Pre-mixed and Diffusion Flames Assessment using CFD Tool for Natural Gas and Biogas Fuels in Gas Turbine Combustion Chambers

LUCILENE DE OLIVEIRA RODRIGUES, MARCO ANTÔNIO ROSA DO NASCIMENTO
Mechanical Engineering Department
Itajubá Federal University – UNIFEI/NEST
BPS, av.,1303, Pinheirinho
Itajubá MG, Brazil
lucilener@unifei.edu.br, marcoantonio@unifei.edu.br, http://www.nest.unifei.edu.br

Abstract: In the past few years, with the development of advanced numerical computational codes, numerical simulation became a promising option to develop and improve technology in different fields. The results obtained through simulations are used to gather important information during the design phase or optimization of industrial equipment. Its employment generates reliable results at low cost due to the reduced number of experiments as well as the opportunity to develop new products and perform many simulations before construction of a new product. With the recent energy shortage and the reduction of the fossils fuel reserves, the industrial sector starts to develop more compact equipment that can be fed with different fuels, attending a wide range of heat and power demand. The aim of the work is to assess thermal-aerodynamics and emission using numerical simulation (Computational Fluid Dynamics CFD) of a 600kW simple cycle gas turbine combustion chamber. The thermal-aerodynamics assessment of the combustion chamber will be presented, showing the profiles of temperature and emissions, when used the premixed and diffusion flame for the natural gas and biogas fuels, respectively.

*Key-Words: C*ombustion Chamber, Flame, Premixed, Diffuse, Different Fuels, Gas Turbine

1 Introduction

Aiming at facilitating the analysis and reducing the number of experimental tests on combustion chambers, there are many computational tools that can be used to help the development of the design. This previous analysis is extremely important because of the reduction in the costs and the reduction in the development of the project as a whole. The CFD (Computational Fluid Dynamics), based on the Finite Volume Method, has been widely used for the analysis of annular and tubular combustion chambers. For example, Lai (1997), used the CFD analyses previewing the exact hot spots, which correspond to the locations of the most critical spots of the combustor; Turrel et al (2004), checked the temperature peaks in the central blade; Darbyshire et al (2006) studied the conditions of the fuel/air mixture input. Through computational fluid dynamics (CFD), it is possible to attain the basic geometry of the chamber and carry out several analyses of the process that happen in this chamber.

Lefebvre and Reid (1966) studied the influence of turbulent flow in the flame, verifying the relation of the flow initial velocity and the initial temperature of the fuel. On the other hand, Kuo (1986) presented mathematic models to calculate the propagation velocity of the flame with laminar and tubular flow.

Cameretti et al (2004), carried out a study of three types of combustors: a diffusive type conventional annular combustor with a poor pre-mixture, one with radial swirls and tubular geometry and a tubular one. The turbulence model used for carrying out the simulations was k-ε and the fuel, natural gas and biomass. Through the analysis, it was possible to assess the behavior of the tow fuels that were used in the three available geometries.

Another significant point is the fuel flexibility. This flexibility in small gas turbines was tested by Andreini et al (2006), involving two combustion systems: one operating with natural gas and another operating with a fuel of low specific heat. The focus of the observation was the ignition limits and the performance of the combustors, analyzing hot spots and the emission of pollutants.

Another aspect that has been widely studied is the pollutant emission factor because of the several environmental problems faced by the world today. Allen (1998) showed a technique that allows one to control the fuel/air mixture, aiming at avoiding high temperature peaks of the flame that favor the formation of thermal NOx, pollutant responsible for the acid rain.

The development and validation of the CFD methodology regarding the emission of pollutants was also studied by Kutsenko et al (2006) in different combustors. The physical and chemical modeling of the process was carried out: turbulent flow of the reactive gases, heat transfer, chemical kinetics and NOx formation. The k-ε, RNG k-ε, RSM, k-ω SST turbulence models, Flamelet and Flamefront combustion models, methane and kerosene were used as fuels and the NO formation model were used.

The present study presents the results of the analysis of the thermal and aerodynamic processes of combustion chambers at steady state and design point, using the CFX numeric code. This analysis considers the distribution of the temperature at the combustor outlet, flame tube, emissions and load loss for combustion regime with pre-mixed and diffused flame, and also the use of different fuels. In addition, a study of the experimental techniques used for analyzing combustion chambers was also carried out. The analyzed chambers go with a 600kW gas turbine that is still being developed.

NOMENCLATURE

$\dot{m}$	mass flow of the primary flow	(kg/s)
A_{ref}	reference area of the combustion chamber	(m^2)
D_{ref}	reference diameter	(m)
T	temperature of the reaction zone	(K)
B	parameters attained in low pressure tests	(-)
P_3	chamber input pressure	

2 Combustion Chamber Modeling

The project of combustion chambers of the gas turbines engine is complex task and involve many parameters in its development. The way to minimize this problem is the use of an existing combustion chamber already tested, as reference, or progenitor.

Taken as a progenitor, the annular combustion chamber of a gas turbine manufactured by Solar Turbines was simulated, aiming at assessing the combustion aerodynamics. Afterwards, a new chamber was designed according to the data attained from the design of the compressor/turbine set for a 600 kW simple cycle one shaft gas turbine engine.

The combustion chamber of the 600kW simple cycle gas turbine engine was designed by using aerodynamic scaling technique, that is, loading coefficient of the Solar turbine combustion chamber, which involves mass flow, pressure, temperature and experimental coefficients. This technique allows one to design a combustion chamber with a reduced number of experiments, for most of the experiments have already been carried out in the progenitor chamber. The chamber used as a reference was a Model T-62T-32 single cycle one shaft gas turbine engine with a power of 60 kW manufactured by Solar Turbines. It is installed in the Micro-turbines and Biomass Gasification in the Federal University of Itajubá. As the design is carried out based on the loading factor, the compressor and the turbine connections with the combustion chamber will be made through ducts.

The injector was designed to attain pre-mixed and diffuse flames, so was the dilution and anchoring system of the flame. The CFD simulation was carried out by using the turbulence and combustion models. In order to validate the turbulence model a Floxcom report (2003) was used for Alencar (2005). This way, it was possible to simulate Solar's combustion chamber and the new combustion chamber of the 600 kW gas turbine engine, operating at steady state for the design point.

