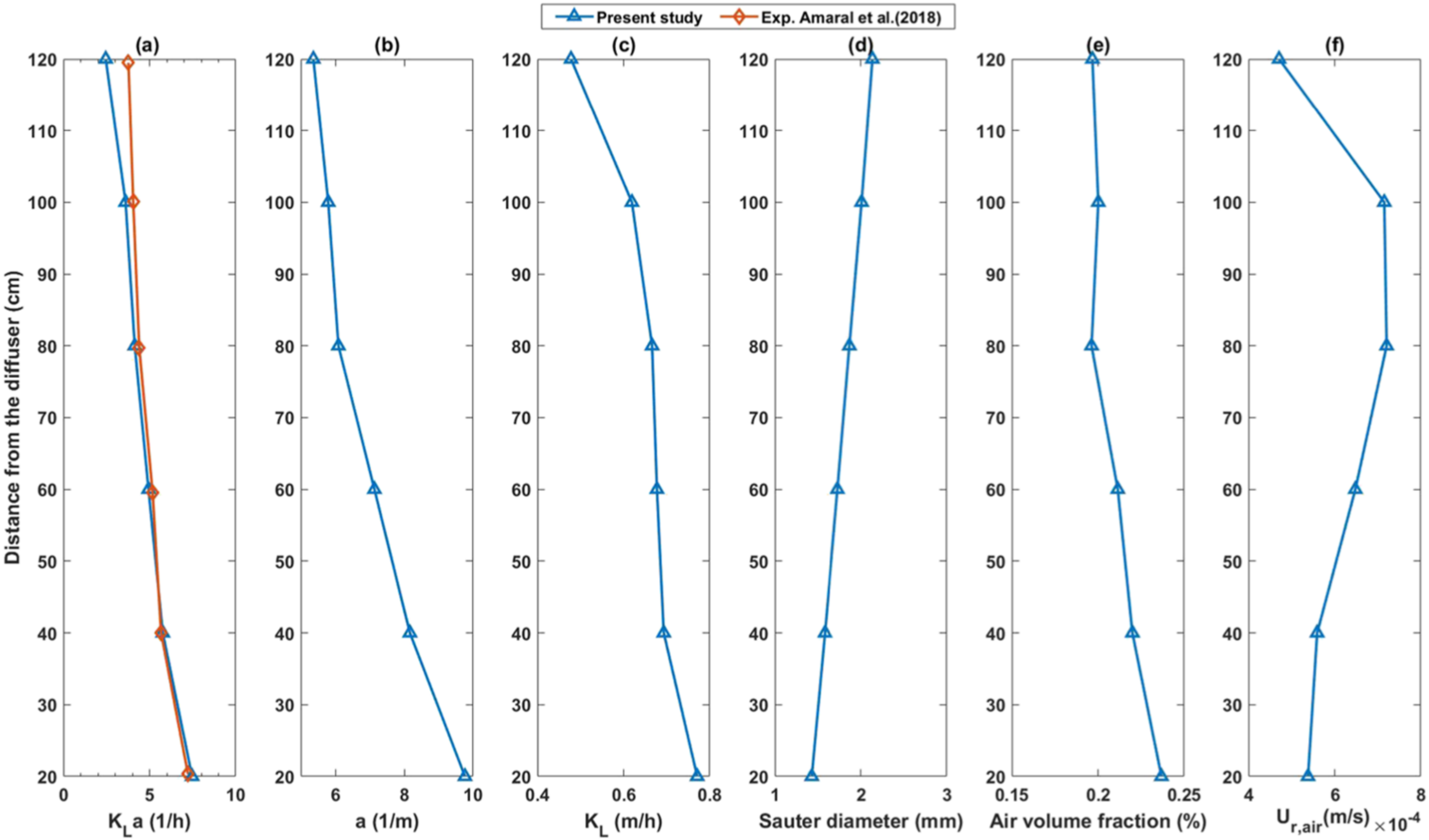
Coalescence can happen for two reasons: a difference in rising velocity (buoyancy) or due to collisions due to turbulent fluctuations.

Including turbulence results in an overestimate of coalescence. Shows that (a) coalescence due to buoyance is dominant and (b) coalescence due to turbulence is not well modelled.

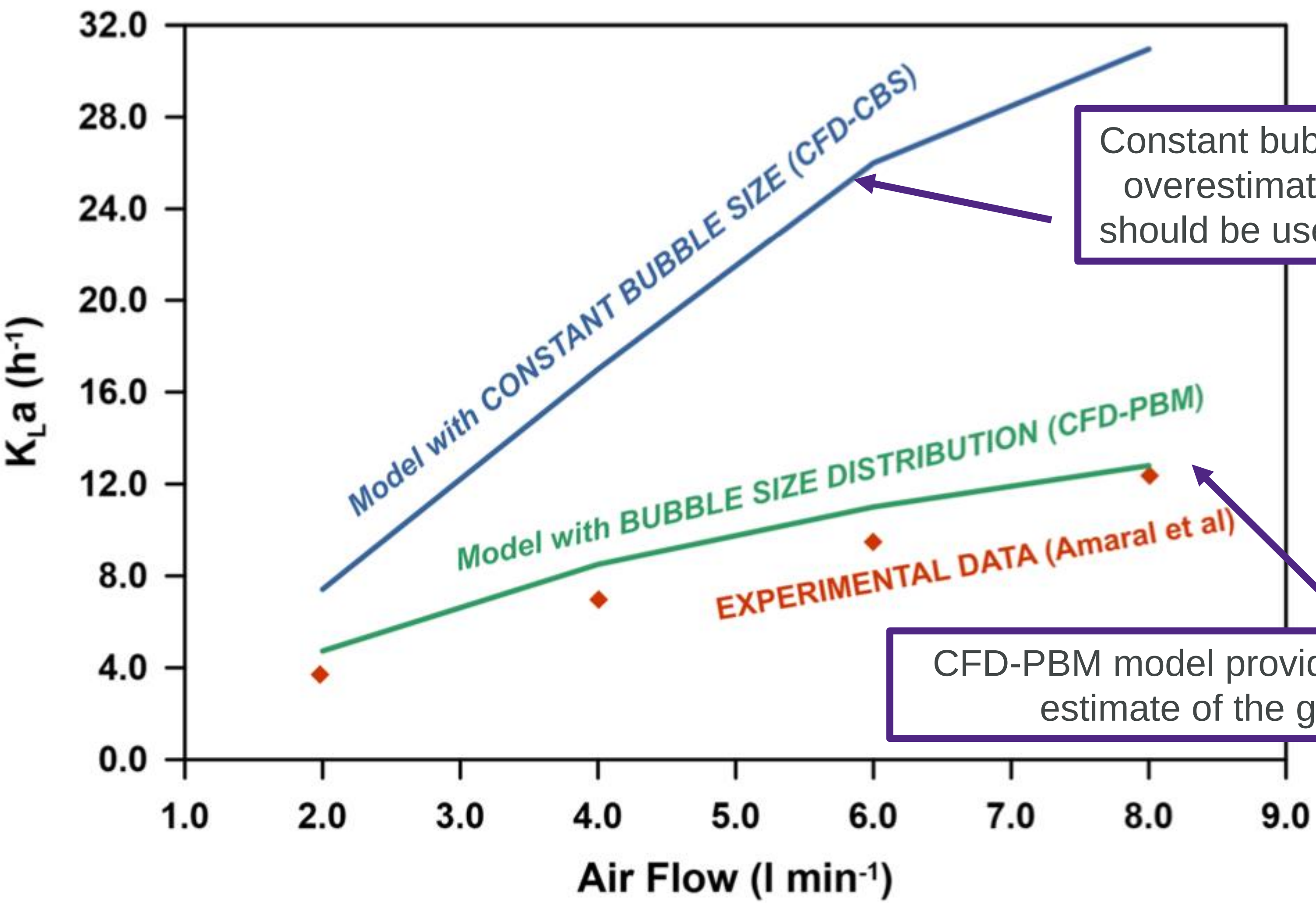
BUBBLE COLUMN STUDY – 8 L/MIN



BUBBLE COLUMN STUDY – 2 L/MIN



BUBBLE COLUMN STUDY – GLOBAL OXYGEN MASS TRANSFER



Constant bubble size model overestimates K_La and is should be used with caution

CFD-PBM model provides an accurate estimate of the global K_La

KEY MESSAGES

- To model the BSD in aeration applications, a CFD-PBM model can be used
- It is important to understand the specific real-world application and conduct validation on the model assumptions if this type of model has not been applied before or is not well-understood
- In the study shown, constant bubble size does not accurately model oxygen mass transfer; for real-world applications the validity of the constant bubble size assumption should be examined and a PBM model should be considered if the BSD cannot be assumed uniform

AGENDA AND HOUSEKEEPING

Presentation 1
Ed Wicklein (Carollo Engineers)

Presentation 2
María Elena Valle de Medina (ENGEES/UNISTRA), Julien Laurent (ENGEES/UNISTRA), Alonso Griborio (Hazen)

Presentation 3
Christopher DeGroot (Western University, Canada)

Presentation 4
Giacomo Bellandi (AM-Team)

Presentation 5
Stephen Saunders (Ibis Group), and Giacomo Bellandi (AM-Team)

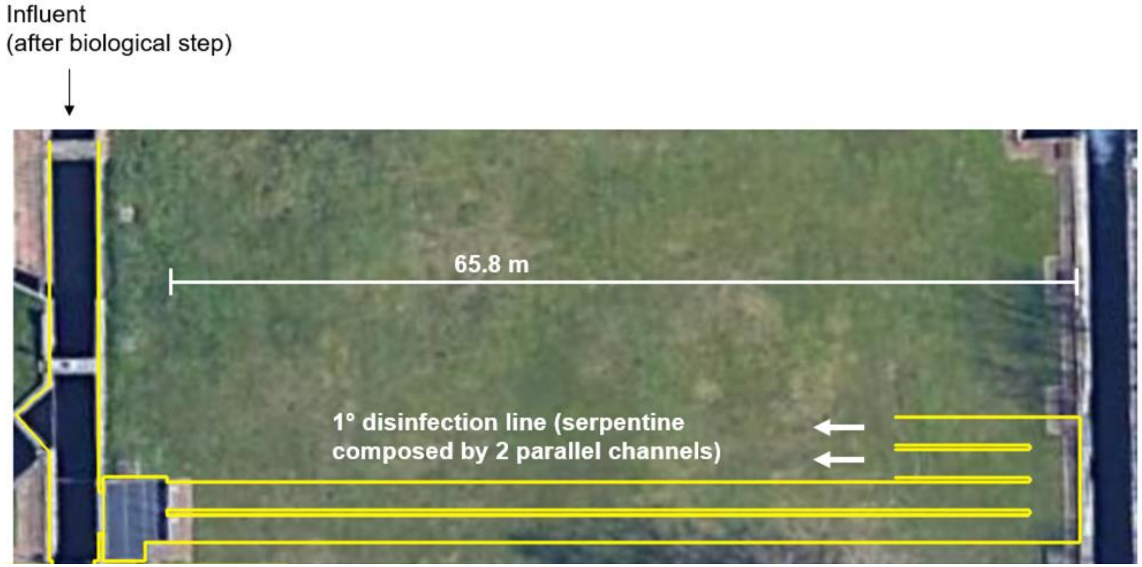
Q&A Session Moderator: *María Elena Valle de Medina (ENGEES/UNISTRA)*

