Modelling Considerations and Advanced Hydraulics For Fishway Structures

Kyle Thomson

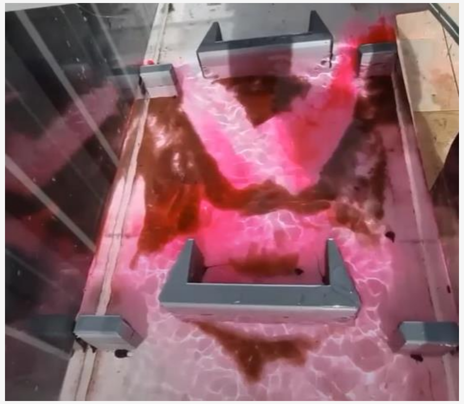
August 2022

Agenda

- Flume vs physical model vs CFD
- CFD – equations and assumptions
- Animations!
  - Entrance attraction
  - Fishway Internals
  - Locks and Gates

Flume vs Physical Model vs CFD

- Flume:
  - Easy to set up
  - Real time optioneering
  - At an early design stage, can eliminate non-viable options
- Flume Limitations:
  - Poor scalability for turbulence, eddies and velocities
  - Larger sites have scalability limitations



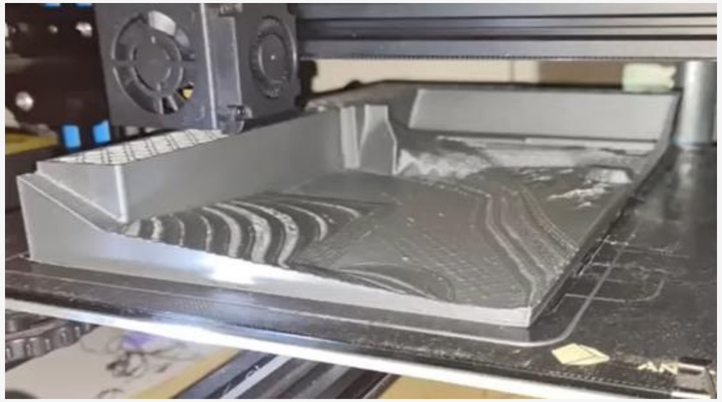
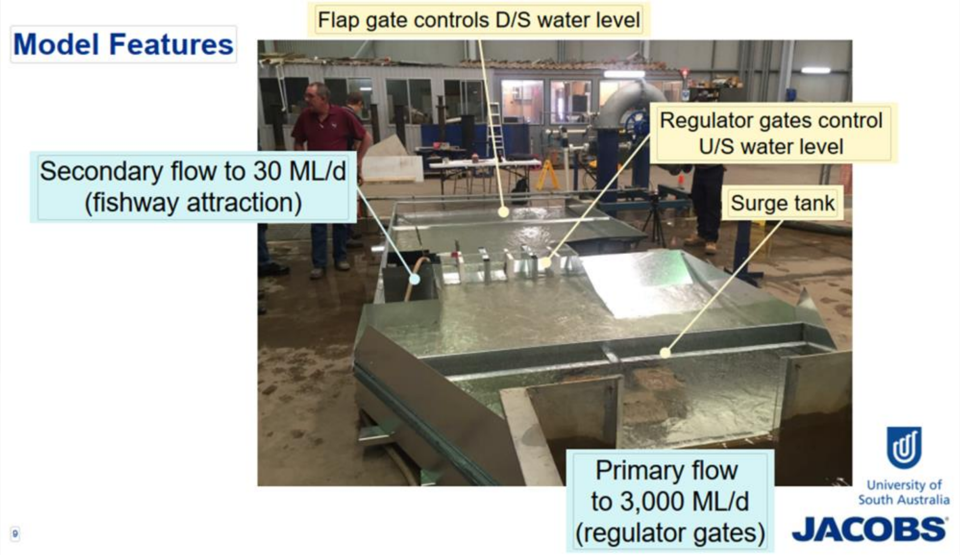
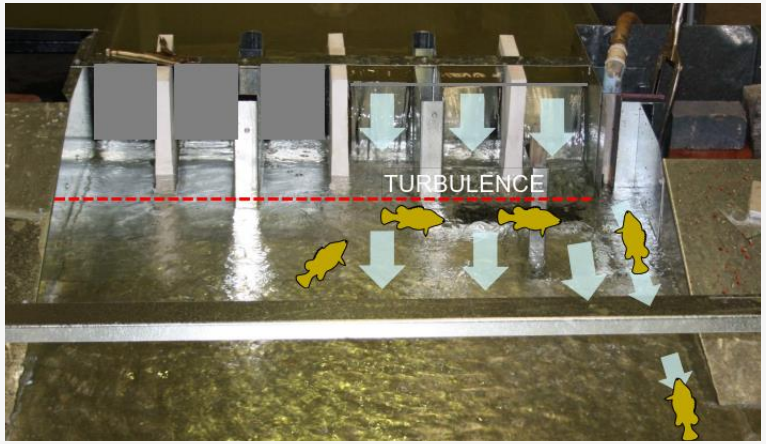


