NUMERICAL ANALYSIS OF FLAMELESS COMBUSTION USING THE TORUS COMBUSTION CHAMBER

[1]SEONG SHIN ONG, [2]YONGSON OOI, [3]KIA WAI LIEW, [4]DIRK RILLING

[1,2,3]Faculty of Engineering and Technology, Multimedia University, Melaka, Malaysia.
[4]Department of Technology (Energy and Environmental Engineering), AKAD University, Stuttgart, Germany
E-mail: [1]oss0517@hotmail.com, [2]yongson.ooi@mmu.edu.my, [3]kwliew@mmu.edu.my, [4]dirk.rilling@akad.de

Abstract- The main objective of this research is to design and analyze a low polluting combustion system, in particular the most hazard and common polluting substance, nitric oxide (NOx). In order to reduce the NOx formation through high temperature combustion, flameless oxidation is the solution. The peak combustion temperature is lowered through flameless oxidation and hence suppresses the formation of NOx. The flameless oxidation or combustion is achieved by inducing large amount of exhaust gases recirculation. This project is focused on designing a new combustion chamber to specially cater to optimizing the performance for flameless combustion using various types of fuels, including the biofuel. A torus shaped of combustion chamber is designed and tested on various fuels in both flame and flameless combustion mode. Using the commercialized numerical code; various numerical models were produced to give the insight of the flameless combustion. The numerical models are validated with the experimental data. From both experimental setup and numerical models, the data shows that the thermal NOx formation has been greatly suppressed with the large amount of recirculation due to the torus shaped combustion chamber.

Index Terms- Flameless combustion, NOx emissions, Torus chamber, Numerical analysis.

I. INTRODUCTION

The emissions from the combustion are getting more attention due to its impact toward the environment, in particular the air pollutions. Among the emissions, Nitric oxide is the most relevant pollutants since it is formed from the nitrogen and oxygen in the air through high temperature combustion. At present, various systems have been developed to reduce the NOx emissions from combustion system using fossil fuel as well as the emphasis in bio-fuel. The flameless oxidation was introduced to avoid temperature peaks, which suppress the thermal NOx formation [1]. The flameless oxidation (FLOX) process is archived by specific flow and temperature conditions. The condition is obtained by inducing large amounts of recirculation of the exhaust gases. The recirculated gases increased the temperature of the air-fuel mixtures of pre-combustion. It is observed that the adiabatic flame temperature of the combustion is reduced and the stage of the flameless combustion is achieved gradually. The FLOX technology is able to reduce the thermal NOx formation, in particular the power plant. A technical assessment of NOx emission from coal or biomass power plant has been carried out using FLOX technology [2]. The FLOX technology is also applicable to gas turbine conditions, at high pressure and overall lean condition [3]. FLOX has technical and economic advantages and would be an attractive option for low NOx combustor in gas turbine system [4].

In order to get the details insight into the combustion and recirculation in flameless mode, which sometimes these information are not easy to obtain from experimental setup, numerical method is considered. The benefit of using computational fluid dynamics (CFD) in the combustion chamber design is to provide some useful information before a chamber design is finalized and sending for fabrication. Apart from that, the numerical method using the commercialized CFD code can be used to model the mixing and transport of chemical species by solving conservation equations describing convection, diffusion and reaction sources for each component species [5]. To ensure the numerical simulation model using commercialized CFD code such as ANSYS are accurate, the simulated models are to be validated and supported by experimental data. Once the models are validated, the combustion chamber designs can be enhanced by modifying the validated numerical models.

The flameless combustion experimental setup requires slightly different instrument from the conventional combustion experiment. As the 'flame' in flameless combustion mode does not emit electromagnetic wave in visible spectrum, the infrared (IR) camera is required to capture the 'flame' in the flameless combustion mode. In flameless combustion mode, the pressure variance is much lesser than the normal combustion, which makes the combustion hardly detected by audible flame. The study of temperature profile of combustion flame using IR camera might be useful in diagnosing the formation of NOx [6].

Experimental and numerical analysis of flameless oxidation using biodiesel such as palm oil are required to evaluate the various parameters of combustion. Similar works done have been carried out to investigate the flameless jet for bio-fuel, such as ammonia (NH3) [7]. The NO emissions from lab-scale burner operating in flameless combustion have been investigated experimentally and numerically [8]. The works done on combustion in general, particular in non-flameless oxidation of biodiesel such as palm oil has been increasing rapidly. The characteristics of

palm methyl ester as an alternative fuel for gas turbine has been determined [9]. The work presents a theoretical and experiment study of the biodiesel from waste vegetable oil in a flame tube furnace provide useful resource for heat transfer analysis from the biodiesel combustion [10]. Conversion of biomass waste from palm oil production to useful energy demonstrates the utilization of biomass energy [11].

