CFD Modelling and Simulations of the HYPROB Regenerative LOX/CH4 Thrust Chamber

Pietro Roncioni, Daniele Cardillo, Mario Panelli, Daniele Ricci, Marco Di Clemente, Francesco Battista
CIRA (Italian Aerospace Research Centre), Via Maiorise, 81043 Capua (CE), Italy

The work described in this paper is conducted in the framework of the HYPROB Program that is carried out by the Italian Aerospace Research Centre (CIRA), under contract by the Italian Ministry of Research. The Program has the main objective to enable and improve National System and Technology capabilities on liquid rocket engines (LRE) for future space propulsion systems and applications, with specific regard to LOX/LCH4 technology.

A first implementation of the Program system line, named HYPROB BREAD, has been launched; it is aimed at designing, manufacturing and testing a LRE demonstrator, of three tons thrust, based on a regenerative cooling system using liquid methane as refrigerant. The HYPROB-BREAD project foresees, in the development of the complete system, the design of various breadboards aimed at the investigation of critical design aspects such as the supercritical behaviour of the methane and the mixing and combustion processes of the propellants (gaseous methane and liquid oxygen). In this paper all the CFD simulations aimed at investigations of the combustion phenomena inside the thrust chamber, including the regenerative cooling of supercritical methane, are reported and discussed. The above-said breadboards are described in [1].

The HYPROB-BREAD Project design activities have been conducted during the preliminary phase mainly by means of engineering methods. However, a cross check with CFD numerical simulations was needed in order to be confident with the results of the design activities. Moreover, an extensive use of CFD codes has been necessary not only for the cross checks with the engineering tools but also as an advanced design tool.

This use of the Computational Fluid Dynamics, as a design tool, can be possible only if simulations can give results in reasonable CPU convergence time. A massive use of "quick" CFD simulations oriented to give phenomenological trends has been used. Deep analyses (characterized by grid convergence and mathematical modelling and numerical schemes sensitivity analyses) have been also conducted but only on a reduced number of tests. At the time of writing these notes, experimental data are not yet available and so a full Verification and Validation procedure (V&V) is not possible for the CFD numerical tools. These quick CFD analyses, conducted following engineering criteria (or requirements) need to be used however in concurrence with engineering tools (ECOSIMPRO, etc.).

Several numerical simulations have been conducted on the Sub Scale Bread Board (SSBB, see Fig1) in order to understand the basic mixing and combustion phenomena. The SSBB is a scaled version of the three tons thrust LRE Demonstrator with only one injector and represents an intermediate step of the project from an experimental and a numerical point of view (validation of CFD codes). Both axisymmetric and full 3D simulations have been conducted aimed mainly at the evaluation of the wall heat fluxes. However other important quantities, as the chamber pressure and the combustion efficiency, have been verified.

The analysis of the thrust chamber (the Demonstrator) aimed, as in the case of previous Bread Board, at the estimation of the thermal and mechanical loads necessary for the structural design and at the characterization of the propellant mixing and combustion and flame structure evolution. The simulations have been performed both with the hypothesis of an axisymmetric flow (Fig4) and in 3D configuration (Fig5 and Fig6). The results showed a good comparison confirming that the use of simplified configurations (axisymmetric) can be used during the first steps of the design. One of the most important aspect of the project is the study of the regenerative cooling fuel (methane) and in particular the coupling between the external cooling counter flow and the internal hot combustion product flowing from the injectors to the nozzle. In figures 7 and 8 some results are already reported.

Acknowledgements

This work has been carried out within the HYPROB program, funded by the Italian Ministry of University and Research (MIUR) whose financial support is much appreciated.

References

1. V. Salvatore et al., Design and Development of a LOX/LCH4 Technology Demonstrator, 48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit 30 July - 01 August 2012, Atlanta, Georgia.

2. ANSYS FLUENT 12.0 Theory Guide, 7.4-7.6, Fluent Inc., Canonsburg, Pennsylvania, USA, 2009.

3. ECOSIMPRO, EA Internacional, Magallanes, 3. 28015 Madrid, Spain.

Nomenclature

CFD	=	Computational Fluid Dynamics
DEMO	=	Demonstrator
GCH4	=	Gaseous Methane
LOX	=	Liquid Oxygen
LRE	=	Liquid Rocket Engine
SSBB	=	Subscale Breadboard

