1D numerical simulations aimed to reproduce the operative conditions of a LOx/LCH$_4$ engine demonstrator

Angelo Romano[1,a*], Daniele Ricci[1,b], Francesco Battista[1,c]

[1]CIRA – Italian Aerospace Research Center, Italy

[a]a.romano@cira.it, [b]d.ricci@cira.it, [c]f.battista@cira.it

Keywords: Liquid Rocket Engine, LOx/LCH$_4$, EcosimPro Simulations, Regenerative Cooling

Abstract. The present paper describes the results of the numerical simulations performed by means of the *"EcosimPro"* software, aimed at reproducing the operative conditions of the regenerative thrust chamber "DEMO-0A" designed by the Italian Aerospace Research Center. The operative conditions simulated are both cold flow and firing conditions. A validation of the numerical cold flow results has been performed by comparing them with the experimental data gathered during a cold flow campaign. Once validated the cold flow numerical model, various hot test conditions of the demonstrator have been simulated by considering different heat wall exchange coefficient correlations, in order to obtain information about the thermal power released during the combustion process and to assess the simulation capabilities of the "*EcosimPro"* software in predicting the behaviour of the demonstrator in firing conditions by modelling it with a 1-D approach.

Introduction

The utilization of liquid oxygen/liquid methane couple (LOx/LCH$_4$) as a potential candidate to substitute hypergolic propellants in the next future propulsion systems has arisen an increasing interest due to the advantages offered in terms of high specific impulse, cooling capabilities, re-usability and low environmental impact [1]. In this perspective, the Italian Aerospace Research Center manages the "HYPROB" research program, which includes also the realization of a LOx/LCH$_4$ demonstrator engine named "DEMO-0A".

The thermal exchange in a liquid rocket engine regeneratively cooled represents a coupled heat transfer problem between the hot gases, the chamber wall and the coolant in the channels. Various approaches have been used to solve this problem: the possibility to use a 3-D modelling for the heat conduction through the wall and a 2-D approach, based on semiempirical correlations for the coolant and the hot gas flows, has been evaluated in [2], [3]. To overcome the complexity and the computational cost introduced by 3-D approaches, simplified quasi-2-D models have been extensively used to solve the heat transfer problem ([4], [5], [6]).

The aim of the present paper is to model the "DEMO-0A" engine designed and realized by the Italian Aerospace Research Center (CIRA) by means of 1-D components offered by the *EcosimPro* software. In particular, a series of numerical simulations have been performed to simulate cold flow and firing conditions, in which the coupled heat transfer problem between the hot gases, the chamber liner and the coolant has been solved by a 1-D approach. The scopes of these simulations, performed with a 1-D approach, consisted in: 1) validating the numerical results of the cooling system by a comparison with the experimental cold flow results; 2) investigating the effects on the thermal power released by the combustion chamber by considering different wall heat exchange semiempirical correlations; 3) assessing the capabilities of the 1-D model implemented in *EcosimPro* to predict the behaviour of the demonstrator in firing conditions.

The “DEMO-0A” Thrust chamber assembly

The “DEMO-0A” is a 30 kN thrust class demonstrator, fed with LOx/LCH$_4$, technologically representative of a thrust chamber assembly of an expander engine, regeneratively cooled by a counter-flow cooling jacket made up of 96 axial channels. The cooling channels of the “DEMO-0A” are obtained by joining an inner liner made of a cooper alloy (CuCrZr) with two outer layers (the first one made of cooper and the second one of nickel). The outer layers are deposited on the inner one by means of the electrodeposition technology. In a former version of the demonstrator the cooling channels were obtained by brazing the inner layer with an Inconel outer layer. Both liquid methane and water can be used to cool the engine ([7], [8]).

Figure 1 shows a model of the demonstrator that includes the igniter, the injector head with 18 coaxial recessed injectors and the thrust chamber with inlet/outlet manifolds for LCH$_4$ (or water).

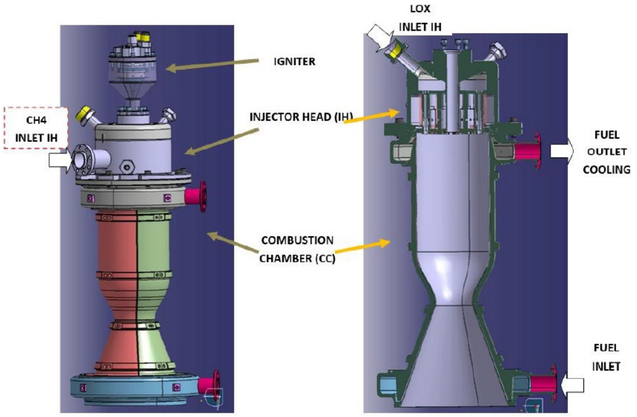

Figure 1:3-D model of the «DEMO-0A» architecture

The main geometrical and performance parameters are reported in Table 1.