II. COMBUSTION CHAMBER DESIGN

As the flameless combustion mode is obtained by large amount of recirculation of exhaust gases, the combustion chamber is designed to maximize the amount of recirculating exhaust gases. The conventional combustion chambers are designed based on the combustion processes, which are mixing, oxidation (combustion), and exhaust. In order to maintain the high momentum of the flow, the flow is kept in the same direction of the three processes, which most of the combustion chambers are in cylindrical shaped. When the flameless combustion was introduced, the cylindrical shaped combustion chamber is modified by placing blockages at the exhaust section of the chamber to allow a certain amount of exhaust gases flowing back to the mixing section through a separated path. Obviously, the design does not optimize the recirculation. A limited amount of recirculation of exhaust gases makes the flameless combustion mode difficult to obtain, in particular the biofuel combustion. As such, a completely new idea of combustion chamber is introduced to resolve the limited recirculation problem. A combustion chamber, which is completely different from conventional cylindrical shaped is designed as illustrated in Figure 2.1.

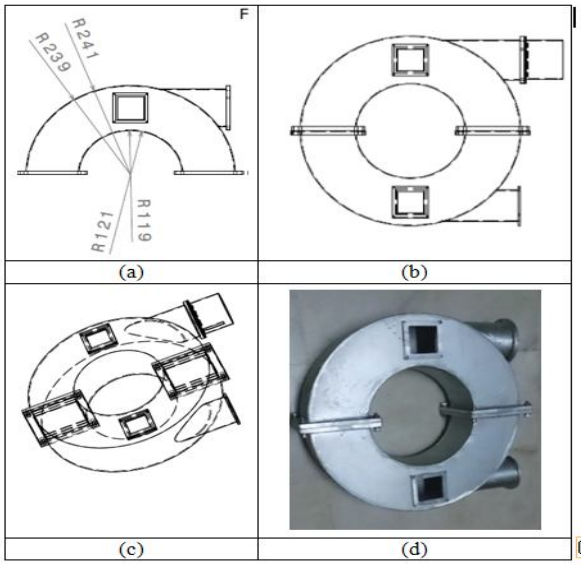


Figure 2.1 Torus shaped combustion chamber design: (a) inlet structure and (b) combined inlet and outlet structures, (c) assembly drawing, and (d) the fabricated chamber.

The chamber consists of two main symmetrical half-torus shaped structures. The two structures are combined with screw mounting. The torus shaped structures are in the size of 120 mm of the inner radius and 240mm of the outer radius. There are two small opening, covered with tempered glass at each structure to observe the combustion. Although there will be no visible flame at the flameless combustion mode, but these two opening will serve as the indication for switching from normal combustion mode to flameless mode. The mixing inlet is attached tangentially with one of the structure while the exhaust outlet is attached tangentially with another structure as illustrated in Figure 2.1 (b). In the inlet section, a static mixer is installed to enhance the mixing of the fuel and the air. A nozzle is placed at the center of the static mixer, where the fuel is mixed with swirling air from the upstream. The nozzle is replaceable to cater for different types of fuel, including methane, butane, diesel and biodiesel. The torus shaped combustion chamber was tested using various types of fuels. The fuel was injected using the high pressure pump through the nozzle. Initially, the combustion started in the flame mode as illustrated in Figure 2.2 (a). After a certain period of time, depending on the combination of fuel and nozzle used, the combustion gradually switched to flameless mode as illustrated in Figure 2.2 (b). The temperatures at various positions are taken using high temperature thermo couples. The composition of the exhaust gases is also measured using the gas analyzer. The experimental data will be served as the validation for the numerical models.



Figure 2.2 Torus combustion chamber in (a) flame mode and (b) flameless mode.

III. NUMERICAL SETUP

In order to obtain the details of the combustion, various numerical models were setup using commercialized code, ANSYS CFX. Using the same dimension of the experimental model, the numerical models are discretized with tetrahedron meshes. Using the average mesh sizing of 0.5 mm, which give the total of around half a million elements (544410 elements) created throughout the entire control volume. The grids of the tetrahedron elements are illustrated in Figure 3.1 (c). Special treatments are done on the elements close to the fuel inlet of the nozzle and the static mixer of the air inlet, in which the elements are gradually reduced in size toward the relative small surface as shown in Figure 3.1 (b) and

(a) respectively.

