DESIGN AND THERMODYNAMIC ASSESSMENT OF A TUBULAR COMBUSTION CHAMBER BASED ON THE FUEL COMPOSITION

Fagner Luís Goulart Dias, fagner@unifei.edu.br
Marco Antonio Rosa do Nascimento, marcoantonio@unifei.edu.br
Lucilene de Oliveira Rodrigues, lucilener@unifei.edu.br
Federal University of Itajubá – UNIFEI, Av. BPS, 1303, Pinheirinho, Itajubá-MG

Abstract. *Numerical studies on combustion chambers resulted in more effective projects, that is, compact and less pollutants, primarily to assist in the conception stage. This ensures a significant reduction in design time and costs. Many of the technologies and digital techniques are frequently used for modeling, emphasizing the computational fluid dynamics (CFD). However, the application of these techniques requires an initial process of modeling and simulation to define, properly, the geometry and initial mesh. In this sense, the purpose of this work is to present a calculation program to automate the preliminary design of a tubular combustion chamber based on the fuel composition. Thus, according to input data and the fuel composition, the main elements of the combustion chamber were designed. Finally, the evaluation of thermo-aerodynamics of the combustion chamber was performed using CFD techniques, in order to compare the results and identify possible improvements in the proposed program.*

Keywords*: gas turbine, can-combustion, combustion chamber, CFD simulations.*

1. INTRODUCTION

The use of gas turbines for power generation, compared with other forms, has increased in recent years. Due to global policies to reduce emissions of pollutants, in association with the context of exhaustion of fossil fuels, research in gas turbines operating in a wide range of fuels has been extensively studied in several research centers in the world.

The combustion chambers are an essential component of the thermodynamic cycle gas, next to the compressor/turbine. Basically, the combustion chamber is intended to promote, effectively, a mixture of an oxidizing agent, previously compressed by the compressor and the fuel agent. Thus, the purpose is a combustion reaction which remains stable and constant throughout the chamber, releasing sufficient heat to raise the gas temperature at the desired level of operation of the turbine. However, this must be done with a minimum pressure loss and maximum efficiency, throughout the operating range of the gas turbine. Finally, the amount of fuel added is due mainly to increase the power required and the desired temperature, the latter being limited by the maximum temperature that the material of the rotor blades and the turbine supports (Lora and Nascimento, 2004).

Authors such as Cohen *et al.* (1987), emphasize that the geometry of the combustion chamber along with the injection system of air and fuel, are designed to promote the occurrence of the following procedures in chronological order:

- To develop a reactive mixture;
- To provide the ignition of the mixture;
- To promote the spread of the flame front;
- To guarantee the mixture of flue gases with excess air to uniform the temperature of the gases that are directed to the turbine.

As seen, there are many requirements that a good design of the combustion chamber must be taken into account. Currently, there are efforts of research institutes and enterprises for projects called "flexible", *i.e.*, allowing the transition to alternative fuels, or simply to allow for variations in fuel composition. Furthermore, this trend has a strong relationship with current environmental laws, which are more restricted and supply of other fuels at an attractive price.

However, the substitution of fossil fuels with low calorific fuels requires significant changes, particularly in the combustion chamber, in terms of geometry, efficiency and emissions. Thus, in applications of gas turbines, changes are necessary and required to attend the various features of the composition and calorific value of fuels, especially in off-design conditions. Depending on the type of fuel, these changes will soon application materials more corrosion resistant to physical changes (to ensure the combustion process) in components of gas turbines (Hung, 1989) (Molière, 2002).

Thus, given the complexity around the combustion chamber, it becomes necessary to develop a calculation procedure for a preliminary design of combustion chamber, which through changes in the composition of fuel used, be possible to determine the necessary changes the chamber of combustion and to examine the feasibility of fuel's replacement.

Bohn and Lepers (2003) analyzed the effects of the burning of biogas in the operating characteristics of microturbines. The authors also showed the major fuels of low calorific value with potential for application in gas microturbines. Finally, they studied the effects of some fuels of low calorific value in the materials and emissions.

Among the fuels with low calorific value, Bonzano and Pollarolo (2004) have intensified their research in the synthesis gas. The authors indicate that modifications in the injection system and combustion chamber should be performed for each composition of fuel burned, since higher speeds are required. Therefore, it requires the use of special control valves, and adding if necessary, a mixing system for recovering gas and steam injection.

Several studies, practice tests and numerical simulations were conducted on the use of biofuels or, more generally, the low calorific fuels in gas turbines for power generation. Lefebvre (1998) and Mellor (1990) have presented an overview of the elements of a biomass combustion chamber and the main equations that help a preliminary design. But there is still no method for calculating the combustion chamber of gas turbines, well defined and allow estimating the impacts associated with replacement of fuel project.

