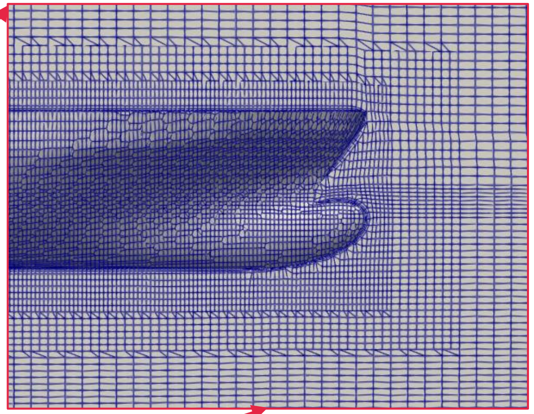
Hydraulic Modelling Using Computational Fluid Dynamics

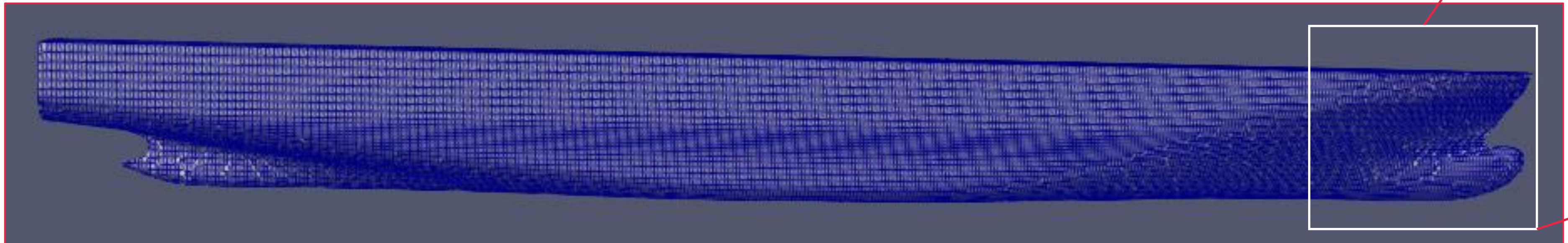
Introduction on OpenFOAM

Ray Shi, Adjunct lecturer,
School of Civil Engineering, UQ
Australia Water School Webinar

Topics to covered today

- ❑ OpenFOAM introduction
- ❑ OpenFOAM Code structures
- ❑ Main solver for large-scale hydraulic modelling – InterFoam
- ❑ Few examples
- ❑ Conclusion





Mesh generated from OpenFOAM tutorial – DTCHull

OpenFOAM introduction

- OpenFOAM == Open **Field of Operation And Manipulation** (OpenFOAM)
- An **open-source** package, built with C++ modules.
- **Linux based** package, based on **Objected Oriented Programming**
- Developed in the 1980s at Imperial College, and now by **OpenCFD Ltd**, ESI Group.
- A set of code libraries for **continuum mechanics** (Solids + Fluids)
- Based on **Finite Volume Method** – cell centred

Prerequisite Skills

- Basic level of Linux interface, C++ coding
- Fundamentals of Fluid Mechanics and Computational Fluid Dynamics (**CFD**)

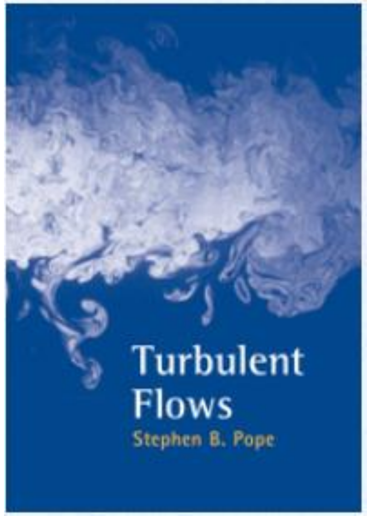
OpenFOAM vs commercial packages

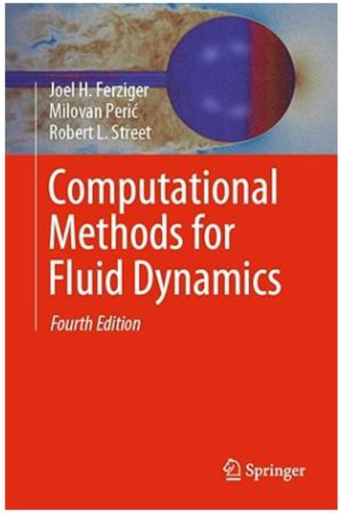
	OpenFOAM	Commercial Packages
1. Module Variety	★ ★ ★ ★ ★	★ ★ ★ ★ ★
2. Learning Difficulty	★ ★ ★ ★ ★	★ ★
3. Source Code	★ ★ ★ ★ ★	★
4. Self Development	★ ★ ★ ★ ★	★
5. Documentation	★ ★ ★	★ ★ ★ ★ ★
6. GUI user friendly	★	★ ★ ★ ★ ★
7. Cost	★ ★ ★ ★ ★	★
8. Code efficiency	★ ★ ★	★ ★ ★ ★ ★
9. Parallel computing	★ ★ ★ ★ ★	★ ★ ★ ★ ★
10. GPU	★	★ ★ ★ ★ ★

