

# CFD Simulation of Free Surface Flows in OpenFOAM

Prof Gavin Tabor

12/11/2017

## CFD at Exeter

Research group : 4 PhD, 2 PDRA, 2 KTP. Working on all aspects of CFD (theory, applications); predominantly using OpenFOAM.

Variety of projects including

- ▶ Tidal Scour behind bridges (EPSRC)
- ▶ Additive Manufacture (InnovateUK)
- ▶ Adjoint Optimisation of vortex separators (KTP)
- ▶ Floating Wind Turbines (UK/China)
- ▶ ML-CFD (EPSRC)

Past projects include Tidal Stream Farm Optimisation, FRMRC-II

# OpenFOAM

OpenFOAM is an Open Source CCM code/code library :

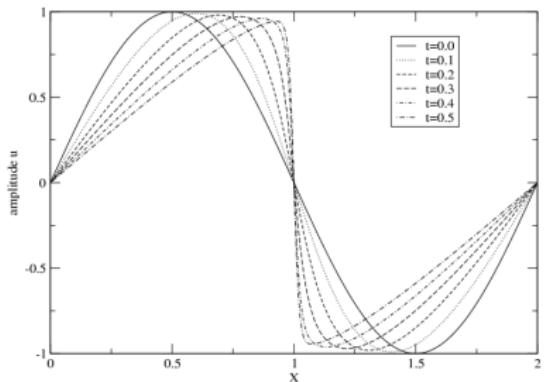
- ▶ Written in C++ – Based on FVM on arbitrary unstructured (polyhedral cell) meshes
- ▶ Fully featured – covers turbulent flow (LES,RANS), multiphase flow (Lagrangian, Eulerian, FSF), combustion, FSI, parallelisation ...
- ▶ Extensive user community – Academic and commercial usage.

OpenFOAM uses the full range of the C++ language – inheritance, polymorphism, templating, operator overloading etc – where appropriate :

Effective result is a high level “language” for encoding CFD.

## Example : Burgers equation

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot \left( \frac{1}{2} \mathbf{u} \mathbf{u} \right) = \nu \nabla^2 \mathbf{u}$$



Implemented as :

```
fvVectorMatrix UEqn
(
    fvm::ddt(U)
    + 0.5*fvm::div(phi, U)
    - fvm::laplacian(nu, U)
);
```

```
UEqn.solve();
```

## Advantages

**Open Source** software development encourages code sharing and information exchange. OpenFOAM's structure and use of C++ is ideal for this and provides a common platform for CFD research.

Basic code designed to make implementing new models simpler (eg. overloading mathematical operators) and more rigorous (physical dimension checking)

Advantages :

- ▶ Complete transparency and code availability (vs. "grey box" approach for commercial codes)
- ▶ Information exchange with other practitioners at a fundamental level
- ▶ CFD use not rationed by license fees.

## VOF modelling

Two types of modelling for free surface flow : interface tracking and interface capturing. *Volume of Fluid* implemented in OpenFOAM.

Solve for single (mixture) fluid

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot \mathbf{u} \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu_t \nabla^2 \mathbf{u}$$

and an *indicator function*  $\alpha$

$$\alpha = \begin{cases} 0 & \text{in air} \\ \text{between 0 and 1} \rightarrow & \text{interface} \\ 1 & \text{in water} \end{cases}$$

Continuity equation for  $\alpha$  :

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot \alpha \mathbf{u} = S_\alpha$$

Given the phase fraction  $\alpha$ , construct mixture properties as

$$\mu_m = \alpha \mu_a + (1 - \alpha) \mu_b$$

Substantial effort involved in compressing the region  $0 < \alpha < 1$  to localise the interface location.