In this sense, the objective of this work is to create a tool to aid the preliminary design of a combustion chamber, which according to the input data and the fuel composition, can provide the main dimensions and design parameters. Finally, an analysis of the effects in the thermo-aerodynamic design must be analyzed using CFD techniques, in order to compare the results and identify possible improvements in the proposed program.

2. INPUT DATA

To start the calculations some parameters are needed. Then, the input data, summarized in Tab. 1, must be inserted in the Excel spreadsheet. In the same tab, it will be allowed to select the user type of fuel to burn some fuel already stored, as well as others that may be entered by selecting the desired composition.

Table 1. Input data required to start the simulations.

Parameters	Units
Mass flow rate of fuel	kg/s
Mass flow rate of air	kg/s
Fuel Inlet Temperature	K
Ambient Temperature	K
Ambient Pressure	bar
Inlet Temperature	K
Inlet Pressure	bar
Outlet Temperature	K
Outlet Pressure	bar

The calculations used in the combustion process and the determination of temperature distribution in the combustion chamber have employed the thermodynamic data base, called NIST Standard Reference Database (NIST Chemistry WebBook), available by the U.S. government in http://webbook.nist.gov/chemistry/ (accessed on 14.10.2010). In addition, the NIST base meets many authors and aims to standardize the values to be used. From the base NIST (National Institute of Standards and Technology) were obtained from the characteristic equation and the coefficients for the calculation of specific heat, enthalpy, and some physical properties of compounds.

3. THE PRELIMINARY DESIGN OF THE COMBUSTION CHAMBER DIMENSIONS

In order to initiate the calculations of the geometry of the combustion chamber, some parameters are needed, as described below:

3.1. Parameters of reference

At the beginning of design of a combustion chamber, some quantities must be defined based on "design point", according to the requirements of air mass flow, temperature, pressure loss and the reference area. Such quantities, according to Lefebvre (1998), are used to analyze and comparison with other arrangements of the combustion chambers, since they are linked to flow characteristics such as speed and Mach number, among others. The main quantities used are:

$$U_{ref} = \frac{\dot{m}}{\rho A_{ref}} \tag{1}$$

$$Ma_{ref} = \frac{U_{ref}}{(\gamma R T_3)^{0,5}} \tag{2}$$

$$q_{ref} = \frac{\rho U_{ref}^2}{2} \tag{3}$$

$$\bar{u}_{ref} = \frac{R_{ar}\dot{m}_3 T_3}{A_{ref} P_3} \tag{4}$$

$$\rho_3 = \frac{P_3}{R_{ar} T_3} \tag{5}$$

$$q_{ref} = \frac{\rho_3}{2}(\bar{u}_{ref})^2 \tag{6}$$

From these quantities, the characteristics of compressible flow to the combustion chamber are given on the following parameters and can be used like a comparison form: $\frac{U_{ref}}{\sqrt{T_3}}$, $\frac{\dot{m}\sqrt{T_3}}{P_3 A_{ref}}$ and $\frac{\dot{m}\sqrt{T_3}}{p_3 A_{ref}}$.

where,
P_3=Inlet total pressure
p_3= Inlet static pressure

3.2. Reference area

The reference area represents one of the most important physical parameters in a design of a combustion chamber, creating great impact on other dimensions of the chamber. According to Lefebvre (1998), this parameter must be calculated by taking a compromise between the limitations given by the chemical reactions and aerodynamic, and the loss of maximum allowable pressure in the combustion chamber. Considering the two above possibilities (aerodynamics and chemical reactions), the calculations are made considering both cases, and then, will choose the area of greatest value. Finally, the reference diameter for tubular combustion chambers can be calculated as follows:

$$A_{ref} = \frac{\pi {D_{ref}}^2}{4} \tag{7}$$

3.2.1 Approach based on aerodynamics

From this approach, calculations concerning the main dimensions of the combustion chamber will be due mainly to loss. The three main quantities that involve the loss of pressure can be related by the Eq. (8) (Lefebvre, 1998):

$$\frac{\Delta P_{3\to 4}}{P_3} = \frac{\Delta P_{3\to 4}}{q_{ref}} \frac{\gamma}{2} {Ma_{ref}}^2 \tag{8}$$

