ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 2, August 2013

Numerical simulations of Jet-A combustion in a gas turbine combustion chamber

A.C. Petcu[1], C.Sandu[1], C. Berbente[2]
[1]Romanian Research & Development Institute for Gas Turbines COMOTI
[2]Aerospace Engineering Department, "Politehnica" University of Bucharest

Abstract*— In order to develop a new multiple-fuel combustion chamber, a series of numerical simulations has been conducted on an existing combustion chamber, in order to determine the modifications required to allow its proper operation for multiple liquid fuels and fuel mixtures. The combustion chamber used in the simulations comes from a Garrett gas turbine engine, model GTP 30-67. A three-dimensional unsteady RANS numerical integration of the Navier-Stokes equations has been carried out, using an Eddy Dissipation combustion Model (EDM) and the k-ε turbulence model, implemented in a numerical simulation conducted using the commercial software ANSYS CFX. To verify the numerical simulations accuracy, the outlet temperature has been recorded along 600 time-averaged iterations and compared with the designed turbine inlet temperature. Numerical simulations results have a good accuracy in this particular case, so it can be concluded that the used numerical models are valid and appropriate for use to simulate the combustion process under different conditions. Thus the results obtained in these simulations will help adapt the existing combustion chamber to different fuels and fuel mixtures.

Index Terms—**3D RANS, ANSYS CFX, combustion chamber assembly, Jet-A**

I. INTRODUCTION

In order to develop a new combustion chamber, a series of numerical simulations have been conducted on an existing combustion chamber, in order to determine the modifications required to allow its proper operation for multiple liquid fuels and fuel mixtures.

The combustion chamber used in the simulations comes from a Garrett gas turbine engine, model GTP 30-67. Garrett GTP 30-67 [1, 2], a shaft power gas turbine engine, is a compact, lightweight, shaft power source which is readily adapted to fit various types of enclosures and installations. The engine provides a mounting pad and drive shaft for installation and drive of an AC generator. The ambient air is compressed by a single stage centrifugal compressor, mixed with fuel in the combustion chamber and the mixture's ignited. The resultant high energy gases drive a radial inward-flow turbine wheel. The rotating shaft power of the turbine wheel drives the compressor impeller and an accessory gear train to provide reduced rpm shaft power. The remaining power is the engine power output. This gas turbine was used mainly to drive the alternator in military, portable, generator sets.

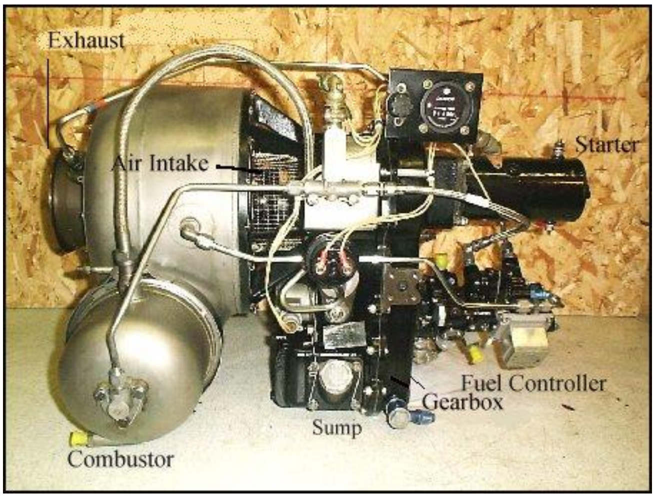
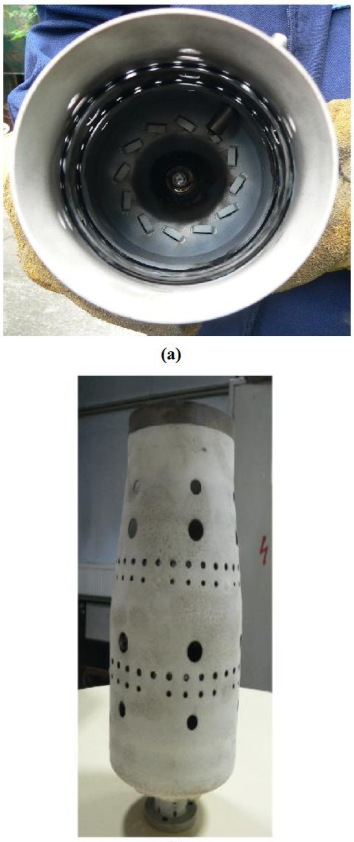


Fig.1 Garrett gas turbine model GTP 30-67 [1]



(a)

(b)

Fig.2 Garrett GTP 30-67 fire tube

In table I are presented a few of the gas turbine engine's technical characteristics.

