Characterization of the Reacting Two-phase Flow inside a Research Ramjet Combustor

Ristori A., Heid G., Brossard C., Bresson A.

Office National d'Etudes et de Recherches Aérospatiales (ONERA),
29 Avenue de la Division Leclerc, F-92322 Châtillon Cedex, France .
email: Arnaud.Ristori@onera.fr

A research ramjet program has been initiated at ONERA with the support of DGA/SPNuc with the aim of improving methodology for ramjet combustion chamber design and tuning by using validated CFD codes. The approach used was to design and manufacture a specific ramjet model in order to setup an experimental database for non-reacting and reacting flows; the final aim is to validate models and CFD codes in order to have an accurate numerical tool able to predict steady performances of ramjet combustors (combustion efficiency, pressure recovery, stability limits,...). Two three-dimensional ramjet research combustors have been defined and are presented in this paper. These motors have been specifically designed to experimentally simulate flows inside real solid-propellant Ducted Rockets (SDR) or Liquid Fueled Ramjet (LFRJ); the first one is dedicated to cold flow experiments at atmospheric pressure (transparent model in Plexiglass®) and the second one to hot flow experiments under more realistic conditions (combustion model equipped with glass windows). Visualization and LDV results obtained for these models in non-reacting flow cases are presented in comparison with numerical results. Visualizations, PLIF, LDV and PIV results obtained for realistic and reacting two-phase flow cases are also presented here.

1. Introduction

Until recently, the methodology used to develop ramjet engine and study ramjet combustor performances was mainly based on an experimental approach consisting of combustion tests in connected pipe mode.

Thanks to the availability of more powerful computers and new 3D CFD codes, it is now possible to compute turbulent and reacting flow fields inside a ramjet combustor. Improvements of the development methodology can be considered and reduction of combustion testing now becomes possible for designing and tuning a ramcombustor. Nevertheless, an efficient use of this tool requires the availability of models and codes in the case of ramjet application.

This is the reason why, in 1995, with the support of French Official Services (DGA/SPNuc), ONERA initiated a research ramjet program with the aim of improving the methodology for ramjet combustion chamber design and tuning by using validated CFD codes.

This paper describes the ramjet research model designed and tested in order to set-up an experimental database for non-reacting and reacting flow cases. Experimental results such as hydraulic visualization, Laser Doppler Velocimetry (LDV) measurements, Particle Image Velocimetry (PIV) and Planar Laser-Induced Fluorescence (PLIF) visualizations obtained on the ramjet research combustor models are presented and discussed. The first calculated results for non-reacting flow cases are compared with LDV data.

2. Description of the Research Ramjet Project

The research ramjet program that has been initiated at ONERA [1] with the aim of validating models and CFD codes for ramjet applications is based on the design, manufacturing and testing of a specific ramjet model in order to set-up an experimental database for non-reacting and reacting flows and to perform, at the same time, numerical simulations using different models (turbulence, combustion,…) and CFD codes.

Comparisons between experimental and numerical data should allow the development, ultimately, of a validated numerical tool able to predict steady state performances of ramjet combustors (combustion efficiency, pressure recovery, stability limits,...).

Ducted Rockets (SDR) and Liquid Fueled Ramjets (LFRJ) are in the scope of this study. In the case of SDR, one-phase flow characterization is concerned whereas, in the case of LFRJ, the investigation of the flow structure is more complicated. A two-phase flow characterization and the validation of two-phase flow models in CFD codes are required.

The program is composed of two parts, each part involving a specific combustor model:

- the first part is focused on cold flow studies (transparent model) on hydraulic and aerodynamic test rigs for internal aerodynamics characterization,
- the second part is focused on hot flow studies (combustion model) for non-reacting (hot air flow without combustion) and reacting cases.

In these two parts, numerical simulations are associated and mainly conducted using a 3D code named MSD [2], which has been developed by ONERA.

3. Ramjet Combustor Models for Cold and Hot Flow Experiments

With the objective to conduct specific combustion studies, a three-dimensional ramjet engine geometry has been defined in order to have operating conditions of the combustor (pressure, velocity, temperature) comparable to real motors. The engine configuration considered here consists of a main combustor with two lateral air inlets on opposite sides; downstream is an axisymmetric nozzle. In order to facilitate optical access (models equipped with specific glass windows for optical measurements) inside the combustor, the geometry was designed with a square section for the air inlets and the duct.

