CFD SIMULATION AND THE DYNAMIC UTILIZATION ANALYSIS IN A GAS TURBINE COMBUSTION CHAMBER REMARK

Engr. Nnadikwe Johnson[1], Engr. Ewelike Asterius Dozie[2], Ibe Raymond Obinna[3], Okereke Godson[4]

[1] *H.O.D in Department of Petroleum and Gas Engineering, Imo State University, Nigeria*

[2] *H.O.D in Agricultural Engineering, Imo State University, Nigeria*

[3] *Lecturer in department of Chemical Engineering, Imo State Polytechnic, Nigeria*

[4] *Lecturer in department of Building, Federal University of Technology Owerri, Imo State, Nigeria*

ABSTRACT

The gas turbine combustion chamber is among the most crucial components to design, since it must maintain a stable functioning throughout a broad variety of air/fuel ratio and load conditions. The Remark area is the most critical physical characteristic in the design of a combustion chamber because of its significant influence on other dimensions. Based on the constraints placed on both chemical processes and aerodynamics, this value must be determined in order to determine the maximum pressure drop in the combustion chamber that may be tolerated. So, the purpose of this work is to examine the effects of the Remark area on the velocity profile, the temperature distribution, the mixing process, and the flame behavior in the combustion chamber, depending on the Remark area employed. Based on the calculations made by Lefebvre [5] for a 600 kW gas turbine engine thermodynamic cycle in GateCycle®, these numerical evaluations were carried out using the ANSYS CFX software. A few tweaks to the Remark area computed by Lefebvre [4] improve the burning process and the behavior of the flame, hence achieving better results.

Keywords: *Combustion chamber, Gas turbine, Can-combustion, CFD simulations, ANSYS CFX®*

1. INTRODUCTION

It has become more common for gas turbine engines to be used for the production of electricity, compared to other technologies. The gas turbine business is estimated to earn $118.4 billion between 2005 and 2014, according to a research issued by the Forecast International consultancy. Gas turbine engines using a variety of fuels have been examined in a number of research institutions across the globe in light of worldwide regulations to minimize pollution and the impending depletion of fossil fuels [4]. Gas turbines' combustion chambers have become more important in this context because of their potential to increase combustion efficiency and reduce emissions [3].

With the rise of renewable energy sources, Pavlas and colleagues [9] have developed options that do not entirely modify the architecture of the original system. As a result of the varied properties, compositions, and calorific value of off-design fuel in gas turbine applications, several adjustments are necessary. Bonzano and Pollarolo [1] stated that gas turbines need low heating value fuels to operate. The primary effects of these fuels on combustion system design have been studied in depth.

Based on Neilson [8's] research on gas turbines that can run on low-heat-value fuels, GE Energy® has developed the LM2500 gas turbine.

In a micro-gas turbine, Cameretti et al. [2] looked at the use of liquid and gaseous biofuels. As a means of reducing the generation of NO in the combustion process, a revised fuel injector shape and placement were paired with a changed position of the pilot injector to achieve flameless combustion.

As shown, various studies have been carried out to emphasize the feasibility of fuel substitution after just a few alterations to the basic design, according to the characteristics of each fuel. If you're going to be changing the fuel mix, you'll need equipment that can help you do it in a simple and efficient manner. However, since Lefebvre's initial Remark area [6] is the most critical parameter in the combustion chamber, this work intends to explore the flame behavior using numerical simulations done by ANSYS CFX®.

2. REMARK AREA

An essential physical design characteristic in a gas turbine combustion chamber is referred to as the "Remark area," which may affect other dimensions.

At least one compromise must be made (1) between the restrictions imposed by chemical reactions and aerodynamics and the maximum pressure loss permitted, according to Lefebvre [6].

As a result, the calculations take into consideration both aerodynamics and chemical processes, with the larger surface area being preferred:

$$A_{ref} = \frac{\pi D_{ref}^2}{4} \qquad (1)$$

2.1. Approach Based on Aerodynamics

The major combustion chamber dimensions are derived from pressure losses using this strategy. The pressure losses may be correlated with [6] a number of different variables:

$$\frac{\Delta P_{3\to4}}{P_3} = \frac{\Delta P_{3\to4}}{q_{ref}} \frac{k}{2} Ma_{ref}^2 \qquad (2)$$

Or, replacing Ma_{ref} and after some manipulations (2):

$$\frac{\Delta P_{3\to4}}{P_3} = \frac{R_a}{2} \frac{\Delta P_{3\to4}}{q_{ref}} \left(\frac{\dot{m}\sqrt{T_3}}{P_3 A_{ref}} \right)^2 \qquad (3)$$

From (3) is possible to get the Remark area value, $A{ref}$:

$$A_{ref} = \left[\frac{R_a}{2} \left(\frac{\dot{m}_3 \sqrt{T_3}}{P_3} \right)^2 \left(\frac{\Delta P_{3 \to 4} / q_{ref}}{\Delta P_{3 \to 4} / P_3} \right) \right]^{0.5} \quad (4)$$

Pressure drop percentages are employed in the calculation of the total pressure loss (P3 4), which corresponds to design values (compressor/turbine) for a certain pressure drop of percent. A preliminary calculation was made that the pressure loss in a flame tube would be roughly 5% due to the fact that this project involves developing a tubular combustion chamber, where pressure losses are around 6-8 percent. The nozzle injector till the end of the main zone, the secondary zone, and the end of the dilution zone were each given a 5 percent pressure drop. The initial design is based on a few trials that yielded typical results. According to Lefebvre [6], some typical values are shown in Table 1 based on the kind of combustion chamber (tubular, annular, and annular-tube)

This value, P3=4 P3, is a design parameter and hence fixed in the Table 1 of the data set. The "pattern factor," pollution emissions, compressor output velocity, and diffuser type all play a role in determining the pressure loss factor, or P3 4/qref. When it comes to the parameter, m T PA 3 3 (3 $_{ref}$), it depends on the Remark area selected. These characteristics can't be calculated in detail, thus they were utilized from the tubular type in the table as an approximation when they were constrained by (4).

