Joint design and simulation of GOX-GCH4 combustion and cooling in an experimental water-cooled subscale rocket engine

Alexandru MEREU*,[1], Dragos ISVORANU[1]

*Corresponding author
[1]National University for Science and Technology POLITEHNICA Bucharest,
Splaiul Independentei 313, 060042, Bucharest, Romania,
mru.alexandru@gmail.com*

DOI: 10.13111/2066-8201.2023.15.4.13

Received: 23 October 2023/ Accepted: 09 November 2023/ Published: December 2023
Copyright © 2023. Published by INCAS. This is an "open access" article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/)

Abstract: *This paper presents the authors' most recent research regarding the feasibility of cooling a 1 kN scaled-down experimental rocket engine, running on gaseous oxygen and gaseous methane, for a ground test. The cooling segment of a rocket engine has always been a delicate problem, increasing the development time and costs of development. Since a series of problems can occur during the first ignition of a rocket engine prototype, removing as many potential issues from the initial test, such as using liquid methane for the cooling system, could result in a more stable experiment. Using water as the cooling agent can contribute to a more accelerated TRL increase of the engine's subcomponents while reducing the risks taken for a whole assembly test. Thus, the combustion chamber, nozzle, and injector can be tested separately from the final cooling method, which can be added subsequently. In the present work, both a steady and transient CFD combustion simulation of a multicomponent compound, consisting of gaseous oxygen and gaseous methane was conducted in the combustion chamber of a small-scale rocket engine. The simulation is based on PDF-flamelet approach for the oxygen and methane combustion, along with real gas equations for the cooling agent.*

Key Words: *rocket engine, test bench water cooling, CFD analysis, PDF combustion, real-fluid.*

1. INTRODUCTION

The environment within a modern high performance rocket combustion chamber is characterized by high gas temperatures up to 3600 K and extremely high heat fluxes up to 160 MW/m^2. To keep the temperatures of the thrust chamber walls within their application limits, an intense cooling effort is necessary. The primary cooling method used in LRE (Liquid Rocket Engines) is regenerative cooling. Regenerative cooling is the method by which either cryogenic fuel or oxidizer is passed through the cooling channel around the thrust chamber. In this respect, large supply pressure requirements are enforced. In the last years, a paradigm shift has been noticed in terms of replacing LH2 (liquid hydrogen) with LCH4 (liquid methane) especially for low orbit missions. In such rocket engines methane is considered as the coolant and it will typically enter the counterflow cooling channels with a supercritical pressure and a subcritical temperature. As methane is heated up, its temperature passes through the critical value (190.56 K); for that reason, the methane flow can be referred to as trans-critical fluid flow for the present applications. The resulting peculiar methane behaviour is caused by the

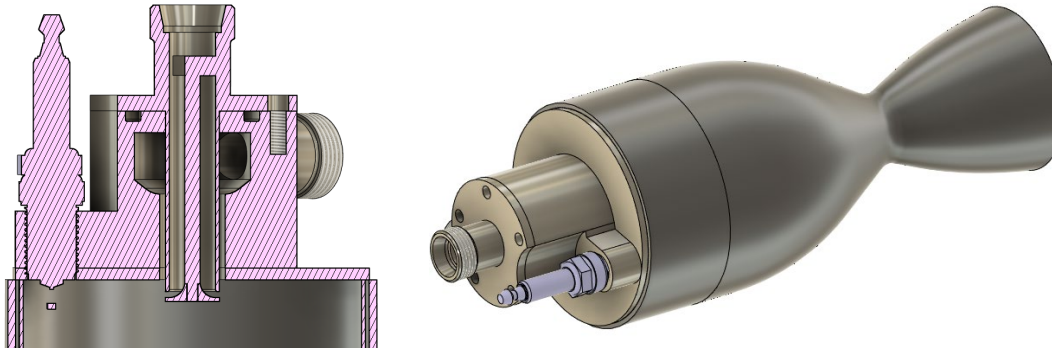
large changes in fluid properties (such as specific heat and density) that occur in the near-critical region, which can greatly affect heat transfer. Although many studies have been carried out for supercritical-hydrogen used in LOX/LH2 engines and for supercritical-nitrogen used in many cryogenic-research laboratories, the typical flow condition of methane inside the cooling channel has only recently been investigated [1-7].

