3D SIMULATION OF LOx/GH2 MASCOTTE TEST CASE AT 10 BAR

*Aurélie NICOLE * - Gérard ORDONNEAU** - Marie THERON****

ONERA– Office National d'Études et de Recherche Aérospatiale, Châtillon, France

BP 72
29 Avenue de la Division Leclerc
92322 CHATILLON CEDEX
FRANCE
Tel : 33 1 46 73 43 03
Fax : 33 1 46 73 41 47
Email : aurelie.nicole@onera.fr
* research Scientist
** Head of "Liquid Propulsion, Heat transfer and Advanced Propulsion Systems" Unit
*** CNES Evry - DLA/SDT/EPM
Rond-point de l'Espace
91023 EVRY-COURCOURONNES Cedex
FRANCE

1 Context

In the context of cost reducing and high concurrence between space launchers, the design of rocket engine appeals more and more simulation tools in order to reduce conception delays and development costs. These simulation tools require a detailed validation which relies on experimental means like MASCOTTE test bench and advanced diagnostics [1]. The work described in this paper has been achieved in this context and in the framework of the CNES/ONERA common programme on "liquid propelled rocket engine" [2]. The objective of this work is the 3D simulation of the LOx/H_2 cryogenic combustion in subcritical regime for the MASCOTTE test configuration. It takes advantage from the work presented in reference [3].

2 Description of the MASCOTTE bench

The MASCOTTE test facility has been developed by ONERA for fifteen years to study elementary processes like atomization, droplets vaporization, turbulent combustion, which are involved in the combustion of cryogenic propellants. Several versions of MASCOTTE are available. In this paper, we have chosen to investigate the A-10 point which corresponds to an Oxygen/Fuel ratio of 2.11 and a chamber pressure of approximately 10 bar. In the experiments considered, MASCOTTE test-bench is equipped with a subscale mono-injector combustion chamber of 50 mm x 50 mm square internal section (Figure 1). The chamber is equipped with side windows which are cooled by a helium film, for visualization purposes and use of several optical diagnostics. The total length of the chamber from the injector exit section to the throat is 478 mm but it should be noticed that the injector overtakes of 10 mm in the chamber in order to see the injector exit through the portholes. The throat diameter is equal to 15 mm.

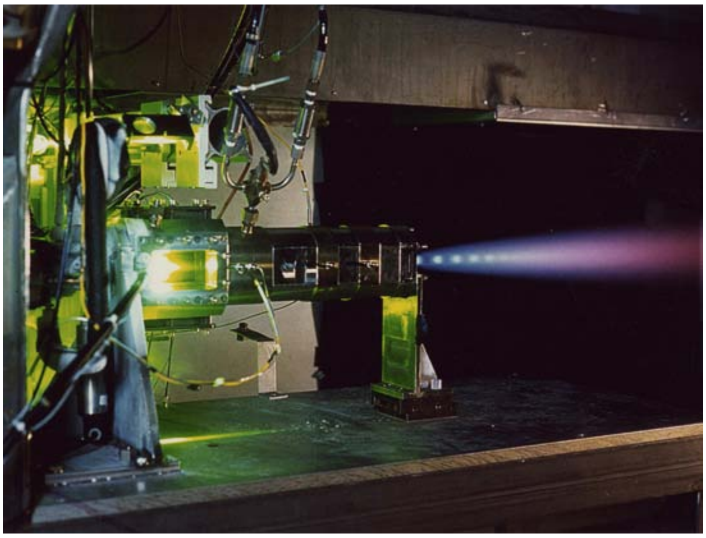


Figure 1 – MASCOTTE test bench.

Temperatures in the hot gas flow are measured using CARS techniques [4]. The MASCOTTE injector is a coaxial injector where the liquid oxygen is injected at the center and the gaseous hydrogen at the periphery.

The studied working point is named A-10. The mass flow rate is 50 g/s, resp. 23.7 g/s and 10.12 g/s for oxygen, resp. for hydrogen and helium. So the mixture ratio is equal to 2.11. And the momentum flux ratio J is equal to 12.63. All the MASCOTTE data for this case are available in reference [4].

3 Numerical simulation of MASCOTTE

The simulation of the MASCOTTE test case is delicate because various complex physical processes take place in this reactive, turbulent, two-phase flow like spraying, vaporization, mixing and combustion. So, to compute this test case, a progressive approach was adopted. First, we simulated an equivalent version of MASCOTTE in 2D-axisymetrical gaseous configuration. Then, we investigated the influence of the various models of CEDRE, our home-made code on the flame shape in a 2D-axisymetrical two-phase flow configuration. From this study [3], we developed and validated a turbulent combustion model based on mixing reactions and kinetic reactions named TPaSR for Transported Partially Stirred Reactor [5]. Concerning the K-L turbulent model, we optimised its parameters set for the cryogenic combustion in subcritical regime. We also showed that a non-uniform droplet distribution along the liquid cone allowed a good coherence between numerical result and MASCOTTE data base [4]. Nevertheless, some discrepancies still remain and they could come from the 2D simplification. So, the next step is the 3D simulation using the optimised parameter set of the 2D simulation to complete the study.

