SWIRL STABILIZED COMBUSTION CHAMBER FOR A LPG FUELED MICRO GAS TURBINE

Guenther C. Krieger Filho, guenther@usp.br
Rafael Cavalcanti de Souza, rafael.cavalcanti.souza@gmail.com
Laboratory of Environmental and Thermal Engineering
Departamento de Engenharia Mecânica da Escola Politécnica da USP
Av. Prof. Mello Moraes, 2231
05508-900 - Sao Paulo - SP BRAZIL

Abstract. *The application of micro gas turbine in cogeneration systems is continuously increasing. The development of combustion chambers for such turbines is a critical issue, as far as the outlet temperature is concerned. In the present paper, the design of a combustion chamber suitable to a micro gas turbine is reported. The main design requirements for the combustion chamber are compactness, outflow gas temperatures and flame stability. The proposed combustion chamber is fueled with LPG and the flame is stabilized by a swirler. The flow inside the combustor was simulated using a commercial CFD code. Special attention was given to the recirculation zone, which is formed behind the swirler. The best results were achieved with the Reynolds Stress Model with standard wall function to account for the near wall effects. The turbulent combustion was accounted for with the adiabatic mixture fraction model. The experimental data of air and fuel mass flow rate as well as temperature at the exit of the combustion chamber were compared to the numerical results. The results concerning the outlet temperature (Turbine Intlet Temperature - TIT) show great discrepancy. Possible reason to the deviation between numerical and experimental results could be the mixture fraction model, which is not able to capture the chemical kinects of the turbulent flame.*

Keywords: *combustion chamber simulation, micro gas turbine, CFD*

1. INTRODUCTION

The application of micro gas turbine in cogeneration systems is continuously increasing. The major advantage of a micro gas turbine is the high power/weight ratio. Therefore, there are research groups, worldwide, working on the development of micro gas turbines. The design of combustion chambers for such turbines is a critical issue, as far as the outlet temperature and emissions are concerned. In the past, the combustion chamber design was based in empirical correlations (Lefvebre, 1983), (Heitor, 1985). Nowadays, the use of Computational Fluid Dynamics (CFD) can help the engineer to simulate the flow and temperature inside the proposed equipment. The present paper is focused on the design of a combustion chamber with the engineering requirements of a micro gas turbine.

2. OBJECTIVES

The main goal of the present paper is to design a combustion chamber suitable to a micro gas turbine. Two approaches are used: empirical correlations and computational fluid dynamics (CFD). The combustion chamber should fulfill the following engineering requirements: air flow rate of 0.2 kg/s; fuel flow rate of 0.0014 kg/s; ambient inlet temperature for both air and fuel; turbine inlet temperature (TIT) of $900\,^{\circ}\mathrm{C}$; combustion stability over a wide range of air flow rate.

3. METHODOLOGY

The main dimensions of the combustion chamber were calculated using the empirical correlations given in Lefevbre, (1983) and Saravanamuto, (2001). It was assumed that 30% of the compressor air flows through the swirler and the remaining through the dilution holes. This assumption should be evaluated by CFD simulation. The geometry of the chamber is shown in Fig. 1 and the air swirler is shown in Fig. 2. The flame is stabilized by a swirler mounted at the rear of the liner. The LPG fuel is injected at the center of the swirler through 2 holes of 1 mm and 6 holes of 0.08 mm. There are two dilution holes placed at the end of the casing. No primary holes were used. The ignition is provided by a spark plug mounted facing the internal surface of the liner. The main dimensions of the chamber are given in Tab. 1.

3.1 Numerical Modeling using CFD

The turbulent flow inside the combustion chamber was simulated using a commercial CFD code, based on finite volume method. The conservation and transport equations are described below.