However, for rather rudimentary subscale test benches lacking turbo-pumps, it seems that alternate cooling has to be imagined. The least expensive cooling agent is water because it has quite a large specific capacity at supercritical pressures and subcritical temperatures. The subject of this paper concerns the combined simulation of combustion and cooling for a 1kN subscale thruster.

Establishing the cooling capacity of an active water-cooling system to be used on a ground-test of the rocket engine prototype represents an important goal, along with the estimation of the flame stability and anchoring of the main thermodynamic parameters. The simulation is done with gaseous methane and oxygen fed from pressurized tanks and water as the cooling agent. To the best of our knowledge, it is the first joint simulation for combustion and heat transfer to the cooling channels in the available literature.

2. ENGINE PARAMETERS

The study is conducted on a small rocket engine prototype of approximately 220 x 95 mm running on gaseous methane and gaseous oxygen. The engine prototype has a designed thrust of 1 kN at sea level with a specific impulse of 273 s. The reaction occurs at a chamber pressure of 20 bar with a total mass flow rate of fuel and oxidizer of 0.435 kg/s at a mixture ratio of 3.6 (0.34043 kg/s oxygen and 0.09457 kg/s methane), just under the stoichiometric ratio of 4. The 3.6 mixture ratio was chosen to protect the engine from an oxidizer-rich combustion which can lead to the oxidation of the combustion chamber and, subsequently, its failure. The injector is a pintle-type injector composed of a circumferential annulus, situated between the snout and the central injector element providing an axial flow of fuel or oxidizer that intersects in the impingement point with the other component, with a uniform velocity profile. The entire engine was manufactured from OFHC by 3D printing and is represented in Fig. 1. The cooling of the engine is performed through a shell-type jacket, which reduces design complexity and manufacturing expense, consequently reducing overall costs of development. For the next engine, the inner chamber shall be manufactured of oxygen-free copper (OFHC), for much better heat transfer capability, a material that was also used in the simulations.



a) b)

Fig. 1 Cross-section view of the injector (a) and full model (b)

The cooling agent shall be water with a mass flow rate of 1.5 kg/s at a pressure of 60 bar. In the final configuration of the engine, methane shall be used as a cooling agent, but for the first combustion test, water shall be used to actively cool the chamber's walls. Thus, in the simulation, both methane and oxygen are injected in a gaseous state, assuming they undergo a phase change from liquid to gas when exposed to ambient temperatures during the feeding process from pressurized tanks to the engine. The shape of the combustor was obtained using RPA code [8].

3. COMPUTATIONAL APPROACH

The approach taken for this study involves the use of ANSYS CFX CFD solver. The computational domain consists of the hot side and part of the ambient environment and is represented in Fig. 2.

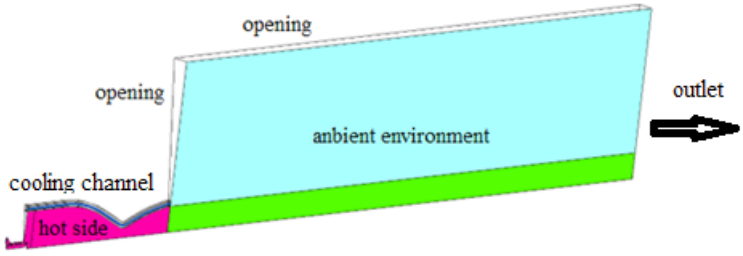


Fig. 2

For the simulation, a 6-degree 3D section of the engine is used, with periodic conditions thus, a denser mesh can be obtained ($0.1\ mm$) while maintaining a relative fast calculation process. The mesh is more than 95% structured being inflated in the vicinity of walls. The first layer of the inflation is placed at 10^{-5} m away from walls and contains 16 layers.
A small detail of the mesh is presented in Fig. 3. It can be noted that close to the injector plate the size of the has been decreased almost 2 times with respect to the mesh size in the rest of the combustion chamber.