Image credit – Tim Marsden
Check out a previous AWS webinar – Technological Advances in the Water Sector

Flume vs **Physical Model** vs CFD

- Physical Model:
  - Set up with a fixed scale (1:15, 1:50 ect)
  - Modelled off well established hydraulic principles (Froude No, Bernoulli Equation ect)
  - Well backed science, extensive literature
  - High degree of certainty for velocities, pressure forces and eddies (with limitations)
  - Often can carry a design through to IFC (bar some extreme cases)
- Physical Model Limitations:
  - Time consuming to set up
  - Real estate costs, increased materials costs ect
  - Output accuracy limited to recording equipment
  - Struggle to replicate extreme conditions

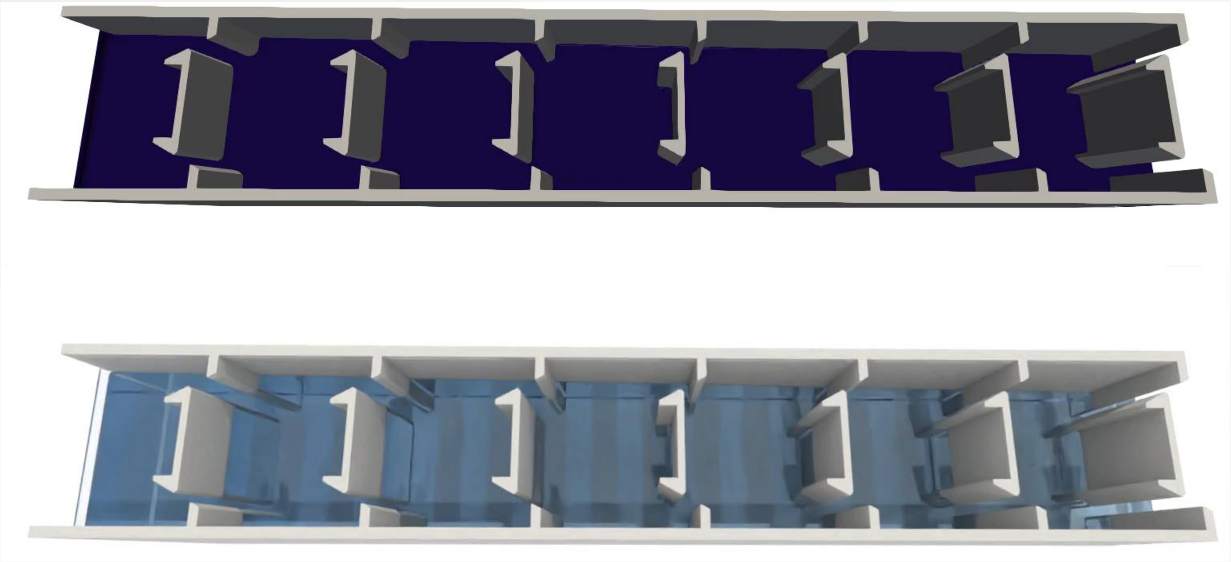




Slarke, S & Stuart, I & Pezanitti, D. "Optimisation of Fishway Entrance and Exit Conditions Using Physical Modelling: SARFIIP Pike Floodplain Regulator and Fishway Designs". 2018.

Flume vs Physical Model vs **CFD**

- CFD Model:
  - Quick to set up, optioneering can be quick (< 1 day per run) if model inputs are dialed back
  - Often can carry a design through to IFC (not for all cases!!!!!!)
  - Once simulated, virtually any output can be post-processed
  - Can model Non-Newtonian flows, sed – transport, air entrainment ect
- CFD Limitations:
  - Best practices still being established (and subjective)
  - Any good model should be backed with an extensive literature review and validation
  - Can cause project timeline drag
  - Limitations to modelling turbulence and unsteady state



Computed Outputs
- Distance traveled by fluid
- Fluid arrival time
- Fluid residence time
- Fluid strain rate
- Fluid vorticity
- › Hydraulic data
- Local maximum velocity
- Shear stress
- Total hydraulic head 3D
- Wall contact time

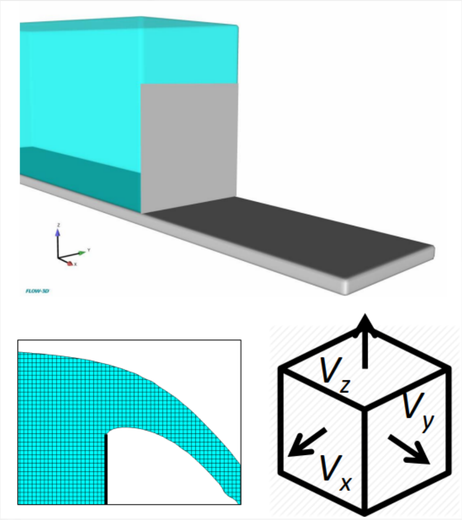
Default Outputs
- Fluid fraction
- Fluid velocities
- Pressure