CEDRE code

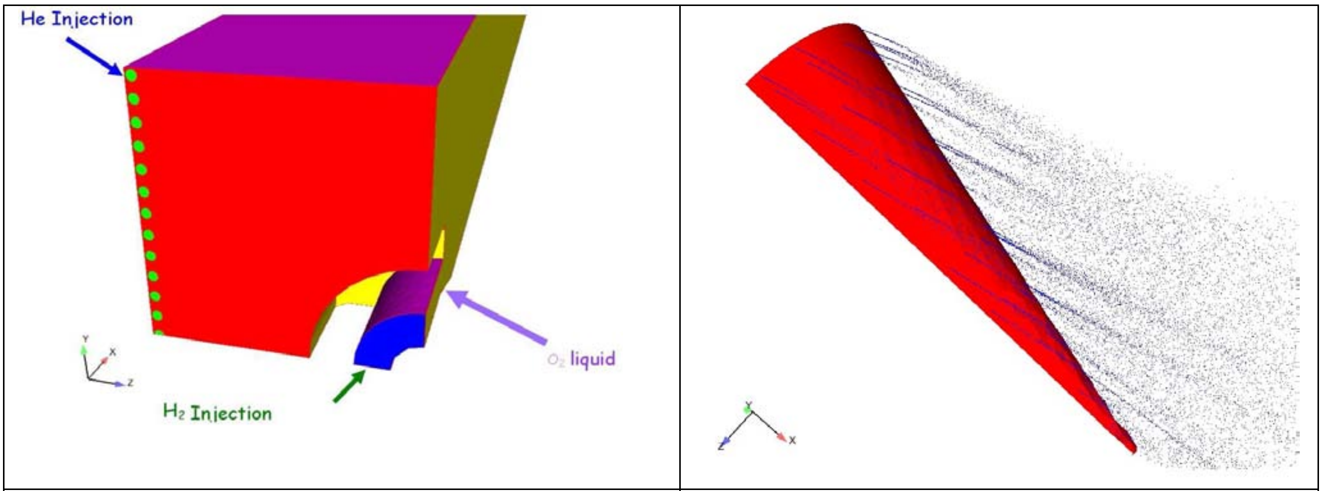
CEDRE [5] is a multiphysical computational tool for numerical simulation in the field of energetics, with a particular emphasis on propulsion applications. The code can handle several coupled physical subsystems, each of them being taken into account by a specialized time-dependent solver. The compressible flow module solves the Navier-Stokes equations with any number of species and various possible models for chemical reactions. The simulated configuration can be 3D or 2D (plane or axisymmetric) and the mesh is unstructured. For turbulence, RANS models are available as well as Large-Eddy Simulation. In this study, RANS simulations were performed. Liquid particles can be modelled in the lagrangian or euleriean approach. We chose the lagrangian solver. In this solver, interactions with the gas include drag forces, heat exchanges and vaporization. We assessed the gaseous solver as well as the lagrangian solver. The simulation was performed in parallel with 32 processors.

Physical models

We used the TPaSR model as combustion model. This model combines mixing equations to take into account the turbulence and kinetic equation to get the right species composition and temperature whatever the mixture ratio is. We use a K-L turbulent model and a non-uniform distribution of the droplet diameter along the liquid cone.

Mesh

Because of the symmetry of the problem, only one quarter of the chamber is simulated. For this first 3D simulation, a relatively coarse mesh was used. It was composed of 100 000 nodes, 600 000 elements and 26 000 boundary faces. Like in the experimental apparatus, the helium cooling film is injected through small holes colored in green in Figure 2 - A.



A – Helium and Hydrogen Injection	B – Lox injectors

Figure 2 : 3D Mesh

The liquid oxygen injection is represented by a solid cone along which oxygen droplets are injected. This cone is meshed by 42 droplet injectors (Figure 2 - B).

Concerning the boundary conditions, inlet conditions for helium and hydrogen set the mass flow rate and the injection temperature. The walls are adiabatic ones and the exit is a supersonic boundary condition.

