OpenFOAM® in Polymer Processing Applications et al.

J. Miguel Nóbrega

i3N/IPC - Institute for Polymers and Composites/, University of Minho, Campus de Azurém, Guimarães, Portugal

Nowadays, aiming an efficient use of resources, the advantages of employing numerical modelling tools to assist any design activity are consensual. Therefore, there are countless modelling software packages available, both commercial and open-source, which can be used to study extremely complex systems.

In the past, the open-source numerical codes presented very specific/limited capabilities and were developed by small research teams. Nowadays, some communities that develop and work with open-source codes are substantially enlarged and better organised. Hence, given the large range of skills covered by the development teams, some of these open-source codes cover a very wide range of applications. One successful case of this type is the OpenFOAM® computational library, which can model complex multiphysics systems, covering both fluid and solid behaviours, being fully parallelized. One of the main advantages of OpenFOAM® is its adaptiveness, resorting to a symbolic programming language, that allows a straightforward development of new applications for the resolution of specific problems, not available on the distributed releases.

This talk will begin with a brief presentation of the OpenFOAM® computational library, its main features and capabilities. Then, several ongoing/recent case studies in the frame of polymer processing and related applications will illustrate its potential.