Coupled analysis of hot-gas and coolant flows in LOX/methane thrust chambers

Marco Pizzarelli, Barbara Betti and Francesco Nasuti
Sapienza University of Rome
Dipartimento di Ingegneria Meccanica e Aerospaziale, Via Eudossiana 18, Rome 00184, Italy

Abstract

The analysis of the flow-behavior and the heat transfer characteristics in a regeneratively cooled thrust chamber is of paramount importance for the development of a high-performance liquid rocket engine and to guarantee its structural thermal design. The accurate analysis of such a problem can only be obtained by costly experiments or by complex numerical simulations, because of the three-dimensional shape of the cooling channels and of the coupling among evolution in hot-gas flow, coolant flow and wall heat transfer. However, parametric investigations of rocket thrust chambers thermal environment can also be efficiently made by accurate engineering tools. An example of the latter approach is provided in the present study: a computational procedure able to describe the typical hot-gas/wall/coolant environment of liquid rocket engines. This procedure provides a quick and reliable prediction of thrust-chamber wall temperature and heat flux. The coupled analysis is performed by means of an accurate CFD solver of the Reynolds-Averaged Navier-Stokes (RANS) equations for the hot-gas flow and a simplified "quasi-2D" approach, which widely relies on semi-empirical relations, to study the problem of coolant flow and wall structure heat transfer in the cooling channels. Numerical results, relevant to a sample engine fed with oxygen and methane are presented and discussed.

1. Introduction

The oxygen/methane propellant combination is currently considered for the development of future European liquid propellant thrust chamber [1, 2, 3, 4]. Compared to kerosene, methane can provide a higher specific impulse, together with lower pressure loss in the cooling channel, better cooling capability and less soot deposition [5]. With respect to oxygen/hydrogen propellant combination, oxygen/methane can be considered "space storable" and it is favored by a higher density [6], although it gives lower specific impulse.

The hot gas environment within a modern high performance rocket combustion chamber is characterized by gas temperatures up to 3600 K and heat fluxes up to 160 MW/m^2. In order to keep the temperatures of the thrust chamber walls within their allowed limits, an intense cooling effort is necessary, which, in case of regenerative cooling, is achieved by flowing one of the two propellants into suitable channels surrounding the thrust chamber. The adequate understanding and accurate prediction of regenerative cooling heat transfer characteristics, heat pick-up and wall temperature distribution in the thrust chamber are considered key features for the development of oxygen/methane engines, especially in the case of expander cycle feed system. In fact, in this case, the driving power for the turbo-pumps comes from the methane used as a coolant in the regenerative cooling circuit. Expander cycle engines are an attractive solution for upper stage propulsion because of their performance and capability for multiple restarts and throttling. They have the simplest configuration among the pump-fed cycles because they do not require components such as gas generators or preburners. Development and testing of expander cycle engines continues to be of interest for the international industries and space agencies, as demonstrated by the studies on RL-10 in all its configurations in US since '60 [7, 8], on RD0146 in Russia [9], on LE-5B in Japan [10], and on Vinci in Europe [11]. All of these expander cycle engines are operated with oxygen/hydrogen propellant combination. Although no known oxygen/methane engines have been flown for aerospace applications, space agencies in the US, Russia, Europe and Japan have been considering methane fueled propulsion system for various applications. Recently, studies on existing oxygen/hydrogen expander cycle engines modified by replacement of hydrogen with methane have been performed by industries [12, 13] and universities [14], in order to better understand feasibility and limits of envisaged oxygen/methane expander cycle engines.

Coupled heat transfer problem of hot gas, chamber wall and coolant in thrust chambers is usually studied on the basis of one-dimensional analysis with Nusselt-type empirical equations [15, 16, 17]. Coupled analysis involving CFD hot-gas side evaluation, three dimensional heat conduction model and one-dimensional hydraulic model for the coolant flow has been performed by [18] for a LOX/hydrogen engine. Fully coupled multidisciplinary CFD simulation with heat conduction and quasi-one-dimensional water cooling channel model for a water cooled thruster nozzle has been conducted by [19]. Two proven thermal analysis codes have been conjugated in [20] via an interface file to design and analyze regeneratively-cooled rocket engine. Refining these models, axis-symmetric CFD model has been adopted for hot-gas side flow instead of the previous thermal analysis code in [21]. Recently, coupled analysis of a water-cooled nozzle has been performed by [22] with CFD for the coolant-cooling channel conjugate heat transfer and two different approaches for the hot-gas side: empirical models and CFD. All of these approaches have their well-defined objective that in some case leads to fast and simplified methods and in other cases to heavy computations which are not suited to engine design phase.

Objective of the present study is to present a computational procedure able to describe the coupled environment among hot-gas wall and coolant that occurs in most liquid rocket engines and to provide a quick and reliable prediction of thrust-chamber wall temperature and heat flux as well as coolant-flow characteristics, which takes into account the near-critical nature of the coolant and thermal stratification in the high-aspect-ratio cooling channels. These parameters, which are of great importance in the development of an expander-cycle engine, will be analyzed for the case of LOX/methane propellant combination. The coupled analysis is performed by means of an accurate CFD solver of the Reynolds-Averaged Navier-Stokes (RANS) equations for the hot-gas flow and a simplified "quasi-2D" approach, which widely relies on semi-empirical relations, to study the problem of coolant flow and wall structure heat transfer in the cooling channels.

