Optimisation of Flame Holder Design for Micro Gas Turbine

I I Enagi1,2, K A Al-attab2* and Z A Zainal2

1Department of Mechanical Engineering, School of Engineering Technology, Federal Polytechnic, P.M.B 55, Bida, Niger State –Nigeria
2School of Mechanical Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Penang, Malaysia

Corresponding author *: khaled@usm.my

Abstract. Micro gas turbine (MGT) is an efficient alternative to costly generation and transmission of electricity especially in combined heat and power (CHP) for remote areas applications. In this study, an optimum geometry of a flame holder of MGT combustion chamber was determined in terms of height and diameter using solid works and ANSYS fluent. To determine and optimise the flame holder geometry, species transport and non-premix with radiation models were used in ANSYS fluent. Optimisation process was performed to select the best geometry of flame holder that produces lowest CO emission. The flame holder was divided in to three zones namely; pre-mix, combustion and dilution zones respectively. The test parameters applied during simulation include: the diameter of holes, number of rows per zone, heat flux through convective heat transfer, percentage excess air and location of dead zone. Optimum diameter of holes for the zones was found out to be 6, 8 and 10 mm with each having four rows of holes. In addition, 60 cm height and 2 inches diameter of flame holder showed a well distributed flame contours. The application of heat flux under radiation model reveals an acceptable CO emission of 102 ppm and average temperature of 1218 °C for the diesel fuel used.

1. Introduction

The most complex technical objects today are modern gas turbine engines and power plants. They are characterized by variety of operating modes under different external conditions, which makes their designing process very complicated [1]. The design of combustion chamber is one of the most difficult process in creating modern gas turbine engines [1]. Combustion chamber plays a significant role in providing proper air-fuel mixing in engines due local fluctuations of the flow field. In addition, it promotes engine performance, combustion and emission formations [2]. In micro gas turbines (MGT), the combustor is a very important part where chemical energy is converted into thermal energy through a combustion process. Many studies on geometry, configuration and dimension of the combustor have been made related to wall cooling, flame stability and emission control [3-4]. Several investigations on MGT by using different design of combustion chamber with different types of fuel were conducted in previous works [5–8]. The application of simulation software has rapidly increased over last decade in improving many complex processes including combustion chambers [9-10]. To explore and design engineering hardware and for the optimisation of combustion chambers, computational fluid dynamics (CFD) [5] has been extensively used as an important design tool [11-12]. Generally, temperature profile is an important characteristics of flame [13]. In addition, the flame location and balance of transport processes affect the performance of micro combustor through its location and shape [3-14]. For the average temperature calculations, only the core air mass flow rate in slide liner at different zones were
considered [15-16]. In the present work, a combustion chamber was designed and optimized using ANSYS-FLUENT simulation software. Species transport and non-premixed combustion models were used to determine the optimum flame holder chamber geometries. The designed combustion chamber was investigated through optimization process, and the effects of different variables and geometries on the combustion performance were studied.

2. Design, development and simulation of combustion chamber
For the combustion chamber design, the initial design parameters were due to constraints concerning compressor exits and turbine inlet temperature. The designed calculations for the dimensions were used to obtain a suitable combustion geometry. The procedure taken for design, development and simulation of the combustion chamber are (1) Design and CAD drawing of the chamber using solid works and (2) CFD simulation using ANSYS Fluent.

2.1. Design and CAD drawing of the chamber using solid works
The chamber is composed of two parts: the jacket and flame holder as shown in Figure 1. The jacket acts as air pre-heater surrounding the flame holder with conical shaped nozzle at the turbine inlet. The flame holder is the inner part where the combustion takes place. Air jacket height was varied in the range of 30-90cm, while flame holder diameter was varied in the range of 2-6 inches. The flame holder is divided into three zones namely; premix, combustion and dilution zones, each with different diameters of hole. The diameter of holes in each zone are in increasing order starting from the premix zone. For the design, all the three zones in the flame holder geometry was made of 8 number of holes, each having 4 rows.