Some geometrical changes are necessary in order to use different fuels in a chamber that was designed for a specific fuel, so as to reach a same or a higher efficiency value. Lourival (2000) carried out the analysis of a combustion chamber operating with natural gas and the consequences of the fuel change for a low calorific gas: the gas from biomass gasification. The need for geometrical changes was necessary in order to change the fuel. An alteration in the number of holes, both primary and in the dilution in combustion chamber was necessary.

Another significantly important factor is that this study used gaseous fuels such as methane or natural gas, and the simulated Solar Turbine chamber was designed to use liquid fuels (kerosene). However, the low heating value of these fuels has similar values, the LCV of the natural gas is 47 MJ/kg and the LHV of kerosene is 43 MJ/kg.

3 Thermodynamic Calculation of Micro-Turbine Cycle

In order to attain the design parameters that are necessary for the CFD simulation of the combustion chamber of the progenitor turbine, a computational program was used for simulating the cycle. This way, the simulation of the Solar gas turbine engine aimed at attaining the approximate parameters of the combustor inlet and outlet at design point. This does not invalidate the CFD analysis of the Solar combustor, given that the values calculated by the computational program are attained based on some parameters provided by the manufacturer. The idea of the simulation is to attain a gas turbine that is equivalent to Solar gas turbine. The software used for the simulation was the Gate Cycle GE Enter Software, which gives the conditions for the air input, for the combustion input and for the output of the gases that are products of the combustion.

The design input parameters used in the GateCycle GE Enter Software simulation for both fuels are shown in Table 1. The parameter values selected in Table 1 were based in the current technologies for radial turbo machineries. The turbine inlet temperature of 850ºC was selected because it is the maximum temperature the material of the radial turbine can support, maintaining the mechanical resistance and the useful life without any blade cooling.

Table 1 Design Point Input Parameters

Description	**Values**	**Unity**
Ambient temperature	288	K
Ambient Pressure	101.32	kPa
Turbine inlet temperature	1123	K
Temperature of fuel	288	K
Pressure ratio	4	-
Compressor adiabatic efficiency	80	%
Combustion adiabatic efficiency	99	%
Turbine efficiency	85	%
Mechanical efficiency	98	%
Combustion chamber pressure loss	2	%
Combustion chamber inlet temperature	461	K
Air Mass flow rate	4.288	kg/s
Fuel mass flow rate (natural gas)	0,072	kg/s
Fuel mass flow rate (biogas)	0,16	kg/s
Air humidity	60	%
Gas turbine power output	600	kW

The compositions of natural gas and biogas are shown in Table 2.

The model used for the CFX simulation was: SST turbulence model, based on the validation carried out by Alencar (2005).

For the use of combustion models, it is necessary to validate them. However, as a testing rig of the chambers in this study is not available, the literature was used as start point for choosing the combustion models, based on the validations that were found. For the validation of the combustion model at diffuse regime, the study developed by Wunning (1996) was considered. It used sensors to read the temperature inside a furnace. It is possible to observe that the Flamelet Model has a behavior that is closer to the experimental results when it is compared to the Global Fast Chemistry.

Table 2 Composition of the natural gas that was used.

Natural gas		
Species	**Composition**	
	Molar fraction	**Mass fraction**
CH_4	0.8795	0.7939
C_2H_4	0.0913	0.1442
CO_2	0.0174	0.0432
N_2	0.0118	0.0186
Biogas		
Species	**Composition**	
	Molar fraction	**Mass fraction**
CH_4	0.695	0.4557
CO_2	0.305	0.5443

4 Combustion Chamber for the 600 kW Turbine

The combustion chamber studied as a reference, Figure 1, was divided into six parts for the CFX simulation. This number represents the amount of injectors it has, according to studies carried out by Gosselin et al (2000) and Rizk and Monglia (1991).

This division is very important due to some facts, among them: the chamber is formed by equal sectors, which favors their analysis; it is possible to make a more refined grid, aiming at improving the carried out analysis; lower computer memory utilization; and a lower time to process the results.

Several aspects must be taken into account in order to scale a combustion chamber, for example, flame adiabatic temperature and value of the CO released in the region of the flame. The scaling strategy is to use the temperature of the progenitor combustor reaction zone, using the loading parameters, according to equation 1.

$$\frac{1}{\theta} = \frac{\dot{m}}{P_3^{1,75} * A * D^{0,75} * e^{\frac{T}{B}}} \qquad (1)$$

Where: $\dot{m}$ = mass flow of the primary flow,
A = combustion chamber reference area (primary zone)
D = reference diameter (primary zone)
T = temperature of the reaction zone, T3 is frequently used (combustion chamber temperature input)
B = parameters attained during low pressure tests. They can be satisfactorily used for several types of combustion cameras with a value of 300, Lefebvre (1983).

The loading calculation, using equation 1, will be the basic reference for the scaling, i.e., the same loading (same value) or a similar one will be searched for the new chamber that will be designed. Table 3 shows the thermal, aerodynamic and geometrical data and the result of the Solar gas turbine loading coefficient calculation.

Table 3 Loading data and values for Solar Turbines

Combustion chamber - Solar Turbines			
Variable	**Unit**	**Value**	
Air mass (m) - total	kg/s	0.956	
Pressure (P_3)	Pa	405300	
Reference area (A_{ref})-tot	m^2	0.038118632	
Reference diameter (D_{ref})	m	0.062901	
Temperature (T_b)	K	473	
B coefficient		300	
Rays		**Loading**	
r_1	0.1279	$1/\theta$	**6.33842E-09**
r_2	0.064999	θ	157768126.5

The 600 kW gas turbine engine combustion chamber must be fit between the previously calculated compressor and the turbine, Figure 2.

The aerodynamic scaling maintains the thermal and aerodynamic characteristics of the chamber taken as a reference, which is a combustion chamber whose experimental data are available. This technique reduces the design cost of the new combustion chamber.

