Synthesis of oxygen-methane combustion activities in In-Space Propulsion (ISP-1) Program

G. Ordonneau,
Onera - The French Aerospace Lab, F-92322 Châtillon, France
Correspondent author: gerard.ordonneau@onera.fr

Abstract

In the frame of the European framework programme 7 (FP7), the In-Space Propulsion (ISP-1) project was initiated in 2009 with the objective of improving knowledge and techniques required by future space missions relying on cryogenic propulsion. One of the work packages addresses remaining open issues for the use of oxygen and methane for in-space propulsion: liquid-liquid injection, ignition and combustion at low pressure, film cooling and soot formation. This paper deals with the work performed since the last Eucass symposium in 2011 and mainly devoted to the use of data acquired and models developed during the initial period in order to validate these models by CFD computations. Another part of the activity was the test of double swirled liquid injector. Through theoretical analyses, modelling and experimental work, this project has enable improving the maturity of technologies which are key elements of cryogenic space propulsion systems.

1. Nomenclature

CEA	Chemical Equilibrium with Applications
CFD	Computational Fluid Dynamics
EUCASS	European Conference for AeroSpace Sciences
DLR	Deutsches Zentrum für Luft- und Raumfahrt (German Aerospace Center)
LCH4	Liquid methane
LOX	Liquid oxygen
ONERA	Office National d'Etudes et de Recherche Aérospatiales (French Aerospace Lab)
URANS	Unsteady Reynolds Averaged Navier Stokes
LES	Large Eddy Simulation
LTCP	Low Thrust Cryogenic Propulsion
NHFR	Net Heat Flux Reduction
$P_{CC,0}$,	Chamber pressure without film
$P_{CC,film}$	Chamber pressure with film cooling
$\dot{q}_0$	Total mass flow rate without film
$\dot{q}_{film}$	Total mass flow rate with film cooling
RCFS	Regenerative Coolant Flow Simulation

2. Introduction

During the last decade, a renewed interest arose in methane as a liquid rocket propellant. There has been sporadic interest in this propellant since the early 1960s in the United States, however, no serious development activity or flight vehicle has used this relatively inexpensive and easily handled cryogenic fuel. Methane/oxygen rocket engines offer potentially significant life cycle mission advantages compared to traditional rocket propellants used in the United States today. Liquid methane (LCH4) and liquid oxygen (LOX) propulsion is very competitive based on bulk density impulse compared to current booster and in-space propellant combinations traditionally used today [1].
The likely potential for new rocket vehicle developments that support space exploration and the commercial market has revived an interest in methane oxygen propulsion systems. These propellants offer multiple advantages compared

to their counterparts. Major advantages include the capability of the fuel and oxidizer being stored at similar temperatures, non- toxic, relative ease of handling during launch vehicle operations.

In Europe, up to now, only preliminary studies and lab-scale testing activities have been carried out on methane ([2]-[5]). These activities have been performed mainly in the frame of medium launcher main stage propulsion, for which the global advantage of methane is presently not clear, considering in particular the development effort. But, if we consider the application to in-space-propulsion combine with innovative concepts, and materials, lox-methane propulsion is a good candidate. Nevertheless, basic researches are required before considering an operational engine development. In addition, in space propulsion introduces new requirements and possibly the use of new technical solutions that haven't been studied so far.

To improve the knowledge and the techniques which are necessary for future space missions' cryogenic propulsion, the project In-Space-Propulsion (ISP-1) was initiated in 2009 within the framework of the European FP7 programme. ISP-1 does not focus on the early launch phase of a space flight, but on the technologies involved in the subsequent phases of a space mission, once the spacecraft or upper stage has already been placed in orbit. Hence the program name “In Space Propulsion”, which applies to launcher upper stages, orbital transfer vehicles or space exploration vehicles. The activities in ISP-1's different work packages are focussed on the concept of Low Thrust Cryogenic Propulsion, which have been presented in previous publications.
The ISP-1 program is structured into five main work packages which deal with various technological difficulties associated to the development of a Low Thrust Cryogenic Propulsion system [6] and serve the purpose of improving the maturity of technologies which are key elements of cryogenic space propulsion systems.

The second work package of ISP-1 deals with oxygen/methane propulsion. Indeed, this propellant combination is a possible answer to the contradictory requirements of performance and long duration in space. Consequently, the project will aim at extending the present knowledge on LOX/methane combustion to its use at operating conditions typical for in-space application through research on injection, ignition, combustion, soot formation and engine cooling six main tasks have been defined to progress along the main research axes. They combine modelling activities for propellant properties and combustion with simple configuration tests to validate modelling tools and more representative tests to increase the technical knowledge in liquid/liquid injection of cryogenic propellants.

Each work package is led by a main contractor assisted by several partners who are in some case major universities, taking advantage of the know-how of large or smaller industrial organizations combined to the theoretical expertise of research laboratories. The LOX-methane combustion work package is led by ONERA (France) with main participation of University of Roma (Italy), DLR and Astrium (Germany) [6].

In 2011, activities on reference engine definition, physical and chemical characterization of LOX/methane system, steady state combustion and ignition data acquisition have been presented in the framework of the 4th Eucass conference [7]. The present paper deal with the remaining activities: CFD simulation of steady state and ignition, liquid/liquid experiment and film cooling experiment and simulation. In each case, several approaches have been considered as far as possible and are presented in this paper.

