THERMODYNAMIC ANALYSIS OF A MICRO GAS TURBINE CYCLE AND NUMERICAL SIMULATION OF ITS COMBUSTION CHAMBER GEOMETRY

Newton Kiyoshi Fukumasu, newton.fukumasu@gmail.com
Rafael Cavalcanti de Souza, rafael.cavalcanti.souza@gmail.com
José Rigoni Júnior, rigoni@usp.br
Guenther Carlos Krieger Filho, guenther@usp.br
Laboratory of Environmental and Thermal Engineering (LETE)
Department of Mechanical Engineering (PME)
Polytechnic School of The University of São Paulo
Cidade Universitária - São Paulo – SP - CEP - 05508-900

Abstract. *Alternative electricity generation is needed due to the high energy demand nowadays. From the many solutions available, gas turbines associated to electric generators is one of the solutions analyzed by many research centers and universities. In its simplest form, this solution is based on an automotive turbo-compressor unit coupled to a combustion chamber. Preliminary data indicate that this system could generate about 5kW of electric power.*
In this work, the gas turbine cycle was thermodynamically analyzed and the combustion chamber geometry was simulated by the finite volume method. The gas turbine was analyzed using the air-standard Brayton cycle, which models the working fluid as an ideal gas and assumes no losses on components. Furthermore, the unit was studied considering the working fluid as a real gas and also the pressure losses through the combustion chamber. These approachs were then compared and the combustion chamber geometry was simulated. Two burners geometries, bluff-body and swirler, and two fuel injectors, radial and axial flow, were studied. Preliminary experimental data of pressure and temperature were obtained at the combustion chamber inlet and outlet positions. The comparison between numerical simulation results and numerical data has pointed the influence of the swirler type mixer in the pressure loss and flame stabilization, which lead to a lower outlet temperature.

Keywords*: Gas turbines, Combustion, Swirl Stabilized Flame*

1. Introduction

Alternative electricity generation is needed due to the high energy demand in the modern life. In this scenario, many solutions are sought and the use of gas turbines is a viable alternative. This class of generator is called 'distributed generation', on which the electric energy is generated closest to the demanding area. Diesel engines are also classified as this type of generator, but gas turbines have advantages over these engines like greater power to weight ratio. It's estimated that large gas turbines have a power to weight ratio over 25 times than showed by diesel engines. Some disadvantages are: smaller efficiency and more sensitivity to fuel quality. The cost of this type of turbo machinery is higher due to the use of high technology engineering and materials.

Gas turbines are usually constituted by a combustion chamber, a compressor and a turbine. In the present work, only the combustion chamber was analyzed, since an automotive turbo-compressor was employed. This set is presented at the Fig.1.

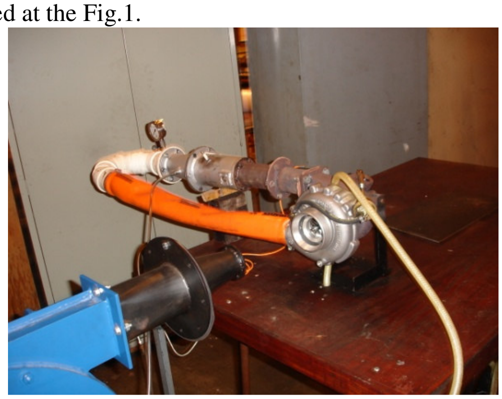


Figure 1 – Complete set of the micro gas turbine

In the present work, a thermodynamic analysis of the system is done by modeling the working fluid both as an ideal gas and a real gas, showing the differences between each approach. It is also analyzed the influence of the inlet temperature at the turbine on the work output. For the combustion chamber, two different geometries are studied for both mixer and fuel injector. Swirler and bluff-body were chosen for the mixer geometry because they are commonly used. For the fuel injector, radial and axial types were studied. It is also shown that pressures losses through the combustion chamber have major influence over the work output as well.

2. Thermodynamics cycles

The schematic of the set for a micro gas turbine is presented in Fig.2. There are three stages: admission of air through the compressor, combustion of the fuel/air mixture and gas expansion at the turbine. This kind of arrangement is not a complete thermodynamic cycle, because the products from the turbine exhaust are not connected to the compressor admission.

