

EDITORIAL

The OpenFOAM[®] transport equation started in the late 1980s, with a source term provided by very bright minds. The computational library was advected for several years, after which, knowing that diffusion is very important for stability purposes and to help avoid unwanted divergences, the appropriate terms were added to the equation, by making it open-source.

Nowadays, the advantages and freedom promoted by OpenFOAM[®] are widely recognized, but it is also clear that its perseverance requires efficiently combining the advances promoted by source terms with the dissemination assured by a proper balance of the transport promoted by advection, and the homogenization granted by diffusion. In fact, the large number of high quality source terms available at different locations of the currently large computational mesh, must be both advected and diffused to maximize its impact.

The current OpenFOAM[®] Governance seeks to provide additional support for promising source terms and is also engaged in several efforts to maximize OpenFOAM[®] advection and diffusion. The OpenFOAM[®] Journal, an initiative from the Technical Committee for Documentation and Tutorials, is an example of one of those efforts. The Journal aims at contributing to an enhanced, rigorous and stable transport of OpenFOAM[®].

The Journals Editorial Team consists of a group of enthusiasts who put forth their best efforts to have a simple but fully functional journal, which we hope will both serve and involve everyone from the multiphase OpenFOAM[®] computational domain. Its success will allow for the enlargement of domain boundaries, an increase in the order of dissemination and continued advancement towards the desired convergence.

We must acknowledge the strong support of the OpenFOAM[®] Governance Committee, OpenCFD and all other contributing parties, whose support was crucial in order to materialize this project.

Looking forward to the participation of all the FOAMers.

Miguel Nóbrega and József Nagy