In the case of ducted rocket simulation, fuel injection occurs at the head end of the combustor and the use of a gaseous fuel (propane) is planned to simulate burnt gases coming from the gas generator. For liquid fueled ramjet simulation, fuel injection can be located either at the head end of the combustor (same location as for ducted rocket simulation) or in the air inlet near the elbow. Kerosene is used as a fuel. The ramcombustor design and the main characteristics are given in **Figure 1**. Further details about the design were published previously [1].

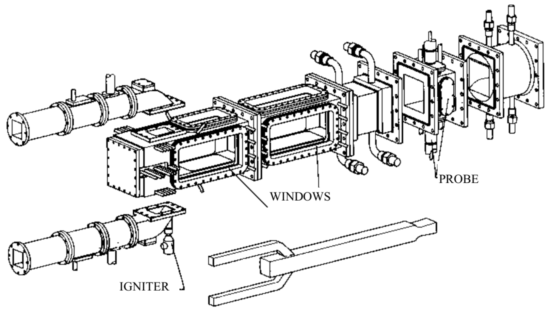


Fig. 1 3-D view of the combustion model.

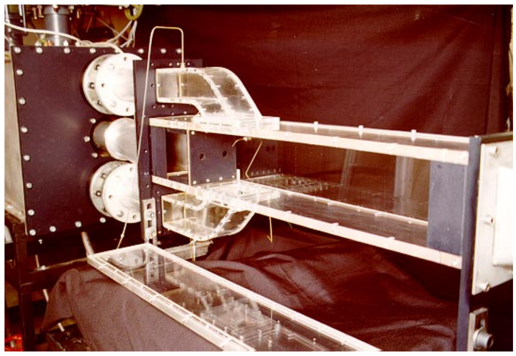


Fig. 2 Transparent SDR model (hydraulic tests).

In the case of LFRJ investigations for three different simulated flight cases, numerical values for inlet air temperature, air mass flow rate, fuel mass flow rate, theoretical combustion chamber pressure and temperature are given in **Table I** for an equivalence ratio equal to 0.5 and 1. A specific transparent model at scale 1.6 with respect to the combustion model has been built to perform cold flow experiments (**Figure 2**). This model is only available for atmospheric tests with water or air flow. For this reason, there is no nozzle at the outlet and the length of the model has been limited compared to the combustion model. In the case of the simulation of a SDR combustor, two fuel holes are located at the head end of the combustor. Their dimensions are determined to keep constant the ratio of momentum quantity between simulated air and fuel with respect to the combustion model [1].

LFRJ Simulation			$m°_{C10H20}$ (g/s)		P_{i4} th (bar)		T_{i4} th (K)	
FLIGHT CONDITION	T_{i2} (K)	$m°_2$ (kg/s)	ϕ=0.5	ϕ=1	ϕ=0.5	ϕ=1	ϕ=0.5	ϕ=1
Low Altitude	520	2.9	98	196	5.7	7.1	1668	2355
Middle Altitude	600	1.9	64	128	3.8	4.7	1722	2380
High Altitude	750	0.9	30.5	61	1.8	2.2	1836	2436

Table I. Test conditions for experimental studies on SDR and LFRJ cases.

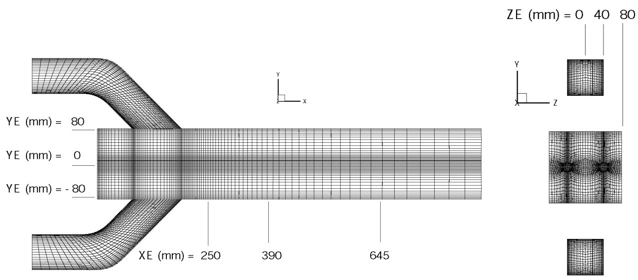
4. Non-reacting flow studies

In the non-reacting flow experiments presented below, mainly SDR and LFRJ simulations were investigated. The experimental data obtained are helpful in understanding mixing processes and flow structures inside such a combustor, and also for partial validation of CFD codes. Three test campaigns have been carried out:

