Proceedings of the 3rd IASME/WSEAS Int. Conf. on HEAT TRANSFER, THERMAL ENGINEERING AND ENVIRONMENT, Corfu, Greece, August 20-22, 2005 (pp317-325)

MODELING OF ANNULAR COMBUSTION CHAMBERS USING CFD

Harley Souza Alencar Helcio Francisco Villanova Marco Antonio Rosa do Nascimento
Departmento de Engenharia Mecânica
Universidade Federal de Itajubá
Benedito Pereira dos Santos Av., n° 1303, Zipcode 37500-000, Itajubá, Minas Gerais
BRAZIL
haarley@terra.com.br *patricia.foroni@itelefonica.com.br* *marcoantonio@nest.unifei.edu.br*

Abstract: - The main goal of this job it is show two combustion models applied for an annular combustion chamber for small gas turbines, considering aspects about the speed, pressure and temperature, which can be used to feature the flame behavior and its effects in efficiency and pollution emission, once that had presented satisfactory results based on others jobs using simulations for tubular combustor chamber for small gas turbines.
The geometric domain is similar to the prototype of small combustion chamber developed by Harvester Company, T-62T-32 model. The thermal-aerodynamic simulation is executed by CFD using the combustion models: Eddy Dissipation Model – EDM and Flamelet Model. The turbulence Model is the RNG K-ε turbulence Model. The Radiation Model is P1 model using the NOx emission model of Zeldovich for fuel methane with 2 steps of chemical reaction.
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.

Key Words: - Combustion Models, Modeling, CFD Simulation and Small Gas Turbine

1 Introduction

The use of gas turbines for the regional development has been increased gradually in early years. In, particular in Brazil, according to a research from National Agency of Electric Energy [2], the participation of this energy source is bigger than 10% and attends from the small power plants with power until 150 kW to the big power plants with power until 100 MW.
Besides, for all designers that work with gas turbines, the development of computers more quick and mathematical models more robust, it has permit to design combustion chambers more efficiency and with low pollution emissions, in a general view, increase its economic feasibility.
The simulation permits to get more quick results for everything that is done to change some characteristics for combustion chambers, mainly for basic parameters: the geometry and fuel.
Because of it, there are two lines of research: one of them, it has the goal to improve the geometry of chambers and analyze the flow behavior in relation to, for instance: volume; distance, position and quantity of holes and nozzles; position of flow, and others. For this case, some jobs have contributed to archive this goal, such as, [3], [21], [30] and [31].
The second line of research has the goal to improve the physical proprieties of fuels, using or not using additives and the analysis of flame behavior based in chemical kinetics and thermo-aerodynamic behavior. In this case, some jobs have importance to archive this goal, such as, [18], [20] and [32].
Besides, others jobs has had importance because of different tools using alternative mathematical models for tubular combustion chambers. Among the tools, it is possible to emphasize the use of Computational Fluid Dynamics – CFD, using Combustion Models, Turbulence Flow Models, Radiation Models and others, which have had satisfactory results.
In [10], [17] and [22] were executed the first simulations using CFD in cylindrical combustors for bi-dimensional and three-dimensional geometry models, using the turbulence models K-ε, RNG K-ε and Reynolds Stresses Model – RSM, and the combustion models Fast Chemistry with -- Probability Density Function – PDF Model and Flamelet Laminar Model – FLM.
In [9] and [11], it was introduced others combustion models, such as, Eddy Dissipation Model – EDM, with models of NOx emission of Fenimore and Zeldovich, for three-dimensional geometry models.

In [13] were applied these models in a tubular combustion chamber for 2-D and 3-D models and concluded that the Eddy Dissipation Model and Probability Density Function have good precision when are compared with experimental results.
Others jobs show that CFD is an important tool, such as, in [14], [19] and [23], where it is shown that turbulence RSM model can esteemed with satisfactory precision the penetration of air in dilution holes for tubular chambers than K-ε Model.

CFD in companies have presented many applications, such as, in [8] and [16] that contributed for Honeywell Engines, Systems & Services in development of a specialist system based on CFD known as ACT in analysis of annular and tubular combustion chambers. Besides, in Cristina (2004) was analyzed the behavior of emissions and the flame in different combustion chambers, considering the aspect about the auto ignition in fuel.

In [4], it is analyzed the behavior of flame in an annular combustion chamber for a small gas turbine developed by Harvester Company, T-62T-32 model T, using Eddy Dissipation Model – EDM. Besides, it is shown that this model has satisfactory results depend on specified conditions, such as, a not pre-mixed flow between air and fuel before the ignition and permanent flow with rotary secondary flow due the position of bending nozzles in relation to the axial axle.

In this job, it is applied other combustion model, the Flamelet Model in the same geometry. So, the goal here is presented a comparison between these two combustion models, considering the aspects of pressure, speed and temperature in next sections.