Fig. 3

The number of elements mounts up to more than 2.6 million. Besides the mass flow rate boundary conditions for methane and oxygen described above, we have imposed normal pressure and temperature conditions on the opening boundaries. For the cooling system, water enters at 60 bar and 310 K from the far end of the combusting chamber and at the outlet side near the injector plate a mass flow rate condition has been set for a parametric analysis for the purpose of ensuring the acceptable temperature distribution on the inner wall of the combustion chamber. The thermodynamic properties of the cooling agent are described in terms of IAPWS IF97 steam tables. In doing so, we were able to capture the phase

transformation from liquid to vapor wherever was needed. For the two solid domains, the thermal conductivity was set at 320 W/m.K corresponding for OFHC. Because the nominal pressure inside the combustion chamber is relatively small (20 bar), it was considered that the perfect gas assumption for the thermodynamic behavior of the combusting mixture and the products of combustion is perfectively valid. On the other hand, due to the domain configuration in which two solid domains and two separate fluid domains are involved, we have noticed that the best option for numerical simulation was Ansys CFX as Fluent cannot handle properly different physics settings (one reacting one non-reacting) for the two fluid domains. As a non-premixed combustion process is taking place, the transport equations for mixture fractions can be written as a single equation, assuming equal species diffusion ($Le = 1$). Although this approach is problematic for laminar flows, it is acceptable for turbulent flows because turbulent convection is greater than molecular diffusion. The transport equation used is the Favre equation for mixture fractions:

$$\frac{\partial}{\partial t}\left(\rho\bar{f}\right) + \nabla \cdot \left(\rho\vec{v}\bar{f}\right) = \nabla \cdot \left(\frac{\mu_l + \mu_t}{\sigma_t}\nabla\bar{f}\right) + S_m + S_{user} \quad (1)$$

Where μ_l is the laminar viscosity, and μ_t is the turbulent viscosity. Assuming chemical equilibrium, all thermochemical scalars are reported in terms of mixture fractions, and the effect of heat release/absorption is parameterized as follows:

$$\phi_i = \phi_i(f, H) \quad (2)$$

The reaction model used is based on chemical equilibrium PDF-flamelet. The flamelet library comprises C1 to C3 chemical species besides the usual reactants and products of reaction: C2H2, C2H4, C2H6, C3H3, C3H4, C3H5, C3H6, CH3, CH4, CO, CO2, H, H2, H2O, HCCO, HO2, O, O2, OH. The temperature dependence on mixture fraction and mixture fraction variance is depicted in Fig. 4.

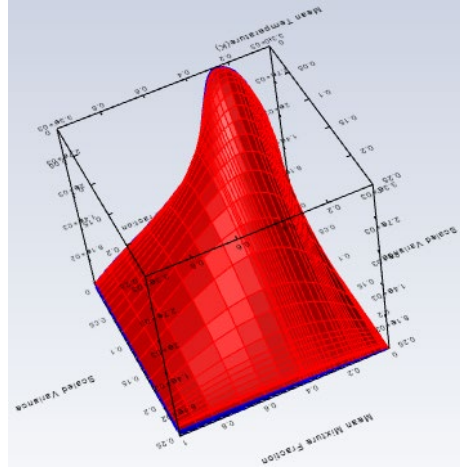


Fig. 4 Visual representation of the PDF function

The first simulation attempt was taken in steady state condition, as we wanted to check the performance of the cooling system. The RANS equations were completed by the mass fraction and mass fraction variance equations together with generalized k-eps equations for the turbulence model (GEKO).