- the first one on a hydraulic test rig to perform SDR flow visualizations of the mixing process between air and gas generator products inside the transparent model (the 3D flowfields were studied experimentally), colorimetric techniques and image processing were combined to allow flow visualization of the mixing process;
- the second one on an aerodynamic test rig to perform LFRJ flow visualizations and PDA measurements in the fuel injection spray either at the head end of the combustor or inside the air inlet in the elbow.
- the third one on aerodynamic test rigs where 2D LDV measurements have been performed inside test combustors (transparent and combustion models).

4.1. Mixing visualizations in water tunnel test facility

For these tests, water flows simulated air and fuel flows. **Figure 2** shows a picture of the transparent SDR model installed on the hydraulic test rig . A tracer dye (fluorescein) was mixed with water in order to simulate the fuel. Different fuel-to-air mixture ratios were simulated by varying the water flow rate. To visualize fuel and air mixing processes, laser light sheet oriented perpendicular and parallel to the flow were moved in order to investigate different sections of the combustor. The intensity delivered by each pixel of the CCD camera is directly proportional to the local fuel concentration. Visualizations of the mixing process, for the lengthwise section ZE = 40 mm (CFD-mesh, **Figure 3**), show (**Figure 4**) that the increase of the equivalence ratio is associated with an increase of the fuel jet penetration into the ram-air stream. However, if the fuel jet momentum becomes too high (simulated E.R = 1.7 and 2.3), a large fraction of the fuel passes through the ram-air stream without mixing with the air; the consequence would be a decrease of the combustion efficiency or the blow-out of the flame. Further results were published on fuel and air mixing [4] and comparisons between local E.R. measured and calculated in water have been done recently [5].



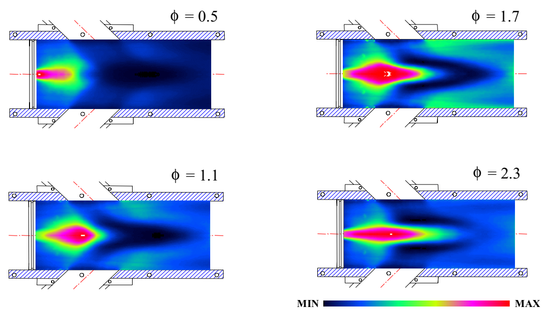


Fig. 3 Mesh of transparent SDR model (scale 1.6).

Fig. 4 E.R.(ϕ) influence on jet-penetration.

4.2. Fuel spray injectors characterization

For the reacting case, the combustor was operated as a liquid fueled ramjet (LFRJ) using two hollow-cone kerosene injectors located either at the head end of the combustor or inside the air inlet in the elbow region (**Figure 5**).

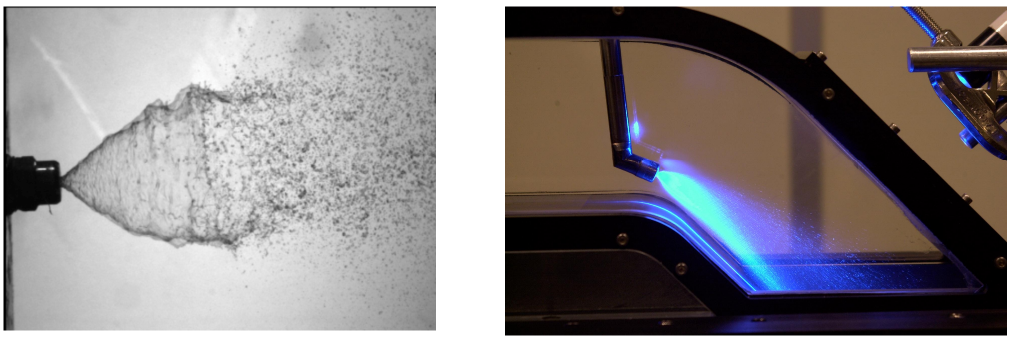


Fig. 5 Visualization of the hollow cone spray

Measurements of axial and radial velocities as well as Sauter Mean Diameter (SMD or D32) were performed, 40 mm downstream from the location of the commercial DELAVAN® BJ3® injector used for combustion experiments (**Figure 6**). PDA measurements results were obtained at atmospheric pressure, for two different fuel injection pressures, without or with two different surrounding air flow rates.