2. Modeling

A comprehensive thermal model for a regeneratively cooled thrust chamber must account for convection from hot-gas, conduction within the wall, and convection to the cooling channels. In fact, heat transfer can be described as the heat flux between two fluids, separated by a solid wall. In its simplest form regenerative cooling can be modeled as a steady heat flux from a hot gas through a solid wall to a cold fluid. This problem can be divided into three sub-problems, which are defined as follows:

Hot gas The turbulent flow of a mixture of gases in a rocket engine, including combustion chamber and converging-diverging nozzle.

Wall The heat conduction through the wall of the rocket engine between the hot gases and the coolant.

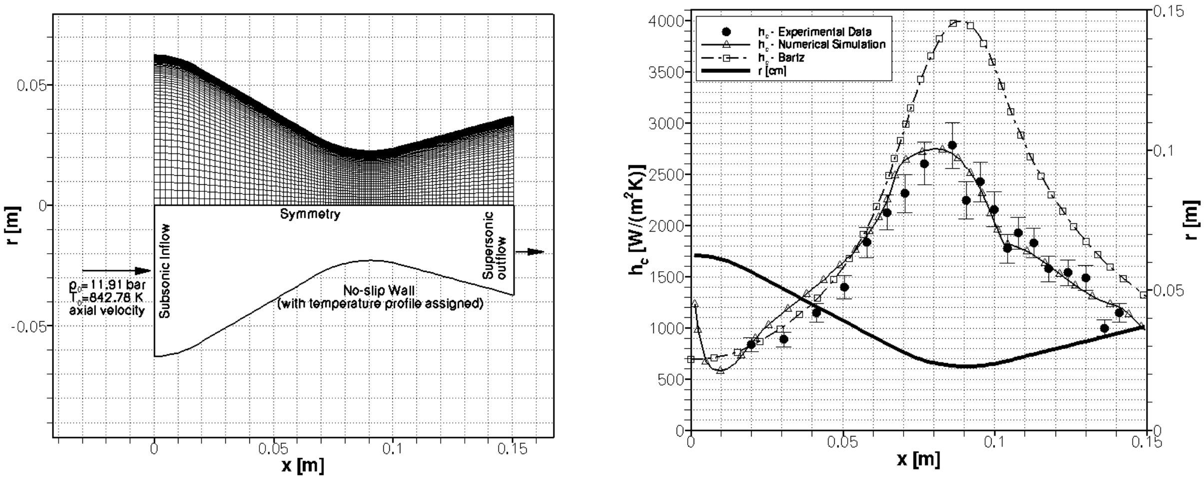
Coolant The turbulent flow of the coolant in the channels surrounding the rocket engine.

These sub-problems are coupled by the two steady-state balances of three heat fluxes: from hot-gas to wall; through the wall; and from wall to coolant.

2.1 Hot gas

A Reynolds-Averaged Navier-Stokes (RANS) approach is used to simulate the hot-gas flow-field and heat transfer in the thrust chamber. In particular, combustion is not simulated, but a flow of combustion products in chemical equilibrium at the chamber temperature and pressure is injected using a "full-inlet" approach [23]. The numerical solutions are carried out by means of an in-house 2D-axisymmetric multi-block finite volume RANS equations solver [24, 25], modified to simulate multi-component mixtures of thermally perfect gases. Turbulence is described by means of the Spalart-Allmaras one equation model [26]. A constant turbulent Schmidt number is adopted to model turbulent diffusivity.

The motivation for introducing hot-gas flowfield simulation in the coupled procedure is that it can provide a quite more accurate evaluation of wall heat transfer compared to empirical approaches, like the widely used approach based on Bartz's correlation [27]. For the sake of understanding the differences and at the same time demonstrate validation of the numerical approach the benchmark experimental test case No. 313 of [28] has been selected and reproduced. This test case is quite useful for validation because authors provide wall temperature and wall heat flux measurements and convective heat transfer coefficient evaluation. The test case is characterized by a supersonic nozzle where compressed and heated air flows with a stagnation pressure of 13.91 bar and a stagnation temperature of 842.78 K. Wall temperature and wall heat flux measurement uncertainties are ±1% and ±8%, respectively. As a consequence, convective heat transfer coefficient is affected at least by the sum of wall temperature and wall heat flux measurements uncertainties (±9%). The numerical reproduction of the experimental test case has been carried out by axis-symmetric computations with grid and boundary conditions shown in Fig. 1(a). The experimental wall temperature enforced has been interpolated with a polynomial function in order to have a smooth temperature profile at the wall. Three grid levels were employed to ensure grid convergence, with the fine grid having 200×180 cells in the axial and radial directions, respectively. The medium grid is obtained by removing one node out of two, in each coordinate direction, from the fine grid. In the same way the coarse grid is obtained from the medium grid. Asymptotic convergence is verified and medium size grid numerical solution for wall heat flux differs from Richardson extrapolated solution at most by 2%. Due to the



(a) Grid (medium grid level) and boundary conditions enforced.