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 2, August 2013

Table I GTP 30-67 technical characteristics [2]

	Characteristics	Value	Unit
1	Idle rotation speed	52800 ± 200	rpm
2	Output drive shaft rotation speed	8000±15	rpm
3	AC generator power	20	kW
4	Compressor inlet maximum temperature	325	K
5	Exhaust gases maximum temperature	1091	K
6	Load functioning regime rotation speed limits	55400 - 58000	rpm
	Fuel		
7	Main: MIL-T-5624, JP4, Alternative: JP5, Kerosene – VV-K-211, Emergency: MIL-G-5572, Gasoline, Diesel VV-F-800		
8	Fuel inlet pressure	0.35 – 1.38	bar
9	Fuel maximum inlet temperature	330	K
10	Maximum fuel consumption at nominal regime	0.0075	kg/s
11	Maximum fuel consumption at idle regime	0.0044	kg/s

II. NUMERICAL SIMULATIONS

A. Geometry

The figures below present the whole combustion chamber assembly and the fire tube used in the simulations.

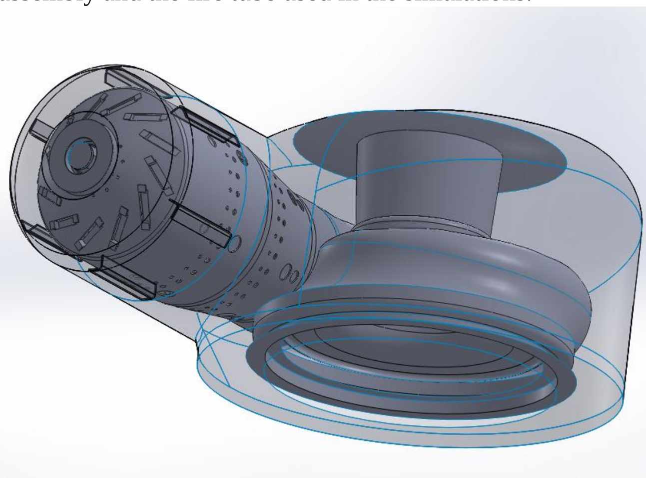


Fig. 3 The combustion chamber assembly

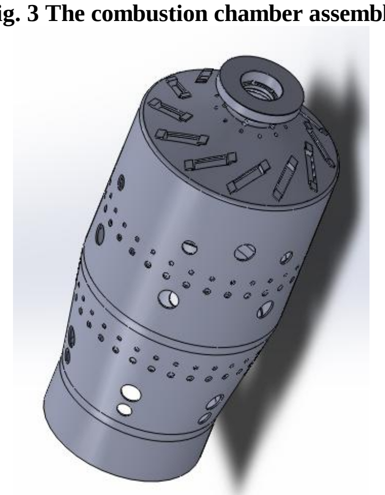


Fig. 4 The fire tube

B. Mesh

Based on the geometry of the combustion chamber assembly presented in fig. 3 an unstructured computational grid of 3.576.588 tetrahedral cells and 592465 nodes has been created using ICEM CFD (fig. 5).

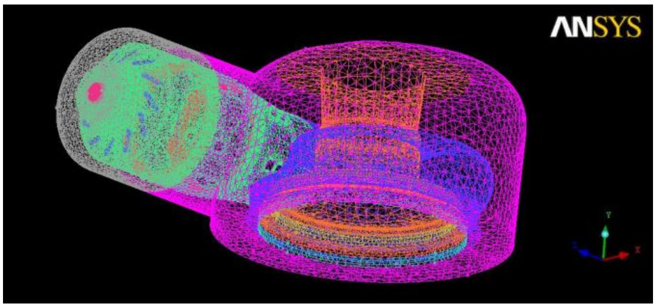


Fig. 5 The computational grid

C. Boundary conditions

Separated fuel and air inlets have been used in the numerical simulations. The air inlet is considered the exit from the compressor – the entrance in the combustion chamber assembly. This is an annular surface of 7540 mm^2. The combustor inlet temperature is of 423K which is the corresponding temperature for the engine's compression ratio of 3 [3]. The fuel, in our case Jet-A, is injected directly in the fire tube through an orifice of 0.017 mm^2. In the simulations, it has been considered that the gas turbine functions at its nominal regime (20kW load) [2], thus the fuel mass flow is of 0.0075 kg/s [2]. The fuel temperature at the entrance in the computational domain is of 300K.

