Micro gas turbine combustion chamber CFD modelling

A C Mangra
National Research & Development Institute for Gas Turbines COMOTI, 220D Iuliu Maniu Bd., sector 6, 061126, Bucharest, Romania

E-mail: andreea.petcu@comoti.ro

Abstract. The interest in micro gas turbines has been steadily increasing. As a result, the researchers' attention has been focused on obtaining optimal configurations for micro gas turbines depending on the applications in which they are used. Micro gas turbines are suitable for both civil and military applications. Presently, they are often used in the development of Unmanned Air Vehicles (UAVs). Micro gas turbines have the following advantages: a wider range of options regarding the used fuel and a significant system weight reduction. This paper presents the CFD modelling results regarding an annular type combustion chamber. This combustion chamber is part of a 80 daN micro gas turbine, destined to equip a small scale multifunctional airplane. The combustion chamber is equipped with eight pressure-swirl injectors, using Jet-A as fuel. A 3D 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. An unstructured computational grid, generated using ICEM CFD, has been used. The fuel is injected directly in the fire tube in the form of droplets. The initial fuel droplet's diameter has been considered to be 50μm and the fuel spraying cone angle has been set at 90°. The fuel droplet break-up and evaporation processes have been simulated using numerical models available in ANSYS CFX library. A two steps reaction mechanism, which takes into consideration the formation of NO, has been used. The results thus obtained are encouraging. The flame developed in the central area of the fire tube, its walls thus not being subjected to high temperatures. Also, the maximum temperatures were obtained in the primary zone of the fire tube. The temperature then decreased in the fire tube's secondary zone and dilution zone, until it reached an average value of 1200 K at the combustion chamber exit. Regarding the pressure, based on the numerical simulation results, a 5% pressure loss has been obtained. The numerical results will be validated by conducting combustion tests on a testing rig which will be developed inside the institute's Combustion Chamber Laboratory.

1. Introduction
Micro gas turbines are smaller-scale versions of gas turbines used today in aviation or various cogenerative applications. The interest in micro gas turbines has been steadily increasing. As a result, the researchers' attention has been focused on obtaining optimal configurations for micro gas turbines depending on the applications in which they are used. Micro gas turbines offer the advantages of compact size, low weight per unit power and multi-fuel capability. Micro gas turbines are suitable for both civil and military applications. The reduced dimensions have transformed them into accessible teaching material that allows to study gas turbines performances. The ones used in industrial applications are able to provide high power/mass ratio, ensuring the required air flow due to the high
speed of the turbocharger group. Presently, micro gas turbines are often used in the development of Unmanned Air Vehicles (UAVs) due to the high traction/mass ratio.

Micro gas turbines usually consist of a single stage compressor connected through a shaft to a single stage turbine. The combustion chamber is of annular type, in order to reduce the overall dimensions.

Worldwide, the first micro gas turbine for aviation applications has been developed by the French manufacturer JPX. Soon two other companies, AMT and Schreckling, have produced their own micro gas turbines.

In this paper are presented the CFD modelling results regarding an annular type combustion chamber, part of a 80 daN micro gas turbine destined to equip a small scale multifunctional airplane. The combustion chamber is composed mainly of an outer casing, an annular type fire tube and 8 pressure-swirl fuel injectors.

The injectors have the role of introducing the fuel into the fire tube in the adequate amount and in the form appropriated for an efficient and stable combustion process. The injector has to ensure a fine spraying of the fuel in order to facilitate a rapid combustion process. A sheet of fuel is formed at the injector exit, from which filaments and then droplets are produced [1-5]. The resulting droplets become smaller and smaller as the spraying process continues, until they evaporate thus resulting fuel vapours.

Inside the fire tube the fuel vapours are mixed with pressurized air received from the compressor. The reacting mixture thus obtained is burned, transforming the fuel's chemical energy into heat. The resulted hot exhaust gases expand in the turbine producing mechanical work.

The fire tube is divided into three zones: the primary zone, the secondary zone and the dilution zone. The main function of the primary zone is to anchor the flame and provide sufficient time, temperature and turbulence to achieve a complete combustion of the fuel-air mixture. In the primary zone a recirculation zone is created to ensure flame stability and to entrain a portion of the hot combustion gases in order to provide continuous ignition to the incoming fuel-air mixture. In the secondary zone, the temperature is dropped to an intermediate level by the addition of air. This also allows the further combustion of CO and other unburned hydrocarbons (UHC) resulted from the primary zone. The role of the dilution zone is to admit the remaining air and to provide an exhaust gases stream with a temperature distribution that is acceptable to the turbine [1].

Over the years CFD techniques have proven useful in designing different component parts of a gas turbine, including the combustion chamber. From the specialized literature it can be seen that a lot of researchers have used CFD modelling to design and develop combustion chambers [6-11].

