Copyright © 2005 by ABCM

ANALYSIS OF FLAME BEHAVIOUR IN SMALL COMBUSTION CHAMBERS USING CFD

Harley Souza Alencar
Universidade Federal de Itajubá, Benedito Pereira dos Santos Av., n° 1303, Zipcode 37500-000, Itajubá, Minas Gerais, Brazil
e-mail halvader@itelefonica.com.br

Helcio Francisco Villanova
Universidade Federal de Itajubá, Benedito Pereira dos Santos Av., n° 1303, Zipcode 37500-000, Itajubá, Minas Gerais, Brazil
e-mail patricia.foroni@itelefonica.com.br

Marco Antonio Rosa do Nascimento
Universidade Federal de Itajubá, Benedito Pereira dos Santos Av., n° 1303, Zipcode 37500-000, Itajubá, Minas Gerais, Brazil
e-mail marcoantonio@unifei.edu.br

Abstract. *The use of small gas turbines in regional electrification in Brazil has been a promising energetic alternative as soon as this alternative can be installed in wherever, with power until 300 kW and small dimension. Although, some limits are known: the small efficiency (until 30 %); unstable flame; and emissions of pollution.*

Its optimized design depends on the flame behavior that is generated in combustion chamber. From this way, many manufactures have looked for the improved performance in operation using experimental and numerical tests. For it, some promising researches have been done to set the affects created by: geometry (cylinder or annular); quantity and position of nozzles; characterization of flames (diffusion or turbulence); quantity of steps for chemical reactions; and emission of pollution and non-combustion materials. Because of it, this work shows a particular study about the behavior of flame that is generated in a model of small gas turbine of annular combustion chamber, and the thermal-aerodynamic simulation using CFD. The results are based in velocity, pressure and temperature sources. The work fluid is methane with combustion for 2 steps. The used mathematical models study the flow, the combustion and heat transfer in radiation, such as: RNG K-ε turbulence Model; Eddy Dissipation Model – EDM for combustion; and P1 radiation model. In this annular combustion chamber, the nozzles have an angle in relation the main axle of combustion chamber, and there is a secondary flow with rotary movement. From this, there is a recirculation near to the combustion zone that permits to increase the residence time of flame, which can be a little diffusive and uniform heat transfer by convection and radiation.

Keywords*: Thermal – Aerodynamic, CFD, Combustion Chambers, and Small Gas Turbine*

1. Introduction

According to the last researches, the use of gas turbines has presented to be a promise alternative to attend the regional development, mainly, in regions that do not have enough hydroelectric sources. In Brazil, for instance, the thermoelectric system has been expanded from 5% to 10%, according to data from Aneel (2003), if it is compared with hydro electrical system, which has approximately 90 % in all electrical system.

Hence, the interesting to install small thermal power plants has been increased too with the function to attend the regional development in last years, using small gas turbines with power until 300 kW, but it has problems with small efficiency (until 30 %) and emission of pollution. According to Lefebvre (1987), others particular problems can be identified, such as: ignition efficiency; cool of metal parts; stable flame without pressure fluctuations; fuel injection; and flammability limits.

A possible solution, that was shown by Melick et al. (1998) and Wakabayashi et. al. (2002) in relation to the unstable flame, for instance, is adopt in fuel injection a valve to adjust the reason air / fuel. This adjustment permits to correct the flow in fuel nozzle.

On the other hand, Yadigaroglu et al. (1998) shows an aerodynamic study in a cylinder combustion chamber in relation to the affects of load losses in emissions. If the load loss is increased, the residence time is increased too, depend on the dimensions of chamber, according to Lefebvre (1995), neither the combustion zone has the tendency to be near to nozzles and the combustion efficiency can be decreased.

Besides, there are others possibly improvements to control the combustion efficiency and the emissions, such as: (a) the change of geometry, with the inclusion of holes for air and/or the change of nozzles; and (b) the use of fuels without sulfur with high reactivity, like as, the natural gas.

In relation to the first alternative, new geometries have been developed like as the Dry Low Nox - DLN chambers, which are the cylindrical combustion chambers with big quantity of holes in Liners and dilution holes in downstream of flame.

Allen (1998) shows that this quantity of holes has the advantage to pull the flame to a zone near to the nozzles, with the function to guarantee the stable flame and increase the dilution of gas in emission. Although, this type of combustion chamber works good using the gas natural, due to its quick dilution, according to Wakabayashi et al. (2002).

Besides, Willian (1998) explored the DLN chambers using other kind of fuel with liquid phase and generated by emulsion process. In relation to the second alternative, Vandebroek (2003) explored the affects of chemical kinetic in combustion of methane with 4 steps, with the function to esteem the temperature for auto ignition. According to this work, in general view, good results can be archived in combustion until 4 steps instead of the full reaction, that could archive more than 20 steps, depend on the fuel.

Besides, Melick et al. (1999) and Lefevbre (1995) show other way to control the emission of NOx based in catalysts with NH_3 and the possible effects in environment. It is described too the importance of natural gas like an alternative fuel.

In general, all of results can be obtained by experimental and numerical tests. While the first, it is necessary to raise enough money with expensive equipments and benches, the numerical tests have been a promise alternative with the advantage use of more quick computers with high capacity to save data. From this way, it is possible to improve the modeling of combustion, Kuo (1986). For it, different numerical methods have been tested to simulate the operation in combustion chambers. Among these methods, the Computational Fluid Dynamic – CFD has more applications.