The air mass flow has been determined from the following relation [4]:

$$\frac{\dot{m}_{fuel}}{\dot{m}_{air}} = \frac{1}{\lambda * \min L} \tag{1}$$

where:

$\dot{m}_{fuel}$ - fuel mass flow (kg/s)

$\dot{m}_{air}$ - air mass flow (kg/s)

λ - air excess

minL – the stoichiometric quantity in kg of oxidizer needed to burn 1 kg of fuel

In our case: $\dot{m}_{fuel}$ *=0.0075 kg/s*, λ *=3.5*, *minL =14.67 kg* [4]. By substituting these values in (1), an air mass flow of 0.4 kg/s has been obtained. The value is in good agreement with previously reported data on a similar engine, GTP 36-51, where the mass flow rate is reported to be 0.41 kg/s [5]. Garrett models GTP 36-51 and GTP 30-67 have very similar technical characteristics, thus we can consider that the nominal exhaust flow is very much the same in the case of the two gas turbines. Hence, we can conclude that the air mass flow of 0.4 kg/s used in the numerical simulations is correct. The air enters the fire tube through a series of holes made in the fire tube's wall. The air which enters the fire tube through the holes situated in its upper region, the primary air flux, is

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 2, August 2013

used in the combustion reaction. The air which enters the fire tube through the lower situated holes, the secondary air flux, is used to cool down the fire tube walls.

The fuel, Jet-A, is uniformly injected into the fire tube, in the form of droplets. The values of the droplets diameter have been chosen based on the diagram presented in [4] (fig. 6).

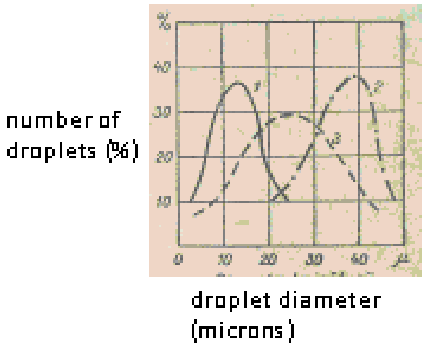


Fig. 6 Droplet distribution in a section of the liquid jet, in 3 cases [4]

The exit from the combustion chamber assembly – entrance in the turbine is the subsonic outlet of the domain. The outlet pressure is set at 2.35 bars, calculated based on the engine data [2].

D. CFD simulation used settings

A three-dimensional unsteady RANS numerical integration of the Navier-Stokes equations has been carried out using the commercial software ANSYS CFX. In these numerical simulations an Eddy Dissipation combustion Model (EDM) [6], based on a one step kerosene-air reaction mechanism from the ANSYS library [7], and an k-ε turbulence model [8] have been used.

To simulate the transformation of the Jet A droplets into Jet-A vapours a Liquid Evaporation Model from the ANSYS library is used [7].

III. RESULTS

In the figure below, the residuals of the numerical simulation after 3000 iterations are presented. It can be noted that, after an initial decrease, the simulation residuals oscillate around a mean value, without further decreasing, as an effect of the unsteady nature of the highly turbulent reactive flow.

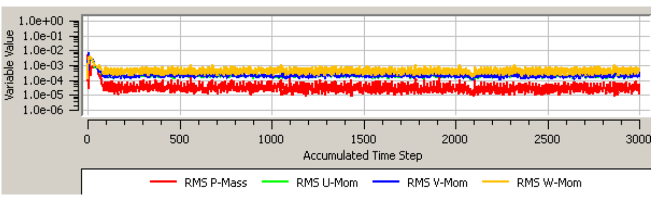


Fig. 7 The residuals after 3000 iterations

To account for the residuals seen above, the pressure and temperature were averaged along 100 iterations using the TECPLOT software [9]. The results in fig. 8 show that the mean pressure is quasi-constant, verifying the typical hypothesis of constant pressure combustion in a gas turbine combustor [4].