The mass conservation for the mean flow is given by

Table 1. Main dimensions of the combustion chamber

Casing diameter (mm)	100
Casing length (mm)	175
Liner diameter (mm)	70
Liner length (mm)	480
Swirler inner diameter (mm)	15
Swirler outer diameter (mm)	40
Number of blades	10
Angle of blades	60 °

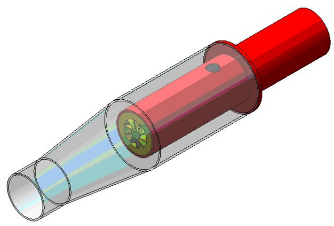

Figure 1. Geometry of combustion chamber

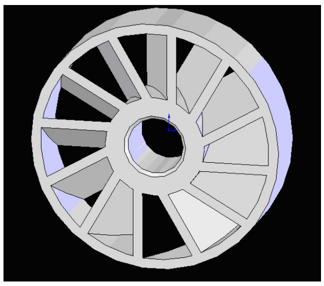

Figure 2. Air swirler

$$\frac{\partial \rho}{\partial t} = div(\rho \vec{u}) \tag{1}$$

where ρ is the density of the mixture, t is the time and $\vec{u}$ is the mean velocity vector.

The 2nd Newton's Law applied to a fluid control volume leads to the averaged momentum equation:

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}(-\rho\overline{u_i' u_j'}) \tag{2}$$

where p is the static pressure and $u_i' u_j'$ is the Reynolds stresses tensor. The first Law of thermodynamic applied to a control volume reads:

$$\frac{\partial \rho e}{\partial t} + div \rho e \vec{u} = -p\, div \vec{u} + div\left(k_{eff} gradT\right), \tag{3}$$

where e is the internal energy of the fluid, T is the temperature of the mixture.

The turbulent flow is calculated using the Reynolds Stress Model [5]. The transport equation of the Reynolds stress tensor is given by:

$$
\begin{aligned}
&\underbrace{\frac{\partial}{\partial t}(\rho\,\overline{u_i' u_j'})}_{\text{Local Time Derivate}} + \underbrace{\frac{\partial}{\partial x_k}(\rho u_k \overline{u_i' u_j'})}_{C_{ij} \equiv \text{Advection}} = \underbrace{- \frac{\partial}{\partial x_k}\left[\rho\,\overline{u_i' u_j' u_k'} + \overline{p\left(\delta_{kj} u_i' + \delta_{ik} u_j'\right)}\right]}_{D_{T,ij} \equiv \text{Turbulent Diffusion}} \\
&+ \underbrace{\frac{\partial}{\partial x_k}\left[\mu \frac{\partial}{\partial x_k}(\overline{u_i' u_j'})\right]}_{D_{L,ij} \equiv \text{Molecular Diffusion}} \underbrace{- \rho\left(\overline{u_i' u_k'}\,\frac{\partial u_j}{\partial x_k} + \overline{u_j' u_k'}\,\frac{\partial u_i}{\partial x_k}\right)}_{P_{ij} \equiv \text{Stress Production}} \underbrace{- \rho\beta(g_i \overline{u_j'\theta} + g_j \overline{u_i'\theta})}_{G_{ij} \equiv \text{Buoyance Production}} \\
&+ \underbrace{\overline{p\left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i}\right)}}_{\phi_{ij} \equiv \text{Pressure Strain}} - \underbrace{2\mu \overline{\frac{\partial u_i'}{\partial x_k}\frac{\partial u_j'}{\partial x_k}}}_{\epsilon_{ij} \equiv \text{Dissipation}} \\
&- \underbrace{-2\rho\Omega_k\left(\overline{u_j' u_m'}\epsilon_{ikm} + \overline{u_i' u_m'}\epsilon_{jkm}\right)}_{F_{ij} \equiv \text{Production by System Rotation}} \qquad (4)
\end{aligned}
$$

In Eq. (4) the terms of convection, C_{ij}, production, P_{ij}, and molecular difusion, $D_{L,ij}$ do not need modeling. The remaining terms need modeling. The turbulent diffusion $D_{T,ij}$ is modeled as:

$$D_{T,ij} = \frac{\partial}{\partial x_k}(\frac{\mu_t}{\sigma_k}\frac{\partial \overline{u_i' u_j'}}{\partial x_k}), \qquad (5)$$

where, μ_t is the turbulent viscosity, defined in Eq. 6. The value of 0.82 is used fot the turbulent Prandtl number, σ_k,

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \qquad (6)$$

The standard value of C_μ is 0.09.

