Computational (Methods in) Hydrodynamics (CIVL 471)

Motivation and Introduction

Fall 2024

Shooka Karimpour, Ph.D., P.Eng.

Associate Professor

Civil Engineering - York University

Visiting Professor at EPFL

What is Computational Hydrodynamics?

- **Computational Fluid Dynamics, *aka* CFD,** is the analysis of systems involving fluid flow, heat transfer, and associated phenomena such as movement of chemical and physical matters.
- **Computational Hydrodynamics**, in particular deals with the study of liquids in motion. Hydro is from Greek hudōr for ‘water’.
- This is of particular interest for Engineers in areas of Civil and Mechanical Engineering.
- The area can expend to simulate the transport of contaminants and sediments, very complex multi-phase flows.

Introduce yourself. And tell us:

- **What fascinates you about fluid mechanics?**
- **What do you aim to achieve (or what problem you like to solve) when you learn CFD?**

Why CFD and Computational Hydrodynamics?

- **Analytical methods:**

Limited in scope to simple geometries, simplified models, asymptotic behavior... They are extremely useful, but not for global system behavior

- **Experimental methods:**

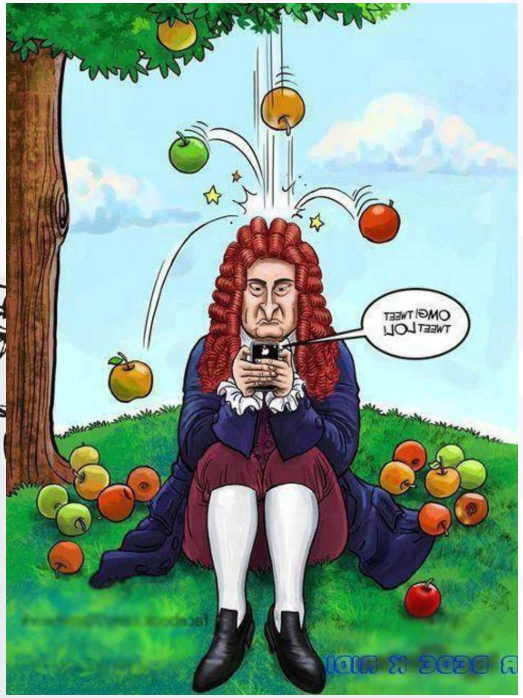
The most realistic, but can be expensive, and sometimes dangerous and/or impossible

- **Computational methods:**

Enable virtual experimentation based on first principles but also suffer from numerical errors and other formulation problems - however, they can provide unbelievable insight into a complex system (time and space)

Why CFD and Computational Hydrodynamics?

- As scientists we **observe the world around us, make hypothesis, and build experiments** to conduct in depth analysis.
- Think of Isaac Newton and Gravity!!



Why CFD and Computational Hydrodynamics?

- BUT!!!!!
- Some phenomena are very impractical and quite impossible to observe:

  Nuclear Explosion!! *(One never wants to observe in real-life)*

  Combustion engines *(How to come up w*

  Hurricanes and other weather events

  limitation)

  Flood and erosion potentials *(Inaccurate*

  Contamination of in a lake *(Field based*

LASSONDE SCHOOL OF ENGINEERING

YORK UNIVERSITÉ UNIVERSITY

EPFL

Why CFD and Computational Hydrody

Satellite image of a 2021 hurricane in the Gulf of Mexico

- BUT!!!!!
- Some phenomena are very impractical and quite imposs

Nuclear Explosion!! *(One never wants to observe*

Combustion engines *(How to come up w initial design)*

Hurricanes and other weather events *(Impossible to generate in the lab due to scale limitation)*

Flood and erosion potentials *(Inaccurate in lab conditions – again due to scaling)*

Contamination of in a lake *(Field based observations are scattered – experiment!)*

Why CFD and Computational Hydrodynamics?

- BUT!!!!!
- Some phenomena are very impractical and quite impossible to

  Nuclear Explosion!! *(One never wants to observe in rea*

  Combustion engines *(How to come up w initial design)*

  Hurricanes and other weather events *(Impossible to*
  limitation)

  Flood and erosion potentials *(Inaccurate in lab conditions – again due to scaling)*

  Contamination of in a lake *(Field based observations are scattered – experiment!)*

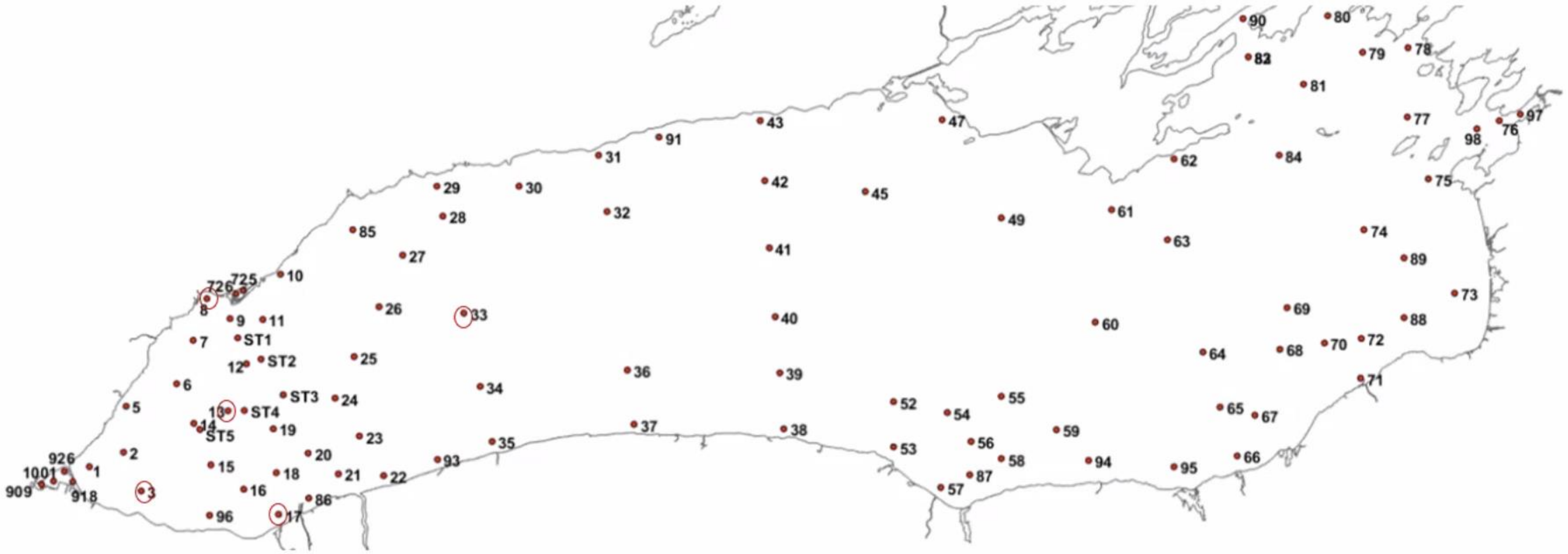
Why CFD and Computational Hydrodynamics?

LASSONDE SCHOOL OF ENGINEERING

YORK UNIVERSITÉ UNIVERSITY

Routine water quality measures in Lake Ontario only in 22 master stations.

22 master stations: 3, 8, 13, 17, 22, 31, 33, 38, 40, 41, 57, 60, 61, 71, 74, 81, 85, 90, 95, 98, 725 and 926;



EPFL

Why CFD and Computational Hydrodynamics?

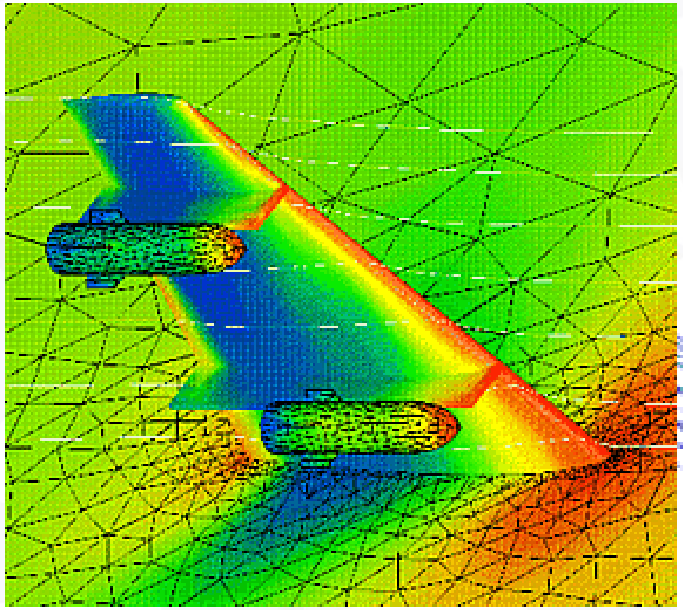
- BUT!!!!!
- Some phenomena are very impractical and quite impossible to observe:

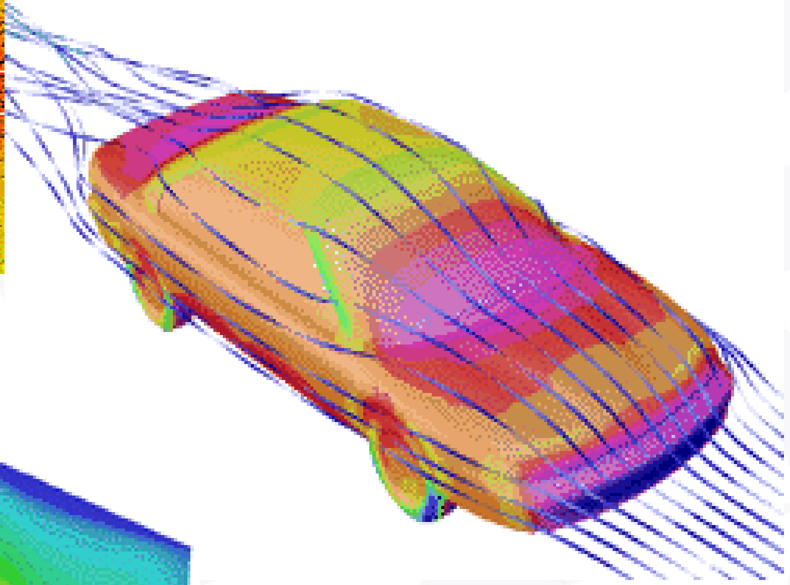
  Impact of Climate Change scenarios on Water Resources!! *(Today's most important existentialist issue!)*

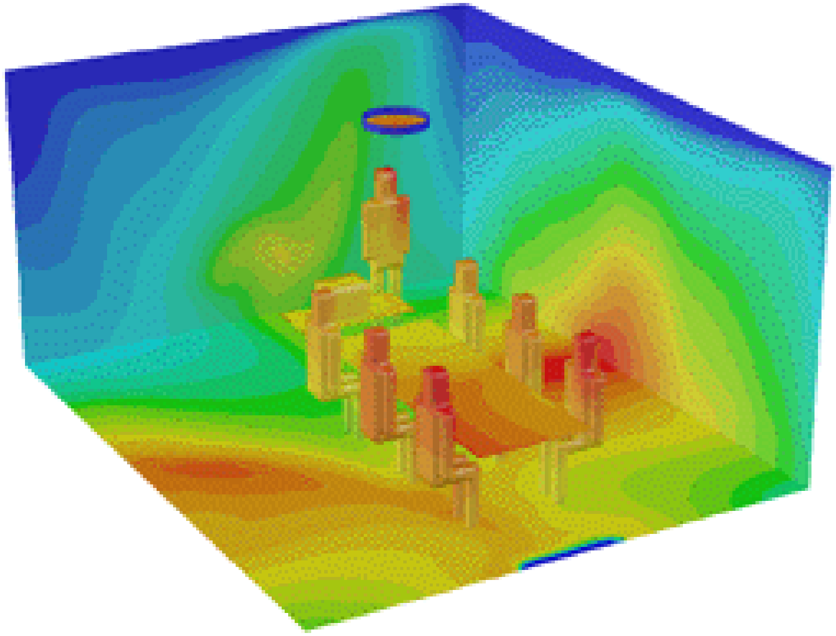
- **Simulation and computational methods (or scientific computing in general) are now an imperative pillar of the scientific method.**

Where is CFD used?

- Aerospace
- Appliances
- Automotive
- Biomedical
- Chemical Processing
- HVAC **(Quite a hot topic - COVID)**
- **Hydraulics**
- **Marine**
- Oil & Gas
- **Power Generation**
- Sports

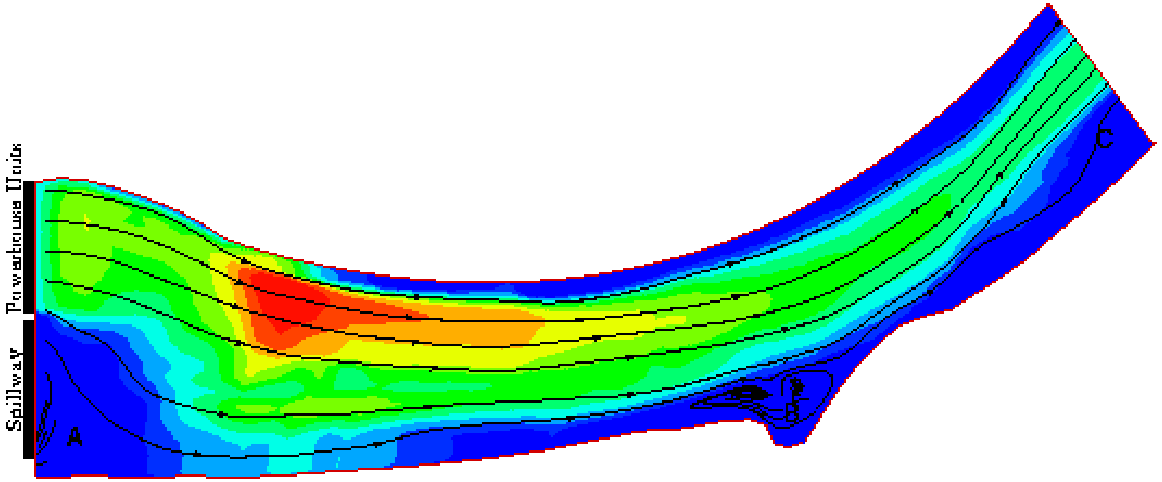






Where is CFD used?