2 Problem Formulation

The methodology applied in this job consists to solver the physical problem relative to the behavior of flame in a model of annular combustor 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, where the physical domain is discretized by volume elements of type tetrahedrical or quadrilateral. The computational tool applied is the CFX® v 5.7 from ANSYS [1].
In the figure 1, it is presented the basic scheme to the modeling of combustor.

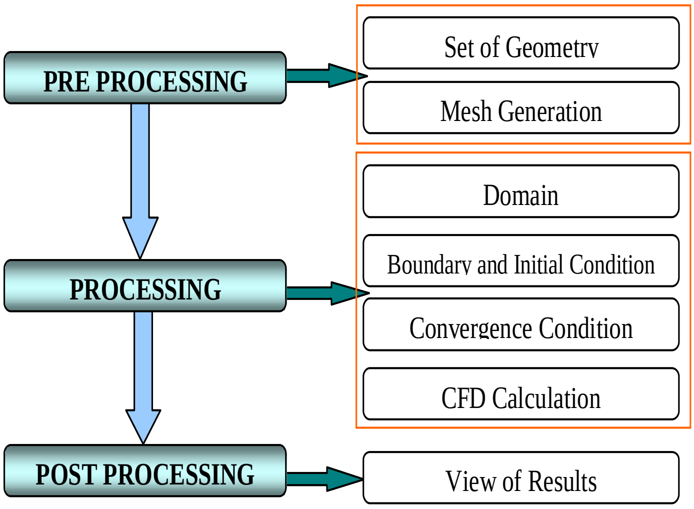

Fig. 1 – Basic scheme to the modeling of combustor

In **Pre-processing**, the geometry model is generated from prototype of an annular combustor, which has 6 nozzles with inclination equal to 60 grades in relation to the axial axle of combustor. 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.
This small gas turbine is designed to work with diesel or kerosene, with pressure 275790 Pa, temperature 25°C, Flow 29 kg/s with consumption 0,85 kg / MWh. The operational air from compressor has pressure 206843 Pa, temperature 52°C and flow 0,75 kg/s. Besides, the maximal temperature in outlet of combustion chamber is 637,9°C, the nominal temperature in outlet is 368 °C, and the maximal temperature of gas in exhaustion after turbine is 493,2 °C.
In combustor, the primary air is mixtured with fuel in nozzle with high pressure and speed. Depend on capillarity and viscosity, the fuel can be drafted by air, due to the decrease of relative pressure inside of nozzle. The figure 2 shows the prototype that is modeled.

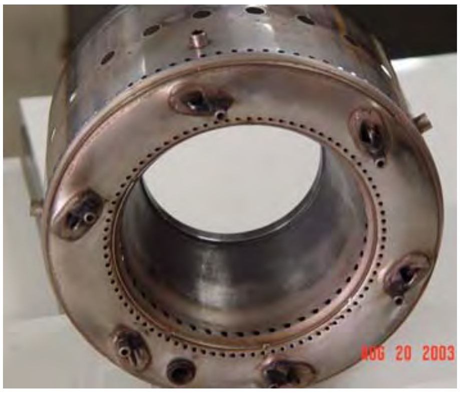

Fig. 2 – Combustor Prototype

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

The flame is rotary and a little diffuse, whose maximum reach is not near the metallic walls. From this way, this combustor is compact, soft with easy transportation and maintenance.

Besides, the works from [10] and [26] show that it is possible to get satisfactory results for a model that corresponds to 1 / 6 from original volume of prototype, considering the symmetry in relation to the axle of combustor. From this way, it is possible to get a more simply model where the mesh is generated with speed and good discretization (refinement) to capture all physical details in propagation of flame, including the convection.

Hence, in this work the model of combustor is 1 / 6 of volume for the prototype, whose adopt fuel is the methane. In figure 3, it is shown this tested model and figure 4 details for CFD in CFX.

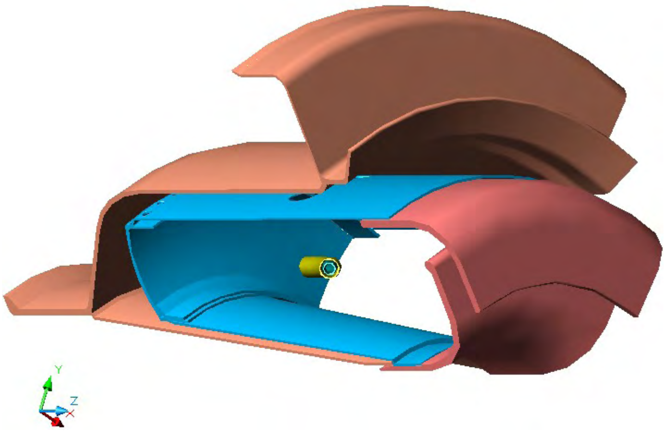

Fig. 3 – Geometry model tested

Fig. 4 – Combustion Chamber tested in details

The mesh generated is non-structured with 1915508 tetrahedral and prismatic elements and 4666141 nodes. In figure 5, it is shown a part of this mesh in a longitudinal surface paralleled to the axial axle of combustor, in a region near to the outlet of nozzle. In figure 6, it is shown details of mesh.