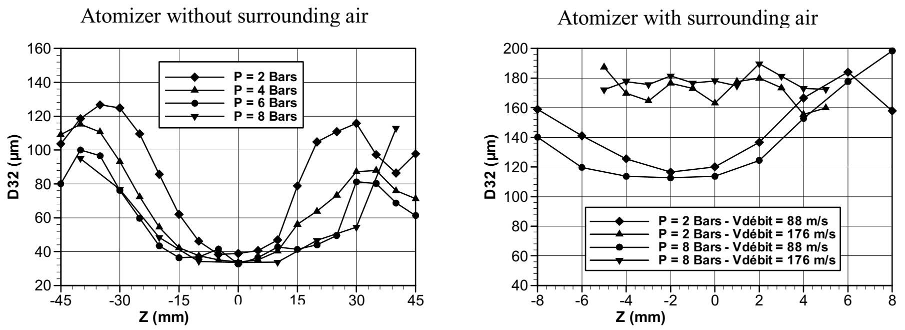


Fig. 6 Atomizer hollow cone spray characterization without/with surrounding air flow

4.3. LDV measurements and comparison with CFD computations in non-reacting flow cases

In order to measure detailed gas-phase velocity, a four-beam, two-color (blue and green) Laser Doppler Velocimeter was used. A fiberoptic probe was used to transmit laser beams and create a probe volume; the forward scattering light intensity was collected by a multimode fiberoptic probe and amplified by two photomultipliers. The LDV measurements were conducted on many different lengthwise and cross-sections inside the transparent and combustion models.

A recirculation zone at the dome region is caused by the jet impingement of the two ram air streams and the existence of free volume at the head end of the combustor. In the cold flow case (transparent model), mean velocities are around 60 m/s at the ram-air outlets and 30 m/s (Mach=0.08) inside the duct. In the hot flow non reacting case (combustion model in middle altitude flight simulated test condition), the flow is preheated to 600 K (test setup in non-vitiated air) and mean ram-air velocity is 340 m/s and axial duct velocity is 170 m/s (Mach=0.35). However, compressibility effects are not taken into account in the cold flow case since the Mach number is equal only to 0.08 inside the duct [1][4] versus 0.35 for the hot flow real case [6]. A general conclusion is that measured and calculated velocity are in agreement (**Figure** 7). Results obtained partially validate the theoretical and numerical models developed in the MSD code.