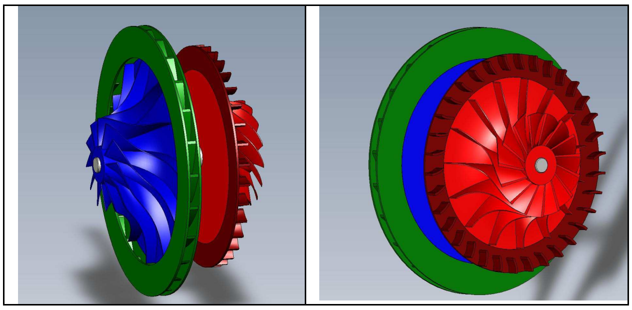

Figure 2. Compressor/turbine sets

Then, the scaling up of the Solar Turbines chamber was carried out, based on the designs of the compressor and of the turbine, Figure 2, introducing some modifications for a better recirculation of the flow and reduction in the load losses of the new designed chamber, for example, wall near the output of the flame tube, according to Figure 3.

Figure 3 shows some of the geometrical changes in the longitudinal plan of the combustion chamber, where the Solar chamber is divided into six parts and the new designed chamber, divided into twenty parts because of the new number of injectors. This division was based on the number of fuel injectors.

As it is possible to observe in Figure 3, a simplified injector was used for the simulation of the Solar Turbines combustion chamber. However, for the design of the new chamber it was necessary to develop a new injector, using appropriate values of the flame adiabatic temperature, aiming at reducing the emission of pollutants.

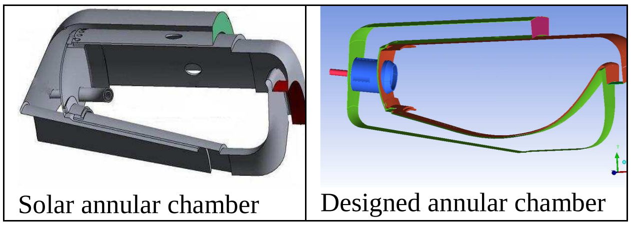

Figure 3 – Changes in the geometry of the chamber

The amount of air necessary for the combustion, i.e., the amount of primary air and dilution air had to be calculated in order to design the new injector. For that, an arrangement for air distribution and fuel input in the combustion chamber was assembled, where the amount of available air is the one coming from the compressor; one part is used as primary air and the rest is used for cooling the walls and for the dilution of the gases that are products of the combustion. The start point for this calculation was the flame adiabatic temperature.

4.1 Chamber with natural gas pre-mixed flame

This stage of the study used natural gas, whose composition is in Table 1. Other input data were also necessary to carry out the calculation of the adiabatic temperature of the flame and the distribution of the air coming from the compressor.

For the stoichiometric calculation, the fuel mass flow is 0.07 kg/s and the flame temperature is 2499 K. Aiming at achieving a flame temperature of approximately 1800ºC, an iterative process in an Excel sheet was used, which resulted in an equivalent ration of 0.5 and the air excess was 80%. Table 3 shows the amount of primary air in the combustion zone, the amount of secondary air destined to cooling and the fuel mass flow that is necessary to reach the temperature of 1800ºC.

With the values of the air and fuel mass flows that are necessary for the pre-mixture area, aiming at attaining low levels of CO emissions. It is possible to start the design of the air and fuel injector for the combustion reaction In general, a proposal regarding the number of injector is made based on a detailed study, and then the experimental analysis or a 3D computational simulation is carried out to help to make the necessary decisions.

Table 3 – Parameters calculated in the iterative process for a pre-mixed flame

Calculated values (1 sector)		
Parameter	**Value**	**Unit**
Primary air mass flow	1.8	kg/s
Secondary air mass flow	2.49	kg/s
Fuel mass flow	0.06	kg/s
Flame adiabatic temperature	1865	K

For the calculations of the injectors the equation contained in Lefebvre (1983) were used. The fuel and air injectors designed for the combustion chamber with a pre-mixed flame can be seen in Figure 4. It is possible to observe in Figure 4 that the fuel injector was assembled inside the primary air injector, in such a position that the air/fuel mixture is favored before the combustion zone.

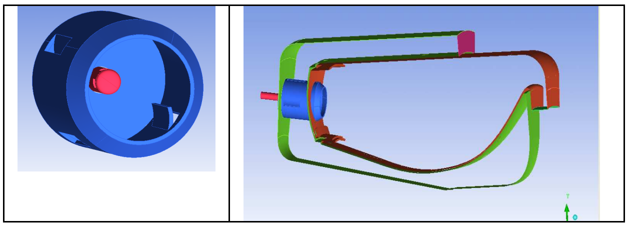

Figure 4 – Isolated assembly of the air and fuel injector and in the sector that will be analyzed, pre-mixture (natural gas)

4.2 Chamber with natural gas diffuse flame

For The design of the fuel and primary air injector for the diffuse flame, the same previous concepts were used, but now, an equivalent ratio of 0.98 was used. An air excess of 3% was used for reaching a temperature of approximately 1800ºC for a diffuse flame regime.

Table 4 shows the amount of primary air in the combustion zone, the amount of secondary air destined to cooling and the fuel mass flow that is necessary to reach the reaction temperature of 1800ºC.

Table 4 – Parameters calculated in the iterative process for a diffuse flame

Calculated values (1 sector)		
Parameter	**Value**	**Unit**
Primary air mass flow	1.17	kg/s
Secondary air mass flow	3.12	kg/s
Fuel mass flow	0.07	kg/s
Flame adiabatic temperature	1928	K

The final result of the design of the injector for the diffuse flame can be seen in Figure 5. It is possible to see that for this case the fuel injector is aligned with the air injector in the combustion area. This alignment allows the fuel and the air to mix only in the primary zone, characterizing the diffuse regime.