Table 1. Typical values used in combustion chambers

Type of chamber	$\Delta P_{3 \to 4}/P_3$	$\Delta P_{3 \to 4}/q_{ref}$	$\dot{m}_3\sqrt{T_3}/(P_3 A_{ref})$
Tubular	0.07	37	0.0036
Annular-tube	0.06	28	0.0039
Annular	0.06	20	0.0046

2.2. Approach Based on Combustion Process

Chemical processes may be described using the parameter,θ, which helps determine the suitable Remark area. The combustion efficiency,η, may be attributed to any fuel/air ratio, depending on the parameter θ, according to Lefebvre [6], as follows (5):

$$\theta = \frac{P_3^{1.75} A_{ref} D_{ref}^{0.75} e^{T_3/b}}{\dot{m}_3} \quad (5)$$

Considering the combustion efficiency about 100%, the parameter θ has a value 73 10_x^{-6} (SI) [6]. Finally, given the values to the Remark area, A_{ref}, obtained by both approaches, the value considered will be the greatest.

3. METHODOLOGY

The Lefebvre [6] approach tries to give the major dimensions of a tubular combustion chamber gas turbine based on input data. To begin, mass flow, temperature and area of application are taken into consideration while calculating Remark parameters. Combustion chamber flow properties like velocity, Mach number, and dynamic pressure are frequently related with these values for comparison and study purposes. According to mass distribution and the combustion regime, the combustion chamber was divided into several portions. The properties of the flow dictated the size of the diffuser, swirler, primary, secondary, and dilution zones. An Excel® spreadsheet was created to automate all of the methodology's computations, as shown in Figure 1. Different plans were made, and the computations were grouped together based on an affinity that had been formed.
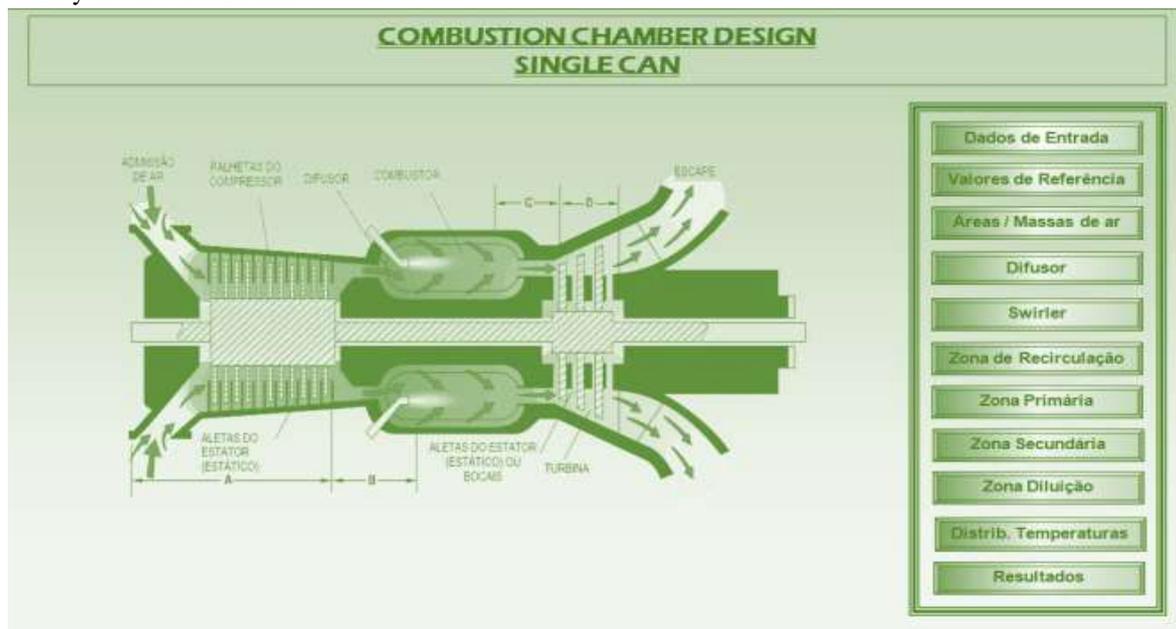
Figure 1. Initial screenshot of the Excel® spreadsheet developed

4. STANDARD DESIGN

Simulating a 600-kW gas turbine combustion chamber, GateCycle® in Rodrigues [10] was used to create the combustion chamber's shape, as shown in Table 2. Table 3 lists the chemical makeup of the fuel utilized in this simulation.