The pressure-strain term Φ_{ij}, which is one of more complex in the RSM model, is decomposed in three components as:

$$\phi_{ij} = \phi_{ij,1} + \phi_{ij,2} + \phi_{ij,w}, \qquad (7)$$

where, $\Phi_{ij,1}$ is known as return-to-isotropy, $\Phi_{ij,2}$ is called rapid pressure-stain and $\Phi_{ij,w}$ is the wall-reflection.

In order to avoid refining the grid at the solid boundaries, standard wall functions were used. The thermochemistry model is based on the conserved scalar model (Malalasekera, 2006). In this model the mixture fraction (Turns,2000) is the variable transported. Due to its definition, the transport equation for the mean mixture fraction, f, does not have source term and reads

$$\frac{\partial}{\partial t}(\varrho f) + \frac{\partial}{\partial x_j}(\varrho f u_j) = \frac{\partial}{\partial x_j}\left(\varrho D_f \frac{\partial f}{\partial x_j}\right) \qquad (8)$$

Under the assumption of unit Lewis number for an adiabatic system, the normalized equation for the mixture enthalpy has the same structure and boundaries conditions as the mixture fraction. Therefore in the conserved scalar model, a solely transport equation for the scalar f is solved representing both thermochemical variables the mixture fraction and the mixture enthalpy. The instantaneous mixture temperature as function of the mixture fraction is shown in Fig. 3. One can see that the stoichiometric mixture fraction is about 0.07 for the propane fuel (C_3H_8).

The boundaries conditions used in the simulations are given in Tab. 2. All external walls were considered adiabatic.

4. RESULTS

The simulated velocity field inside the liner and casing is shown in Fig 4.

One can see that there are simetrical recirculation zones behind the swirler. This is a necessary feature of the flow in order to stabilize the flame. The experimental tests have shown that the more the air flow increases, the more stable the flame is. So one can conclude that the swirler is working properly, at least as far as the stabilization is concerned. The flow through the dilution holes can also be seen.

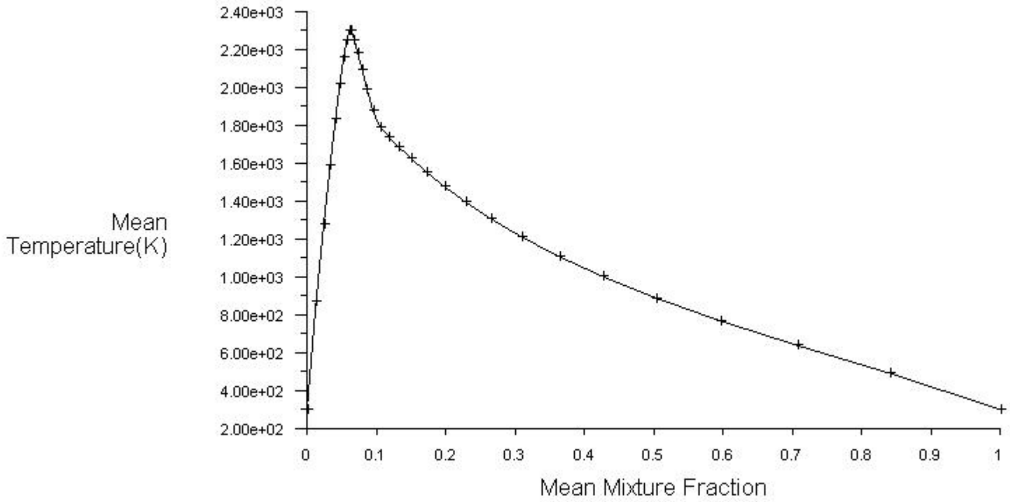

Figure 3. Temperature as funciton of the mixture fraction

Table 2. Boundary Conditions

Variabla	Prescribed value	Other
Air inlet	0.15 kg/s	
Outlet		Developed flow
Fuel inlet	0.0012 kg/s	