DESIGN SUPPORT FOR DRINKING WATER DISINFECTION

Giacomo Bellandi

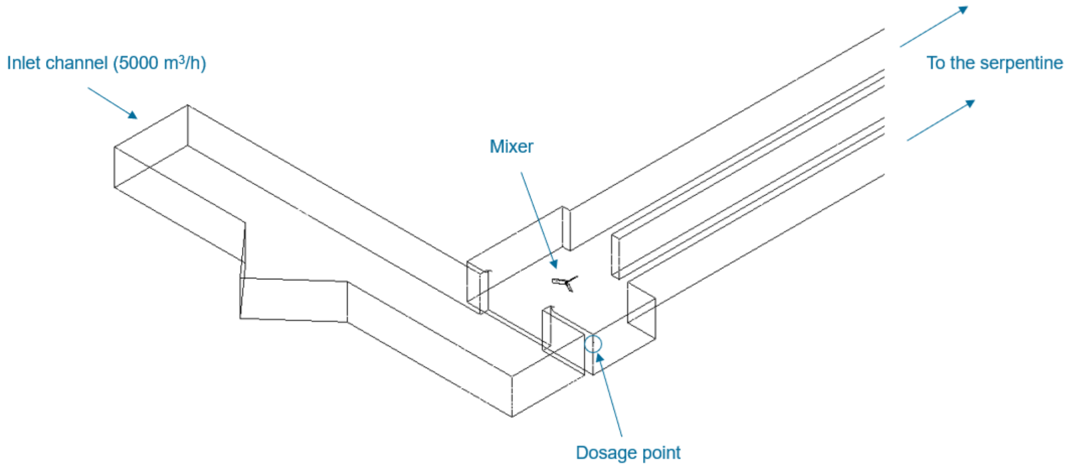
AM-TEAM

DISINFECTION UNIT IN PERO (ITALY) – INITIAL CONFIGURATION

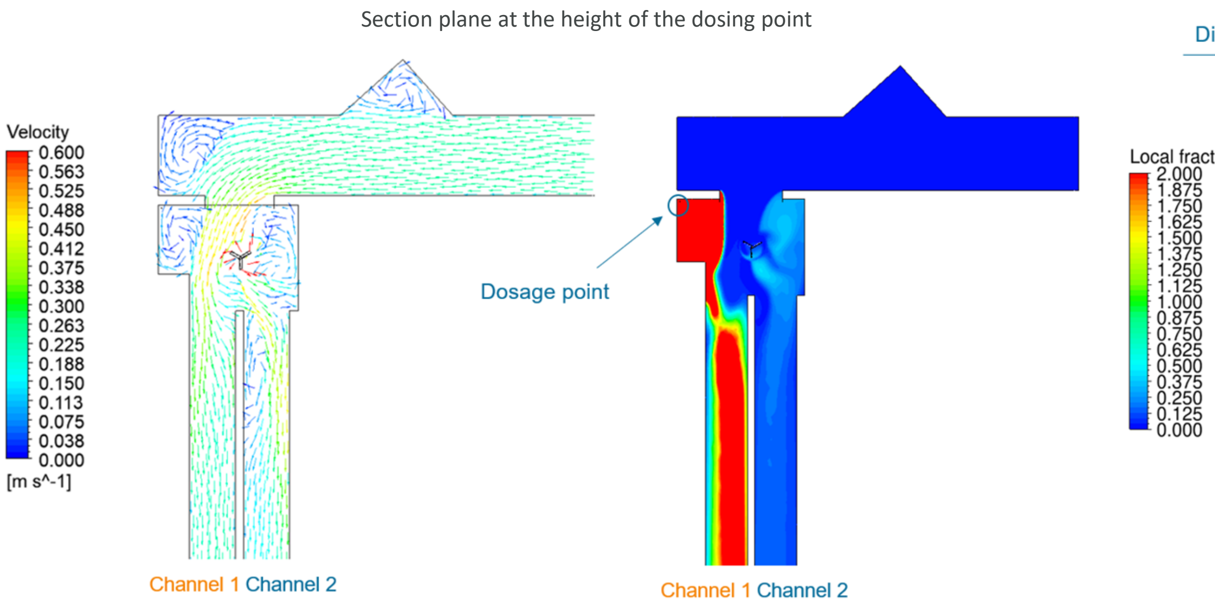


Main objective:

- Ensure optimal disinfectant usage (Is the dosage location good? Are we wasting disinfectant?)



INITIAL CONFIGURATION



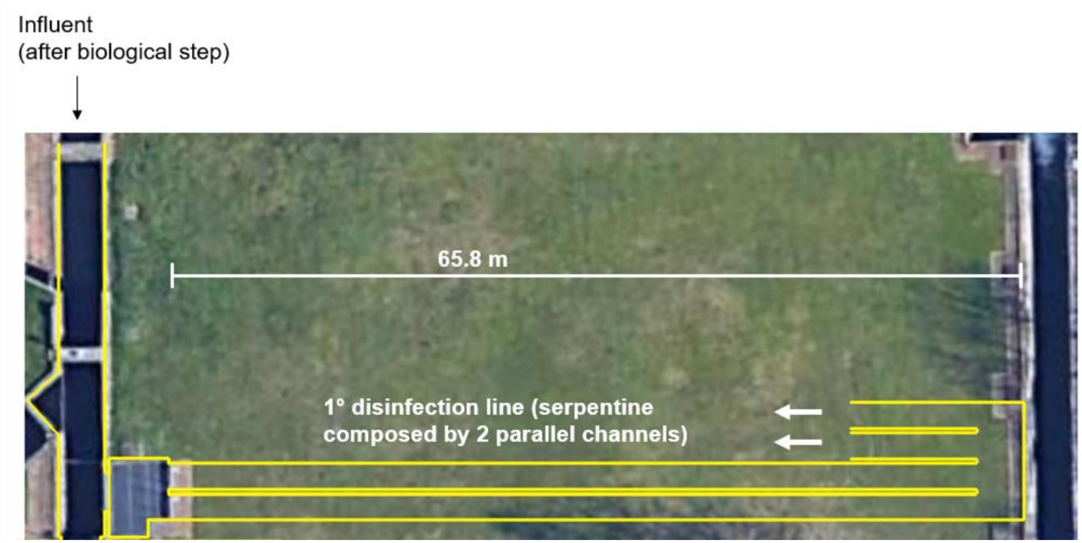
Water velocity

Disinfectant distribution

The dosage point is placed in a stagnation zone.

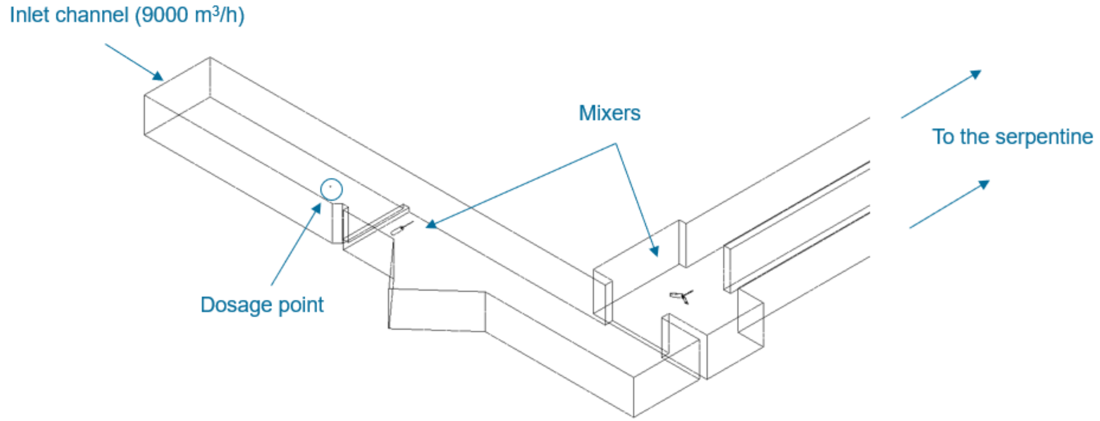
Channel 1: *Local fraction / Desired fraction* = 1.8
Channel 2: *Local fraction / Desired fraction* = 0.14

IMPROVED CONFIGURATION



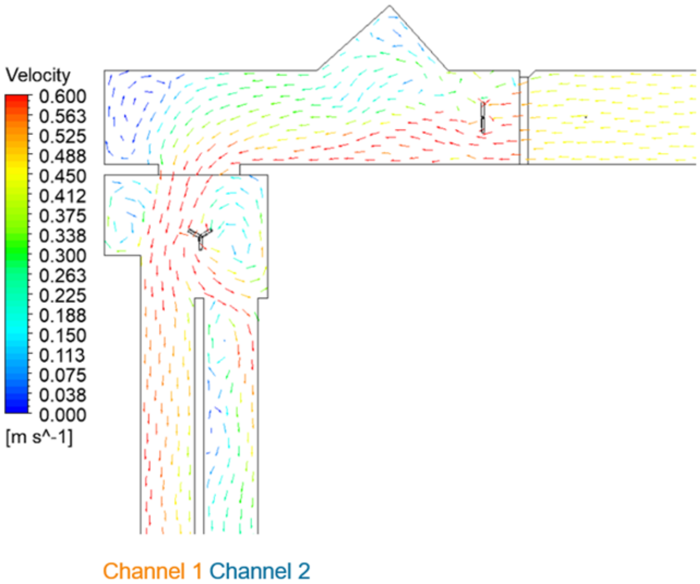
The dosing point was moved in the middle of the inlet channel, with a mixer placed afterwards.

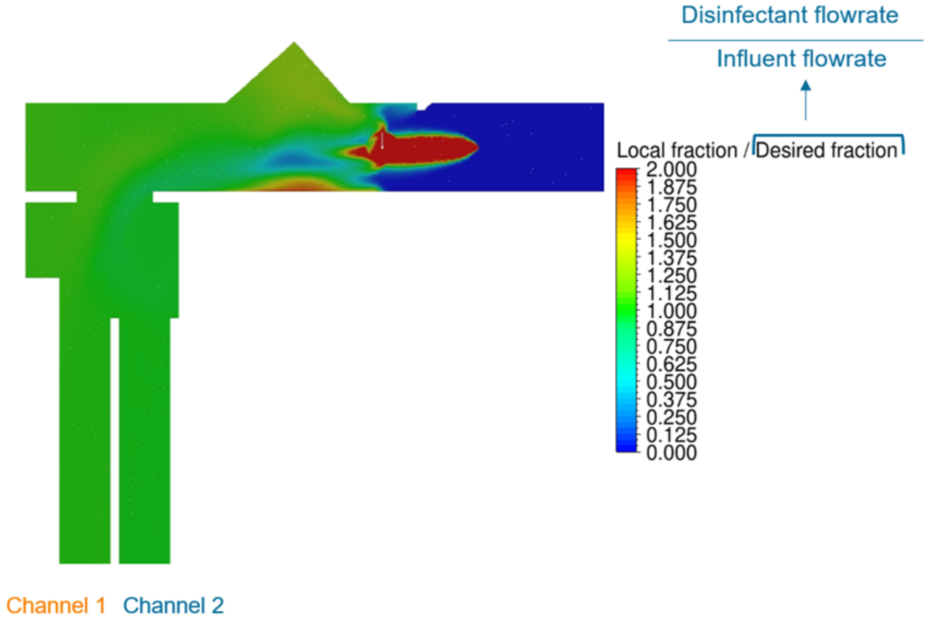
This configuration was tested with the maximum operating flowrate (9000 m^3/h)