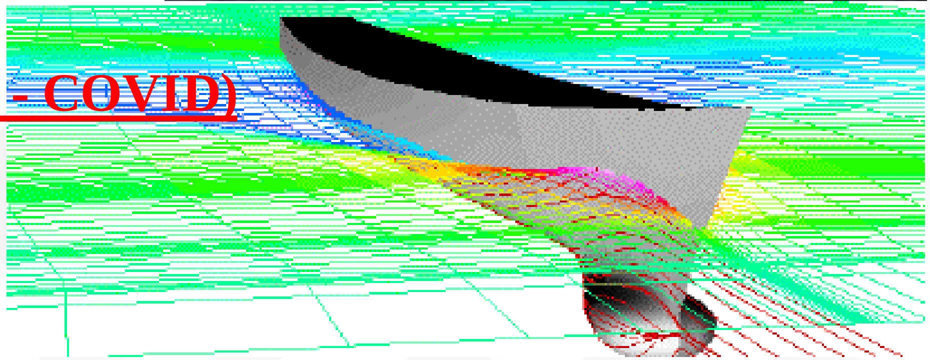
- Aerospace
- Appliances
- Automotive
- Biomedical
- Chemical Processing
- HVAC **(Quite a hot topic - COVID)**
- **Hydraulics**
- **Marine**
- Oil & Gas
- **Power Generation**
- Sports



Total Discharge = 125,000 cfs (no flow through spillway)

Where is CFD used?

- Aerospace
- Appliances
- Automotive
- Biomedical
- Chemical Processing
- HVAC **(Quite a hot topic - COVID)**
- **Hydraulics**
- **Marine**
- Oil & Gas
- **Power Generation**
- Sports

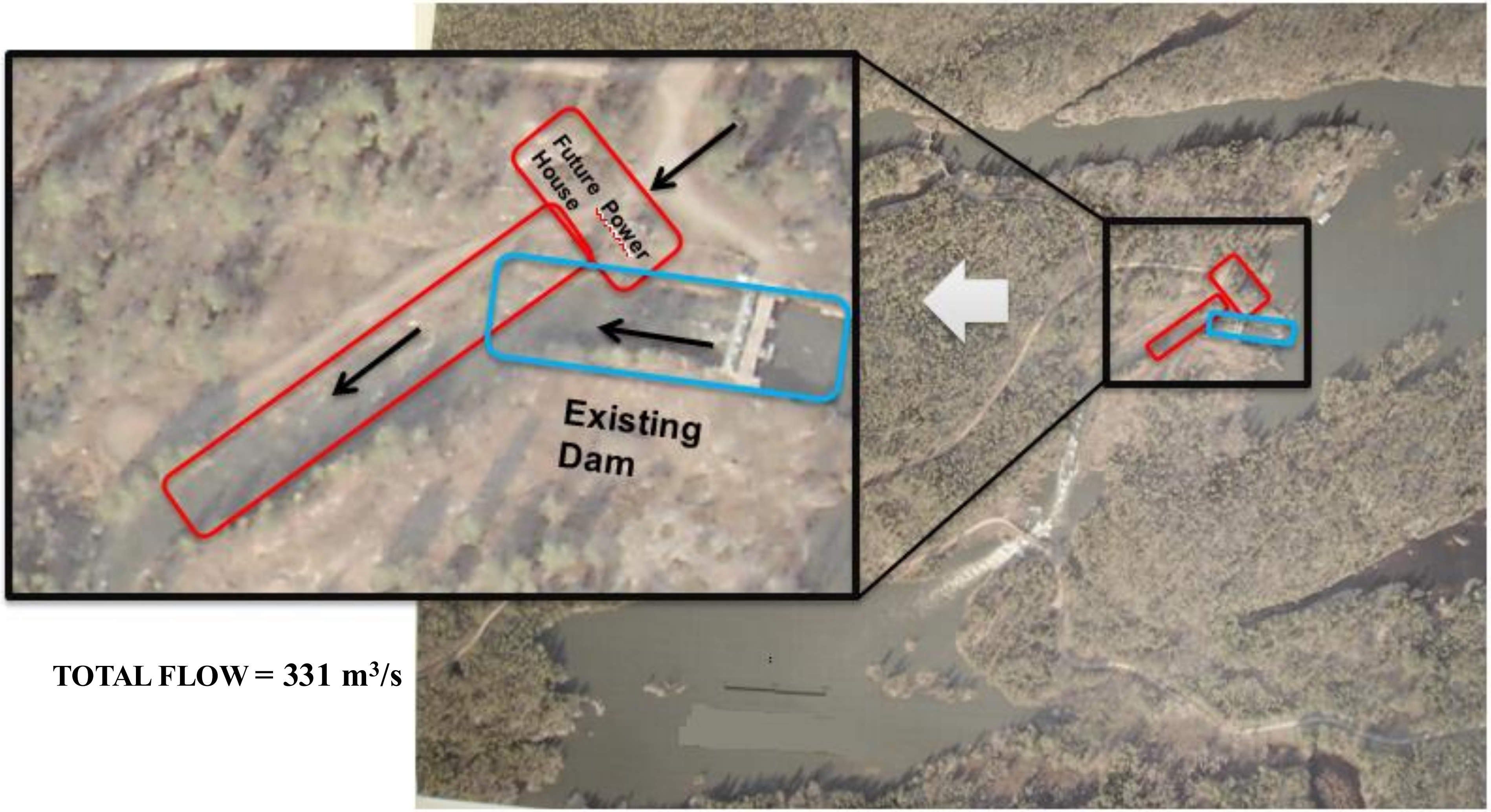


GENIVAR

(Now WSP)

LASSONDE SCHOOL OF ENGINEERING

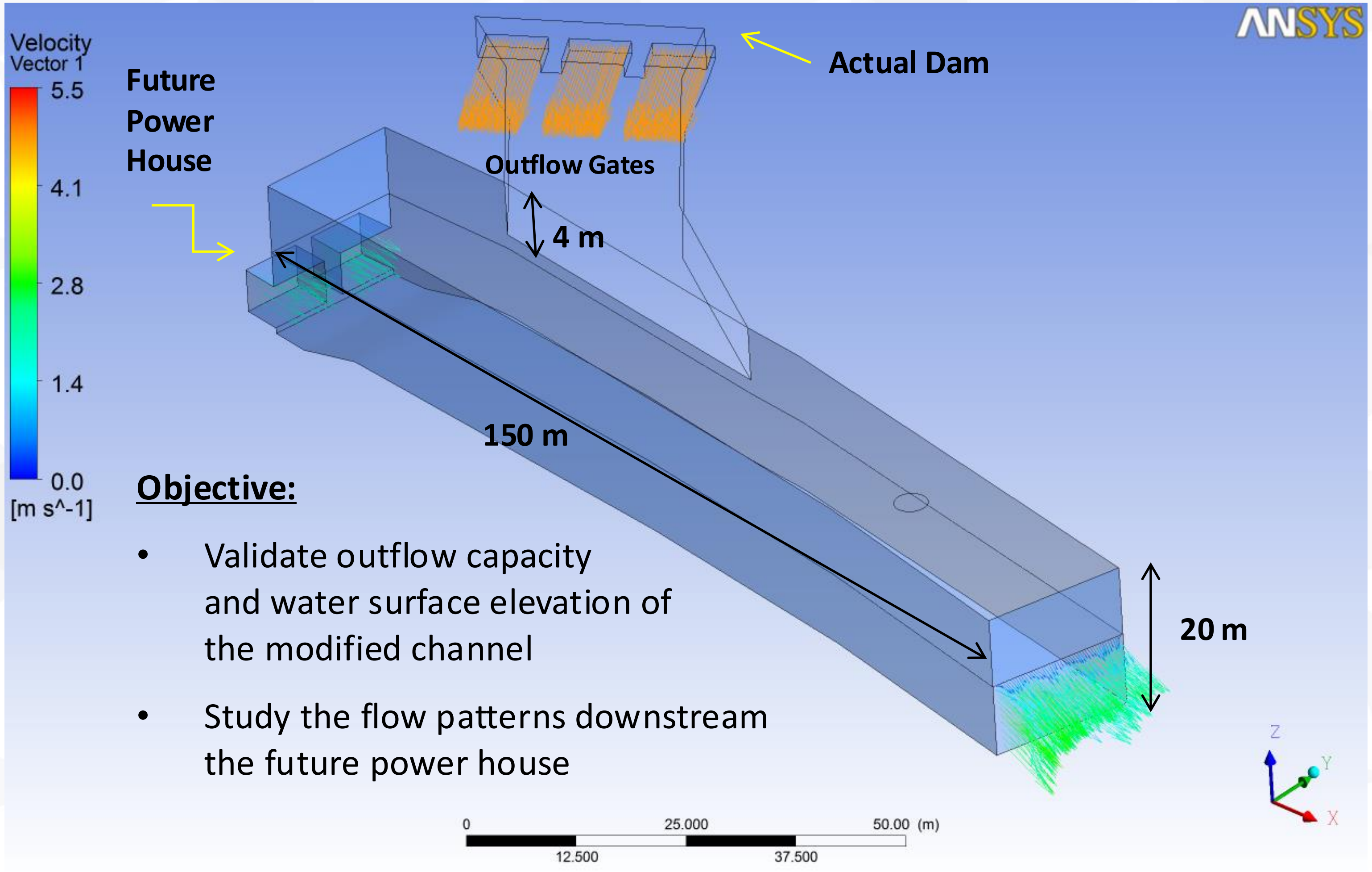
YORK UNIVERSITÉ UNIVERSITY



TOTAL FLOW = 331 m^3/s

GENIVAR

(Now WSP)

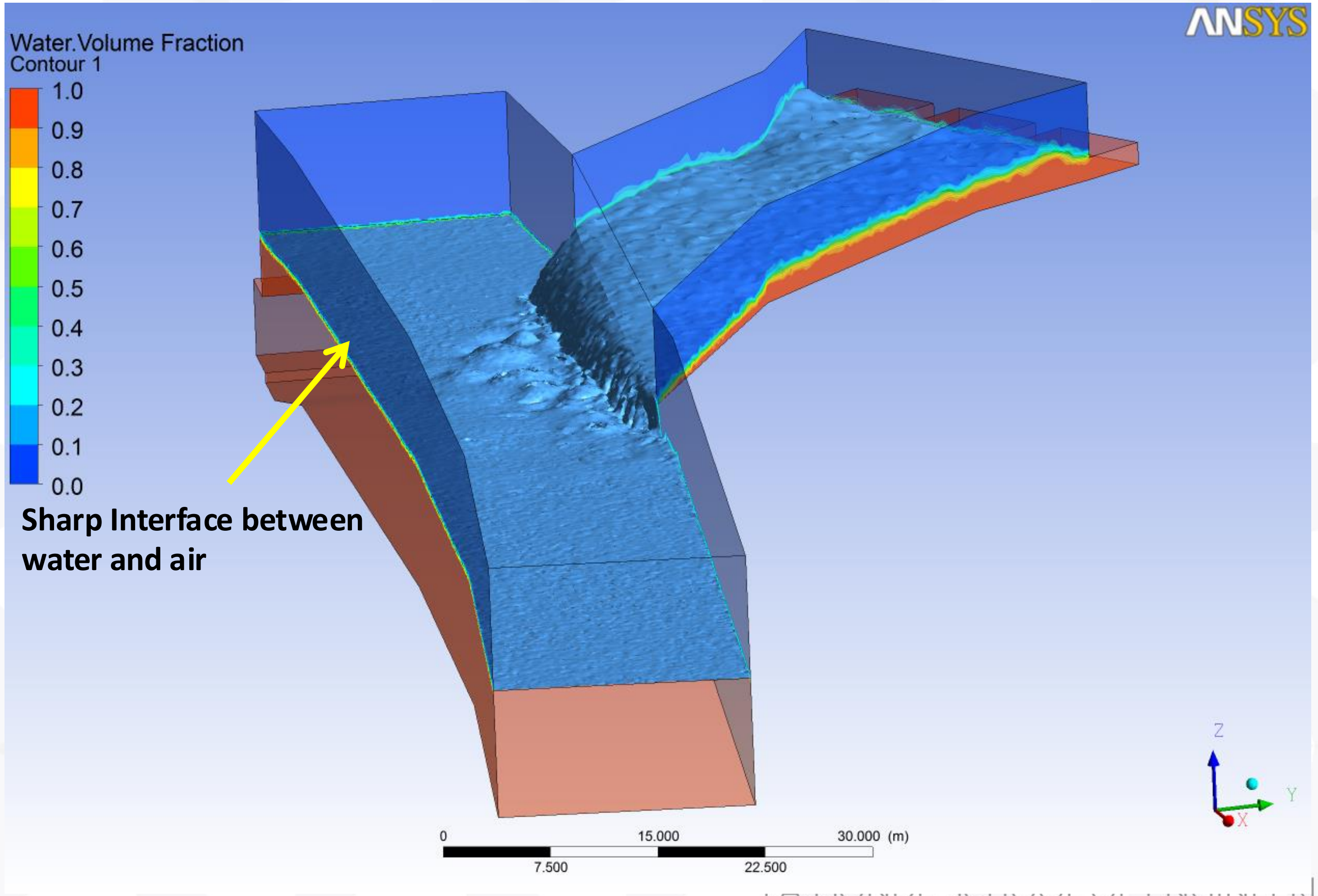


Objective:

- Validate outflow capacity and water surface elevation of the modified channel
- Study the flow patterns downstream the future power house