Physical Model Outputs
- Dynamic viscosity
- Turbulent quantities (tke/dtke)

Animation CFD – FLOW3D Team

CFD – Equations and Assumptions

- CFD (Navier-Stokes Equation) solver:
  - Conservation of mass (mass in = mass out)
  - Conservation of momentum (F = ma)
  - Conservation of energy (relevant when modelling heat)



Conservation of mass

$$\frac{D\rho}{Dt} + \rho\left(\nabla \cdot \vec{V}\right) = 0$$

$$\frac{D\rho}{Dt} = 0 \longrightarrow \nabla \cdot \vec{V} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

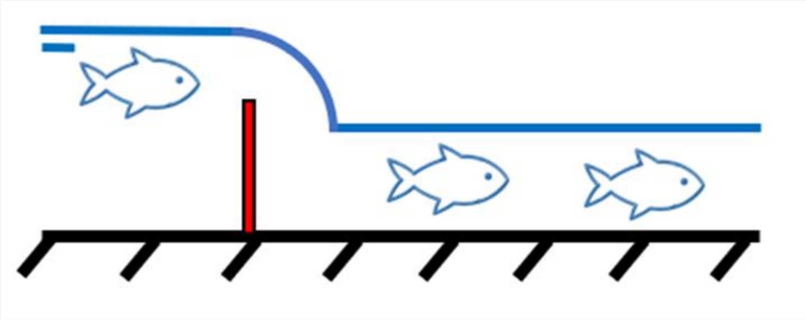
When density is assumed as constant and fluid is incompressible

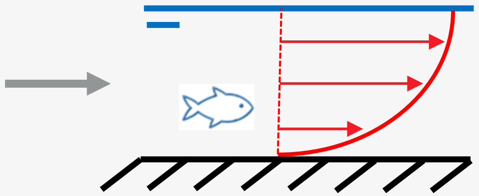
Conservation of momentum

$$\rho\left[\frac{d\vec{V}}{dt} + (\vec{V}\,\nabla)\,\vec{V}\right] = \rho\, g - \nabla p + \mu\, \nabla^2\, \vec{V}$$

- ρ: Fluid density (mass)
- $\left[\frac{d\vec{V}}{dt} + (\vec{V}\,\nabla)\,\vec{V}\right]$: Change in velocity and direction of fluid (acceleration)
- $\rho\, g - \nabla p + \mu\, \nabla^2\, \vec{V}$:
  - External force (ie from gravity)
  - Pressure gradient
  - Fluid stresses (viscous effects)

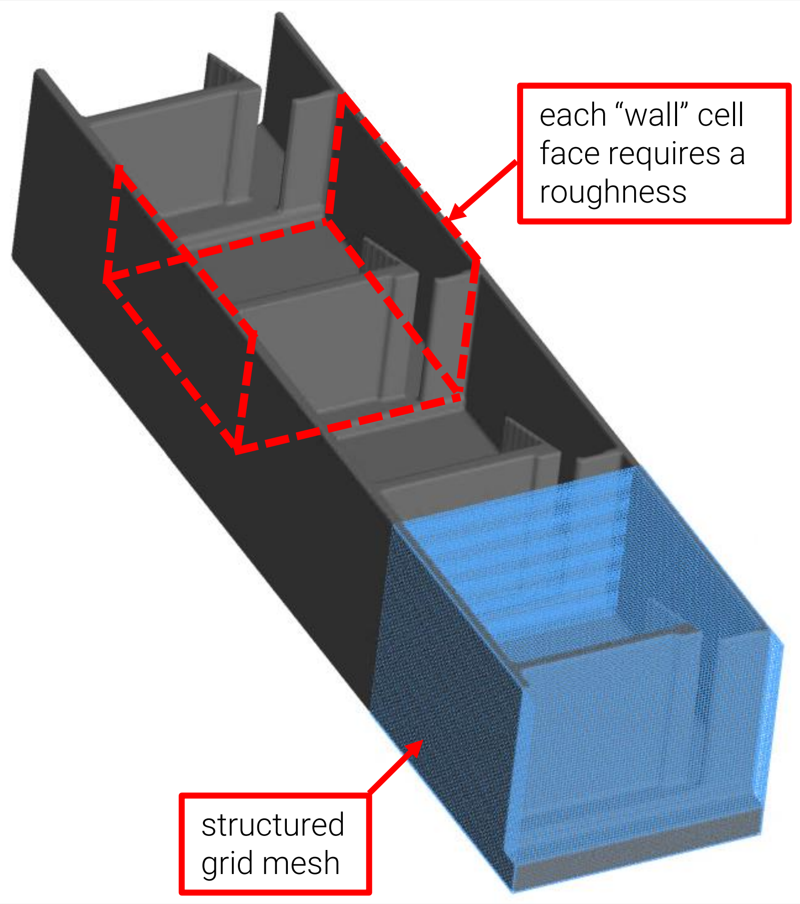
  (Force)