IMPROVED CONFIGURATION

Section plane at the height of the dosing point



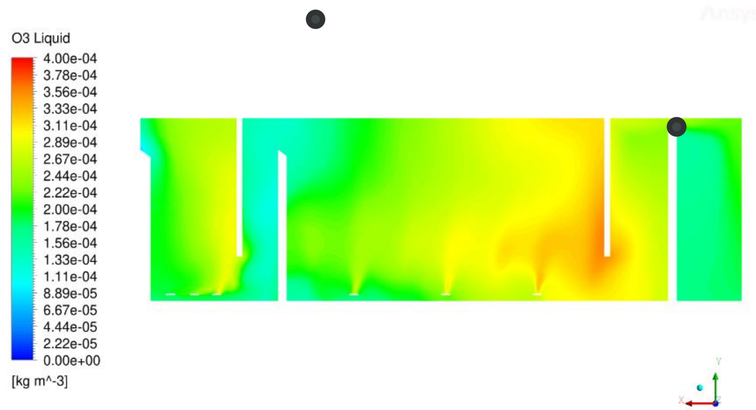


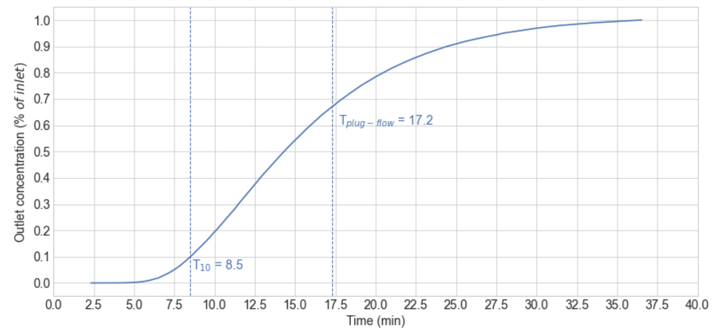
Water velocity

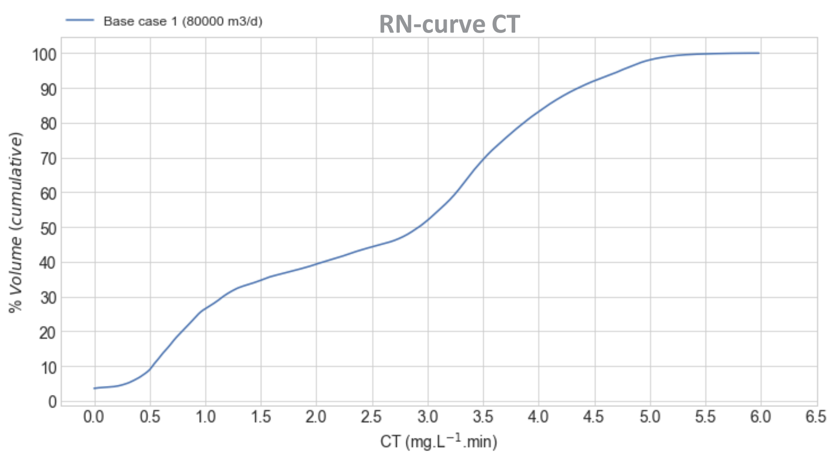
Disinfectant distribution

CT CALCULATION

▪







- Cumulative distribution of the CT in the tank. The CT is calculated in each cell as follows:

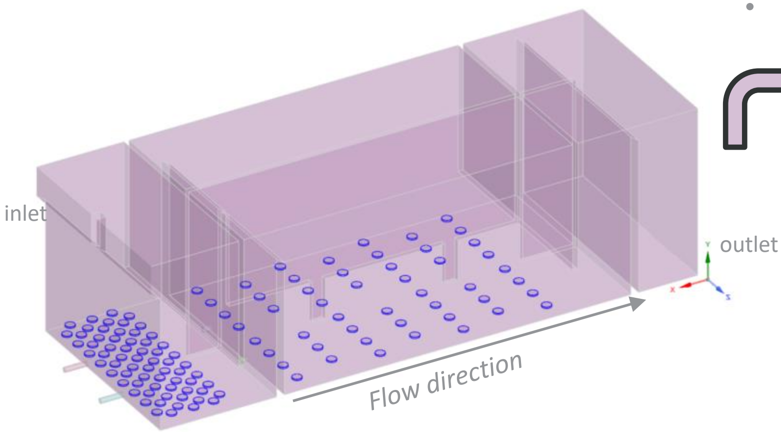
$$CT_i = Ci_{O_3} * \tau_i$$

Where Ci_{O_3} is the O_3 concentration in the cell i, and τ_i is the local water age in the cell i

- CT slows down in the central part due to suboptimal mean age and not uniform O_3 in compartment 2
- CT slows down towards the end due to O_3 consumption and decay after the second ozone injection

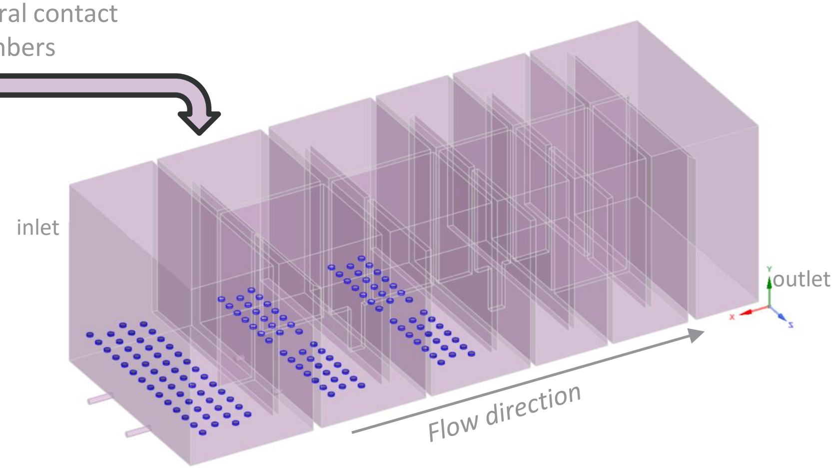
OZONATION PLANT REVAMPING FOR DISINFECTION

Existing design



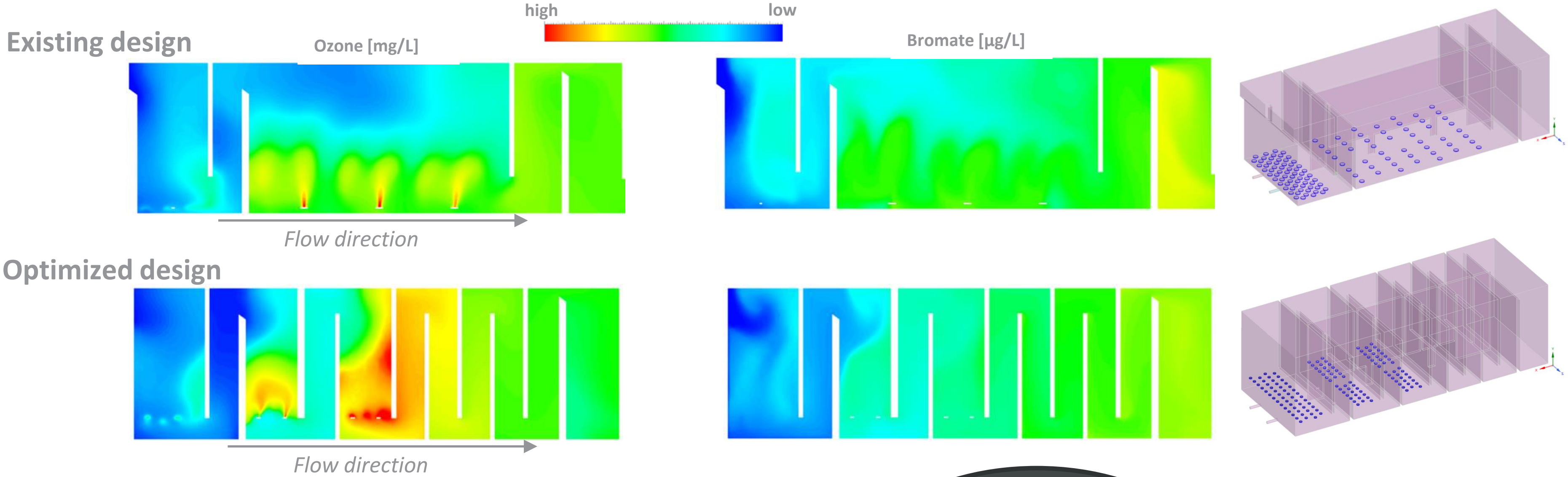
- Three ozone chambers
- Several contact chambers

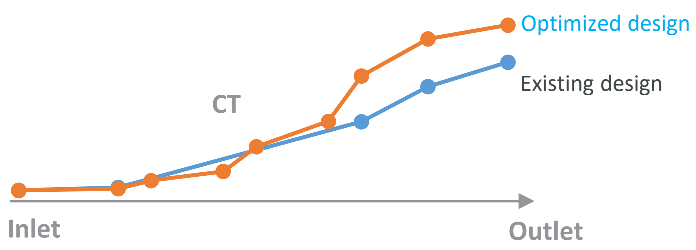
Optimized design



- Ozonation reactor for disinfection (Revamping)
- Consideration of the water matrix
- Local gas-liquid transfer model
- What is the ozone dose requirement to reach:
  - minimum CT (in winter and summer conditions)
  - minimize the bromate

OZONATION PLANT REVAMPING FOR DISINFECTION





Same ozone dose
Optimized design:

- **Improved Baffling Factor**
- **Improved CT (substantial dose saving)**
- **Same bromate**

AGENDA AND HOUSEKEEPING

Presentation 1
Ed Wicklein (Carollo Engineers)

Presentation 2
María Elena Valle de Medina (ENGEES/UNISTRA), Julien Laurent (ENGEES/UNISTRA), Alonso Griborio (Hazen)

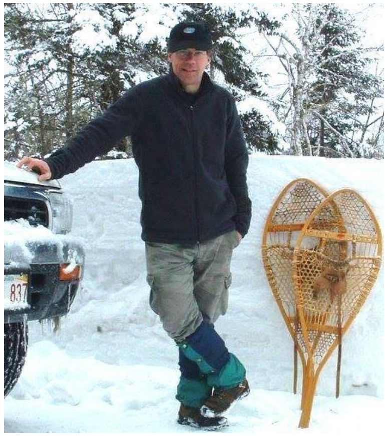
Presentation 3
Christopher DeGroot (Western University, Canada)

Presentation 4
Giacomo Bellandi (AM-Team)

Presentation 5
Stephen Saunders (Ibis Group), and Giacomo Bellandi (AM-Team)

Q&A Session Moderator: *María Elena Valle de Medina (ENGEES/UNISTRA)*

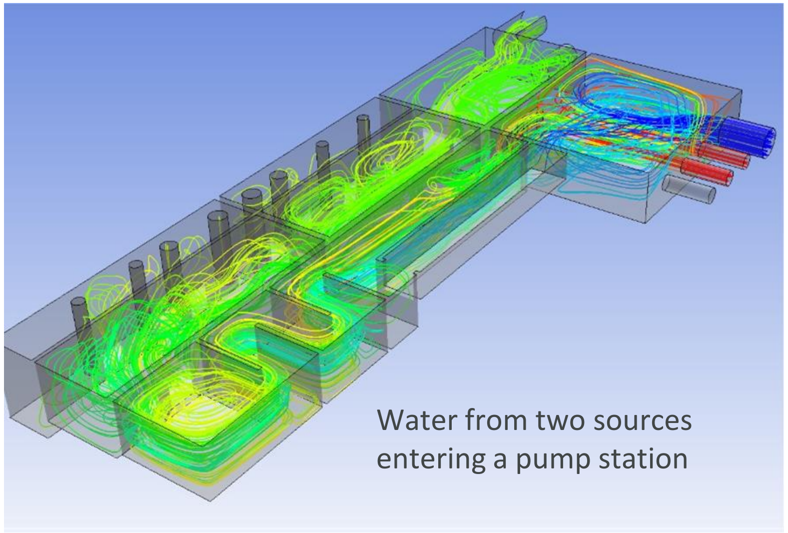
HOW TO SIMULATE MIXING IN WATER TREATMENT WITH CFD AND THE MIXING EFFECTS ON BIOLOGICAL ACTIVITY IN SECONDARY TREATMENT