4 Simulation results

3D effect

Unlike the 2D-axisymetrical simulation, the cooling helium film is injected on only two sides of the chamber where the visualisation windows are set. These sides are parallel to the XY plan (Observation plan). This cooling film generates a dissymmetry in the flow field and the flame does not have an axial symmetry as it is shown in Figure 3 :

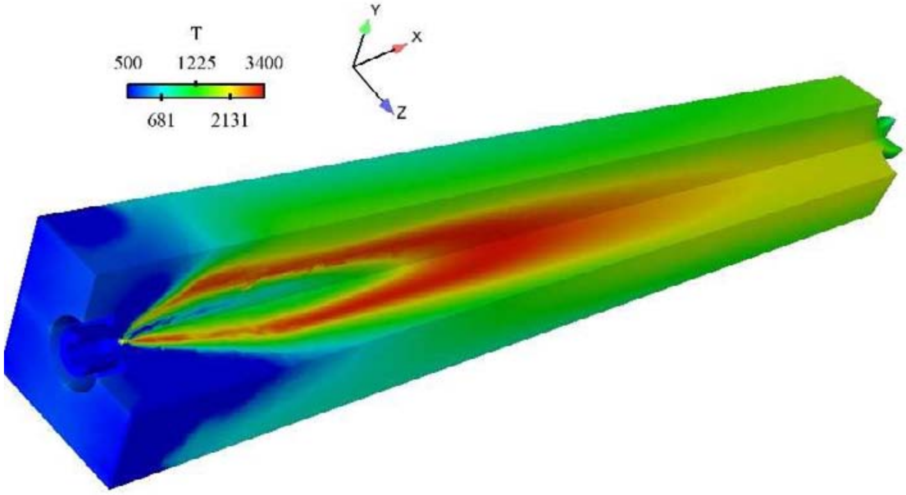


Figure 3 : Dissymmetry of the flame

The Figure 4 compares the temperature field in the observation and transverse (Plan XZ) plans .

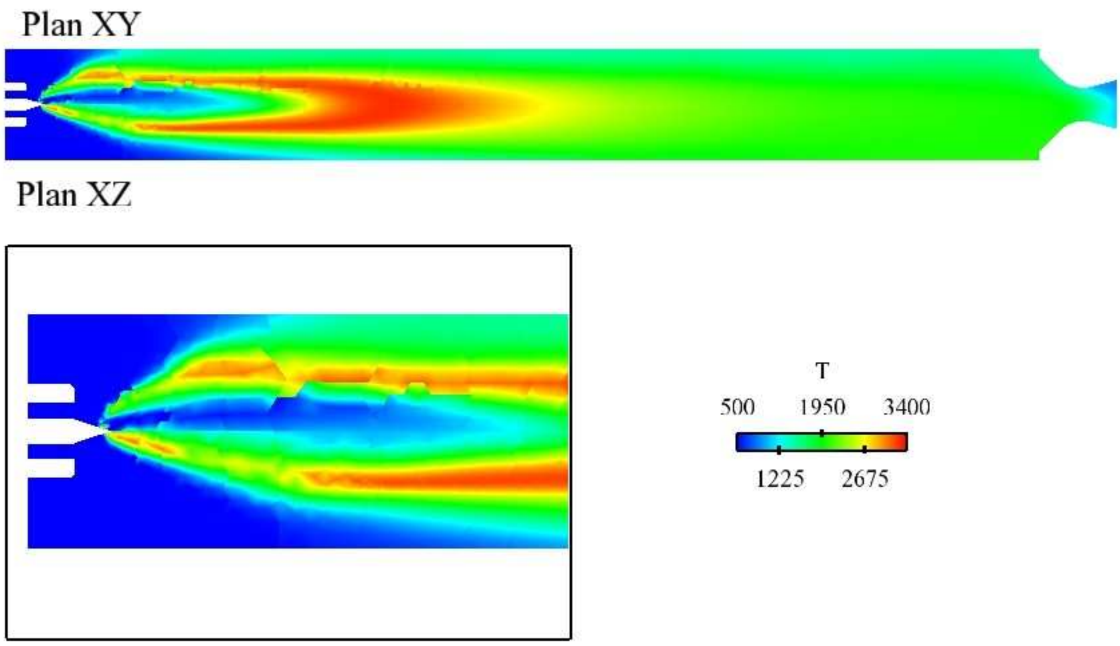


Figure 4 : Comparison of the temperature field for two cut plans

The flame opens with a wider angle and more quickly in the observation plan than in the transverse one. The temperature field of the 2D-axisymetrical simulation represents an average of these two plans. The helium film cools only two sides of the chamber, sides parallel to XY plan. So, the flow is colder in this direction, and it slows down the combustion. Thus, the flame widens less quickly in the transverse plan than in the observation one. In addition to cooling effect, the helium injection leads to a dissymmetry of the flow and of the eddies around the injector like it is shown in Figure 5 :