## Other modelling

Also implemented in OpenFOAM :

1. Actuator disk modelling for turbine rotors (several times)
2. Sliding mesh (detailed blade motions)
3. 6 d.o.f modelling/mesh adaption
4. Overset meshing

## Relevant Projects

Several projects involving free surface flows + VOF :

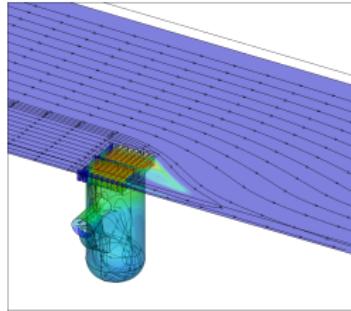
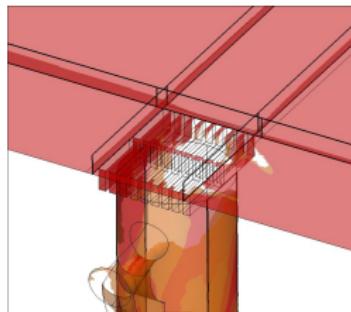
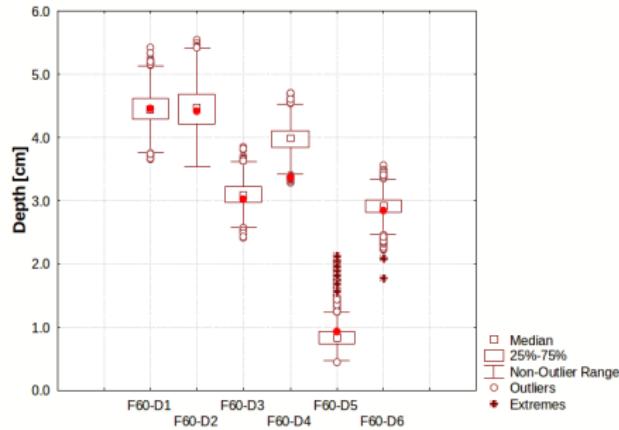
1. FRMRC-II – modelling runoff from road surfaces into gully pots
2. Tidal farm modelling project
3. CCP-WSI – wave-structure interaction
4. Scour behind bridge piers
5. Wind farm optimisation

# FRMRC-II

Study water runoff from road surface into gully pot

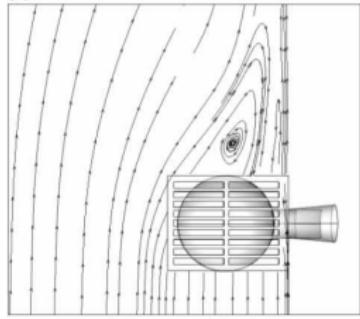
CFD used to extend flood modelling for drainage network model

## Comparison of observed and simulated depths



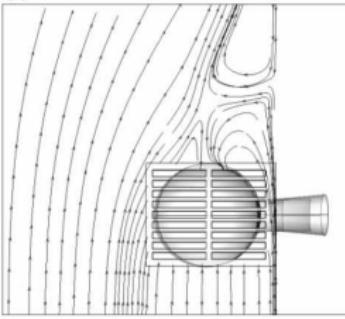


(a)



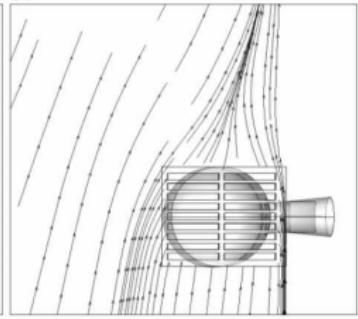
$S_L = 1/200, S_T = 1/200$

(b)