2. Fuel mass flow determination
As starting point for developing the combustion chamber, the following calculated data was available (table 1):

| Parameter                              | Value  | Unit |
|----------------------------------------|--------|------|
| air mass flow                          | 1.4    | kg/s |
| P^2                                    | 476201 | Pa   |
| T^2                                    | 481.452| K    |
| Compressor stator exit flow angle      | -18    | °    |
| T^3                                    | 1173   | K    |

The air excess has been determined using equation (1) [12]:
where $H_i$ is the inferior calorific power of the fuel, $c_{pg}$ is the exhaust gases specific heat at constant pressure, $c_{pa}$ is the air specific heat at constant pressure, $T_3^*$ is the total temperature at compressor exit, $T_3'$ is the total temperature at combustion chamber exit, and $minL$ is the theoretical air quantity necessary for complete combustion of the fuel. In this case the used fuel is Jet A. Thus $H_i$ is 42800kJ/kg and $minL$ is 14.6. Replacing in equation (1), an excess of air of 3.5 is obtained.

Using equation (2) [12]:

$$
\alpha = \frac{H_i - c_{pg} \cdot T_3}{c_{pg} \cdot T_3 \cdot minL - c_{pa} \cdot T_2 \cdot minL}
$$

$$
mc = \frac{ma}{\alpha \cdot minL}
$$

where $mc$ is the fuel mass flow, $ma$ is the air mass flow, $\alpha$ is the air excess, and $minL$ is the theoretical air quantity necessary for complete combustion of the fuel, a fuel mass flow of 0.0274 kg/s has resulted.

3. Numerical simulation setting

In order to develop the new combustion chamber a 3D steady RANS numerical integration of the Navier-Stokes equations has been carried out using the commercial software ANSYS CFX. The computational domain is composed of the combustion chamber, starting from the compressor's stator exit and ending at the turbine stator entrance.

3.1. Geometry and mesh

In figures 1 and 2 is presented the fire tube used in the simulations.

![Figure 1. The fire tube.](image1)

![Figure 2. Section through the fire tube.](image2)

Based on the geometry of the fire tube presented in figures 1 and 2, an unstructured computational grid of 14818116 elements and 2466339 nodes has been generated using ICEM CFD (figure 3). The mesh has been refined inside the fire tube, as it can be seen in figure 4.

![Figure 3. Fire tube mesh.](image3)

![Figure 4. Mesh refinement.](image4)
3.2. Boundary conditions
The reference pressure has been set at 101.325 Pa.

The air inlet has been considered the entrance in the combustion chamber. The following conditions have been imposed: an air relative total pressure of 376.201 Pa, an air total temperature of 481.452 K and a flow angle of -18°.

The fuel has been introduced in the computational domain using 8 Particle Injection Regions, one for each injector. The initial fuel droplet diameter of 50 μm and the spraying angle of 90° have been imposed. The fuel mass flow corresponded to each injector was set at 0.003425 kg/s and the fuel temperature at 288 K. The fuel droplets primary break-up process has been simulated using the Blob Method model. From the resulting droplets, by continuing the spray process, smaller and smaller droplets are obtained. This phenomenon (secondary break-up) has been simulated using the Cascade Atomization Break-up (CAB) model. Finally, the droplets reach a diameter small enough to vaporize. This process has been modelled using the Liquid Evaporation Model.

The walls have been considered adiabatic. At outlet, the total mass flow (air mass flow plus fuel mass flow) has been imposed, namely 1.4274 kg/s.

![Figure 5. Boundary conditions.](image)

A RANS type turbulence model has been chosen, namely the k-ε model, which is a numerically stable and robust model and very popular in the realization of technical applications numerical simulations [13-16].

The chosen combustion model has been the Eddy Dissipation Model (EDM), based on a two-step kerosene-air reaction mechanism, imported from the ANSYS library, which also takes into consideration the formation of NO. A simple reaction mechanism has been chosen because the pollutant emissions level has not been of interest at this point. The EDM model has been chosen because of its simplicity and robust performance in predicting turbulent reacting flows. This model is very popular in the realization of technical applications numerical simulations [17-20].

4. Results
The average total pressure along the fire tube has been monitored in order to better understand the phenomena that takes place inside the fire tube during the combustion process (figure 6). In figure 7 is presented the total pressure field on a longitudinal plane through the fire tube. The total pressure values presented in figure 7 are relative to the reference pressure set at 1 bar.
From figure 6 it can be observed a normal behaviour of the total pressure inside the fire tube. The points represent mean values of the total pressure on traverse planes at different values of the z coordinate along the fire tube. The total pressure decreases towards the fire tube exit. The same behaviour is observed in figure 7. A pressure loss of 5% has been obtained at the fire tube exit.

From figures 8-11 it can be observed that the flame developed inside the fire tube, not exceeding its length. Also, the maximum temperatures were obtained in the primary zone of the fire tube. The flame developed in the central area of the fire tube, its walls not being subjected to high temperatures.
Figure 12. Average total temperature along the fire tube.

