

Object-Oriented CFD: Free Surface Flow Solver

April 7 2005

Dr. Hrvoje Jasak

Wikki Ltd, United Kingdom

E-mail: h.jasak@wikki.co.uk

Web: <http://www.h.jasak.dial.pipex.com>

Web: <http://www.openfoam.org>

Abstract

Continuing improvements in numerics, decreasing price/performance of computer hardware and deeper understanding of physics is opening new areas for numerical modelling in Computational Continuum Mechanics. However, increasing complexity on the modelling side poses new challenges both in terms of understanding the complexity and handling model implementation and model-to-model interaction in software.

This presentation describes **FOAM (Field Operation And Manipulation)**, a numerical simulation package for continuum mechanics designed to answer the complex physics demands. The package is designed to allow easy, quick and reliable implementation of physical models by mimicking the form of partial differential equations in the software. Object orientation which is a core of the new approach, naturally leads to code re-use and layered approach to development and software validation. More importantly, code separation between abstract interfaces and run-time loadable implementation eliminates the need for traditional “user-coding” and opens the software for user-defined extension, not only in terms of considered equation sets, but also for various discretisation elements, differencing schemes, boundary conditions, solver technology and physical models.

FOAM implements several Continuum Mechanics modelling paradigms (Finite Volume, Finite Element, Lagrangian particle tracking) in library form, handles complex geometries through polyhedral mesh support and allows for close coupling between various models.

In the second part, capabilities of FOAM will be illustrated on free surface flow modelling, including both the **surface capturing** approach capable of handling dominant surface tension and a **surface tracking** solver built around an automatic vertex-based mesh motion algorithm.