Table 2. inputs for the simulations– case 1

Parameters	Values	Units
Air mass flow rate	4.288	kg/s
Inlet air temperature	461.0	K
Inlet total pressure of air	4.053	bar
Inlet static pressure of air	3.899	bar
Mass flow rate of fuel	0.745	kg/s
Inlet fuel temperature	430.0	K
Inlet total pressure of fuel	4.053	bar
Velocity of fuel injection	15.0	m/s
Outlet temperature	1123.0	K
Pressure loss	6.0	%
Combustion efficiency	0.99	-

Table 3. Composition of natural gas that was utilized for the simulations – case 1

Chemical formula, -	Mass fraction, -
CH_4	0.8856
C_2H_6	0.0917
CO_2	0.0065
C_3H_8	0.0042
N_2	0.0120

The preliminary geometry of a tubular combustion chamber gas turbine, as shown in Figure 2, was calculated using an Excel® spreadsheet.
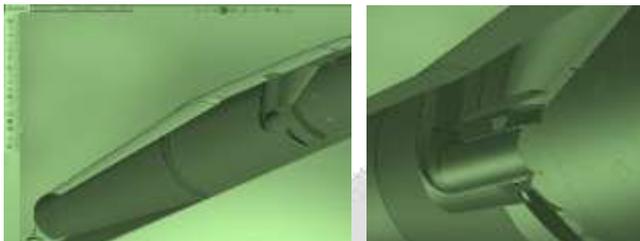
Figure 2. Excel® spreadsheets were used to create geometric shapes.
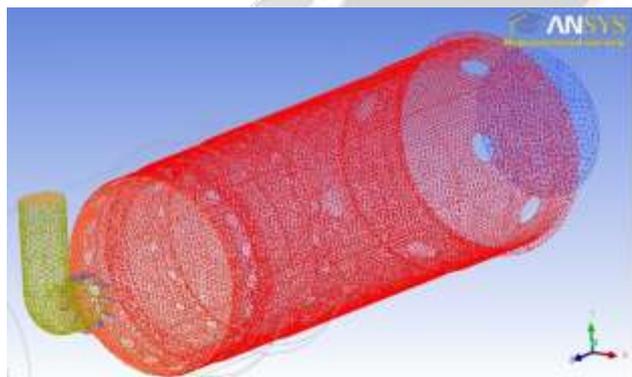
Figure 3. mesh information for Simulation 1

As seen in Figure 3, the resultant solid model must be transformed into a mesh. Complex geometries benefit most from the unstructured tetrahedral kind of mesh. We utilized elements with a size of 0.010 m with 15 iterations of modification in the global network. As a result, additional modifications were made to the dilution holes and the swirler blades, which had a maximum element size of 0.008 meters. Mesh creation ended up with 4600 nodes and 2.7 million pieces.

Because of the natural gas's nature, the following reactions were used: WD1 Ethane Air, WD1 Methane Air, and WD1 Propane Air. SST, a shear stress transfer model, was utilized to simulate the turbulence, while the Eddy Dissipation model was employed to simulate the combustion process. Due to previous success in comparable applications, the turbulence model was used. As a broad model, the Eddy Dissipation model takes into account a turbulent burning and dispersed flames, among other things. Finally, the P1 model, as advocated by Rodrigues [10], was employed to determine radiation heat transport. In this section, ANSYS CFX® simulation results are shown after construction of geometry and the accompanying boundary conditions.

5. INPUT FROM NUMERICAL MODELS

With the use of an Excel® spreadsheet, the shape of the combustion chamber was created in SolidWorksTM with the assistance of an ANSYS CFX® tool to analyze the geometry suggested by Lefebvre [6]. Each of the five systems included eight Intel Xeon QuadCore 5420 GHz/12MB processors and 16 GB of RAM. There is no commercial usage for the ANSYS CFX® license. These simulations produced the following results, which are detailed below..

5.1. Case 1

Figure 4a shows that the diffusion process was halted at a certain point, resulting in a high velocity at the center of the distribution. Despite the fact that the arrangement of streamlines in Figure 4b seems to have had no effect on the mixing process between fuel and air in the first area, the high periphery velocity should draw some of the fuel to the chamber's end.

(a)
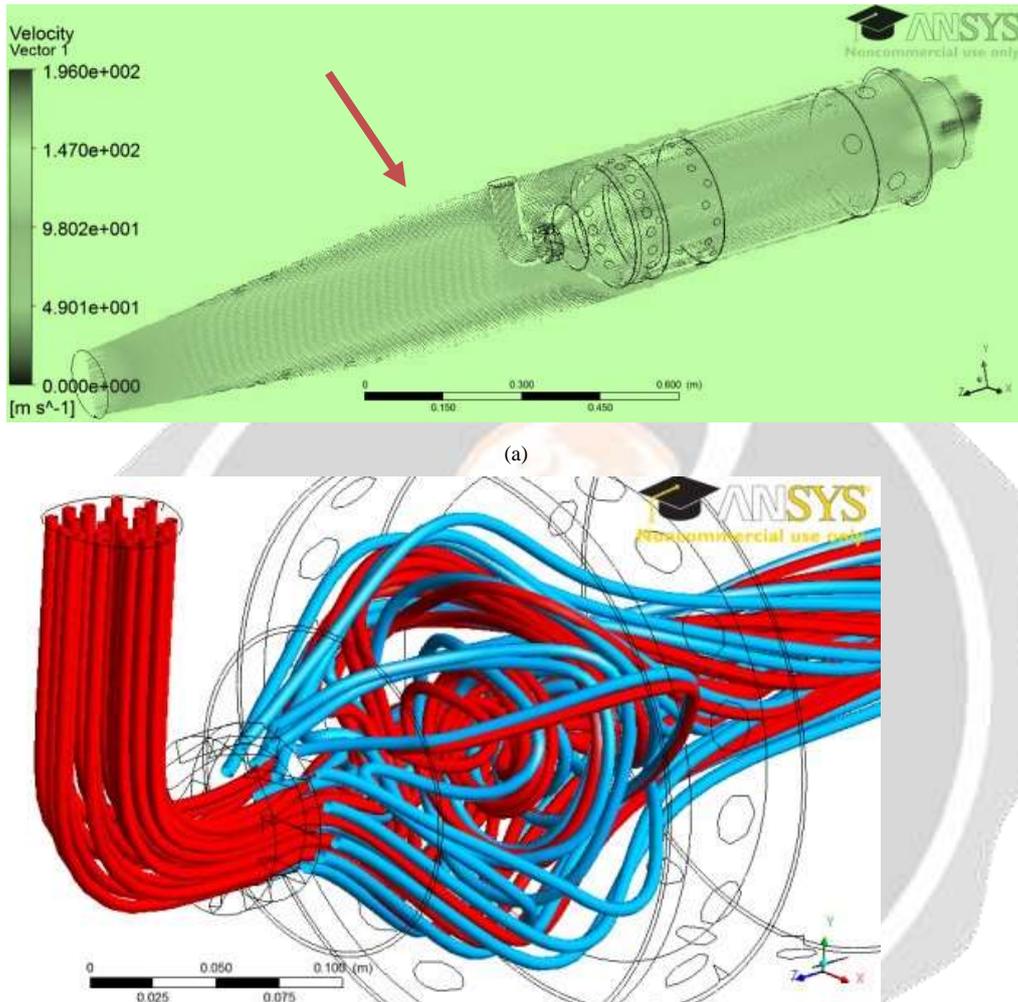
(b)