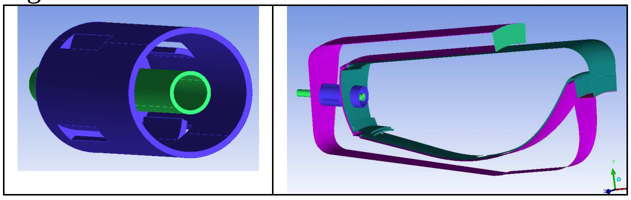

Figure 5 – Isolated assembly of the air and fuels injector and in the area that will be analyzed, diffuse flame (natural gas)

4.3 Chamber with biogas pre-mixed flame

Biogas is a fuel gaseous mix that results from the anaerobic fermentation of organic matter. The proportion of each gas depends on some parameters such as the typo of biodigestor and the matter that will be digested. Anyway, biogas is essentially composed of methane (CH4) and carbon dioxide (CO2) and its calorific value is directly linked to the amount of methane in the mixture. Biogas can came from liquid or solid residue from rural, urban or industrial area (CENBIO, 2004).

The composition of the biogas used in this study is the composition of the biogas that is present in the sewage treating station (ETE) of the sanitation company of the state of São Paulo (SABESP) in the city of Barueri. The composition is shown in Table 1.

The geometry of the combustion chamber that was used for the simulations with biogas for pre-mixed flame was the same one used for simulation with natural gas with pre-mixed flame.

The biogas mass flow used for this type of combustion is 0.16 kg/s, according to the results attained by simulation with GateCycle GE Enter Software 600 kW simple cycle one shaft gas turbine engine.

4.4 Chamber with biogas diffuse flame

In the same way as for natural gas, the same composition of the biogas of the pre-mixture was used for the simulation of the diffuse flame, aiming at verifying the behavior of another fuel with a different calorific value at a geometry that was not designed for its use.

The geometry of the combustion chamber used for the CFD simulations for biogas diffuse flame was the same for natural gas diffuse flame.

5 CFX Simulation

Modeling of gas turbine combustion chambers involves frequent configuration change. A modern

CFD tool must have the flexibility and robustness to quickly evaluate a many of geometric and operating cycle changes. CFD solutions of modified geometry must be obtained very quickly in order to be of value in a typical, schedule driven combustion chamber development program.

Gas turbine combustion involves a number of complex, closely coupled physical and chemical processes, including the three-dimensional dynamics of evaporating fuel sprays interacting with flowing multi-component gases undergoing turbulent mixing, chemical reactions, heat/mass transfer, and pollutant formation and oxidation.

As CFD becomes mature enough to predict reliable and accurate combustion system flow processes, attention should be focussed on how to deploy CFD as a design process with sufficient speed to drive down the design cycle time.

According to Lefebvre (1983), more than 80% of the amount of heat transferred in combustion chambers, where the fuel is liquid or gaseous occurs through radiation. This study, however, does not consider the presence of sooth, i.e., the flow is adopted as transparent for radiation. Supposing that the walls reflect all the radiation that is transmitted by the fluid, and are not influenced by the combustion process, the heat transfer model initially adopted is the Discreet Transfer Model (DTM). This model is appropriate, given that the radiation interferes in the flow through the heating of the walls, once it is considered transparent. The validation for the pre-mixed combustion model was not found in the literature. That is the reason why the combustion models of the CFX library, appropriate for this type of flame, were used.

1)Geometry

The geometry of the combustion chambers used for all of the CFX simulations for the 600kW gas turbine engine, both for natural gas and biogas, was the same, except for the fuel and air injectors, which were separately for pre-mixed and diffuse flames.

2)Grid

For the discretization of the combustion chamber that will be studied, a non-structured grid with tetrahedral elements was used.

3)Turbulence model

The turbulence model that was selected for the simulations was the SST (Shear Stress Tensor) due to the geometric complexity of the chamber that will be studied, which is full of protuberances and holes, as well as the validation of this model carried out by Alencar (2005)

4)Combustion model

The selection of the appropriate combustion model has a significant influence on the results attained by the simulation. That is the reason why the selection of the models must be carried out considerably carefully.

The combustion models used for this study were selected due to the flame regime present in the simulation such as:

-BVM (Burning Velocity Model), appropriate for pre-mixed or partially pre-mixed flame;

-FM (Flamelet Model), appropriate for diffuse flame (not pre-mixed).

The two aforementioned models were used for the different types of fuel that were studied.

6 Results Analysis

In all of the CFD simulations analyzed by this study, 1,000 iterations with 1,105,983 tetrahedral elements for the combustion chamber with pre-mixed flame and 661,989 tetragonal elements for the combustion chamber with diffuse flame were carried out. The processing time, using 40 cores, was approximately 9 hours for all of the simulations.

For all of the analyses carried out in the combustion chambers, one plan, based on which the velocity, temperature, pressure and NO and CO concentration profiles were generated. This plan is shown in Figure 6.

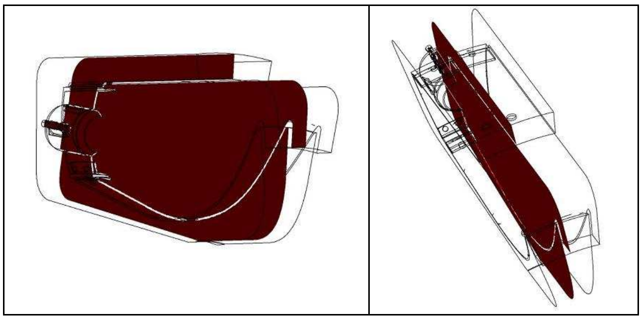

Figure 6 – Analyzed longitudinal plans

6.1 Chamber with natural gas pre-mixed flame

In the combustion chamber with pre-mixed flame, the injector was designed aiming at the air and fuel pre-mixture. The Swirl created for this fact to occur is shown in Figure 7.

The analysis of the temperature profiles is significant important to studies regarding combustion chambers, for based on them it is possible to verify the position and temperature of the flame, the temperature close to the walls, the temperature at the turbine inlet and the formation of CO and NO. The temperature at the turbine inlet must be close to 850ºC, according to what was defined in the operation and design parameters of the turbine, Table 1.