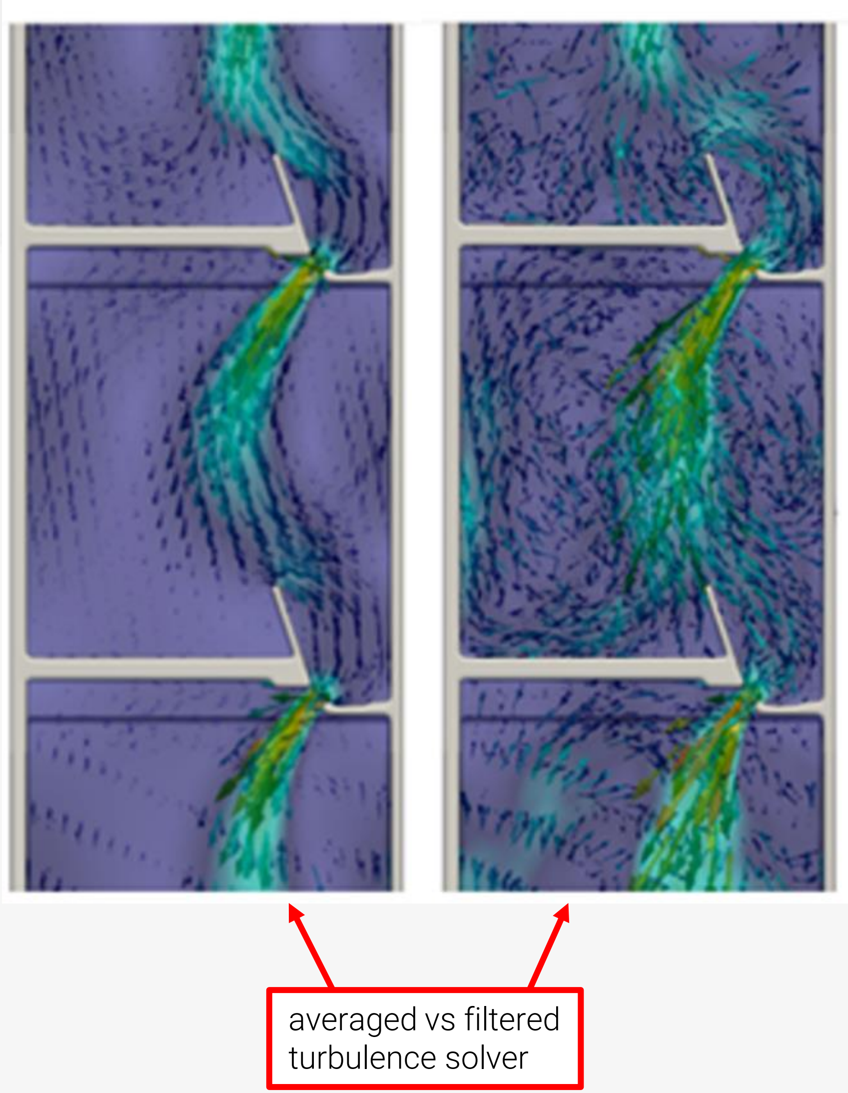




CFD

- Modelling assumptions can significantly impact model results, some common traps are:
  - Mesh resolution
  - Turbulence model
  - Wall roughness
- Different software packages have different assumptions and modelling processes that need to be followed. My preference is FLOW3D