Figure 4. Simulated flow patterns in the combustion chamber the YZ longitudinal plan a) and b) of the major zone c) velocity vectors

In Figure 5 the flame temperature presents high values from the extremities to the central region of the chamber, showing the difficulty of mixing the air flow with fuel in the primary region, previously observed. Thus, as the airflow dragged just a part of the fuel, the fact that temperatures were elevated in the extremities is compatible. Furthermore, this reflects in a difficulty of mixing and in long flame, with high temperatures near the wall and output the chamber.
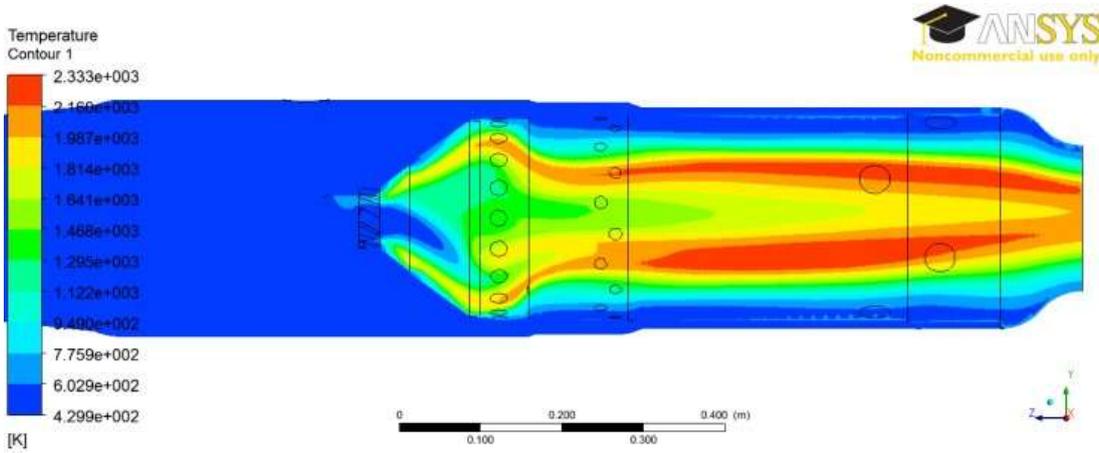
Figure 5. Temperature patterns along the YZ axis
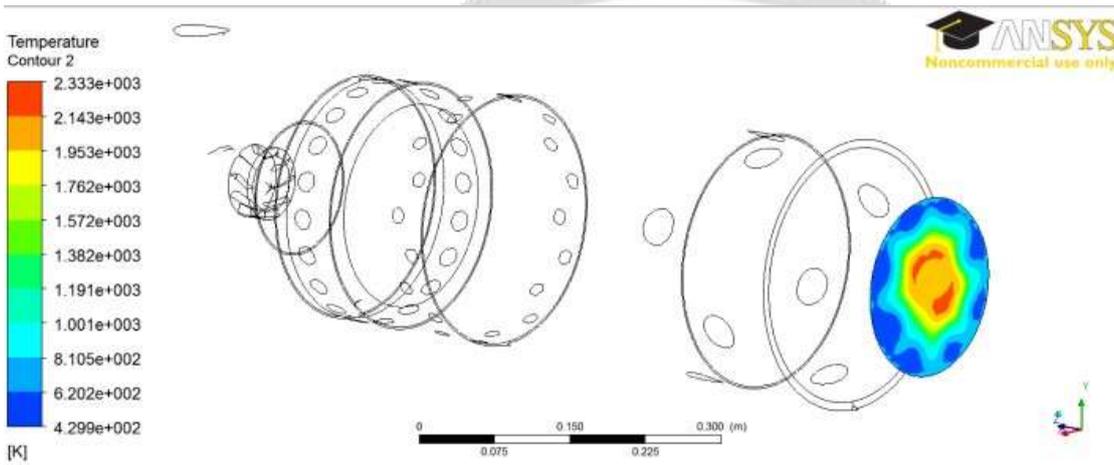
Figure 6. Exhaust flow temperature distribution
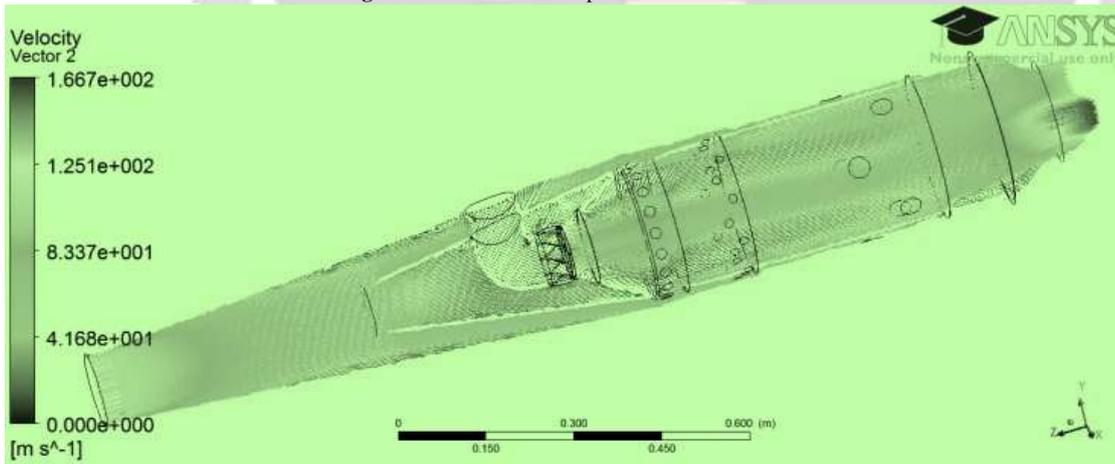
Figure 7. Simulation 2's flow distribution