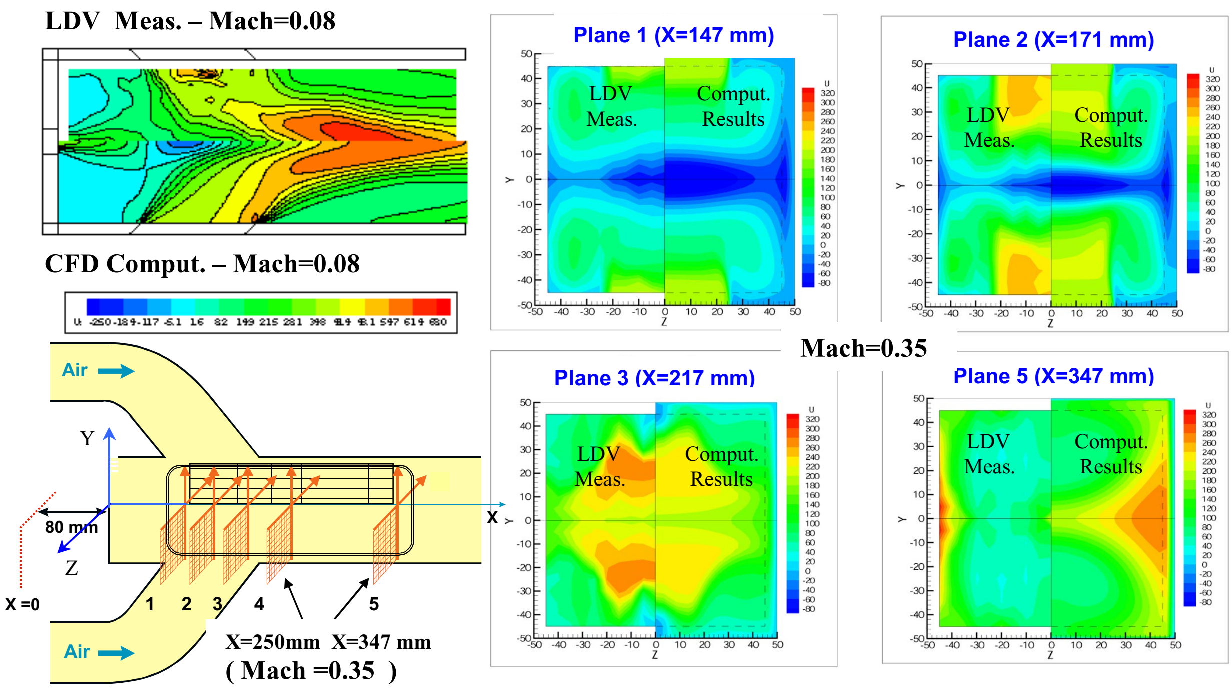


Fig. 7 Axial velocity U measured (LDV) and compared with computational results

5. Reacting flow studies

The second part of the program is focused on combustion studies. This is an important phase because global and local measurements are performed exactly at the operating conditions of a real motor. For this purpose the combustion model is used: visualizations, LDV, PIV, PLIF and gas analysis measurements have been done for this model in LFRJ combustor simulations under realistic thermodynamical test conditions (**Table I**).
Experimental studies in connecting pipe test setup with non-vitiated air have been carried out in reacting flows at test conditions corresponding to the high altitude case for an equivalence ratio equal to 0.5. Pressures and temperatures are measured on the combustion model to characterize the global operation of the combustor. The combustor was operated as a liquid fueled ramjet (LFRJ) with two kerosene injectors located at the head end of the combustor. Visualizations, LDV and PIV measurements as well as the OH* emission and OH-PLIF methods are used to describe the flow pattern.

5.1. High-speed Visualizations

Figure 8 shows an instantaneous image of the turbulent flame inside the duct section, obtained from a high speed movie (1,000 frames/sec) recorded using a high speed digital video camera. Movies at a higher speed (4,000 frames/sec) have also been recorded and are currently being analyzed to deduce some information about the unsteady characteristics of the flame, such as instability frequencies.

Fig. 8 Instantaneous Image of the Flame.

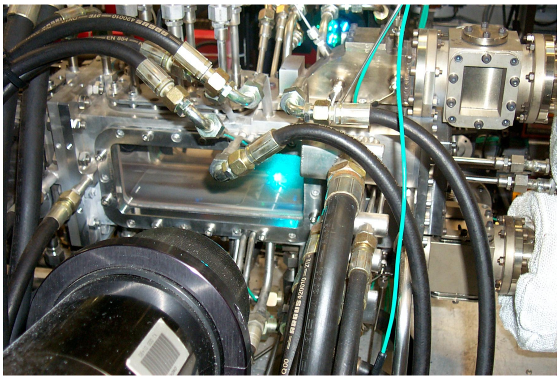


Fig. 9 Combustor equipped with quartz windows and the LDV system.

5.2. Laser Doppler Velocimetry and Particle Image Velocimetry measurements

The LDV system used for those experiments has been presented elsewhere [6]. A photograph of the combustor, showing in particular the two ram air inlets, the duct section equipped with quartz windows, and the LDV receiving head, is provided in **Figure 9**.
Compared to the non-reacting flow case, the analysis of the flow [6] shows that no significant changes in the flow structure between X=250 mm and X=350 mm were observed in the reacting flow case (**Figure 10**). Therefore, the impingement of the two ram air streams did not result in a deviation of the flow towards the side-walls as in the non-reacting flow case (plane 5 – **Figure 7**). This difference can be attributed to a much lower ram air velocity in the reacting flow case than in the non-reacting flow case due to the pressure increase caused by the combustion; the mean velocity in the ram air inlet ducts was 345 m/s in the non-reacting flow case versus only around 180 m/s in the reacting flow case. The axial profile in **Figure**

10 also indicates that the downstream limit of the recirculation zone in the reacting flow case was located more upstream (X=176 mm) than in the non-reacting case (X=184 mm). The existence of a strong recirculation zone in the dome region with two large counter-rotating vortices was evidenced by the LDV data (**Figures 7 and 10**). The downstream limit of this recirculation zone in the reacting flow case was located 8 mm upstream of the limit obtained for the non-reacting case.