From this way, the works from Nicklaus et. Al. (2002), Gosselin et. Al. (2000), Willian (1998) and Lee et. Al. (1990) execute the first simulations using CFD in cylindrical combustion chambers for bi-dimensional and three-dimensional geometry models. The turbulence models tested by them are K-ε, RNG K-ε and Reynolds Stresses Model – RSM, and the combustion models Fast Chemistry with PDF Model and Flamelet Laminar Model – FLM.

Besides, the works from Fuller et al. (1994), Hamer e Roby (1997) introduced others combustion models, such as, Eddy Dissipation Model – EDM, with models of NOx emission of Fenimore and Zeldovich, for three-dimensional geometry models.

Besides, new studies in annular combustion chambers for small gas turbines have been increased with the function to solve its problems with low efficiency and emission of pollution and, in a general view, its economic feasibility.

Hence, this work shows an analysis using CFD about the behavior of flame in an annular combustion chamber from the thermal-aerodynamic phenomenon discretized in space by characteristic curves of working in relation to the pressure, the velocity and the temperature. The geometry model is based on the prototype developed by Harvester Company, T-62T-32 model.

2. Methodology

The methodology consists to solver the physical problem relative to the behavior of flame in a model of annular combustion chamber for a small gas turbine, using the Computational Fluid Dynamics – CFD that represents a part of science for Numerical Methods that simulate the behavior of flows, based on Eulerian Method known as Finite Volume Method.

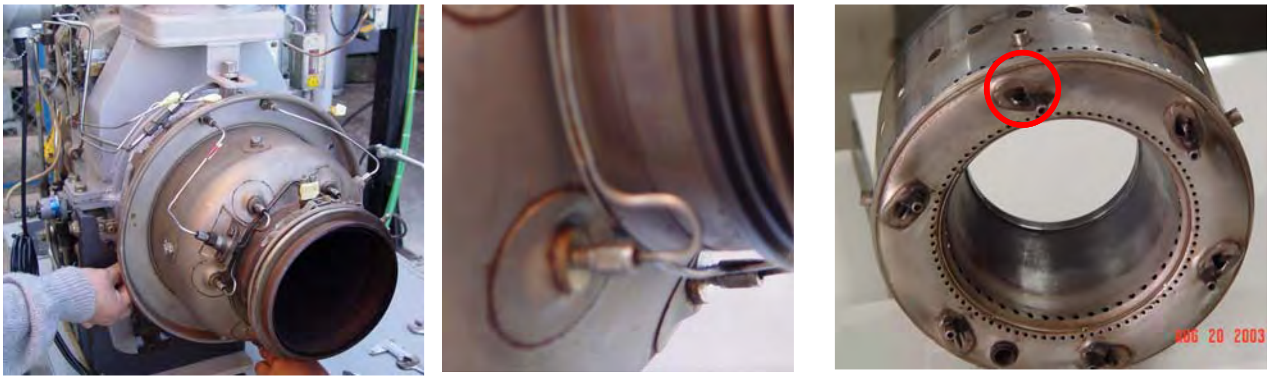
For this method, the physical domain is discretized by volume elements of type tetrahedrical or quadrilateral, whose geometries can be structured or not, i.e., the physical greatness is distributed in center of these elements, that permits to realize detailed analysis in geometrical model. The computational tool applied is the CFX® v 5.7 from ANSYS, which is used to define the three basic steps for the modeling: pre-processing, processing and post-processing.

In **Pre-processing**, the geometry model is generated from the prototype of an annular combustion chamber, which has 6 nozzles with inclination equal to 60 grades in relation to the axial axle of combustion chamber. This model has 86 primary holes and 46 secondary holes to the inlet of air that is used to the combustion and to the dilution and cooling of hot gas, which is used to work a small turbine.

This prototype of small gas turbine is designed to work with diesel and kerosene according to the manufacturer Harvester Company, Solar Model type T-62T-32, whose main operation conditions are:

a) Fuel pressure in inlet of nozzle: 275790 Pa;
b) Environment temperature and pressure according ISO condition: 25°C and 1 atm;
c) Mass flow of fuel for 6 nozzles: 29 kg/s;
d) Fuel consumption 0,85 kg / MWh;
e) Pressure, temperature and mass flow of air in outlet from compressor, respectively: 206843 Pa, 52°C and 0,75 kg/s;
f) Maximal temperature of gas in outlet of combustion chamber: 637,9 °C; and
g) Maximal temperature of gas in exhaustion after turbine: 493,2 °C.

In combustion chamber, the primary air is mixtured with fuel in nozzle with high pressure and velocity. Depend on capillarity and viscosity, the fuel can be drafted by air, due to the decrease of relative pressure inside of nozzle. Figure 1 shows the prototype that is modeled.



(a) (b) (c)

Figure 1 – (a) Small gas turbine; (b) External detail of nozzles; and (c) Combustion chamber with 6 nozzles (one of them is shown by red cycle)

Due to the inclination of the nozzles in relation to the axial axle of combustion chamber, it is generated a secondary flow with rotation. From this way, the time residence tends to be big in annular combustion chamber, which permits to improve the combustion.

