Near wall recombination effects on the prediction of heat flux in O_2/CH_4 combustion chambers

B. Betti*, D. Bianchi*, F. Creta*, E. Martelli**, F. Nasuti*, M. Onofri*

* *"Sapienza" University of Rome*
Department of Mechanical and Aerospace Engineering
via Eudossiana 18, I-00184 Rome, Italy

** *Second University of Naples*
Department of Industrial and Information Engineering
Via Roma 29, I-81131 Aversa, Italy

INTRODUCTION

In the frame of the "In-Space Propulsion 1" (ISP-1) project funded by the European Community 7^{th} Framework Program (FP7) [1], film cooling in oxygen/methane thrust chambers has been investigated by means of experimental tests and numerical simulations [2, 3]. Heat flux and wall temperature have been measured and computed in the case with and without film cooling to evaluate film cooling efficiency. In fact, inside the thrust chamber, the hot gas flow reaches very high temperature which exceeds by far the safe temperature limit of metallic materials, hence requiring active cooling to preserve the thrust chamber walls from mechanical failure. The thrust chamber wall heat transfer has been numerically predicted in a satisfactory way with a simplified numerical approach for most cases, as discussed in [3]. Nevertheless, as long as frozen gas chemistry is assumed, the important role of recombination reactions that may take place in the near wall region is not taken into account. Aim of the present study is to quantify the effect of finite-rate chemistry inside the boundary layer on the wall heat flux using the same simplified numerical approach as in [3]. The analysis will be performed on the experimental test case investigated in the frame of the "In-Space Propulsion 1" (ISP-1) project.

NUMERICAL METHOD AND APPROACH

A Reynolds-Averaged Navier-Stokes (RANS) approach is used to simulate the hot-gas flow-field and heat transfer in the thrust chamber. The numerical solutions are carried out by means of an in-house 3D multi-block finite volume RANS equations solver able to treat multi-component mixtures of thermally perfect gases [4]. Turbulence is described by means of the Spalart-Allmaras one equation model [5]. Constant turbulent Prandtl number and Schmidt number are adopted to model turbulent conductivity and diffusivity, respectively. The present numerical investigations are carried out adopting a simplified approach aiming to reduce the computational cost of the simulation capturing the basic phenomena driving the heat transfer processes in liquid rocket engine thrust chambers.

In full scale engines, turbulent mixing and combustion are typically confined to a short distance from the injector plate. Models able to resolve accurately these complex phenomena involved in the near injection region, can hardly be extended to full scale dimension unless dramatically increasing the computational cost. As a compromise between global understanding of the problem inside the thrust chamber and low computational cost, a simplified approach is therefore adopted here to evaluate the heat load coming from the hot combustion products inside the thrust chamber. In particular, near injector plate phenomena are neglected and equilibrium combustion products are injected in the chamber at the adiabatic flame temperature. Inlet mixture composition and adiabatic flame temperature are evaluated by means of the "Chemical Equilibrium and Applications" (CEA) program [6], prescribing stagnation chamber pressure and propellant mixture ratio. This mixture can evolve in the flow field according to a finite-rate chemical reaction mechanism. Gas thermodynamic and transport properties are evaluated as a function of the local temperature. The hot gas inlet section is chosen preserving the distance between the outer injector ring and the thrust chamber wall as shown in red in Fig. 1.

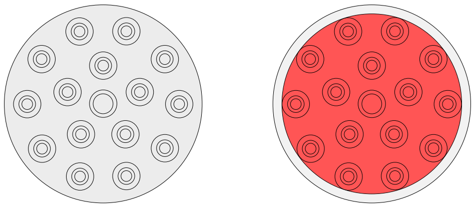


Figure 1: Pseudo-injector simplified approach inlet section.