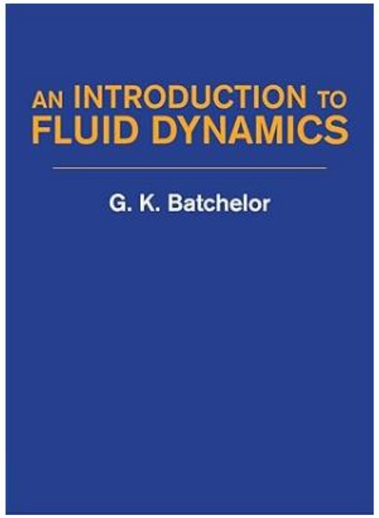
- **Above ranking only represents author's personal opinions !!!**
- In short, OpenFOAM is free to **download**, **use**, **modify** and **distribute**.

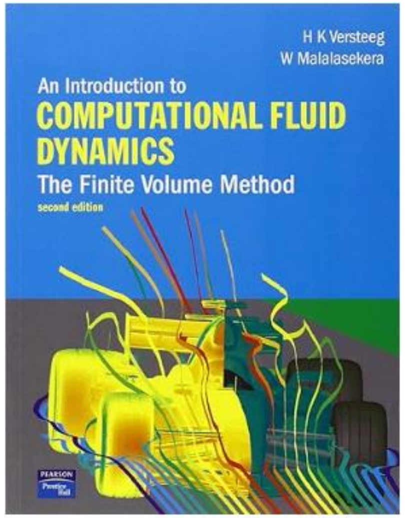
Theoretical background

- ❖ Introduction to fluid dynamics
- ❖ Finite volume method
- ❖ Turbulence theory









Batchelor, G. K. (1967). ***An introduction to fluid dynamics***. Cambridge university press.

Ferziger, J. H., Perić, M., & Street, R. L. (2002). ***Computational methods for fluid dynamics***. springer.

Pope, S. B. (2001). ***Turbulent flows***, *Cambridge University Press, Cambridge*.

Versteeg, H. K. and Malalasekra, W. (1995). **An introduction to computational fluid dynamics the finite volume method** Harlow, Essex, England ; Longman Scientific & Technical :New York : Wiley,.

Workflows

Pre-processing

Pre-processing is mainly related to mesh generation

OpenFOAM functions
- blockMesh
- snappyHexMesh
- foamyHexMesh
- checkMesh
- extrudeMesh
- Other mesh utilities

External Tools
- Gmsh
- enGrid
- SALOME

Modelling

Standard flow-motion solvers:

- Incompressible flow e.g. icoFoam, simpleFoam
- Compressible flow e.g. rhosimpleFoam, sonicFoam
- Multiphase flow e.g. interFoam, twoPhaseEulerFoam

Solution algorithms for N-S equations :
- SIMPLE, PISO and PIMPLE

Turbulence modelling
- RAS, LES, DNS

Post-processing

Post-processing functions has two parts:

Post-processing utilities:
- fieldAverage
- Probes
- codeFunctionObject

External tools
- ParaView
- SALOME
- Gnuplot

Model structures

OpenFOAM model folder structures

<<model>>

- 0 — Setting initial values
  - k — Kinematic energy
  - nut — Turbulent viscosity
  - P — Pressure
  - T — Temperature
  - U — Velocity
  - Other parameters, nuTilda, epsilon
- constant — Constant values
  - transportProperties — Fluid properties
  - turbulenceProperties — Definition of turbulence model
  - polyMesh — Mesh information after model run
- System — Numerical modelling setups
  - blockMeshDict — Control file for **blockmesh** function
  - controlDict — Simulation times and any post-processing functions
  - fvSchemes — Numerical schemes
  - fvSolutions — Numerical solvers, smoothers, tolerance

Solver interFoam – large-scale hydraulic modelling

- For large-scale hydraulic modelling (e.g. **fish passage**, **spillway**, **dam-break**), free surface is important to be captured, and air and water phases are typically considered.
- The InterFoam solves the two incompressible, isothermal immiscible flows (e.g. air and water for most case).
- **Volume of Fluid (VOF)** used to track free surface.