The flame is rotary and a little diffuse, whose position must not be near to the metallic walls. From this way, this combustion chamber is compact, soft with easy transportation and maintenance. Besides, the works from Willian (1998) and Gosselin et al. (2000) show that it is possible to get satisfactory results for a model that corresponds to 1 / 6 from original volume of prototype, including the convection heat transfer, considering the symmetry in relation to the main axle of combustion chamber. From this way, it is possible to get a more simply model, where the mesh is generated quickly and with good discretization (refinement) to capture all data about the flame (velocity of propagation, pressure, temperature and emission).

Hence, in this work the model of combustion chamber is 1 / 6 of volume for the prototype from combustion chamber of small gas turbine, Solar Model type T-62T-32 from Harvester Company, whose alternative fuel adopt is only the methane CH_4. Figure 2 shows this geometric model with details.

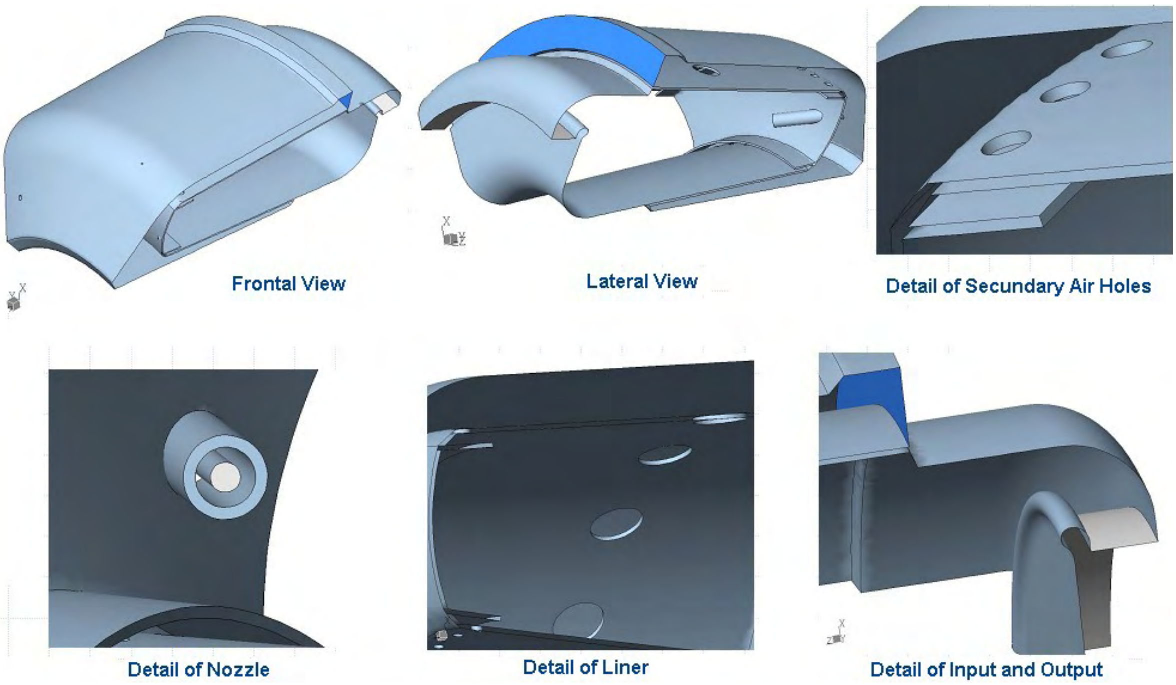


Figure 2 – Geometry model tested in details

The mesh generated is unstructured with 1915508 tetrahedral and prismatic elements and 4666141 nodes. Figure 3 shows a part of this mesh in a longitudinal surface paralleled to the axial axle of combustion chamber, in a region near to the outlet of nozzle. Figure 4 presents the details of this mesh.

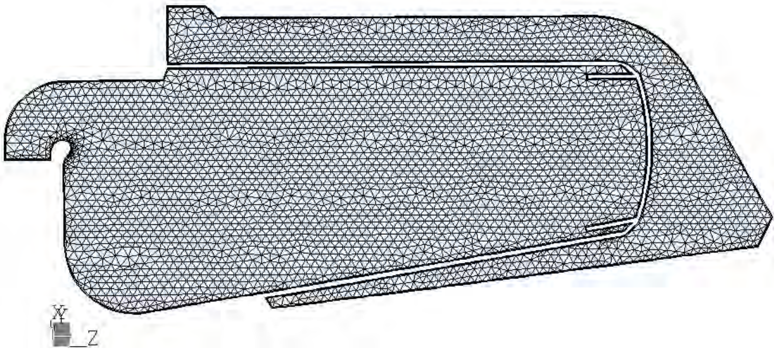


Figure 3 – Lateral view of mesh

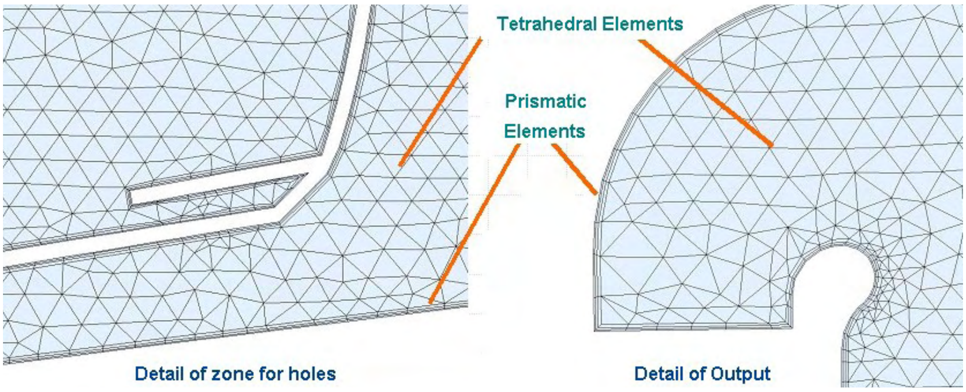


Figure 4 – Details of mesh: (a) in region near to the nozzle; and (b) in outlet of hot gas