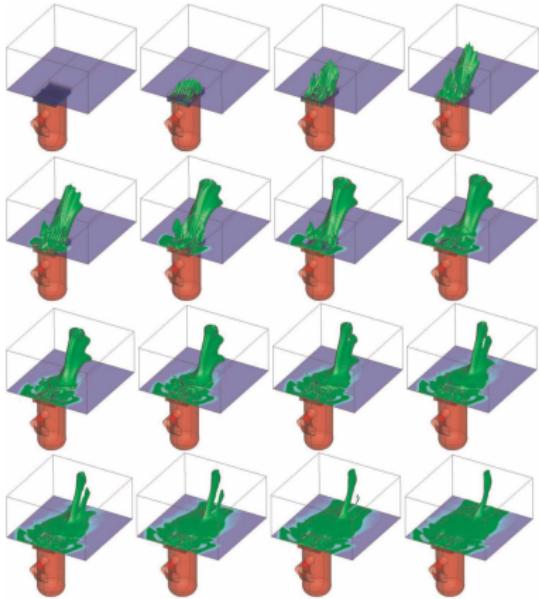
$S_L = 1/200, S_T = 1/60$

(c)

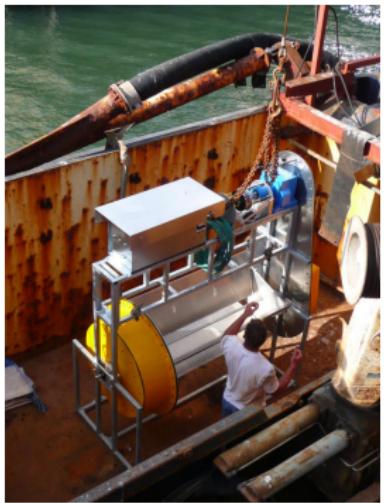


$S_L = 1/200, S_T = 1/20$

# Surcharging

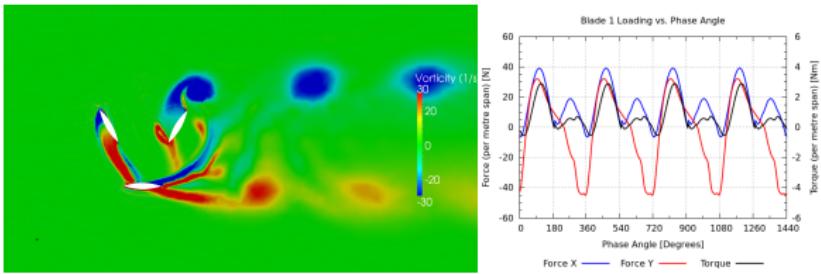


# Tidal turbine project

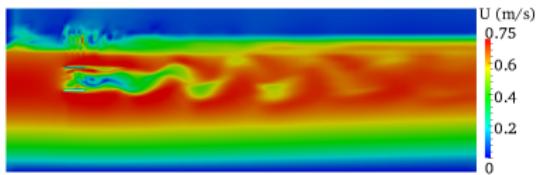
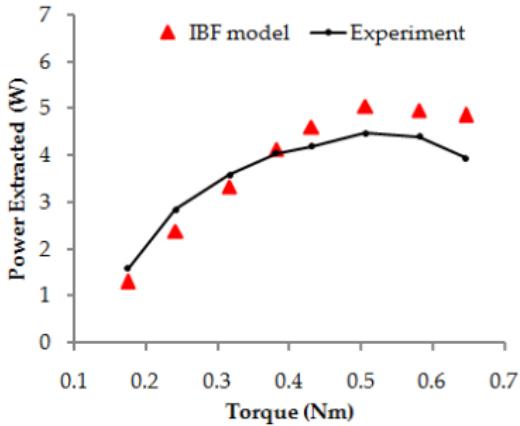
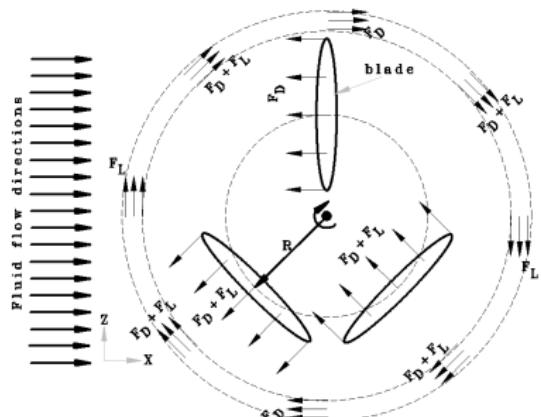


Project investigating estuary-based tidal turbines (+ arrays) based on novel lift/drag design.