Steve Saunders
Ibis Group

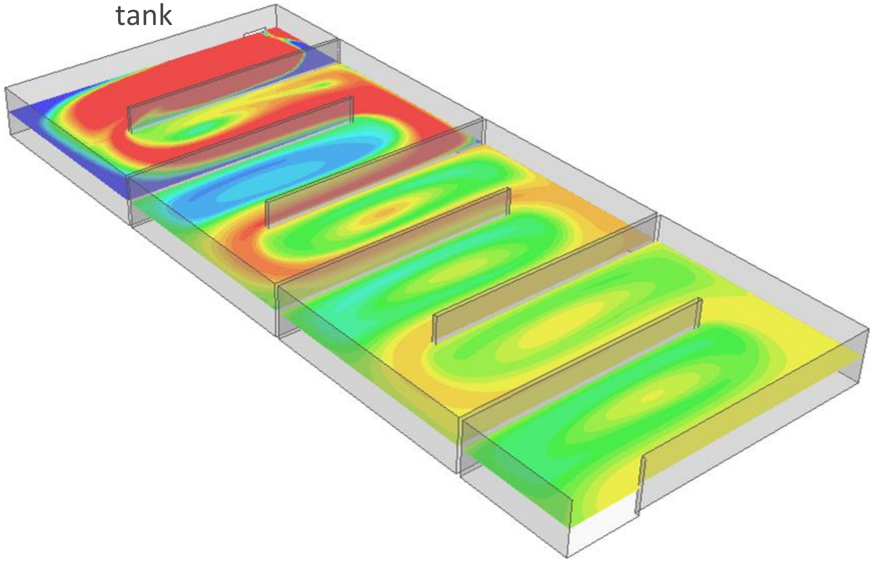
Giacomo Bellandi
AM-TEAM Group

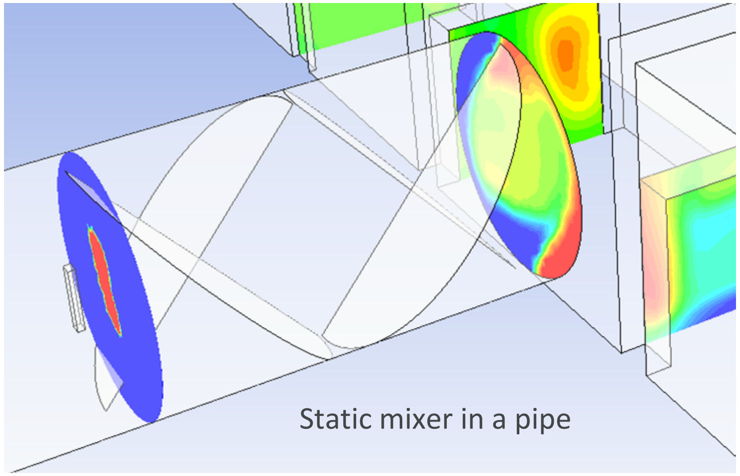
PASSIVE MIXING



Water from two sources entering a pump station

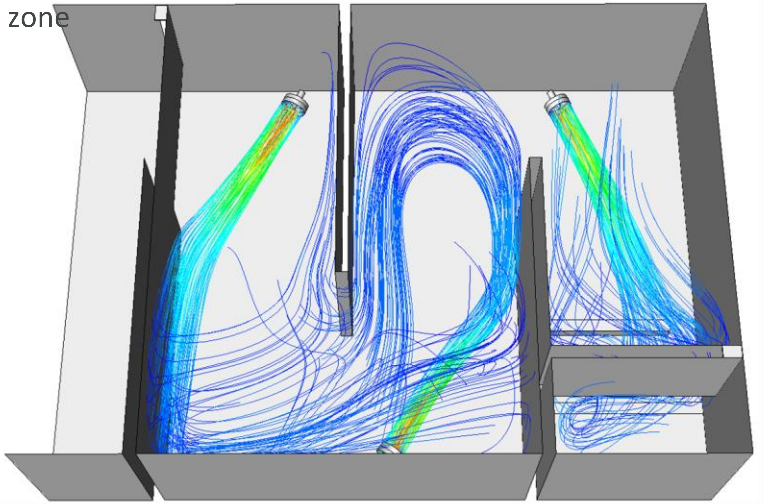
Disinfectant introduced at the head of a contact tank

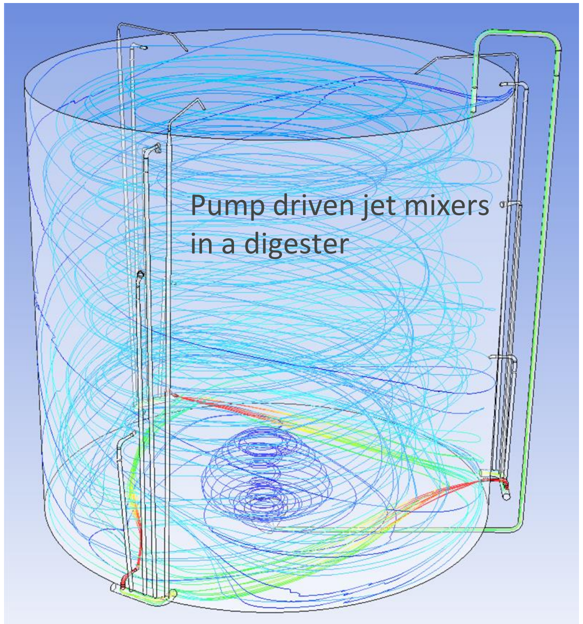


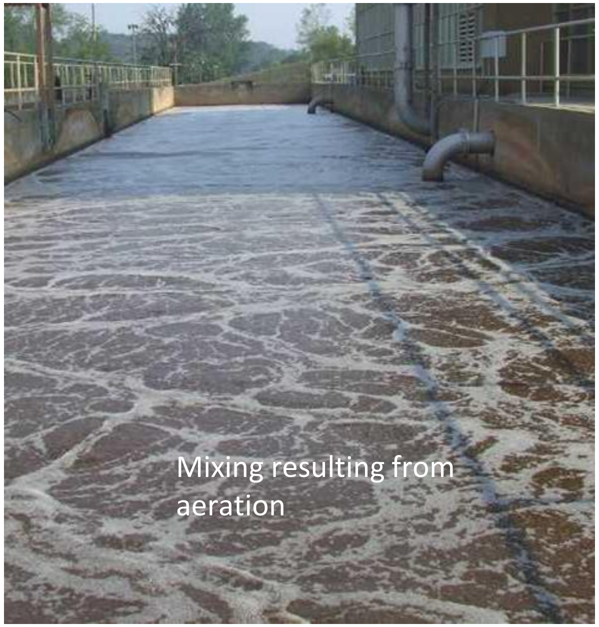


Static mixer in a pipe

MACHINE DRIVEN MIXING



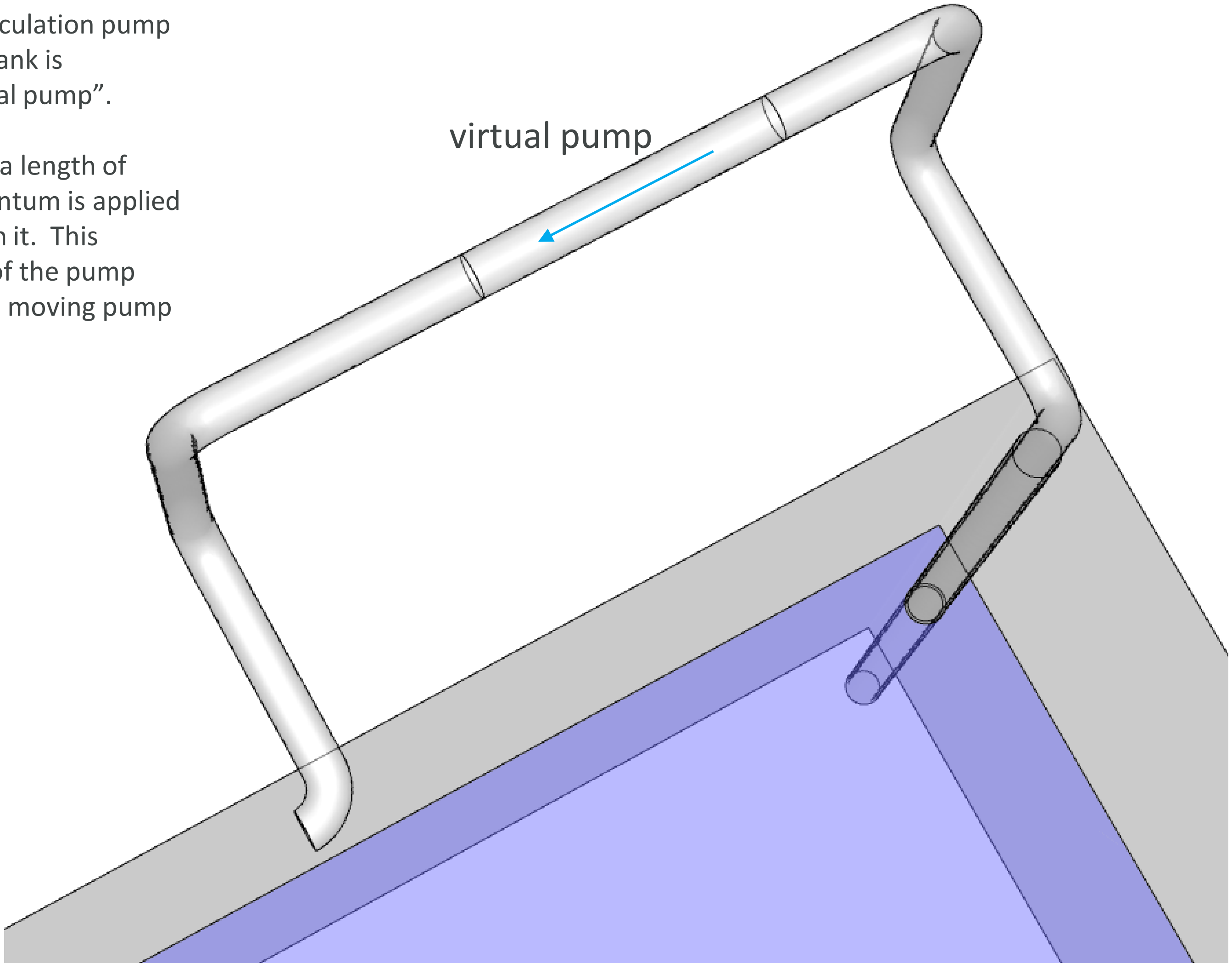




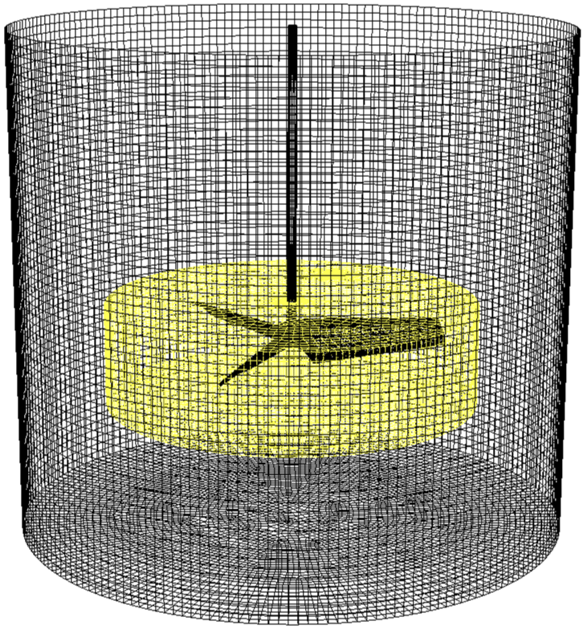
Pump driven Mixing

Water drawn up by the recirculation pump and discharged back into a tank is simulated by way of a “virtual pump”.

The virtual pump comprises a length of pipe wherein a linear momentum is applied to the water passing through it. This replicates the energy input of the pump without having to model the moving pump parts.



Propeller Mixer Input

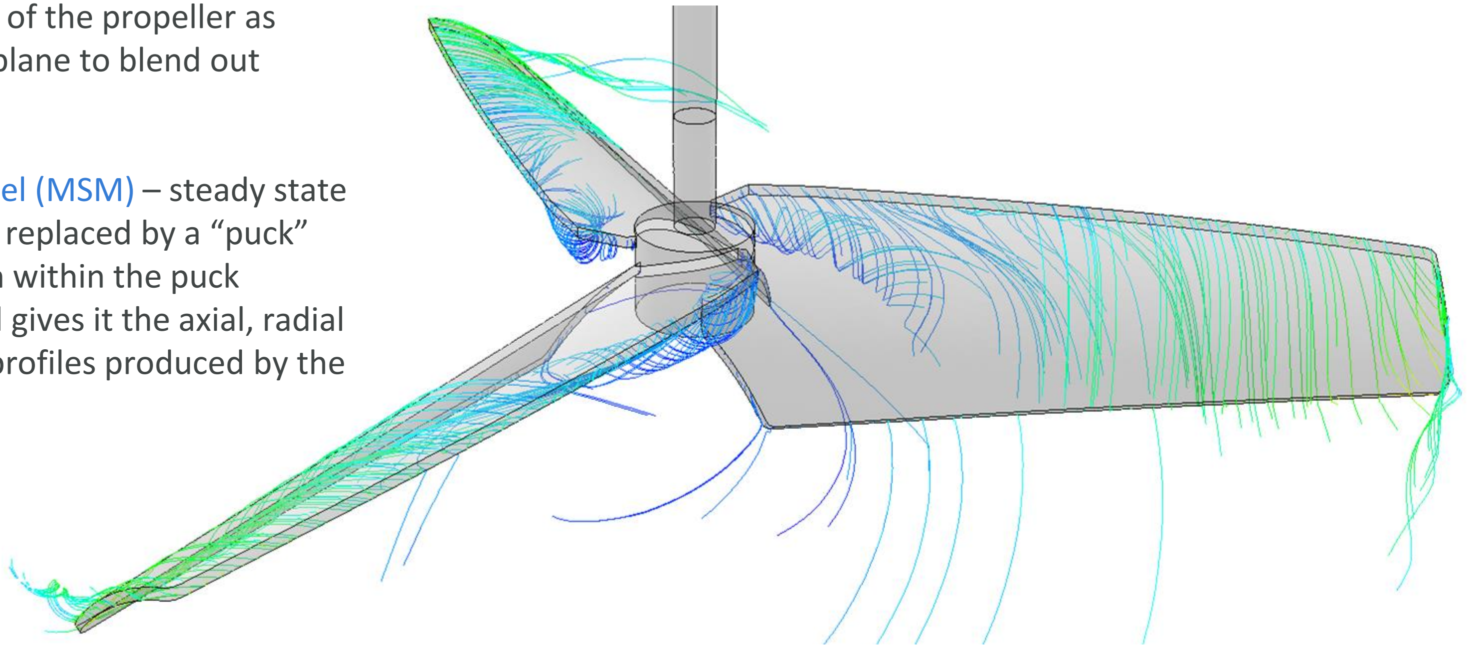


Sliding mesh (SM) – transient model with an accurate representation of the propeller rotating incrementally with each computational time step

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.