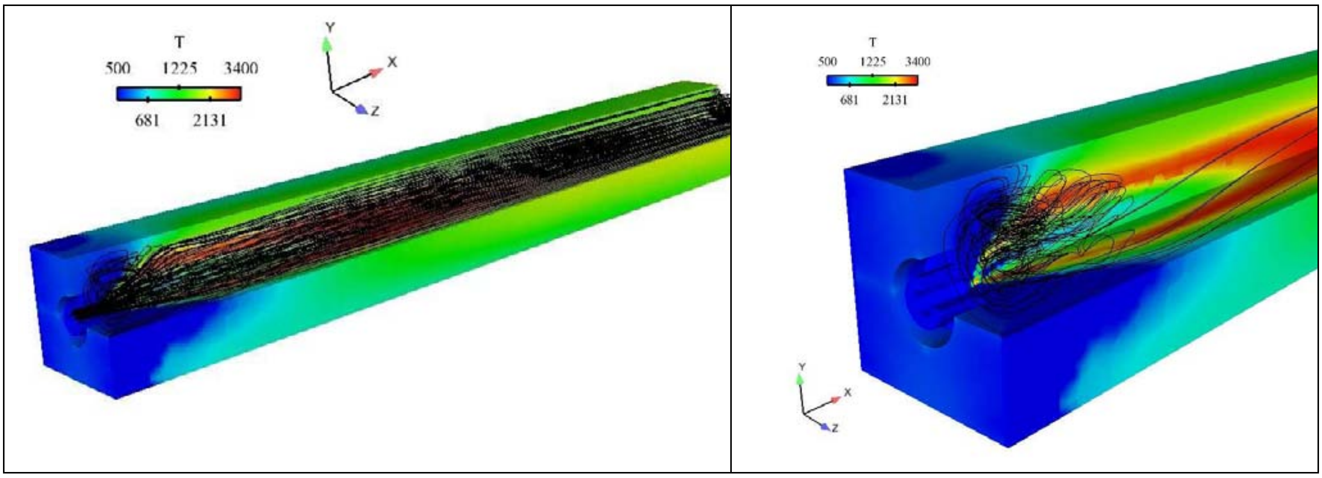


Figure 5 : Stream-line in the chamber

According to Figure 5, we can consider that the flow is approximately 2D-axisymetric downstream the injector. On the contrary, around the injector, the flow nearby the injector is strongly 3D and the flame shape is significantly altered compared to the 2D simulation. In the future, if we need to compute a similar configuration and if we are not interested in the injection zone, a 2D simulation turns out to be sufficient.

Helium cooling film

The helium film is injected nearby the visualisation windows in order to cool down them. The Figure 6 represents the helium mass fraction in the chamber :

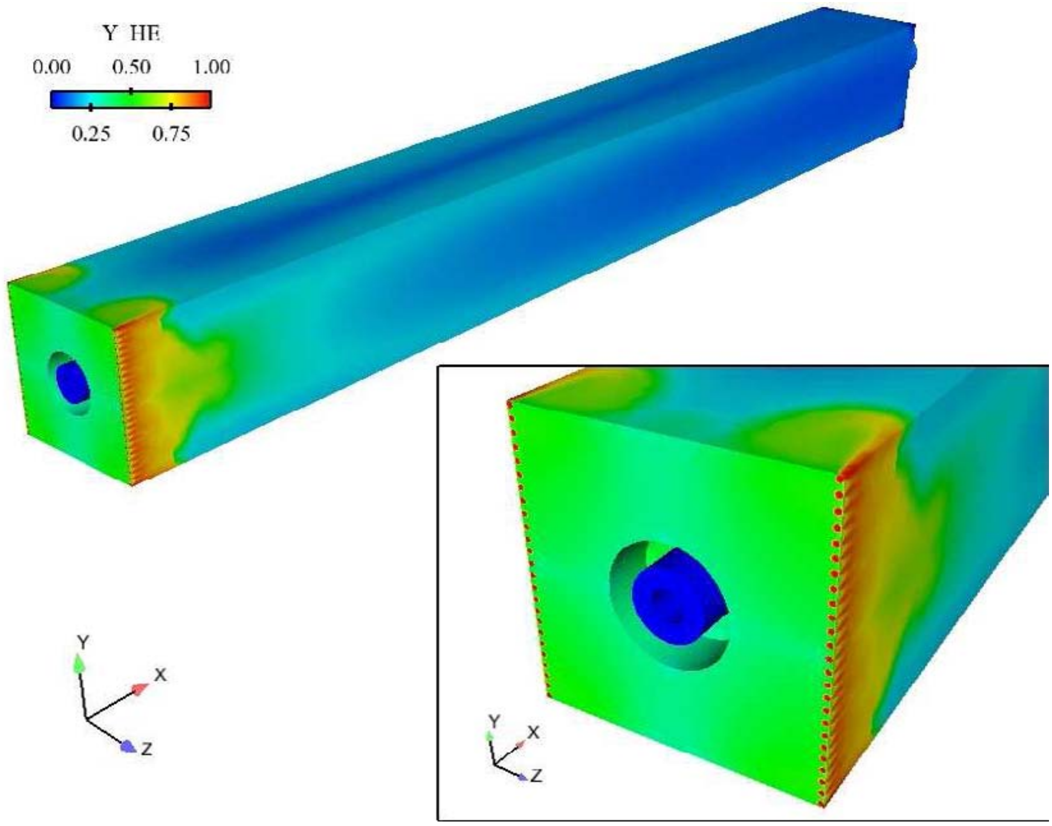


Figure 6 : Helium mass fraction in the chamber

The windows are 100 mm long, 50 mm wide and 10 mm thick. They are made of melt silica. For this kind of material, there is no a fusion temperature, like for metal for instance, but a softening temperature. In the present case, this temperature is approximately equal to 1850 K but the suppliers advocate a maximum use temperature of 1500 K.