The solid work drawing of the chamber involves several procedures beginning with 2D transforming in to 3D using boss-extrude. Reference geometry plane was then, created to enable a clear drawing of other salient features of the flame holder. To build and replicate holes in reference for the top edge of the chamber, features like cut extrude and linear pattern were used. Finally, the flame holder body is subtracted from the chamber in order to simplify meshing and simulation process by using combine and subtract feature.

![Figure 1. Solid works drawing of the combustion chamber](image_url)

2.2. Simulation in ANSYS Fluent
Simulation in ANSYS fluent involves the use of different models to achieve optimization process of selecting the best chamber geometry. Table 1 shows the different models applied and their respective functions.
Table 1. Shows different combustion models applied during simulation

| S/N | Models                              | Functions                                                                 |
|-----|-------------------------------------|---------------------------------------------------------------------------|
| 1.  | Species transport                   | Used to determine the geometry that will produce combustion              |
| 2.  | Non-premixed combustion             | Used to determine CO released in ppm (parts per million) at the outlet.  |
| 3.  | Non-premixed with radiation         | Used to determine the optimisation in terms of CO emission and outlet temperature. |

2.2.1. Species Transport model. Using this model for the first part, each geometry, ranging from 900mm as the maximum to 300mm minimum height of the chamber is simulated in Fluent, while testing each geometry for combustion. After 100 iterations, if any geometry attains temperature of above 2000K, then it is considered to produce combustion, otherwise, combustion is not sustainable.

The process includes simulation of the different geometries of flame holder, starting with minimum diameter of 50.8mm (2 inches) to the maximum of 152.4mm (6 inches). The manipulated parameters that make difference in this process are the height and the diameter of flame holder, however, other variables that were kept constant include number of rows for flame holder’s holes, holes diameter, and conic shape of the bottom part of the chamber. The overall stepwise flowchart for simulation performed in ANSYS fluent is shown in Figure 2.

Figure 2. Flowchart of steps taken in ANSYS Fluent
2.2.2. Non-premixed Combustion model. In order to optimise a geometry that will produce minimum CO emission, species transport model is therefore replaced with non-premixed combustion model in which the PDF files of propane and diesel are generated and calculated. The same procedure in species transport is also applicable here, but the calculation is running for 300 iterations to ensure the combustion is stable for CO formation at the outlet. The average mole fraction of the CO at the outlet of each geometry is then calculated based on the XY plot and in Microsoft excel.

3. Setting up the ANSYS Fluent
In Setting up the Fluent for the simulation, the parameters are set out for each part named in Mesh feature. For non-adiabatic and flow analysis, energy equation with k-epsilon viscosity model are activated. They are both used for laminar and turbulent flow with species transport and non-premix combustion model. Finite rate and turbulent eddy dissipation equations were also used in volumetric combustion model. The fuel is then set to either propane-air or diesel-air depending on what type of fuel is required for the simulation. The boundary conditions are set out using several parameters as shown in Table 2.

| Parameters | Variables |
|------------|-----------|
| Fuel Inlet | 300 K     |
| Temperature | 2atm     |
| Pressure | 0.0058kg/s |
| Mass flow rate (70% excess air) | 0.0058kg/s |
| Air Inlet | 530 K     |
| Temperature | 1.5atm |
| Pressure | 0.15kg/s |
| Mass flow rate | 1atm |
| Outlet | 1atm     |
| Pressure | 600 K     |
| Back flow Temperature | 1atm |

3.1. Optimization
Optimization of the chamber involves further simulation of the selected geometries that produces best CO emissions. The variables use during optimization include, the number of rows for holes of flame holder and its diameter, percentage of excess air, length of dead zones of the chamber and also height of the chamber.