GENIVAR

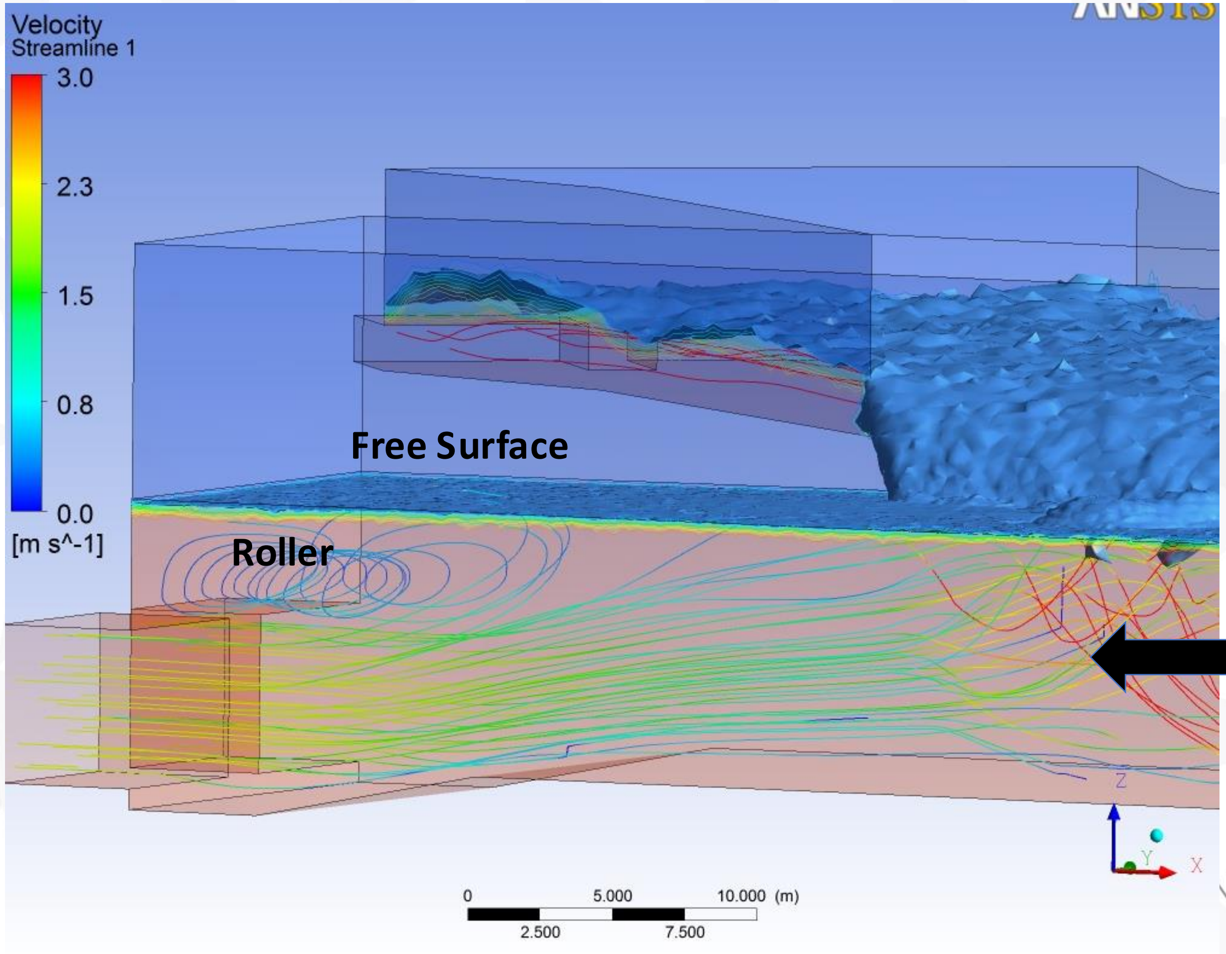
(Now WSP)

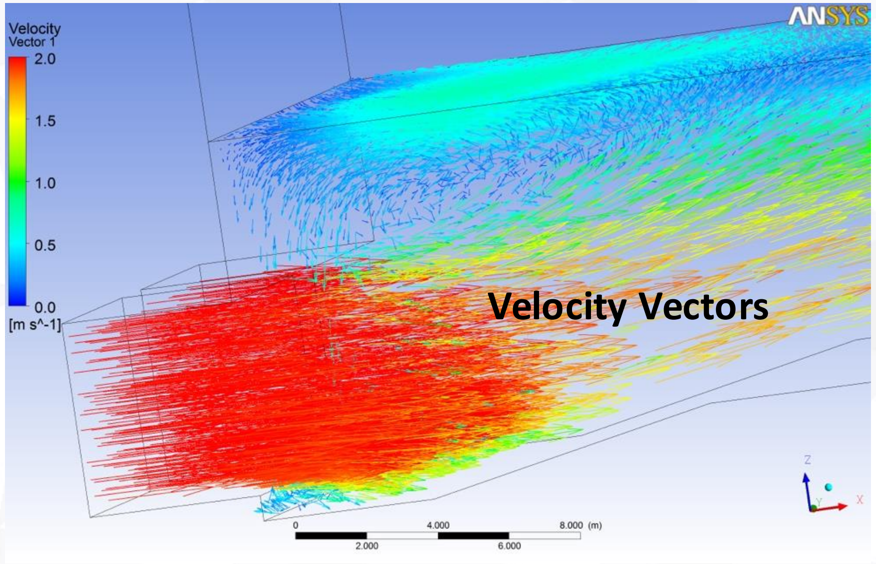


GENIVAR

(Now WSP)

Studied flow patterns (circulation, flow paths, roller, back-eddy) downstream of the Powerhouse



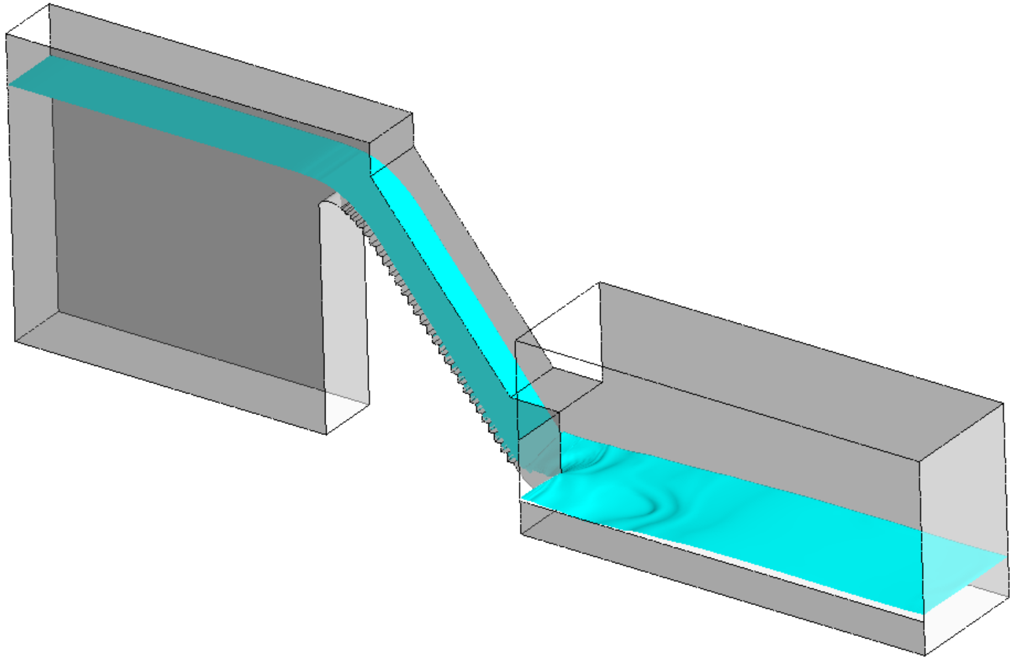


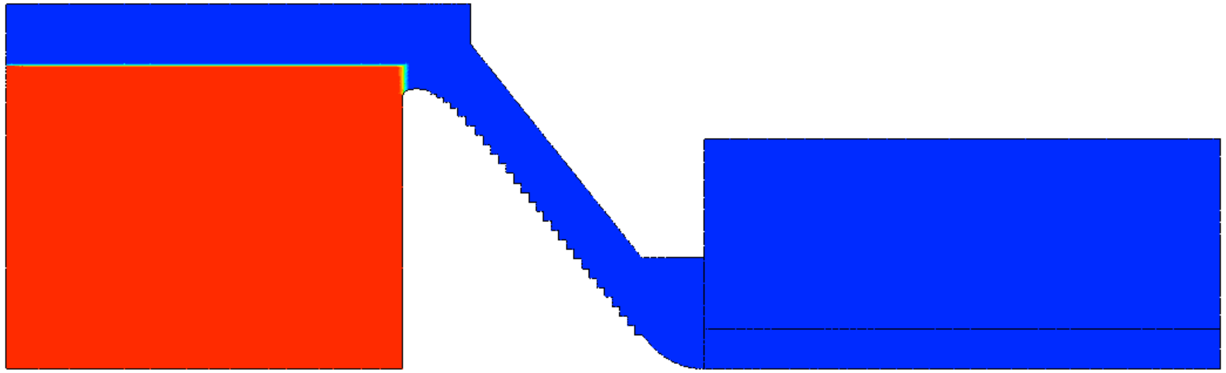
Flow paths

GENIVAR

(Now WSP)

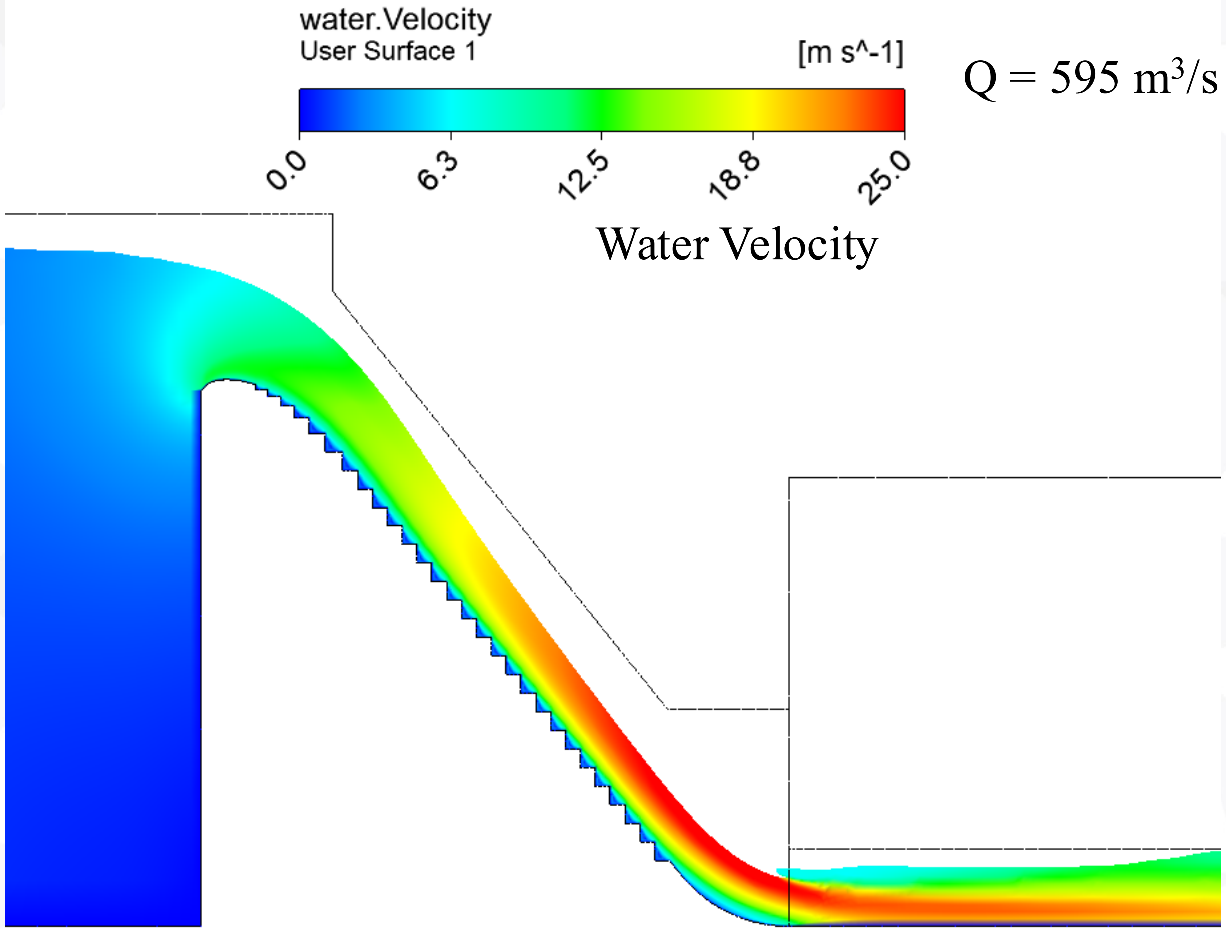
- This is a CFD simulation of a free surface flow of water down a stepped slope
- The aim is to determine free surface height, velocity and Froude number