Figure 7 shows the temperature field using a scale which maximum is 1500 K and the window position as a black rectangle.

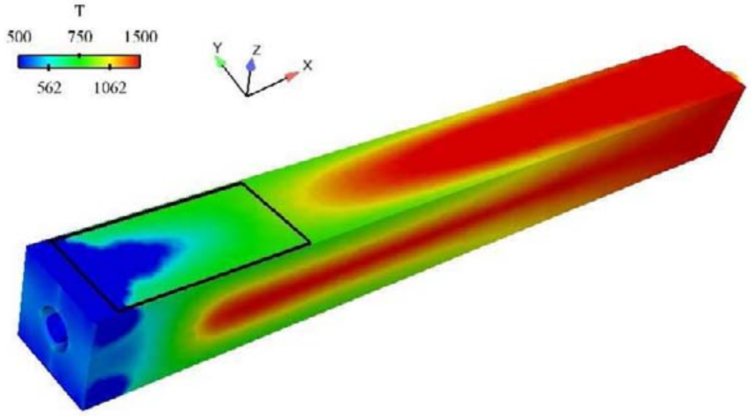


Figure 7 : Temperature field and window

The simulation confirms that the window is well protected by the helium film. Yet, we can notice that this film has a limited area of efficiency. Or there are three positions of the visualisation window module (Figure 8). So, the simulation corroborates the experimenters' choice [4] to have linked the helium injection to the window module. In the future, the numerical simulation could help to proportion the cooling system of the visualisation windows. We can also notice the sides which are perpendicular to the helium film are hotter than those where the helium film is injected.

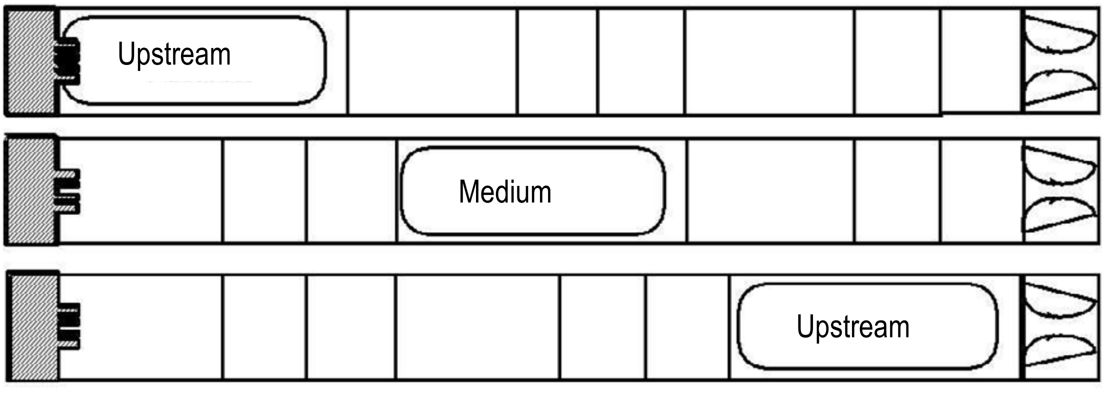


Figure 8 – The three positions of the visualisation module.

1 Comparison between 2D simulation and 3D simulation

Figure 9 and Figure 10 compare the temperature field for the 2D-axisymmetrical simulation and the 3D simulation for the observation plan (XY-Plan) and for the transverse plan (XZ-Plan). In both plans, the flame is shorter in the case of 3D simulation. But, the flame aperture angle is wider in the observation plan of the 3D simulation than in the 2D simulation. In the transverse plan, the aperture angle is almost the same for the 2D and the 3D simulations.

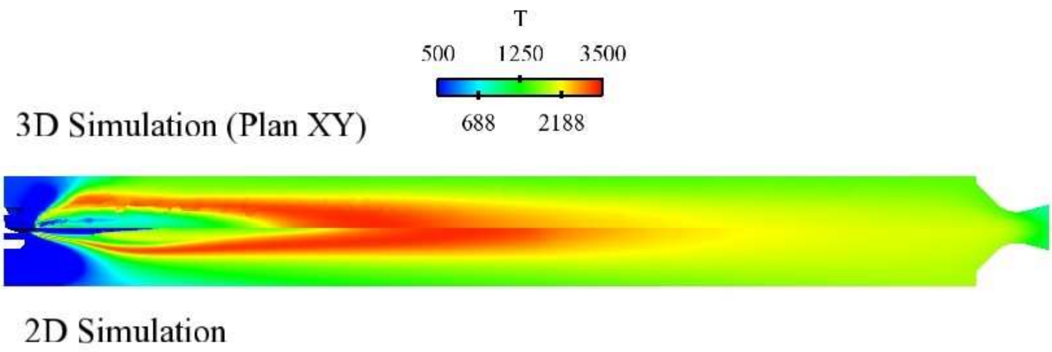


Figure 9 : Comparison of the temperature field for 2D (lower part) and 3D simulations (upper part) (Observation plan)