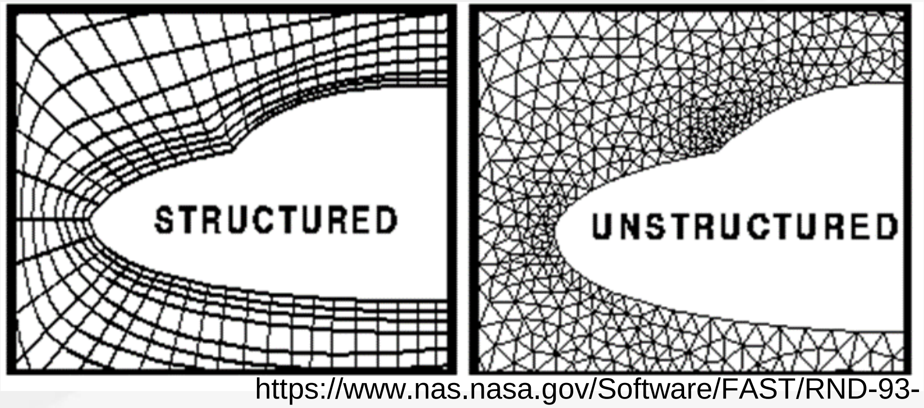




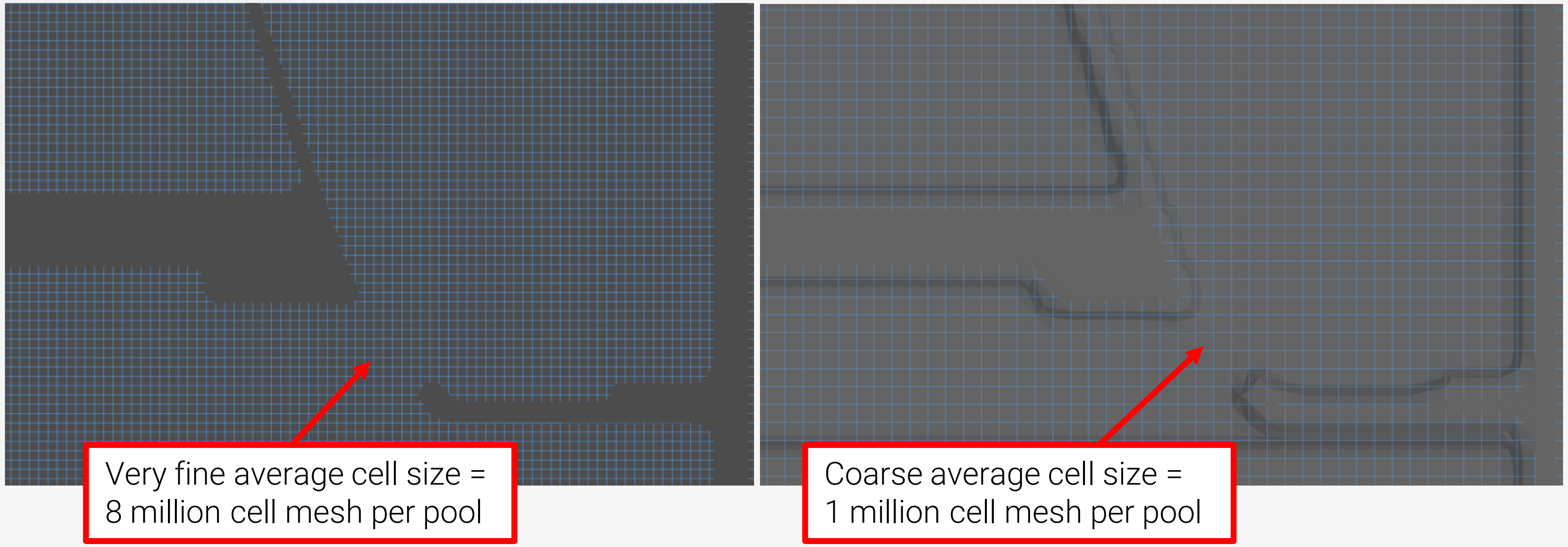
Thomson, K. *"Practical Application of CFD for Fish Passage Design"*. 2022.

CFD – Mesh Resolution

- An appropriate mesh is one that captures the geometry adequately, maintains a reasonable run time and provides defendable results.
- An adequate cell size is critical for meeting several rules of thumb including:
  - Number of cells for orifice / baffle opening
  - Number of cells for flow depth / flow over a structure
  - Cell height related to roughness
  - Turbulence model
  - Universal law of the wall and y^+

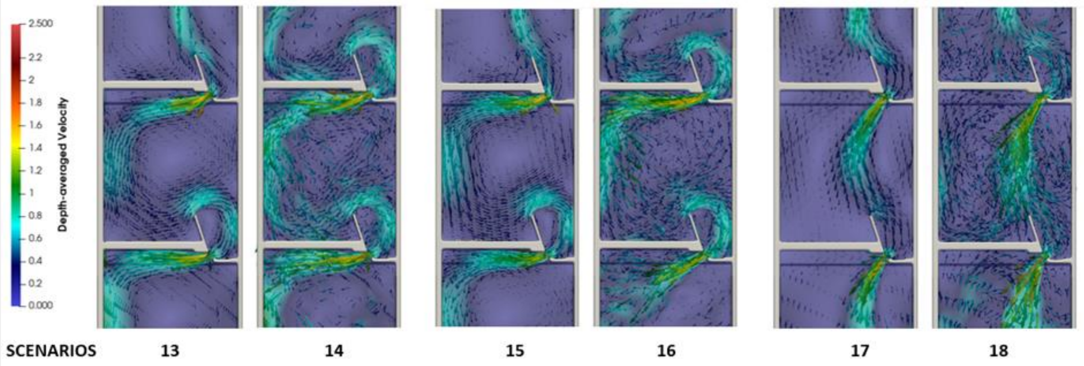


https://www.nas.nasa.gov/Software/FAST/RND-93-010.walatka-clucas/htmldocs/chp_16.surferu.html

