11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

PLATE COOLING DESIGN BY MEANS OF CFD ANALYSIS

P. Natale[1], D. Ricci[2], M. Fragiacomo[3] and F. Battista[4]

[1] System engineer at C.I.R.A., Capua (CE) – Italy, p.natale@cira.it
[2] PhD system engineer at C.I.R.A., Capua (CE) – Italy, d.ricci@cira.it
[3] System engineer at C.I.R.A., Capua (CE) – Italy, m.fragiacomo@cira.it
[4] PhD senior system engineer at C.I.R.A., Capua (CE) – Italy, f.battista@cira.it

Key Words: *Firing Plate, CH_4, Cooling System, Liquid Rocket Engine, Rocket Engineering.*

The HYPROB program is carried out by CIRA under contract with the Italian Ministry of Research with the main objective to improve National system and technology capabilities on liquid rocket engines (LRE) for future space applications, with specific regard to LO_x/LCH_4 (liquid oxygen / liquid methane) technology. Among its objectives the design and development of technology LRE demonstrators, including intermediate breadboards, represent one of the most important parts [1].

In this framework, different subsystems have been designed and verified in order to produce and test a technological demonstrator, which is the final goal of the entire programme. This consists in a fully regenerative rocket engine where fuel (methane) is used to cool the combustion chamber (by a counter-flow cooling jacket subsystem) and then to feed injectors. Since very high heat fluxes are associated with combustion, the injector-plate has to dissipate high temperatures and a cooling channel has been designed in the back-side of the plate itself. Naturally, the coolant is represented by the methane coming from the cooling jacket [2].

An engineering tool has been developed in MATLAB® to design a baseline configuration for starting design analyses. This tool uses empirical laws (from the literature) written for cooling channels. From this baseline configuration, detailed analyses have been carried out in order to define the final configuration of the back-plate component and so cooling channel.

The aim of this work is to show a methodology based on the use of a commercial CFD solver (in particular Fluent® code [3]) to optimize the final design of this component. Moreover, the implementation of semi-empirical laws for fluid (i.e. methane) thermo-physical properties has been considered in order to explain their strong variability as a function of temperature and pressure in the ranges foreseen for this specific problem; in fact, methane is in supercritical thermodynamic state, therefore, its properties depend intensely on pressure and especially on temperature. Properties laws are taken from NIST table interpolations [4].

Moreover, a coupled run (solid/fluid) has been undertaken to simulate the behaviour of the firing-plate. In this way, heat fluxes have been imposed as boundary condition at the hot side of it (where injectors burn) and resulting temperature distribution on the cold side (facing at coolant gas methane) has been calculated by simulation. Specific laws have been implemented to simulate the actual characteristics of real material (copper alloy) that will be used for manufacturing this component.

The target for design loop was pressure drop minimization, from channel inlet towards injectors outlet, taking into account the design constraints, linked to other subsystems requirements. In particular, injectors pressure drop and combustion chamber operating pressure were considered as driving constraints.

Simulations have revealed some unwanted behaviours that flow exhibits in terms of unbalance of mass flow rate. The results have been considered in the design loop of the thruster (demonstrator). Moreover, these results will be used as input data for structural analyses for design validation [5].

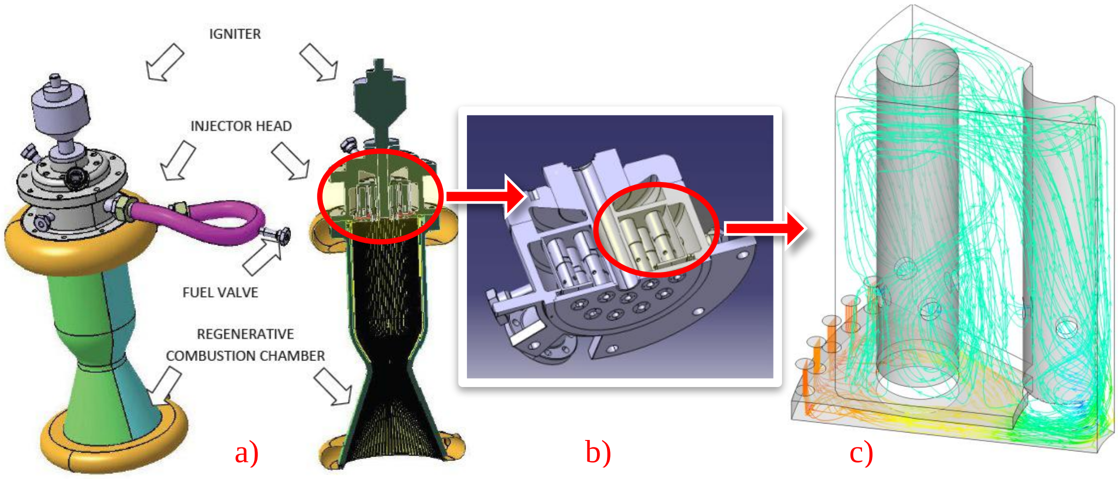

Figure 1: a) demonstrator full assembly; b) detail on injector head; c) CFD results in control volume considered

In Figure 1 a), the whole assembly provided for the demonstrator is reported. The Injector head is visible at the top of the assembly. This is closed, at down side, by the firing plate (or injector-plate), as it is visible in b). In c), it is possible to appreciate the calculation volume domain considered in these simulations obtained taking advantage of radial symmetric planes.

All the laws have been implemented by means of *User Defined Functions* (Fluent® UDF scripting) and methodology approach have been reported.

This work has to be considered as a methodological inspiration for further simulations where the implementation of NIST properties are strongly recommended

REFERENCES

[1] V. Salvatore, et al., *Recent Progress On The Development Of A LOx/LCH4 Rocket Engine Demonstrator In The Framework Of The Italian Hyprob Program*, 64th International Astronautical Congress, Beijing, Cina, 2013

[2] D.K., Huzel, and D. H. Huang, *Modern engineering for design of liquid-propellant rocket engines*, American Institute of Aeronautics and Astronautics. 1992.

[3] *Ansys Fluent User's Guide, release 13.0*, Ansys Inc., Canonsburg, PA, 2011.

[4] *NIST Chemistry WebBook, REFPPROP v7*, http://webbook.nist.gov/chemistry/fluid/

[5] P. Roncioni, et al., *CFD modelling and simulations of the HYPROB regenerative LOx/CH4 thrust chamber*, 5th European Conference for Aerospace Sciences, Munich, 2013.