Application of OpenFOAM to Modelling Hydraulic Structures

Gavin Tabor[1]

[1]College of Engineering, Maths and Physical Sciences, University of Exeter, Harrison Building, North Park Road, Exeter EX4 4QF

E-mail: g.r.tabor@ex.ac.uk

ABSTRACT: OpenFOAM is a fully-functioned Open Source Computational Continuum Mechanics (CCM) code library written in C++ and originally developed for Computational Fluid Dynamics (CFD). Technically it is a C++ class library of classes covering all aspects of CFD; mesh manipulation, tensor field algebra, calculus using the Finite Volume method and solution of implicit partial differential equations. At the top level it is designed to provide a `pseudo-mathematical' syntax for modeling PDE's which can be treated as a high level programming language for writing CFD (and more generally CCM) codes. In addition the standard distributions contain pre-written codes solving most standard CFD problems (including turbulent flow, free surface and multiphase flow) and it is perfectly possible to use it as a 'black box' code with capabilities similar to or exceeding those of commercial multipurpose codes. In this Keynote presentation I will outline the capabilities and key features of OpenFOAM and present key benefits of using OpenFOAM in both academic and industrial environments. Following on from this I will present results from academic and industrial projects in the water systems area, including work on simulation of vortex flow controls, runoff from roadways into gullys, and air entrainment in stepped spillways; and look at future work at Exeter using OpenFOAM to investigate hydrodynamic scour behind bridges.

Keywords: Computational Modelling, Computational Fluid Dynamics, Open Source software