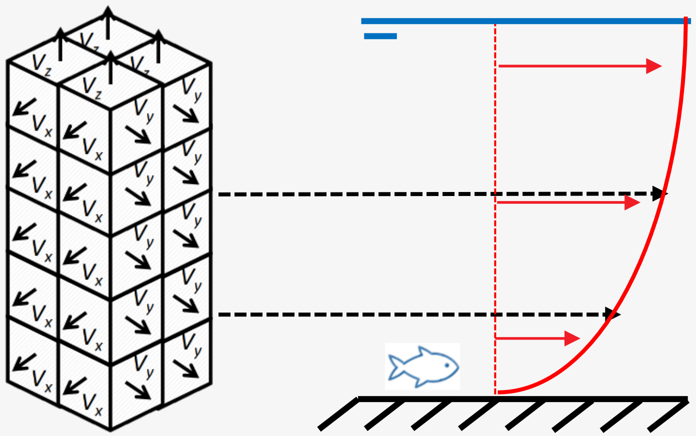


CFD – Turbulence and Roughness

- Common turbulence models include:
  - $k - \epsilon$, assumes model is turbulent, RANS based
  - RNG (Renormalization-Group, $k - \epsilon$), better handles low Reynolds and near wall functions
  - LES (Large Eddy Simulation)
- Wall roughness is applied to the model using the Nikuradse sand-grain-equivalent roughness length:
  - Roughness varies depending on the surface (ie concrete or excavated rock)
  - Roughness trends towards an asymptotic value as its length approaches the cell centroid
  - Alternative approached may be more suitable
- Cell size, turbulence and roughness length aren't interchangeable!



Run #	Differential Head (mm)	Cell Size	Wall Roughness (mm)	Turbulence Solver
13	150	Coarse	1.25	RNG
14	150	Coarse	1.25	LES
15	150	Fine	1.25	RNG
16	150	Fine	1.25	LES
17	150	Very Fine	1.25	RNG
18	150	Very Fine	1.25	LES



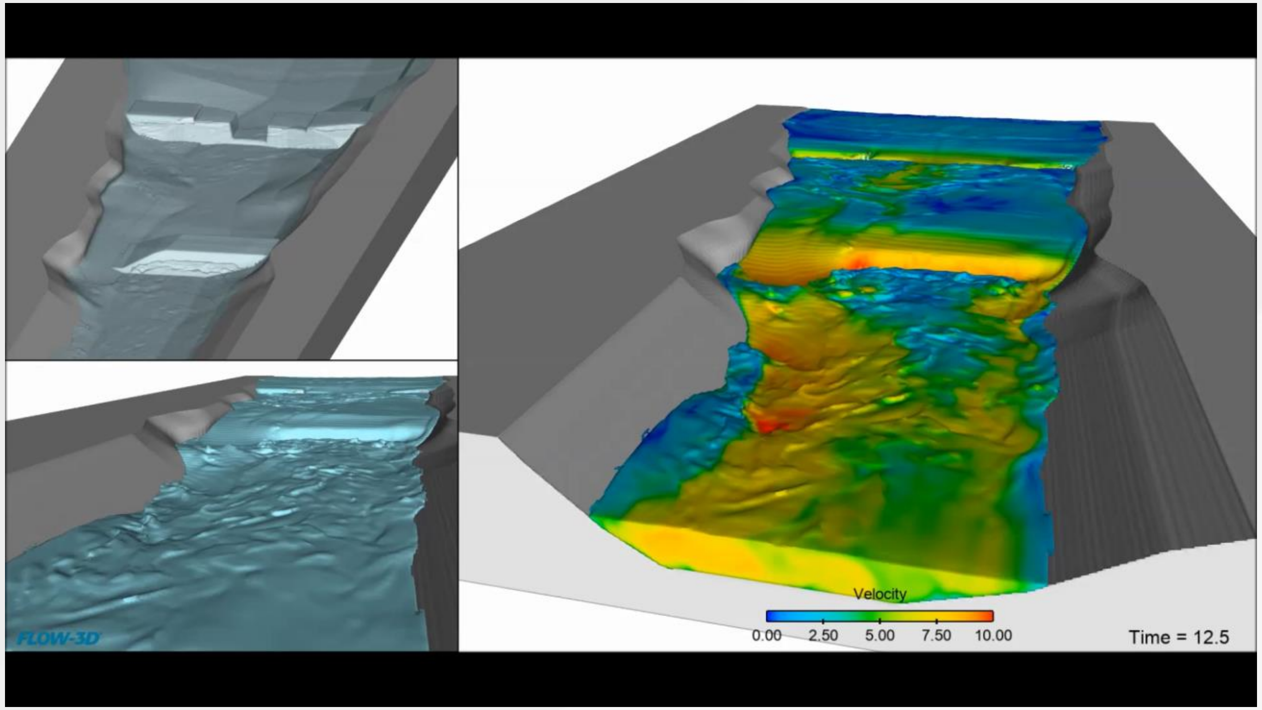
Thomson, K. *"Practical Application of CFD for Fish Passage Design"*. 2022.