Or, replacing Ma_{ref} and doing some manipulations in the Eq. (8), we have:

$$\frac{\Delta P_{3\to 4}}{P_3} = \frac{R_{ar}}{2} \frac{\Delta P_{3\to 4}}{q_{ref}} \left(\frac{\dot{m}\sqrt{T_3}}{P_3 A_{ref}}\right)^2 \tag{9}$$

From the Eq. 9, it was possible to get the value of the reference area, A_{ref}:

$$A_{ref} = \left[\frac{R_{ar}}{2}\left(\frac{\dot{m}_3\sqrt{T_3}}{p_3}\right)^2 \left(\frac{\Delta p_{3\to 4}/q_{ref}}{\Delta p_{3\to 4}/p_3}\right)\right]^{0,5} \tag{10}$$

The total pressure loss in the combustion chamber, obtained from the previous expression corresponds to a design value, i.e., corresponds to a value given by the simulation cycle (compressor/turbine) for a given pressure drop.

Due to the fact that this project develops a tubular chamber, where pressure losses throughout the combustion chamber are around 6-8%, was defined as an initial approximation that the pressure loss in flame tube to be around 5%. Then, these 5% were equally divided into three regions that the chamber was divided, *i.e.*, the injector until the end of the primary zone, then zone between the primary and secondary, ending with the dilution zone.

The preliminary design begins with some typical values, resulted of experiments. Lefebvre (1998) suggests some typical values, according to the type of combustion chamber (tubular, annular and annular-tube), listed in the Tab. 2:

Table 2. Typical values used in combustion chambers

Type of chamber	$\Delta P_{3\to4}/P_3$	$\Delta P_{3\to4}/q_{ref}$	$\dot{m}_3\sqrt{T_3}/(P_3A_{ref})$
Tubular	0.07	37	0.0036
Annular-tube	0.06	28	0.0039
Annular	0.06	20	0.0046

In the Tab. 2, among these parameters, the loss of pressure in the chamber, $\Delta p_{3\to4}/p_3$, is a design parameter and therefore, fixed. The value of loss factor of pressure, $\Delta P_{3\to4}/q_{ref}$ is a function of the "pattern factor ", the emission of pollutants, the velocity of the compressor and of the diffuser's type (Lefebvre, 1998). Finally, the parameter $\dot{m}_3\sqrt{T_3}/(P_3A_{ref})$, varies according to the reference area chosen.

As until now, the detailed calculation of these parameters is not feasible, and since they are related by the Eq. (10), were used in the Excel® spreadsheet data into the combustion chamber of the tubular type contained in Tab. 2.

3.2.2 Approach based on combustion process

The determination of the appropriate reference area for chemical reactions can be obtained through the parameters, θ. According to Lefebvre (1998), to any fuel/air ratio, the combustion efficiency, η, can be related, based on the parameter θ, as follows:

$$\theta = \frac{{P_3}^{1,75}A_{ref}D_{ref}^{0,75}e^{T_3/b}}{\dot{m}_3} \tag{11}$$

For a combustion efficiency about 100%, the parameter θ has a value $73\text{x}10^6$ (SI) (Lefebvre, 1998). Finally, given the values to the reference area, A_{ref}, obtained by both approaches, the value considered will be the greatest.

3.3. A_0 area

This section can be defined assuming that the flow velocity in this section is equal to the velocity through the area, A_{an}. Based on this account, obtains the value of the area through the following expression, A_0 (Lefebvre, 1999).

$$\frac{A_0}{A_{an}} = \frac{\dot{m}_3}{\dot{m}_{an}} \tag{12}$$

$$A_0 = \frac{\dot{m}_3A_{an}}{\dot{m}_{an}} \tag{13}$$

3.3. A_s area

In most projects in the combustion chamber, an initial estimate is that about 20% of the air compressor that provides, or even, half the air needed for the primary zone, passes through this nozzle also known as "snout". Then, the air finds many routes to follow, as the vortex and the cooling holes of the torch, before reaching the primary combustion zone. According to Lefebvre (1998), the area A_s can be calculated by the Eq. (14):

$$A_s = \frac{\dot{m}_sA_0}{\dot{m}_3C_{d,s}} \tag{14}$$

Therefore, the discharge coefficient in the "snout", $C_{d,s}$, was initially considered as unit.

3.4. A_n area

The A_{an} area, simply corresponds the difference between the reference area, A_{ref}, and the area of the flame tube A_{ft}:

$$A_{an} = A_{ref} - A_{ft} \tag{15}$$

3.5. Temperature distribution along the combustor

For this calculation, the combustor was divided into four zones: a zone of recirculation, the primary zone, secondary zone and dilution zone. For each zone, the local temperature was considered as a linear variation between T_{in} and T_{out}.

