

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/0xRsjozt2kLv0rh22>



Combustion for
Low Emission
Applications of
Natural Gas



POLITECNICO
MILANO 1863



TECHNISCHE
UNIVERSITÄT
DARMSTADT