3. CFD ignition

The CFD computation simulates tests realized on the test bench M3 at DLR Lampoldshausen, which has been presented in details in several papers ([7]-[10]).
The quantitative analysis of the events connected with the ignition sequence in the M3 chamber can be carried out by adopting the assumption inherent in the so-called "well-stirred reactor" (WSR) or "continuously stirred tank reactor" (CSTR) models. These models allow carrying out the sensitivity analysis with respect to many important system parameters (pressures at manifolds and at outflow, chamber volume, fuel and oxidizer time sequence parameters, type of reactants, power level of the energy deposition, equivalence ratio of reactants, type of inert

filler, geometry of injector pipes and nozzle). Therefore, a preliminary question involves motivating the making of costly and tedious CFD analyses, such as those based on URANS or LES models. In fact, there are at least two very significant transient phenomena that cannot be replicated with the assumption of perfect mixing, because they are inherently multi-dimensional in nature (at least two-dimensional axisymmetric), and are (i) the development of a large recirculation region surrounding the jet emanating from the co-axial injector, and (ii) the occurrence of a strong blast wave [11] triggered by the laser pulse used for ignition. Both phenomena crucially affect the modalities of ignition and re-ignition of the cold reactant jet entering the chamber from the single co-axial injector.
URANS and LES approaches have been envisaged for this calculation. All the details for the ambient calculations can be found in ([10],[12]). Both URANS and LES simulations predicted larger-than-experiment chamber pressure peak value and growth rate. However, 2D axisymmetric URANS and 3D LES simulations agree each other by a large extent. It is therefore conjectured that a possible mismatch with the experimental data might be the cause of this discrepancy. Other possible causes can be searched among those that might produce a larger-than-experiment amount of CH4 and O2 in the chamber at initial time T0, say for example the neglect of the amount of nitrogen filling the propellant manifold (volume of pipe(s) between the probe and the boundary conditions, or, a too quick spread of the supersonic oxygen jet by numerical dissipation, which might cause an excess of oxidant in the chamber at the instant where calculation begins. Given that the URANS simulation has been extended for a small fraction of the test duration (150ms vs 4s), there are important questions still unanswered, and among the most important we have: might other re-ignition occur given that a significant amount of hot products are still present in the chamber? If re-ignitions occur, will the flame be able to stay "attached"? And in this case, what is the anchoring mechanism? The experience gained in this work indicates that (i) URANS axisymmetric calculations can be effectively able to provide a rather detailed picture of the ignition events, albeit there remains a number of issues for the quantitative accuracy of the URANS predictions, (ii) 2D axisymmetric URANS and 3-D LES provide predictions in satisfactory agreement, even when rather different kinetic mechanisms have been adopted. The CFD analyses here reported have offered interesting contributions in the understanding of a number of critical ignition phenomena, which are difficult to appreciate on the basis of experimental diagnostics alone.

The low pressure calculation allows to improve turbulent combustion modelling and to apply the methodology previously developed at ambient pressure and smooth ignition conditions to a different case at low pressure and strong ignition conditions.

Instead of using a mixing step to take into account turbulent limiting effects, as proposed by Grenard and Vingert [13] and used in ambient case CFD study [10], a thickened flame model was used to better discretize the flame front and to reproduce flame wrinkling with turbulence. The Thickened Flame Model for LES (TFLES), which was implemented in Cedre by Sainte-Rose ([14],[15]) and used in this study, increases the flame front thickness to spread it on more points on coarse grids. The aim is to give access to the accurate laminar flame speed in spite of too few points to discretize the flame front. The Dynamic Thickened Flame (DTF) formulation of the model, proposed by Legier et al. [16], was preferred to the original formulation because reactant mixing in the present study is partial due to separate injections.
This computational strategy was able to reproduce the combustion chamber pressure rise during ignition and flame growth with a strong ignition process (Figure 1). High-frequency pressure fluctuations are discrepancies may caused by the ignition procedure and combustion modelling, which have to be improved. Finally, flame lift-off is not well reproduced because of not detailed chemistry modelling (Figure 2).

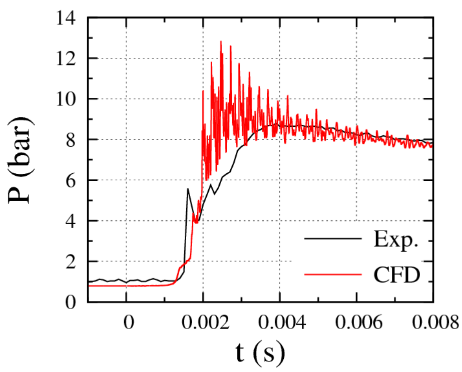

Figure 1 : Temporal evolution of chamber pressure compared to measured signal..