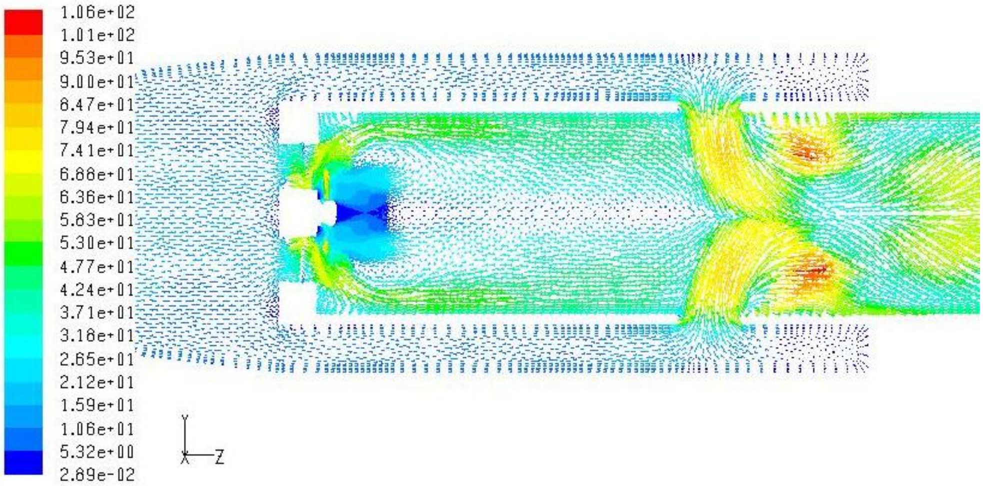

Figure 4. Velocity field near the fuel injector

The temperature distribution inside the liner and casing is shown in Fig. 5. The high temperatures of about 2000K can be seen just upstream of the dilution holes. It means that the flame extends from the fuel injector until the dilution holes. In Fig. 6 the whole temperature field is depicted. It is important to note that the outlet temperature reaches values of 800K. The experimental data, obtained with N-type thermocouple, show temperature of 1250K. There is a large discrepancy between simulated and measured values.

In order to better understand the flame structure inside the liner, one has to investigate the mixture fraction distribution. The Fig. 7 shows the values of mixture fraction ranging from 0 to 0.1, since the stoichiometric mixture fraction is 0.07. One can observe that just behind the dilution holes there are spots of mixture fraction of 0.035. It means that the mixture processes between fuel and oxidizer is still going on, what can explain the high temperatures at the combustion chamber outlet.

The experimental data measured during the gas generator operation are resumed in Tab. 3