In **Processing**, the domain is controlled by the **Conservative Equations of Continuous, Momentum (Navier Stokes) , Energy** in differential way in relation to the time and the space, respectively:

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho . U) = 0 \quad \Rightarrow \quad \frac{1}{\rho} . \frac{D \rho}{D t} = -\nabla \bullet U = \text{Compressibility} \tag{1}$$

$$\frac{\partial U}{\partial t} + U \bullet \nabla U = -\frac{1}{\rho} . \nabla p + \frac{\partial}{\partial x_i}\left[\nu . \frac{\partial Ui}{\partial x_j} - (Ui . Uj) \right] + g \tag{2}$$

$$\frac{\partial T}{\partial t} + U \bullet \nabla T = \alpha . \nabla^2 T + \overline{Q} + \varphi \tag{3}$$

Where: ρ is the density of fluid, in [kg/m^3]; **U** is the velocity, in [m/s]; **t** is the time, in [s]; **p** is the relative pressure, in [Pascal]; **Ui**, **Uj** are the velocity in direction **xi** and **xj**, respectively, in [m/s]; **i** and **j** is **the Indicial Notation of Einstein**, in relation to direction **x** and **y**, respectively, to discretization of space; ν is the kinetic viscosity, in [kg/m. s]; **g** is the gravity acceleration, in [m/s^2]; α is the **Thermal Diffusibility Coefficient**, in [m^2/s]; **T** is the absolute temperature, in [Kelvin]; $\overline{Q}$ is heat generated locally, in [kJ]; and φ is the **specific energy due to the friction between the fluid and the walls**, in [J/kg].

Besides, the heat transfer for radiation can be analyzed by the **Transportation Equation for Spectral Radiation (RTE)**, Beer et al. (1971):

$$\frac{d I_\nu (r,s)}{d s} = -(K_{av} + K_{sv}) . I_\nu (r,s) + K_a . I_b (\nu, T) + \frac{k_{sv}}{4.\pi} . \int_{4.\pi} dI_\nu (r, s') . \Phi(s \bullet s') \, d\Omega + S \tag{4}$$

Where: ν is the emission frequency; **r** is the position vector; **s** is the direction vector; ***s´*** is the distance traveled by radiation; $\mathbf{K_{av}}$ is the absorption coefficient; $\mathbf{K_{sv}}$ is the reflexion coefficient; $\mathbf{I_b}$ is the intensity of emission in black body; $\mathbf{I_n}$ is the intensity of spectral radiation that depend on the position **r** and the direction **s**; $\mathbf{\Omega}$ is the solid angle; $\mathbf{\Phi}$ is function to phase of reflexion due to a immersed solid in domain; and **S** is the source term for the radiation (combustion).

From this way, the modeling of flow is done by application of different turbulence models to solve the term represented by **Reynolds Stress Tensor**, **Ui. Uj**, in Eq. (2).

Among the turbulence models applied by others authors, such as **K-ε**, **RNG K-ε** and **Reynolds Stress Model - RSM**, the **RNG K-ε** model is adopted because permits to describe flows in curved surfaces, where there is rotary flows, and have the capacity to capture the smallest vorticities generated by small holes in combustion chamber model. For this model, the **Reynolds Stress Tensor Ui. Uj** can be calculated by expression, Yadigaroglu (1998):

$$-\rho\,.\,(U_I\,.\,U_J) = 2\,.\,\mu_t\,.\,S_{IJ} - \frac{2}{3}\,.\,\rho\,.\,k\,.\,\delta_{IJ} \tag{5}$$

Where δ_{IJ} is the Dirac Function; and S_{IJ} is the Average Shear Tensor calculated by expression:

$$S_{IJ} = \frac{1}{2}\,.\left(\frac{\partial U_I}{\partial x_J} + \frac{\partial U_J}{\partial x_I}\right) \tag{6}$$

Where the **turbulent viscosity** μ_t is given by:

$$\mu_t = C_\mu\,.\,\rho\,.\,\frac{k^2}{\varepsilon} \tag{7}$$

Where:

$$k = \frac{1}{2}\,.\,(U_I\,.\,U_J) \qquad \varepsilon = \nu\,.\left(\frac{\partial U_I}{\partial x_J}\,.\,\frac{\partial U_J}{\partial x_I}\right) \tag{8}$$