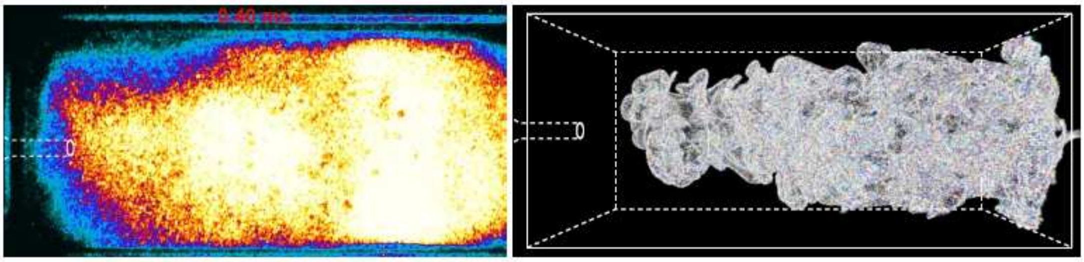

Figure 2 : Comparison between measured OH* field (left) and CFD heat release rate (right).

Regarding CFD, firstly a very important work has been done to simulate ignition of DLR M3 combustor. Model comparison, on one hand, between RANS and LES, and on the other hand, between ambient pressure conditions and low pressure conditions, has shown that modelling allows simulating the main phases of the ignition sequence. Areas for improvement and “good practice” were identified during a dedicated workshop. Particularly, code used must be able to deal correctly with highly compressible flows, since there are shocks and high Mach number in the injector and a blast wave at ignition. The increasing complexity approach should be generalized, as it permits at very low cost to evaluate a first order of magnitude of global phenomenon. On counterpart, ignition procedure seems to have quite low influence on later development of the flame: all procedures that give sufficient amount of energy in the chamber on a sufficient spatial zone are equally good for starting the ignition.

4. Film cooling

4.1. Experiment

An important open issue of the LTCP engine is the chamber cooling. Indeed, regenerative cooling requires a higher pump power due to the pressure loss in the regenerative circuit, which is contradictory to the stage requirement. Consequently, it is assumed that film cooling is the only cooling technique to be applied. Film cooling experiments were performed on P6.1 test bench of DLR [17].
For this purpose DLR has manufactured a gas/gas injector head, which has been installed a segmented (6 segments each 50 mm long) water-cooled combustor using an existing nozzle with a contraction ratio of 3.2 (Figure 3).

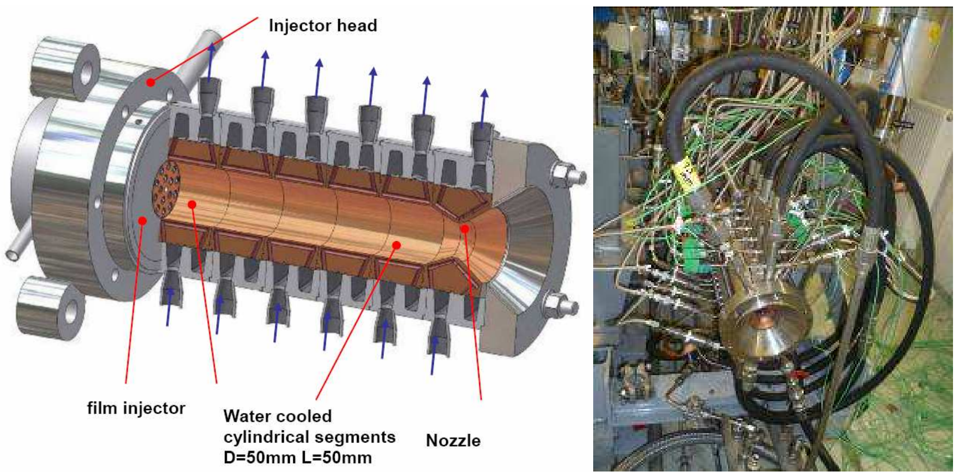

Figure 3 : DLR Segmented combustion chamber model “B”

Injector head contains central igniter and 15 coaxial injector elements without recess and with tapering of 10°. The injector elements are positioned on two different pitch circles with two diameter d1 = 19 mm and d2 = 38 mm. The 5 injector elements have been arranged in inner row and 10 injector elements in outer row. The elements form 5 identical triangles. This pattern design provides uniformly distribution on the local combustion zones and corresponding ensures the uniform thermal loads on the hot gas surface of the combustion chamber wall. Film injection is implemented between the injector head and the measurement segment. The methane at ambient temperature flows from ring slot [17]. In the presented test row two slot dimensions, 0.46 mm (small) and 1.0 mm (large), have been employed. The first slot dimension corresponds to a velocity ratio v_{cc}/v_{film} (v_{cc}- calculated velocity of the hot gas, v_{film}- film injection velocity) of nearly 1 and the second slot dimension to a velocity ratio of nearly 0.5.