In the second attempt, we went for an unsteady computation and changed GEKO model for the SAS-SST method [9, 10]. It is an improved URANS formulation for unstable flows based

on the von Karman length-scale introduction in the turbulence scale equation. Thus, a LES-like behavior is obtained in unsteady regions of the flow due to the dynamically adjusted resolved structures based on the provided von Karman information, while also having standard RANS capabilities in a stable flow field. Ansys CFX uses the latest model of the SST-SAS model, presented in the work of Egorov and Menter [11], which uses the quadratic length scale ratio $(L/L_{vK})^2$ instead of the linear form. The quadratic length scale is more consistent with the derivation of the model.

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j k) = P_k - \rho c_\mu k\omega + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] \tag{3}$$

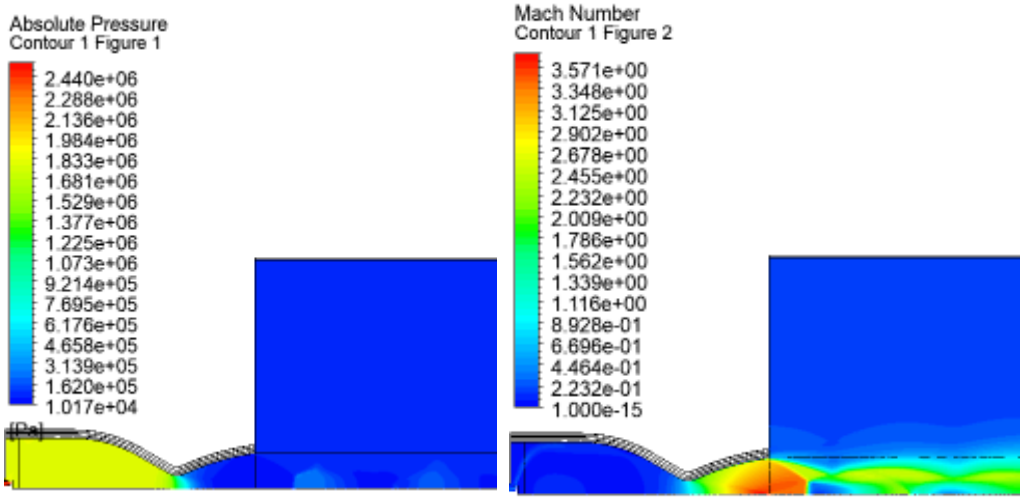
The difference between the SST-SAS and SST RANS models is in the addition of the Q_{SAS} term in the transport equation for the turbulence eddy frequency ω [11, p. 151]:

$$\begin{aligned}\frac{\partial \rho\omega}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j \omega) \\ = \alpha\frac{\omega}{k}P_k - \rho\beta\omega^2 + Q_{SAS} + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] \\ + (1 + F_1)\frac{2\rho}{\sigma_{\omega 2}}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}\end{aligned} \tag{4}$$

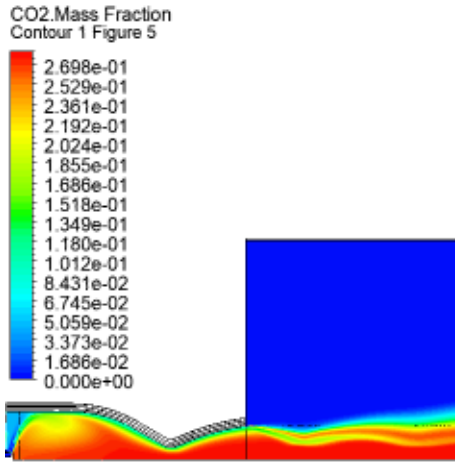
In this unsteady case, the computational effort is more demanding and for this reason we chose to restrict the hot flow domain only to the main combustion chamber. In this endeavor, we took advantage of the fact that in the outlet section of the nozzle, the flow is completely supersonic, meaning that no perturbations from downstream are propagating upstream. The initialization of the flow field, in this case, was done starting from the steady solution and preserving all boundary settings.