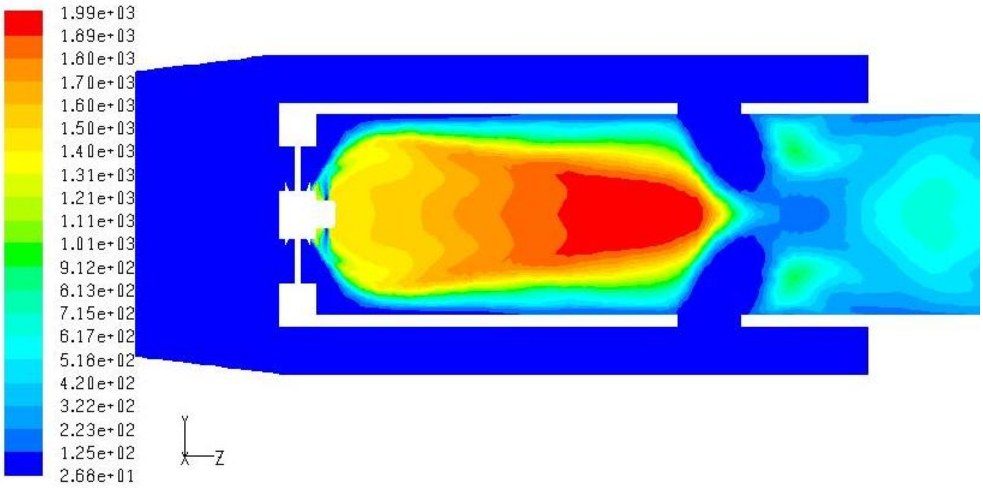

Figure 5. Temperature field near the fuel injector

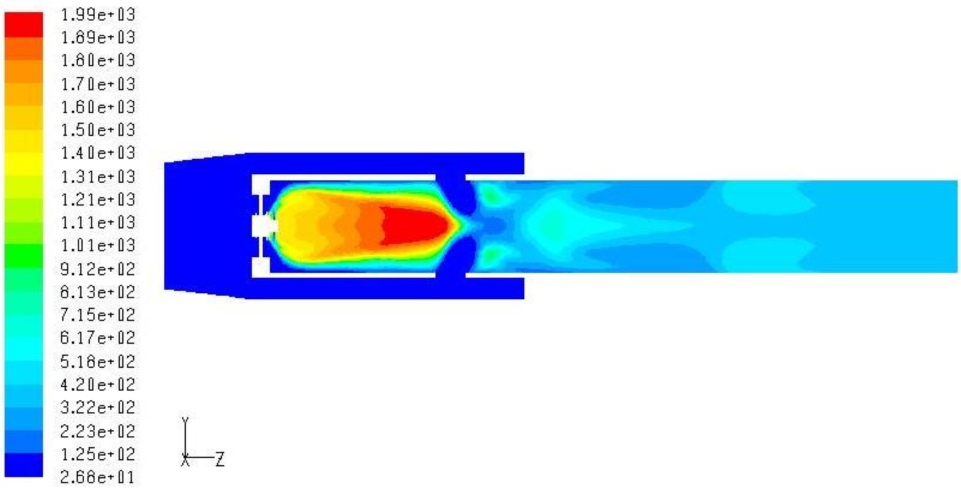

Figure 6. Temperature field in the combustion chamber

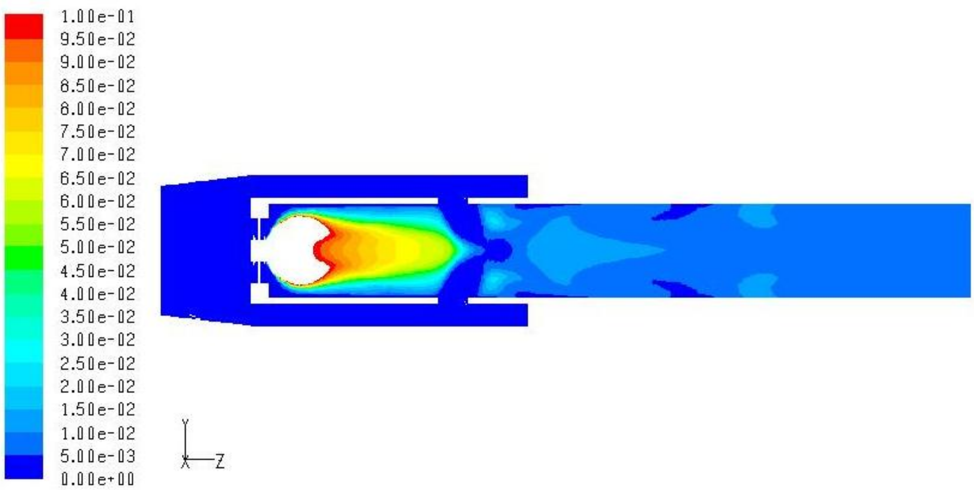

Figure 7. Mixture fraction field in the combustion chamber

5. CONCLUSION

In this paper, a micro gas turbine combustion chamber, fueled with LPG, was simulated using CFD and the main results were compared to experimental data. The combustion chamber basic design was done using empirical correlations. The

Table 3. Experimental Data

Air flow rate	0.15 kg/s
LPG flow rate	0.0014 kg/s
Outlet temperature (TIT)	1250 K
Pressure in the chamber	1.0 (gauge)

numerical simulation has captured the recirculation zones behind the swirler and has revealed that the mixing process reaches the region downstream of the dilution holes. This flow pattern increases the outlet temperature (the TIT turbine temperature). The experimental data of TIT were higher than the simulated one, despite of the adiabatic model. One possible reason for this disagreement is the mixture fraction model, which is based on chemical equilibrium and therefore is not able to capture chemical kinetics effects. This issue is, however, under investigation by the authors.

6. ACKNOWLEDGEMENTS

This project is supported by FAPESP under the grant 2007/03072-1, ANP- PRH-19 and BorgWarner Turbo Systems.

7. REFERENCES

Heitor,M. F. T. V.,“ Experiments in Turbulent Reacting Flows”, Ph.D Thesis, Imperial College of Science and Technology, UK, 1985

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

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

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

Versteeg, H.K., Malalasekera, W, "An introduction to computational fluid dynamis - the finite volume method", 2nd edition, Prentice Hall, London, UK, 2007.

8. Responsibility notice