Lastly, it can be seen in Figure 6 that there are certain hot patches, even if they are minor, that might have an impact on the turbine blades' integrity. Temperature at the combustion chamber's outlet was 1028.17 K, close to the figure calculated using Excel® spreadsheet calculations and in compliance with the design value of 1123K.

5.2. Case 2

A number of modifications were made to Lefebvre [6]'s original design in order to slow down the chamber's velocity and enhance the burning process, as seen in the diagram in Figure 7. For the swirl, this will help to further slow the speed at which it arrives. Figure 9 shows the addition of a radial swirler to help prevent the flow from adhering to the dome's walls and concentrating it in the centerA comparison of simulations 1 and 2 in Figure 8 reveals that snout-imposed velocity decrease is evident.
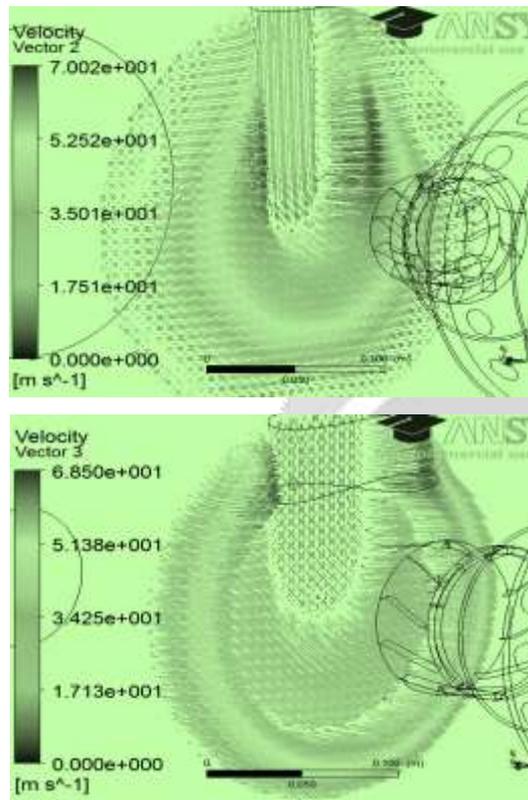
(a)

(b)

Figure 8. the intensity of local velocity vectors without and with the snout are compared
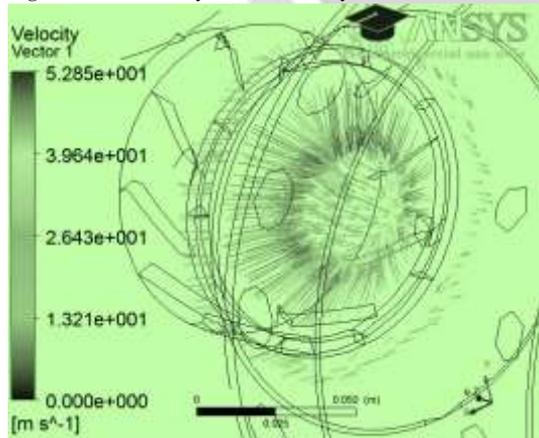
Figure 9. Axially and radially configured new swirler's exit flow details

Figure 10 shows that the temperature profile remained more uniform after the snout was added to the chamber, for example, because of the reduced velocity in the chamber.

Because lower velocities give the reactions of the combustion process more time, Figure 11 shows that this uniform temperature distribution is reflected in the outflow plane. NO emissions will be reduced as illustrated in Figure 12 if temperatures are evenly distributed.

The suggested changes, in general, were deemed to be beneficial. A significant temperature gradient, as well as a flame with a long duration and a low intensity, may be seen in the chamber's temperature distribution. This might have an adverse effect on the burning process. When Lefebvre [6] first estimated the Remark area, as reported in instance 3, he recommended a revision.