Thus, the main equations used to simulate the turbulence by **RNG K-ε Model** are:

$$\frac{\partial(\rho\,.\,U_J\,.\,k)}{\partial x_J} = \frac{\partial}{\partial x_J}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right).\frac{\partial k}{\partial x_J}\right] + 2\,.\,\mu_t\,.\,S_{IJ}\,.\,S_{IJ} - \rho\,.\,\varepsilon \tag{9}$$

$$\frac{\partial(\rho\,.\,U_J\,.\,\varepsilon)}{\partial x_J} = \frac{\partial}{\partial x_J}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right).\frac{\partial \varepsilon}{\partial x_J}\right] + 2\,.\,C_{\varepsilon 1}\,.\,\frac{\varepsilon}{k}\mu_t\,.\,S_{IJ}\,.\,S_{IJ} - C_{\varepsilon 2}\,.\,\rho\,.\,\frac{\varepsilon^2}{k} \tag{10}$$

Where there are five important constants, **Cμ, Cε1, Cε2, σk e σε**, whose typical values based on experimental tests for isotropic flows are presented by tab.1, where are shown others three auxiliary constants η, η_0, and β:

Table 1 – Typical values for constants in turbulence model **RNG K-ε**

C_μ	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	σ_k	σ_ε	η	η_0	β
0,076 to 0,094	$1{,}42 - \dfrac{\eta\,.\left(1 - \dfrac{\eta}{\eta_0}\right)}{1 + \beta\,.\,\eta^3}$	1,51 to 1,85	0,6103 to 0,8256	0,6461 to 0,7897	$\left(\dfrac{k}{\varepsilon}\right).(2\,.\,S_{IJ}\,.\,S_{IJ})^{1/2}$	3,94 to 4,82	0.012 to 0,017

Besides, among the combustion models such as, **Eddy Dissipation Model – EDM**, **Finite Rate Chemistry – FRM**, and **Laminar Flamelet Model – LFM**, the **EDM Model** is adopted because it can describe combustion reactions with pre-mixture between methane and air in nozzles, where the mechanism of reaction can be described by the following equations of combustion:

Phase 1: the main reaction is given by expression:

$CH_4 + (O_2 + 3.76\,N_2) \rightarrow CO_2 + 2\,H_2O + 3.76\,N_2$ (11)

Phase 2: the reaction is described using stoichiometric air in two steps:

$CH_4 + (O_2 + 3.76\,N_2) \rightarrow CO + 2\,H_2O + 3.76\,N2 + ½\,O_2$ (12.a)

$CO + ½\,O_2 \rightarrow CO_2$ (12.b)

According to Lefebvre (1987), the reaction rate can be affected by turbulence and esteemed by following expression:

$$\text{Reaction Rate} \propto \frac{\varepsilon}{k} \tag{13}$$

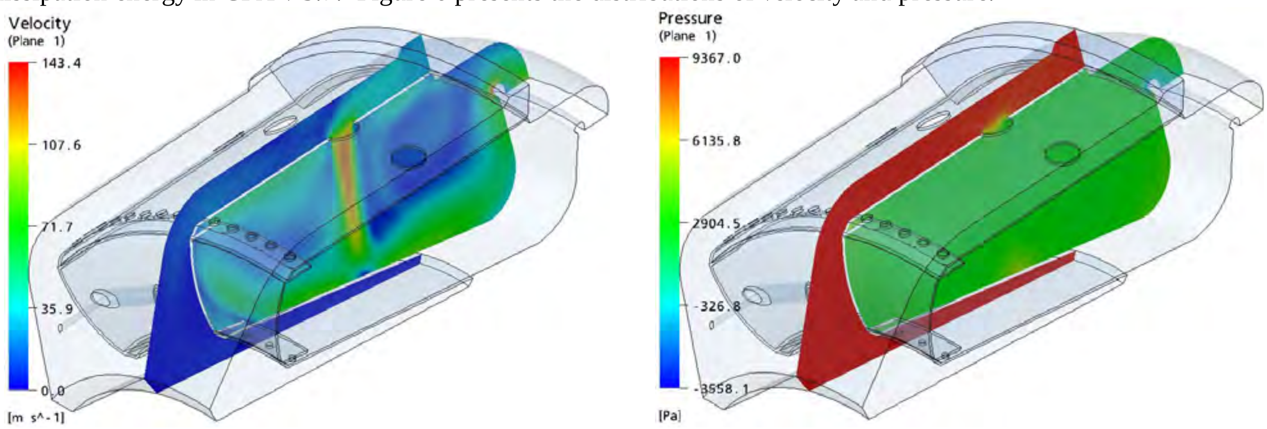
On the other hand, among the models for heat transfer by radiation, such as, **Rosseland** (to opaque domains), **P1**, **Monte Carlo**, **Discrete Transfer** (the three to semi-transparency domains) and **Spectral** (to full transparency domains), the **P1 Model** is adopted because permits to describe the radiation in domains with unitary emissive and unitary absorption in walls, which can be considered not catalytic (not affect the reactions).