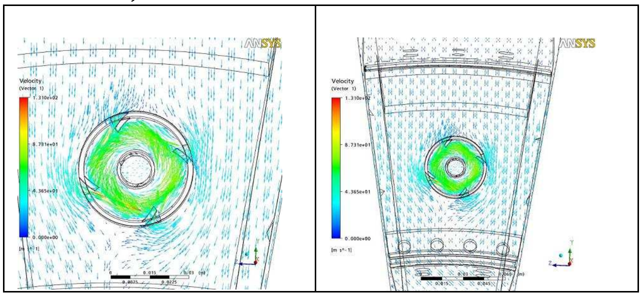

Figure 7 – Circulation details in the pre-mixed zone

Figure 8 shows the temperature profile along length of center plan. The used fuel mass flow was 0.06 kg/s. It is possible to observe through these plans that the flame is anchored in the primary zone. However, it is possible to see an outlet temperature of approximately 950 K, i.e., the value of the temperature is lower than expected, which would be 1123 K (850ºC), Table 1. This happened due to the insufficient amount of fuel used for this analyzed case.

The flow velocity for this simulation is 29.88 m/s, whereas the velocity of the flame is 13.32 m/s. This velocity difference associated with the amount of fuel makes the flame concentrate in the primary zone, avoiding flame flash back.

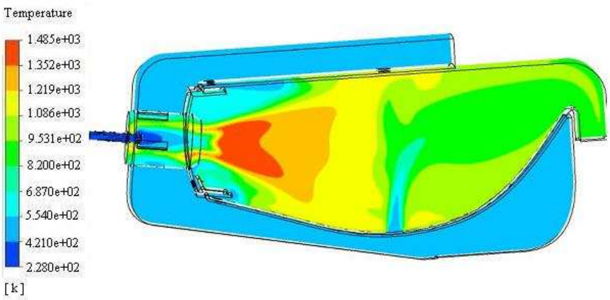

Figure 8 – Distribution of temperature in center plan

According to the analysis, the average temperature at the combustion chamber outlet is 937 K, a value that is lower than the calculated one in the cycle analysis (1123K) to maintain the performance of the engine. To solve this problem, a modification in the amount of fuel mass flow supplied to the combustion process would be necessary.

6.2 Chamber with natural gas diffuse flame

Figure 9 presents the temperature distribution profile of the studied longitudinal plan. It is possible to observe that the high temperature region is considerably larger than the one attained for the pre-mixed flame using the same amount of fuel.

The flow velocity for this simulation was 23,23 m/s, whereas the flame velocity was 4,42 m/s.

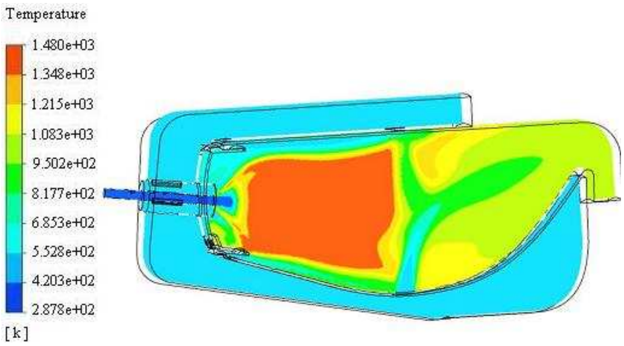

Figure 9 – Temperature distribution in center plan

6.3 Chamber with biogas gas pre-mixed flame

In order to use biogas as a fuel, the mass was changed to 0.16 kg/s because of the difference in the low heating value between biogas and natural gas.

The temperature distribution along longitudinal center plan, Figure 10, shows a larger flame, whose high temperatures reach the walls of the tube of the flame. One can observe that the amount of air destined to the dilution and cooling of the hot gases that are products of the combustion is not sufficient, making the flame spread to the region of the chamber outlet.

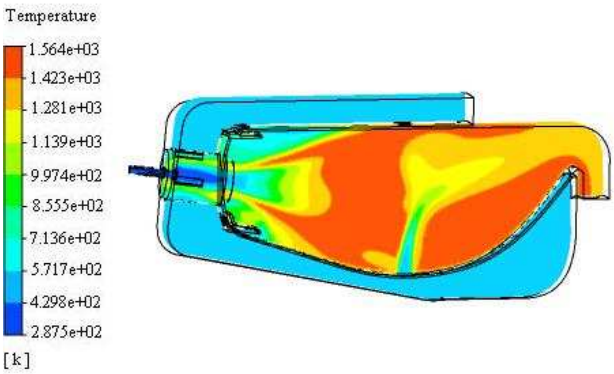

Figure 10 – Temperature distribution on center plan

This behavior of the flame is highly influenced by the fuel additional mass, which increases the injection velocity of the fuel inside the tube of the flame. Through the distribution of the temperature, Figure 10, it is possible to see the existence of a flame at the turbine inlet, which indicates that the combustion chamber will have to suffer some changes in its geometry, in order not to compromise the material of the turbine blades, as well as the material of the combustion chamber as a whole.

The flow velocity at the pre-mixture chamber is 30.21 m/s and the velocity of the flame is 9.26 m/s, which justifies the flame to extend towards the turbine inlet.

6.4 Chamber with biogas diffuse flame

Through Figure 11, which shows the temperature distribution along longitudinal center plan, it is possible to observe that when biogas is used as fuel, the shape of the flame changed significantly. When natural gas was used, the flame was anchored in the primary zone, which does not happen with biogas. In this case, according to Figure 11, the flame has a longer length inside the tube of the flame, extending itself as far as the combustion chamber outlet. This behavior takes place due to the large amount of biogas that is injected. Another point that should be analyzed deals with the high temperatures in the walls of the tube of the flame, both in the upper and lower parts. As there are high temperatures close to the turbine outlet, the possibility of damages to the material of the blades of the turbine exists, as well as the formation of non-recombined polluting elements inside the chamber.

As the velocity of the flame is significantly lower than the flow velocity, there is a high amount of non-burnt fuel at the turbine inlet.