4. RESULTS

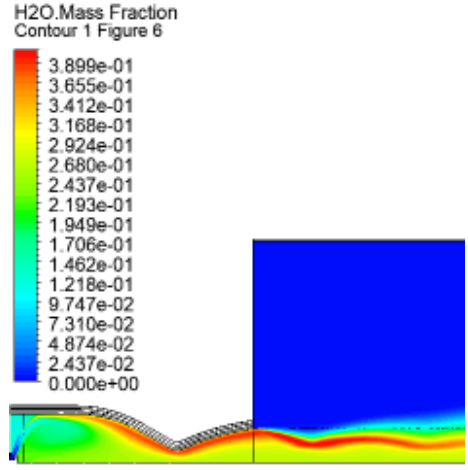
In the following pictures the main parameters distribution is depicted for the steady case. Two thirds of the outside environment have been cut for Fig. 5 a)-f) due to space constraints.



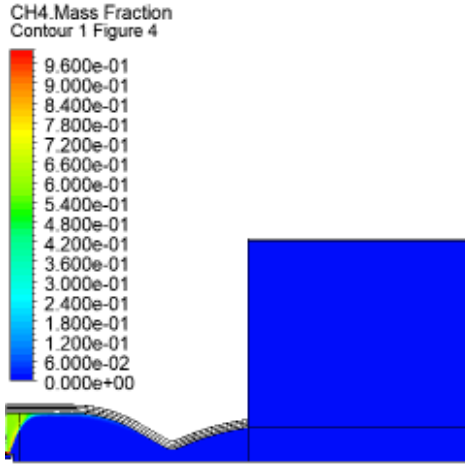
a) Absolute pressure b) Mach number



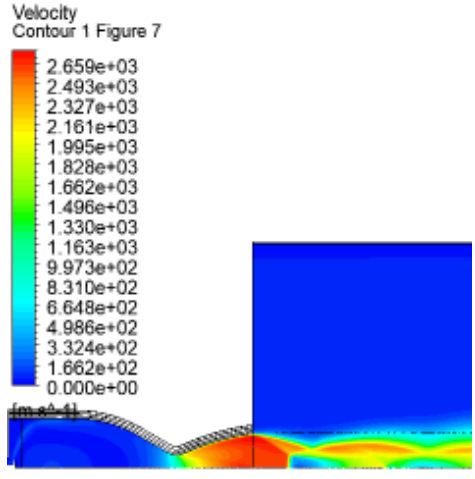
c) Carbon dioxide



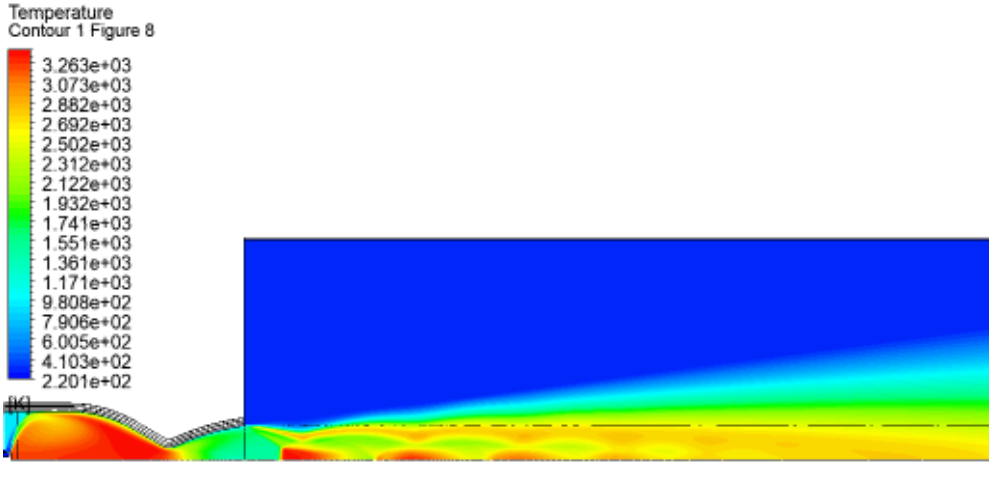
d) Water



e) Methane mass fraction



f) Velocity field

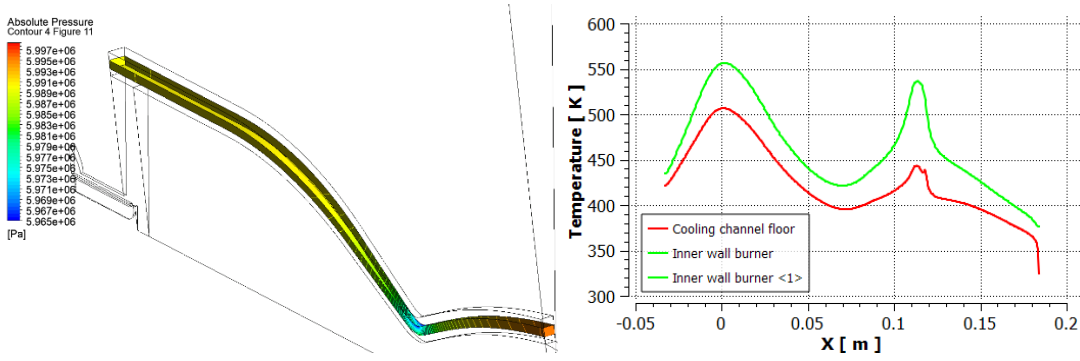


g) Temperature field

Fig. 5 Main parameters distribution in the front periodic plane

Fig. 6 Snapshot of the ground test of the engine

We noticed a realistic distribution of the major parameters for this steady solution. The combustion chamber has been properly designed so that no unburned methane flows through the nozzle. Regarding the temperature field (Fig. 5-g), the similarity between the computed field and the actual ground test image (Fig. 6) is encouraging. The computed thrust from CFX simulation was 1043 N and matches quite well the estimated design thrust. The measured thrust (the only parameter from the ground test) was 950 N.



a) b)

Fig. 7 Absolute pressure distribution in cooling channel (a); Temperature distribution on inner and outer wall of the combustion chamber (b). 1.5 kg/s case

Regarding the cooling water mass flow rate, two cases have been analyzed: 1.5 kg/s and 3 kg/s. It was noted that in the second case, the maximum temperature inside the channel is almost 100 K less than in the first case, not exceeding 450 K. Hence, the first cooling option was chosen. In this case (Fig. 7-b), the maximum temperature is still below the admissible operating temperature for OFHC which is around 1100 K. Also, at lower mass flow rate, the pressure loss in the channel is almost insignificant (Fig. 7-a) as the water does not vaporize and remains in liquid state all along the channel. Also, it can be noticed that due to the way the pintle injector operates, the temperature at the wall peaks two times: one time in the vicinity of the impingement point (x=0 m) and second time in the nozzle throat section. Even if the simulation did not account for radiation from the hot gases to the walls of the combustion chamber, we consider to be covered by the cooling system as the contribution from radiation does not add more than 100 K to the temperature distribution already estimated. With respect to the unsteady simulation, we were interested in comparing the steady RANS solution with URANS solution and check if the flame stays still well anchored on pintle injector.

Additionally, in this case we considered the film cooling of the pintle with gaseous oxygen. The results presented below were computed 0.6 ms later after the initialization with the RANS GEKO solution. The time step used in the unsteady simulation was 10^{-7} s.