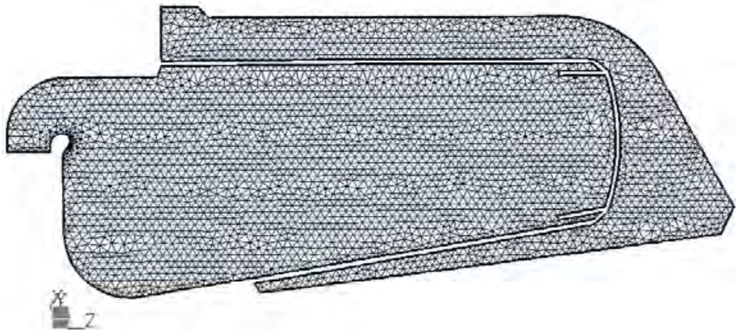

Fig. 5 – Lateral view of mesh

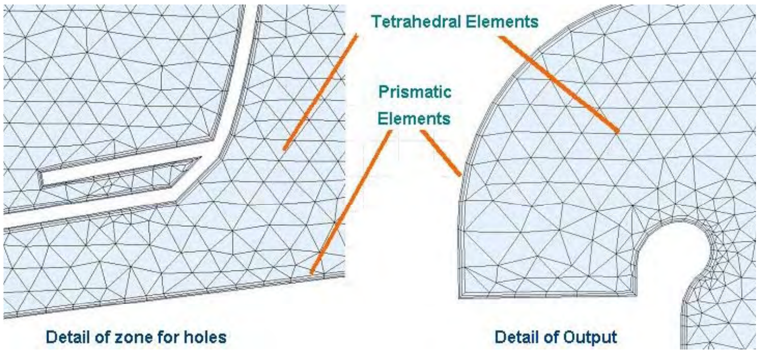

Fig. 6 – Details of mesh: (a) in region near to the nozzle; and (b) in outlet of hot gas

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

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \,.\, U) = 0 \qquad (1)$$

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

$$\frac{\partial T}{\partial t} + U \bullet \nabla T = \alpha \,.\, \nabla^2 \; T + \overline{Q} + \varphi \qquad (3)$$

Where: ρ is the density of fluid, in [kg/m^3]; **U** is the speed, in [m/s]; **t** is the time, in [s]; **p** is the relative pressure, in [Pascal]; **Ui, Uj** are the speed in direction **xi** and **xj**, respectively, in [m/s]; **i** and **j** is the Index 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**), [5]:

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

Where: ν is the emission frequency; **r** is the position vector; **s** is the direction vector; **s´** is the distance traveled by radiation; **Kav** is the absorption coefficient; **Ksv** is the reflexion coefficient; **Ib** is the intensity of emission in black body; **In** is the intensity of spectral radiation that depend on the position **r** and the direction **s**; **Ω** is the solid angle; **Φ** 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 that are known, 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 that can be used to describe the flow in combustion. More details about these turbulence models are in jobs made by [30] and [31].
Besides, among the combustion models such as, **Eddy Dissipation Model – EDM**, **Finite Rate Chemistry – FRM**, and **Laminar Flamelet Model – LFM**, the **EDM** and **LFM** are adopted because it can describe combustion reactions with or without pre-mixture, such as the current geometry model in this work.

In **EDM**, the mechanism of reaction can be described by the following steps:

a) Phase 1: the main reaction is given by expression:

C_XH_Y + a (O_2 + 3.76 N_2) ➔ XCO_2 + (Y/2) H_2O + 3.76 a N_2

b) Phase 2: the reaction is described using stoichiometric air:

C_XH_Y + a (O_2 + 3.76 N_2) ➔ X CO +
+(Y/2) H_2O + 3.76 a N_2 + ½ O_2

CO + ½ O_2 ➔ CO_2

From this way, the reaction rates for reactants and products are, respectively (kinetic chemical):

$$R_K = A\,.\,\frac{\varepsilon}{k}\,.\,\min\left(\frac{[I]}{\nu^{*}_{KI}}\right) \quad (5)$$

$$R_K = A\,.\,B.\,\frac{\varepsilon}{k}\,.\left(\frac{\sum_P [I].\,W_I}{\sum_P \nu^{**}_{KI}\,.\,W_I}\right) \quad (6)$$

Where **[I]** is the molar concentration of each species between the reagents; **A** is a proportionality constant, that depends on the physical properties of the reactants and if there is or not pre mixture and the conditions of reactions with or not free radicals; **P** tracks all the products generated in reaction **K**; $\mathbf{W_I}$ is the molecular weight of each component of the product; **B** is a numerical parameter that indicates if the simple reaction of or multiple stage. If **B** is negative, the formation of the products for determined reaction **K** is not performed; **ε** is the dissipation energy due the turbulence **k** is the turbulence in flow; and ν is the stoichiometric coefficient for reactants and products.