Finally, among the models to simulate the NOx emission, such as, **Thermal Model of Zeldovich**, **Prompt Model of Fenimore** and **Fuel - NOx Model**, the adopted models are **Thermal Model of Zeldovich** and **Prompt Model of Fenimore**, because of emission from methane combustion depends on oxidation of atmosphere nitrogen in front of flame and the high velocity reactions, respectively, given by the following basic reactions:

- Thermal NOx:
  - $N_2 + O \leftrightarrow N + NO$
  - $N + O_2 \leftrightarrow NO + O$
  - $N + OH \leftrightarrow NO + H$
- Prompt NOx:
  - $N_2 + CH \leftrightarrow HCN + N$
  - $N_2 + CH_2 \leftrightarrow HCN + NH$

3. Analysis of Results

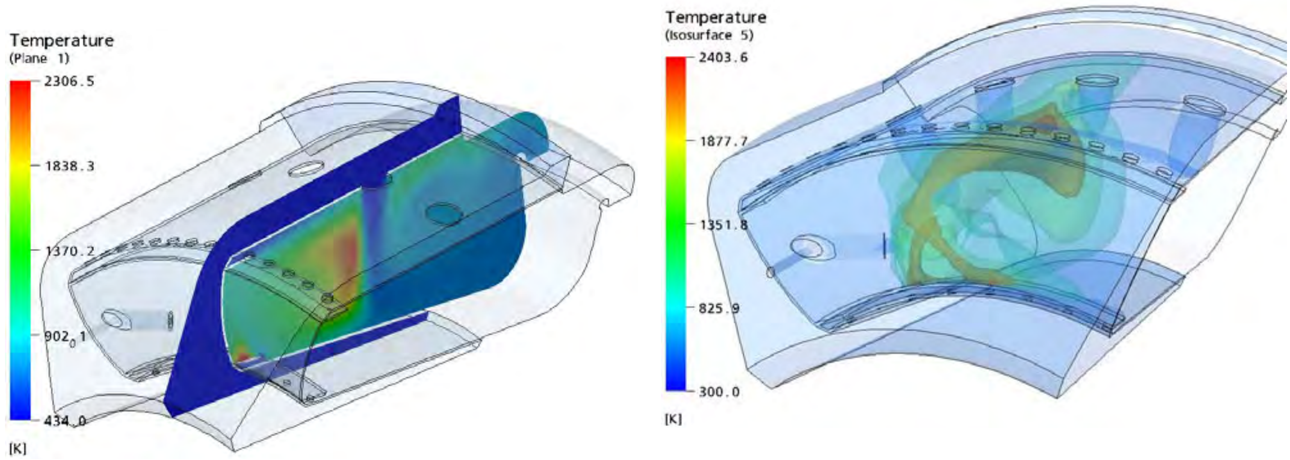
In general, the duration for the calculations is 3 hours and 14 minutes using a computer DELL with double processor of 3,4 GHz and 1GByte of RAM, considering a goal error equal to 10^{-5} for mass flow, velocity, turbulence and turbulent dissipation energy in CFX v 5.7. Figure 6 presents the distributions of velocity and pressure.



(a) (b)

Figure 6 – (a) Velocity contour; (b) Pressure contour

Figure 7 shows the distribution of temperature and the format of flame from iso-surfaces of temperature, with gradients equal to 200 °C, approximately, where is shown that the flame is formed from specific distance in relation to the nozzle with low dispersion. It is a characteristic for this kind of annular combustion chamber.



(a) (b)

Figure 7 – (a) Temperature contour; (b) Iso-surfaces of temperature

The distribution of iso-surfaces of temperature shows that the flame has dispersion in annular combustion chamber, as well as, the temperature is affected by rotary flow. This flow changes the position where the flame has its maximum temperature after the ignition. Maybe, it is necessary to change the dimensions of holes of dilution air.

Besides, in longitudinal surface that is shown in fig. 6 and 7, it is possible to determine the variation for velocity, pressure and temperature along the local axle from nozzles, which is parallel to main axle of combustion chamber, according to fig. 8. Table 2 presents the values for these greatnesses for this local axle obtained after the convergence.

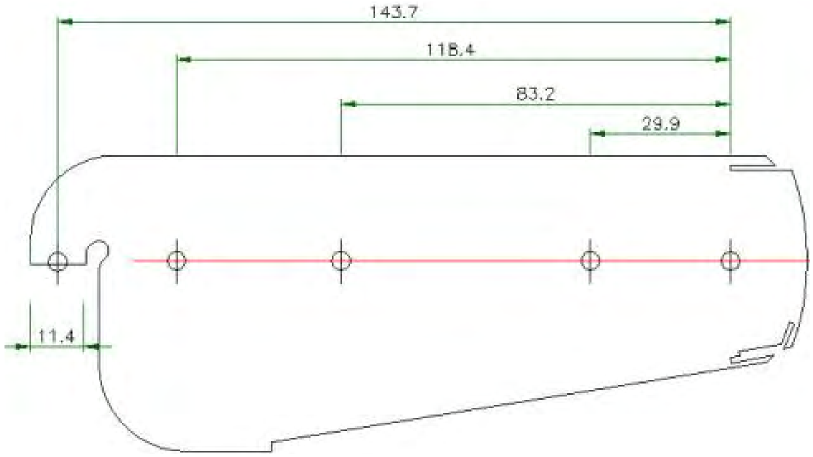


Figure 8 – Position of **Z** axle from nozzle

Table 2 – Values obtained after the convergence

Greatness	Symbol	Unit	Values				
Position in relation to nozzle	Z	mm	**0**	**30**	**83**	**118**	**144**
Average velocity of flow	V_{GAS}	m/s	52	45	32	27	62
Average pressure	P_{GAS}	Pa	2350	3200	3800	4100	0
Gas temperature	T_{GAS}	°C	1180	2306	912	810	434
Gas density	ρ_{GAS}	kg/m^3	0,35	0,18	0,36	0,42	0,52
Reynolds Coefficient	RE_{GAS}	1	4,71E+07	4,07E+07	2,90E+07	2,44E+07	5,61E+07

Besides, others important parameters that can be determined are the maximum temperature of flame and the temperature of gas in exhaustion, whose values are 2403 (°C) and 434 (°C), respectively, which are coherent with data from manufacturer.

