OpenFOAM: A Year in Review

Prof. Hrvoje Jasak
Wikki, United Kingdom
University of Zagreb, Croatia

With every year, the continued expansion on the number of users, quality of work and contributions and results of extensive validation studies continues to impress my both privately and in my professional work. In my opinion, OpenFOAM Community and the spirit of Open Source development, sharing of results and experiences is a critical driver for this expansion.

In this talk, an overview of the development, validation and community activities in the OpenFOAM community will be given. The main topics include:

- New developments in the upcoming release of `foam-extend-4.0` and notable community developments:
  - Major performance improvement for the coupled implicit p-U solver;
  - Numerics: Consistency improvements in the pressure-velocity coupling (`ddtPhiCorr`); Accuracy of forces and pressure field fluctuation in transient sliding mesh/GGI;
  - Linear solver improvements for segregated and block-coupled systems, including ILU-Cp preconditioning with user-defined fill-in level;
  - Feature completion and performance improvement for compressible turbomachinery applications: rothalpy equation, interface jump conditions, density-based solver upgrade with run-time selection, tutorials and validation;
  - Real gas thermodynamics and steam properties library;
  - Major upgrade of the fluid-structure interaction library;
  - General performance improvement features, including gradient cacheing on static and moving meshes;
- Code certification exercise for Naval Hydrodynamics CFD: Tokyo 2015 Workshop;
- Update on developments of the dynamic Immersed Boundary and dynamic Overset Mesh capability;
- Harmonic balance solver: implementation and results;
- Unified development line of the source code, natively compiling on Linux, Mac OS X and Microsoft Windows. Raspberry Pi port;
- Efforts in bringing various forks of the OpenFOAM project under a joint umbrella: unified input format and cases portable between different forks.

The presentation will be completed with a brief update on community projects and local User Groups as centres of activity and collaboration.

Hrvoje Jasak graduated at the University of Zagreb, Croatia in 1992 and earned his PhD in numerical modelling and Computational Fluid Dynamics (CFD) at Imperial College London in 1996. During his PhD he has started developing FOAM, released today as the leading Open Source CFD package under the name OpenFOAM.
With the open source release of OpenFOAM, Prof. Jasak has founded Wikki Ltd in 2004 to provide industrial consulting, support and custom development services with OpenFOAM. Starting in 2003, prof. Jasak is working at University of Zagreb as professor at the Chair of Turbomachinery, where he runs a CFD research group.