SPACE PROPULSION SYMPOSIUM (C4)
Propulsion System (1) (1)

Author: Mr. Daniele Cardillo
CIRA Italian Aerospace Research Center, Capua, Italy

Dr. Mario Panelli
CIRA Italian Aerospace Research Center, Capua, Italy
Dr. Daniele Ricci
CIRA Italian Aerospace Research Center, Capua, Italy
Dr. Pietro Roncioni
CIRA Italian Aerospace Research Centre, Italy
Dr. Francesco Battista
CIRA Italian Aerospace Research Centre, Italy

# NUMERICAL SIMULATION OF A LO2-CH4 ROCKET ENGINE DEMONSTRATOR

**Abstract**

The HYPROB Program, carried out by the Italian Aerospace Research Centre with the aim to increase system design and capabilities on liquid oxygen-methane engines, foresees the designing, manufacturing and testing of a ground demonstrator of three tons thrust, made of 18 injectors and regenerative cooled using methane. The project foresees the design of various breadboards, i.e. intermediate test articles, aimed at the investigation of critical design aspects. A numerical CFD campaign on a single injector sub-scale breadboard has been initially carried out, whose results allowed to understand important numerical and modeling aspects of the computation and so to consolidate the proper mathematical models and numerical schemes. Then, the analysis of the combustion phenomena has been switched on the more complex demonstrator configuration, with the main purpose to estimate the heat release at the chamber wall. The prediction of this variable is one of the major issue for regenerative thrust chamber design, since its correct estimation is fundamental in order to properly design the chamber cooling channel. In order to fulfill this purpose, several CFD reactive simulations have been performed in steady and transient states, with real gas model because of the super-critic oxygen condition. The output, in terms of wall heat flux, has been used as input for the numerical simulation of the cooling channel. Once obtained, the temperature profile along the channel has been used as new wall boundary condition for the reactive computation, allowing more accurate results. The solution dependence on the computational mesh was evaluated using different kind of grids, both structured and un-structured, with different global and local refinement levels. Numerical simulations have been performed both with the hypothesis of axi-symmetric flow than in 3D configuration, considering a 60 periodic edge. The simulations of the cooling channel were performed by means of structured grids, considering only a single cooling channel. Simulations were solved under the hypothesis of steady state, NIST real gas model and turbulent flow with thermo-physical properties, evaluated by REFPROP v7.0 database (http://webbook.nist.gov/chemistry/fluid/), according to the pressure and temperature operative conditions inside the channel. Also the conduction effects were taken into account both in the liner part, and in the close-out one. The channel surfaces were considered rough with 6.3x10-6 m. The full paper will report the results of the numerical computations carried for the demonstrator combustion chamber (reacting simulations) and for the cooling channel (non reacting with a transcritical phase change of methane).