Continuity equation

$$\frac{\partial u_i}{\partial x_i} = 0$$

Momentum equation

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial (\rho u_j u_i)}{\partial x_j} - \frac{\partial \tau_{ei,j}}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \rho g + \sigma \kappa \frac{\partial \alpha_f}{\partial x_i}$$

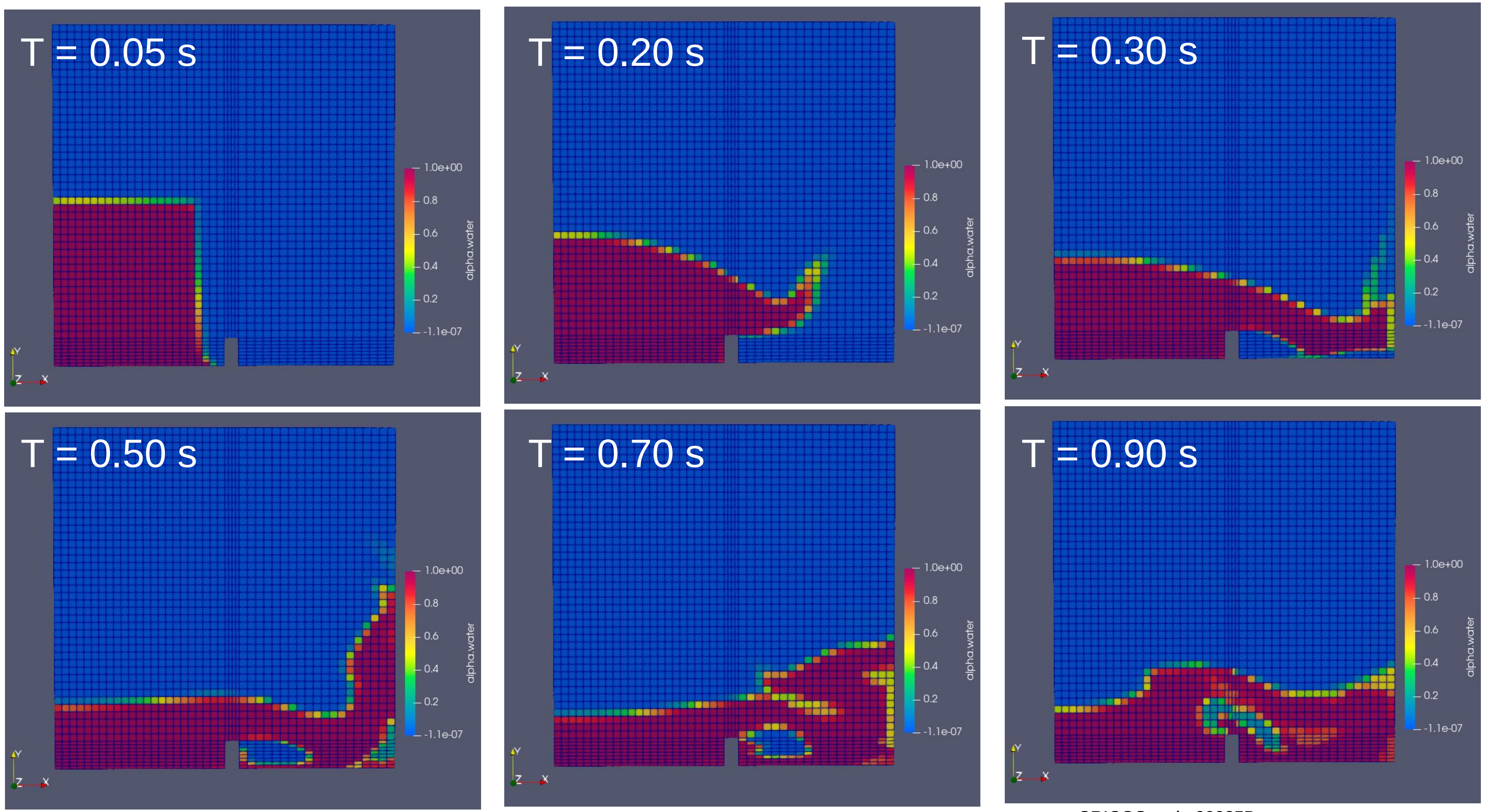
Note:

- α_f is an indicator function between 0 for fluid and 1 for fluid
- κ is the interface curvature, calculated as

$$\kappa = \frac{\partial}{\partial x_i}\left(\frac{\partial \alpha_f / \partial x_i}{|\partial \alpha_f / \partial x_i|}\right)$$