From the data in table above, it is possible to determine the characteristic curves of operation for the model, as it is shown in next figures, whose reference is the **Z** axle in longitudinal surface (see Fig. 8).

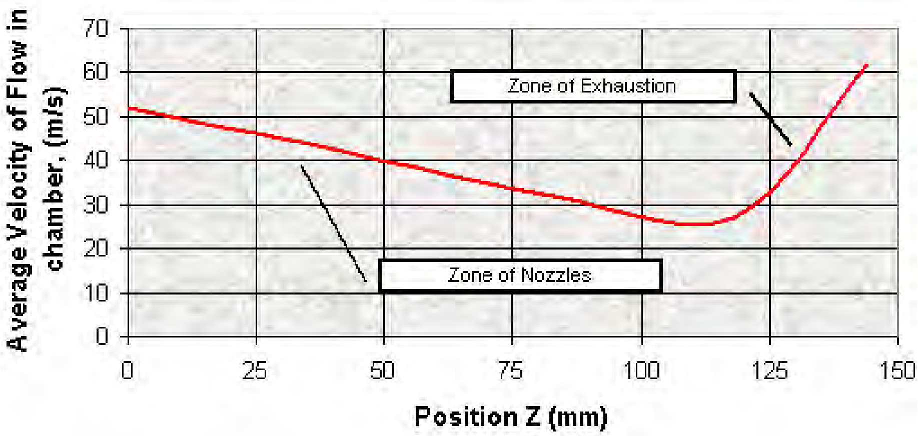


Figure 9 – Characteristic velocity curve

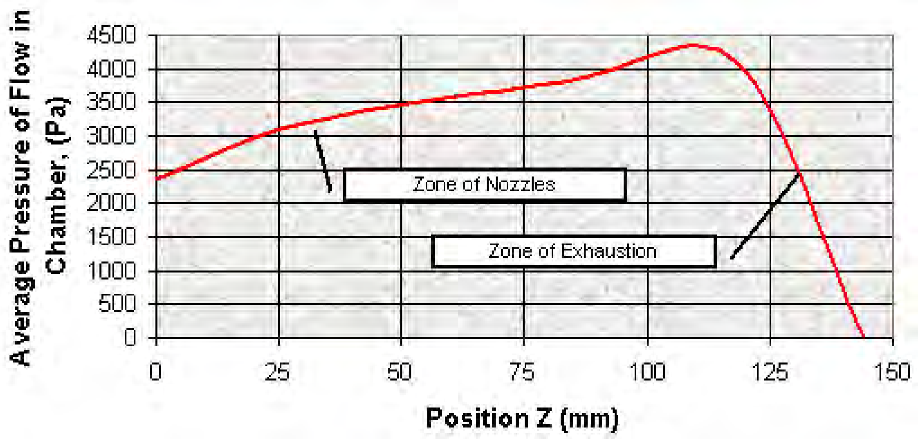


Figure 10 – Characteristic pressure curve

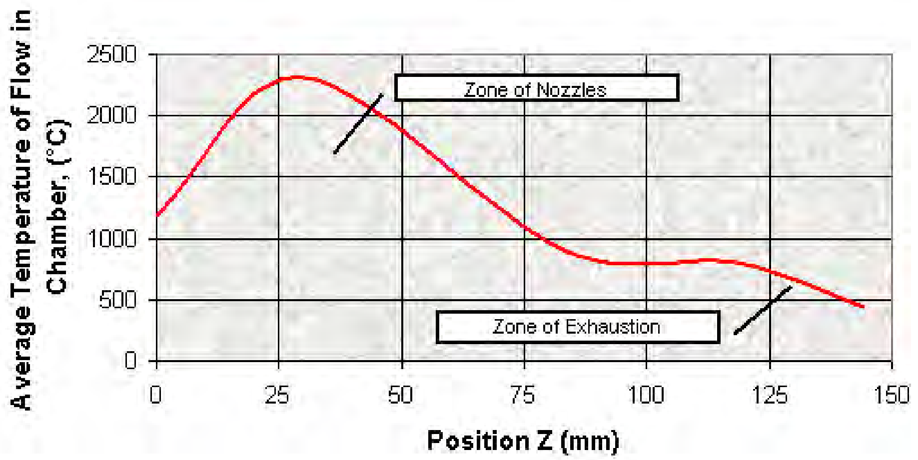


Figure 11 – Characteristic temperature curve

From fig. 11, it is possible to identify the maximum temperature of flame is near to the nozzles, but its position is affected by the secondary flow, which can be represented by high flow of air from dilution holes in lateral surface (for distances between 60 and 90 mm from nozzles) and by the rotary flow due to the inclination of nozzles in relation to the main axle on combustion chamber, respectively, figs 6 (a) and 7(a). Because of it, it is possible to avoid the approximation of hot gases in relation to the metallic walls, too.

The velocity and pressure values obtained in figs. 9 and 10, respectively, are affect by the turbulence zone near to the nozzles and exhaustion zones, see fig. 6(a).

