Best practices guidelines for CFD of turbulent combustion
Including hydrogen combustion, emission modeling, spray atomization modeling and machine learning tools

SALC 5, Seminar and Learning centre, Imperial College, London

Tuesday, December 11, 2018

Day 1: Best practices guidelines for CFD of turbulent combustion

8:30	**Registration and welcome**	
9:00	**Turbulent combustion modeling**	**Luc Vervisch**
10:00	**Best practice for model validation in LES (and RANS) I**	**Andreas Kempf**
10:30	**Refreshments**	
11:00	**Best practice for model validation in LES (and RANS) II**	**Andreas Kempf**
12:00	Discussion	
12:30	**Lunch**	
13:30	**Best practices in CFD of thermal radiative heat transfer**	**Dirk Roekaerts**
14:30	**Chemistry reduction for CFD**	**Luc Vervisch**
15:30	**Refreshments**	
16:00	**Artificial Neural Networks for chemistry tabulation**	**Stelios Rigopoulos**
17:00	**Final discussion**	
17:30	**Close**	

Evening: course dinner

Wednesday, December 12, 2018

Day 2: CFD for emission modeling and emerging methods

8:30	**Registration and welcome**	
9:00	**Detailed simulation of spray atomization**	**Salvador Navarro-Martinez**
10:00	**Fundamentals of soot modelling: chemical kinetics and aerosol dynamics**	**Stelios Rigopoulos**
11:00	**Refreshments**	
11:30	**Soot modelling in CFD of turbulent combustion**	**Stelios Rigopoulos**
12:30	**Lunch**	
13:30	**Hydrogen combustion**	**Dirk Roekaerts**
14:30	**Mini workshop on combustion CFD applications:** Participants and lecturers are invited to give a short presentation on a combustion CFD application using the CFD tools of their interest, with emphasis on challenging issues. The discussion will focus on which best practices can be identified for the presented cases.	Chairman tbd
16:00	**Final conclusions and closure**	

Background and objectives:

Design and operation of modern combustion systems (gas turbine engines, IC engines, process furnaces) faces the need to combine high efficiency with low pollutants emissions.
Computational Fluid Dynamics has become a powerful tool in design of these systems.
Many numerical models exist, each having a range of applicability, computational cost and accuracy.
Consequently, CFD experts involved in combustor simulations, in addition to usual CFD skills, need specific insight and knowledge in combustion, heat transfer and emission modelling in order to conduct thorough analysis. They must be able to respond to societal demands (e.g. larger role for hydrogen as fuel) or opportunities from other fields (data science, machine learning).
The present course addresses this need.

The participants will learn the best practices in CFD of combustion systems.
They will discover how to select models, how to validate numerical simulations, and which accuracy to expect.
Interactions between fuel injection, turbulence, heat release and thermal radiation are critical in determining flame structure and pollutant emissions, and a major part of the course is devoted to them.

The lectures of this course, all by well-known experts in the field, cover from basics to applications.

The course is partially based on the ERCOFTAC Best Practice Guide on CFD of combustion, a copy of which will be provided to the participants.

In the course also the link will be made with the CFD programs and cases of interest for the participants.
As a result, the course provides the means for CFD analysts to significantly enhance their use of commercial and open-source CFD software for combustion engineering applications.

Lecturers:

Prof. Andreas Kempf, University Duisburg-Essen, Germany
Dr. Salvador Navarro-Martinez, Imperial College London, United Kingdom
Dr. Stelios Rigopoulos, Imperial College London, United Kingdom
Prof. Dirk Roekaerts, Delft University of Technology, The Netherlands

Prof. Luc Vervisch, National Institute of Applied Sciences, Rouen Normandy University, France