Besides, in **LFM**, the mechanism of reaction can be described using the **Probably Density Function – PDF**, [24], whose main advantage is in the fact of not have the necessity of the modeling of the not linear term of the chemical power source in the **Equation of Conservation of Energy**. On the other hand, the term represented for the variation of the static pressure and the term represented for the diffusion in the **Equation of Conservation of the Momentum** must be modeled.
In [24] it is shown that the **PDF** model for **LFM** is given by the following expression:

$$P\left[\left|\frac{1}{n}\,.\sum_{j=1}^{n}\delta_j - m\right| < \frac{3\,.\,\nu}{\sqrt{n}}\right] \approx 0{,}997 \quad (7)$$

Where **m** is the hope of **n** arbitrary variables **δJ** chosen in **x**, with standard deviation ν.

Besides, 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** and **Discrete Transfer** are adopted because can describe the radiation in domains with unitary emissive and absorption in walls, as well as, not catalytic, that not affect the reactions. More details about these radiation models are in [12], [25] and [28].

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 speed 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:
  $N_2 + CH \leftrightarrow HCN + N$
  $N_2 + CH_2 \leftrightarrow HCN + NH$

More details about these NOx models are in [18] and [28].

3 Validation of CFD Calculation

In general, all CFD calculations shall be performed in comparison to the experimental results for simple geometries, in function to guarantee the right result with quality in relation to the behavior of physical phenomena studied for every geometries.

A good example for it, which can be applied for combustion chambers, is the CFD validation for radiation calculation in a geometry proposed by [27], which is used to validate the numerical method developed by [29], who used it in a particular case of annular combustion chamber.

This geometry domain consists of a little cylinder with length 5 m and diameter 2 m. The walls have temperature of 773 K and an emissivity of 0.8. The gas contains a hot region at a temperature of 1700 K and an absorption coefficient of 0.6 m^{-1}. The remainder of the cylinder is at a temperature of 1100 K and an absorption coefficient of 0.05 m^{-1}, as shows figure 7.

The CFD calculation uses mesh with 89782 tetrahedral elements. The convergence condition has 50 iterations with goal error equal to 10^{-4} for heat transfer. Besides, the radiation model is **Heat Transfer Discrete Model**, where the radiation source can be discretized by 8, 64 and 128 rays with enough dimension for that the average refraction is smaller than emission, because of work fluid is a hot gas without particles (photons). The adopted model to discrete the spectrum is **Gray Model** because the spectrum is uniform. Besides, The **Scattering Model** is considered null because that the term in transport equation for energy due the particles (photons) is not necessary and the domain is isotropic.

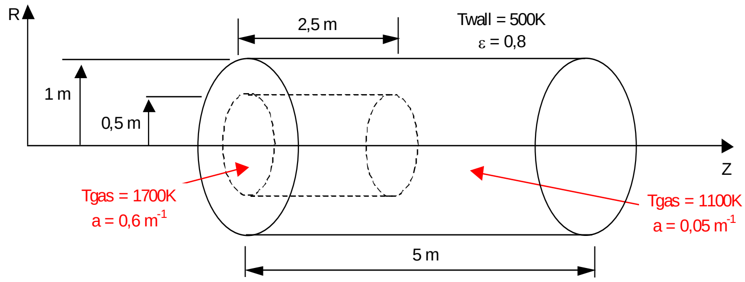

Fig. 7 – Geometry of [27] and boundary conditions for radiation calculation

In figure 8, it is presented the intensity of radiation in non-dimensional form obtained by experimental test and by CFD calculation, where **Io** is the maximum radiation intensity and equal to 80000 W / m^2 and **L** is the length of cylinder and equal to 5 m.

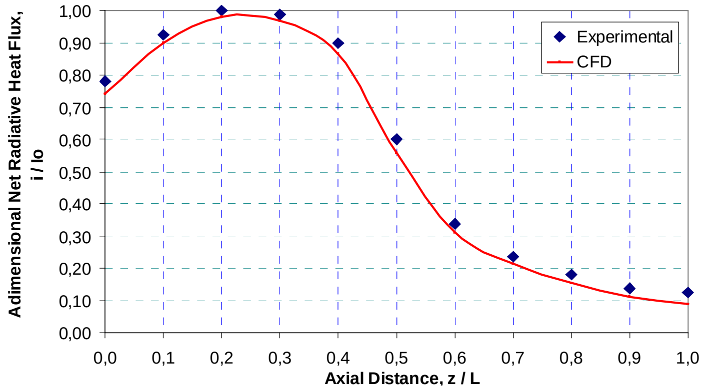

Fig. 8 – Non-dimensional Radiation Heat Flux for Axial Distance

The values of experimental and numerical tests are near and the CFD calculation permits to get satisfactory results in a simple geometry for radiation calculation. The maximum difference between the results is 4 % in non-dimensional axial distance 0.5.

From this, the CFD can be used to study more complex geometries, where the heat transfer is done by radiation too, like as, in the combustion cases.