Although the used cycle is not closed, it is usually assumed as an air-standard Brayton cycle with an isoentropic turbine and compressor. The fluids are modeled as ideal gases, the kinetic energy variation on the fluid flow, between inlet and outlet of each component is negligible and there are no pressure losses through the combustion chamber, accordingly to Turns (2000).

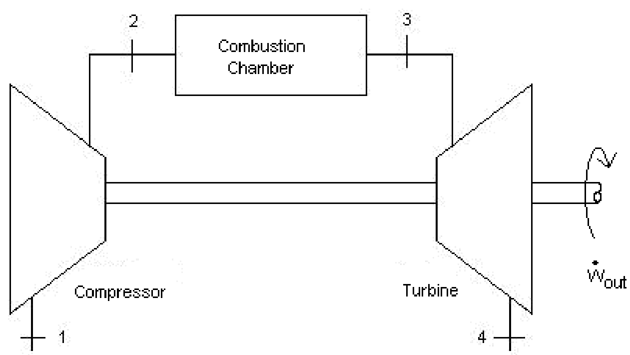


Figure 2 – Gas turbine cycle

For this arrangement, the first law of thermodynamics is given by Eq. 1. In this equation, Q and W are, respectively, the heat and the work transfer per unit mass flow. For each component of the cycle, compressor, combustion chamber and turbine, the first law reads respectively Eq. 2, 3 and 4.

$$Q = (h_o - h_i) + W \tag{1}$$

$$W_{comp} = W_{12} = -(h_2 - h_1) = -c_p(T_2 - T_1) \tag{2}$$

$$Q_{comb} = Q_{23} = (h_3 - h_2) = c_p(T_3 - T_2) \tag{3}$$

$$W_{turb} = W_{34} = (h_3 - h_4) = c_p(T_3 - T_4) \tag{4}$$

Using the isoentropic relation to couple pressure and temperature, it can be shown that the cycle efficiency is described by Eq. 5. In this equation, r is the pressure ratio ($p_2/p_1=p_3/p_4$) and k ($=c_p/c_v$) ratio between the specific heats.

$$\eta = 1 - \left(\frac{1}{r}\right)^{\frac{(k-1)}{k}} \tag{5}$$

Specific work output (w_{out}) is presented in Eq. 6. This expression shows the effect of inlet temperatures of both turbine and compressor ($t = T_3/T_1$), the pressure ratio (r) and the ratio between specific heat coefficients (k) on the specific work output. It can be seen that the available work output is tightly dependent on the turbine inlet temperature (T_3) , which is limited by material properties.

$$\frac{W_{out}}{c_p T_1} = t\left(1-\left(\frac{1}{r}\right)^{\frac{k-1}{k}}\right)-\left(r^{(k-1)/k}-1\right) \tag{6}$$

3. Combustion chamber geometry

The main objective of a combustion chamber is to elevate the enthalpy of the gas mixture by the chemical reaction of oxidation, Lefebvre (1983). Usually the combustion chamber is constituted by a casing, a liner, a mixer, a fuel injector and a diffuser, as presented at the Fig. 3.

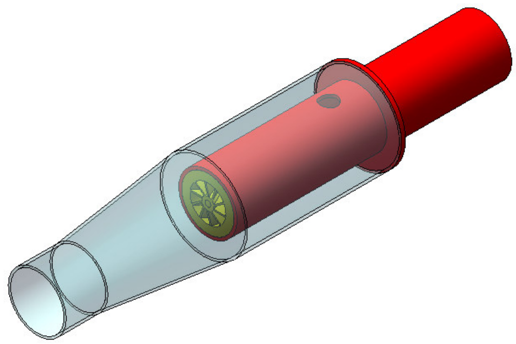


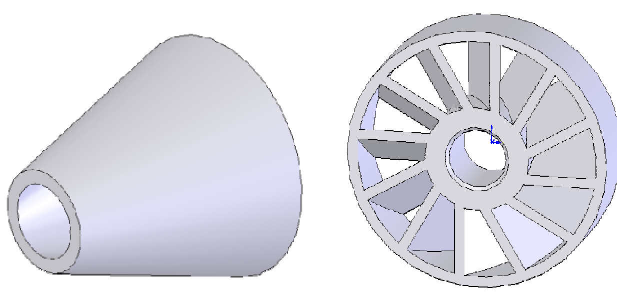
Figure 3 – Schematics of a combustion chamber composed by a diffuser and a casing (transparent blue), a liner (red), a mixer and a fuel injector (dark yellow)