WMS CIVIL | WATER | ENVIRONMENT

Entrance Attraction

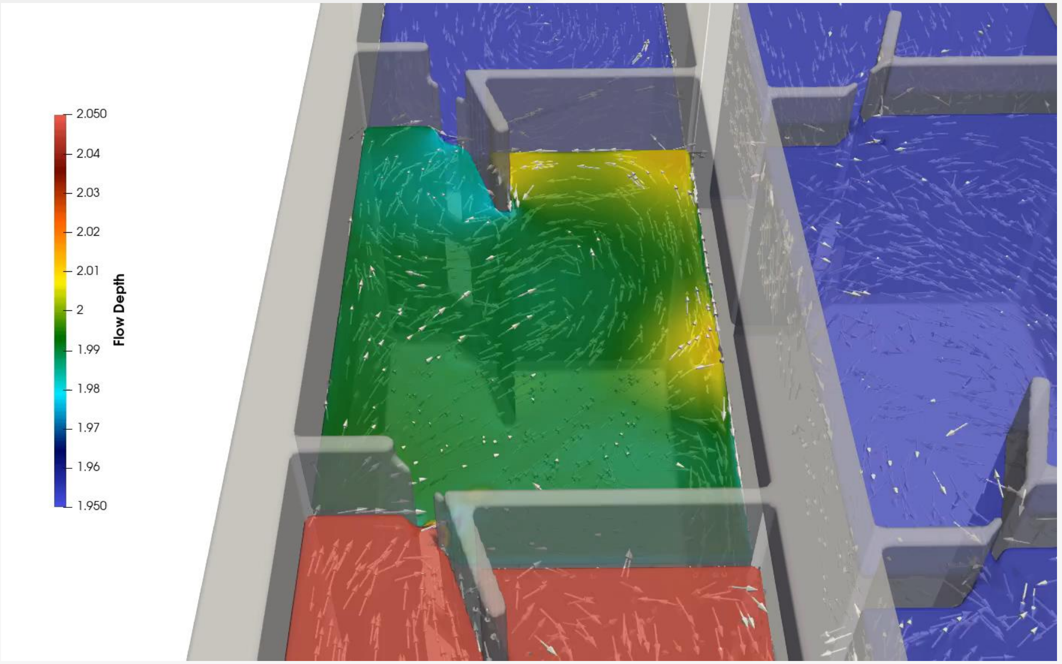
MDBA Animation Credit – MDBA Media (2012)
CFD Animations – FLOW3D and WMS