3.5.1 Recirculation zone

It was considered that the minimum temperature in this region is given by the inlet temperature of the gases in the combustion chamber, T_3, and the maximum temperature that will correspond to a stoichiometric mixture, T_{esteq}. Thus, the gas temperature that will achieve the primary zone will correspond to an average between the values of T_3 and T_{esteq}, as defined by the following expressions:

$$T_{in} = T_3 \tag{16}$$

$$T_{out,max,ZR} = T_3 + \eta_{RZ} \Delta T_{\phi=1} \tag{17}$$

The parameter, η_{RZ}, represents the efficiency of combustion in this region, and can be defined by Eq. (18).

$$\eta_{ZR} = 0{,}56 + 0{,}44\, tanh[1{,}547510^{-3}(T_3 + 108\, ln(p_3) - 1863)] \tag{18}$$

where,

p_3= Inlet static pressure

According to Lefebvre (1998), because this region presents a flow agitation, this temperature value is restricted only to a specific location within this region. Thus it was assumed a temperature given by Eq. (19).

$$T_{out,ZR} = \frac{1}{3} T_3 + \frac{2}{3} T_{out,max,ZR} \tag{19}$$

Finally, faced with a finite reaction rate and a continuous addition of cold air, one can consider the temperature distribution in the recirculation region as linear, beginning with the temperature T_3 until $T_{out,ZR}$ (Lefebvre, 1998).

a) Remaining Primary zone: The temperature distribution in the rest of the primary zone follows the same principle employed in the recirculation region, as given by Eq. (19). But, the parameters ΔT_{ZP}, now represents an increase in temperature of the fluid from the entrance, T_3, until $T_{out,ZP}$, as shown by Eq. (20):

$$T_{out,ZP} = T_3 + \eta_{ZP} \Delta T_{ZP} \tag{20}$$

Where, the overall efficiency of combustion, η_{ZR}, until the end of the primary zone may be defined as:

$$\eta_{ZR} = 0{,}71 + 0{,}29 tanh[1{,}547510^{-3}(T_3 + 108\, ln(p_3) - 1863)] \tag{21}$$

The gain in the temperature, ΔT_{ZP}, in Eq. (20) corresponds to an increase of T_3 until a temperature near the stoichiometric condition, because it is a diffusion flame. Thus, the designer defines the desired excess air in the Excel spreadsheet carried out in order to obtain a temperature near the recommended amount, and through a thermo-chemical calculation, the gain desired temperature is set. Finally, from Eq. (21), the temperature of flue gases in the primary zone can be estimated.

b) Secondary zone: Due to the complexity of the combustion process, a precise determination of chemical composition and temperature of flue gases in the secondary zone are difficult to solve. Thus, the increase in temperature calculated in the previous sections it becomes an indefinite quantity, since the partial combustion reactions were not considered in this project.

In order to determine the temperature of flue gases in the secondary zone, was used same methodology used in the primary zone. As illustrated in Fig. 1, the combustion chamber was divided into sections and was applied the Laws of Conservation of Energy and Continuity. At the end, was possible to estimate the mass flow rate and temperature of gas in various parts of the combustion chamber.

c) Dilution zone: The temperature distribution in the dilution zone was defined similarly to other areas, applying the equations for mass and energy balance.

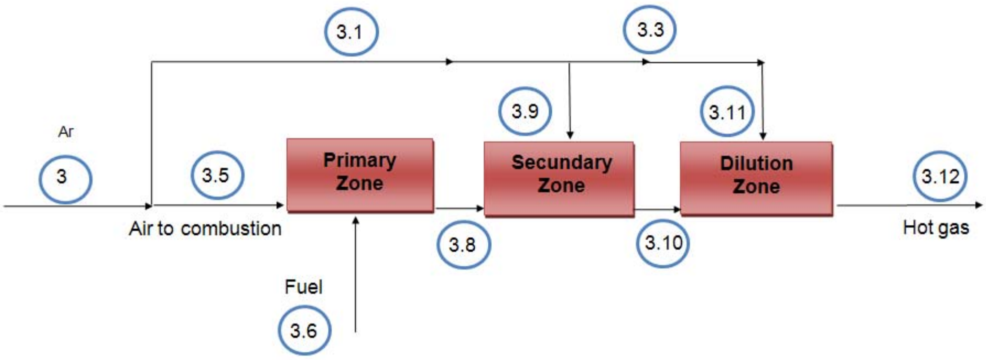

Figure 1. Scheme of division of regions in the combustion chamber.