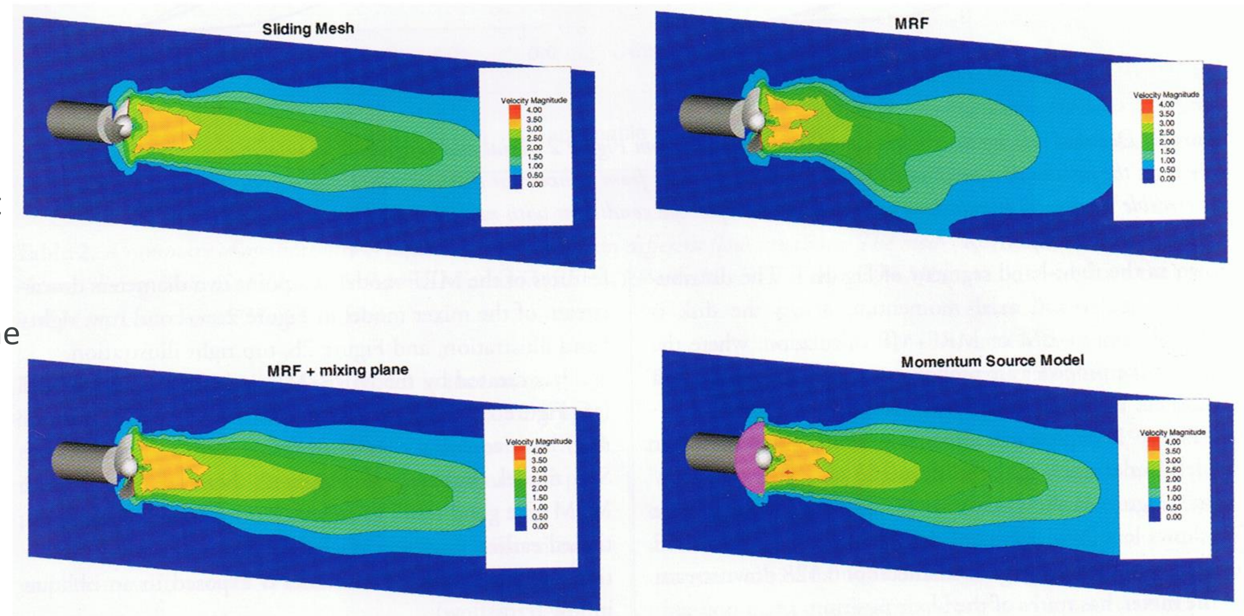
Propeller Mixer Input

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.

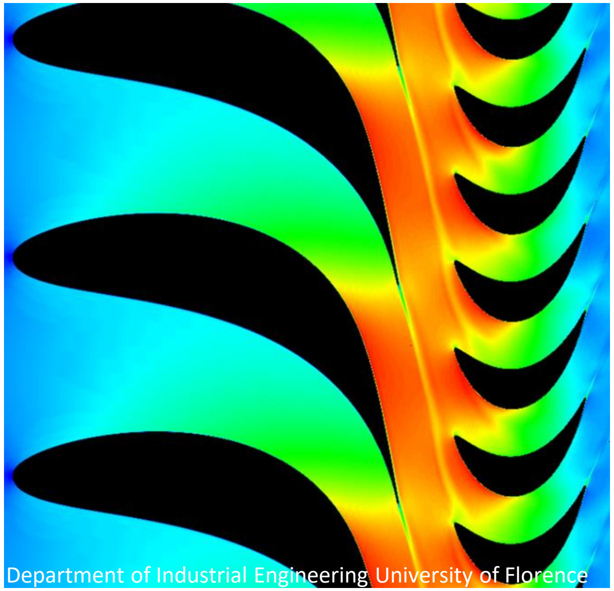
Momentum source model (MSM) – good reproduction of the mixer jet and computationally efficient. The downside is the user must obtain velocity profile data.

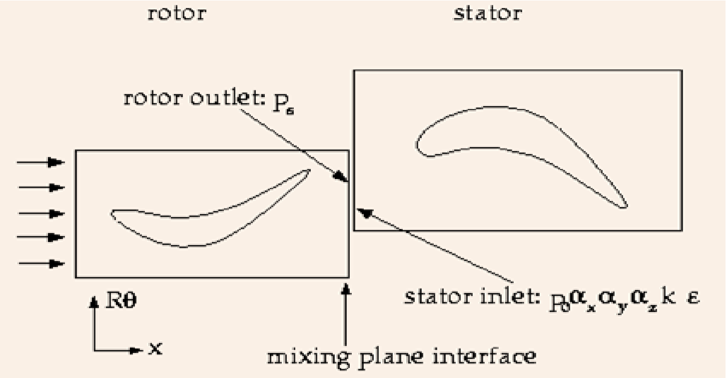


Börgesson, T. Fahlgren, M. *Mechanics of Fluids in Mixing*, 2001

Propeller Mixer Input

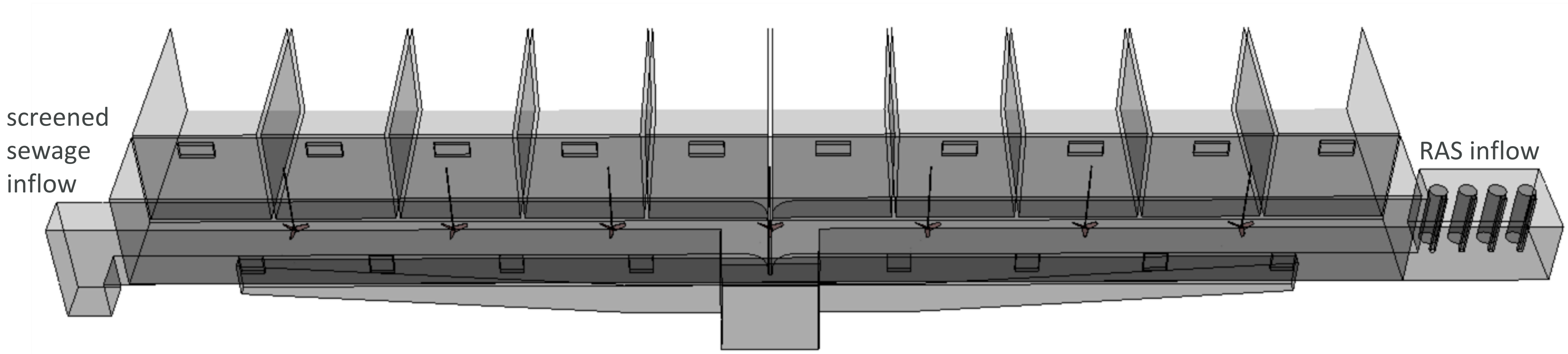
- What is a mixing plane?
  - Developed by turbo machinery modelers who wanted to analyze individual stator/rotor passages.
  - Blends out wakes of individual blades by circumferentially averaging flow properties.
  - Facilitates the use of small steady state models for rapid optimization of blade shapes





Excerpt from Ansys Inc. Fluent User Manual

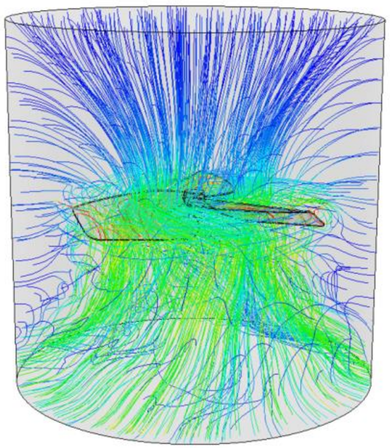
Propeller Mixer Input
Example



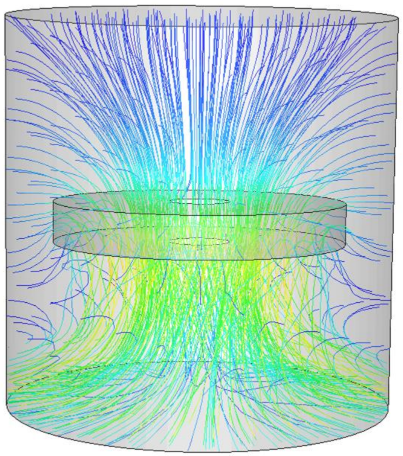
- Anoxic zone feeding 10 parallel aeration lanes
- 7 low speed top entry mixers
- Engineer wants
  - Flow splits on aeration lanes
  - Ratio of RAS to screened sewage flow entering each aeration lane
  - …next week

Propeller Mixer input

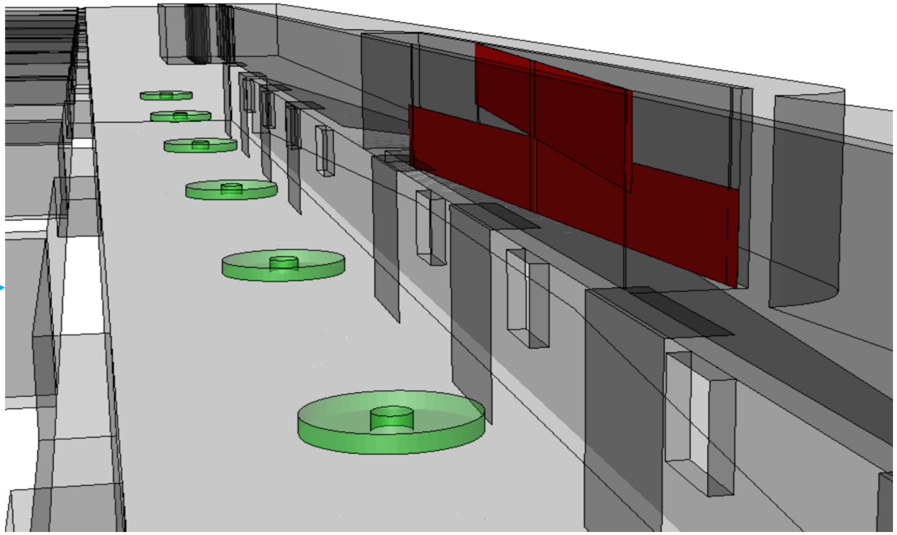
Use a Sub-Model



Sliding mesh sub-model of rotating impeller



Momentum source sub-model



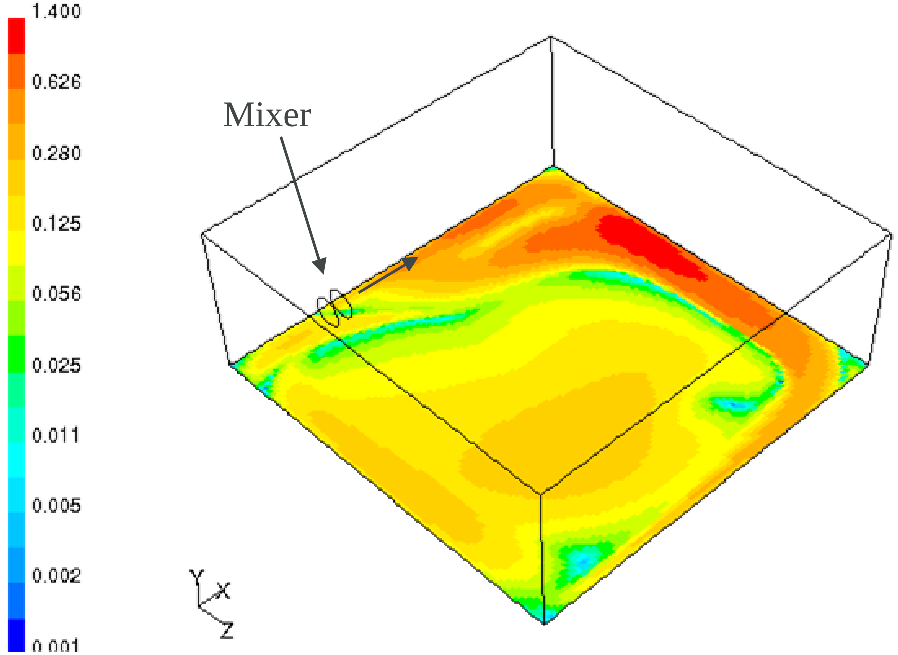
rotating impellers replicated by momentum sources