5.3. Case 3

In the simulations, it was found that lowering the arrival velocity at the swirler improved both the flame dispersion and the velocity profile. In addition, the swirler output flow was increased by focusing it in the center.
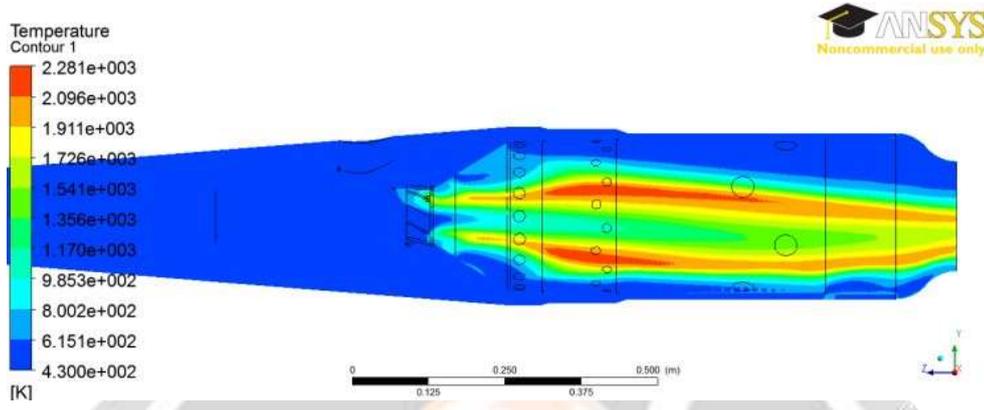
Figure 10. Simulation 2's temperature distribution

(a)   (b)
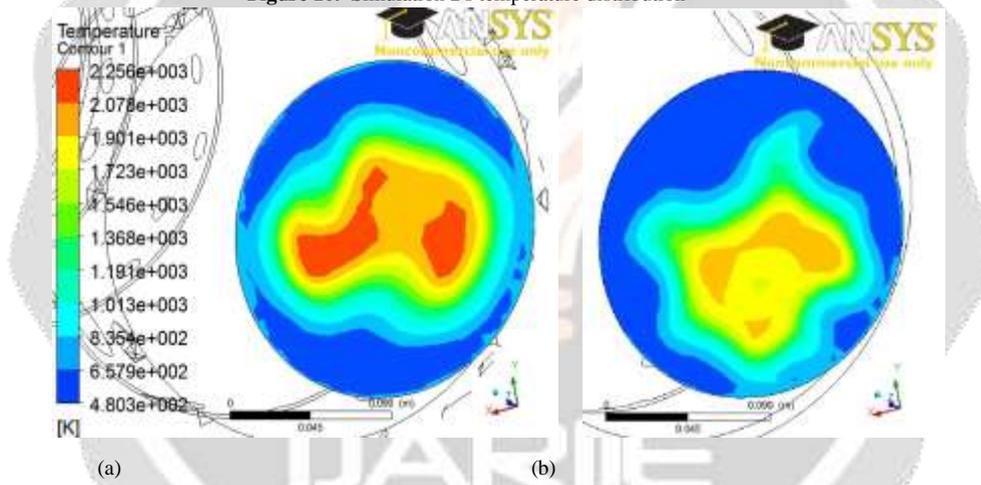
Figure 11. Output temperature distribution: (a) Simulation 1, (b) Simulation 2.

(a)   (b)
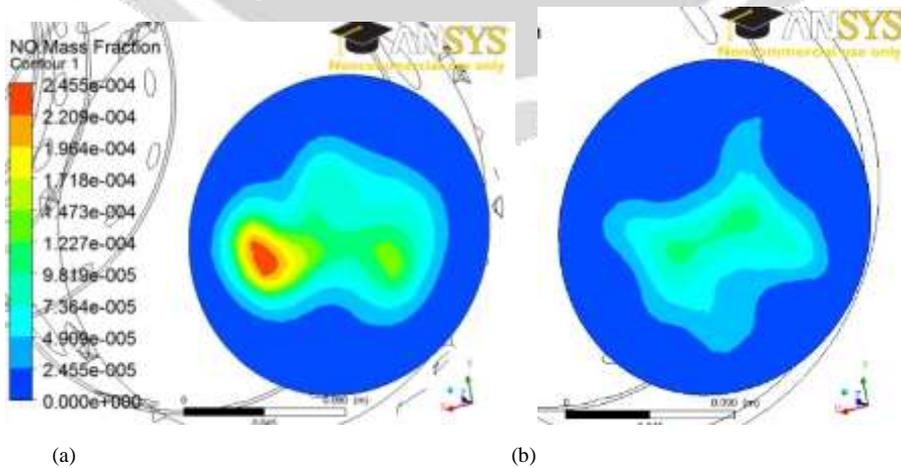
Figure 12. Comparing the NO distribution at the outflow: (a) Simulation 1, (b) Simulation 2

However, despite the recommended improvements, the chamber's high flow velocity continues. Lefebvre's [6] initial calculation of the Remark area, which can only be increased if the diffuser, the snout, and the swirler have previously been studied, is the only way to reduce the flow velocity within the chamber.