The casing serves as a wall protection of the internal parts of the combustion chamber and isolates the hot kernel from the environment. As its diameter is usually a few centimeters greater than that of the liner, it improves the heat extraction of the liner with the enhancement of the heat exchange coefficient of the air flow between those walls.

Inside liner the combustion takes place. It is divided into two main regions: the primary and secondary zones. The primary air injection zone is used to stabilize the flame by the recirculation of the reactants flow. The secondary air injection region is designed to minimize the bulk temperature of the gas mixture to a specified level.

The diffuser is designed to reduce the air velocity and elevate the pressure at the entrance of the liner.

The mixer is the device that improves the mixture of fuel/air due to the generation of turbulence. The common used mixers are the bluff-body and the swirler. The first usually has a conical geometry as presented in Fig. 4a. The second consists of a series of blades as shown in Fig. 4b. The fuel can be injected in an axial or a radial manner, as presented in Fig. 5.



(a) (b)

Figure 4 – Configuration of the mixers: bluff-body (a), swirler (b)

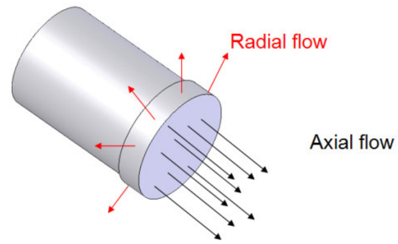


Figure 5 – Fuel injector geometry for an axial flow (black) and a radial flow (red)

In the present work, the flow pattern of each mixer and the two possible fuel injectors were studied by a computational fluid dynamics approach, as shown below.

4. Numerical simulation setup

To analyze the influence of the geometries of the mixer and the fuel injector in the mixture process, a computational fluid dynamics (CFD) simulation with the finite volume method (Versteeg, H.K., Malalasekera, W., 1998) of four geometries was done. Table 1 summarizes the studied cases. For the analysis of the bluff-body, case 3, it was composed with an axial fuel injector and for the swirler type mixer, case 4, the radial fuel injector was used. Those configurations characterize the common applications for those mixers. On all four cases, an isothermal and adiabatic flow was simulated, i.e., there were no reactions in the flow.

Table 1 – Studied cases by CFD simulations

Case	Mixer	Fuel Injector
1	-	Axial
2	-	Radial
3	Bluff-body	Axial
4	Swirler	Radial

For all cases, a steady flow with an ideal gas was considered. The SSG Raynolds Stress (ANSYS, 2003) turbulence model was applied due to the high rotational fluid motion observed at the swirler configuration. The physical domain was decomposed by a tetrahedral mesh which varied in size, due to the different geometries, but in the range from 1.5 to 3.2 million volumes.

A CFD study of the combustion chamber, with all its parts, was also carried on to evaluate the total pressure loss due to the use of a radial fuel injector and a swirler type mixer. Is this study, the computational fluid domain size was about 6 million volumes and it was executed on a cluster with twelve Xeon processors.

5. Experimental setup

The experimental setup built at University of São Paulo is composed by the combustion chamber described above and an automotive turbo-compressor unity type K27.2 manufactured by BorgWanner Turbo Systems Inc. The complete set is presented in Fig. 6. This figure shows also a 5 CV centrifugal fan used to start up the microgasturbine.

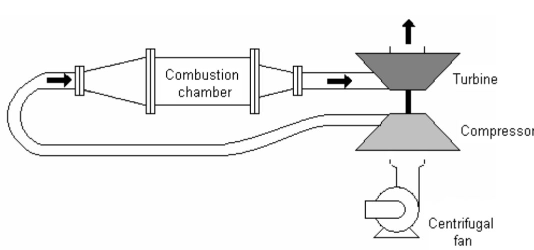


Figure 6 – Experimental setup schematic

The total work generated in the turbine is almost completely consumed by the compressor. The system starts with a centrifugal fan to supply the initial air. After the design operation point has been reached, which happens, for a rotational speed around 10.000 rpm, the external aid is turned off and the air is supplied solely by the compressor.