This is required to reduce the number of CO per mole released at the outlet. Thus, P1 radiation model is introduced which has taken account of heat fluxes around the wall of the chamber. The calculations here therefore, considered the effect of convection and radiation of heat transfer. The value of emissivity applied is 0.5 that is suitable for steel being the material of the existing chamber in the Engine Lab. Parameters set in radiation model are shown in Table 3.

| Feature | Value |
|---------|-------|
| Type of radiation model | P1 |
| Material of the chamber | Steel |
| Emissivity | 0.5 |
| Wall thickness | 6mm |
| Heat fluxes at the outer wall | 10800 W |
4. Results and discussions
This section includes results and discussions for simulations in ANSYS Fluent that makes up the optimisation process. Species transport model in ANSYS Fluent shows the rough result of the combustion process that occurred inside the chamber, which is evidently an adiabatic reaction. Since species transport model only shows the basic and simple model of the chamber, non-premixed combustion model that can provide the formation of CO emission after combustion process was introduced. For optimization process, the use heat flux through convective heat transfer calculation allows for selection of best geometry of the flame holder.

The variables used during the simulation to achieve the optimum geometry are:

- The diameter of flame holder and height of the chamber.
- The number of holes for flame holder’s zone.
- The diameter of holes in each zone for flame holder.
- The convective heat flux and radiation model.
- The excess air supplied
- The dead zones of the flame holder.

4.1. Effect of flame holder diameter and chamber height
The results obtained from species transport model are tabulated in a Table 4 for further optimisation of the chamber, which will be determining which geometry to produce the least CO emission. The results shows that chamber heights above 80 cm are not sustainable for combustion of flame holder diameters below 4 inches. However, there was a significant sustainability between 30 to 70 mm chamber heights. For reasons bothering manufacturing cost, and minimum volume and area of flame holder, a preferred choice of 60 cm chamber height and 2 inches flame holder diameter is chosen for further optimisation.

| Chamber Height (cm) | 30  | 40  | 50  | 60  | 70  | 80  |
|---------------------|-----|-----|-----|-----|-----|-----|
| Flame Holder Diameter (inches) |     |     |     |     |     |     |
| 2                   |     |     |     |     | ✓   | ✓   |
| 3                   | ✓   | ✓   | ✓   | ✓   | ✓   | ✓   |
| 4                   |     | ✓   | ✓   | ✓   | ✓   | ✓   |
| 5                   |     | ✓   | ✓   | ✓   | ✓   | ✓   |
| 6                   |     |     | ✓   | ✓   | ✓   | ✓   |

4.2. Effect of rows and diameter of holes
For the species transport model simulation, the flame holder diameter of holes was first fixed at 6, 8 and 10 mm and later 8, 10 and 12mm for premix, combustion and dilution zones. In addition, 15 and 4 number of rows per zone were separately varied for the first test considerations. The variables for the chamber height was from 300 to 700 mm and 50.8 to 152.4 mm for flame holder diameter. The temperature contour at 600 mm chamber height for 4 rows of holes in 6, 8, and 10mm configuration shows a well distributed undisturbed flame inside the flame holder, while for 15 rows configurations, the flame is quite disturbed as shown in Figures 3 A and B.
Figures 4 A and B shows the temperature contours for the two set of holes diameters. The configuration with 8, 10 and 12 mm in Figure 4A does not produce good combustion performance as it clearly showed early start of combustion at premix zone rather than combustion zone. On the other hand, configuration with 6, 8 and 10 mm shows a well-distributed combustion starting at the combustion zone as shown in Figure 4 B. While Figures 5A and B show an outlet temperature contours of mole fraction of CO for (6, 8, 10) mm configuration which is better than that of the (8, 10, 12) mm.
4.3. Effect of excess air supplied

The excess air supply to MGT combustion chamber is needed to ensure complete combustion of the fuel used. The excess air supplied were tested from 20% to 70% excess air. The 70% excess air produces the lowest number of CO emissions as compared to others put under investigation. Figure 6A shows the average mole fraction using diesel fuel with 20% and 70% excess air supplied. The effect of having excess air for combustion is to improve the combustion efficiency. The results reveal that combustion efficiency increases with increased in excess air until the heat loss during the supply of excess air is greater than the heat provided.