Detailed blade modelling and simplified body force model constructed



# IBF Model

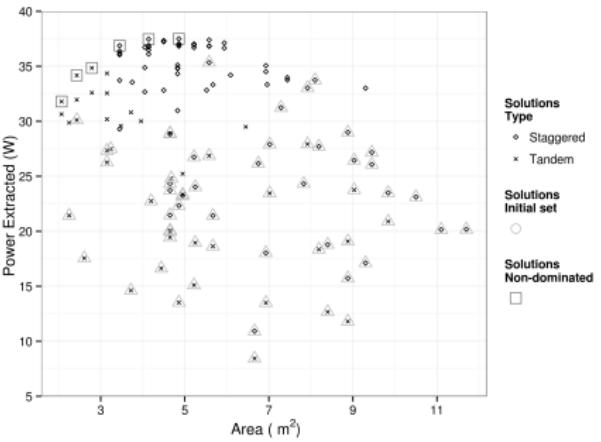
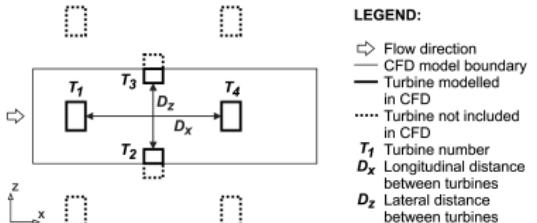
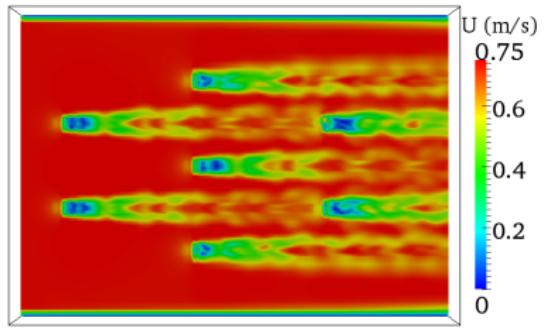


# Farm Modelling

Case	Details	Spec	Time
GGI	160k cells, 30 revolutions	16 cores	5 days
IBF	1 turbine, 148k cells	12 cores	17 hrs
IBF	7 turbines, 1M cells	12 cores	44 hrs

Need to develop *surrogate model* – run 10's of simulations and use *Artificial Neural Network*, *Kriging* to mine results and create correlation.

## Array optimisation carried out using surrogate modelling and GA.



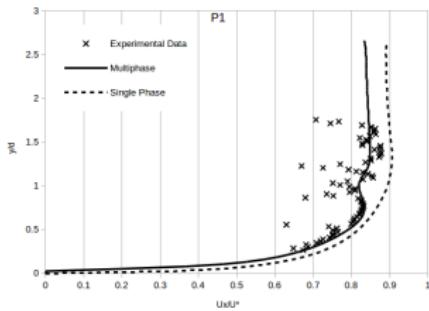
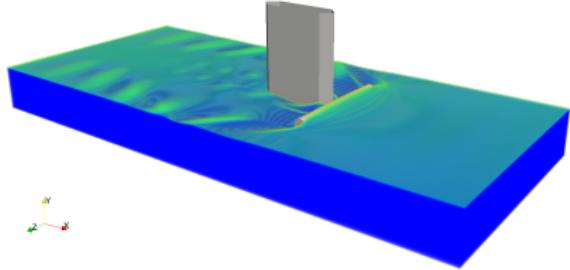
## Bridge Scour project : Motivation

