Review of Recent and Ongoing Developments of the OpenFOAM library

H.Rusche[1]

In this talk, an overview of the major developments and community contributions to the OpenFOAM library will be presented. Recently, a number of notable contributions has been made by the community, enhancing the software structure, physical modelling capability and numerical underpinnings of the library.

In base numerics, the block-matrix solver has been applied and validated on new applications, including strongly coupled n-phase free surface VOF solver. The block matrix is now complete with parallelisation. In the future work, the block matrix will be used to replace the existing linear solver classes in the library.

The thin liquid film solver has been used in a number of projects and is now considered mature. Its performance tuning, validation and deployment is complete; furthermore, the Finite Area discretisation that underpins it has proven to be very versatile.

Substantial improvements in dynamic mesh handling have come from multiple sources. The RBF mesh motion technique proves to be versatile and efficient: in the next period, it needs to be tested on a wider application area. The ”missing link” of automatic dynamic tet-remeshing using 3-D edge swapping has been contributed: its power and elegance warrants a complete re-think of dynamic mesh features currently in use. Along the similar lines, the work on Immersed Boundary Method and Overlapping Grid is ongoing. This would make OpenFOAM a true powerhouse for the dynamic mesh cases.

The list of new developments is not comprehensive and certainly does not give credit to all contributors (my apologies). One can only conclude that the vigorous community effort makes for fascinating and complementary code developments and making OpenFOAM better for all of us.

[1]WIKKI GmbH, Görgestraße 24, 38118 Braunschweig, Deutschland