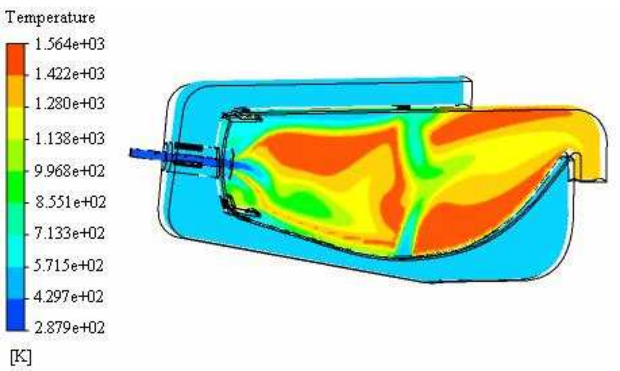

Figure 11 Temperature distribution in center plan

6 Modifications in the Geometries of the Simulated Chambers

Aiming at attaining a configuration that meet the needs of temperature in the chamber outlet of 850ºC and a minimum level of CO and NO emissions, some small modifications were carried out in the combustion chamber, particularly in the fuel injector due to the change in the mass flow input of the used fuel.

In order to reach these goals, several simulations were carried out for the combustion chamber with a pre-mixed and a diffuse flame with natural gas and biogas. The new mass flow to satisfy the aforementioned conditions is 0.1 kg/s for the two used fuels and for the two flame regimes. Aiming at reducing the speed of the fuel injection, the Mach number at the injector inlet was also changed from 0.15 to 0.1. With this change in the fuel mass, the injectors were once again designed for the two flame regimes that were studied. For the natural gas the mass flow was increased and it was reduced for the biogas. Due to this change in the fuel mass flow the generated energy for both machines will also be changed.

7.1 Modified chamber with natural gas pre-mixed flame

The diameter of the fuel injector was modified, given that the fuel mass flow that was necessary to reach the temperature of 850ºC was increased. With the rise in the fuel mass flow, maintaining the geometry of the injector constant would increase the flow velocity, making the flame extend towards the combustion chamber outlet. This way, there are regions of high temperature closer to the combustor outlet, but the average value of the temperature at the turbine inlet is high, and so is the value of the emission of pollutants, for there is not sufficient space and time for the NO and CO formed in the region of the chamber to recombine, forming non-polluting elements.

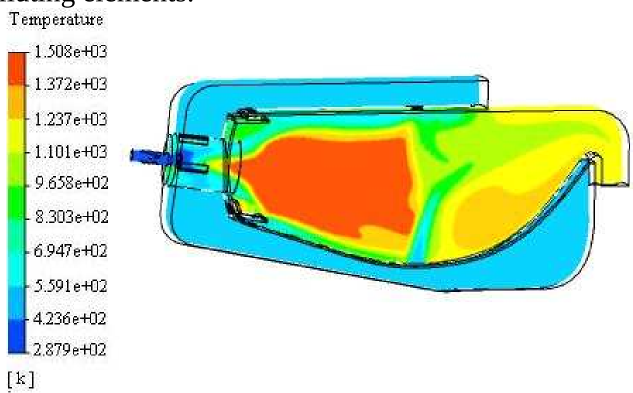

Figure 12 Temperature distribution along the longitudinal plan for the modified combustion chamber with pre-mixed flame

The results of the temperature distributions in the studied plans for the new geometry, using natural gas as fuel, can be seen in Figure 12. It is possible to see that the larger anchorage of the flame in primary region, as well as some regions close to walls with temperatures of approximately 1200 K, for now, the flow velocity is slower. The flow velocity at the pre-mixed chamber outlet is 15.85 m/s and the flame velocity is 6.21 m/s.

Figure 13 shows the temperature and NO and CO emission profiles in the outlet plan of the combustion chamber with its modified geometry. It is possible to observe that there is a temperature gradient in the combustor outlet plan, and its average value in this plan equals 852ºC. For the analysis of the NO emission in this plan, there is an average value of 0.215 ppm and a maximum value of 0.344 ppm. For the emissions of CO, the values are 16.6 ppm and 94.8 ppm, respectively.

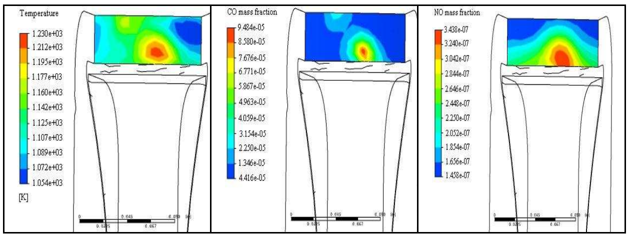

Figure 13 Temperature, NO and CO distribution in the outlet plan of the modified combustion chamber with pre-mixed flame

7.2 Modified chamber with natural gas diffuse flame

For the new geometry of the combustion chamber with diffuse flame, Figure 14 shows an anchorage of flame in the primary region. However, there is a high temperature region after the dilution holes that may compromise the emission of pollutants and the average temperature at the combustor outlet. Another point that must be taken into account is the endangerment o the walls of the combustion chamber, a consequence of the high temperatures close to the material of the combustion chamber.

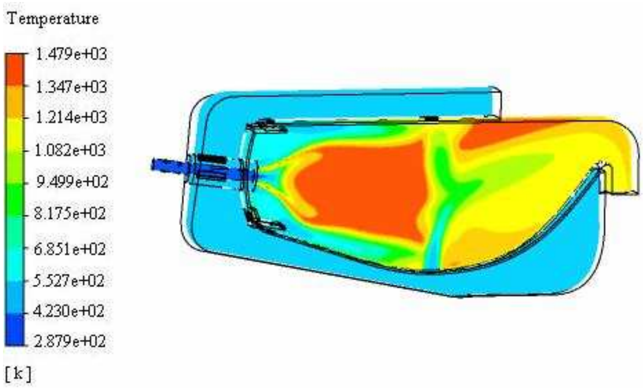

Figure 14 – Distribution of the temperature along the longitudinal plan for the modified combustion chamber with diffuse flame

The flow velocity is 17.58 m/s and the flame velocity is 6.20 m/s, values that provide a flame that is more concentrated in the primary area.

Through the temperature and CO and NO emission profiles at the outlet of the combustion chamber, Figure 15, it is possible to analyze the whether the goal of the geometry change was achieved or not. For an average temperature at the outlet of the combustion chamber the value was 841ºC. The value close to the desired one is 850ºC. For the emission analysis, there is an NO average emission value of 0.359 ppm and the maximum value released in that region was 0.599 ppm. For the CO emission the average value is 23.8 ppm.

