Getting started with OpenFOAM using the tutorial collection at wiki.openfoam.com

Jozsef Nagy[1], Andrew Heather[2], J. Miguel Nobrega[3]

[1]eulerian-solutions e.U., jozsef.nagy@eulerian-solutions.com, Linz, Austria (presenter)

[2] OpenCFD Ltd, Bracknell, UK (co-author)

[3] Institute for Polymers and Composites, University of Minho, Guimaraes, Portugal (co-author)

OpenFOAM is a widely used computational toolkit with applications spanning industry and academia. Since release by OpenCFD in 2004 its range of functionality and user base has grown dramatically, and yet it is still perceived as challenging use.

Getting started is usually the first major difficulty faced by new users since the learning curve can be daunting. Users coming from GUI-driven workflows, or without a lot of experience with the Linux command line, often face challenges that feel very difficult to overcome when attempting their first simulations. Moreover, with so much disparate information available on the Internet that lacks of organization, e.g. the actual version/release of OpenFOAM is not described, or offering conflicting advice, frustration can quickly set-in.

To ease beginners onto their OpenFOAM journey, a site hosting a collection of tutorials was launched at wiki.openfoam.com/tutorials in September 2016 by a group of OpenFOAM enthusiasts, with the full support of OpenCFD Ltd. The idea was to create and maintain a governing structure to guarantee an up-to-date and organized collection of tutorials for beginners. This led to the creation of a Board of Editors with many years of experience in teaching and training with OpenFOAM to guide the efforts. Since 2017, twice per year, the collection is updated to the latest versions of OpenFOAM (official release, openfoam.org release, and foam-extend release).

The best place to start is the "First glimpse" series (https://wiki.openfoam.com/%22first_glimpse%22_series). Here, users can quickly identify whether the OpenFOAM toolkit is suitable for their needs. For users wanting to learn the basic and advanced features of OpenFOAM, the "3-weeks-series" (https://wiki.openfoam.com/index.php?title=%223_weeks%22_series) offers a wide variety of learning topics, appropriate for individuals without previous experience.

In addition users can search the collection by topic (https://wiki.openfoam.com/Collection_by_topic) or by contributor (https://wiki.openfoam.com/Collection_by_authors) to find the tutorials that best fit their needs.

In the last 4-5 years the collection has shown sustainable growth of up-to-date tutorials, and is now part of the Technical Committee for Documentation and Tutorials in the Governance Structure of OpenFOAM.