Table 1

“DEMO-0A” performance and geometrical parameters			
Performance		Geometry	
T [kN] @ sea level	23.7	L [mm]	440
I_{sp} [s]	286	$D_{chamber}$ [mm]	119.6
ṁ LCH$_4$ [kg/s]	1.92	D_{throat} [mm]	59.8

Methodology

The numerical simulations have been performed by means of the *EcosimPro* software, a simulation tool that allows to model continuous and discrete systems.

The equations solved in the cold flow condition simulation are the mass conservation and the momentum conservation equations written under steady state conditions hypothesis ([9], [10]).

On the contrary, the system of governing equations of the combustor and cooling jacket coupling consists in the continuity equation written for the vaporized propellants, the momentum equation and the energy equation, all written under the hypothesis of steady state conditions [9]. The vapours are assumed to be fully released at the first node of the grid discretizing the combustor component assuming a characteristic vaporization time τ_{vap} set to 0.1 ms. The energy equation contains the term modelling the heat exchange between the hot gases, the chamber liner and the cooling jacket. Both convection and radiation are considered and *EcosimPro* offers three empirical correlations to calculate the wall heat exchange coefficient h_c: the Bartz, the modified Bartz and the Pavli correlations. The heat conduction in the cooling jacket liner is modelled by the Fourier equation. Regarding the combustion modelling a delayed equilibrium model has been chosen, in so doing a non-equilibrium combustor is simulated by introducing a time delay between the equilibrium condition and the actual burnt gases composition.

The discretization scheme considered is the AUSM with a 2nd order accuracy and the integration method is the «CVODE_BDF_SPARSE» [9] with transient and steady tolerances set to $1 * 10^{-6}$ as recommended in [9] in order to reduce the simulation time. The final time of the simulation has been set to 20 s.

Results

The cold flow experimental campaign was carried out at the AVIO/ASI FAST2 facility in Colleferro (Rome) and was devoted to perform tests in order to measure the pressure drops along the cooling channels of the regenerative cooling system and to define a characteristic law to predict the pressure drops of the "DEMO-0A" cooling system.

A mesh sensitivity analysis has been performed in order to individuate the discretizing mesh that allows to obtain a good quality solution with the minimum number of nodes and in the end the mesh independence turned out to be a discretizing grid of 35 nodes.

The results of three out of the six cold flow tests simulations performed in *EcosimPro* in terms of coolant pressure drop are reported in Table 2. The numerical results slightly underestimate the experimental ones and it is possible to see that the estimation error is between 2% and 3%.

The water pressure profile along the cooling jacket is reported in Figure 2(a) (red line) and allows to appreciate that the larger pressure gradients are concentrated in the throat region, as expected.

Table 2

	Cold flow tests results			
		EcosimPro	Experimental	$\varepsilon = \frac{\Delta X_{Eco} - \Delta X_{Exp}}{\Delta X_{Exp}}$
Test01	Δ_P_cool. [bar]	60.73	62.37	-2.63%
	Water MFR [kg/s]	5.12	5.12	0%
Test02	Δ_P_cool. [bar]	46.96	48.03	-2.23%
	Water MFR [kg/s]	4.47	4.49	-0.4%
Test03	Δ_P_cool. [bar]	35.83	36.68	-2.32%
	Water MFR [kg/s]	3.91	3.91	0%

By relating the pressure drops obtained during the six experimental tests with the squared water mass flow rates it is possible to note that their relation can be modelled as linear (Figure 2(b)).

The firing test campaign has been carried out at the AVIO/ASI FAST2 facility in Colleferro (Rome) and consisted in three tests. The mesh independence analysis, considered for the coupling of the cooling jacket with the combustor, differs from the one used in the cold flow simulations and is a discretizing grid made of 25 nodes.

In order to predict the temperature rise of the water inside the cooling channels, the three empirical correlations offered by *EcosimPro* to compute the wall heat exchange coefficient have been considered and the results in terms of water temperature rise and heat flux have been compared. These comparisons are here presented and they refer to the first firing test, labelled as «FT01».