(b) Comparison between experimental data, CFD numerical solution and Bartz's semi-empirical model.

Figure 1: Convergent-divergent nozzle [28]

close values of wall temperature and hot air stagnation temperature, wall heat flux along the nozzle is strongly affected by the interpolation law of the experimental temperature enforced at the wall. As a consequence, here is decided to compare emperimental and numerical heat transfer coefficients which are weakly related to wall temperature values. In Fig. 1(b), h_c numerical solution along the nozzle is plotted together with the heat transfer coefficient evaluated by the classic Bartz's correlation [27]:

$$h_c = \left[\frac{0.026}{D_*^{0.2}} \left(\frac{\mu^{0.2} c_p}{Pr^{0.6}} \right) \left(\frac{p_c}{c^*} \right)^{0.8} \right] \cdot \left(\frac{A_*}{A} \right)^{0.9} \cdot \sigma \tag{1}$$

for the sake of comparison. In Eq. (1), μ, c_p and Pr are the viscosity, the specific heat and the Prandtl number of the combustion gases, respectively, evaluated at the chamber conditions, p_c is the chamber pressure, c^* is the characteristic exhaust velocity, D^* is the nozzle diameter at the throat, A_*/A is the nozzle area ratio at the actual axial position and σ is a factor which contains the correction for property variations across the boundary layer. As shown in Fig. 1(b), Bartz correlation clearly overpredicts the convective heat transfer coefficient along the nozzle reaching a value 45% higher than experimental measurement at throat, whereas CFD numerical solution follows the experimental data within the error estimated in the reference work along the nozzle. The comparison between accurate numerical tools and semi-empirical models against experimental data shows that one-dimensional semi-empirical models cannot be employed for hot-gas side heat transfer evaluation due to their low accuracy, especially in the throat region where the maximum heat flux occurs.

2.2 Coolant and wall

A simplified “quasi-2D” approach [29, 30] is used to study the coupled problem of coolant flow and wall structure heat transfer in rectangular cooling channels of liquid rocket engine thrust chambers. This approach is based on a 1D hydraulic model of the coolant mass and momentum governing equations and on a 2D stratified model of the coolant energy equation which is coupled to the wall heat transfer balance in the radial direction. The turbulent thermal conductivity, fluid skin friction and coolant-wall heat transfer coefficients are evaluated by semi-empirical relations provided in the literature. This model permits to have a fast prediction of both the coolant flow evolution and the temperature distribution along the whole cooling channel structure. The model, aiming to the study of any fluid evolving through cooling channels, considers a generic equation of state, and thus the coolant fluid can be either a compressible gas, or a supercritical fluid or a liquid. To consider fluids other than perfect gas or incompressible liquid is mandatory in the case of methane as coolant in rocket engines, because its thermodynamic conditions in the cooling circuit of an expander cycle engine are quite close to the critical point. In this regime, the fluid cannot be modeled either as a perfect gas or as a liquid. The thermodynamic properties of methane can be evaluated by any accurate model or database for equation of state and transport properties like viscosity and thermal conductivity. Among them, thermophysical properties provided in [31] are selected in the present study.

2.3 Coupling

The quasi-2d model is able to solve the coupled wall and coolant flow evolution once thermal boundary conditions are provided. In particular, the model, suited to rectangular channels, requires thermal boundary conditions at the top and bottom walls whereas symmetry is exploited for the channel side wall boundary conditions. The thermal boundary condition can be one of the following:

- assigned wall temperature;
- assigned heat flux;
- assigned convective heat transfer coefficient and adiabatic wall temperature.

On the outer side of the rectangular channel an “assigned heat flux” boundary condition is given, assuming zero heat flux, that is adiabatic wall. On the inner side, the hot gas side where heat enters the channel, thermal boundary condition is obtained by the RANS solver. Of course, in principle, neither temperature nor wall heat flux are known, because they depend on the thermal equilibrium between flows and wall, so a first tentative solution has to be computed. This could be done for instance enforcing a first tentative distribution of wall temperature, then passing the resulting heat flux to the quasi-2d solver, compare the wall temperature computed by the quasi-2d model with the first tentative value, and iterate. Analogously it could be done by enforcing a first tentative heat flux distribution. Although this process could successfully converge, a more efficient way is that of enforcing convective heat transfer coefficient and adiabatic wall temperature. In fact in this case only the heat transfer coefficient has to be adjusted which only weakly depends on wall temperature. Therefore the convective heat transfer coefficient approach is used to provide the boundary condition to the quasi-2d solver. In particular, the coupling procedure is described by the following steps:

1. RANS computation of the hot-gas flow, with adiabatic boundary condition to determine the adiabatic wall temperature T_{aw}.

2. RANS computation of the hot-gas flow, imposing the wall temperature ($T_w = 800$ K at first iteration, the result provided by quasi-2d computation for the next iterations) as boundary condition and evaluation of the heat transfer coefficient h_c from the computed heat flux, the enforced wall temperature and the adiabatic wall temperature T_{aw} computed in step 1.