Table 4. Solar Turbine Design Data [7]

Variable		Value	Unit
Air mass flow rate		0,95	kg/s
Inlet total pressure		4,053	bar
Remark area		0.038118224	m^2
Inlet temperature		473	bar
Output power		60	kW

According to Rodrigues [10], we may use the geometric data of a Solar Turbines Model T-62T-32 to 60kW combustion chamber annular to figure out the best Remark area for our application. The remark area achieved was 0.015 m2 and had a remark velocity of 21.322 m/s, as indicated in Table 4, which was much less than the value used by solar turbines. Accordingly, the beginning Remark radius was changed until the resultant area was near to the Table 4 figure. Remark's velocity reached 8.329 m/s when it had to enlarge its diameter by 60%.

This enlarged combustion chamber was expected to need the use of fresh calculations, with the diameter of the Remark raised by 50%. There is now a greater amount of space available for the combustion process and lower flow velocities in the core area, as seen by Figure 13.

According to the alterations made, it can also be seen that the flow velocity in the flame tube was reduced, which resulted in an improved combustion process and the passage of air via the perforations.

Figure 16 shows that the mixing process has been improved as a result of the enlarged diameter of the Remark. In the output plane, a more even distribution and more consistent temperature were achieved. With an average temperature of 905.2K, there were certain areas with a temperature of 2000.0K in Simulation 2. With an average temperature of 1110.9 K (near to the design value of 1123K) in Simulation 3, the distribution was more equal, albeit there were still some hot patches.

(a)   (b)

Figure 13. Comparison of the longitudinal plan YZ velocity distribution between simulations 2 and 3 (a) and 3 (b)
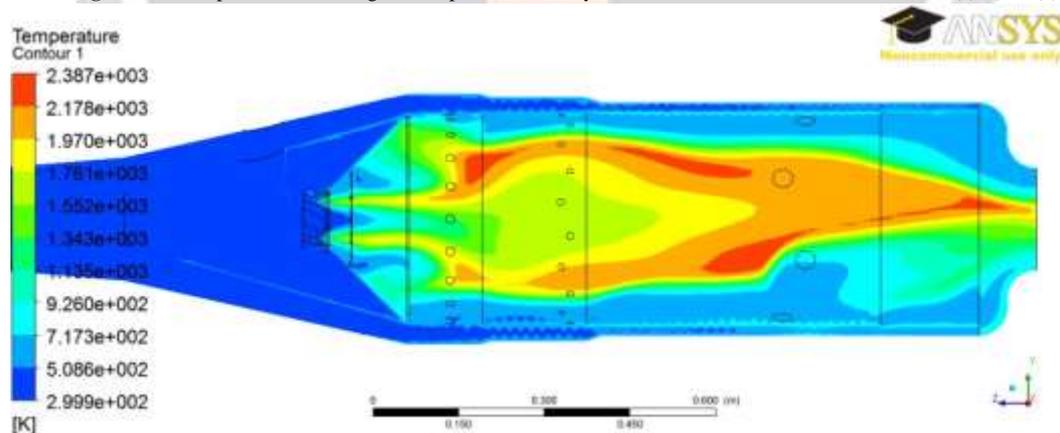
Figure 14. Simulation 3's YZ longitudinal temperature distribution

(a)   (b)
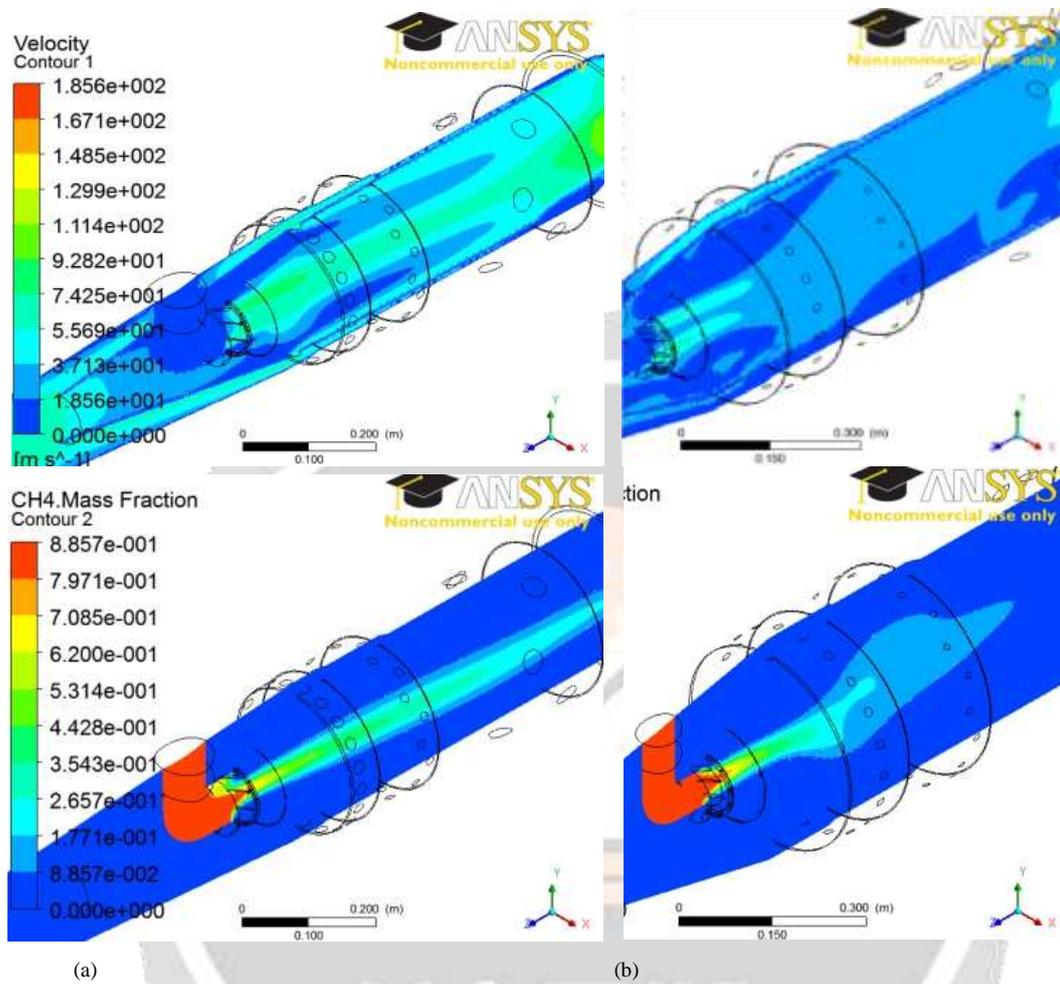
Figure 15. (a) Simulation 2, (b) Simulation 3, comparative comparison of CH$_4$ mass percent

