11th. World Congress on Computational Mechanics (WCCM2014)
5th. European Conference on Computational Mechanics (ECCM V)
6th. European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

COMPUTATIONAL CONTINUUM MECHANICS WITH OPENFOAM™

G.TABOR* AND G.MONTENEGRO†

* College of Engineering, Maths and Physical Sciences, University of Exeter
Harrison Building, North Park Road, Exeter EX4 4QF UK
g.r.tabor@ex.ac.uk

† Department of Energy, Politecnico di Milano University
Campus Bovisa, Via Lambruschini, 4, 20156 Milano, Italy
gianluca.montenegro@polimi.it www.engines.polimi.it

Key words: Computational Continnum Mechanics - Computational Fluid Dynamics – Open Source Software

ABSTRACT

OpenFOAM™ represents a new departure from the traditional model of commercial, pay-per-use, grey-box engineering codes. Released by OpenCFD in 2004, and based on the academic code FOAM developed in the 1990's at Imperial College, it is a complete, fully functional general purpose Computational Continuum Mechanics (CCM) solution released under the GNU General Public License. Rather than a single monolithic code, it is in fact a C++ code library of classes for developing CCM codes, including classes for tensor field manipulation, implicit and explicit numerical solution of matrix equations derived from the finite volume formulation, related mesh handling and matrix inversion classes, and a wide range of physical models for turbulence, combustion and reaction, mesh motion, and so forth. The canonical release provided by OpenCFD contains numerous pre-written apps to solve common CFD and CCM problems such as incompressible steady state turbulent flow (simpleFoam) together with mesh generation, preprocessing and post-processing apps written within the OpenFOAM framework. Code users can use these, or develop their own apps; under the Open Source license even the base library can be modified if necessary. The result has been a rapidly-growing user base numbering in the 10's of thousands who are able to collaborate at a very fundamental level using the OpenFOAM library and classes as a high level language for exchanging ideas; a sort of Matlab for CCM. Whilst there is undoubtably a steep learning curve for the code, since it is free it has enormously expanded the use of CCM (at a personal level anyone can download it and use it; whilst at an institutional or company level, use is restricted by the hardware available, not by licensing costs). The Minisymposium will showcase use of OpenFOAM in academic and commercial applications and demonstrate some of the cutting edge research being undertaken by its users.