4 Solution of Problem

In general, the duration for the calculations is 3 hours and 14 minutes. In figures from 9 to 11, it is presented the sources of pressure, speed and temperature, for **Eddy Dissipation Model**, where is shown that the flame is formed from specific distance in relation to the nozzle with low dispersion.

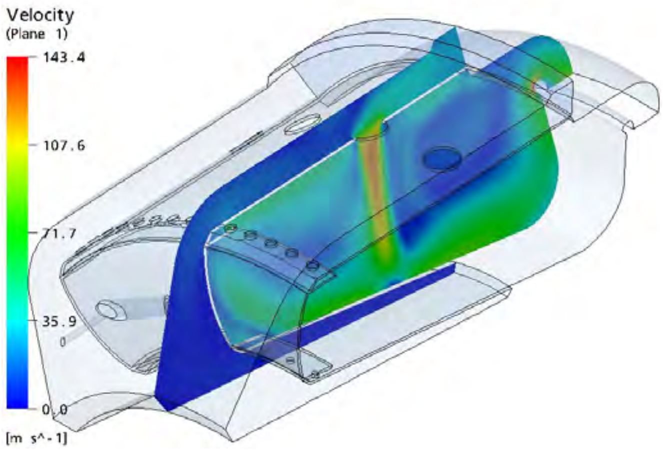

Fig. 9 – Speed contour

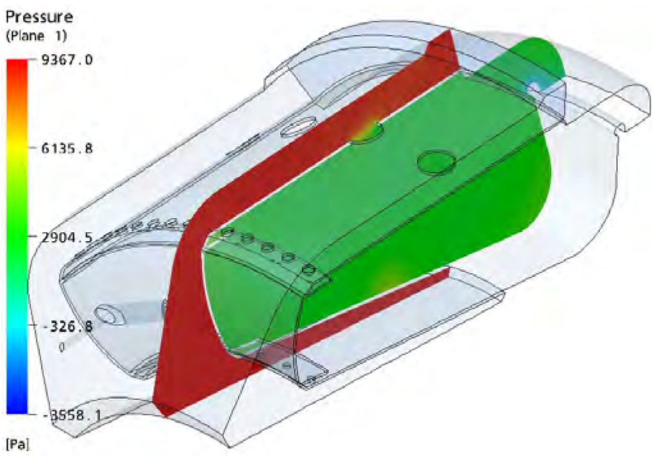

Fig. 10 – Pressure contour

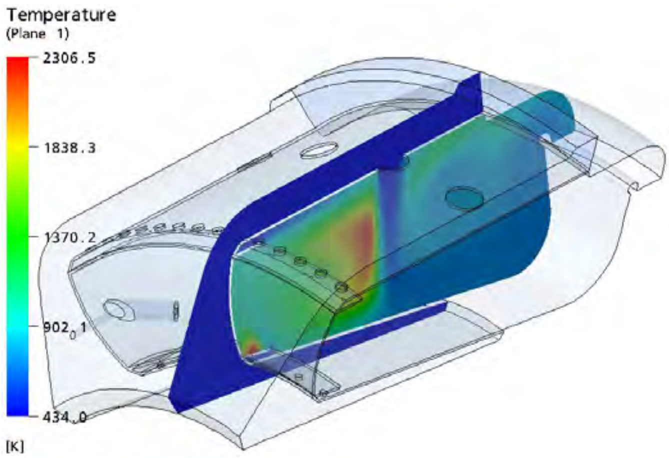

Fig. 11 - Temperature contour

In figure 12, it is presented the profile of flame from iso-surfaces of temperature, with gradients equal to 200 °C, approximately.

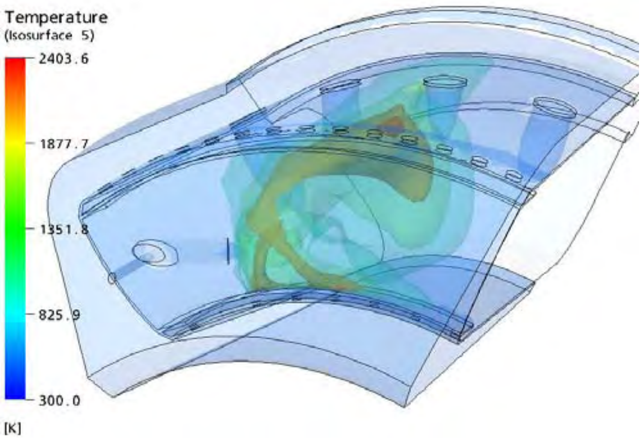

Fig. 12 – Iso-surfaces of temperature, volume of flame

In figures from 13 to 15, it is presented the sources of pressure, speed and temperature, for **Laminar Flamelet Model**.