4.4. Effect of convective heat flux

Heat transfer surfaces are commonly present in combustion chambers of many types of processes and power equipment. Figure 6B shows the effect of convective heat flux that was applied to further reduce the outlet CO emission. The convective heat flux (heat transfer) of 10800W was applied to the outer wall with the emissivity of 0.5 and 6mm wall thickness of the stainless steel material used.
4.5. Effect of dead zones
Creation of dead zone between the premix, combustion and dilution zone was also used to improve the combustion. The dead zone between combustion and dilution zones (dead zone B) produced a better combustion in terms of CO emission, than the zone between premix and combustion zones (dead zone A) as shown in Figure 7.
4.6. Optimization

Optimising the combustion chamber geometry promotes the reduction of CO emission and further enhance efficiency of MGT. For the optimisation process, 60, 70 and 80 cm chamber heights and 2, 3 and 4 inches’ flame holder diameters were separately simulated. Figures 8A & B show the outlet average CO mole fractions and temperatures.

![Figures 8. A & B: CO and temperature plots for optimum geometries](image)

4.7. Optimum chamber design

The optimum chamber geometry with dimensions as shown in table 5 and design parameters in Table 6 was therefore adopted after optimisation, in addition, are 4 holes of (6,8,10) mm, and dead zone between the combustion zone and dilution zone. It produces an outlet average CO mole fraction of 102 ppm and temperature of 1218 °C. It also confirms as having the best combustion stability, with the flame well distributed within the flame holder. This shows that the flame does not rush outside the flame holder, confirming that the outer surface temperature of the chamber is cooler than the other parts. Figures 9 (A-D) show the outlet average CO emissions and temperatures of the optimum combustion chamber.
Figure 9. (A-D): Average CO emissions and temperatures of the optimum Combustion chamber

| Table 5. Main dimensions of the combustion chamber |
|-----------------|------------------|
| Dimensions      | Value            |
| Height of the Jacket (mm) | 700             |
| Diameter of the Jacket (mm) | 152             |
| Height of the flame holder (mm) | 690             |
| Diameter of the flame holder (mm) | 50              |
| Diameter of air inlet (mm)    | 50              |
| Diameter of the fuel inlet (mm) | 3               |
| Diameter of the Outlet (mm)   | 50              |
| Number of holes per row       | 8               |
| Number of rows of holes       | 4               |
5. Conclusion

The design optimisation of a combustion chamber for MGT was performed, using CAD solid works and ANSYS-FLUENT 16.1 version CFD simulation software. Various design parameters and combustion models were used to produce an optimum design configuration. The simulations were first performed on different 3D geometries of chamber heights/flame holder diameter using species transport combustion model. The non-premixed combustion models, which employs the infinitely fast chemistry assumptions, were further applied to predict the diffusion flames.

The design parameters used with non-premixed transport model include: the effect of holes diameters, excess air supplied, convective heat flux and dead zones. In addition, optimisation was performed to further reduce the number of CO emissions and outlet temperature. The result revealed that, design of the optimum chamber produced overall combustion stability with least outlet CO emission and well-defined temperature contours. In addition, a 2-inch flame holder diameter, 60cm chamber height, having 4 holes of (6, 8, and 10) mm with dead zone between combustion and dilution zone was adopted as the optimum chamber geometry. The simulations results show that an average CO mole fraction of 102 ppm and temperature of 1218 °C were produced at the outlet.

Acknowledgments

The authors would like to thank the Universiti Sains Malaysia, RUI grant number: 1001/PMEKANIK/814282, for the financial support of this study.