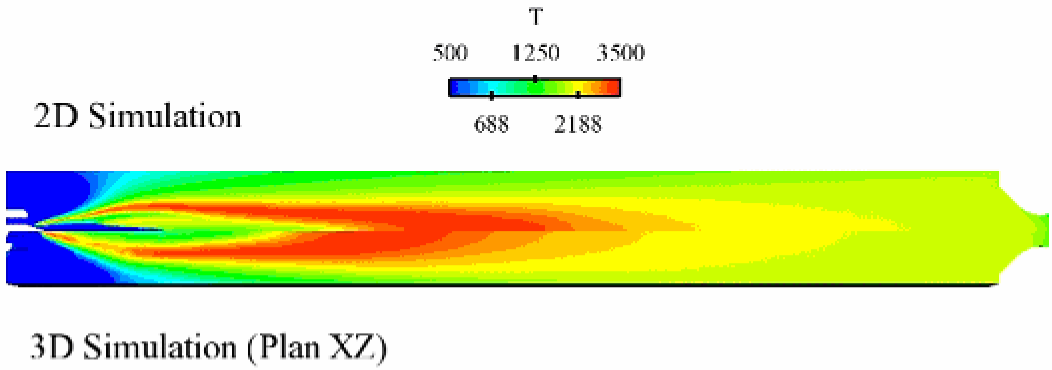


Figure 10 : Comparison of the temperature field for 2D (upper part) and 3D (lower part) simulations (Transverse plan)

2 Comparison 3D / 2D-axi / Experiments

First, the simulations (2D and 3D) are compared to the CARS measurement [4]. These comparisons are presented in the following figures for various abscissa:

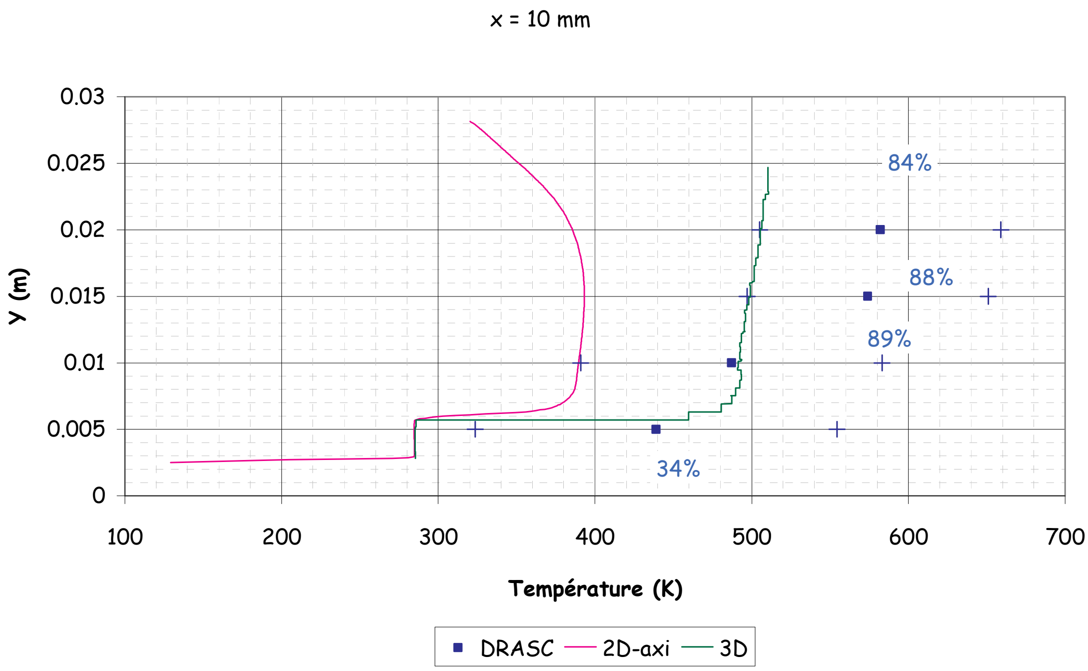


Figure 11 : Comparison CARS / simulation 2D and 3D (X = 10 mm)