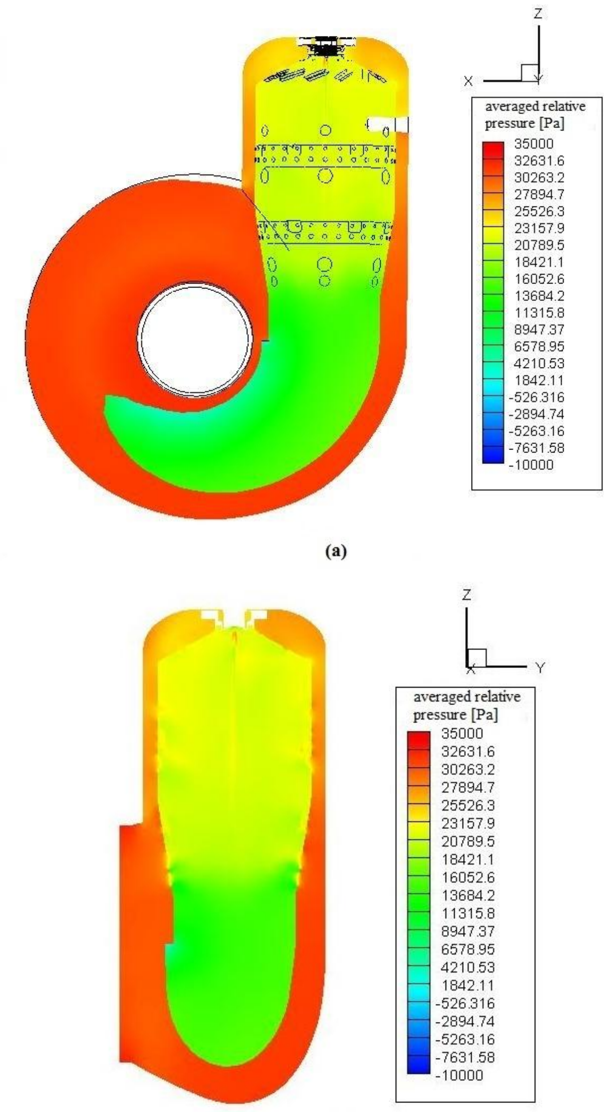
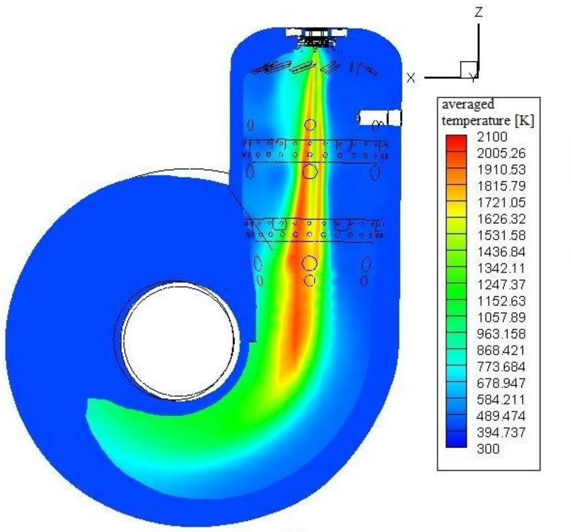
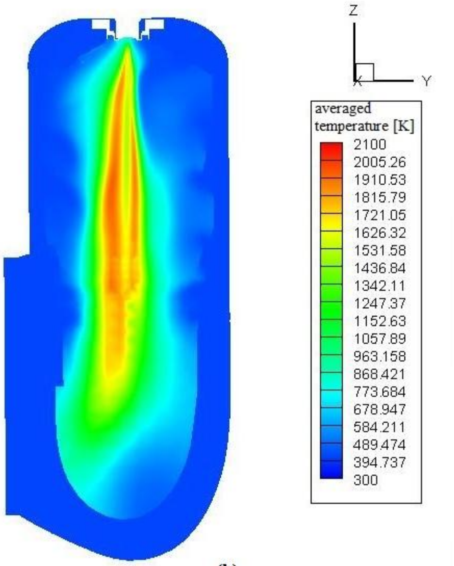


Fig. 8 The averaged relative pressure along 100 iterations (reference pressure = 2.46 bars)



(a)

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 2, August 2013



(b)

Fig. 9 The averaged temperature along 100 iterations

The mean temperature field, presented in fig. 9, shows that the high temperature region is contained in the axial region of the combustor and does not extend into the volute that directs the burned gas to the downstream turbine, thus not endangering the physical integrity of the engine.