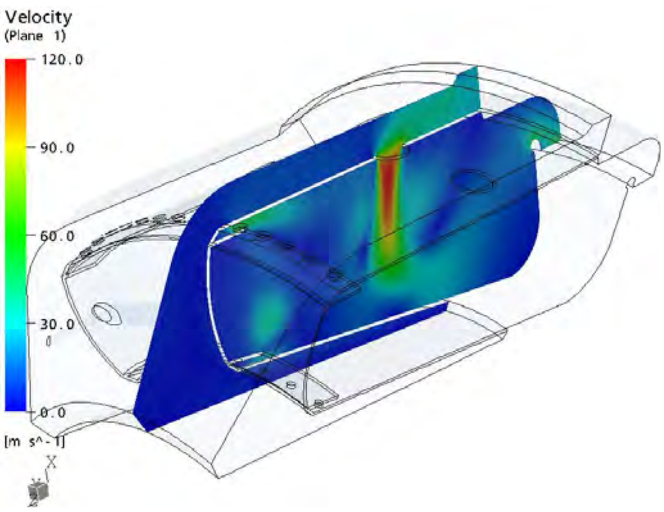

Fig. 13 – Speed contour

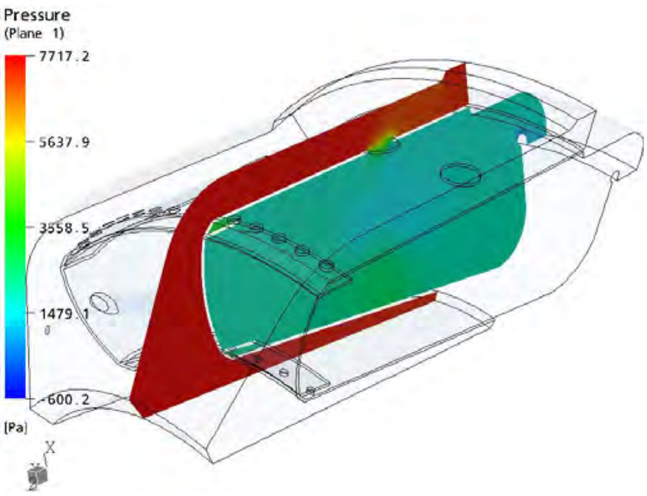

Fig. 14 – Pressure contour

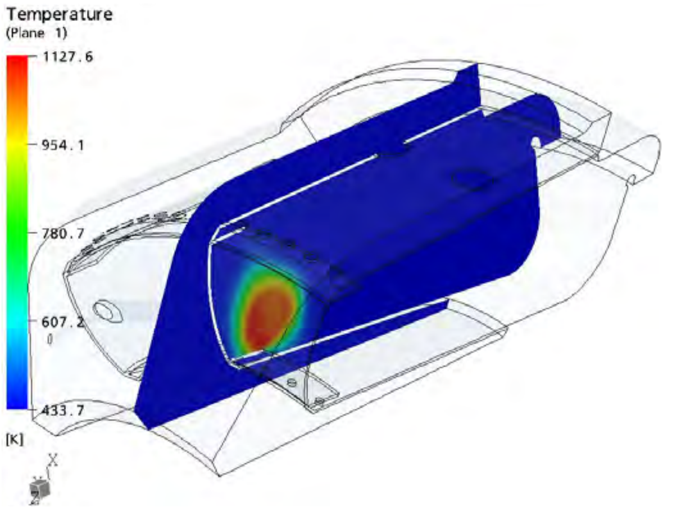

Fig. 15 - Temperature contour

In figure 16, it is presented the profile of flame from iso-surfaces of temperature, with gradients equal to 200 °C, approximately.

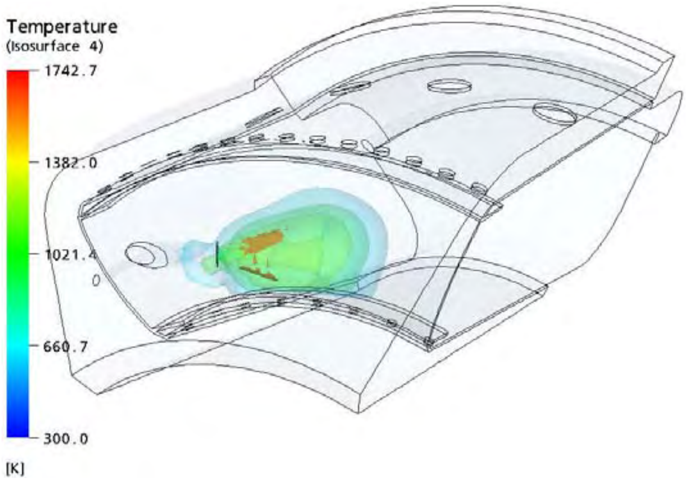

Fig. 16 – Iso-surfaces of temperature, volume of flame

From figure 16, it is seen that the flame is concentrated next to nozzle, while in figure 10, the flame is more dispersed.

The experimental visual inspection detects that the flame in prototype is dispersed due to secondary rotary flow in relation to main axle.