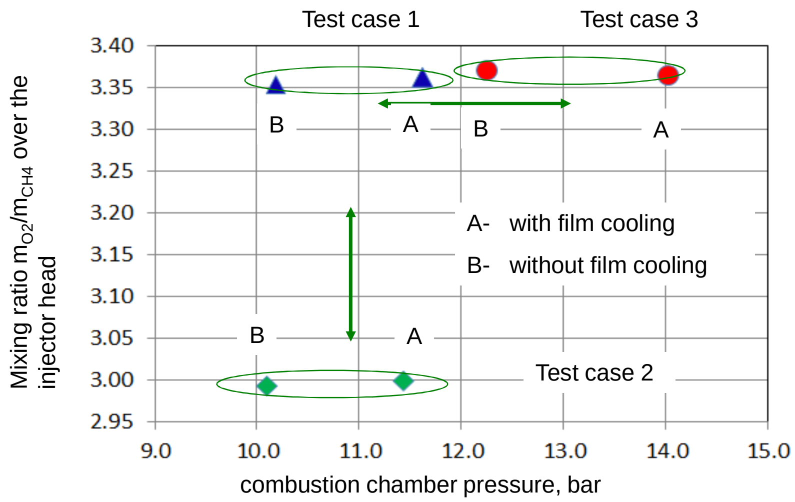

Figure 4 : Test conditions

Each test case has two load levels; with film cooling (A) and without film cooling (B). The test conditions without film cooling have been used as a reference for the determination of the cooling film efficiency. To make a better comparison of the test results the mass flow over the injector head has been kept constant independent from cooling film mass flow. It leads to an increase of the combustion chamber pressure by film cooling due to additional methane mass flow in to combustion chamber

The heat flux to the water-cooled wall and heat transfer coefficients will be calculated using a calorimetrical method. The flux is computed knowing inlet and outlet temperature, and the mass flowrate, deduced from pressure measurement as well. Additional temperature sensors measure the surface temperature in the segment structure 1mm from the hot gas surface. Combining with the measurement of wall temperature and chamber pressure leads to the evaluation of the heat transfer coefficient for each segment. The measurement error has been estimated as ca. ± 5% for heat flux density and ca. ± 15 K for surface temperature at current thermal loads.

By investigation at the rocket engine typical test conditions it is impossible to reach adiabatic conditions by low cooling film mass flow. For the protection of the test specimen additional convective cooling is necessary. Therefore it would be not correct to use for the adiabatic the wall typical cooling film efficiency criteria. For the determination of the cooling film efficiency the (NHFR) has been used:

$$NHFR = 1 - \frac{\dot{q}_{film}}{\dot{q}_0} \cdot \left(\frac{P_{CC,0}}{P_{CC,film}} \right)^{0.8} \tag{1}$$

The equation part $\left(\frac{P_{CC,0}}{P_{CC,film}} \right)^{0.8}$ is based on the proposal by Bartz and considers the influence of the increasing combustion chamber pressure due to an injection of the coolant film. Examples of results are given Figure 5

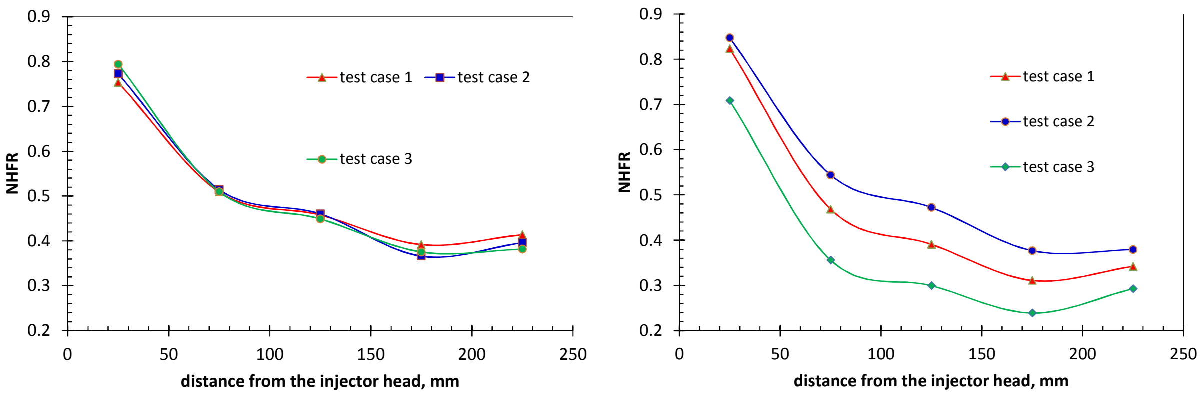

Figure 5 : cooling film efficiency, small slot (left) and large slot (right)

Test results will be used to evaluate CFD tools. For this purpose computations will be carried out at Astrium, DLR and University of Rome. Preliminary results are reported in [18].

4.2. Calculation

Computations are performed following two approaches: in the first one, load points are simulated with Rocflam-II, Astrium's standard tool for combustion chamber analysis and design. In the second one, load point simulations are carried out with a suitably simplified RANS approach. The results are evaluated and compared to test data in order to identify possible subjects for simulation model improvements.

Rocflam-II is Astrium's axis-symmetric Navier-Stokes Solver with incorporated Lagrange module, which allows the injection and tracking of liquid droplet parcels. A standard k-ε turbulence model is implemented with a 2-layer approach at the wall. The CH_4/O_2 combustion chemistry is treated by a ppdf chemistry model (equilibrium, taking into account turbulent combustion). The following species are considered: CH_4, O_2, CO, CO_2, H_2, H_2O (gaseous, liquid and solid), OH, H, O. Rocflam-II is Astrium's standard tool for analysis and design of rocket combustion chambers and in-orbit thrusters. The coolant channel system is simulated with RCFS-II, Astrium's Nußelt correlations based engineering tool for the design of cooling channels and coupled with the Rocflam-II simulation for the combustion process. Within this coupling the wall heat flux is derived from the Rocflam-II simulation and the resulting wall temperature is determined with RCFS-II. As the measured flow rates are prescribed, calculated pressure can be directly compared to the experimental results.