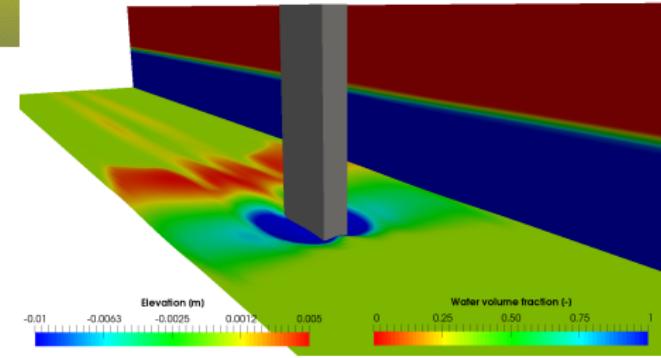
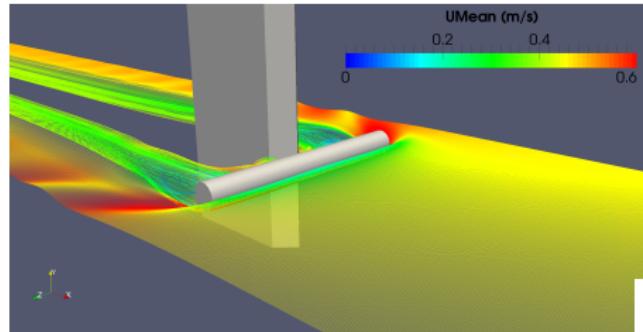


Figure : Rotherham Bridge,  
Devon

- Debris blockages at bridges can change the flow behaviour thus increasing scour and hydraulic loading on the structure.
- Masonry arch bridges are the most commonly used bridge type in the UK.
  - ▶ Form 40% of UK bridge stock, many of which are heritage structures.
  - ▶ Have short span and low clearance increasing the chance of debris accumulation.
  - ▶ Past decade failures (e.g. in Cumbria flood of 2009).
- A pressing need to update UK safety codes to account for the effect of debris blockages on scour.

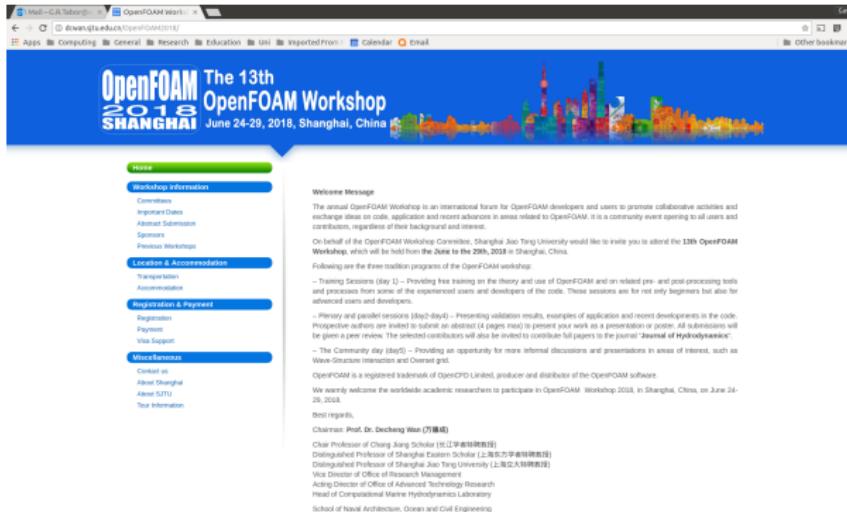
- ▶ Validated VOF modelling of flow around bridge pier
- ▶ Developed scour model using mesh (bed) motion
- ▶ Developing scour + deposition model using kinetic theory





# OpenFOAM Workshop 2018 – Shanghai, June 24-29th –

<http://dcwan.sjtu.edu.cn/OpenFOAM2018/>



The screenshot shows the homepage of the "The 13th OpenFOAM Workshop 2018 SHANGHAI" website. The header features the workshop title, date (June 24-29, 2018), and location (Shanghai, China) against a background of a colorful city skyline silhouette. A navigation menu on the left includes links for Home, Workshop Information, Location & Accommodation, Registration & Payment, and Workshops. The main content area contains a "Welcome Message" about the workshop's purpose and a call for participation. It also lists the three main program components: Training Sessions, Poster and Parallel Sessions, and a Community Day. At the bottom, there is a section for Chinese speakers featuring the names of several key figures.