3. “quasi-2D” computation of the coupled coolant flow and wall thermal evolution using h_c values from step 2 and T_{aw} values from step 1; the coupled coolant flow and wall analysis provides a resulting value of T_w.

Iterations are made repeating steps 2 & 3. The process ends when the difference between the computed values of T_w in two consecutive iterations is smaller than an assigned value.

3. Test case

Being interested to LOX/methane thrust chambers, a reference test case of such thrust chambers has been searched for in the literature. The selected reference liquid rocket engine is the one discussed in [14], which is a LOX/methane expander cycle engine studied theoretically with particular focus on regenerative cooling jacket design. On the basis of the system and parametric analysis carried out in [14] the following features of the thrust chamber, nozzle and cooling channel have been selected. For the thrust chamber the assumed data are reported in Table 1. Gas mixture composition and chamber total temperature are calculated as the equilibrium condition [32]. The resulting chamber temperature is 3603.20 K, and the most important species accounted for the mixture are: CO, CO_2, H, HO_2, H_2, H_2O, O, OH, O_2. Nozzle geometry is designed by method of characteristics assuming a truncated ideal nozzle contour (Fig. 2). As attention was focused on cooling channel/thrust chamber coupling, only the whole convergent section and the part of the divergent section cooled with channels (up to $x_i = 25.1$ cm and $\varepsilon_i = 15$, see Table 2) have been considered in the RANS simulations.

Constant cross section cooling channels are considered whose main characteristic are reported in Table 2. Note

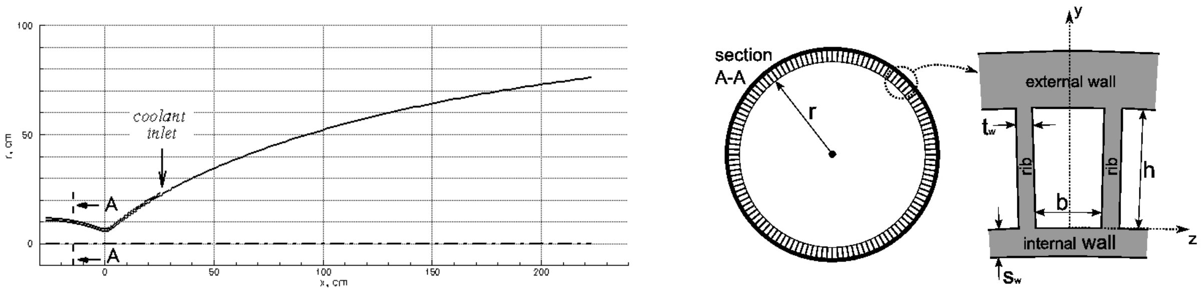


Figure 2: Schematic of nozzle and regenerative cooling channels.

Chamber pressure (p_c), bar	58.6
O/F	3.5
Expansion ratio (ε_d)	180
Contraction ratio (ε_c)	3.5
Throat radius (r_t), cm	5.66
Divergent section length (L_d), cm	223.0
Convergent section length (L_c)	33.1

Table 1: Thrust chamber and nozzle data

width (b), mm	1.08
height (h), mm	5.40, 6.48, 8.63
Aspect ratio (λ)	5, 6, 8
Number of channels	150
Inlet section area ratio (ε_i)	15
Inlet section abscissa (x_i), cm	25.1
Internal wall thickness (s_w), mm	0.7

Table 2: Cooling channel data

that, as will be discussed in the following, three different sets of cooling channels have been considered, all featuring the same width but different height and thus aspect ratio and cross-section area. Moreover, it is assumed that the wall material is hydrodynamically smooth and its thermal conductivity is $k_w = 390$ W/m/K, that is typical of high-conductivity copper-alloys. The dimensions of the cooling circuit composed of 150 rectangular channels are plotted in Fig. 3 together with the engine radius for the case with aspect ratio $\lambda = 8$.

4. Results

The approach described in Section 2 has been used to evaluate the performance of the regenerative cooling system of the engine test case described in Section 3. CFD simulations were carried out on the grid shown in Fig. 4 featuring 100×90 cells in the axial and radial direction, respectively. Cells have been clustered towards the wall to correctly capture the turbulent kinematic and thermal boundary layer. Grid and clustering law have been selected after a grid convergence analysis. In particular the present grid feature a minimum cell height such that $y^+ = 0.75$ at its maximum at throat. The basic CFD simulation, with adiabatic wall has been the first step leading to the solution shown in Fig. 5. As can be seen the computed value of adiabatic wall temperature slightly decreases from a maximum of about 3600 K, at the combustion chamber entrance.

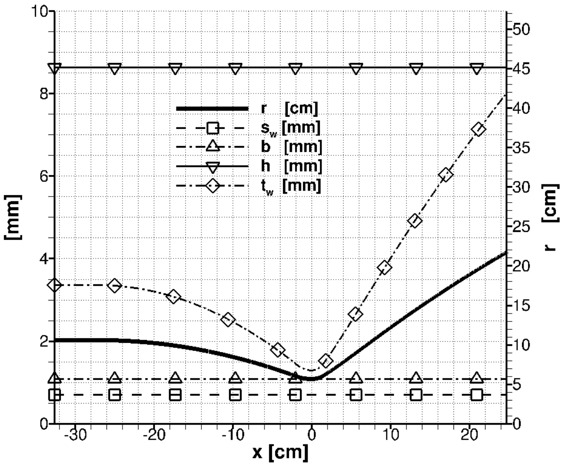


Figure 3: Features of the cooling circuit.