4. THE EXCEL SPREADSHEET DESIGNED

The Excel® spreadsheet carried out applies the main calculations necessary for the preliminary design. Thus, it was created many components, where the calculations have been grouped according to some affinity established, as follows:

- Menu shortcuts
- Input Data
- Calculation of Adiabatic Flame Temperature
- Reference Values
- Areas and Distribution of Mass Flow
- Diffuser
- Swirler
- Recirculation Zone
- Primary Zone
- Secondary Zone
- Dilution Zone
- Scheme of the combustion chamber designed

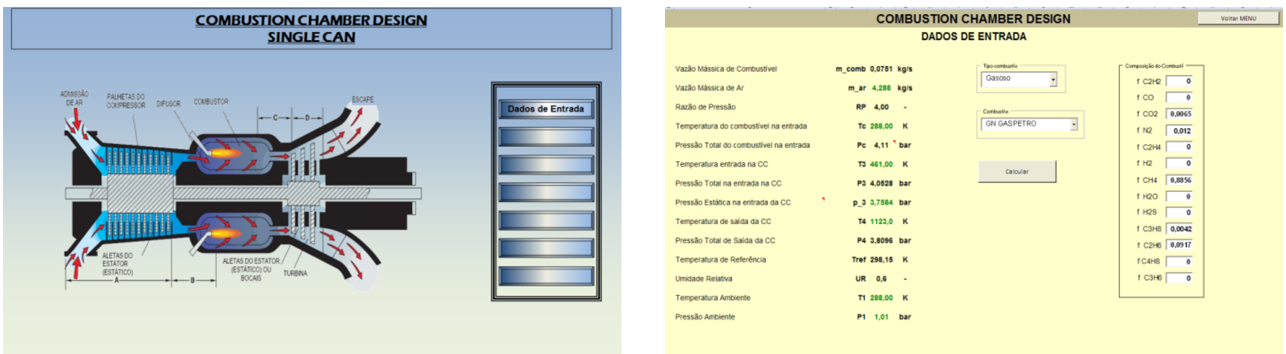

Figure 2. Screenshots of the Excel® spreadsheet developed.

5. GEOMETRY CALCULATED

From the calculations carried out by Excel® spreadsheet developed, was obtained the preliminary geometry of a tubular combustion chamber gas turbine, as illustrated in the Fig. 3.

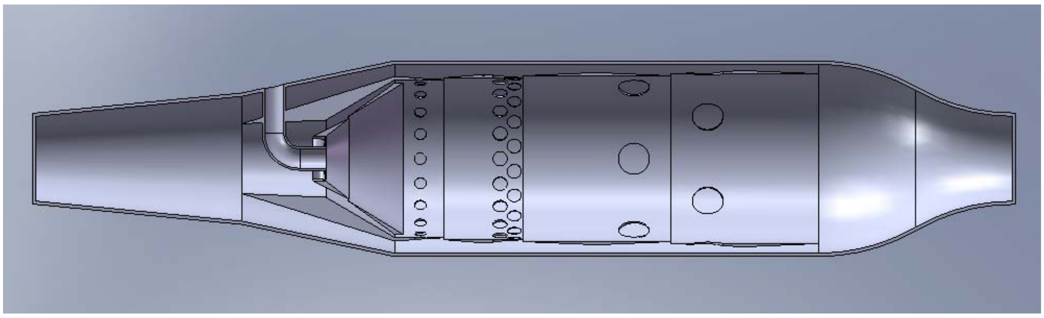

Figure 3. Geometry designed by the Excel spreadsheet developed.

6. SIMULATION IN THE GATECYCLE™

In order to obtain the thermal parameters required for the CFX simulation to the combustion chamber, the software GateCycle ™ GE Energy was used to simulate. The input and output design parameters used in the GateCycle™ for natural gas fuel, are shown in Tab. 3. The parameter values selected in Tab. 3 were based in the current technologies for radial turbomachineries. The turbine inlet temperature of 1123 K corresponds the maximum temperature supported by a typical material in a radial turbine, maintaining the mechanical resistance and the useful life, without any blade cooling (Rodrigues, 2009).

Table 3. Input and output parameters to design (Rodrigues, 2009)

Description	Values	Units
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	6	%
Combustion chamber inlet temperature	461	K
Air Mass flow rate	4.288	kg/s
Fuel Mass flow rate (natural gas)	0.075	kg/s
Air moisture content	60	%
Gas turbine power output	600	kW

The compositions of the fuels that were used are presented in Tab. 4.