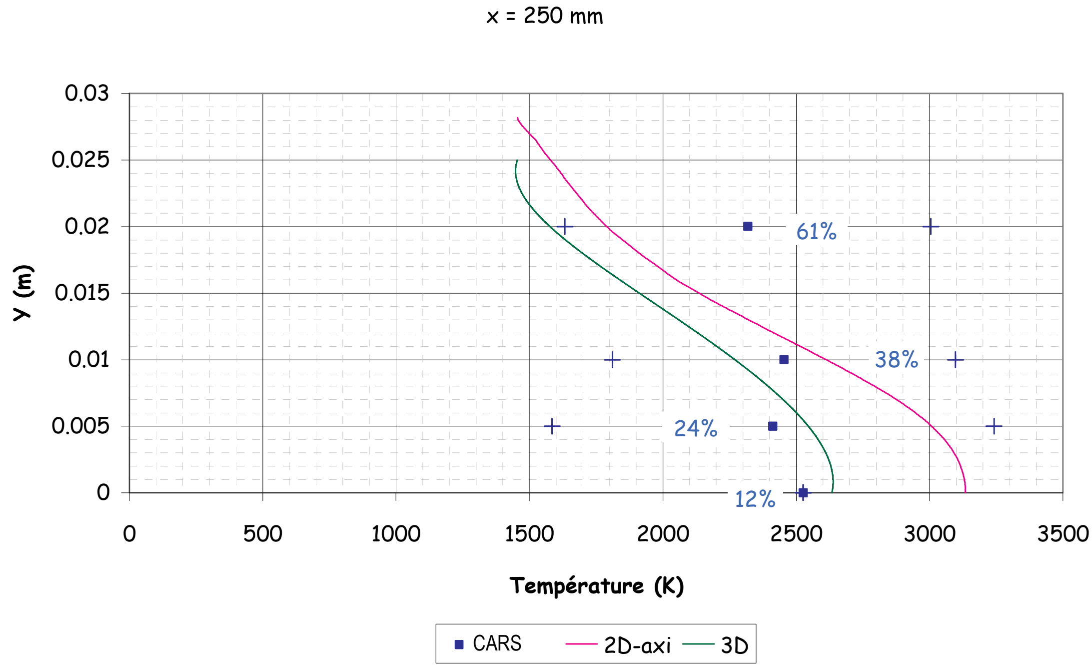


Figure 12 : Comparison CARS / simulation 2D and 3D (X = 250 mm)

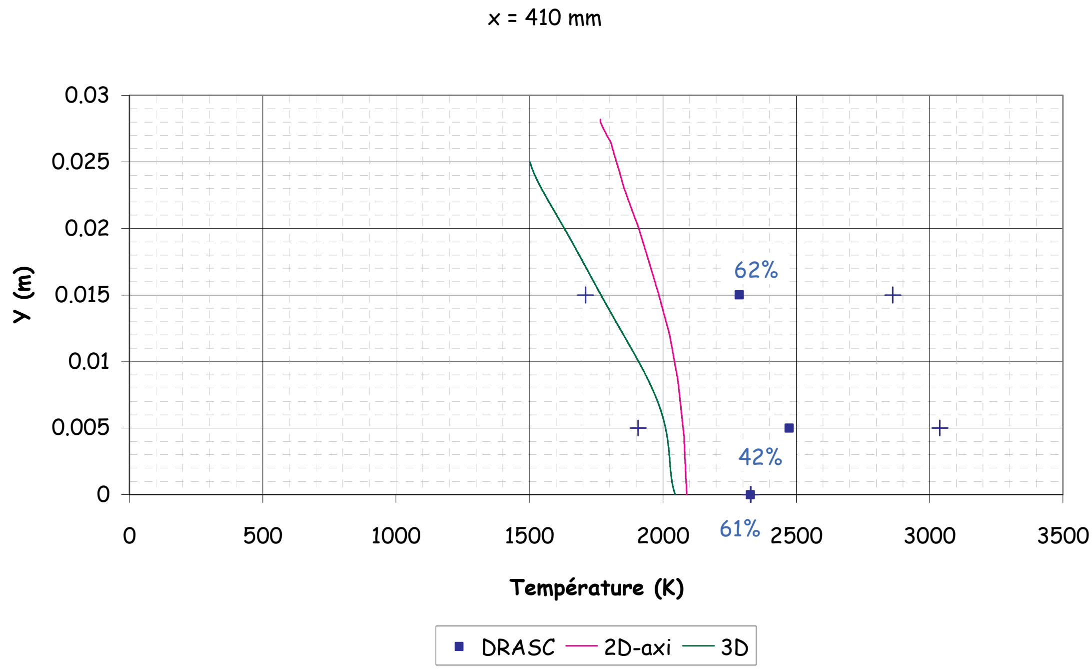


Figure 13 : Comparison CARS / simulation 2D and 3D (X = 410 mm)

CARS measurements give the mean value (blue square), the standard deviation and the validation rate of this measurements.

The Figure 11 to Figure 13 give the transverse evolution of temperature in the median plan. Nearby the injector, the 3D computation improves the comparison with experiment but for the other profiles, the difference between 2D and 3D simulations is not significant.