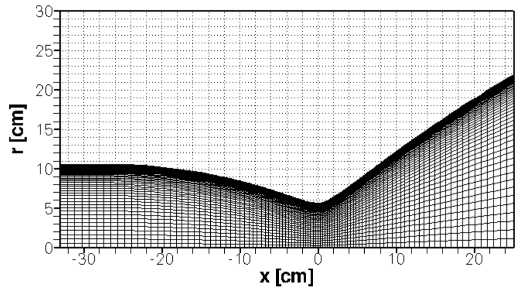


Figure 4: Numerical grid for the hot gas flow simulation.

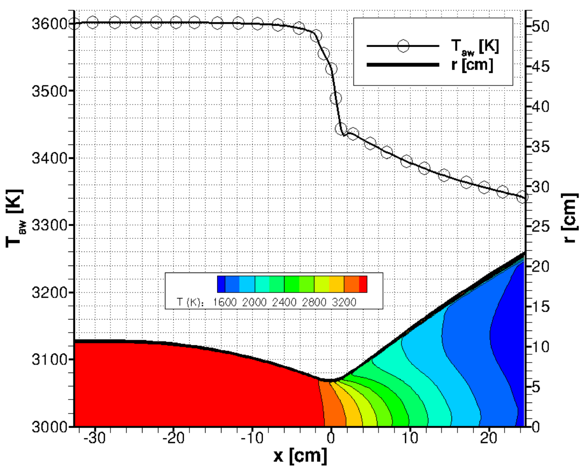


Figure 5: Wall temperature and temperature field in the nozzle in case of adiabatic wall.

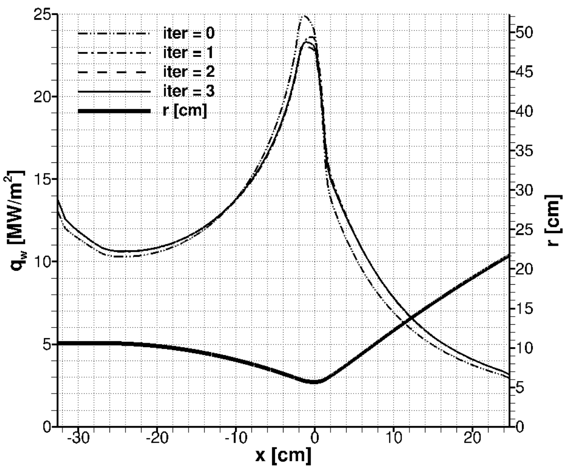


Figure 6: Wall heat flux convergence history of the quasi-2D simulation.

On the basis of the adiabatic CFD solution and of the CFD simulation carried out enforcing isothermal wall with $T_w = 800K$ the first tentative solution for the convective heat transfer coefficient provided by CFD is obtained and passed to the quasi-2D solver of coolant flow and channel wall thermal evolution.

The semi-empirical relations used in the quasi-2D model to describe the skin friction factor f_w and the Nusselt number Nu are those of Colebrook [33] and Sieder & Tate [34], respectively:

$$\frac{1}{\sqrt{f_w}} = -2\log\left(\frac{2.51}{Re\sqrt{f_w}}\right) \tag{2}$$

and

$$Nu = 0.027Re^{0.8}Pr^{0.4}\left(\frac{\mu}{\mu_w}\right)^{0.14} \tag{3}$$

where the properties are evaluated at the fluid pressure and temperature but the viscosity μ_w in Eq. (3), which is evaluated at the wall temperature and takes into account for the variable properties of the coolant. The computed heat flux at the end of the first computation by the quasi-2D model is displayed in Fig. 6 (iter =0). Then the computed wall temperature is passed as a boundary condition of the CFD solver, an updated value of h_c computed by the CFD solution is passed back to the quasi-2D solver. The result is the new iteration solution for the wall heat flux (iter=1 in Fig. 6). Note that, convergence is rather fast and that the second quasi-2D computation provides a result quite close to the final one. After four quasi-2D computations convergence is achieved with accuracy.

For the sake of comparison, beside using the present model, coupling has been carried out also computing coolant flow by a basic 1D model. Of course, physical and mathematical modeling of in the 1D computation is similar to that used in the quasi-2D model, with the difference that in the former case the coolant velocity and temperature are constant in each channel cross-section and the energy equation is written in 1D rather than 2D form. This means that in the 1D case all the coolant parameters vary only along the axial abscissa x and thus no thermal stratification can be evaluated. For the 1D flow model the same semi-empirical expressions as the quasi-2D model have been adopted. Note that heat conduction through ribs is taken into account by suitable discretization also in case on 1D flow assumption.

