Numerical Modelling in Continuum Mechanics

22nd March 2005

Dr. Hrvoje Jasak
Wikki Ltd, United Kingdom
E-mail: h.jasak@wikki.co.uk
Web: http://www.openfoam.org
Web: http://www.h.jasak.dial.pipex.com

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. Looking beyond well-established areas of structural analysis and fluid flow simulations, new interests include a variety of constitutional laws as well as complex inter-equation and inter-domain coupling. However, increasing complexity on the modelling side poses new challenges both in terms of understanding the complexity and handling model implementation 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. Moreover, object orientation which is a core of the new approach, naturally leads to code re-use and layered approach to development and software validation. FOAM implements several 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.

A combination of easy model implementation and layered design provides a new perspective in numerical continuum mechanics. Capabilities of FOAM will be illustrated over a range of complex continuum models including fluid flow, combustion, free surface flows, non-linear stress analysis and solid-fluid interaction.