References

[1] Aleksandrov Y B and Mingazov B G 2017 Optimal design of a combustion chamber of gas turbine engine by a Combustion chamber 1D-2D computer program IOP Conf. Ser. Mater. Sci. Eng. 240:012006. doi:10.1088/1757-899X/240/1/012006
[2] Varun, Singh P, Tiwari S K, Singh R and Kumar N 2017 Modification in combustion chamber geometry of CI engines for suitability of biodiesel: A review Renew Sustain Energy Rev 79:1016–33. doi:10.1016/j.rser.2017.05.116
[3] Li J, Chou S K, Li Z W and Yang W M 2009 Characterization of wall temperature and radiation power through cylindrical dump micro-combustors Combust Flame 156:1587–93 doi:10.1016/j.combustflame.2009.05.003
[4] Wan J, Fan A, Yao H, Liu W. Effect of thermal conductivity of solid wall on combustion efficiency of a micro-combustor with cavities. Energy Convers Manag 2015;96:605–12 doi:10.1016/j.enconman.2015.03.030.
[5] Krieger G C, Paulo S, Filho F L S and Souza R C De 2012 A Swirler Stabilized Combustion Chamber for a Micro-Gas Turbine Fuelled with Natural Gas J. Braz. Soc. Mech. Sci. Eng. XXXIV:441–9
[6] Zornek T, Monz T and Aigner M 2015 Performance analysis of the micro gas turbine Turbec T100 with a new FLOX-combustion system for low calorific fuels Appl Energy 159:276–84 doi:10.1016/j.apenergy.2015.08.075
[7] Cavarzere A, Morini M, Pinelli M, Spina P R, Vaccari A and Venturini M 2014 Experimental analysis of a micro gas turbine fuelled with vegetable oils from energy crops Energy

Table 6. Parameters used in designing the flame holder.

| Zones     | Diameter (mm) | Hole/Row Number of rows of holes | Hole Area (m²) | Area/Row (m²) | Total Area (m²) | Area compared to air inlet (%) |
|-----------|---------------|----------------------------------|----------------|---------------|----------------|-------------------------------|
| Premix    | 0.006         | 8                                | 4              | 0.000028      | 0.000023       | 0.00090                      | 46.08                        |
| Combustion| 0.008         | 8                                | 4              | 0.000050      | 0.00040        | 0.00161                      | 81.92                        |
| Dilution  | 0.10          | 8                                | 4              | 0.000079      | 0.00063        | 0.00251                      | 128                          |
[8] A H Lefebvre 1998 GAS Turbine Combustion, Second Edition - Books.google.com
[9] M M Noor, A P Wandel and T Yusaf 2013 Design and development of mild combustion burner
J. Mech. Eng. Sci. 5:662–76
[10] K A Al-attab and Z A Zainal 2014 Performance of a biomass fueled two-stage micro gas turbine
(MGT) system with hot air production heat recovery unit Appl Therm Eng 70:61–70
doi:10.1016/j.applthermaleng.2014.04.030
[11] Computational Fluid Dynamics in Industrial Combustion - Google Books n.d.
[12] Bott R 2014 Applied modelling for bio-and lean gas fired micro gas turbines Igarss 2014:1–5
doi:10.1007/s13398-014-0173-7.2
[13] Jiaqiang E, Peng Q, Liu X, Zuo W, Zhao X and Liu H 2016 Numerical investigation on hydrogen
/ air non-premixed combustion in a three-dimensional micro combustor Energy Convers
Manag 124:427–38 doi:10.1016/j.enconman.2016.07.048
[14] Li Y, Chen G, Cheng T, Yeh Y and Chao Y 2013 Combustion characteristics of a small-scale
combustor with a percolated platinum emitter tube for thermophotovoltaics Energy 61:150–7
doi:10.1016/j.energy.2013.09.003
[15] Mark C P and Selwyn A 2016 Design and analysis of annular combustion chamber of a low
bypass turbofan engine in a jet trainer aircraft Propuls Power Res 2016;5:97–107
doi:10.1016/j.jprr. 2016.04.001
[16] Carlos A and Filho P 2004 Basic design principles for gas turbine combustor. Proc 10o Brazilian
Congr Therm Sci Eng -- ENCIT 2004 Braz Soc Mech Sci Eng -- ABCM, Rio Janeiro, Brazil,
Nov 29 -- Dec 03, 2004 BASIC