(a)   (b)
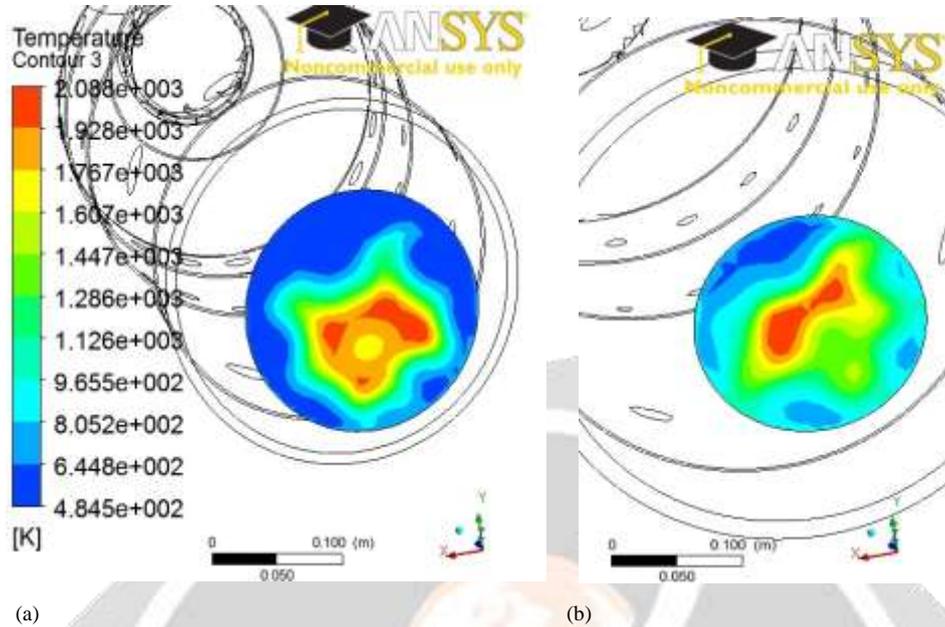
Figure 16. (a) Simulation 2, (b) Simulation 3: temperature distribution in the outflow plane

6. Conclusions

The Lefebvre [6] approach worked well in numerical simulations. However, based on the methodology's geometric data, the flow velocity inside the combustion chamber remained high even after minor geometry adjustments. So, faults in the burning process harmed the mixing process and increased pollution emissions. Increasing the Remark area, as shown in Simulation 3, reduced the rate of burning in the zone and improved the combustion process. The factors employed in Lefebvre's [6] approach for calculating Remark area must be assessed to match fuel properties via their relative flame velocity.

ACKNOWLEDGEMENT

REFERENCES

[1]   Bonzani F., Pollarolo G., Ansaldo Energia - Gas turbine operating experience with low BTU fuels, Turbo Expo 2004, Power for Land, Sea, and Air, Vienna, Austria, 2004.

[2]   Cameretti, M.C., Tuccillo R., Piazzesi, R, Study of an exhaust gas recirculation equipped micro gas turbine supplied with bio-fuels, Applied Thermal Engineering, Volume 59, Issues 1–2, 162–173, 2013.

[3]   Dias, F.L.G., Preliminary Design and Computational Simulation of Combustion Chamber of Gas Turbine Considering the Biofuels Combustion, Itajubá, 118p. Msc. Dissertation, Federal University of Itajubá, 2011.

[4]   Gupta K.K., Rehman, A., Sarviya, R.M, Bio-fuels for the gas turbine: A review, Renewable and Sustainable Energy Reviews 14, 2946–2955, 2010.

[5]   Hung, W.S.Y., Gas turbine fuels. Gas turbine technology seminar, San Diego, USA. 1989.

[6]   Lefebvre A.H., Gas turbine combustion 2nd edition. USA: Taylor and Francis; 1998.

[7]   Mollière, M., Benefiting from the wide fuel capability of gas turbines: a review of application opportunities, ASME 2002 GT-2002-30017, 2002.

[8]   Neilson C.E., LM2500 gas turbine modifications for biomass fuel operation. Biomass and Bioenergy 1998; 15(3):269-273, 1998.

[9]   Pavlas M., Stehlik P., Oral J., Sikula J., Integrating renewable sources of energy into an existing combined heat and power system; 31:2499-2511, 2006.

[10]   Rodrigues L.O., Parametric analysis of the combustion chamber of gas turbines using CFD [PhD thesis] Federal University of Itajubá, 2009.