There is an automotive spark plug on the liner to start the combustion. As the combustion process begins, the rotational speed of the turbo-compressor unit increases. When the micro gas turbine sustains itself, the speed of the turbo system can be adjusted by the inlet fuel pressure at the combustion chamber.

6. Thermodynamic results

From the Eq. 5, the Brayton cycle efficiency is a function of the pressure ratio. This means that for a high efficiency, a high pressure ratio must be obtained. When working with real gases, it is noted that the efficiency reaches a peak for a certain pressure ratio and then begins to fall down. The efficiency versus pressure ratio for both cases is shown in Fig. 7, in which the ratio between heat coefficients was assumed as 1.4.

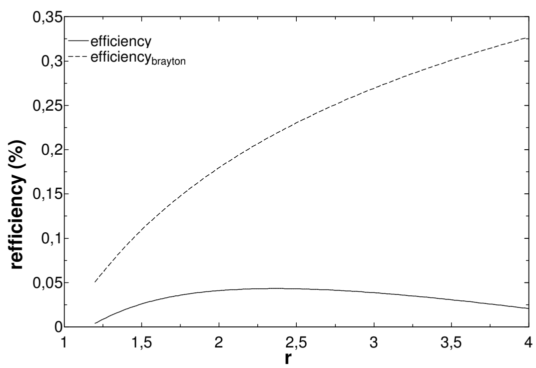


Figure 7 – Efficiency versus pressure ratio (r)

Fig. 8 shows the work output dependence of temperature T_3 for a designed constant pressure ratio of 1.8, based in Eq. 6. It is shown that the work output can reach as far as 7 kW for a practical temperature of turbine operation.

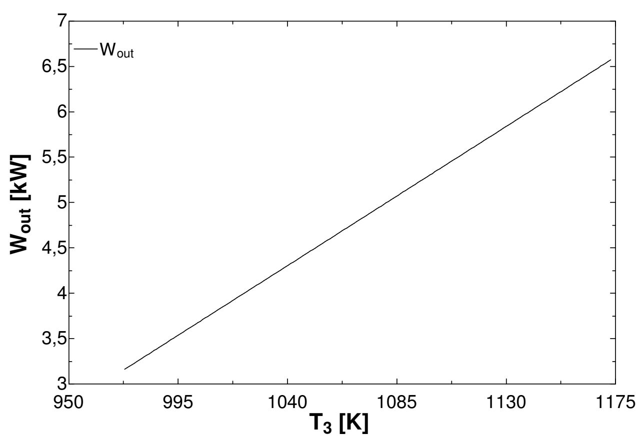


Figure 8 – Work output variation with temperature at the inlet of the turbine

As presented in Fig. 9, the pressure loss through the combustion chamber (P_3/P_2) is an important factor, because the ratio of pressures during expansion (P_4/P_3) is directly related to the amount of output work.

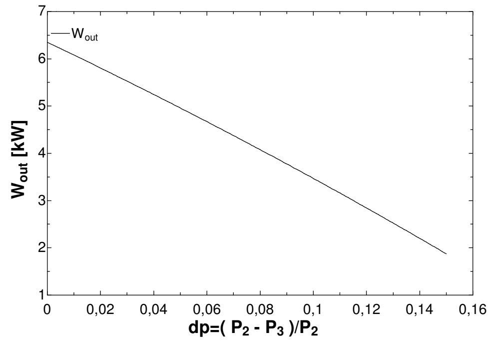


Figure 9 – Work Output variation with pressure loss through the combustion chamber

7. Numerical simulation results

The distribution of fuel mass concentration is presented in Fig. 10 and 11 for an axial and radial fuel injector, respectively. Fig. 12 presents the radial distribution of fuel mass concentration at the chamber outlet. It can be seen that a radial fuel injector has an almost homogeneous distribution along the radius, implying in a better mixing process compared to the axial type.