Table 4. Mass fraction of natural gas used (Rodrigues, 2009)

Natural Gas	Mass fraction	
CH_4	0.8856	%
C_3H_8	0.0042	%
C_2H_6	0.0917	%
CO_2	0.0065	%

6. CFD SIMULATIONS

The combustion model used was Eddy Dissipation, along with the package of reactions Methane Air WD2 NO PDF, provided by CFX. The turbulence model used was SST, Stress Shear Transfer. Both models used in CFD simulations of this paper are similar to the conditions studied by Rodrigues (2009), who throughout her work detailed the whole process of validation.

6.1 MESH APPLIED

In the CFD simulations were used unstructured meshes of tetrahedral type. In order to obtain a well-refined mesh, were used different sizes of elements in the overall order of 0.008 m. In the swirl and dilution holes the size of 0.004 elements have me on the inside wall and the dome region of 0.006 m. Thus, resulting in well refined mesh with 3650671 elements distributed in 596231 nodes. According to the Fig. 3, details of the meshes employed are presented.

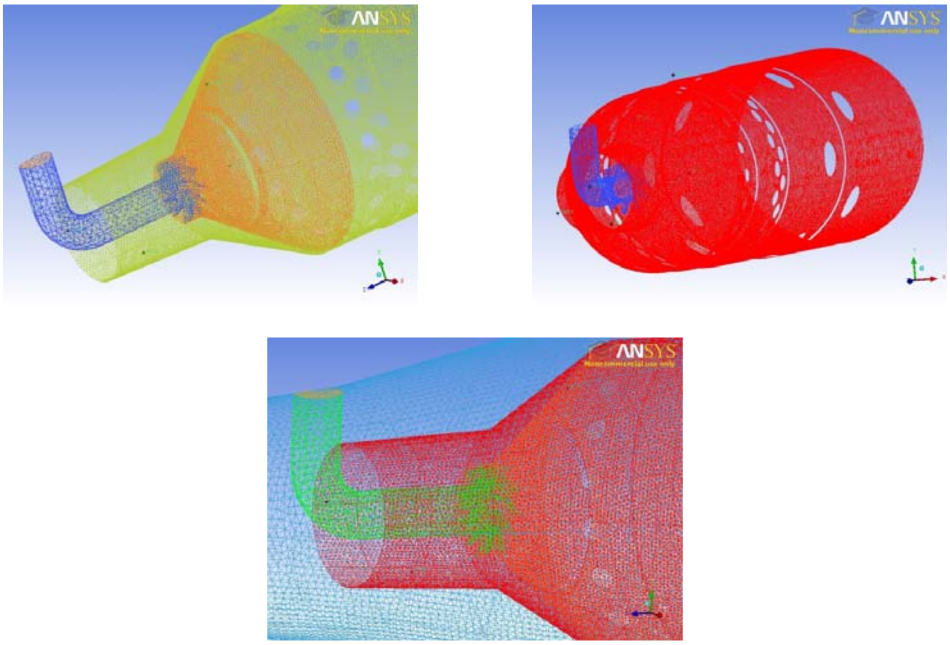

Figure 3. Mesh details

7. RESULTS AND DISCUSSION

7.1 Distribution of velocity vectors

From the Fig. 4 was described the distribution of velocity vectors along the combustion chamber designed. It can be noted interference of a large duct in the fuel injection flow. Also, the swirler created a recirculation region in front of the injector, with low speed, facilitating the anchorage of the flame in this region. Finally, can be noted the interaction created by the dilution holes in the flow. According to Rodrigues (2009), the flame tends to stabilize in the region of low velocity, and thus, from Fig. 4, it's possible to estimate the region of flame.

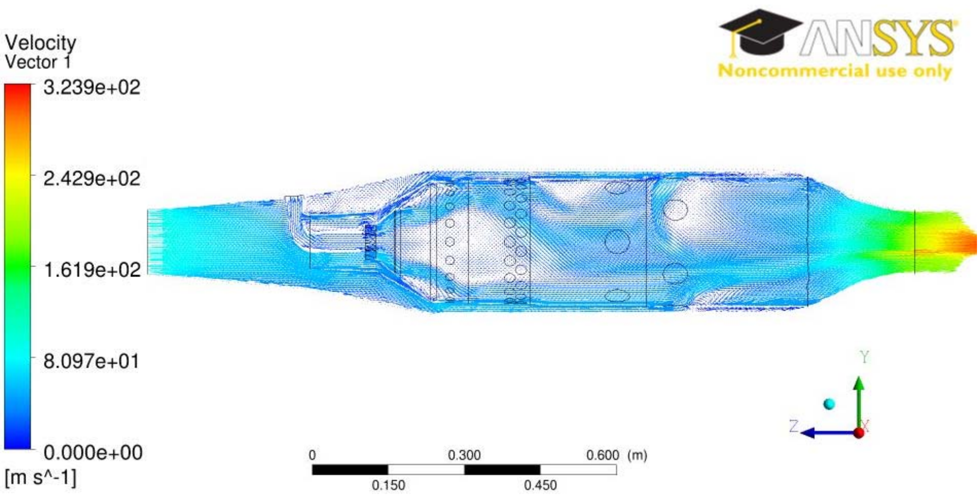

Figure 4. Distribution of velocity vectors.