**Welcome Message**

The annual OpenFOAM Workshop is an international forum for OpenFOAM developers and users to promote collaborative activities and exchange ideas on code, application and recent advances in areas related to OpenFOAM. It is a community event open to all users and contributors, regardless of their background and interest.

On behalf of the OpenFOAM Workshop Committee, Shanghai Jiao Tong University would like to invite you to attend the 13th OpenFOAM Workshop, which will be held from the June 24 to 29th, 2018 in Shanghai, China.

Following are the three main program of the OpenFOAM workshop:

- Training Sessions (day 1) – Providing free training on the theory and use of OpenFOAM and on related pre- and post-processing tools and processes. Many some of the experienced users and developers of the code. These sessions are for not only beginners but also for advanced users.
- Poster and parallel sessions (days 2-4) – Presenting validation results, examples of application and recent developments in the code. Prospective authors are invited to submit an abstract (4 pages max) to present your work as a presentation or poster. All submissions will be given a peer review. The selected contributors will also be invited to contribute full papers to the journal "Journal of Hydrodynamics".
- The Community day (day 5) – Providing an opportunity for more informal discussions and presentations in areas of interest, such as Wave-Surface Interaction and Oceanic grid.

OpenFOAM is a registered trademark of OpenCFD Limited, producer and distributor of the OpenFOAM software.

We warmly welcome the worldwide academic researchers to participate in OpenFOAM Workshop 2018, in Shanghai, China, on June 24-29, 2018.

Best regards,

Chairman Prof. Dr. Decheng Wu (吴德成)  
Chair Professor of Chung Liang School (竺江良副校长)  
Distinguished Professor of Shanghai Jiao Tong University (上海交通大学特聘教授)  
Vice Director of Office of Research Management  
Acting Director of Office of Advanced Technology Research  
Head of Computational Marine Hydrodynamics Laboratory  
School of Naval Architecture, Ocean and Civil Engineering

# Thank you for listening

## Any questions?

# Papers

"Experimental and numerical investigation of interactions between above and below ground drainage systems", S. Djordjevic, A.J. Saul, G.R. Tabor, J.Blanksby, I. Galambos, N. Sabtu, G. Sailor. *Water Science and Technology* **67**(3) pp. 535 – 542 (2012).

"CFD Simulations for Sensitivity Analysis of Different Parameters to the Wake Characteristics of a Tidal Turbine", M.G.Gebreslassie, G.R.Tabor, M.R.Belmont, *Open Journal of Fluid Dynamics* **2** pp. 56–64 (2012).

"Numerical modelling of a new class of cross flow tidal turbine using OpenFOAM I:calibration of energy extraction", M.G.Gebreslassie, M.R. Belmont, G.R.Tabor, *Renewable Energy Journal* **50** pp.994-1004 (2013).

"Numerical modelling of a new class of cross flow tidal turbine using OpenFOAM II: investigation of turbine to turbine interaction", M.G.Gebreslassie, M.R. Belmont,G.R.Tabor, *Renewable Energy Journal* **50** pp.1005-1013 (2013).

"Investigation of the performance of a staggered configuration of tidal turbines Using CFD", M.G. Gebreslassie, G.R.Tabor, M.R.Belmont *Renewable Energy Journal* **80** pp. 690–698 (2015).

"Comparison of multiple surrogates for 3D CFD model in tidal farm optimisation, W. Moore", H. Mala-Jetmarova, M. Gebreslassie, G. Tabor, M. Belmont, D.Savic, *Procedia Engineering* **154** pp. 1132–1139 (2016).

"The impact of tidal energy extraction in estuaries: analysis of the influence of channel geometry". M. Garcia-Oliva , S. Djordjevic, G. R. Tabor,*Renewable Energy Journal* **101** pp.514 – 525 (2017)