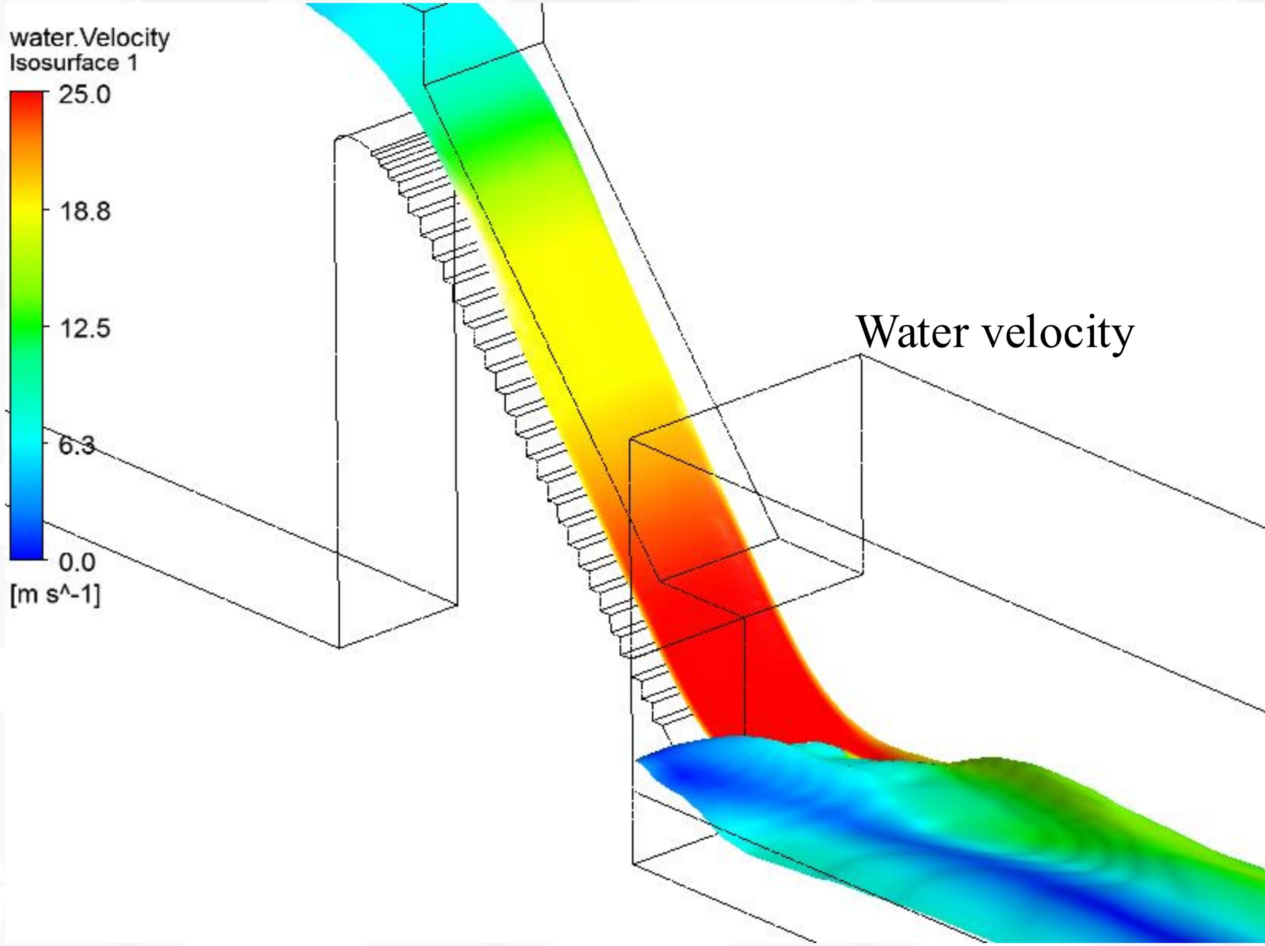
GENIVAR

(Now WSP)



$Q = 595\ m^3/s$

Water Velocity



Water velocity

Side Cavity - Groynes

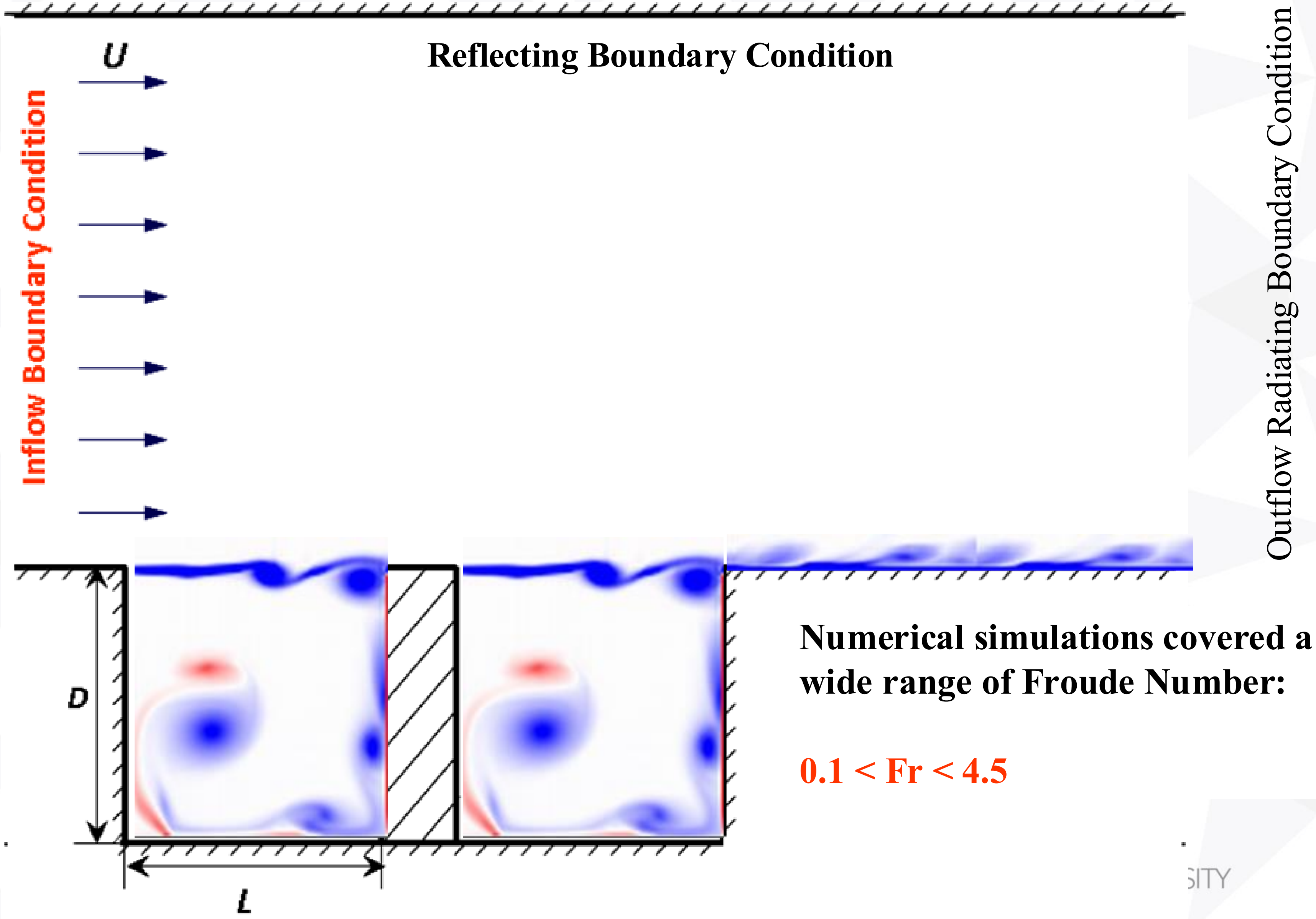
- Prevents bank erosion and ice jamming
- Generates navigable depth

- Stabilize beaches and protects the shoreline

Side Cavity – Groynes (in-house CFD)

Numerical Setup:

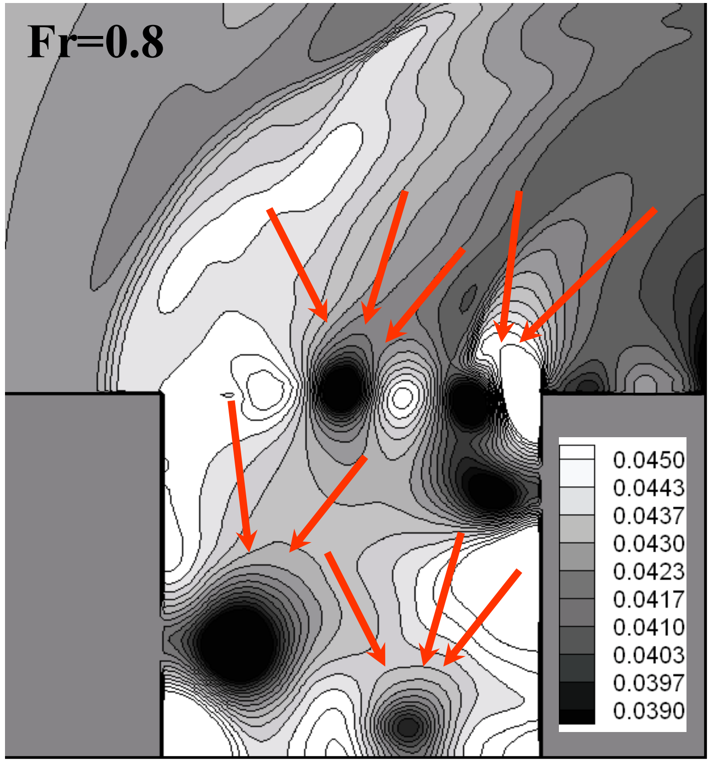
- Radiating Boundary Condition, allowing the incoming waves to scape;
- Reflecting boundary condition, simulating the opposite bank;
- Allowing uniform inflow, and adjusting the slope to balance the friction;
- Alternating the water depth to achieve wide range of Froude Numbers.



Numerical simulations covered a wide range of Froude Number:

$0.1 < Fr < 4.5$

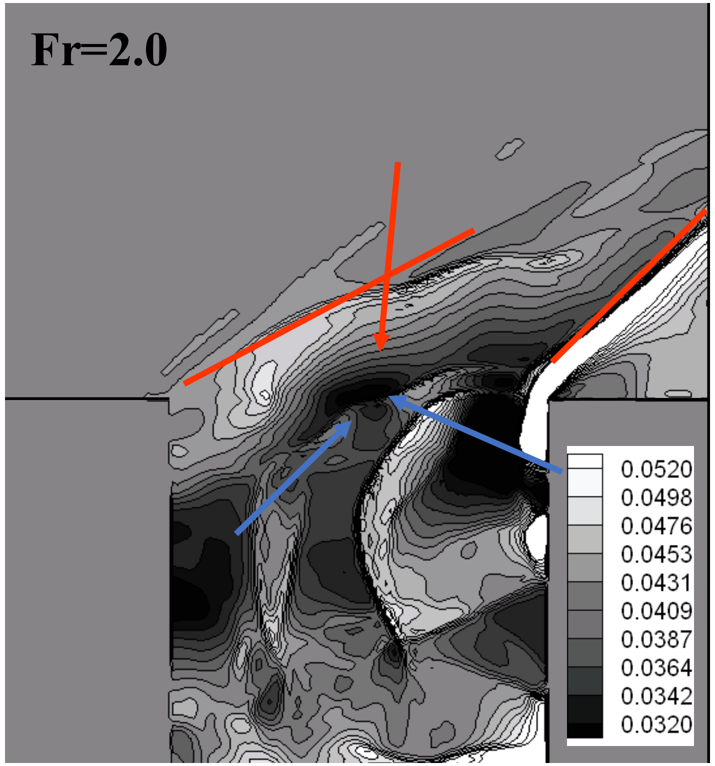
Side Cavity – Groynes (in-house CFD)



Sub-Critical

- Depth variation only occur in the center of the vortices.

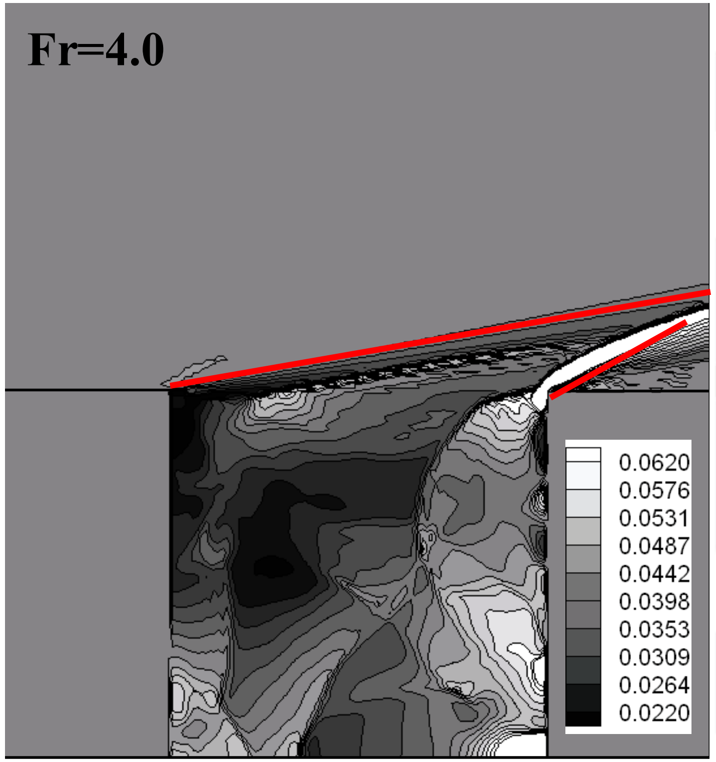
- vortex shedding is dominant.



Super-Critical

- Depth Variation dominated by wave. Anisotropic eddies accompanied by wave radiation.