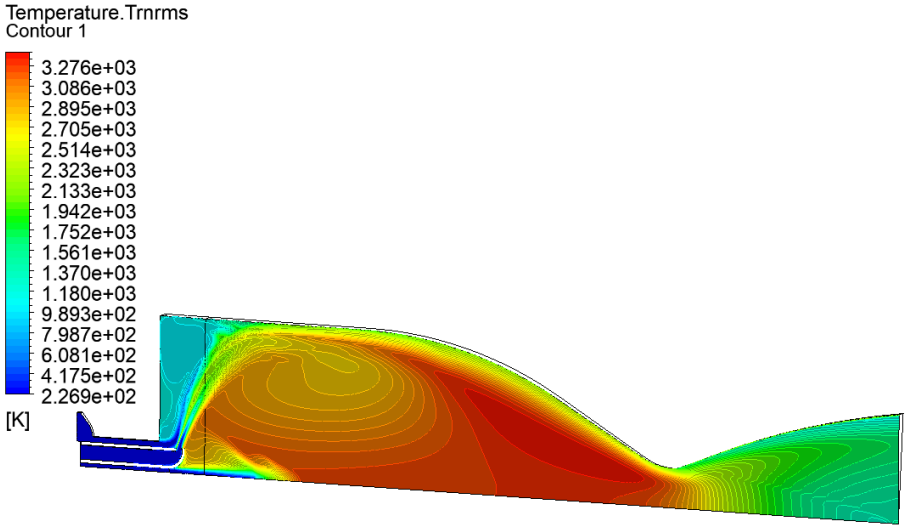


Fig. 8 RMS averaged temperature field at 0.6 ms

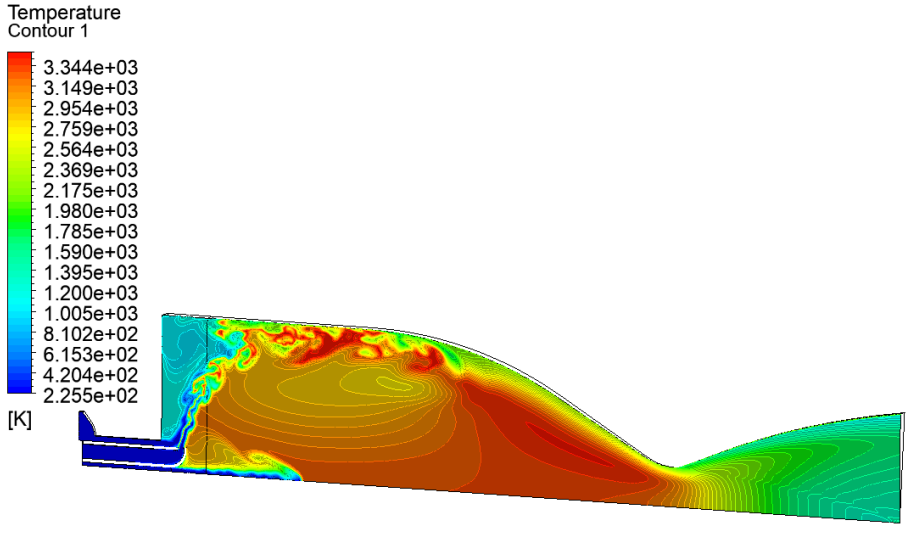


Fig. 9 Instantaneous temperature field at 0.6 ms

When comparing Fig. 8 with Fig. 5-g, we notice a strong consistency. In terms of the instantaneous distribution, we can notice that the flame front is very wrinkled especially in the vicinity of the combustion chamber wall.

Pockets of burning mixture are found also in the mixing zone and even close to the perpendicular wall of the pintle.

We conclude that the flame front remains anchored to the injector benefitting from the high temperature recirculation zone downstream the vertical wall of the pintle. Even if the pintle itself was film-cooled, the major part of the combustor domain was not sensibly influenced in terms of temperature distribution, at least far away from the axis.