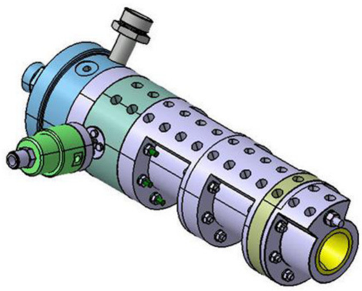

Figure 1: Subscale Bread Board

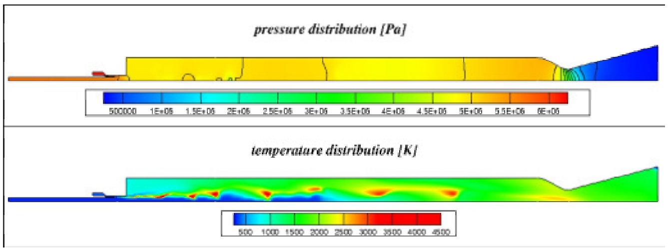

Figure 2: SSBB axisymmetric simulation

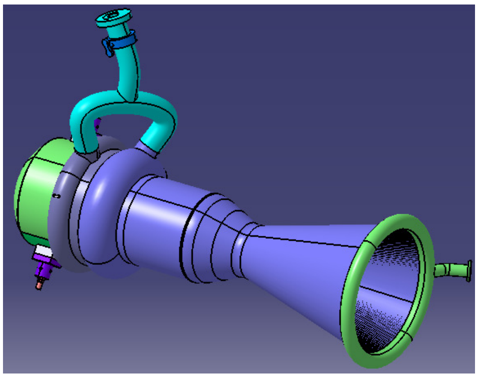

Figure 3: LRE Demonstrator

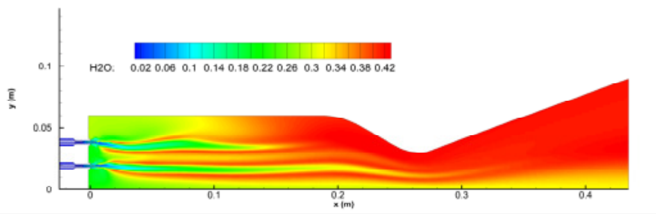

Figure 4: Demo axisymmetric simulation

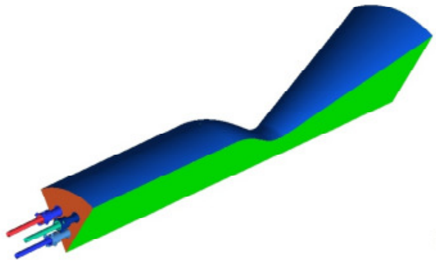

Figure 5: Demo: 60° wedge for 3D simulations

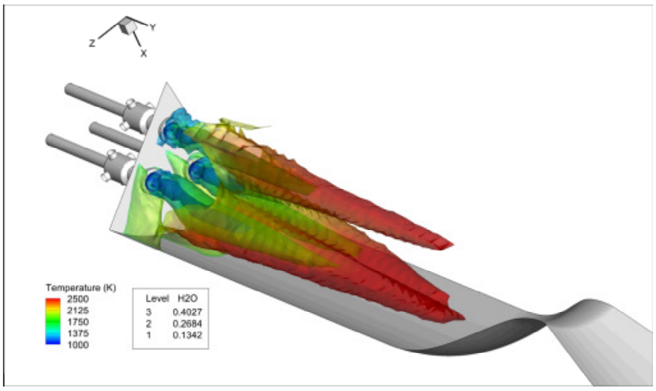

Figure 6: Demo 3D results

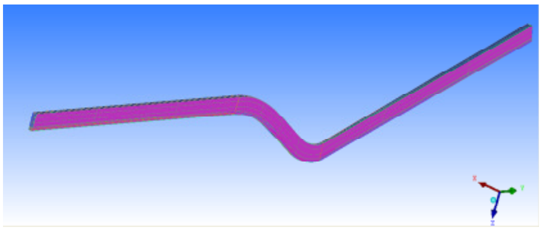

Figure 7: Demo: Regenerative cooling channel

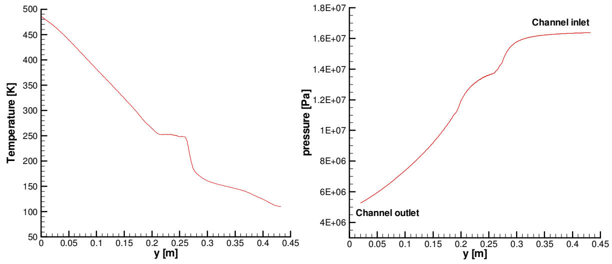

Figure 8: Demo 3D results