Presenting the Results

Off-Bottom Suspensioning

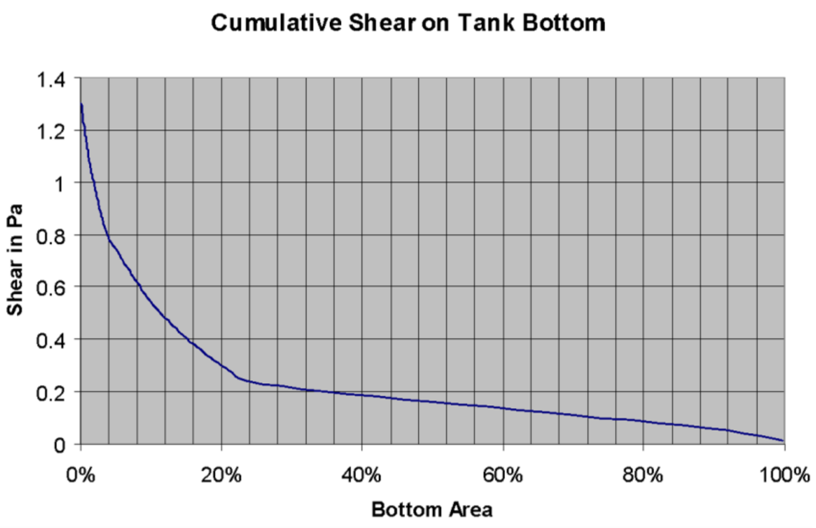
Qualitative Representation

Contour Map of Bottom Shear



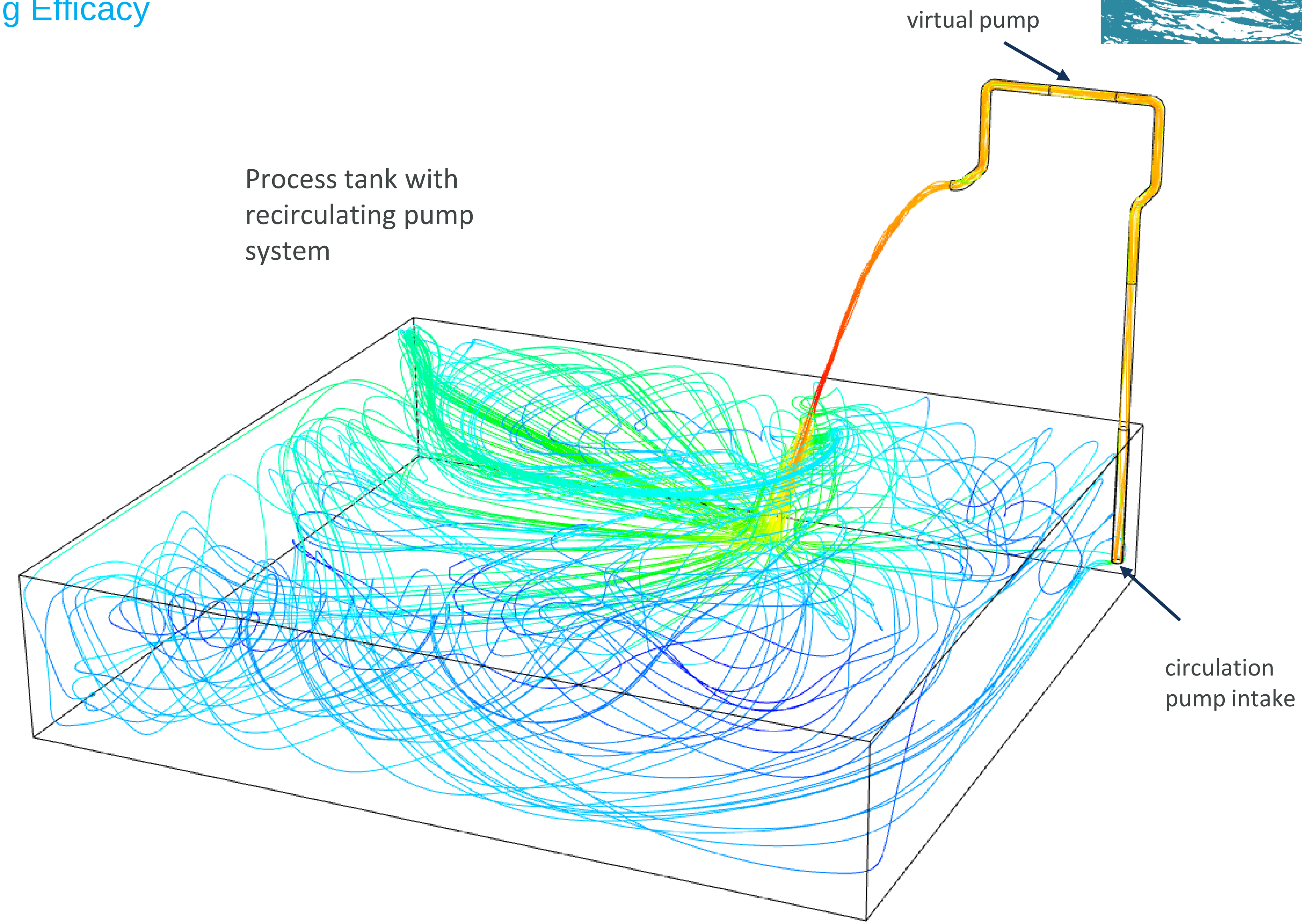
Fluid shear (Pa)

Quantitative Representation

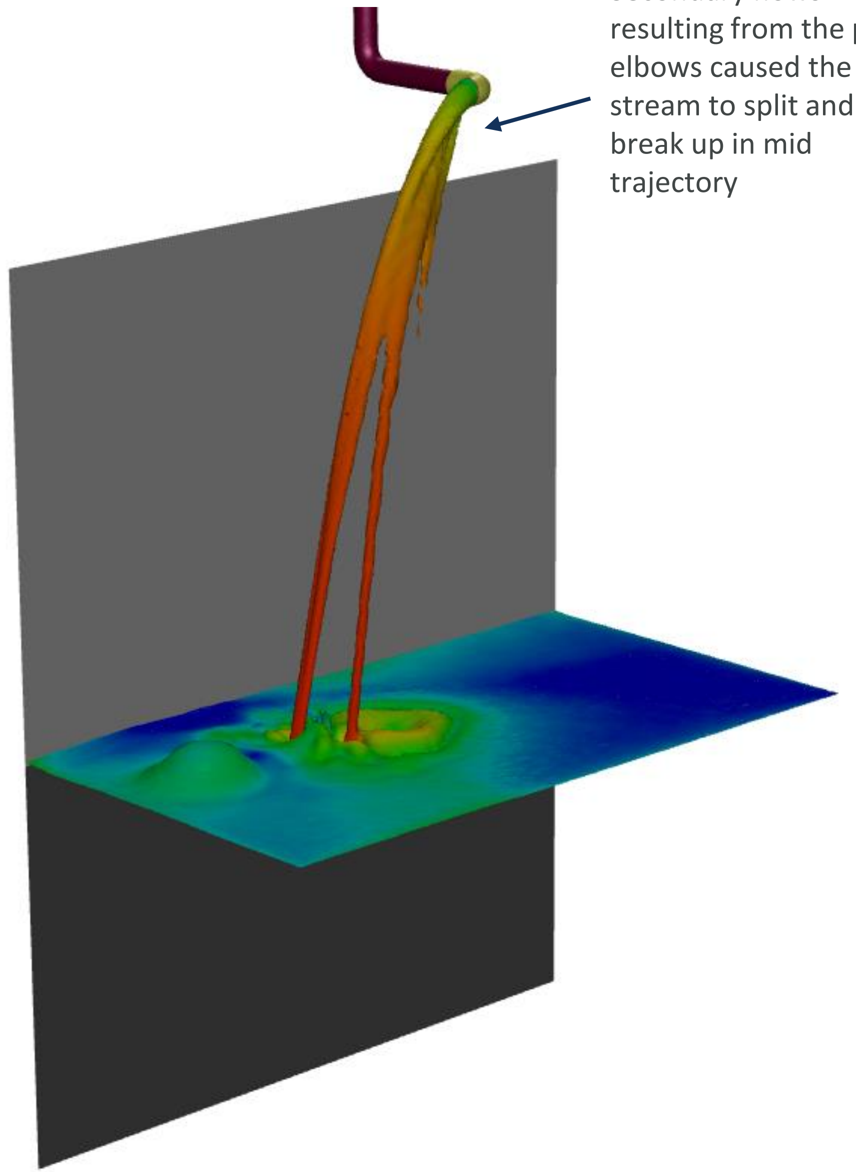


A CFD model provides a database for the flow activity within the volume being studied. Here, in the interest of off-bottom solid suspensioning, both qualitative and statistical representations of bottom shear are shown

Presenting the Results
Mixing Efficacy



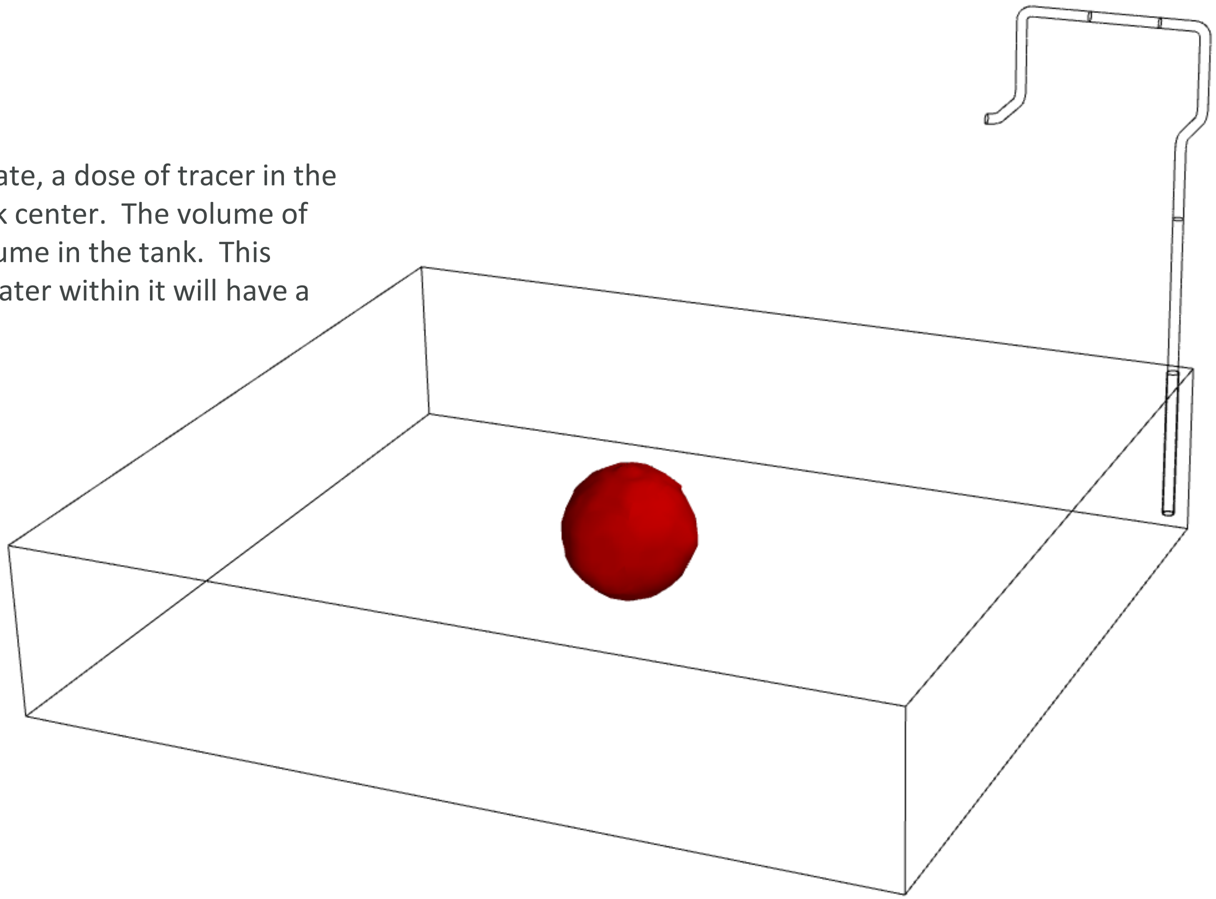
Presenting the Results
Mixing Efficacy



Secondary flows resulting from the pipe elbows caused the stream to split and break up in mid trajectory