Entrance Attraction

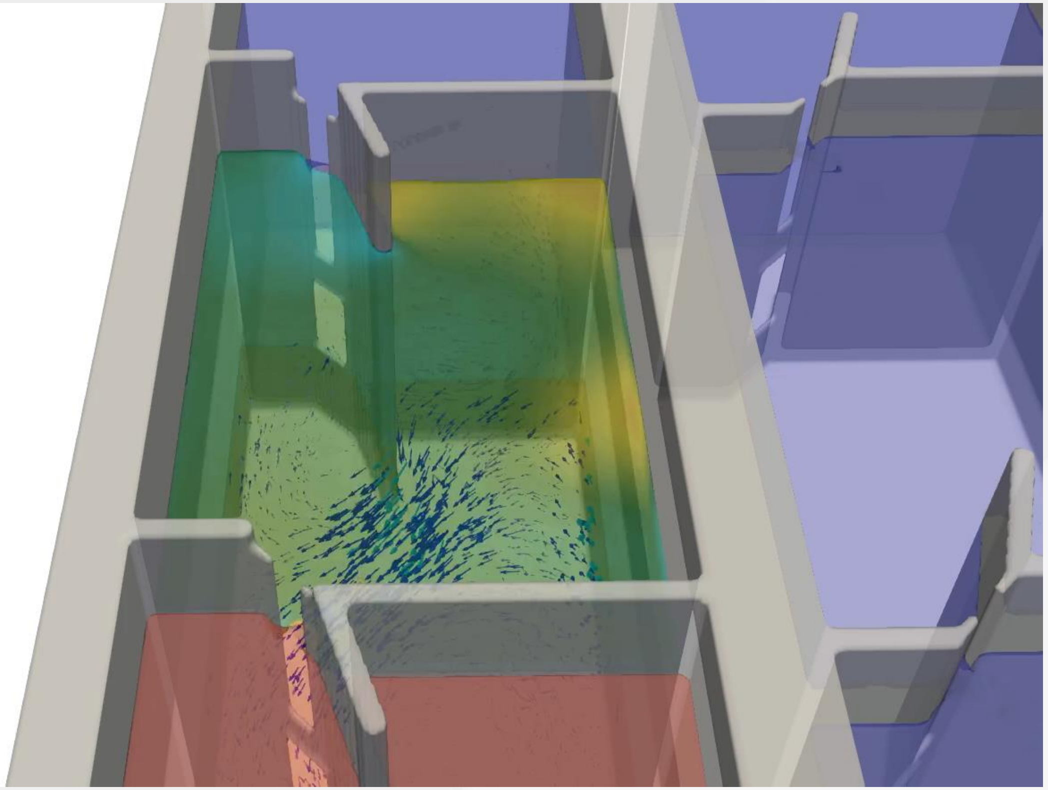


Animation Credit – FLOW3D

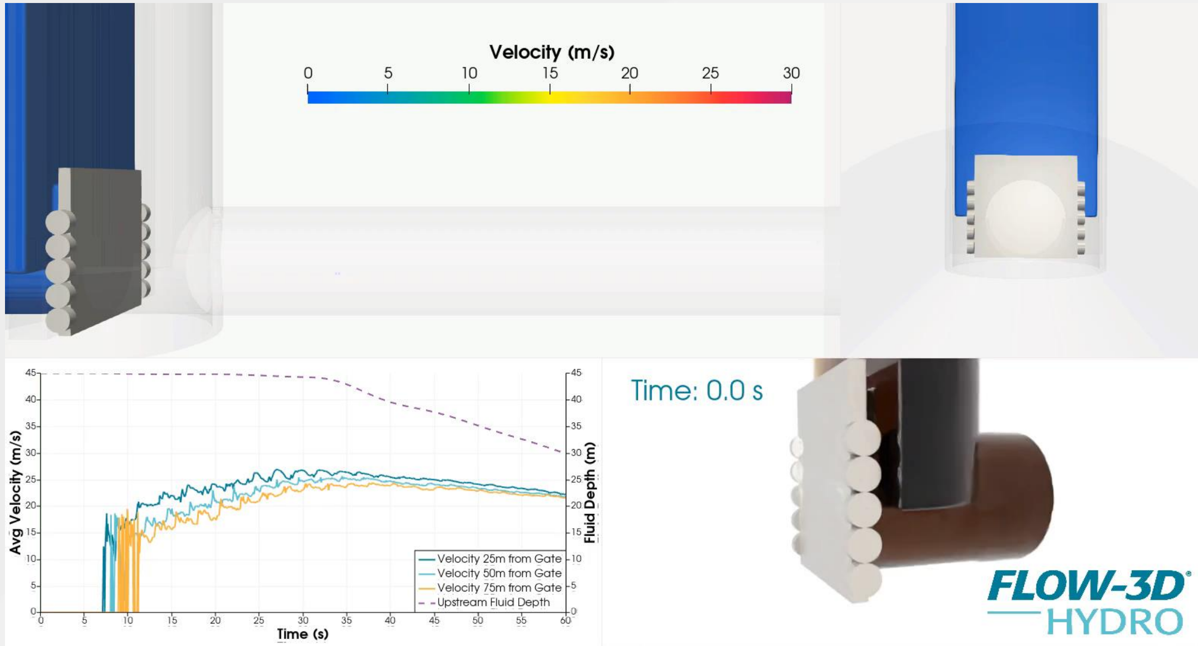
Internals

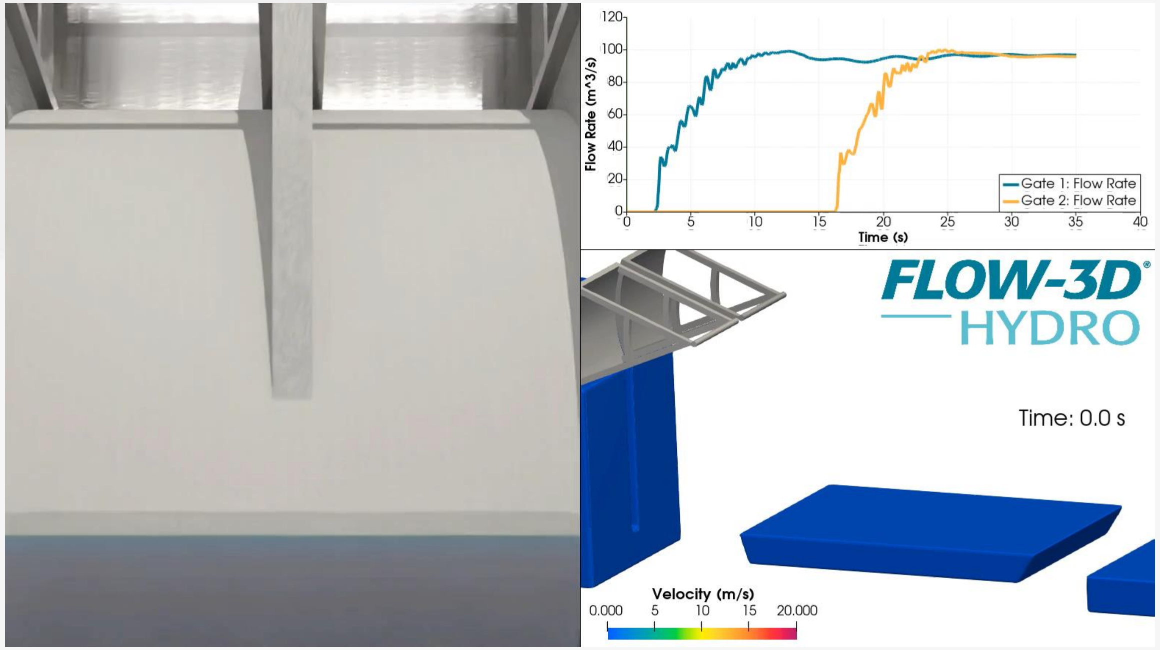


Internals



Locks / Gates





Animation Credit – FLOW3D

Thank You!