Other characteristic that can be identified is the rotary flow in zone of nozzles, which can benefit the combustion, because it contributes to increase the residence time.

4. Conclusions

From the simulation about the annular combustion chamber model using methane, it is possible to identify:

- The high flow of air from dilution holes in lateral surface affects the position where it is possible to identify the maximum temperature of flame in relation to nozzles;
- The inclination of nozzles induces the rotary flow, whose main axle can be coincident to combustor axle and can increase the residence time for combustion, too;
- The methane that is used by combustion in model signs the dispersed flame, which obligates in new design of combustion chamber with the function to avoid the approximation of hot gases in relation to the metallic walls;
- According to the values studied by Dudebout et al. (2004) about the definition of flame in function to its origin, the flame generated is by Deflagration when the temperature reason between the maximum temperature of flame and the temperature of fuel in nozzle is between 4 and 16. From this, in current combustion chamber studied, this reason is near to 8, therefore, the flame is generated by deflagration, where the velocity of propagation for the flame is sub sonic.

In future jobs, it intends: first, to model this combustor with different inclination and dimension of nozzles; second, to model a different combustor volume to control the efficiency and the emission, considering a big time to attend the convergence condition; and third, to apply a pre mixtured model with 04 steps of chemical reaction, using methane or hydrogen.

5. References

AEA Technologies, 2003, "Tutorial do CFX v5 .7"., www.ansys.com/cfx.

ANEEL, 2003, Guia para Cálculo das Tarifas de Venda de Energia – Tarifação Praticada, www.aneel.gov.br

Allen J. W., 1998, "Low Nox Burner Designs, Proceedings of the American Power Conference", Vol. 60 – II, pp 869 – 874.

Chigier N. A ., Berr J. M., 1983, "Combustion Aerodynamics, Robert E. Krieger Publishing Company", Malabar, Florida, USA.

Dudebout R., Reynolds B., Khosro M. H., Integrated Process for CFD Modeling and Optimization of Gas Turbine Combustors, Proceedings of ASME Turbo Expo, Power for Land, Sea and Air, Vienna, Austria, GT 2004-54011, June 14-17, 2004.

Fuller E. J., Smith C. E., 1994, "CFD Analysis of a Research Gas Turbine Combustion chamber Primary Zone", 30th AIAA / ASME / SAE / ASEE Joint Propulsion Conference, Juny 27 – 29, Indianapolis, IN, USA.

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 and Exhibit, pp 11, Huntsville, Alabama.

Hamer A . J. and Roby R. J., 1997, "CFD Modeling of a Gas Turbine Combustion chamber Using Reduced Chemical Kinetic Mechanisms", AIAA 1997 – 3242, 33rd AIAA ASME / SAE / ASEE Joint Propulsion Conference and Exhibit, July 6 – 9, Seattle, WA, USA.

Hottel, H.C. e Sarofim, A.F., 1967, "Radiative Transfer", McGraw-Hill, New York, USA.

Lefebvre A . H. , 1995, "The Role of Fuel Preparation in Low Emission Combustion", ASME Journal of Engineering for Gas Turbines and Power, Vol 117, pp. 617-654.

Lefebvre A . H., 1987, "Gas Turbine Combustion", Mc-GrawHill Book Company, New York , USA.

Kuo K.K., 1986, "Principles of Combustion", John Wiley & Sons Edition, New York, EUA.

Lee D., Yeh C., Tsuei Y., Jiag W., Chung Y., 1990, "Numerical Simulation of Gas Turbine Combustion chamber Flows", 26th AIAA / ASME / SAE / ASEE Joint Propulsion Conference, July 16 – 18, Orlando, FL, USA.

Melick T. A . et al., 1999, "Burner Modifications for Cost Effective NOx Control", Proceedings of the American Power Conference, Vol. 61 – I, pp 478 – 482.

Melick T. A . et al., 1998, "Burner Modifications for Cost Effective Nox Control", Proceedings of the American Power Conference, Vol. 60 – II, pp 855 – 860.

Nickolaus D. A., Croker D. S., Smith C. E., 2002, "Development of a Lean Direct Fuel Injector for Low Emission Aero Gas Turbine", ASME

Technical Manual Overhaul, 2001, "Gas Turbine Engine Assembly", Part 100617-0, Model No. T-62T-32, Solar Division of international Harvester Company.

Yadigaroglu G. et al., 1998, "Numerical and Experimental Study of Swirling Flow in a Model Combustion chamber", Heat Mass Transfer Journal, Vol. 41, No. 11, pp. 1485-1497.

Wakabayashi T. et al., 2002, "Performance of a Dry Low Nox Gas Turbine Combustion chamber Designed with a New Fuel Supply Concept", Engineering for Gas Turbines and Power Journal, ASME, Vol. 124, pp. 771-775.

Willians. F., 1965, "Combustion Theory", Addison – Wesley Publishing Company Inc., London, UK.

Vandebroek L., Winter H., Berghmaus J, 2003, "Numerical Study of the Auto Ignition Process in Gas Mixtures Using Chemical Kinetics", Heat Mass Transfer Journal.

Yadigaroglu G. et al., 1998, "Numerical and Experimental Study of Swirling Flow in a Model Combustor", Heat Mass Transfer Journal, Vol. 41, No. 11, pp. 1485-1497.

6. Responsibility notice

The author(s) is (are) the only responsible for the printed material included in this paper.