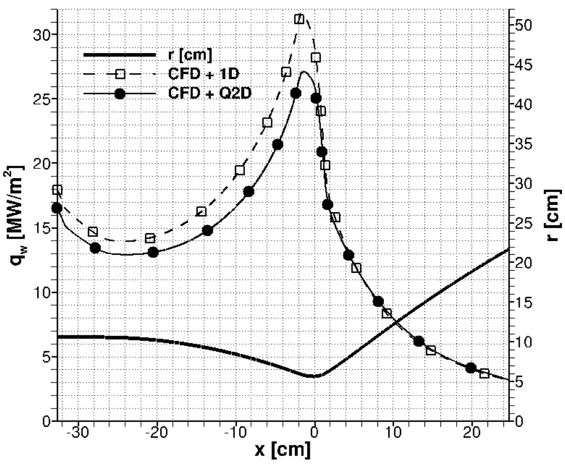


Figure 7: Wall heat flux.

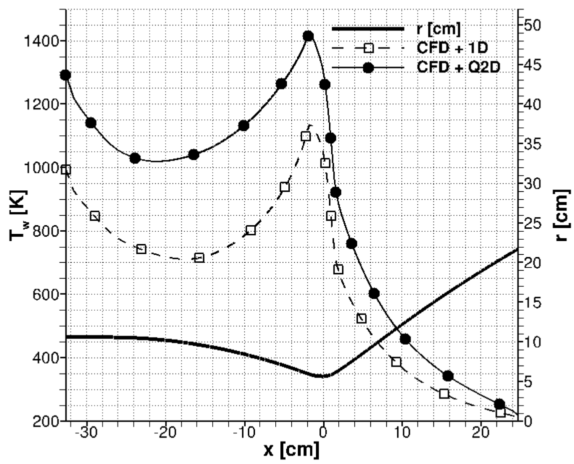


Figure 8: Hot-gas side wall temperature.

Comparison of the converged solutions achieved by means of both the 1D and the quasi-2D solvers coupled with CFD are displayed in Figs. 7-8. The plotted variables are wall heat flux and maximum wall temperature. Looking at Fig. 7, it is clear that the computed wall heat flux at the hot gas side is higher in case of 1D modeling. In fact, the 1D model predicts a peak value of about 32 MW/m^2 in the throat region, whereas this value is about 27 MW/m^2 for the quasi-2D model, that is 16% lower. However, in the divergent section of the nozzle, the two models exhibit similar predictions of the wall heat flux. Figure 8 shows that the wall temperature at the hot-gas side is strongly dependent on the coolant flow modeling. This is mainly due to the different wall heat flux prediction (Fig. 7): the higher the heat flux the lower the wall temperature. Maximum wall temperature is about 1100 K for the 1D model and about 1400 K for the quasi-2D model. While the former result, although hardly acceptable for a copper-based thrust chamber, is similar to that presented in [14] for the same engine, the quasi-2D model predicts a maximum wall temperature which is far larger than the limit temperature of high conductivity copper alloys. The discrepancy between the two models is due to the fact that the quasi-2D model takes into account the reduced cooling capability in case of radial thermal stratification in the channel cross section.

Because the quasi-2D predictions have shown that the achieved temperature of wall and coolant are far beyond design requirements, a different design is considered in the following, which aims to satisfy system requirements. To improve the cooling capability, that is, increasing the coolant flow velocity, the total cross section of the cooling channels must be reduced. This goal can be achieved reducing the channel height and retaining the number of cooling channels. In this way, the cross section aspect ratio reduces. Two more different cooling channel designs have been thus considered: channels with aspect ratio $\lambda = 6$ and $\lambda = 5$, that is, total area reduction of 25% and 37.5%, respectively.

Decreasing cross section area lead to a decrease of heat flux (Fig. 9) and to a significant reduction of wall temperature (Fig. 10). In particular, heat flux peak decreases from about 32 MW/m^2 to about 27 MW/m^2 in case of $\lambda = 5$. The maximum wall temperature decreases from the about 1400 K of $\lambda = 8$ to about 1200 K in case of $\lambda = 6$ and

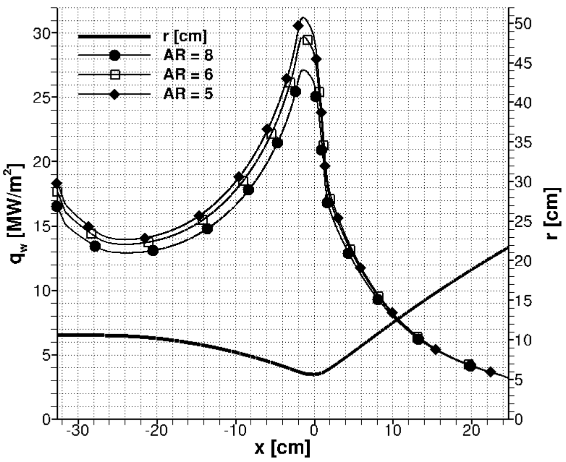


Figure 9: Wall heat flux.