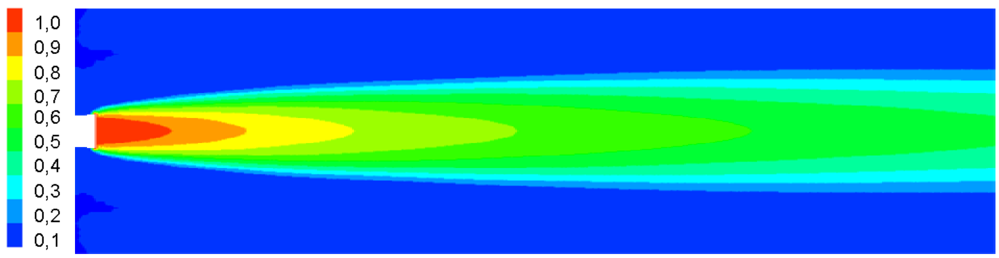


Figure 10 – Distribution of fuel mass concentration for an axial fuel injector

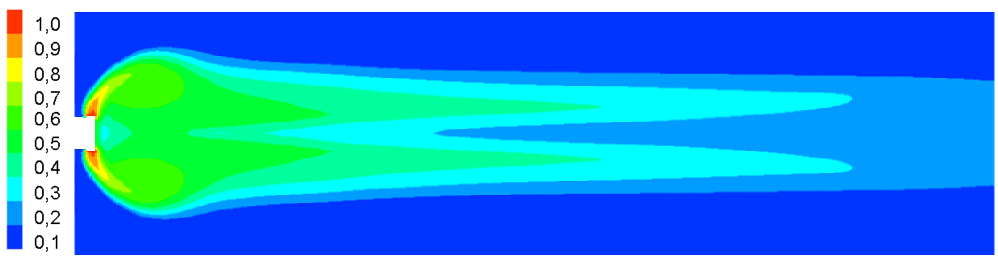


Figure 11 – Distribution of fuel mass concentration for a radial fuel injector

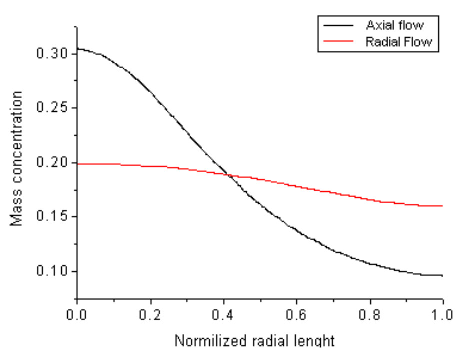


Figure 12 – Distribution of fuel mass concentration for both axial and radial fuel injectors at the outlet of the combustion chamber

Fig. 13 and 14 present the distribution of fuel mass concentration for cases 3 and 4, respectively. Comparing those figures, one can see that the swirler mixer improves the mixing process near the inlet of the liner, since this position presents a more homogenous distribution in the radial direction.

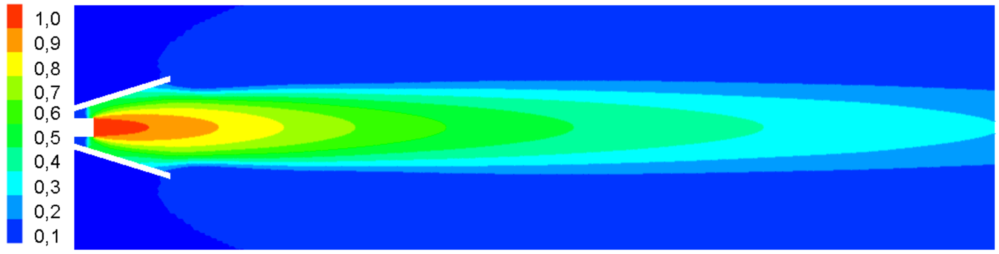


Figure 13 – Distribution of fuel mass concentration for an axial fuel injector with a bluff-body type mixer (case 3)