- The standing shock waves do not interact with shear layer

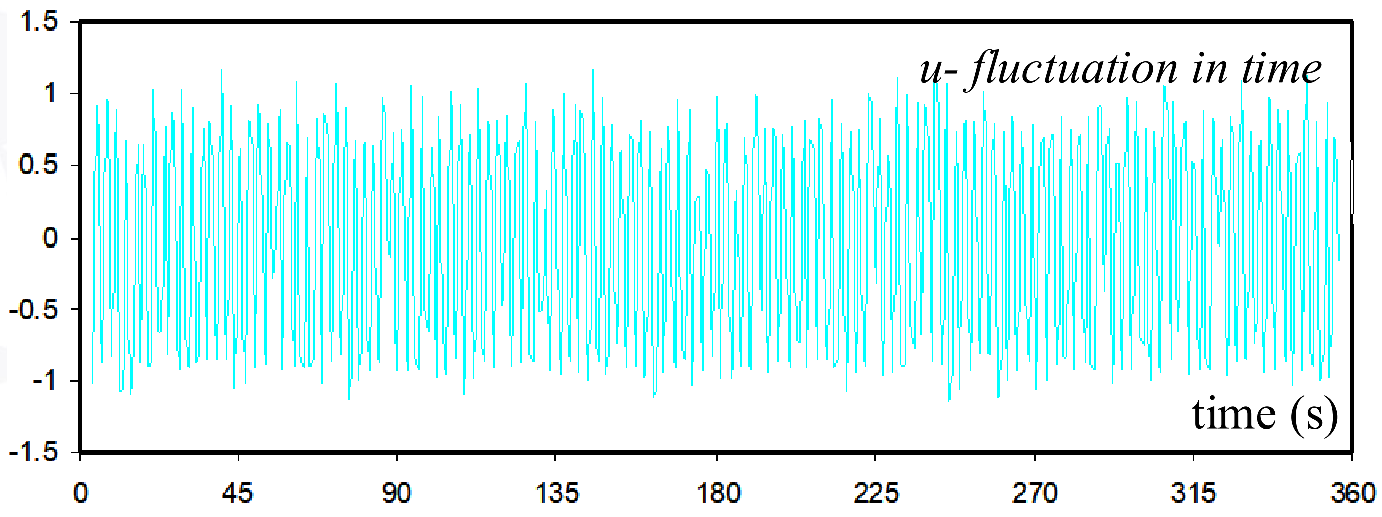


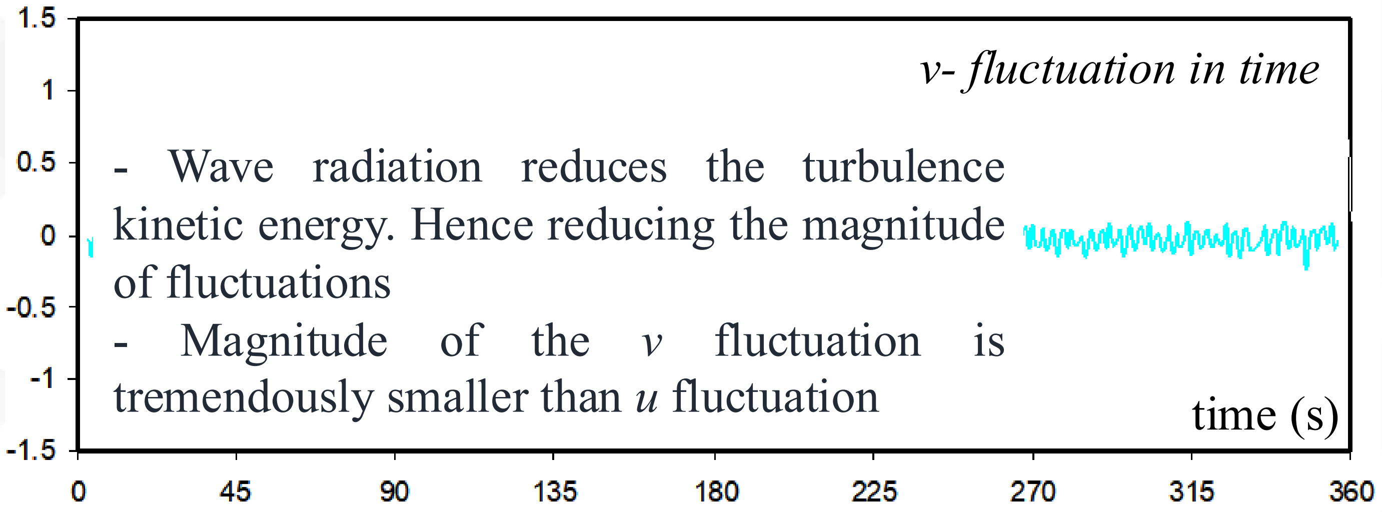
Hyper-Critical

- Depth Variation dominated by wave. Anisotropic eddies are hardly formed.

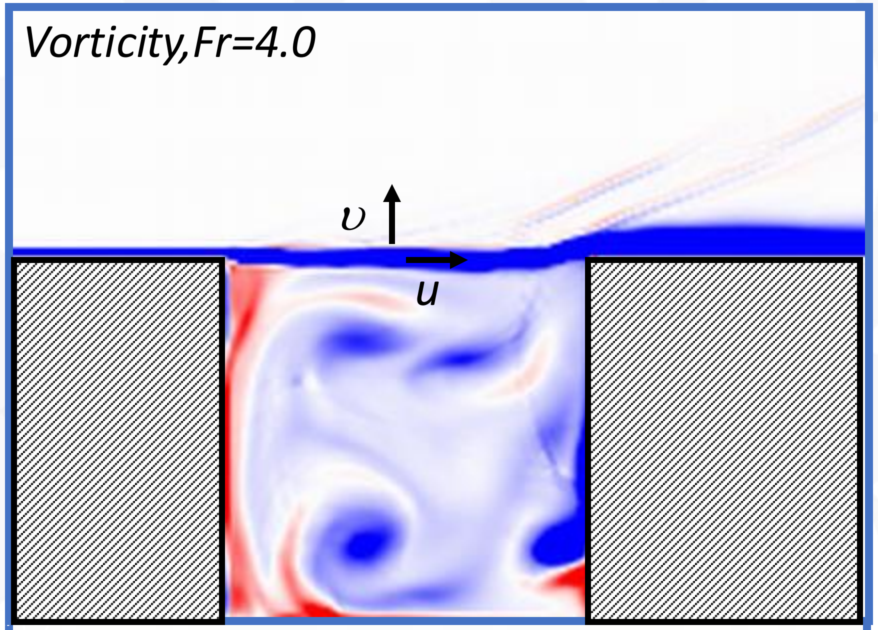
- The standing shock waves strongly interact with shear layer

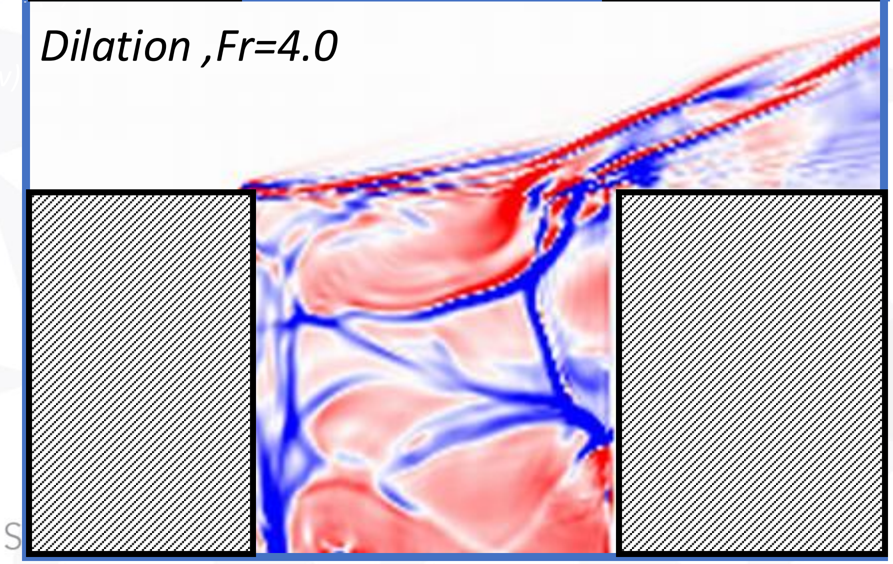
Side Cavity – Groynes (in-house CFD)





- Wave radiation reduces the turbulence kinetic energy. Hence reducing the magnitude of fluctuations
- Magnitude of the *v* fluctuation is tremendously smaller than *u* fluctuation



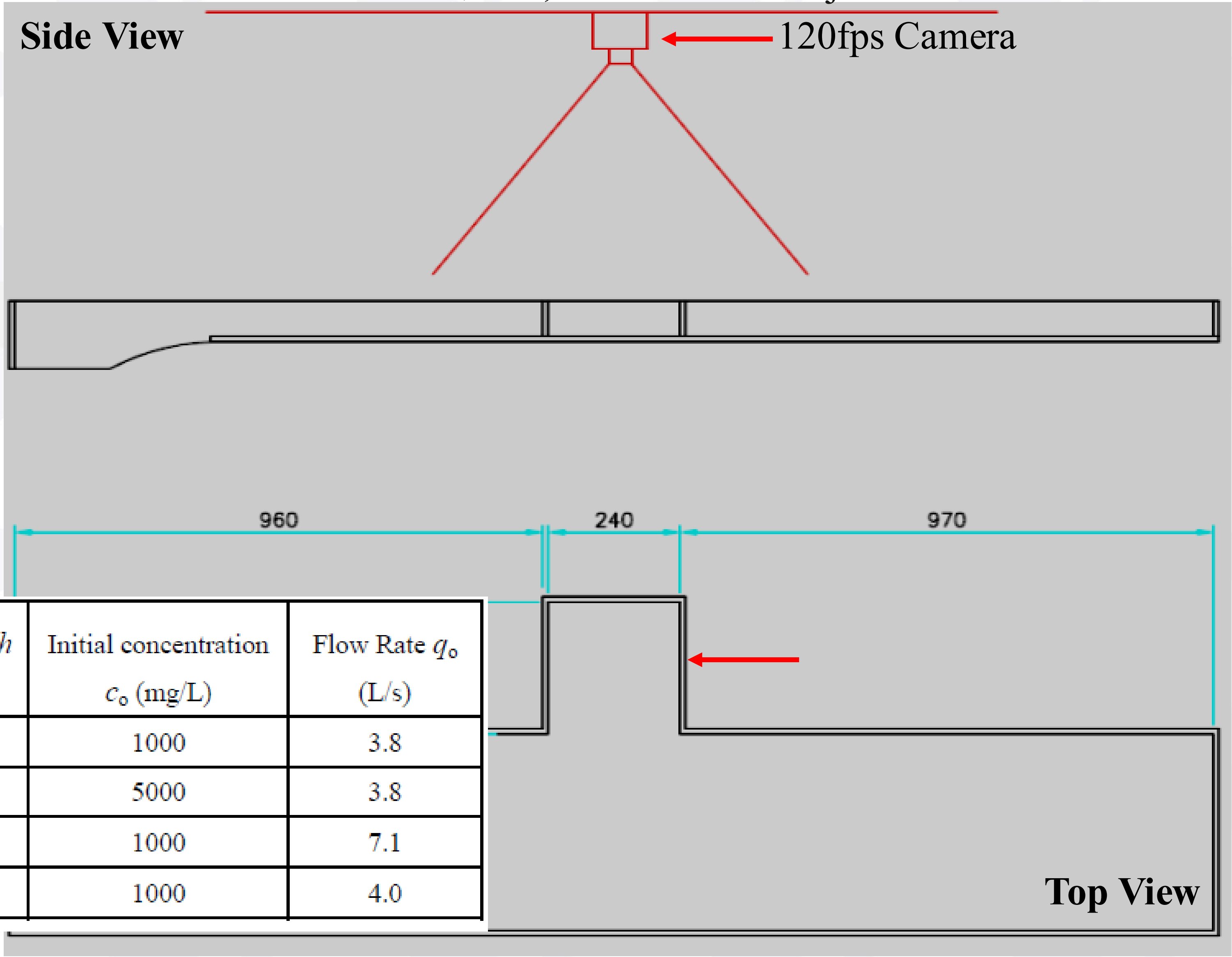


Side Cavity – Groynes (in-house CFD)

Karimpour, S., Wang, T., & Chu, V. (2021). The exchanges between the mainstream in an open channel and a recirculating flow on its side at large Froude numbers. *Journal of Fluid Mechanics, 920*, A8. doi:10.1017/jfm.2021.418

Experimental Setup:

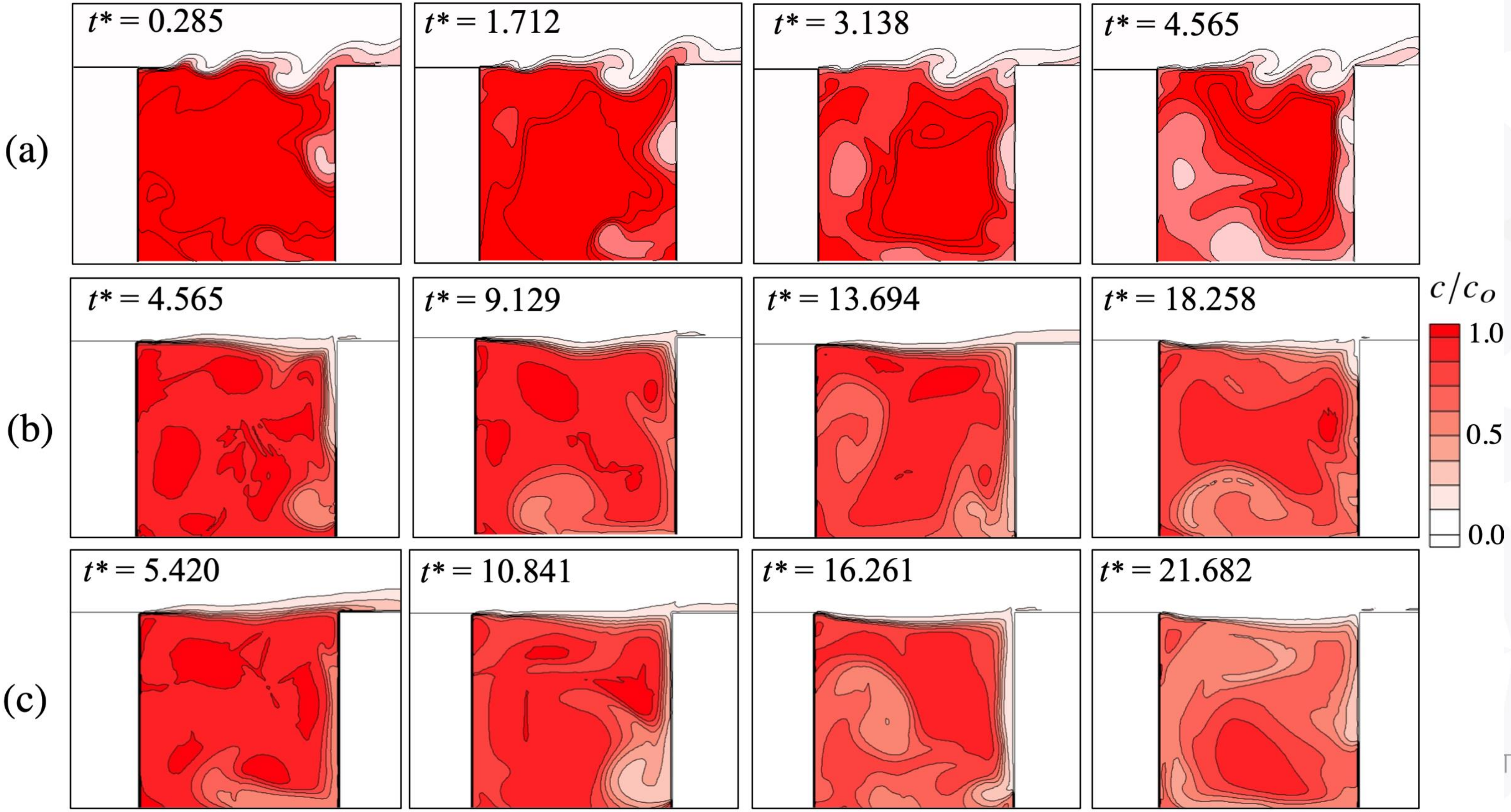
- 24x24cm^2 basin;
- Video Camera mounted on the top;
- Uniform flow through honeycomb;
- Dark room set-up.