The discrepancies between the 3D simulation and the experiment are more considerable in Figure 14 where OH mass fraction is compared with the Abel transform of OH emission. In the observation plan, the flame bump is not located at the same position. It appears earlier in the 3D simulation than in the experiment. The 3D simulation does not improve the flame shape.

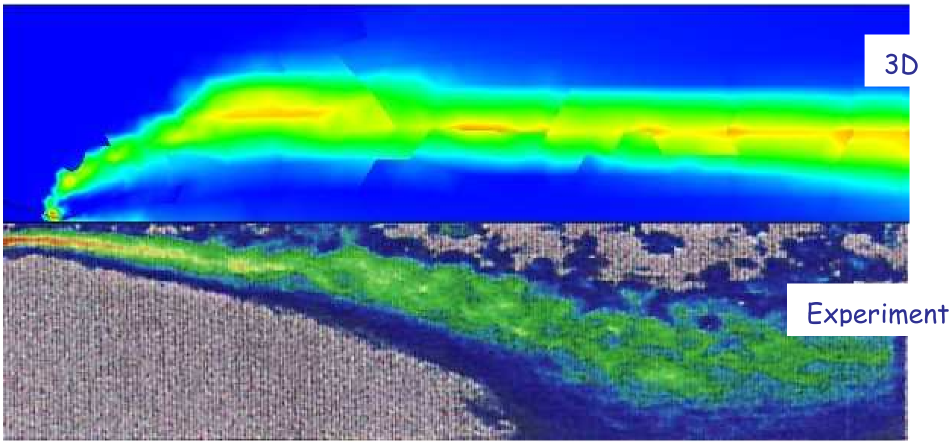


Figure 14 : OH mass fraction (Observation plan)

Nevertheless, this qualitative comparison between simulation and experiment should be qualified. Indeed, the experimental OH image was recorded by a camera which detects the emission intensity of OH radical and integrated it along the ray in the direction of the camera. And then, an Abel transformation of this image allows us to get information on the emission intensity for concentric ring by supposing that the mean field is 2D-axisymmetric. Yet, the 3D simulation shows that the flow field is not 2D-axisymmetric. Moreover, the 3D simulation gives a cut of the OH fraction field and does not integrate the OH emission. So before concluding on this qualitative comparison, we should concentrate our effort on adapting the simulation and the experimental results to compare exactly the same kind of data.

5 Conclusion

This study shows the 3D simulation of MASCOTTE test-bench. It follows the 2D-axisymmetrical study in which the influence of the CEDRE model was investigated in detail. The 3D simulation proves that the flame is not 2D-axisymmetrical. Indeed, the injector and the helium film injection on only two sides of the chamber lead to this dissymmetry. The helium film alters the flame shape by shortening it .Its efficient to cool down the visualisation window was also emphasised. The numerical code could be used to proportion correctly the helium mass flow rate and the window size in future experimental apparatus. There is a good agreement between experiment and 3D simulation concerning the temperature and the composition inside the chamber but the flame shape is not satisfactory. One reason could be the inconsistency of the comparison between Abel transformed image of experimental OH emission and the OH mass fraction field. So, an effort will be done to compare the same kind of data between experiment and simulation. Yet, the 2D study showed that the droplet distribution of the liquid cone has a great influence of the flame shape. So, a detail investigation will also be carried out to clarify this influence.

6 Acknowledgement

The partial support of CNES is gratefully acknowledged for this study.

7 References

1. MASCOTTE : A test bench for cryogenic combustion research - M. Habiballah, L. Vingert, J.C. Traineau, P. Vuillermoz - 47th International Astronautical Congress – Beijing (China), October 7-11, 1996
2. Proceeding of the 2nd International Workshop Rocket Combustion Modeling (atomization, Combustion and Heat Transfer) – 25/27 march 2001
3. Simulation of a subcritical LOX/GH2 MASCOTTE test case (10 bar) – A. Nicole, F. Dupoirieux, L. Vingert, M. Habiballah, M. Theron - 5th International Spacecraft Propulsion Conference - 5-8 May 2008, Heraklion (Greece)
4. Sélection de cas tests MASCOTTE – cas subcritique LOX/GH2 (10 bar) et cas supercritique LOX/GH2 (60 bar) - L. Vingert - ONERA RT n°1/11785- Novembre 2006
5. Large Eddy Simulation of a turbulent premixed flame stabilized by a backward facing step - MURRONE A., SCHERRER D. - 1st INCA Workshop – 2005 - Villaroche (France)
6. CEDRE: development and validation of a multiphysic computational software - P. Chevalier, B. Courbet, D. Dutoya, P. Klotz, E. Ruiz, J. Troyes, P. Villedieu - 1st European Conference for Aerospace Sciences (EUCASS), Moscow, Russia, 2005