7.2 Distribution of temperature

From Fig. 5, can be seen the strong tendency of the flame to concentrate near the region of low velocity. It is also observed that the primary/secondary region tends to concentrate most of the flame and that the holes distribution contributes to anchoring the flame from the central region. However, it appears that a part of the flame spreads throughout the combustion chamber.

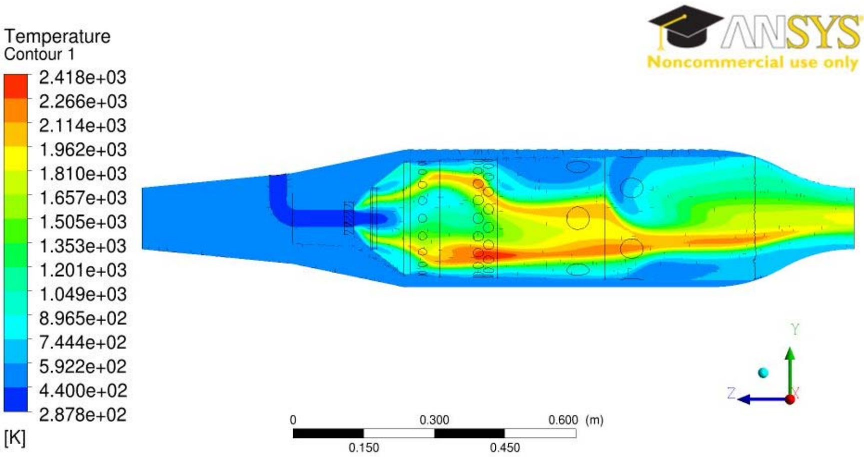

Figure 5. Distribution of temperature along the combustion chamber.

In addition, the temperature distribution in the combustion chamber, it was possible to obtain the temperature of flue gas at the exit. As mentioned in the Tab. 3, the design output temperature is 1123K, and after the CFD simulations, it was obtained an average of 1125.28K in the outlet region, calculated by the CFX program, like showed in the Fig. 6. This value is very close to what was expected, showing that the combustion chamber has been well planned. But, according to Fig. 6, it was observed a local maximum temperature of 1980K, which can damage some turbine blades.

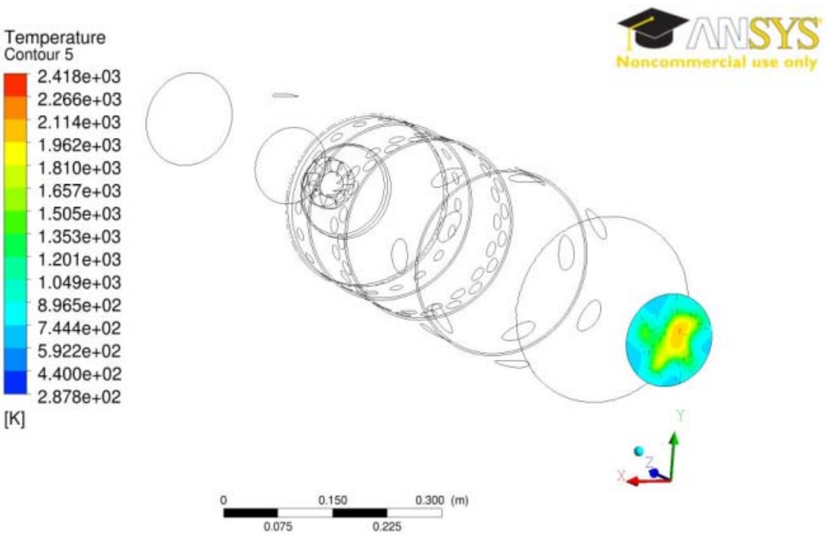

Figure 6. Detail of the temperature distribution in the outlet region.

About the radiation model, it was used the P1 Model, like suggested and validated for the same problem by Rodrigues (2009).

7.3 The swirler effect

The swirler has intended to move the streamlines of flow in order to create a little turbulence and thus reduce the flow rate, facilitating the mixing process. Through the Fig. 7 is possible to visualize this effect being created.