Solver interFoam – large-scale hydraulic modelling

Simple examples on a two-dimensional dam-break type flow

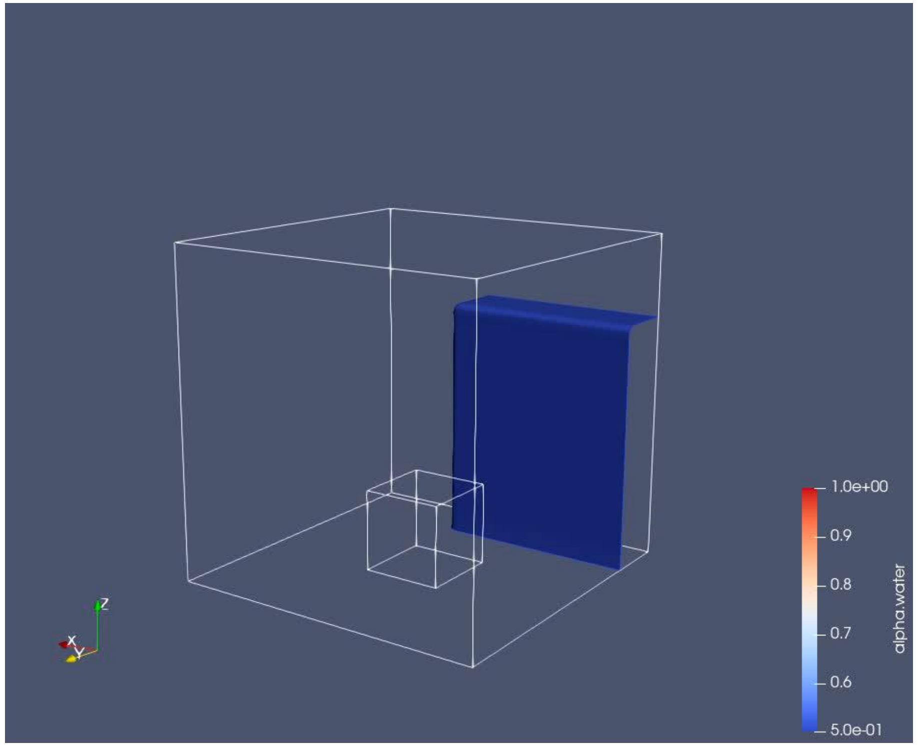


OpenFOAM tutorial – damBreak

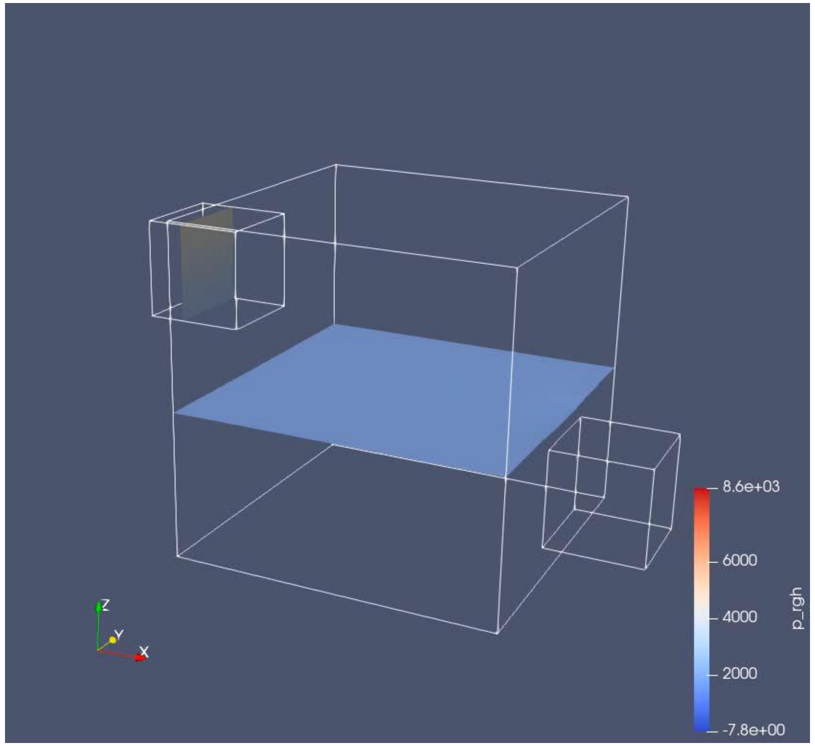
Solver interFoam – large-scale hydraulic modelling

Simple 3d modelling

OpenFOAM tutorial – 3D dam break



OpenFOAM tutorial – iobasin



Solver interFoam – large-scale hydraulic modelling

Complex modelling

- Large scale modelling – breaking bore
- Cross-sectional view of the breaking bore
- Large eddy simulation used
- Mesh size – 2.5 mm
- Large computational resources – HPC