Exp. Number	Forude number Fr	Channel Slope	Velocity U (m/s)	Depth h (cm)	Initial concentration c_o (mg/L)	Flow Rate q_o (L/s)
1	0.48	0.00%	0.29	3.7	1000	3.8
2	0.52	0.00%	0.30	3.6	5000	3.8
3	2.77	0.30%	1.14	1.7	1000	7.1
4	1.35	0.30%	0.58	1.9	1000	4.0

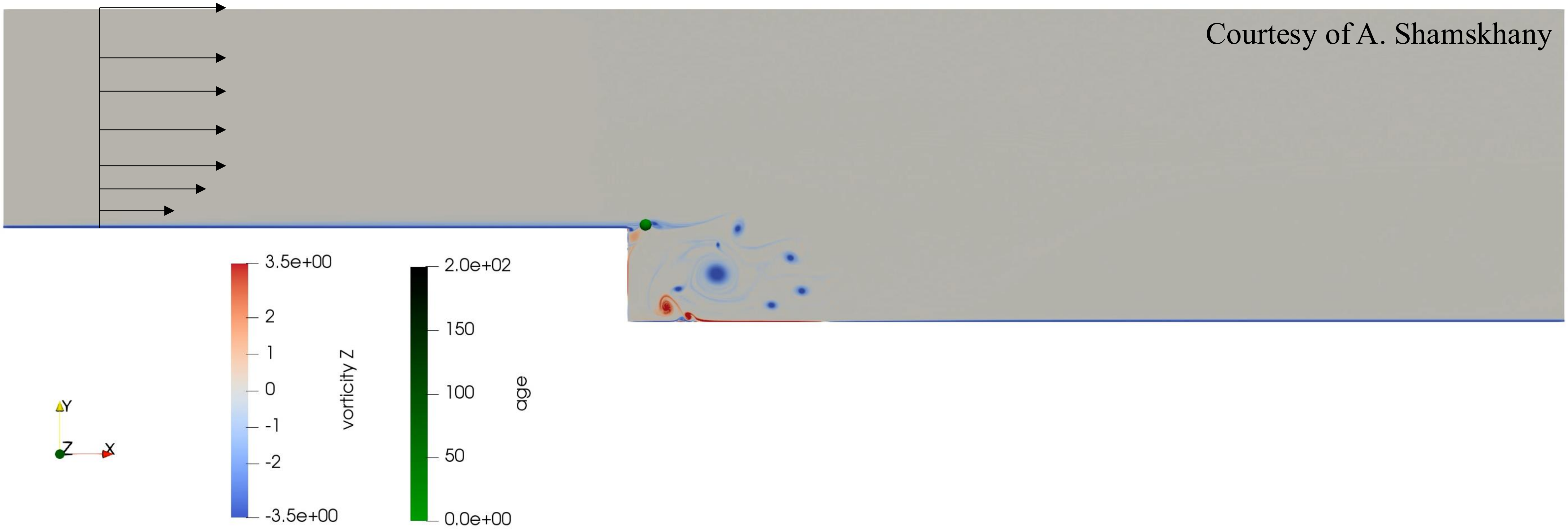
Side Cavity – Groynes (in-house CFD)

Karimpour, S., Wang, T., & Chu, V. (2021). The exchanges between the mainstream in an open channel and a recirculating flow on its side at large Froude numbers. *Journal of Fluid Mechanics, 920*, A8. doi:10.1017/jfm.2021.418

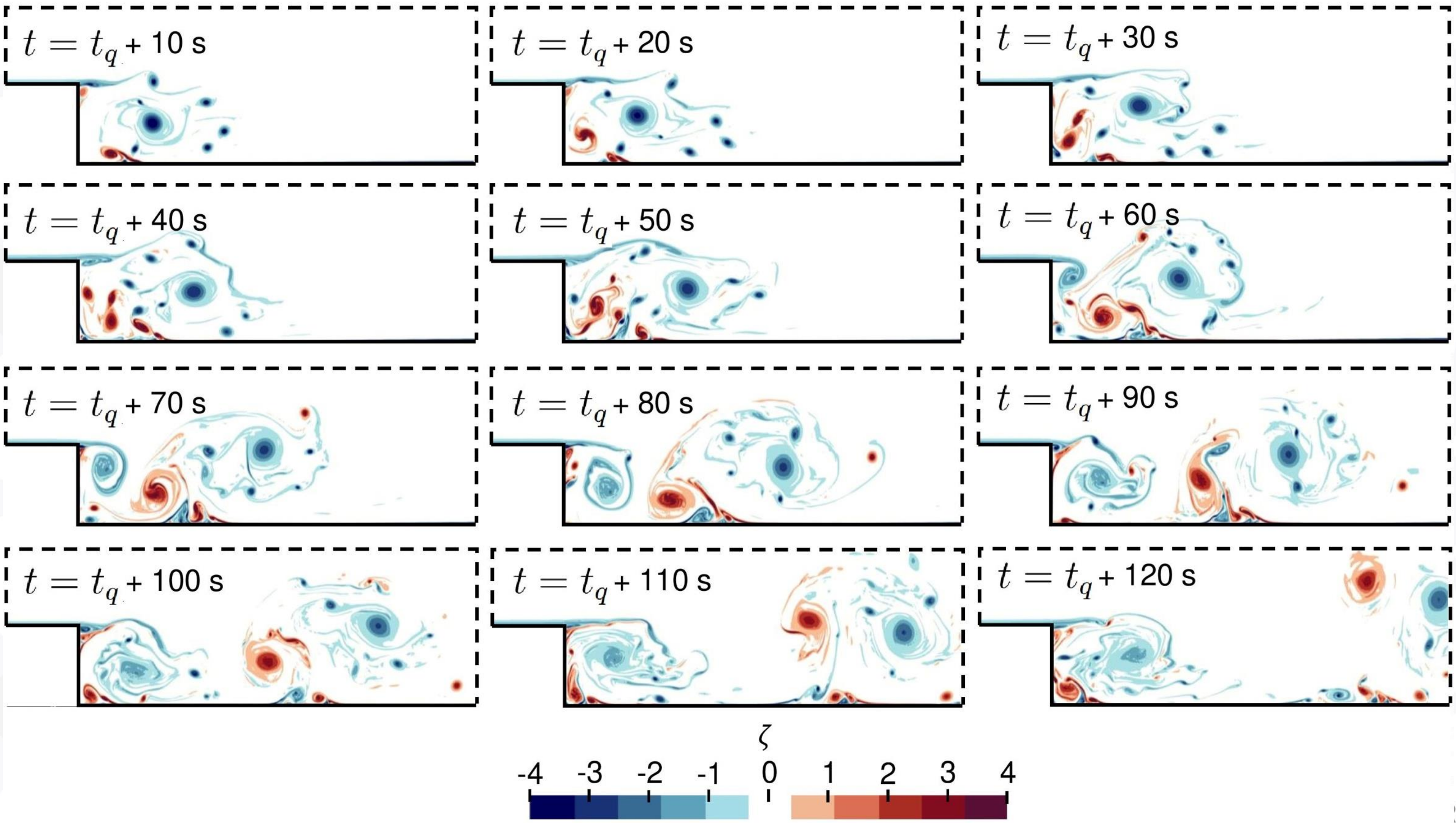


MP entrainment - Backward Facing Step (BFS) – using OpenFoam

Radiating Boundary Condition, allowing the incoming waves to scape;
Reflecting boundary condition, simulating the opposite bank;
Allowing uniform inflow, and adjusting the slope to balance the friction;
Turbulent model (RANS not appropriate – LES used)

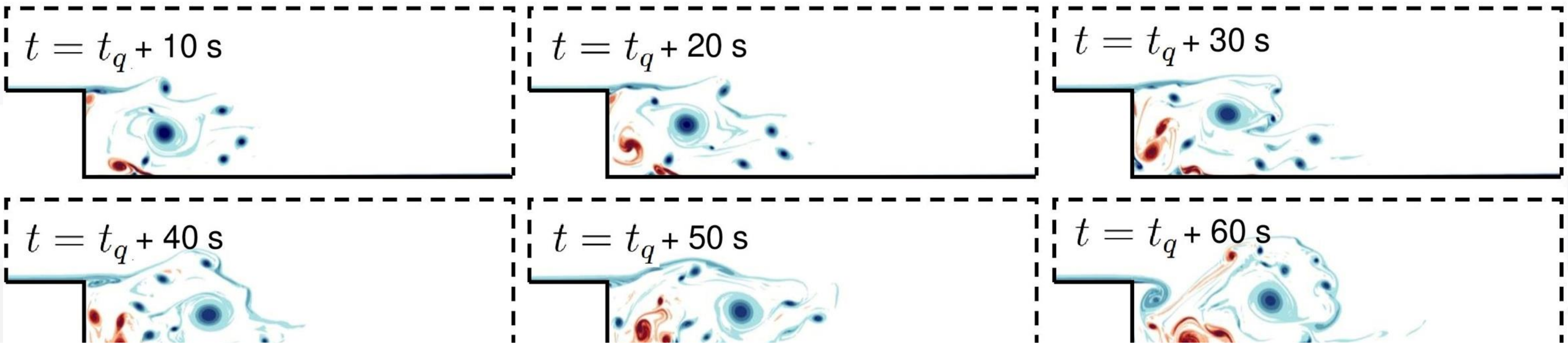


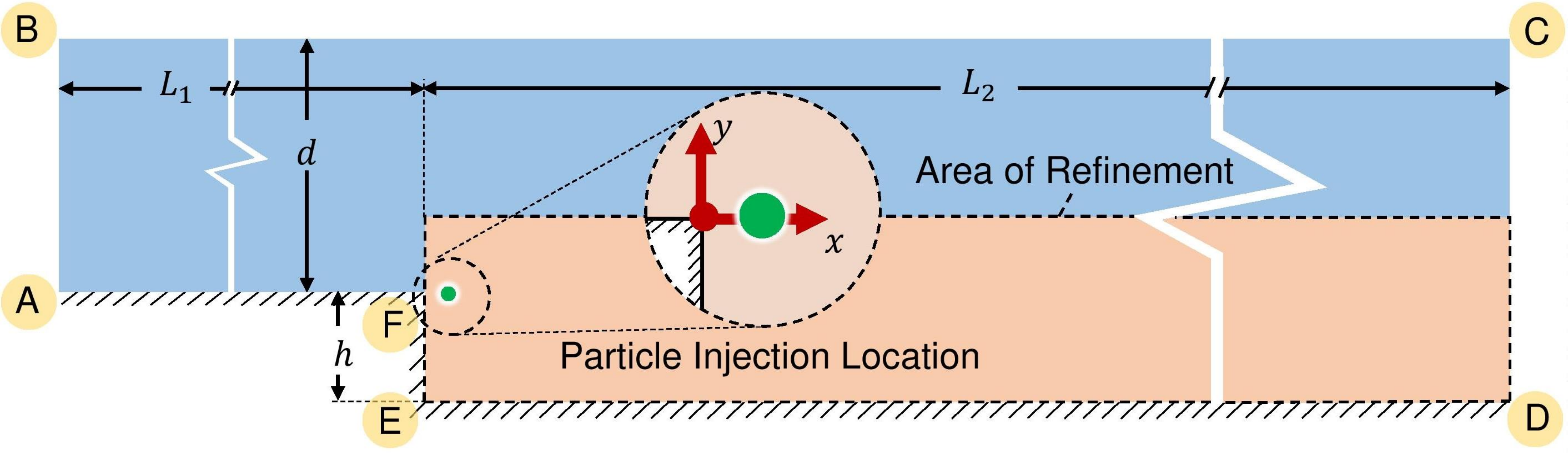
MP entrainment - Backward Facing Step (BFS) – using OpenFoam