From figure 12 it can be observed a normal behaviour of the total temperature inside the fire tube. The points represent mean values of the total temperature on traverse planes at different values of the z coordinate along the fire tube. The temperature rises in the primary zone of the fire tube, as the combustion reaction takes place, reaching a maximum. The temperature then decreases in the secondary zone and the dilution zone of the fire tube. This is achieved by introducing a larger quantity of air through holes placed on the fire tube walls.

In figures 13-15 are presented the outlet pressure field, temperature field, respectively velocity field.

Figure 13. Outlet total pressure field.  
Figure 14. Outlet total temperature field.  
Figure 15. Velocity outlet field.
The average parameters at the combustion chamber exit are presented in table 2.

**Table 2. Combustion chamber exit average parameters.**

| Parameter | Value | Unit |
|-----------|-------|------|
| $P^*$ | 453924 | Pa |
| $T^*$ | 1195 | K |
| velocity | 106 | m/s |
| flow angle | -19 | ° |

Comparing the average temperature value obtained through numerical simulation (1195 K) with the one calculated (1173 K), an error of 1.9% has been obtained.

5. Conclusions

So far the numerical simulations results are encouraging. The flame develops inside the fire tube, not exceeding its length. The maximum temperatures are obtained in the primary zone of the fire tube. The walls are not subjected to high temperatures.

The next step is to extend the computational domain by adding the compressor. Thus the air pressure and temperature fields at the combustion chamber entrance will be closer to the ones existent during the micro gas turbine functioning. The new results will be compared with the ones presented in this paper.

The numerical results will be then validated by testing the fire tube on a testing rig which will be designed and built in INCDT COMOTI Combustion Chamber Laboratory.

6. Reference

[1] Lefebvre A H and Ballal D R 2010 *Gas Turbine Combustion* (New York: CRC Press Taylor & Francis Group)
[2] Pimsner V, Vasilescu C A and Radulescu G A 1964 *Energetica turbomotoarelor cu ardere interna* (Bucharest: Academia Republicii Populare Romane Publishing House)
[3] Lemneanu N, Cristea E D and Jianu C 1982 *Instalatii de ardere cu combustibili lichizi* (Bucharest: Tehnica Publishing House)
[4] Stambuleanu A 1971 *Flacara industriala. Procese de ardere cu combustibili lichizi* (Bucharest: Tehnica Publishing House)
[5] Lefebvre A H and McDonell V G 2017 *Atomization and Sprays* (New York: CRC Press Taylor & Francis Group)
[6] Crocker D S, Nickolaus D and Smith C E 1999 *J. Eng. Gas Turbine Power* **121** 89
[7] Sreenivasarao K and Bhatti S K 2013 *International Journal of Engineering Research & technology* **2**
[8] Pradhani N L, Rajesh A and Ganesh Prasad M S 2016 *Journal of Aeronautics & Aerospace Engineering* **5**(1) doi:10.4172/2168-9792.1000170
[9] Hadi H A, Kamran M S and Abbas N 2016 *Science International (Lahore)* **28** 5121
[10] Vilag V, Vilag J, Carlanescu R, Mangra A and Florean F 2019 CFD application for gas-turbine combustion simulations *Computational Fluid Dynamics Simulations*, ed G Ji and J Zhu (London: INTECH)
[11] Rodrigues L O, Alencar H S, Nascimento M A and Venturini O J 2007 *Proc. of ASME 2007 POWER Conf. (San Antonio)* p 547
[12] Popa B and Vintilca C 1973 *Termotehnica, masini si instalatii termice* (Bucuresti: Didactical and Pedagogical Publishing House)
[13] Dumitrescu O, Gherman B G and Porumbel I 2018 *INCAS Bulletin* **10** 63
[14] Carlanescu R, Prisecaru T, Petcu A C and Porumbel I 2015 Proc. Int. Conf. on jets, wakes and separated flows (Stockholm) p181
[15] Petcu A C, Sandu C and Berbente C 2013 International Journal of Engineering and Innovation Technology 3 487
[16] Enache M, Mangra A C, Carlanescu R and Florean F G 2017 Proc. of the 6th CEAS Air and Space Conference (Bucharest) p 342
[17] Gherman B, Malael I, Florean F G, and Porumbel I 2019 E3S Web of Conferences 85
[18] Florean F G, Gherman B G, Porumbel I, Carlanescu R and Dumitrascu G 2013 Proc.1st International Conference Newchallenges in Aerospace Sciences (Bucharest) p 154
[19] Petcu A C, Gherman B, Florean F G, Sandu C and Porumbel I 2012 Nonequilibrium processes in Plasma, Combustion and Atmosphere, ed Starik A M and Frolov S M (Moscow: TORUS PRESS) p 395
[20] Dias F L, Rosa do Nascimento M A and de Oliviera Rodrigues L 2014 Journal of Mechanical Engineering and Automation 2 73

Acknowledgments
This work was carried out within “NUCLEU” Program, supported by the Romanian Minister of Research and Innovation, project number PN 19 05 01 05 - 2.3