OpenFOAM Training: Combustion

3-5 July 2017, Brussels

Apply OpenFOAM to your combustion case and unlock its various functionality.

This course consists of various practical lectures and hands-on tutorials organised over three days:
Monday morning (optional):
   Crash course on programming in OpenFOAM

Monday afternoon: The combustion models in OpenFOAM
   How to use them
   How to modify them or implement new ones

Tuesday morning: Chemistry acceleration
   How to configure ISAT for various cases
   How to use mechanism reduction techniques
   How to implement new ones

Tuesday afternoon: Use of external libraries for chemistry
   Getting started with OpenSMOKE++

Wednesday morning: Practical cases
   Based on request from participants, reference turbulent flames and autoignition in rapid compression machine

Venue: Brussels (more details communicated soon)
Registration: https://goo.gl/forms/0xRsjovt2kLv0rh22