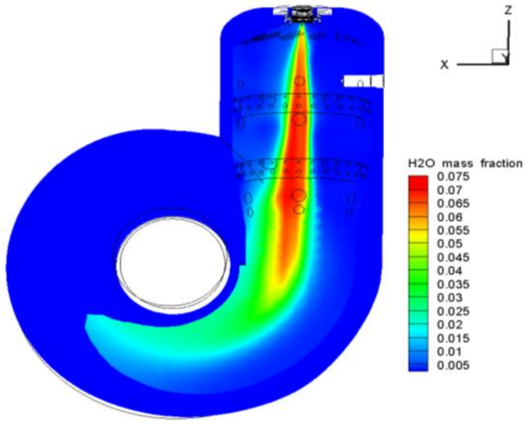


Fig. 10 Avereged H_2O mass fraction field

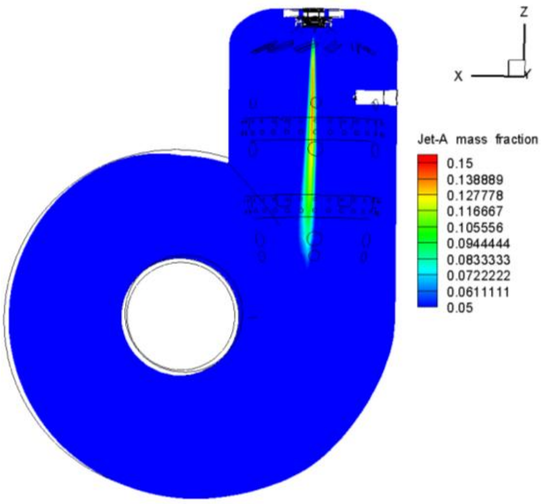


Fig. 11 Avereged Jet-A mass fraction field

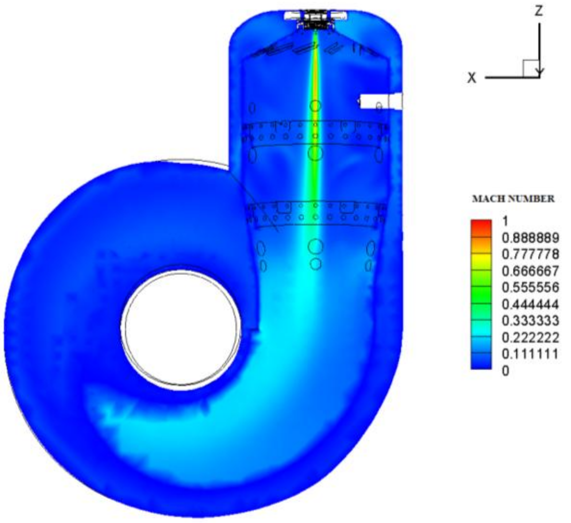


Fig. 12 Avereged Mach number field

In the above figures are presented the averaged H_2O mass fraction field (fig. 10), averaged Jet-A mass fraction field (fig. 11), respectively the averaged Mach number field (fig. 12).

It can be noted that the fuel is completely consumed inside the combustor, thus validating the design. The maximum velocity value is reached in the fuel jet and in the surrounding region, but the velocity decreases downstream fast enough to provide complete combustion inside the flame tube, as indicated by fig. 12.

To verify the numerical simulations accuracy, the average outlet temperature has been recorded along 600 iterations and compared with the designed turbine inlet temperature. A larger time interval was used to ensure high result accuracy. The engine's turbine inlet temperature is around 1100 K [2], while the average outlet temperature obtained in the numerical simulations is near 1200 K, as it can be observed in fig. 13. The accuracy of the numerical simulation is quite good, and the mean outlet temperature oscillations are small, indicating proper operation of the combustor.

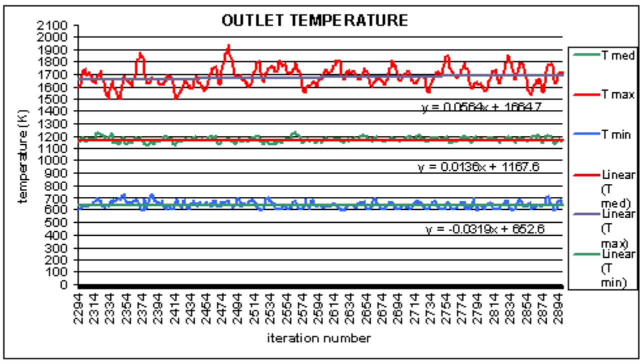


Fig. 13 Outlet temperature

IV. CONCLUSION

The purpose of the numerical simulations presented in this paper was to validate the numerical models chosen to simulate the combustion process of Jet-A in an existing gas turbine combustion chamber (the EDM combustion model, the turbulence model, the liquid to vapours transformation model).

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 2, August 2013

Taking in consideration that the above numerical simulations results have a reasonable accuracy, in this particular case, it can be concluded that the used numerical models can also be used to simulate the combustion process under different conditions: using different combustion chambers or using different fuels.