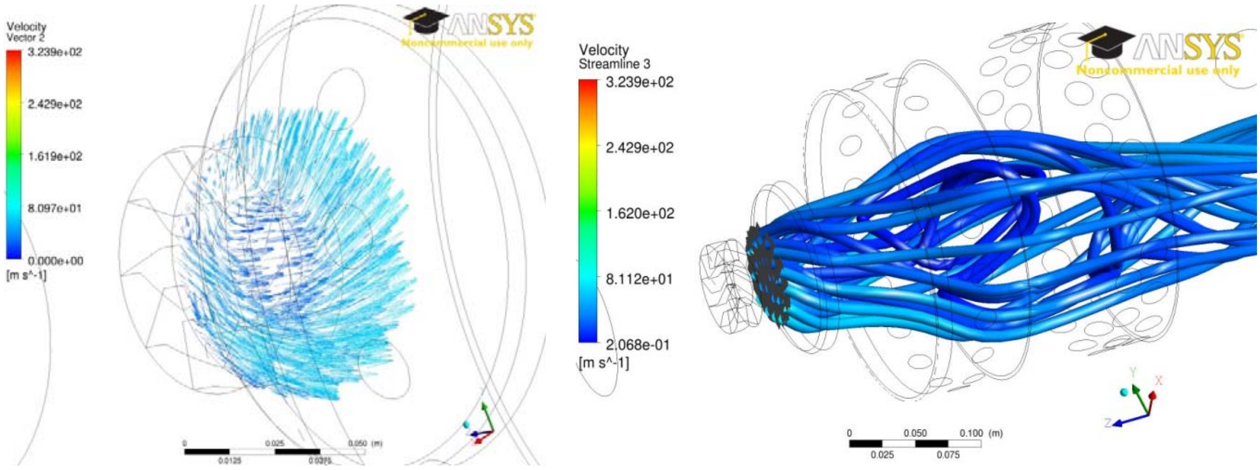

Figure 7. The swirler effect being created.

8. CONCLUSIONS

From the CFD simulations carried out, several equations and the methodology proposed by Lefebvre (1998) for the preliminary design of tubular combustion chambers may be verified. Some points that were observed by the distribution of temperature need to be reviewed, such as the number of holes and their distribution throughout the chamber. It was observed that these parameters are directly related to the stabilization of the flame in primary region and centered. Studies in the fuel injection rate must also be made with the objective of keeping the flame front closest of the nozzle.
In general, the behavior of the flame inside the chamber was adequate, requiring only some adjustments in the equations in order to consider some deficiencies noted by the CFD simulations.

9. ACKNOWLEDGEMENTS

The authors gratefully acknowledge the financial support given by funding agencies like CAPES, CNPq, FAPEMIG and CENPES-PETROBRAS, which were essentials to the development of this work. Also, we would like to acknowledge the support given by the student Thiago Oliveira, during the CFD simulations, as part of his scientific initiation report.

10. REFERENCES

Bohn, D. and Lepers, J., 2003, "Effects of biogas combustion on the operation characteristics and pollutant emissions of a micro gas turbine", Proceedings of ASME Turbo Expo 2003, Atlanta, Georgia, USA.
Bonzani, F. and Pollarolo, G., 2004, "Ansaldo Energia gas turbine operating experience with low Btu fuels", Proceedings of ASME Turbo Expo 2004, Vienna, Austria.
Cohen H., Saravanamutto H. and Rogers G., 1987, "Gas Turbine Theory", 3ª Edition, Ed. Longman Scientific & Technical, New York, USA.
Hung, W.S.Y., 1989, "Gas Turbine Fuels", Gas Turbine Technology Seminar - Solar Gas Turbines, San Diego (CA), USA.
Lefebvre, A.H., 1998, "Gas Turbine Combustion", Ed. Taylor and Francis, USA.
Lora, E.E.S. and Nascimento, M.A.R., 2004, "Geração Termelétrica – Planejamento, Projeto e Operação", vol. 1, Ed. Interciência.
Mellor, A.M., 1990, "Design of Modern Turbine Combustors", Ed. Academic Press, England, UK.
Mollière, M., 2002, "Benefiting from the wide fuel capability of gas turbines: a review of application opportunities", ASME-Paper GT-2002-30017.
Rodrigues, L.O., 2009, "Análise Paramétrica de Câmara de Combustão de Turbinas a Gás Utilizando Dinâmica dos Fluidos Computacional", Tese de Doutorado, UNIFEI.

11. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.