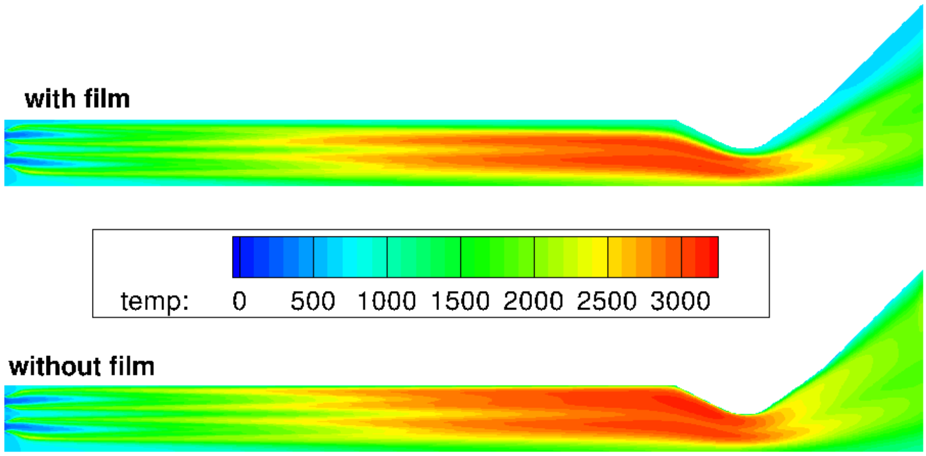

Figure 6 : Temperature distribution inside combustion chamber B for test cases 1 (bottom, without film) and 2 (top, with film)

All the nine computation results showed that a good agreement between test and simulation can be reached. The maximum deviation reaches about 3 % for test cases without film, about 6 % for test cases with film for a large film slot and an almost perfect matching for a small film slot. This comparison can be used as an indication for the

prediction quality of combustion efficiency achieved. Concerning heat fluxes, a comparison of the simulated wall heat flux with the measured values reveals that the plateau value, corresponding to the downstream chamber wall before nozzle entrance, is predicted a little bit too high for all cases; however, the overall agreement between test and simulation is quite good. Nevertheless, hot gas wall temperature distribution obtained for test case 1 and 2 compared to values derived from test data (approximated values from thermo couples installed in a bore at 1 mm distance of the hot gas surface) show that values derived from test data are significantly lower than the predictions. However, the test values seem to be very low − approx. 400 K along the entire combustion chamber without film cooling at an increasing combustion temperature reaching more than 3000 K towards the throat.
To take into account the mass flow variation, NHFR is also evaluated for the performed simulations and compared to test data. For all six diagrams the NHFR in test is significantly higher for the first chamber segment compared to simulation. For the segments further downstream the agreement is very good for the NHFR between test case 1 and 2 and between 1 and 6 but still improvable for the other test cases. However, it has to be mentioned that the prediction of the mixing of the cold coolant film and the hot combustion gas is a challenging task, especially when taking into account that this is the first approach in modelling the combustion of gaseous CH4 and gaseous O2 with Rocflam-II.

At the same time, a simplified two-gas approach has been pursued by Uniroma ([17],[19]), using an in-house Reynolds-Averages Navier Stokes equations solver, in order to understand the main phenomena affecting wall temperature and heat flux in film cooled combustion chambers.

In fact, in large engines turbulence 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. 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 the present simplified approach, near injector plate phenomena are neglected and equilibrium combustion products are injected in the chamber at the adiabatic flame temperature. Mixture composition and adiabatic flame temperature are evaluated by means of the CEA program [20], prescribing stagnation chamber pressure and propellant mixture ratio. Mixture composition is used to evaluate hot gas properties throughout the chamber. In fact, the simulations are carried out considering the interaction of two gases with frozen composition: “hot gas” and “coolant”. Adiabatic flame temperature is enforced together with mass flow rate and the assumption of axial flow direction as hot-gas inflow boundary condition. Mass flowrate, total temperature and axial flow direction are also assumed as inflow boundary conditions for the coolant. Note that the coolant injector slot is actually an axisymmetric slot in the experiments, so the enforced boundary conditions enable to reproduce all experimental coolant injection features, including injection velocity. On the contrary, the simplified injection assumption for hot gas does not allow preserving all injector parameters. In fact, once hot-gas mass flow rate is prescribed, the axial injection velocity is a result of the axisymmetric inlet section chosen in the simplified approach. In the present study, it has been considered important to preserve distances between the hot gas flow and the coolant film and having a hot gas injection velocity that is representative of the actual average velocity of the burned mixture in the chamber. The injection section has been therefore chosen as the annular section within the outer injector ring. As no coupling with the coolant circuit is available, experimental wall temperature are prescribed as thermal boundary condition at the chamber wall.

The numerical grid has been selected after a grid convergence study on similar test carried out for oxygen/hydrogen thrust chamber. The adopted grid together with a typical computed temperature flowfield is shown in Figure 7.

