*International Workshop on* Coupled Methods in Numerical Dynamics
*IUC, Dubrovnik, Croatia, September 19th-21st, 2007*

# OpenFOAM: A C++ Library for Complex Physics Simulations

Hrvoje Jasak[1,2,*], Aleksandar Jemcov[3], Željko Tuković[2]

[1] Wikki Ltd, United Kingdom
[2] Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Croatia
[3] Development Department, Ansys/Fluent Inc, USA

**Abstract.** This paper describes the design of OpenFOAM, an object-oriented library for Computational Fluid Dynamics (CFD) and structural analysis. Efficient and flexible implementation of complex physical models in Continuum Mechanics is achieved by mimicking the form of partial differential equation in software. The library provides Finite Volume and Finite Element discretisation in operator form and with polyhedral mesh support, with relevant auxiliary tools and support for massively parallel computing. Functionality of OpenFOAM is illustrated on three levels: improvements in linear solver technology with CG-AMG solvers, LES data analysis using Proper Orthogonal Decomposition (POD) and a self-contained fluid-structure interaction solver.

**Key words:** *Object oriented, C++, Scientific computing, FSI, Fluid-structure interaction, POD, Proper orthogonal decomposition, Iterative solver, CG-AMG.*

## 1. Introduction

Expansion of Computational Continuum Mechanics (CCM) in engineering design is mirrored in the maturity of commercial simulation tools in the market. In terms of numerical techniques, structural analysis is dominated by the Finite Element Method (FEM), while fluid flow is regularly handled using the Finite Volume Method (FVM) of discretisation. Leading software combines accurate and robust numerics with an impressive range of physical models under a general-purpose Graphical User Interface (GUI). Current simulation challenges are related to integration and automation of simulation tools in a Computer Aided Engineering (CAE) environment, including automatic geometry retrieval, surface and volume meshing and sensitivity and optimisation studies.

In terms of solver settings and user expertise, Computational Fluid Dynamics (CFD) is considerably behind structural analysis, making the problems of software development and usability more acute. Range and quality of physical

*Correspondence to: Hrvoje Jasak, E-mail: h.jasak@wikki.co.uk.