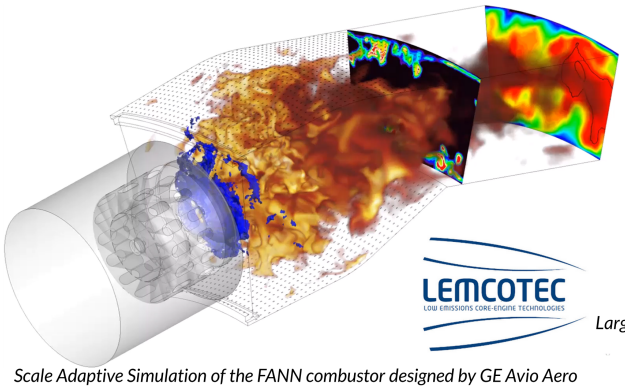
Numerical Investigations of turbulent spray flames for aeronautical applications

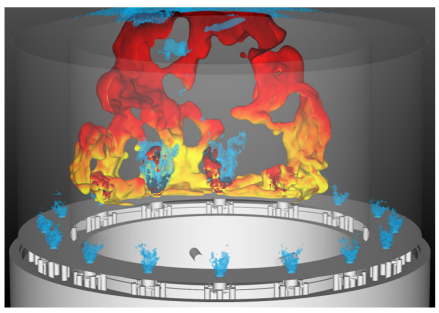
Dr. Stefano Puggelli

Friday, June 28th 2019 - 14h - Amphi V

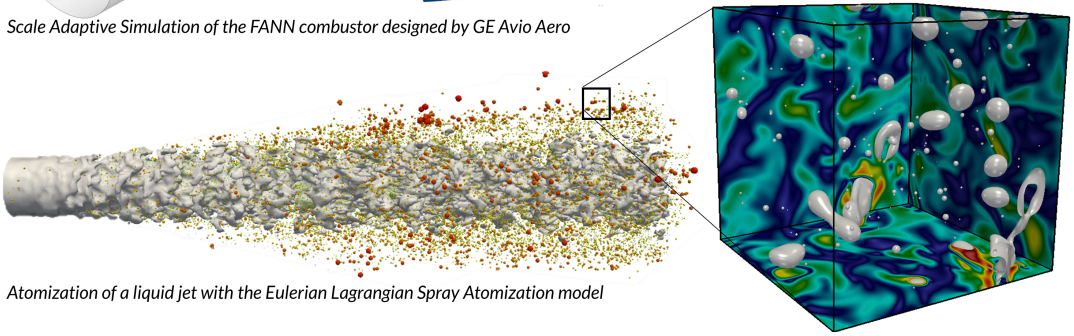
Over the last years, aero-engines are progressively evolving towards design concepts that improve engine safety, fuel economy and pollutant emissions. In this context, lean burn technology has been recognized as a reliable solution even if it must face safety risks and technical issues. A deep insight on lean burn combustion is required and Computational Fluid Dynamics (CFD) is a powerful tool for this purpose. This presentation aims at presenting an overview of the numerical research works carried out over the last years as an early-stage researcher in the field of lean combustion.
First, the recent CFD activities, performed at the University of Florence in collaboration with GE-AVIO and intended to set up a numerical procedure for the scale-resolving calculation of aero-engine combustors, are shown. Then, the attention is moved on the numerical description of ignition, which represents a further challenging problem to be tackled from an industrial perspective. Recent works at the EM2C laboratory focused on the effects of turbulent combustion and heat transfer on ignition and light-round are presented. Finally, the complexity of atomization modeling, which is a common problem between all the previous activities, is faced and the recent advances carried out in collaboration with the CORIA research center on the Eulerian Lagrangian Spray Atomization model are described.



Scale Adaptive Simulation of the FANN combustor designed by GE Avio Aero



Large Eddy Simulation of light-round in the MICCA-Spray chamber



Atomization of a liquid jet with the Eulerian Lagrangian Spray Atomization model