Computational Modelling with OpenFOAM(TM)

TRACK Number 2000

Gavin R Tabor\*, FRED MENDONCA†

\* CEMPS, University of Exeter

Harrison Building, North Park Road, Exeter, EX4 4QF, UK

g.r.tabor@ex.ac.uk

† Managing Director, OpenCFD Ltd,

Unit 6, The Courtyard, Eastern Road, Bracknell, RG12 2XB, UK

fred.mendonca@esi-group.com

**Key words:** Computational Fluid Dynamics, Open Source, Computational Modelling

ABSTRACT

OpenFOAM is probably the most used Open Source CFD code in use at the moment and one of the most commonly used CFD tools in research and industry. Initially developed (as Field Operation And Manipulation) in the 1990’s [1], its open source release has resulted in phenomenal growth in its use and applications in a wide range of fields of CFD and Computational Continuum Mechanics more widely. Based on the finite volume methodology on arbitrary polyhedral meshes, its object oriented design makes it an ideal platform for research in CFD methodology, and numerous toolkits for specific modelling projects have also been developed by members of the worldwide community (such as IHFoam[2], fireFoam[3], mdFoam[4], to name just a few).

The objective of the minisymposium is to showcase and disseminate all aspects of research using OpenFOAM, from fundamental developments of the code through to applications in science, technology and medicine. Contributions using any branch of the code are welcome and we encourage the open release of code developments and online archival of any data presented (space for this will be provided on the community Wiki at <https://wiki.openfoam.com/Main_Page>).

**REFERENCES**

1. H.G. Weller, G.Tabor, H.Jasak & C.Fureby, “ATensorial Approach to CFD using Object Orientated Techniques”, *Computers in Physics*. Vol **12#6** pp. 620 – 631 (1998).
2. P. Higuera, J. L. Lara, & I.J. Losada, “Realistic Wave Generation and Active Wave Absorption for Navier-­Stokes Models. Application to OpenFOAM®”, *Coastal Engineering* Vol. **71**, pp. 102-118 (2013).
3. Y. Wang, P. Chatterjee and J.L. de Ris, “Large Eddy Simulation of Fire Plumes”, *Proc. Combustion Institute* Vol **33** pp. 2473 – 2480 (2011).
4. S.M. Longshaw, M.K. Borg, S.B. Ramisetti, J. Zhang, D.A. Lockerby, D.R. Emerson and J.M. Reese, “mdFoam+: Advanced molecular dynamics in OpenFOAM”, *Comput.Phys.Comm*., Vol **224** pp. 1-21 (2018).