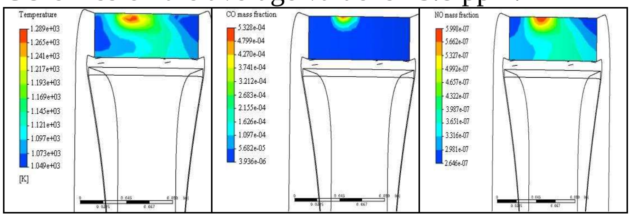

Figure 15 Temperature, NO and CO distribution at the outlet plan of the modified combustion chamber with diffuse flame

7.3 Modified chamber with biogas pre-mixed flame

With the modification of the geometry of the combustion chamber with pre-mixed flame, Figure 16 shows a more anchored flame, but it is closer to the injector due to the reduction in the flow velocity, which now is 15.86 m/s, and in the flame, which became 6.37 m/s. This location of the flame increases the probability of finding a smaller temperature gradient at the outlet of the chamber, as well as a smaller value of released pollutants.

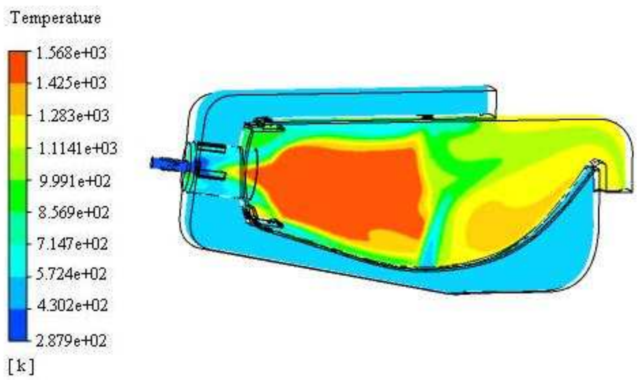

Figure 16 – Temperature distribution along the longitudinal plan for the modified combustion chamber with pre-mixed flame

Bit can be noticed in Figure 17, that for the outlet of this modified combustion chamber the average temperature is 867ºC, i.e., the value is in accordance with the desired one. However, it is necessary to observe the temperature gradient that is present in this region. For the polluting emission values, the average value for NO is 0.438 ppm and the maximum value released in this region is 0.709. The average emission of CO is 17.6 ppm.

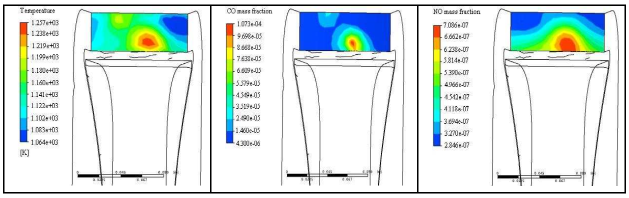

Figure 17 Temperature, NO and CO distribution at the outlet plan of the modified combustion chamber with pre-mixed flame

7.2 Modified chamber with biogas diffuse flame

Figure 18 shows that the flame is mostly located in the primary region, a similar behavior to the biogas diffuse flame. However, the temperature is higher close to the walls that are located after the dilution holes, which may compromise the temperature and the emissions at the combustor outlet.

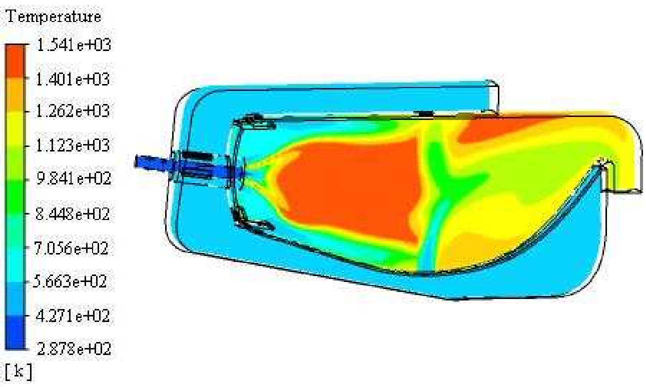

Figure 18 Temperature distribution along the longitudinal plan of the modified combustion chamber with diffuse chamber

As the flow velocity is now 17.53 m/s and the flame velocity is 5.64 m/s, the flame gets more concentrated in the primary zone, making the temperature at the outlet be lower, and so is the emission of pollutants.

Figure 19 shows the temperature gradient at the combustor outlet plan. The average temperature at this plan is 857º C. The average amount of NO released at the outlet plan is 0.813 ppm. The maximum released value in this region is 1.37 ppm. For the CO analysis a released average value of 24.2 ppm was found.

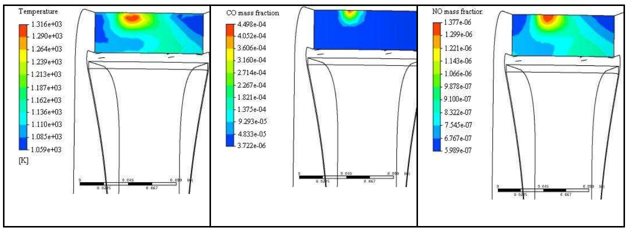

Figure 19 Temperature, NO and CO distribution at he outlet plan of the modified combustion chamber

6 Conclusions

The CFD numerical simulation enabled the execution of changes in the geometry of the combustion chamber in order to adjust the pressure losses and to improve the dilution of the gases that are products of the combustion process.

The data regarding the distribution of primary air masses and dilution are of the most importance for the design of the air injectors, which is an important ally regarding the behavior of the flame and the formation of pollutants.

For the new combustion chamber, it was necessary to develop a new methodology to design the air and fuel injectors for the pre-mixed and non-pre-mixed flame for both fuels that were used.