For the future we have in mind the research of different fuels combustion in a gas turbine, thus the results obtained in these simulations will help adapt the existing combustion chamber to these new fuels and fuel mixtures.

REFERENCES

[1] http://www.technologie-entwicklung.de/Gasturbines/GTP30-67/gtp30-67.html

[2] "Shaft Power Gas Turbine Engine. Mode GTP 30-67," Technical Manual, 1966.

[3] T.Bose, "Airbreathing Propulsion: An Introduction", Springer Aerospace Technology, DOI 10.1007/978-1-4614-3532-7, 2012.

[4] V. Pimsner, C.A. Vasilescu, and G.A. Radulescu, "Energetica turbomotoarelor cu ardere interna", Ed. Academiei Republicii Populare Romannia, Bucuresti, 1964.

[5] R.H. Bode, J.S. Howland, and G.E. Pollak, "Definition of an Improved Wet Support Bridge Concept and Related System Analysis", report to US Army Mobility Equipment Research and Development Command (MERADCOM), 1981.

[6] N. Peters, "Turbulent Combustion", Cambridge University Press, 2000.

[7] ***, Product-Manual for ANSYS CFX v11, ANSYS INC, 275 Technology Drive Cannonsburg, PA, Released 13.0, November 2010.

[8] S. Pope, "Turbulent flows", Cambridge University Press, 2000.

[9] ***, Tecplot 360 User's manual, Tecplot, Inc., Bellevue, WA, 2012.

AUTHOR BIOGRAPHY

A.C.Petcu: Researcher since 2010 at Romanian Research & Development Institute for Gas Turbines COMOTI, in the Combustion and Heat-resistant Coatings Department, where she carries out experimental activities and numerical analysis, and is involved in the development of combustion algorithms. Doctoral student at the "Politehnica" University of Bucharest, Department of Aerospace Engineering, under the advisement of Prof. Dr. Corneliu Berbente. Earned a Master of Science degree in 2011, from the "Politehnica" University of Bucharest, Department of Aerospace Engineering, on the subject of Software Development for Aerospace Engineering, and a Diplomat Engineer degree, in 2009, from the "Politehnica" University of Bucharest, Department of Applied Sciences, focusing on Applied Mathematics and Informatics in Engineering. Publications: "Numerical simulations of round turbulent jet flames", „Camelina oil – kerosene mixtures combustion", "Flow control through an airplane's intake". Member in the research teams of several national and European projects: "Research on the development of biofuel-powered gas turbines", "Tangential Impulse Detonation Engine", "Efficient Systems and Propulsion for Small Aircraft".

C. Sandu: Senior researcher since 2006 at Romanian Research & Development Institute for Gas Turbines COMOTI, in the Combustion and Heat-resistant Coatings Department, where he carries out experimental activities. Good experience in experimental investigation techniques such as laser PIV (Particle Image Velocimetry), LIF (Laser Induced Fluorescence). Earned a Mechanical Engineer Diploma in 1981, from the Polytechnic Institute of Bucharest, Mechanics Department. Member in the research teams of several national and European projects: "Efficient Systems and Propulsion for Small Aircraft","Turboshaft Engine Exhaust Noise Identification", "Development of Helicopter Exhaust Engine Noise Reduction technologies".

C. Berbente: Professor Emeritus at "Politehnica" University of Bucharest, Aerospace Engineering Department. Earned a Ph.D. degree in 1970 from the Polytechnic Institute of Bucharest in the field of aerodynamics and the engineer diploma in 1960 from the Polytechnic Institute of Bucharest. Member of American Mathematical Society, of American Romanian Academy of Arts and Sciences and of the Academy of Technical Sciences of Romania. Scientific Papers: 131. Among them: "Numerical simulation of pollutant emission in a turbine combustor", "Numerical Simulation of Blast Wave Propagation and Focusing Using Fluent and Other Finite Volume Finite Solvers", "Large - Eddy Simulation of Unsteady Rotor Aerodynamics and Acoustic Emissions". Books: 19. Among them: "Numerical Methods in Aviation", "Numerical Methods in Fluid Dynamics", " Gasdynamics and Aerothermochemistry". Scientific research contracts: 36. Among them: "Experiments on combustion of liquid fuel jets injected to a supersonic hot steam", "Multiphase multicomponent turbulent flow", "Research regarding the chemical pollution diminution of turbojet engines by multistage fuel injection in turbines".