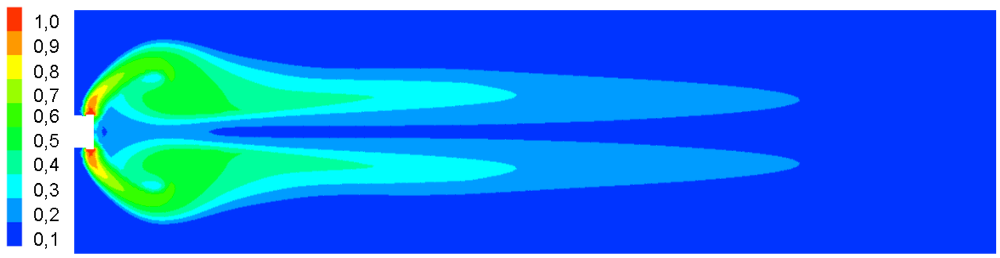


Figure 14 – Distribution of fuel mass concentration for a radial fuel injector with a swirler type mixer (case 4)

To better understand the 3D process of air/fuel mixture, Fig. 15 and 16 present the stream lines for cases 3 and 4, respectively. The intensity of the mixing process is higher in the case 4 with the generation of a huge turbulent kernel of recirculating fluid flow against the jet like distribution provided by case 3.

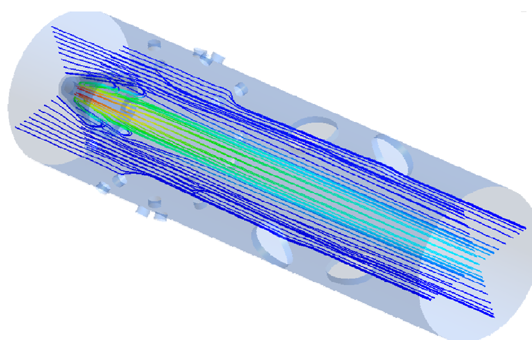


Figure 15 – Stream lines for the bluff-body type mixer configuration (case 3)

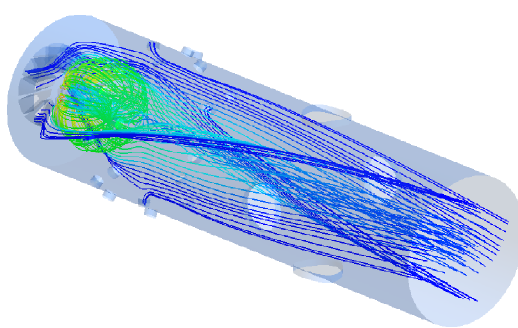


Figure 16 – Stream lines for the swirler type mixer configuration (case 4)

The simulation of the complete geometry of the combustion chamber, Fig. 3, allows one to verify that for an inlet mass flow of 0.01 kg/s, the air mass flow through the swirler is about 53% and and the remaining 47% is injected at the secondary zone of the liner. Fig. 17 shows the stream lines for the complete combustion chamber. Due to the fact that almost half of the air flow goes directly to the secondary zone, it leads to lower turbulent kernel intensity at the primary zone, compared to Fig. 16, but improves the turbulent mixture at the secondary zone, as shown in Fig. 18.

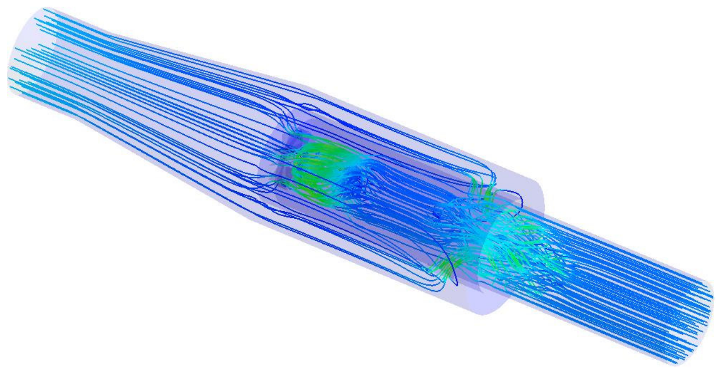


Figure 17 – Stream lines for the complete combustion chamber with a swirler type mixer

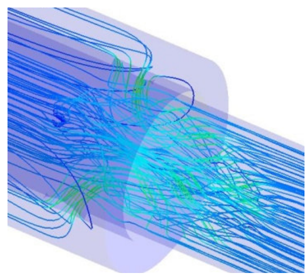


Figure 18 – Detail of the stream lines in the combustion chamber at the secondary zone of the liner

8. Experimental results

Using an air-standard Brayton cycle, the fluid states are defined for every point at Fig. 2. Table 2 shows those data for the system analyzed in this work. The efficiencies of the turbine and compressor are obtained from their operation maps and are, respectively, η_t=0.63 and η_c=0.77. The air was treated as ideal gas in steady flow.