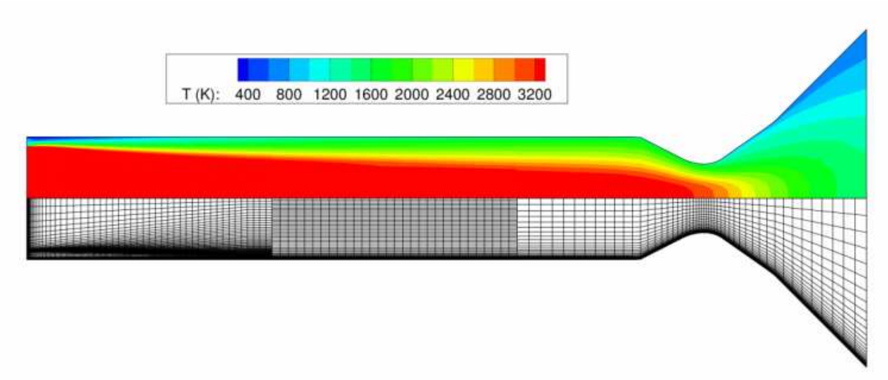

Figure 7: Temperature flowfield and mesh arrangement

Results show that, in this case of low wall temperature and large film coolant mass flowrate, the most relevant phenomena for the determination of heat exchange are mixing of hot gas with methane and convective wall heat transfer. In fact, a good agreement between experimental measurements and the present simplified approach is obtained for all the three test cases. It can be observed that a slightly smaller heat flux is computed at the measurement point closer to injector plate. Discrepancies in this region are expected because of the present simplified modelling of the region close to injector plate. In particular, before diffusion of the hot gas through the whole available cross section of the chamber, actual hot gas velocity is locally larger than the average value used in computations, thus affecting mixing and heat exchange. Considering the two other test cases, differences between computations and measurements are of the same order. Comparison between test cases 1A and 3A shows the role of chamber pressure, and the increase of heat flux as a consequence of increased chamber pressure. Note that data of two experimental runs of test case 3A are shown to emphasize the experimental measurement spreading error. On the basis of the available experimental inflow and wall temperature data, the resulting numerical computations provide quite superimposed lines, whereas heat flux measures show a more evident shift. The second comparison, between test cases 1A and 2A, shows that, in agreement with experimental data, changing the mixture ratio of the hot gas does not affect much heat flux. There is only a slight effect in the region close to the injector plate (first thermocouple) which is not captured by the numerical solution.

As a conclusion on heat transfer modelling, both approaches show overall good agreement with experimental data. The two-gas approach provides a reliable wall heat flux prediction in the case with film cooling, which could be used for preliminary design in case of film cooling. However, at least a simplified flame computation is needed in the case without film cooling to reproduce the experimental wall heat flux trend, as shown in the results of the coupled approach.

5. Liquid/liquid swirl injector

One of the objectives of ISP-1 project is getting a know-how on liquid/liquid injector. During the first part of the ISP-1 project, a double swirl injector has been selected. The selection has been made on the basis of several weighted criteria combined with several injector types [7]. Cold flows tests, with water and then with actual propellants, have been carried out, followed by hot fire tests [21]. In fact, two injector configurations were used; they differ from each other only by the number of methane inlet holes and the thickness of the lip of the LOX post. Indeed, the first one, referred as Inj#1, is composed of a LOX post of 0.3 mm lip thickness and the LCH4 injector is fed with 6 holes of diameter dpCH4. Inj#2, is composed of a LOX post of 1 mm lip thickness and the LCH4 injector is fed with 4 holes.
In such an atomization device, a gas core exists in the jet, and the liquid is limited to a thin sheet of a few hundred microns of thickness, the exact value depending on the fluid. The recess number (*RN=Lr/Lc* where *Lc* is the length between the LOX post exit and the point where the LOX sheet impacts the internal wall of the CH_4 post) is an important parameter which quantifies the internal impinging position of the LOX sheet on the fuel post. The break-up length seems to increase with the increase of the recess number. Kim et al. [22] defined three different injection regimes: external, tip and internal mixing injection. The recess number in our case belongs to $\{0; 0.7; 1.3\}$. Thus, the first two cases (*RN*=0 and *RN*=0.7) are considered as external mixing injection, for which the interaction of the two liquid sheets is not affected by the outer wall. *RN*=1.3 is considered as internal mixing injection, i.e. the interaction of liquid sheets occurs inside the injector.

For optical measurements, we used two high-speed intensified cameras, one operated by Onera and one by DLR, to record the emission signals of OH* and CH radicals. Results were compared with the ones obtained from a shear coaxial injector, commonly used in cryogenic rocket engines. The combustion stability behaviour of the flame was also investigated, based upon the injector geometry.
When actual fluids were injected separately (Figure 8), the cone angle of the oxygen spray was about 90°, whereas the one of methane was approximately 120°, as indicated by dotted lines in Figure 8. For the recessed injector *RN*=1.3, and *ROF* = 1, the two sheets interpenetrate on the LCH_4 post wall, and probably lead to a much better mixing.

During the hot fire test, combustion exhibits some rough combustion with frequency around 50 Hz which is visible for recessed injector. Without recess this level of fluctuation is quite low. This instability was clearly coupled with the liquid oxygen feed line. This fluctuation didn't appear during preliminary tests where methane temperature was probably slightly higher.