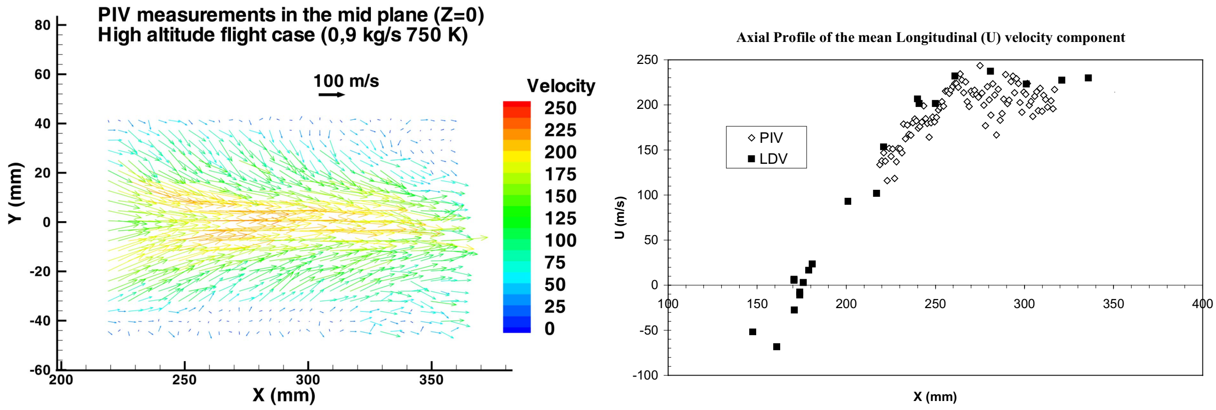


Fig. 10 PIV and LDV measurements in the high altitude simulated reacting flow case

5.3. OH emission and OH-PLIF measurements*

The OH* emission and OH-PLIF imaging techniques [7] were used to visualize the structure of the reacting mixing layer. The main elements of the PLIF system are shown in **Figure 11**. A sample result is presented in **Figure 12** which shows the OH^* radical emission from the reacting flow region at a high altitude flight condition and an equivalence ratio ϕ=0.5. The analysis of PLIF measurements show more precisely the 3D flame structure. Results on this matter will be published in a later paper.

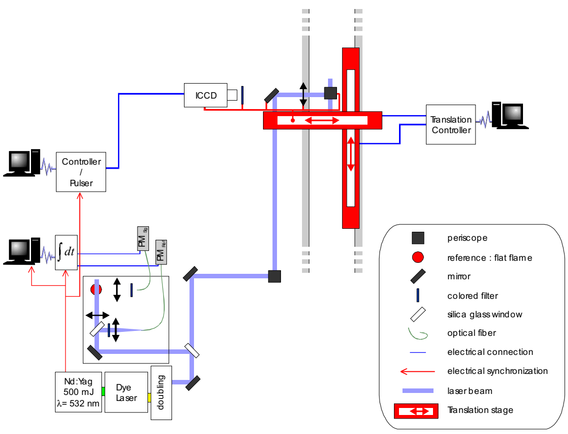


Fig. 11 PLIF experimental setup.