Presenting the Results
Mixing Efficacy

Once the tank volume reaches a steady state, a dose of tracer in the shape of a sphere is introduced at the tank center. The volume of tracer in the sphere is 1% of the liquid volume in the tank. This way, once the tank gets fully mixed, the water within it will have a uniform tracer concentration of 1%.

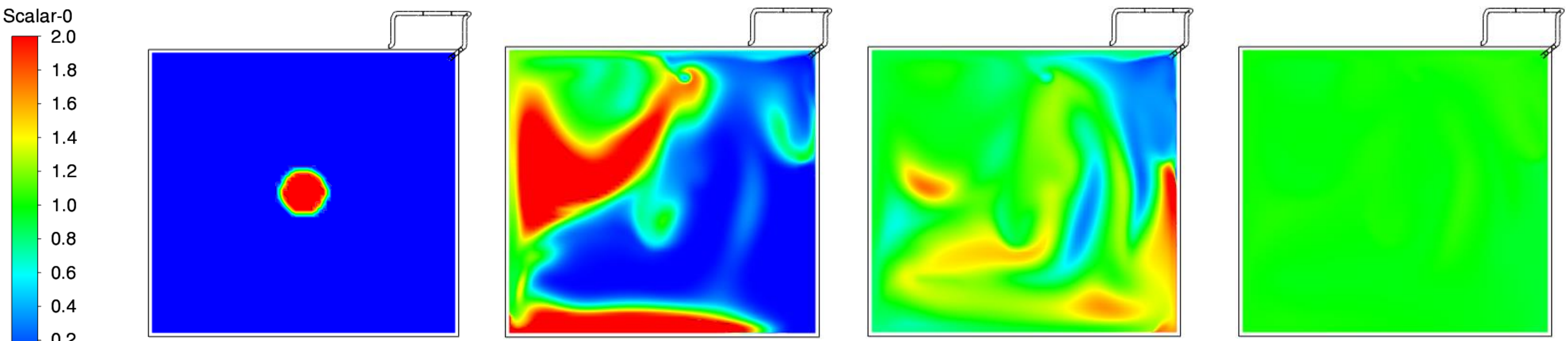


Presenting the Results

Mixing Efficacy

Qualitative

The display below shows sequential contours of tracer. These contours are on a plane cutting through the tank at mid-depth. The color scale shows tracer concentration in percent. Note that red denotes regions having concentrations of 2% or higher.



T = 0 | T = 5 min | T = 10 min | T = 20 min

concentration (%)

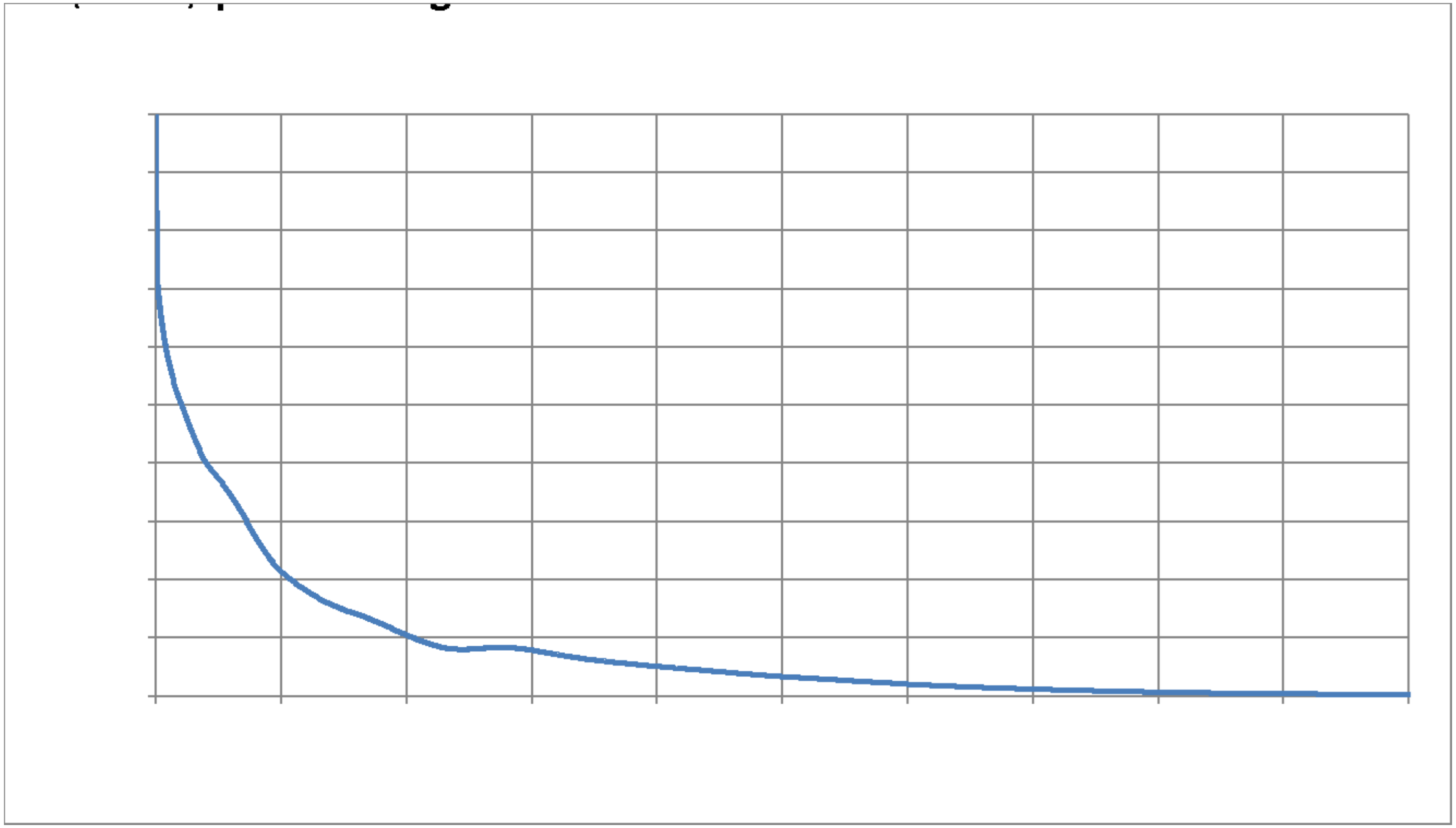
Presenting the Results

Mixing Efficacy

Quantitative

$$CoV = \frac{s}{C_{ave}}$$

Coefficient of variation (CoV) is an excellent means of applying a numerical value to the level of mixing. It is the ratio of the standard deviation of tracer concentration, s, to the average concentration, C_{ave}. It tends towards 0 as the mixture becomes homogenized.



PLANT NO. 1: DESIGN IMPROVEMENTS

N2O in liquid (mg/L)

SCENARIO
- Winter conditions
- Dry weather

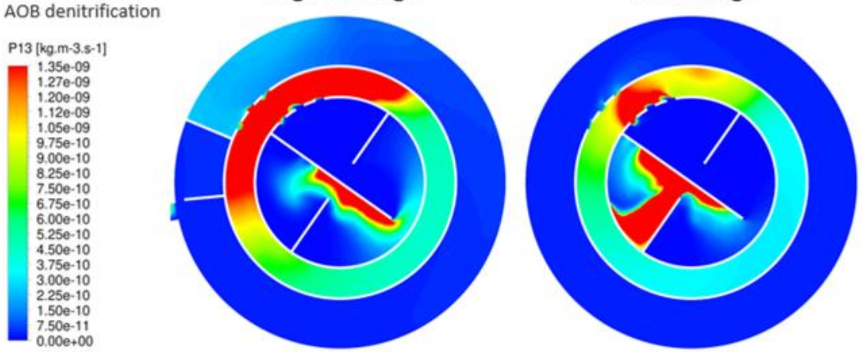
SCENARIO
- Winter conditions
- Wet weather

SCENARIO
- Summer conditions
- Dry weather

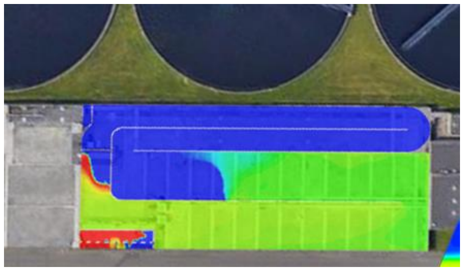
SCENARIO
- Summer conditions
- Wet weather