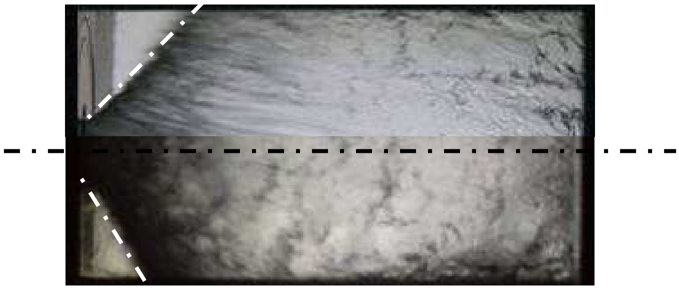

Figure 8 - Results obtained for actual propellants cold flow tests with the biswirl Inj#1without recess: Oxygen side alone (top) and methane side alone (bottom). The dotted lines indicate the fluid cone angles.

Beside low frequency instabilities and thermal stresses of the windows, a third difficulty was encountered during this campaign: the frequent occurrence of hard starts. A pressure peak occurred in the LCH4 injection line, not immediately at the ignition but during the transient at the moment when the liquid methane first reached the flame zone.
Moreover, an ice film always appeared on the upstream side of the windows due to the mixing of the water vapour contained in the combustion products and liquid methane, yielding to dark images for the close view images and inducing difficulties to obtain information on the corresponding part of large view images. Thus, in order to visualize the vicinity of the injector, images were also taken during the ignition period. These are nevertheless not completely relevant for the objective of the study because in the beginning of this phase, the injector doesn't work in liquid/liquid but in liquid/gas conditions (liquid oxygen and gaseous methane). The flame anchoring is located at the tip of the swirl injector. This scheme is different from the coaxial injector case, for which the flame behaves more like a jet flame, with a narrow radial spread (~30°) and a high OH* signal in the central zone along the injection axis. Indeed, the bi-swirl exhibits a bulb probably due to the presence of a central recirculation zone, which is typical for this kind of swirl-stabilized flames Figure 9.

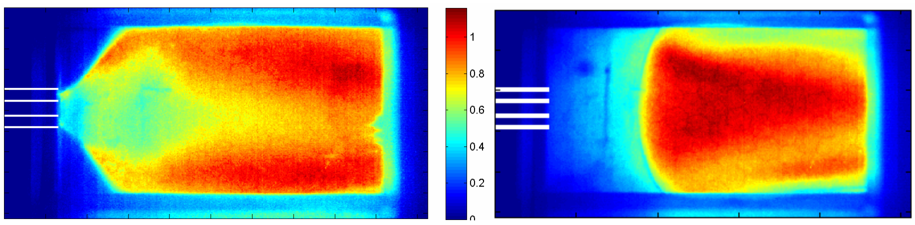

Figure 9 : Example of flame during ignition (left) and main stage (right)

As a conclusion, results are compared with a coaxial injector reference case and with literature. The double swirl injector produces a more compact flame, compared to the coaxial injector for which the flame looks more like a jet flame. The flame seems to be anchored at the tip of the injector for all cases. For the bi-swirl injector alone, the effect of the recess seems to produce a narrower and longer flame. Based upon the results obtained in this study, the best setup for bi-liquid swirl injection seems to be a moderate LOX post recess length, (*RN* < 1), as it produces a relatively smooth combustion and a better mixing of the propellants.

6. Conclusion

This 3-year project combined modelling and experimental works leading to test case use in CFD computation.
Over these three years, several models have been derived following three directions: species thermodynamics. First of all, concerning the species involved in Lox/Methane combustion a database and equation of state for mixture have been derived for large range of temperature and pressure, at the operating conditions for ISP-1 target but also for high pressure.
For combustion, a low pressure chemical kinetics, with 24 species and 103 reactions has been established. But it requires a lot of computing time, thus it was partially applied in ignition CFD computation. Such calculation could be a good candidate for European HPC projects (PRACE).
Combustion with hydrocarbons means also soot. A lot of activities have been carried out for methane combustion and several models exist in literature but in air. A trial of adaptation of such a model to pure liquid oxygen has been

attempt but this yields unrealistic concentration at high temperature. It appears clearly that an effort must be continued and new data on soot production must be acquired for combustion in pure oxygen.
Despite some difficulties related to any experimental work, we managed to provide databases for ignition and combustion of oxygen / Methane jet and heat transfer in the presence of film cooling as well.
Various type of CFD calculations, from simplified 2D two-fluid approach up to fully 3D LES, applied to these test cases have also enabled to implement a wide range of model and they have broadened the range of validity of the tools.
Finally, the design, fabrication and testing of a double swirl injector have enabled to collect useful information on the efficiency and stability of combustion as well as the shape of the flame that will be very useful for the final selection of a engine for in-space propulsion.

7. Acknowledgments

This work was performed within the "ISP-1" project, coordinated by SNECMA, and supported by the European Union within the 7th Framework Program for Research & Technology. (Grant agreement N° 218849.). Many colleagues and researchers participated to the making of the "ISP-1" project, namely Prof. F. Nasuti, Prof. D.Lentini, Prof. M.Valorani, Dr. E. Martelli, Dr. A. Minotti, Dr. M. Pizzarelli, PhD students B. Betti, P. Ciottoli, C.M. Mazzoni, and graduate fellow G.Gargiulo from university of Rome, Pr. O Haidn from TUM, J. Sender, Dr. R. Stützer, Dr. D. Suslov from DLR, P. Grenard, N. Fdida, L. Vingert, from ONERA.