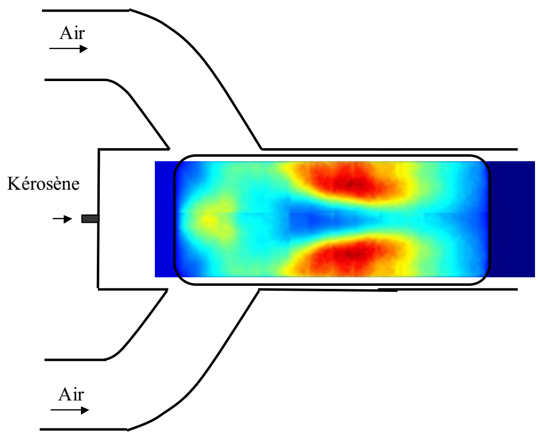


Fig. 12 OH^* radical emission from the reacting flow.

6. Conclusions

In order to improve the methodology used to develop ramjet combustors, ONERA initiated in 1995 a research program on ramjet motors with the aim of establishing detailed experimental databases for real operating conditions and validate physical models and CFD codes for SDR or LFRJ applications. A specific 3D ramjet geometry design with a square section well-adapted to optical measurements has been defined. Values of dimensions and area ratios are comparable to real motors ones. The program has an important experimental phase associated with numerical simulations mainly conducted using the MSD code developed by ONERA.

Up until now, experiments were done either for cold flow, or hot flow with or without combustion. Based on the 3D ramjet geometry design previously defined, one model for each kind of experiment has been built. The model for cold flow studies is entirely transparent and used at atmospheric pressure on hydraulic or aerodynamic test rigs. The combustion model is equipped with quartz windows to allow local optical measurements.

In the case of cold flow studies, SDR simulation was studied; mixing characteristics between fuel and air have also been examined on a hydraulic test rig whereas aerodynamic flowfields have been measured on a aerodynamic test rig. Cold flow studies were performed in the frame of this work to ascertain whether this semi-empirical approach, previously used in the development methodology, is justified although the compressibility effects are not taken into account in this kind of simulation.

The main part of the experimental program is based on hot flow experiments. Up until now, LDV and PIV measurements were carried out for the non-reacting case (no fuel injection) and the reacting case (LFRJ case with fuel injection at the head end of the combustor). Velocity profiles and turbulence characteristics were obtained in the horizontal and vertical mid-sections of the combustor duct section. PLIF measurements were also performed for LFRJ cases at middle and high altitude simulated flight cases.

The experimental results obtained in this study already represent a significant data base, which will be used for comparisons with flow field computations. This data base will be completed in the near future with additional measurements obtained for high speed flows corresponding to high and middle altitude flight conditions by using other techniques.

Acknowledgments

The authors would like to thank the French Délégation Générale de l'Armement, Service des Programmes Nucléaires (DGA/SPNuc), for its financial support in this research program.

References

[1] Ristori A, Heid G, Cochet A, Lavergne G, "Experimental and numerical study of turbulent flow inside a research SDR combustor ", 35th Joint Propulsion Conference & Exhibit, June 20-24, Los Angeles, USA, AIAA Paper-99-2814, 1999.

[2] Ristori A, Dufour E, "Numerical simulation of ducted rocket motor ", 37th Joint Propulsion Conference & Exhibit, July 8-11, Salt Lake City, Utah, AIAA Paper-2001-3193, 2001

[3] AGARD Advisory Report 323 - Working Group 22 on "Experimental and Analytic Methods for the Determination of Connected-Pipe Ramjet and Ducted Rocket Internal Performance", AR-323 , 1994.

[4] Ristori A, Heid G, Cochet A, Lavergne G, "Experimental and numerical study of turbulent flow inside a dual inlet research ducted rocket combustor ", XIVth Symposium ISOABE, September 5-10, 1999, Florence, Italy.

[5] Heid G and Ristori A, "An optical method for local equivalence ratios measurements applied to hydraulic simulation of a ramjet combustion chamber", Proceedings of PSFVIP-4, June 3-5, 2003, Chamonix, France.

[6] Gicquel P, Brossard C, Barat M, Ristori A, 2002, "Experimental study of a high speed flow inside a dual research ducted rocket combustor using laser doppler velocimetry", Proceedings of 2002 joint US ASME-European Fluid Engineering Summer Conference : Forum on High Speed Jet Flows, July 14-18, Montreal.

[7] Bresson A , « Techniques d'imagerie quantitatives : fluorescence induite par laser appliquée aux écoulements et aux combustions », Ph. DThesis , Rouen University, (2000).