The analyses were carried out for pre-mixed and diffuse chamber, using natural gas and biogas as fuels. Through the numerical simulations it was possible to notice that:

- for diffuse combustion chambers, the temperature and the emission of CO and NO at the outlet is higher that for the pre-mixed chamber, as it was expected;
- the change in the fuel from natural gas to biogas changes the aerodynamic and thermal behavior significantly due to the variation of the fuel mass flow, which is higher for the biogas, given that it has a lower heating value that the natural gas and different chemical composition as well. This change in the amount of fuel alters the energy generated by the analyzed engine;
- The residence time for biogas is higher than natural gas fuel, this leads to modification in the primary zone, increasing length and width, for biogas fuel;
- The length flame is inversely proportional to the stoichiometric fuel mass fraction. This implies that fuels that require less air for complete combustion produce shorter flames. In the case analysed in this paper, the natural gas is 16 percent smaller than the biogas.
- The mass flow provided by the Gatecycle GE Enter Software was based on simplified combustion models, assessed at stoichiometric conditions, aiming at reaching a temperature of 1123 K at the

turbine inlet. However, the CFD simulation showed that the fuel mass flows provided by the Gatecycle GE Enter Software were not enough for such temperature to be reached at the turbine inlet. This way, it was necessary to change the fuel mass flows. For the natural gas, a rise by 42% was needed and for the biogas a reduction of 60% was necessary, implying in alterations in the amount of energy generated by this equipment.

- For these cases analysed the conditions were not stoichiometric, so, the combustion efficiency with the natural gas was 70% and for the biogas 60%. Its show that modifications were necessary in the engine cycle simulation for a good performance.

- The adjustment of the geometry of the fuel injector allowed the adjustment of the flow velocity with the flame velocity, making the flame stable in the primary zone. Low flame velocities associated with high flow velocities make the flame extend towards the turbine inlet, and consequently there are higher temperatures and NO and CO emissions, which are not desired.

Through the presented results, it is possible to notice the need of geometric changes in the combustion chamber designs when the fuel is switched. The results clearly show the consequences of using no design fuel in combustion chambers that were not designed to use it.

6 Acknowledgements

The authors wishes to thank the CAPES and PETROBRÁS, for the financial support which made investigation possible. The authors also please the Jim Kesseli for the attention excused to this research.

References:

[1] ALENCAR H.S., NASCIMENTO M.A, VILLANOVA H., (2007), “Estudo Numérico termo-aerodinâmico para Câmaras de Combustão para Turbinas a Gás: Aplicação ao Caso da Micro turbina”, Tese de doutorado, UNIFEI.

[2] ALLEN J.W., (1998), “Low Nox Burner Design”, Proceeding of the American Power Conference, Vol.60-II, pp. 869-874.

[3] ANDREINI A., FACCINI B., MANGANI L., COCCHI S., MODI R., (2006), “Fuel Flexibility Test Campaign on a GE10 Gas Turbine: Experimental and Numerical Results”, Proceeding of ASME Turbo Expo, Power and Land, Sea and Air, May 8-13, GT2006-90510, Barcelona, Spain.

[4] CAMERETTI M.C., TUCCILO R., (2004), “Comparing Different Solutions for the Micro Gas Turbine Combustor”, Proceeding of ASME Turbo Expo, Power for Land, Sea and Air, June 14-17,GT2004-53286, Vienna, Austria.

[5] CENBIO, (2004), “Relatórios de Atividades – Projeto ENERG-BIOG”, São Paulo.

[6] DARBYSHIRE O. R., EVANS A., WILSON C. W., BECK S. B. M., (2006), “CFD-Based Analysis of Burner Fuel Air Mixing Over Range of Air Inlet and Fuel Pre-heat Temperatures for Siemens V94.3A Gas”, Proceeding of ASME Turbo Expo, Power and Land, Sea and Air, May 8-13, GT2006-90944, Barcelona, Spain.

[7] GOSSELIN P., DECHAMPLAIN S.K., KRETSCHMER D., (2000), “Three Dimensional CFD Analysis of a Gas Turbine Combustion Chamber”, 36th AIAA/ASME/SAE/ASEE Joint Propulsion Conference an Exhibit, pp11, Huntsville, Alabama.

[8] ISRAEL INSTITUTE OF TECHNOLOGY (2003), “FLOXCOM – Low NOx Flameless Oxidation Combustor for High Efficiency Gas Turbines”, Project n° NNES-1999-20246, Israel, p. 156.

[9] KUO K.K., (1986), “Principles of Combustion”, John Wiley & Sons Edition, New York, EUA.

[10] KUTSENKO Y.G., ONEGIN S.F., (2006), “Development and Application of CFD-Based Analysis Methodology to Evaluate Efficiency of Low Nox Combustion Technologies”, Proceeding of ASME Turbo Expo, Power for Land, Sea and Air, May 8-11, GT2006-90530, Barcelona, Spain.

[11] LAI M. K., (1997), “CFD Analysis of liquid Spray Combustion in a Gas Turbine Combustor”, ASME, GT1997-309.

[12] LEFEBVRE A. H. (1983), “Gas Turbine Combustion”, 1ª Edition, Mc-GrawHill Book Company, New York , USA.

[13] LEFEBVRE A. H. e REID R, (1966), “The Influence of Turbulence on The Structure and Propagation of Enclosed Flames”, Combustion and Flame, Volume 10, Issue 4, pp. 355-366.

[14]]NETO L. J. M. (2003), “Projeto Preliminar de uma Câmara de Combustão de Microturbina a Gás operando com Gás Natural e sua Modificação para Gás de Baixo Poder Calorífico”, Dissertação de mestrado, UNIFEI.

[15] RIZK N. K. e MONGLIA H. C. (1991), “Three dimensional Analysis of Gas Turbine

Combustor", Journal of Propulsion and Power, Vol. 7, N° 1.

[16] TURREL M. D., STOPFORD P.J., SYED, K. J., BUCHANAN E., 2004, "CFD Simulation of the Flow Within and Downstream of a High-Swirl Lean Premixed Gas Turbine Combustor", Proceeding of ASME Turbo Expo, Power for Land, Sea and Air, June 14-17, GT2004-53112, Vienna, Austria.

[17] WUNNING J. G. (1996), "Flamlose Oxidation von Brennstoff", PhD thesis, RWTH.