SITY

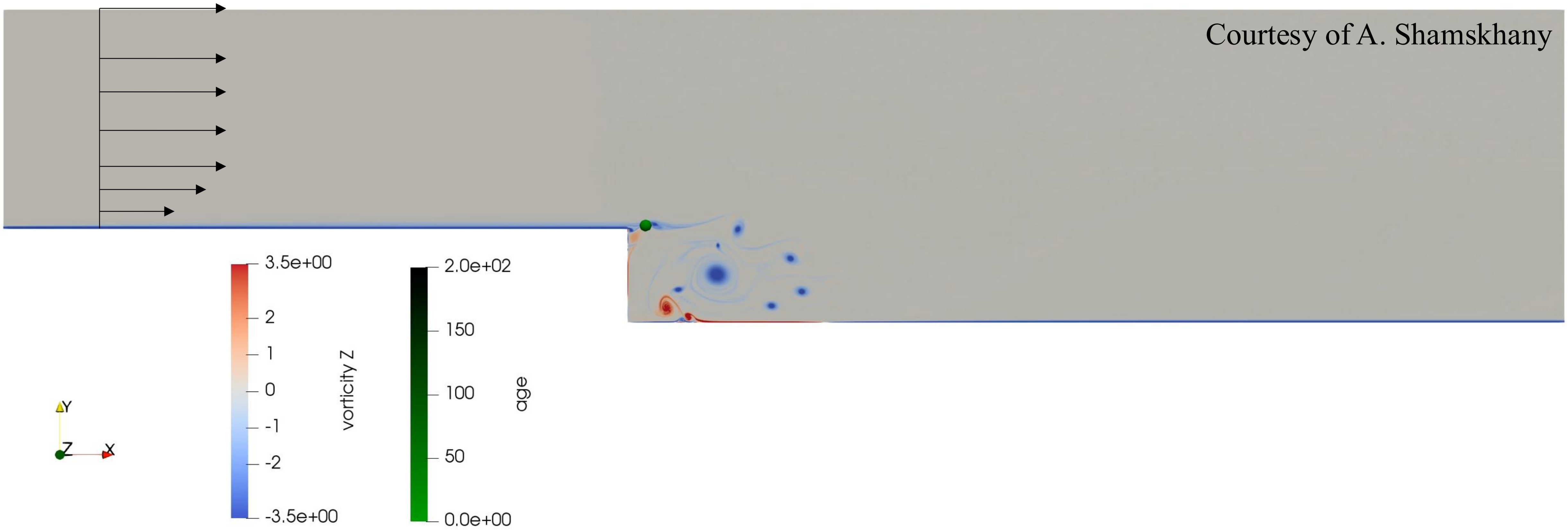
MP entrainment - Backward Facing Step (BFS) – using OpenFoam



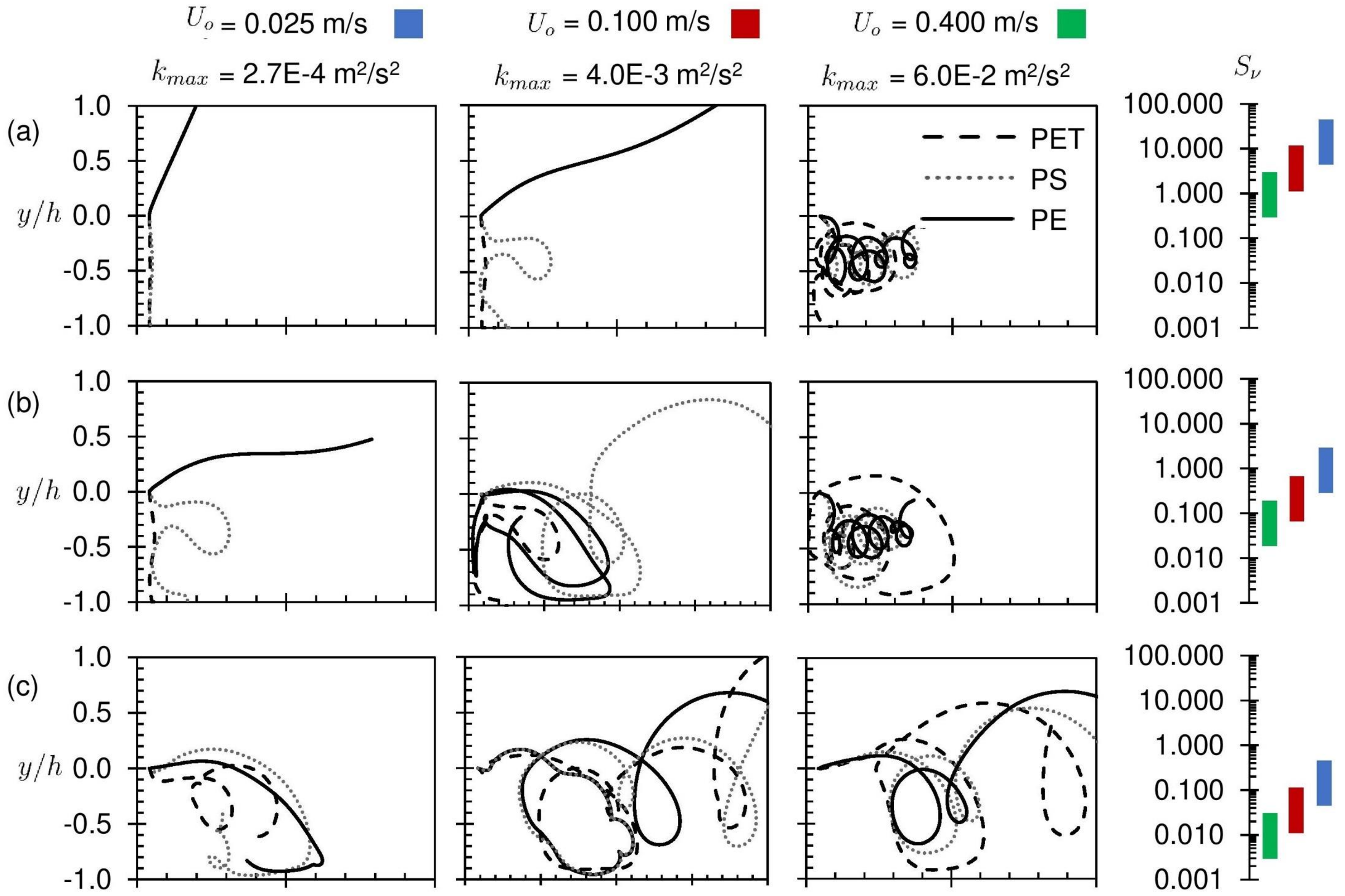


MP entrainment - Backward Facing Step (BFS) – using OpenFoam

Radiating Boundary Condition, allowing the incoming waves to scape;
Reflecting boundary condition, simulating the opposite bank;
Allowing uniform inflow, and adjusting the slope to balance the friction;
Turbulent model (RANS not appropriate – LES used)



MP entrainment - Backward Facing Step (BFS) – using OpenFoam



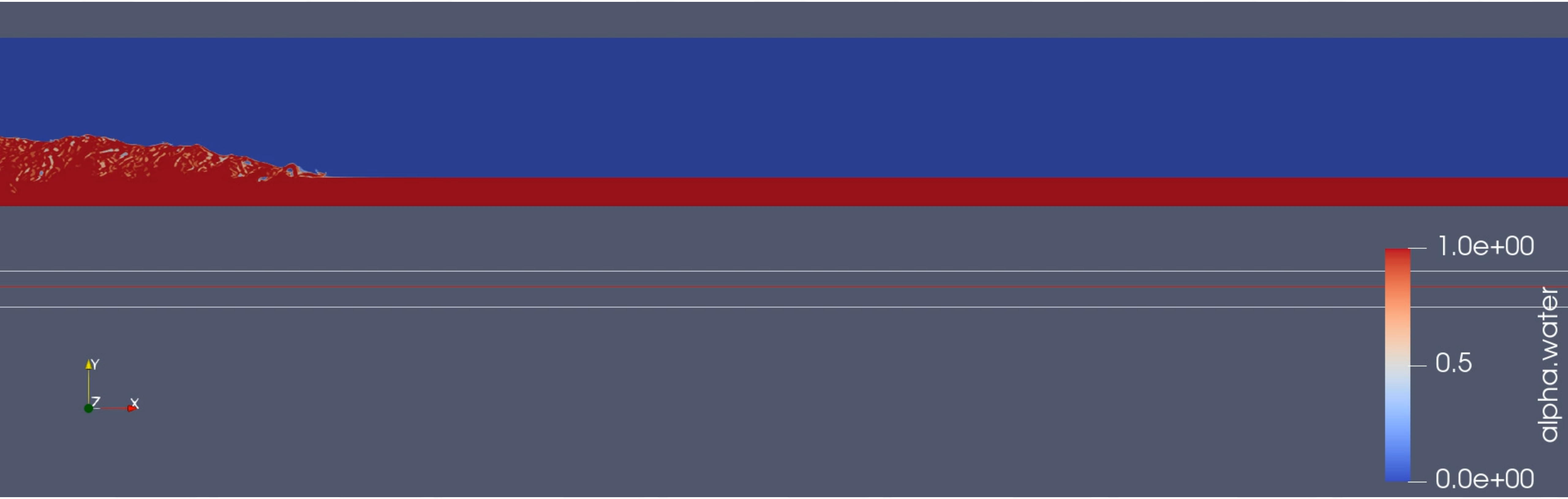
UNIVERSITY

Dam Break Wave (DBW) Anisotropy– using OpenFoam

2-phase solver;

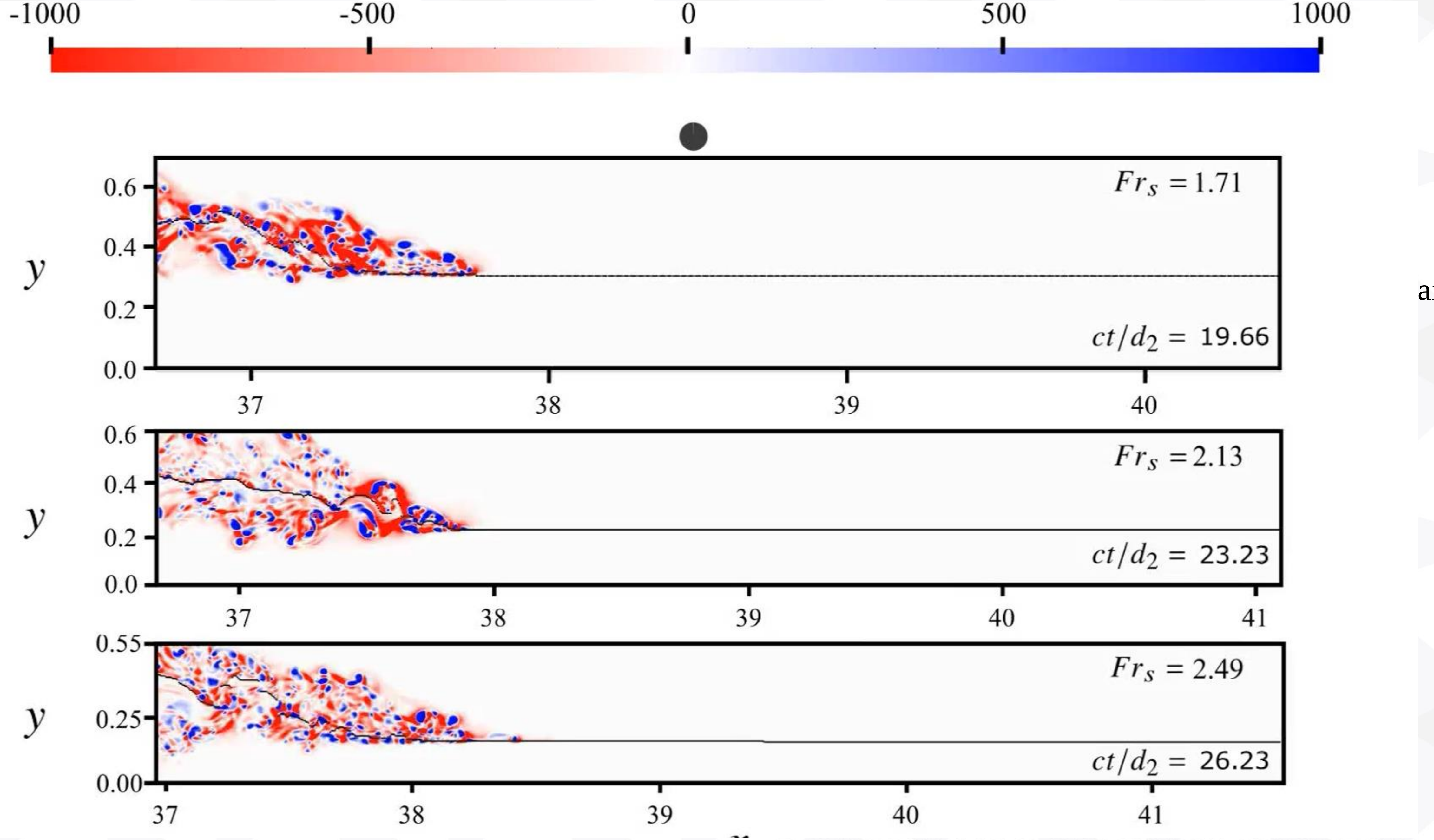
3D behaviour (initially attempted 2D) ➔ Cannot produce the energy cascade

Turbulent model (RANS not appropriate – LES used)