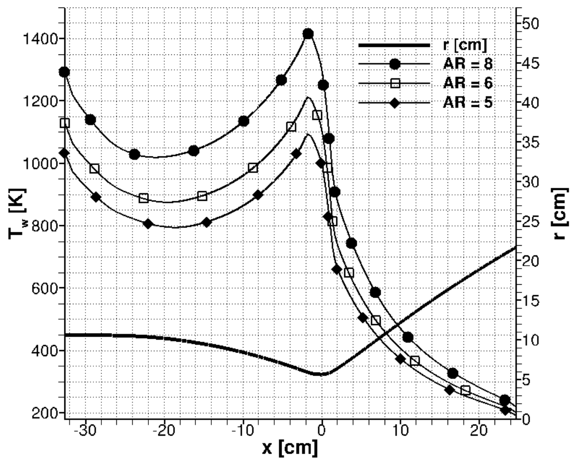


Figure 10: Hot-gas side wall temperature.

finally to the value of about 1150 K in case of $\lambda = 5$ which is close to the value accepted by the system analysis of [14]. It is worth noticing that the maximum temperature always occur at both nozzle entrance and throat and that the curves preserve the same general behavior but at lower and lower temperature for decreasing λ. Of course the 18% maximum temperature reduction has a cost, which is in term of increased total pressure loss and total temperature increase. In fact increasing coolant velocity, due to the cross section area reduction, increases pressure drop. Moreover, also pressure loss due to heating increases with velocity where coolant can be considered as a compressible gas.

The present approach, relying on coupled CFD for the hot gas and shows that stratification has a significant effect on the prediction of maximum wall temperature and heat flux and thus it cannot be neglected in the engine design phase. On the contrary it has to be included by suitable models like the quasi-2d approach proposed in the present study.

5. Conclusions

A loosely coupled procedure for the simulation of the behavior of wall and coolant flow in a regeneratively cooled liquid rocket engine, has been set up and demonstrated on a literature test case. The numerical approach is based on validated software based on CFD computation of RANS equations for the hot gas and on a quasi-2d modeling for the coupled coolant/wall evolution. Results point out that stratification has a role that, if not considered (or poorly considered like in 1D approaches) could lead to significant errors in the design. In particular, without accounting for thermal stratification temperature peak reached at the wall and in the coolant fluid are far higher than those which can be reasonably foreseen by the 1D analysis of cooling channel.

References

[1] Haeseler, D., Gotz, A., Mading, C., Roubinski, V., Gorokhov, V., and Khrissanfov, S., "Testing of LOX-Hydrocarbons Thrust Chambers for Future Reusable Launch Vehicles," AIAA Paper 2002-3845, July 2002, 38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[2] Preclik, D., Hagemann, G., Knab, O., Mading, C., Haeseler, D., Haidn, O., Woschnak, A., and DeRosa, M., "LOX/Hydrocarbon Preparatory Thrust Chamber Technology Activities in Germany," AIAA Paper 2005-4555, July 2005, 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[3] Ierardo, N., Biagioni, M., and Pirrelli, P., "Development of the LM10-MIRA LOX-LNG Expander Cycle Engine for the LYRA Launch Vehicle," Proceedings of the 2nd conference space propulsion 2008, May 2008.

[4] Arione, L., Ierardo, N., Rudnykh, M., Caggiano, G., Lobov, S., Shostak, A., De Lillis, A., and D'Aversa, E., "Development status of the LM10-MIRA LOX-LNG Expander Cycle Engine for the LYRA Launch Vehicle," Proceedings of space propulsion 2010, May 2010.

[5] Pempie, P., Frohlich, T., and Vernin, H., "LOX/Methane and LOX/Kerosene High Thrust Engine Trade-Off," AIAA Paper 2001-3542, July 2001, 37th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[6] Preuss, A., Preclik, D., Mading, C., Gorgen, J., Soller, S., Haidn, O., Oschwald, M., Clauss, W., Arnold, R., and Sender, J., "LOX/Methane Technology Efforts for Future Liquid Rocket Engines," Proceedings, May 2008, Proceedings of the 2nd Conference Space Propulsion 2008.

[7] Pugh, R., "The Many Factes of the RL-10 Liquid Rocket Engine, a Continuing Success Story," AIAA Paper 1998-3680, July 1998, 34th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[8] Peery, S. and Minick, A., "Design and Development of an Advanced Expander Combustor," AIAA Paper 1998-3675, July 1998, 34th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[9] Rachuk, V. and Titkov, N., "The First Russian LOX-LH_2 Expander Cycle LRE: RD0146," AIAA Paper 2006-4904, July 2006, 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[10] Fujita, M., Aoki, H., Nakagawa, T., Tokunaga, T., Yasui, M., and Iwama, K., "Upper Stage Propulsion System for H-IIA Launch Vehicle," AIAA Paper 2002-4212, July 2002, 38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[11] Knab, O., Fröhlich, A., Wennerberg, D., and Haslinger, W., "Advanced Cooling Circuit Layout for the VINCI Expander Cycle Thrust Chamber," AIAA Paper 2002-4005, July 2002, 38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[12] Crocker, A. M. and Peery, S. D., "System Sensitivity Studies of a LOX/Methane Expander Cycle Rocket Engine," AIAA Paper 1998-3674, July 1998, 34th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[13] Brown, C., "Conceptual Investigations for a Methane-Fueled Expander Rocket Engine," AIAA Paper 2004-4210, July 2004, 40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[14] Schuff, R., Mayer, M., Sindiy, O., Ulrich, C., and Fugger, S., "Integrated Modeling and Analysis for a LOX/Methane Expander Cycle Engine: Focusing on Regenerative Cooling Jacket Design," AIAA Paper 2006-4534, July 2006, 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[15] Frohlich, A., Popp, M., Schmidt, G., and Thelemann, D., "Heat Transfer Characteristics of H_2/O_2: Combustion Chambers," AIAA Paper 1993-1826, July 1993, 29th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[16] Kirchberger, C., Wagner, R., Kau, H., Soller, S., Martin, P., Bouchez, M., and Bonzom, C., "Prediction and Analysis of Heat Transfer in Small Rocket Chambers," AIAA Paper 2008-1260, July 2008, 44th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[17] Marchi, C. H., Laroca, F., da Silva, A. F. C., and Hinckel, J. N., "Numerical Solutions of Flows in Rocket Engines with Regenerative Cooling," *Numerical Heat Transfer, Part A: Applications*, Vol. 45, No. 7, 2004, pp. 699–717.