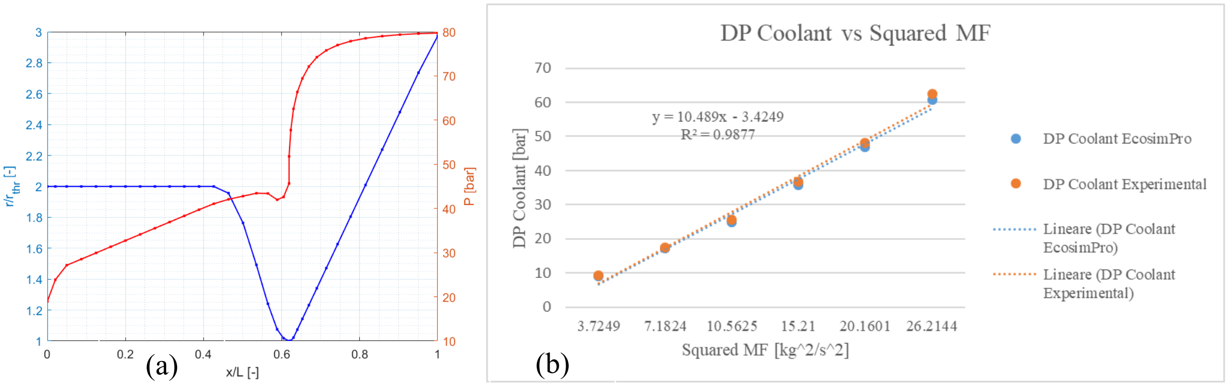

Figure 2: (a) Pressure variation along the cooling jacket. (Test01 results). (b) Pressure drop vs squared water MFR linear relation.

Figure 3(a) compares the water temperature profiles obtained with the Bartz, modified Bartz and Pavli correlations. The Bartz correlation underestimates the temperature increase while the modified Bartz and the Pavli correlations overestimate it.

The heat fluxes coming from the combustor are compared in Figure 3(b) and show how a peak is reached in the throat section, phenomenon due to the fact that in this region the exchange area (i.e. the lateral surface through which the thermal power coming from the combustor is exchanged) is the minimum. The heat flux obtained with the Pavli wall heat exchange correlation is characterized by the highest peak, on the contrary the Bartz correlation provides the lowest peak. Since the modified Bartz correlation provides the lowest error in estimating the water temperature rise, it has been considered as the correlation to use to predict the firing condition behaviour.

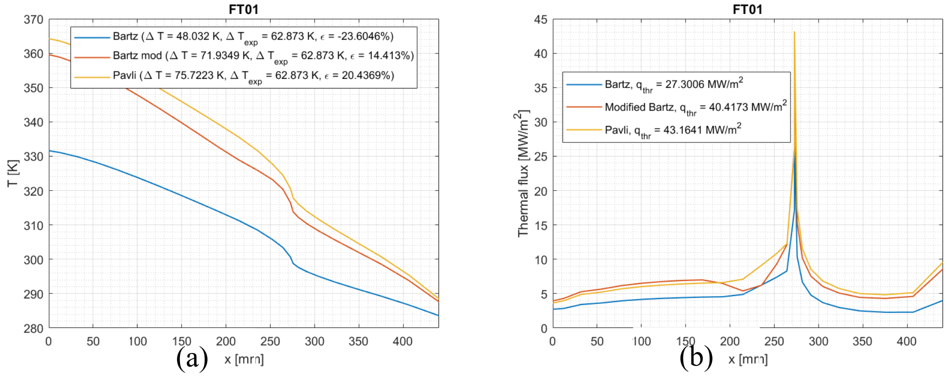

Figure 3: (a) Water temperature profiles comparison (FT01). (b) Heat fluxes comparison (FT01).

Table 3 reports a summary of the numerical and experimental results of the three firing tests. The numerical results well predict the pressure chamber, the coolant pressure drop and the thrust, on the contrary the coolant temperature rises are overpredicted. As expected, the higher thrusts occur during the tests where the chamber pressure is higher. Furthermore, the combustion efficiencies for the three tests are 93%, 89% and 93 %, respectively.