5. CONCLUSIONS

The research regarding the development of a cheap active cooling system using water, for a ground test, has proven to be a success, with wall temperatures being well under the maximum operating temperatures for the chosen material. ANSYS proved to be an efficient tool for accelerating the development time of the cooling system, even though Fluent seems to lack some of the advantages CFX has regarding the use of multiple fluid types in the same domain. The combustion simulations also provided important information regarding the temperature distribution in the recirculation zone around the pintle tip, which exceeded the maximum operating temperature of the material, information which led to a design change in the geometry of the injector, preventing a catastrophic result. Also, as shown in recent papers on hydrogen and oxygen combustion [12, 13], the temperature falls within acceptable ranges when using the PDF flamelet model.

REFERENCES

[1] M. Pizzarelli, F. Nasuti, M. Onofri, Flow Analysis of Transcritical Methane in Rectangular, *44th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, 21 - 23 July 2008, Hartford, CT, AIAA-2008-4556, 2008.

[2] G. Waxenegger-Wilfing, K. Dresia, J. C. Deeken, and M. Oschwald, 2020, Heat Transfer Prediction for Methane in Regenerative Cooling Channels With Neural Networks, *Journal of Thermophysics and Heat Transfer*, Vol. **34**, No. 2, 347-357.

[3] M. Arun, Raj R. Amith and Prakash M. Jose, Numerical Analysis of Combustion and Film Cooling in LOx-Methane Rocket Engine, *International Conference on Aerospace and Mechanical Engineering, ICAME'18, Journal of Physics: Conference Series*, 1355 (2019) 012009, 2019.

[4] R. Schuff, M. Maier, O. Sindiy, C. Ulrich, and S. Fugger, Integrated Modeling and Analysis for a LOX/Methane Expander Cycle Engine: Focusing on Regenerative Cooling Jacket Design, *42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibi*t, 9 - 12 July 2006, Sacramento, California, AIAA 2006-4534, 2006.

[5] M. Pizzarelli, F. Nasuti, M. Onofri, Investigation of Transcritical Methane Flow and Heat Transfer in Curved Cooling Channels, *45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, 2 - 5 August 2009, Denver, Colorado, AIAA-2009-5304, 2008.

[6] M. Pizzarelli, B. Betti, F. Nasuti, D. Ricci, P. Roncioni, F. Battista, V. Salvatore, Cooling Channel Analysis of a LOX/LCH4 Rocket Engine Demonstrator, Propulsion and Energy Forum, July 28-30, 2014, Cleveland, OH, *50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference*, AIAA 2014-4004, 2014.

[7] M. Arun, M. Jose Prakash, Heat Transfer Deterioration Effects of Cryogenic Methane in Rocket Engine Cooling Channels, *Frontiers in Heat and Mass Transfer (FHMT)*, **1**1, 9 (2018), 2018.

[8] A. Ponomarenko, *RPA: Tool for Liquid Propellant Rocket Engine Analysis C++ Implementation*, May 2010.

[9] * * * *ANSYS Fluent Theory Guide Release 2022 R1*, January 2022

[10] * * * *ANSYS CFX-Solver Theory Guide Release 2022 R*1, January 2022

[11] Y. Egorov, and F. Menter, Development and Application of SST-SAS Turbulence Model in the DESIDER Project, *Second Symposium on Hybrid RANS-LES Methods*, Corfu, Greece, 2007.

[12] V. M. Zubanov, D.V. Stepanov, L.S. Shabliy, The technique for Simulation of Transient Combustion Processes in the Rocket Engine Operating with Gaseous Fuel Hydrogen and Oxygen, *Journal of Physics: Conf. Series 803* (2017) 012187, 2017.

[13] G. Ya. Gerasimov, O. P. Shatalov, Kinetic mechanism of combustion of hydrogen-oxygen mixtures, *Journal of Engineering Physics and Thermo physics*, 2013.