[18] Wang, T. S. and Luong, V., "Hot-Gas-Side and Coolant-Side Heat Transfer in Liquid Rocket Engine Combustors," *Journal of Thermophysics and Heat Transfer*, Vol. 8, No. 3, 1994, pp. 524–530.

[19] Liu, Q., Luke, E., Cinnella, P., and Tang, L., "Coupling Heat Transfer and Fluid Flow Solvers for Multidisciplinary Simulations," *Journal of Thermophysics and Heat Transfer*, Vol. 19, No. 4, 2005, pp. 417–427.

[20] Naraghi, M. H., Dunn, S., and Coats, D., "A Model for Design and Analysis of Regeneratively Cooled Rocket Engines," AIAA Paper 2004-3852, July 2004, 40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[21] Naraghi, M. H. and Jokhakar, J., "A CFD-RTE Model for Thermal Analysis of regeneratively Cooled Rocket Engines," AIAA Paper 2008-4557, July 2008, 44th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[22] Kang, Y. D. and Sun, B., "Numerical Simulation of Liquid Rocket Engine Thrust Chamber Regenerative Cooling," *Journal of Thermophysics and Heat Transfer*, Vol. 25, No. 1, 2011, pp. 155–164.

[23] Betti, B., Martelli, E., Nasuti, F., and Onofri, M., “Numerical Study of Film Cooling in Oxygen/Methane Thrust Chambers,” 4th European Conference for Aerospace Sciences, July 2011.

[24] Pizzarelli, M., Nasuti, F., Paciorri, R., and Onofri, M., “Numerical Analysis of Three-Dimensional Flow of Supercritical Fluid in Asymmetrically Heated Channels,” *AIAA Journal*, Vol. 47, No. 11, Nov. 2009.

[25] Betti, B., Martelli, E., and Nasuti, F., “Heat Flux Evaluation in Oxygen/Methane Thrust Chambers by RANS Approach,” AIAA Paper 2010-6721, July 2010, 46th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[26] Spalart, P. and Allmaras, S., “A One-Equation Turbulence Model for Aerodynamic Flows,” *La Recherche Aerospatiale*, Vol. 1, 1994, pp. 5–21.

[27] Bartz, D. R., “A Simple Equation For Rapid Estimation of Rocket Nozzle Convective Heat Transfer Coefficients,” *Jet Propulsion*, Vol. 27, No. 1, 1957, pp. 49–51.

[28] Back, L. H., Massier, P. F., and Gier, H. L., “Convective Heat Transfer in a Convergent-Divergent Nozzle,” NASA CR-32-415, 1965.

[29] Pizzarelli, M., Nasuti, F., and Onofri, M., “A Simplified Model for the Analysis of Thermal Stratification in Cooling Channels,” 2nd European Conference for Aerospace Sciences, July 2007.

[30] Pizzarelli, M., Carapellese, S., and Nasuti, F., “A Quasi-2D Model for the Prediction of Wall Temperature of Rocket Engine Cooling Channels,” *Numerical Heat Transfer, Part A: Applications*, In press.

[31] Younglove, B. and Ely, J., “Thermophysical Properties of Fluids. II. Methane, Ethane, Propane, Isobutane, and Normal Butane,” *Journal of Physical and Chemical Reference Data*, Vol. 16, No. 4, June 1987, pp. 577–798.

[32] Gordon, S. and McBride, B. J., “Computer Program for Calculation of Complex Chemical Equilibrium Compositions and Applications, I. Analysis,” NASA-RP 1311, Oct. 1994.

[33] Colebrook, C. F., “Turbulent Flow in Pipes with Particular Reference to the Transition Region Between the Smooth and Rough Pipe Flow,” *J. Inst. Civ. Eng.*, , No. 11, 1938, pp. 133–156.

[34] Sieder, E. N. and Tate, G. E., “Heat Transfer and Pressure Drop of Liquids in Tubes,” *Industrial and Engineering Chemistry*, Vol. 28, No. 12, 1936, pp. 1429–1435.