AOB denitrification

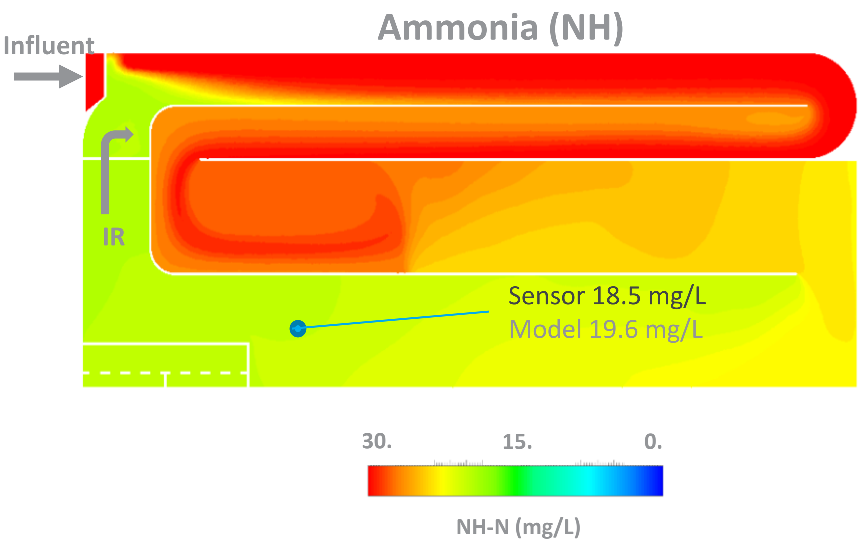
Original design **New design**

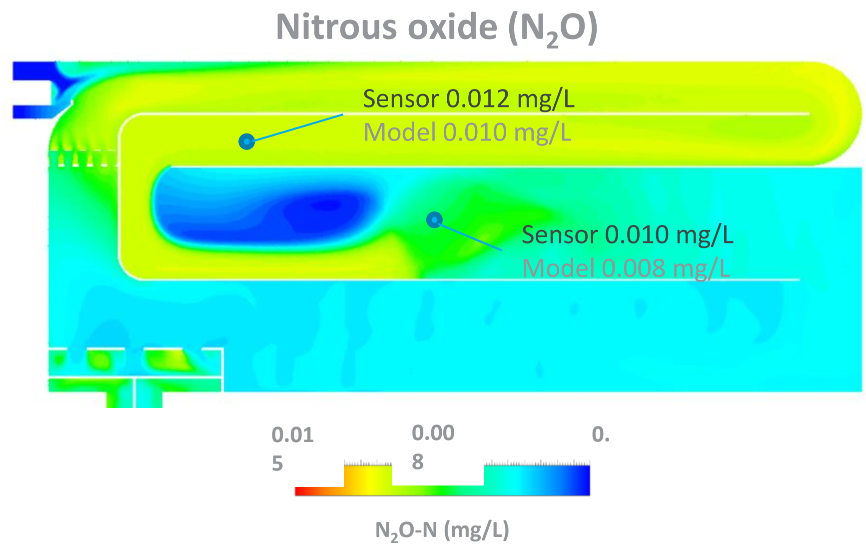


PLANT NO. 2: IMPORTANCE OF SENSOR LOCATION



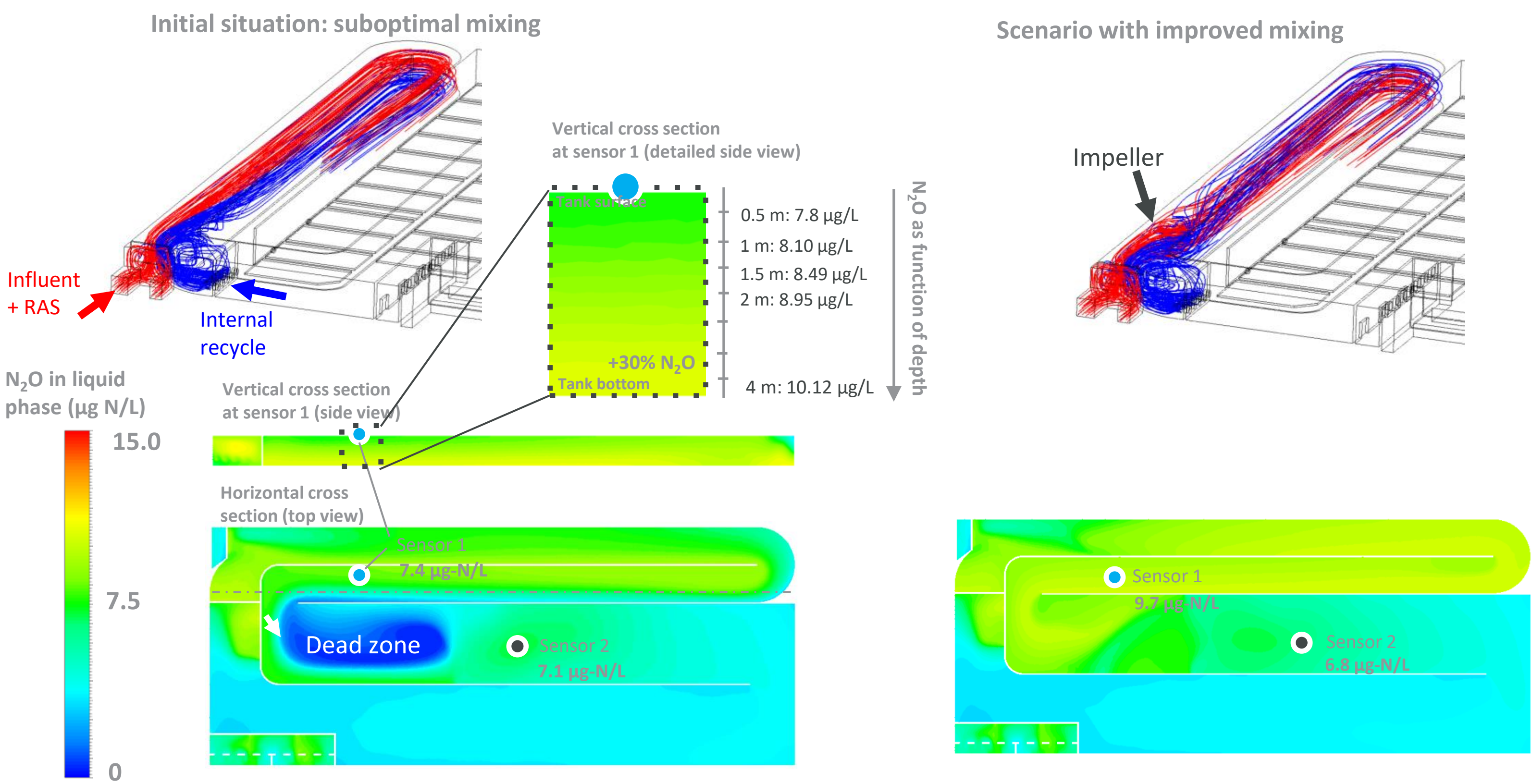
TOP VIEW OF BIOREACTOR



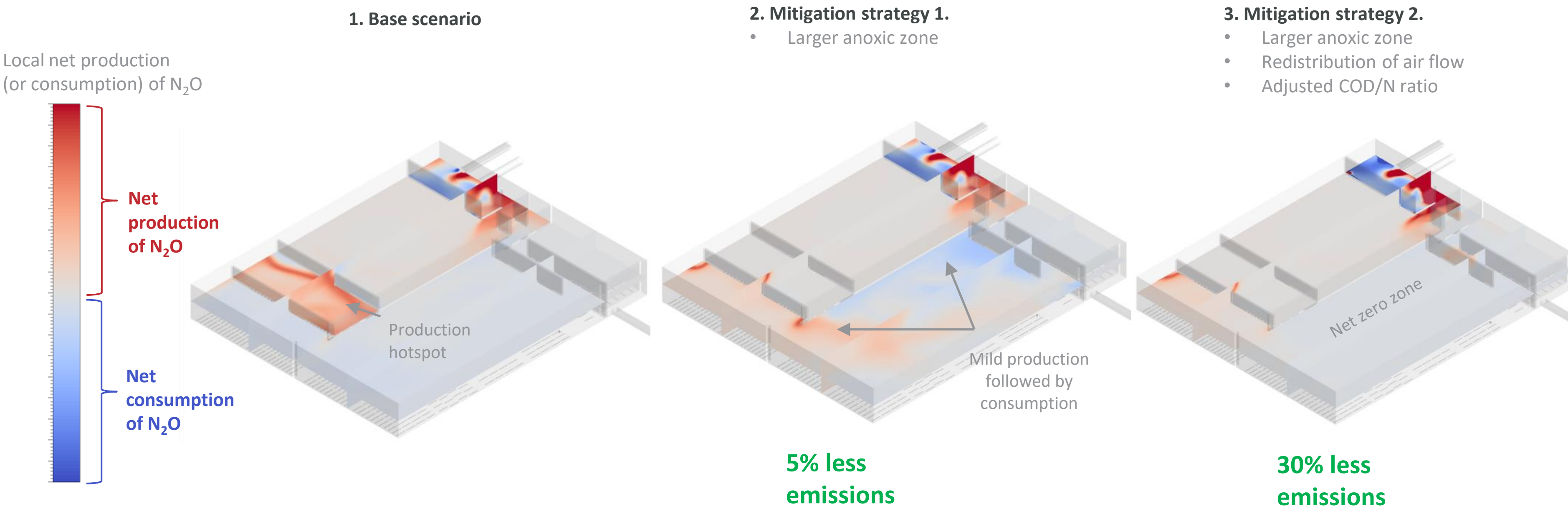


14/12/2023

PLANT NO. 2: IMPORTANCE OF SENSOR LOCATION

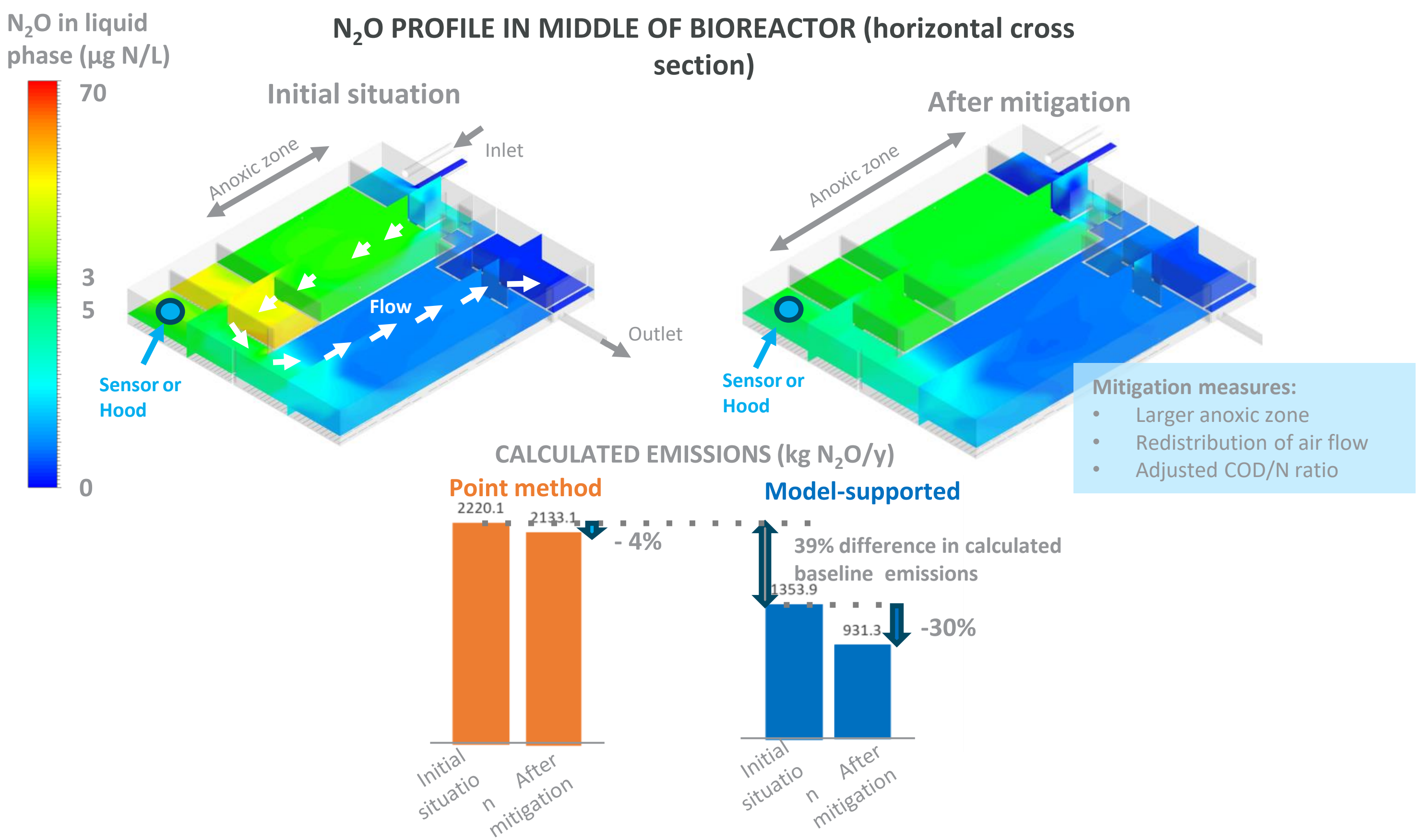


PLANT NO. 3: SCENARIO TESTING FOR MITIGATION



Watercare

PLANT NO. 3: EVALUATION OF MITIGATION STRATEGIES



AGENDA AND HOUSEKEEPING

Presentation 1
Ed Wicklein (Carollo Engineers)

Presentation 2
María Elena Valle de Medina (ENGEES/UNISTRA), Julien Laurent (ENGEES/UNISTRA), Alonso Griborio (Hazen)

Presentation 3
Christopher DeGroot (Western University, Canada)

Presentation 4
Giacomo Bellandi (AM-Team)

Presentation 5
Stephen Saunders (Ibis Group), and Giacomo Bellandi (AM-Team)

Q&A Session Moderator: *María Elena Valle de Medina (ENGEES/UNISTRA)*

CLOSING REMARKS

Great thanks to all presenters for a wonderful show!

Look out for MIA's NEXT webinar on 30th or 31st January 2024:

"Water reuse modelling?" (working title)

If you have ideas for your own future webinar then contact MIA MC and we will help you make it happen!

Find out more at

http://iwa-mia.org/

https://iwa-connect.org