Dam

2-phas
3D bel
Turbul



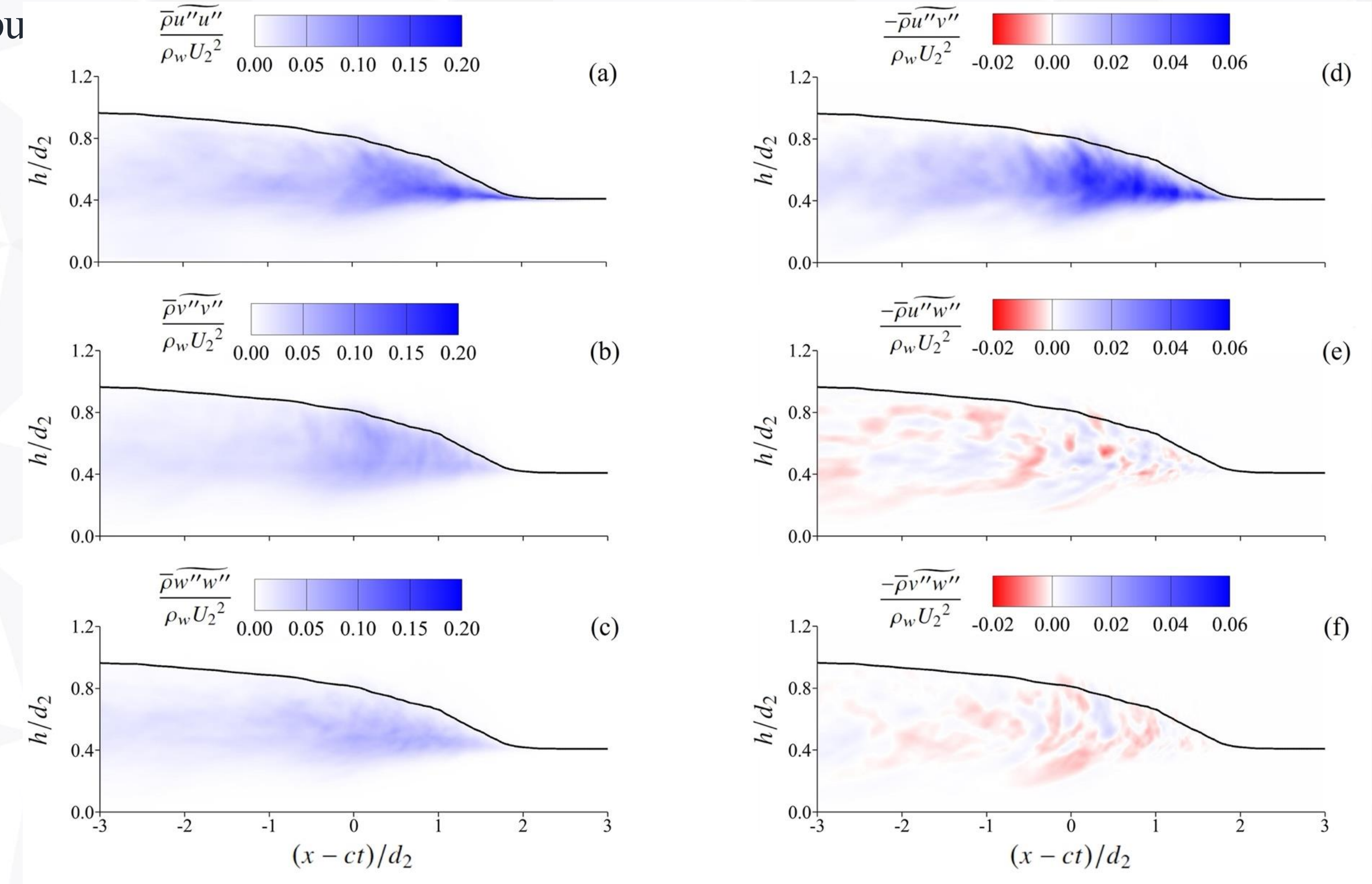
an

Dam Break Wave (DBW) Anisotropy– using OpenFoam

2-phase solver;

3D behaviour (initially attempted 2D) ➔ Cannot produce the energy cascade

Turbu



Courtesy of A. Venkateshwaran

CFD and Computational Hydrodynamics platforms?

- **In-house platforms:**

Defined and programmed by the use. Limited scope, applications, but often tailored to a specific purpose. **(The Cavity example)**

- **Proprietary (commercial) platforms:**

Closed box approach. Often comprehensive in terms of governing eqs. And schemes, but user has no access to the source code. No access to modify or improve limitations. ANSYS Fluent, Flow 3D, HecRAS (1D, 2D, also 3D but very narrow governing eqs.) **(Stepped spillway and Dam outlet)** ➔ Ease of use. Often adopted/preferred by engineering firms.

- **Open-source platforms:**

Best of both worlds! **(Dam break wave and BFS)**

CFD and Computational Hydrodynamics platforms?

- **In-house platforms:**

Defined and programmed by the use. Limited scope, applications, but often tailored to a specific purpose. **(The Cavity example)**

- **Proprietary (commercial) platforms:**

Engineers are often in this category!!

Closed box approach. Often comprehensive in terms of governing eqs. And schemes, but user has no access to the source code. No access to modify or improve limitations. ANSYS Fluent, Flow 3D, HecRAS (1D, 2D, also 3D but very narrow governing eqs.) **(Stepped spillway and Dam outlet)** ➔ Ease of use. Often adopted/preferred by engineering firms.

- **Open-source platforms:**

Best of both worlds! **(Dam break wave and BFS)**

CFD and Computational Hydrodynamics platforms?

- **In-house platforms:**

Defined and programmed by the use. Limited scope, applications, but often tailored to a specific purpose. **(The Cavity example)**

- **Proprietary (commercial) platforms:**

Engineers are often in this category!

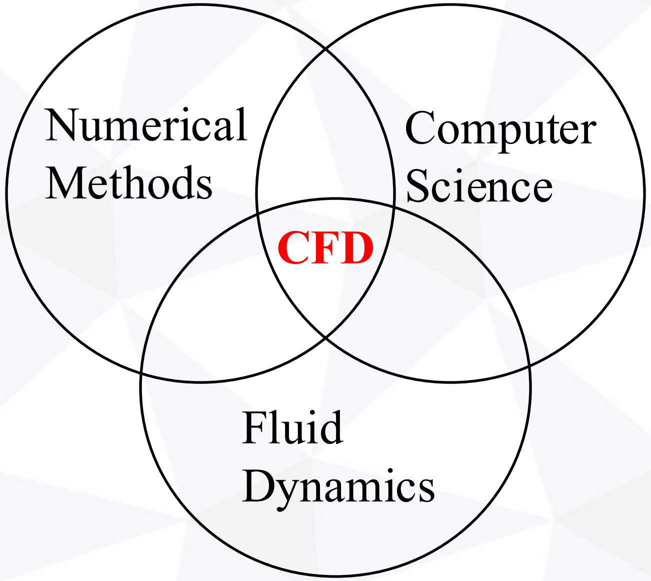
Closed box approach. Often comprehensive in terms of governing eqs. And schemes, but user has no access to the source code. No access to modify or improve limitations. ANSYS Fluent, Flow 3D, HecRAS (1D, 2D, also 3D but very narrow governing eqs.) **(Stepped spillway and Dam outlet)** ➔ Ease of use. Often adopted/preferred by engineering firms.

- **Open-source platforms:**

Best of both worlds! **(Dam break wave and BFS)**

Computational Hydrodynamics is inter-disciplinary

- **In nature:**



- **Applications:**

Diverse applications (see earlier slides)

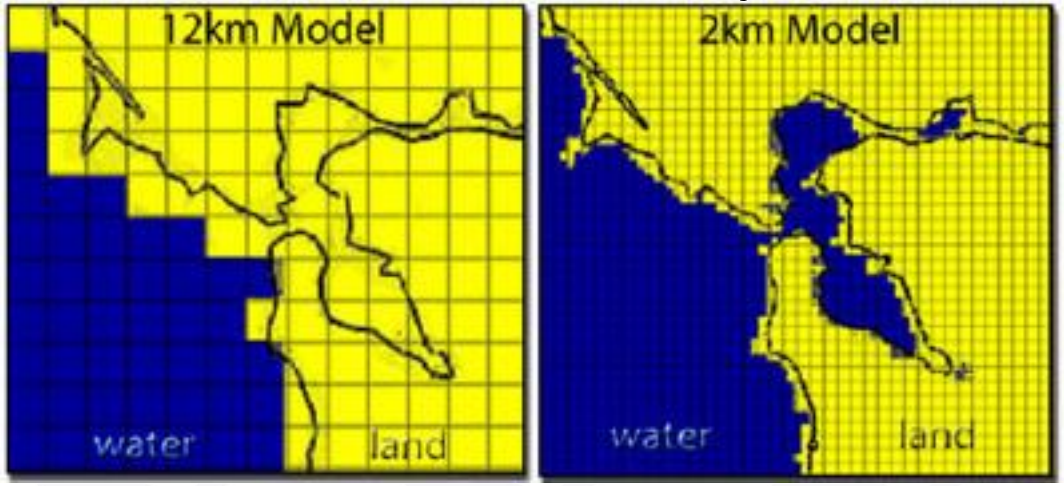
Computational Hydrodynamics is inter-disciplinary

$$\boxed{\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}.\nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p - g\nabla h + \nu\nabla^2\mathbf{u}}$$

$$\boxed{\frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = 0}$$

$$\frac{\partial \alpha}{\partial t} + \frac{\partial u_i \alpha}{\partial x_i} + \frac{\partial}{\partial x_i}[u_{ci}\alpha(1-\alpha)] = 0$$

Weather modelling mesh – San Francisco Bay



Computational Hydrodynamics is inter-disciplinary

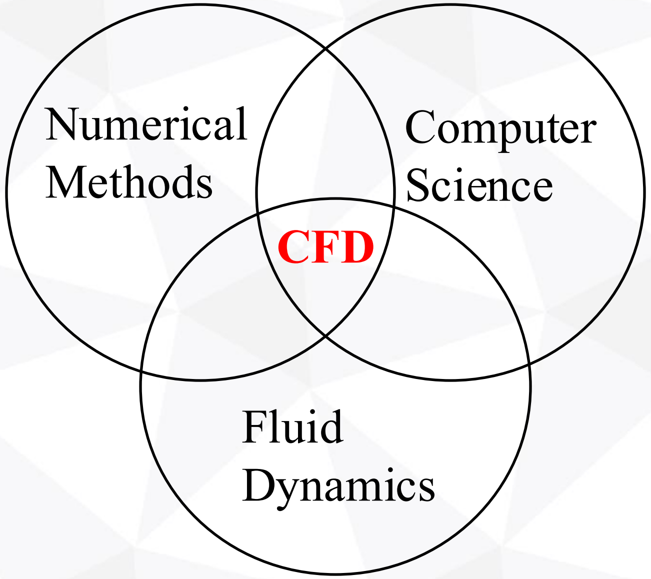
$$\boxed{\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}.\nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p - g\nabla h + \nu\nabla^2\mathbf{u}}$$

$$\boxed{\frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = 0}$$

$$\frac{\partial \alpha}{\partial t} + \frac{\partial u_i \alpha}{\partial x_i} + \frac{\partial}{\partial x_i}[u_{ci}\alpha(1-\alpha)] = 0$$

These partial differential equations describe a continuum of real number and often there is no exact solution.
We aim to convert these into algebraic equations to solve over particular locations across the flow.
Using CFD techniques, you ensure stability and reliability;
Implement in a programming language;
Post-processing to make sense of the results;

As a user/engineer do I need to know fundamentals of CFD?



CFD Process:

- **Formulate the Geometry and Flow Domain** (Fluid Dynamics)
  what is the objective of the analysis? what is the easiest way to obtain those objective? what geometry should be included? what are the freestream and/or operating conditions?
- **Formulate the Flow - Chose the Governing Equations** (Fluid Dynamics)
  what dimensionality of the spatial model is required? (1D, quasi-1D, 2D, axisymmetric, 3D) what temporal modeling is appropriate? (steady or unsteady) what is the nature of the viscous flow? (inviscid, laminar, turbulent)
- **Establish the Boundary and Initial Conditions** (Fluid Dynamics/numerical scheme)
  Since a finite flow domain is specified, physical conditions are required on the boundaries of the flow domain. The simulation generally starts from an initial solution and uses an iterative method to reach a final flow field solution.
- **Construct the grids** (All 3)
  Since a finite flow domain is specified, physical conditions are required on the boundaries of the flow domain. The simulation generally starts from an initial solution and uses an iterative method to reach a final flow field solution.
- **Establish the computational Scheme** (Numerical Scheme / Comp Sci)
  Eulerian vs Lagrangian - Finite Difference - Finite Volume - Spatial and temporal breakdown
- **Monitor the Simulation for Completion** (All 3)
  As the simulation proceeds, the solution is monitored to determine if a "converged" solution has been obtained. Monitor the speed of computation. Ensure the convergence and stability mid-way.
- **Post-Processing and Validation** (All 3)
  Post-Processing involves extracting the desired flow properties (Vorticity, lift, turbulent characteristics, etc...) from the computed flow-field. The computed flow properties are then compared to results from analytic, computational, or experimental studies to establish the validity of the computed results.

CFD Process:

Questions to answer in formulating the geometry - Step#1:

What is the area of interrogation?
Should I simulate the entire length of the stream?
How many groynes to consider?

Questions to answer in formulating the geometry - Step#2:

Turbulent or laminar?
What governing equation is proper?
How to account for pressure? What about slope?
How is friction accounted for?
Can I reduce the problem from 3D to 2D?
What turbulent model is appropriate?

Plan for Computational Hydrodynamics course :

See course outline!

CFD to-do list this week:

- **Programming: We will use exclusively Python.**
- **Install Python, depending on your operating system.**
  - Install Python itself – version 3 (version 2 has different commands)
  - Install an Integrated Development Environment (IDE)
  - Python Free learning resources:

    Python Essential Training *by Bill Weinmann* (LinkedIn Learning)

    Learn Python - Full Course for Beginners [Tutorial] (YouTube)
  - **My preference is PyCharm – Community version. Follow this instruction for installation:**

    https://www.guru99.com/how-to-install-python.html (WINDOWS)

    https://www.educademy.co.uk/how-to-install-python-and-pycharm-on-mac (MAC)
- **Assignment#1: Warmup with Python features and plot of a time/space dependent problem**

House-keeping items:

- **Assignments matter a lot! It guides you through the course and the final project.**
- For a grad level course, I expect you to produce grad-level and professional results and documents.
- As with other grad courses, the material is open ended.
- **Weekly lectures 2 hours of in-person lectures**
- Professionalism and courtesy.
- Template Latex and MS word will be provided
- Assignments- What to submit:

    1- Your source Python file including those used for post-processing

    2- A comprehensive report.