8. References

[1] T. Neill, D. Judd, E. Veith, D.Rousar, Practical uses of liquid methane in rocket engine applications, Acta Astronautica 65 (2009) 696 –705

[2] Preuss, A., Preclik, D., Maeding, C., Soller, S., Haidn, O., Oschwald, M., Clauss, W, Torres, Y and Sender, J.: 2008, LOX/Methane Technology Efforts for Future Liquid Rocket Engines, *Space Propulsion Conference, Heraklion*, Crete,

[3] Zurbach, S., Thomas, J. L., Vuillermoz, P., Vingert, L. and Habiballah, M.: Recent Advances on LOX/Methane Combustion for Liquid Rocket Engine Injector, AIAA-2002-4321, *38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit*, July 7-10, 2002, Indianapolis, IN

[4] Zurbach, S., Thomas, J. L., Verplancke, C., Vingert, L. and Habiballah, M.: LOX / Methane Studies for Fuel Rich Preburner, AIAA 2003-5063, *39th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit*, July 20-23, 2003, Huntsville, AL

[5] M. Calabro, Innovative LOX/Methane Upper Stage for Future Launchers, In proc. of *International Symposium on Space Propulsion 2004*- Shanghai- August 25-28

[6] M. Muszynski, P. Alliot 2010. The In-Space Propulsion (Isp-1) Project. *61th International Astronautical Congress; Prague.*

[7] G. Ordonneau, O. Haidn, S. Soller, M. Onofri, Oxygen-methane combustion studies in the In Space Propulsion Programme, *4th European Conference For Aerospace Sciences (EUCASS) 2011*

[8] V. Schmidt, D. Klimenko, O. Haidn, M. Oschwald, A. Nicole, G. Ordonneau, M. Habiballah, 2004, *ONERA, TP no. 2004-49*

[9] Lacaze G.; Cuenot B. ; Poinsot T. ; Oschwald M., Large eddy simulation of laser ignition and compressible reacting flow in a rocket-like configuration, 2009, *Combustion and Flame, vol. 156, no6, pp. 1166-1180*

[10] M.Valorani, P.Grenard , J.Sender, Experimental and Numerical Analysis of Ignition in a GO2/GCH4 Co-Axial Injector at Ambient and Low Pressures, *Space Propulsion 2012* (May 2012)

[11] E.A. Hurlbert, R.J. Moreland, and S.Candel, Propellant ignition and flame propagation, in Liquid rocket thrust chambers : aspects of modeling, analysis, and design, vol. 200, 2004, pp. 406-435.

[12] M. Valorani, E. Martelli, G. Gargiulo, P. Grenard, J. Sender, CFD Simulation of Laser-Pulse Ignition of GCH4/GO2 in M3-DLR Test Bench, *Space Propulsion 2012 (May 2012)*

[13] P. Grenard and L. Vingert. CFD simulation of the GCH4/GO2 MASCOTTE test bench. *In Proceedings of The 3rd Int. Conference on Space Propulsion*, 2012.

[14] B. Sainte-Rose, N. Bertier, S. Deck, F. Dupoirieux, Numerical simulations and physical analysis of an overexpanded reactive gas flow in a planar nozzle, *Combustion & Flame Volume 159, Issue 9, Pages 2777-3002 (September 2012)*

[15] A. Refloch, B. Courbet, A. Murrone, P. Villedieu, C. Laurent, P. Gilbank, J. Troyes, L. Tessé, G. Chaineray, J.B. Dargaud, E. Quémerais, F. Vuillot, CFD Platforms and Coupling -CEDRE Software, Aerospacelab-

journal n°2 (mars 2011)
(http://www.aerospacelab-journal.org/al2)

[16] J. P. Legier, T. Poinsot and D. Veynante. Dynamically thickened flame LES model for premixed and non-premixed turbulent combustion. *In Proceedings of The Center for Turbulence Research, 2000*

[17] D. Suslov, B. Betti, S. Soller, Prof. F. Nasuti, Prof. O. Haidn, G. Ordonneau Experimental Investigation and CFD-Simulation of the Film Cooling in O2/CH4 subscale Combustion Chamber, *Space Propulsion 2012*

[18] Pizzarelli M., Betti B., Nasuti F., Coupled analysis of hot-gas and coolant flows in LOX/methane thrust chamber, 2011, *4th European Conference for Aerospace Sciences, EUCASS, Saint Petersburg, Russia*.

[19] B. Betti, E. Martelli, F. Nasuti , M. Onofri, Numerical Study of Heat Transfer in Film Cooled Thrust Chambers, *48th AIAA Joint Propulsion Conference AIAA 2012-3907* (2012)

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

[21] N. Fdida, L. Vingert, G.Ordonneau, A. Ristori, C. Brossard, R. Stützer, J. Sender, Characterization of a double swirl injector in a high-pressure LOX/LCH4 rocket engine combustor, *5th European Conference For Aerospace Sciences (EUCASS) 2013*

[22] Kim, D., Han, P., Im, J., Yoon, Y., Bazarov, V. G. 2007. Effect of the recess of liquid-liquid swirl coaxial injectors, *Journal of Propulsion and Power*, vol.23, no. 6.