Table 3: Firing tests results

	FT01		FT02		FT03	
	Experimental	EcosimPro	Experimental	EcosimPro	Experimental	EcosimPro
P_CC [bar]	37.99	38	36.61	35.98	45.66	45.76
Water MFR [kg/s]	5.55	5.55	5.46	5.43	5.52	5.52
Water ΔP [bar]	67.86	69.19	65.72	66.99	66.85	68.46
Water ΔT [K]	62.87	71.93	61	73.16	71.92	83.23

Conclusions

The simulations performed by means of the *EcosimPro* software on the «DEMO-0A» to reproduce the cold flow operative conditions turned out to be very accurate for the prediction of the pressure drops experienced by the water in the cooling jacket that are slightly underestimated with respect to the experimental ones. This underestimation can be ascribed to the fact that the schematic used does not include the inlet and outlet manifolds that add further pressure drops. The simulations reproducing the firing conditions of the demonstrator had the aim to investigate the effects of the different wall heat exchange coefficient correlations on the thermal behaviour of the engine and the outcome of this study shows that the results obtained by considering the Bartz correlation underestimates the coolant temperature rise and thermal flux, on the contrary the modified Bartz and Pavli correlations overestimate them and the empirical correlation that provides results comparable to the experimental ones is the modified Bartz correlation. The results in terms of chamber pressure and temperature, pressure drop of the cooling jacket and thrust delivered are better predicted. In the end, the outcomes of the numerical simulations aimed at reproducing the firing conditions suggest that the *EcosimPro* software can be used to perform preliminary simulations able to provide accurate results in terms of chamber pressure, temperature and thrust, but with thermal results characterized by a lower accuracy due to the fact that the model developed is 1-D and does not take into account some phenomena that can occur in an engine and that are typically 2-D or 3-D , such as the thermal stratification of the coolant inside the cooling channels that changes the coolant thermophysical properties and consequently the thermal exchange occurring between the combustion chamber and the cooling jacket.

References

[1] C. D. Brown, "Conceptual Investigations for a Methane-Fueled Expander Rocket Engine.," *40th AIAA/ASME/SAE/ASEE Joint Propuls. Conf. & Exhib.,* 2004. https://doi.org/10.2514/6.2004-4210

[2] J. Jokhakar and M. Naraghi, "A CFD-RTE model for thermal analysis of regeneratively cooled rocket engines.," *44th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit. 2008.,* p. 4557, 2008. https://doi.org/10.2514/6.2008-4557

[3] H. Kawashima, H. Negishi, T. Tomita, K. Obase and T. Kaneko, "Verification of Prediction methods for Methane Heat Transfer Characteristics.," *48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit.,* p. 4120, 2012. https://doi.org/10.2514/6.2012-4120

[4] M. Pizzarelli, S. Carapellese and F. Nasuti, "A quasi-2-D model for the prediction of the wall temperature of rocket engine cooling channels.," *Numerical Heat Transfer, Part A: Applications,* pp. 1-24, 2011. https://doi.org/10.1080/10407782.2011.578011

[5] M. Leonardi, F. Di Matteo, J. Steelant, B. Betti, M. Pizzarelli, F. Nasuti and M. Onofri, "A zooming approach to investigate heat transfer in liquid rocket engines with ESPSS propulsion simulation tool.," *8th Aerothermodynamics Symposium,* 2015.

[6] C. H. Marchi, F. Laroca, A. F. C. D. Silva and J. N. Hinckel, “Numerical solutions of flows in rocket engines with regenerative cooling.,” *Numerical Heat Transfer, Part A: Applications,* pp. 699-717, 2004. https://doi.org/10.1080/10407780490424307

[7] F. Battista, D. Ricci, P. Natale and et al., “The HYPROB demonstrator line: status of the LOX/LCH4 propulsion activities.,” *8th European Conference for Aeronautics and Space Sciences, EUCASS2019-FP0621,* 2019.

[8] Empresarios Agrupados, “EcosimPro ESPSS User Manual,” 2020.

[9] S. Omori, W. G. Klaus and A. Krebsbach, “Wall temperature distribution calculation for a rocket nozzle contour.,” *NASA-TN-D-6825,* 1972.

[10] B. Betti, M. Pizzarelli and F. Nasuti, “Coupled Heat Transfer Analysis in Regeneratively Cooled Thrust Chambers.,” *Journal of Propulsion and Power,* pp. 360-367, 2014. https://doi.org/10.2514/1.B34855

[11] D. Ricci, F. Battista, M. Ferraiuolo and et al., “Development of a Liquid Rocket Ground Demonstrator through thermal analyses,” *Heat Transf. Eng.,* pp. 1100-1116, 2020. https://doi.org/10.1080/01457632.2019.1600879