Table 2 – Air state on each section based on Air-standard Brayton cycle

Location	Temperature [K]	Pressure [Pa]	Enthapy [kJ/kg]
1	300.0	101325.0	300.4
2	354.7	182385.0	355.5
3	973.0	182385.0	1016.0
4	837.1	101325.0	863.1

From the experimental set, it was possible to obtain the values of the pressure at the inlet position of the combustion chamber (about 170kPa) and the temperature at the inlet of the turbine (about 1115K). The comparison between experimental T_3 and numerical T_3 shows a discrepancy of about 142K. The reason for this departure between theorical and experimental value is probably due to a inefficient dilution process inside the liner. The swirler type mixer elevates the pressure loss along the chamber but, for high mass flows, it tends to stabilize the flame creating a high turbulent kernel, as shown in Fig. 16, that holds the flame attached.

9. Conclusion

In this work, a micro gas turbine cycle was thermodynamically analyzed and the combustion chamber geometry was studied by a finite volume method. It was shown that some initial assumptions may change the work output estimative. The air-standard Brayton cycle is the ideal cycle for gas turbines, where the mixture flow is modeled as an ideal gas and pressure losses are neglected. From these assumptions, some mathematical work is performed to find Eq. 5, which shows that the gas turbine efficiency is only a function of pressure ratio. Then, the higher the pressure ratio, the higher is the efficiency. But when the mixture flow is modeled as a real gas and pressure losses are taken into account, the efficiency reaches a peak for a certain value of pressure ratio. After this peak, no matter how great the difference between inlet and outlet pressures, the efficiency will not raise. The assumption of no pressure losses through the components also alters the net work. When pressure losses through the combustion chamber are taken into account, there's a significant reduction of the work output as shown in Fig. 9.

For the combustion chamber, it is shown that the combination of radial fuel injector and swirler type mixer produces the best mixing process, leading to a more stable combustion. This geometry was chosen because it is commonly used in this type of application.

When the complete combustion chamber geometry is simulated, it is shown that the secondary zone has an important effect over the mixing process. As it was shown, nearly 50% of the mass flow enters the liner through the secondary zone, reducing significantly the turbulent kernel intensity at the primary zone, generated by the swirler type mixer.

From the temperature and pressure measured on the experimental setup, one can see that the proposed combustion chamber works well, as far as flame stabilization is concerned. The discrepancy between theorical and experimental value on turbine inlet temperature (T_3) is probably due to inefficient dilution process.

The combustion chamber geometry proposed in the present work is a first approach to a low cost micro gas turbine. There are some issues concerning primary zone, secondary zone and swirler, that should be further studied. These issues will be addressed in future works.

10. Acknowledgments

The authors would like to acknowledge BorgWarner Turbo Systems Inc, Ventiladores Bernauer Ltda, ANP, FAPESP, Polytechnical School of the University of São Paulo.

11. References

ANSYS Company (2003) "CFX-5 Solver Theory Manual", CFX Ltd., Oxfordshire.

Lefebvre, A.H., "Gas turbine combustion", 1983, Ed. McGraw-Hill, S. Paulo, Brazil.

Turns, S.R., "An introduction to combustion: concepts and applications", 2000, Ed. McGraw-Hill, 2° ed., S. Paulo, Brazil.

Saravanamuttoo, H.I.H., Rogers, G.F.C, Cohen, H, 2001, "Gas turbine theory", Pearson Education Limited, Harlow, England.

Sonntag, R.E., Borgnakke, C., Van Wylen, G.J., "Fundamentos da termodinâmica", Ed. Edgard Blücher, S. Paulo, Brazil.

Versteeg, H.K., Malalasekera, W., 1998, "An introduction to computational fluid dynamics: The finite volume method", Addison Wesley Longman Ltd., England.

12. Responsibility notice

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