While the Eddy Dissipation Model has the convergence to 100 iterations with goal error 10^{-5} in relation to speed, mass and turbulence dissipation energy, the convergence condition to Laminar Flamelet Model considered a total time of 10 seconds. Although, for more time, the laminar Flamelet model describe flame volume with the same format next the nozzle, with small deviation due to secondary rotary flow in annular chamber.

Besides, the Eddy Dissipation Model permits to detect that the flow from a specific distance in relation to nozzle, can change 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 to avoid the maximum temperature occurs near to the lateral walls.

In relation to the longitudinal surface that is shown in figures from 9 to 11 and from 13 to 15, it is possible to determine the variation for speed, pressure and temperature along the **Z** axle from nozzle, as it is presented in figures 17, 18 and 19.

In figures 17, 18 and 19, it is possible to identify zones, where there is a high intensity of recirculation.

These zones are in different positions in combustor, where the pressure fluctuations and temperature are, as it is shown for distances until 80 mm (combustion zone) from nozzle.

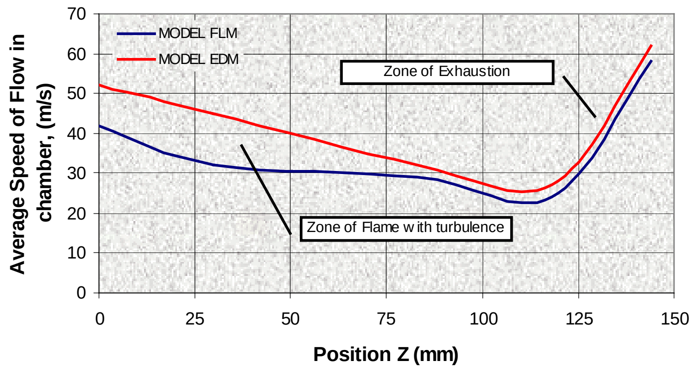

Fig. 17 – Characteristic speed curve for two combustion models

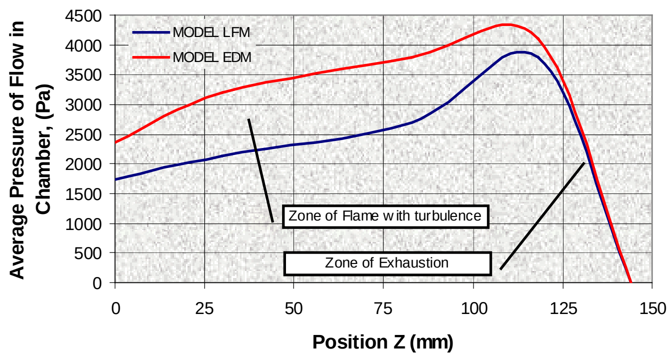

Fig. 18 – Characteristic pressure curve for two combustion models

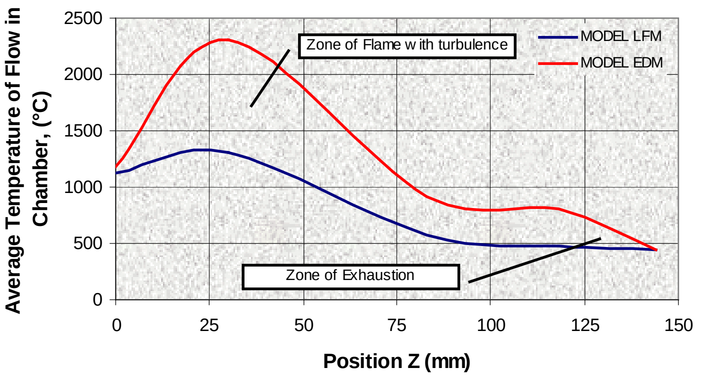

Fig. 19 – Characteristic temperature curve for two combustion models

But, the formation of recirculation in zone near to nozzle can benefit the combustion, because it contributes to increase the residence time. Besides, it is identified that the airflow from lateral holes cannot be enough to avoid the approximation of hot gases in relation to the metallic walls in combustor. This sign shows that the prototype of combustor using methane, can work in flammability limit and mechanic resistance.

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, for Eddy Dissipation Model, respectively, which are coherent with data from manufacturer, without problems in metallic walls due to the satisfactory efficiency of dilution air. For Laminar Flamelet Model, these temperature values are 1270 °C and 434 °C, respectively.

5 Conclusion

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

- Zones where there is high intensity of recirculation, that affect the residence time and can benefit the full combustion;
- The sloping of nozzles induce the rotary flow, whose main axle can be coincident to combustor axle and can increase the residence time for combustion;
- The methane, that is used by combustion in model, signs the dispersed flame, which obligates in new design of combustor with the function to avoid the hot gases near to the metallic walls;
- The origin of flame in annular combustion model tested by CFD can be defined by deflagration, because the temperature reason between the maximum temperature of flame and the temperature of fuel in nozzle is near to 8, according to [18], where this temperature reason for deflagration of flame is defined between 8 and 21; and
- These results obtained by CFD can be considered satisfactory in relation to the true behavior for prototype based on CFD validation for radiation calculation, which was shown in section 3.