RESULTS

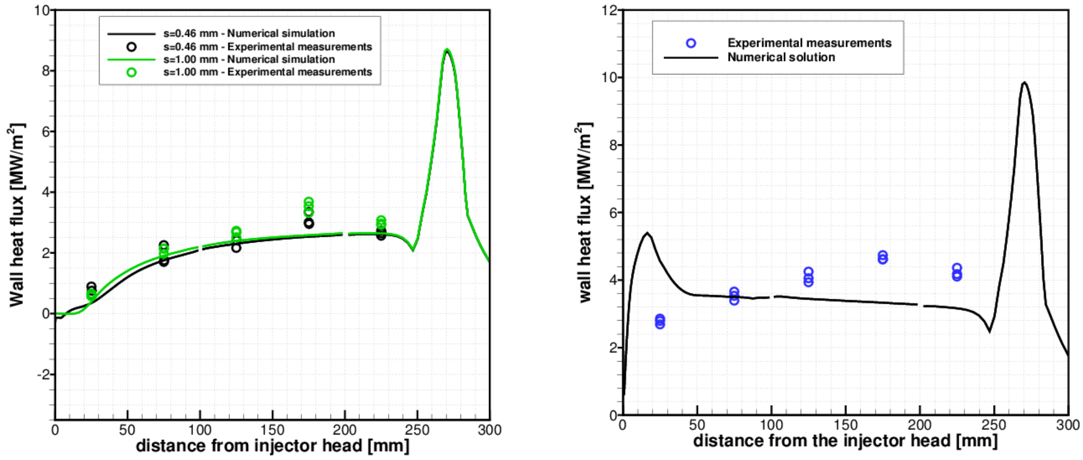
The experimental test case[1, 2] consists in a low pressure calorimetric subscale combustion chamber where gaseous oxygen and gaseous methane are injected at ambient temperature. Film cooling is realized with gaseous methane injected through a 2D axis-symmetric circular slot. In the experimental test campaign, different chamber pressures, mixture ratios and film slot heights have been investigated providing detailed axial distribution of heat pick-up based on water mass flow rate and surface temperatures.

In numerically reproducing the present experimental test matrix, wall temperature has been enforced as a boundary condition at the wall and the predicted wall heat flux has been compared to the integral wall heat flux measured by the calorimetric method. In the case with film cooling, very good agreement has been achieved between experimental and numerical solution as shown in Fig. 2(a). On the contrary, in the case without film cooling, the numerical solution is able to provide the average value along the chamber whereas the trend is not captured (Fig. 2(b)).

Note that in the test case without film cooling, dealing with extremely high gas temperature near the wall and low wall temperature may lead to a significant heat source coming from the partial or complete recombination of dissociated species inside the boundary layer and at the surface. An upper bound of this *recombination heat flux* can be evaluated by assuming chemical equilibrium at wall. This can be expressed in terms of heat flux percentage increase $\Delta q_{wall,\,\%}$ by means of Eq. 1:

$$\Delta q_{wall,\,\%} = \left(\frac{h_{eq} - h_{wall,eq}}{h_{eq} - h_{wall,frozen}} - 1 \right) \times 100 \tag{1}$$

where h_{eq} is the enthalpy of the chemical equilibrium mixture at the chamber pressure and adiabatic flame temperature (i.e. chamber equilibrium mixture); $h_{wall,frozen}$ that of the chamber equilibrium mixture at the wall temperature; and $h_{wall,eq}$ that of the chemical equilibrium mixture at the wall temperature. Depending on the wall temperature, the chamber pressure, and the propellant combination, this term can



(a) With film cooling (slot height effect)

(b) Without film cooling

Figure 2: Wall heat flux: comparison between experimental measurements and numerical results.

be significant reaching the same order of magnitude of the convective wall heat flux. Being related to the residence time and the assumption of chemical equilibrium, this term may be more significant in the cylindrical section of the chamber rather than in the nozzle.

In the final work, a quantitative analysis of the effect of both finite rate chemistry inside the boundary layer and surface recombinations on wall heat flux will be discussed focusing on wall temperature, chamber pressure and mixture ratio effects.

REFERENCES

[1] Ordonneau, G., Haidn, O., Soller, S., and Onofri, M., "Oxygen-methane Combustion Studies in In Space Propulsion Programme," *4th European Conference for Aerospace Sciences, EUCASS, Saint Petersburg, Russia, 3-7 July*, 2011.

[2] Suslov, D., Betti, B., Aichner, T., Soller, S., Nasuti, F., and Haidn, O., "Experimental Investigation and CFD-Simulation of the Film Cooling in an O_2/CH_4 subscale Combustion Chamber," *Space Propulsion, Bordeaux, France, 7-10 may*, 2012.

[3] Betti, B., Martelli, E., Nasuti, F., and Onofri, M., "Numerical Study of Heat Transfer in Film Cooled Thrust Chambers," 48^{th} AIAA/ASME/SAE/ASEE Joint Propulsion Conference& Exhibit 30 July - 01 August 2012, Atlanta, Georgia. AIAA 2012-3907.

[4] Pizzarelli, M., Nasuti, F., Paciorri, R., and Onofri, M., "Numerical Analysis of Three-Dimensional Flow of Supercritical Fluid in Cooling Channels," *AIAA Journal, Vol. 47, No. 11 (2534-2543)*, 2009.

[5] Spalart, P. and Allmaras, S., "A One-equation Turbulence Model for Aerodynamic Flows," *La Recherche Aerospatiale 1, 5-23*, 1994.

[6] McBride, B. J. and Gordon, S., "Computer Program for Calculation of Complex Chemical Equilibrium Compositions and Applications," *NASA Reference Publication 1311*, 1994.