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.

References:

[1] AEA Technologies, *CFX v5.7 Tutorial*, www.ansys.com/cfx, 2005.

[2] ANEEL, *Guide for Calculation of the Tariffs for Energy Sale – Practice Tariffs*, www.aneel.gov.br, 2003.

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

[4] Alencar H. S., Villanova H. F., Antonio Rosa M. N., *Analysis of Flame Behavior in Small Combustion Chambers Using CFD*, Proceedings of COBEM 2005, 18th International Congress of Mechanical Engineering by ABCM, Ouro Preto, Brazil, November 6-11, 2005.

[5] Beer, J.M., Foster, P.J. and Siddall, R.G., *Calculation Methods of Radiative Heat Transfer*, HTFS Design Report No. 22, AEA Technology (Commercial), 1971.

[6] Chigier N. A ., Berr J. M., *Combustion Aerodynamics*, Robert E. Krieger Publishing Company, Malabar, Florida, USA, 1983.

[7] Cristina M. C., Tuccilo R., *Comparing Different Solutions For The Micro-Gas Turbine Combustor*, Proceedings of ASME Turbo Expo, Power for Land, Sea and Air, Vienna, Austria, GT 2004-53286, June 14-17, 2004.

[8] 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.

[9] Fuller E. J., Smith C. E., *CFD Analysis of a Research Gas Turbine Combustor Primary Zone*, 30th AIAA / ASME / SAE / ASEE Joint

Propulsion Conference, Indianapolis, IN, USA, June 27 – 29, 1994.

[10] Gosselin P., DeChamplain S. K., Kretschmer D., *Three Dimensional CFD Analysis of a Gas Turbine Combustor*, 36th AIAA / ASME / SAE / ASEE Joint Propulsion Conference and Exhibit, pp 11, Huntsville, Alabama, 2000.

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

[12] Hottel, H.C. e Sarofim, A.F., *Radiative Transfer*, McGraw-Hill, New York, USA, 1967.

[13] Jiang L. Y., Campell I., *A Critical Evaluation Of Nox Modeling In A Model Combustor*, Proceedings of ASME Turbo Expo, Power for Land, Sea and Air, Vienna, Austria, GT 2004-53641, June 14-17, 2004.

[14] Koutsenko I. G., Onegin S. F., Sipatov A . M., *Application of CFD-Based Analysis Technique For Design And Optimization of Gas Turbine Combustors*, Proceedings of ASME Turbo Expo, Power for Land, Sea and Air, Vienna, Austria, GT 2004-53398, June 14-17, 2004.

[15] Kuo K.K., *Principles of Combustion*, John Wiley & Sons Edition, New York, EUA, 1986.

[16] Lai M. K., Reynolds R. S., Armstrong J., *CFD-Based, Parametric, Design Tool For Gas Turbine Combustors From Compressor Deswirl Exit To Turbine Inlet*, Proceedings of ASME Turbo Expo, Power for Land, Sea and Air, Amsterdam, The Netherlands, GT 2002-30090, June 03-06, 2002.

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

[18] Lefebvre A . H., *Gas Turbine Combustion*, Mc-GrawHill Book Company, New York , USA, 1987.

[19] Lyckama N. J. A ., Komen E. M. J., Hermanns R. T. E., Goey L. P. H., Van Beek M. C., Verhage A . J. L., *CFD Modeling Of Biogas Co firing In A Gas Turbine*, Proceedings of ASME Turbo Expo, Power for Land, Sea and Air, Amsterdam, The Netherlands, GT 2002-30103, June 03-06, 2002.

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

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

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

[23] Parente J., Anisimov G. M. V. V., Croce G., 2004, *Micro Gas Turbine Combustion Chamber Design And CFD Analysis*, Proceedings of ASME Turbo Expo, Power for Land, Sea and Air, Vienna, Austria, GT 2004-54247, June 14-17, 2004.

[24] Peters N., *Turbulent Combustion*, Cambridge Monographs on Mechanics, Cambridge University Press, UK, 2000.

[25] Raithby, G.D.,. *Equations of Motion for Reacting, Particle-Laden Flows*, Progress Report, Thermal Science Ltd., EMR, 1991.

[26] Rizk N. K., Monglia H. C., *Three dimensional Analysis of Gas Turbine Combustor*, Jounal of Propulsion and Power, Vol. 7, No. 1, 1991.

[27] Shah, N. G., 1979, *New Method of Computation of Radiant Heat Transfer in Combustion Chambers*, PhD Thesis, University of London, UK, 1979.

[28] Siegel, R e Howell, J.R., *Thermal Radiation* Heat Transfer, Heat Mass Transfer Journal, 1997.

[29] Stuttaford P. J., *Preliminary Gás Turbine Combustion Design Using a Network Approch*, PhD Thesis, Cranfield University, USA, 1997

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

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

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