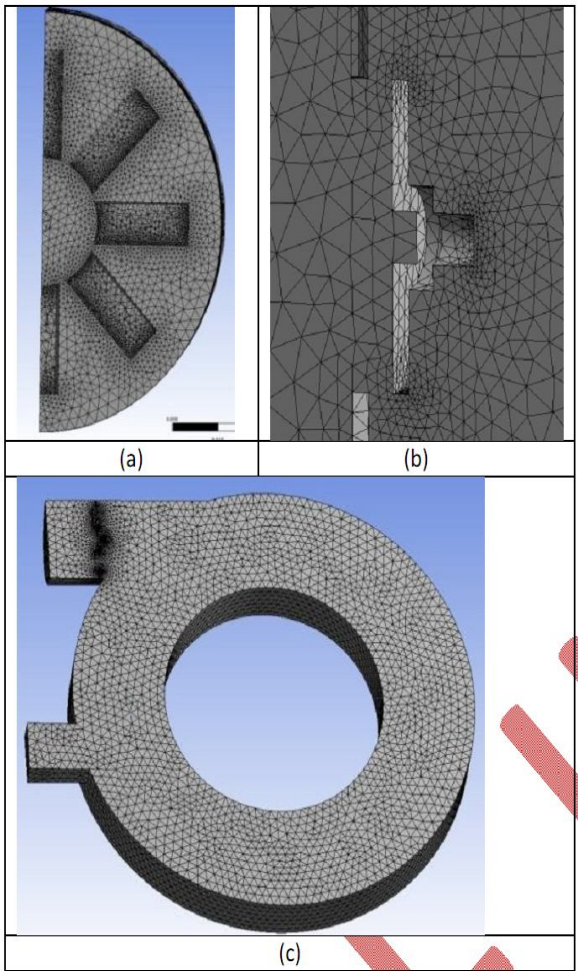


Figure 3.1 Tetrahedron meshes on (a) air inlet static mixer, (b) nozzle fuel inlet and (c) entire chamber

As the torus shaped combustion chamber is symmetrical, only half of the chamber is considered for the numerical models in order to save computation time. The inlet conditions were set as the mixtures of air and the specific fuel. Both air and fuel are set at inlet temperature of 300 K. As the air was blown into the chamber using blower and the inlet velocity of the air was set at 3 m/s, corresponding to the blower average exist velocity measured. The air blower cross section area is similar to the inlet cross section area of the combustion chamber. For the fuel, the volume flow rate was obtained by simply spraying the fuel into a cup and the time taken to fill up a specific volume was measured. The inlet velocity of the fuel into the combustion chamber was obtained by considering the fuel volume flow rate and the nozzle area. The fuel inlet velocity was set at 50 m/s. Although the fuel pressure is high in the nozzle, but only the pressure exist of the nozzle was considered. The control volume of the numerical model is enclosed by fuel inlet (the nozzle spray), air inlet (air blower), solid wall (combustion chamber structures), and exhaust outlet (exhaust gases release to air at atmospheric pressure). As the combustion chamber is coated with heat insulating surface, the amount of heater transfer through the combustion chamber structure is considered insignificant as compared to the heat generation inside the combustion chamber.

IV. RESULT AND DISCUSSION

The numerical models were solved using the commercialize CFD code, ANSYS CFX. The converged steady state solutions were obtained after 200 to 300 iterations (depending on the models), with the convergence criteria of 1 x 10-4 applied for all equations. The solutions were used to compute various informative charts for evaluating the combustion quality. In this paper, the combustion of methane gas is used to illustrate the flameless combustion using the torus shaped combustion chamber. From Figure 4.1 (a), the methane fuel depleted at just a couple of centimeters from the fuel injection. The combustion is well mixed and complete. Figure 4.1 (b) shows that the composition of the nitrogen oxide in mass fraction is kept below 0.0025 % throughout the combustion, which agree with the data obtained from the experiments where the NOx composition measured is always below 20 ppm. The torus shaped combustion chamber produce much lesser nitrogen oxide. Figure 4.1 (c) and (d) shows the composition in mass fraction of the oxygen and the carbon dioxide respectively. The oxygen is having a reducing of composition flowing from upstream (inlet) to downstream (outlet), while the carbon dioxide is having an increasing of composition.

In the aspect of flow pattern, the streamlines in Figure 4.2 (a) shows that quite a significant amount of flow is diverted back to the inlet section by simply following the curvature of the torus shaped chamber. This torus shape design obviously induces much greater recirculation than the conventional cylindrical shaped chamber. The velocity vector diagram in Figure 4.2 (b) also shows the similar flow pattern. It is observed from the velocity vector diagram that the low pressure suction from the outlet is causing the flow separated from the inner radius of the chamber. If the flow can be prevented from separation, it will obviously increase the recirculation further. Some vane can be placed at the outlet to prevent direct suction in the normal direction of the flow stream. Alternatively, the outlet can be reposition so that the outlet suction is applied along with the stream flowing direction.

From Figure 4.3 (a), the temperature region above 2000 K is limited to a small region not too far from the inlet. In comparison with adiabatic flame temperature of methane, which is 2200 K, the average temperature of the combustion chamber is much lower. The pressure in the combustion chamber stays uniformly across the chamber and it is slightly above the atmospheric pressure. The inlet pressure is much higher than the rest of the combustion chamber, but the magnitude is just about 1 % greater than the atmospheric pressure.

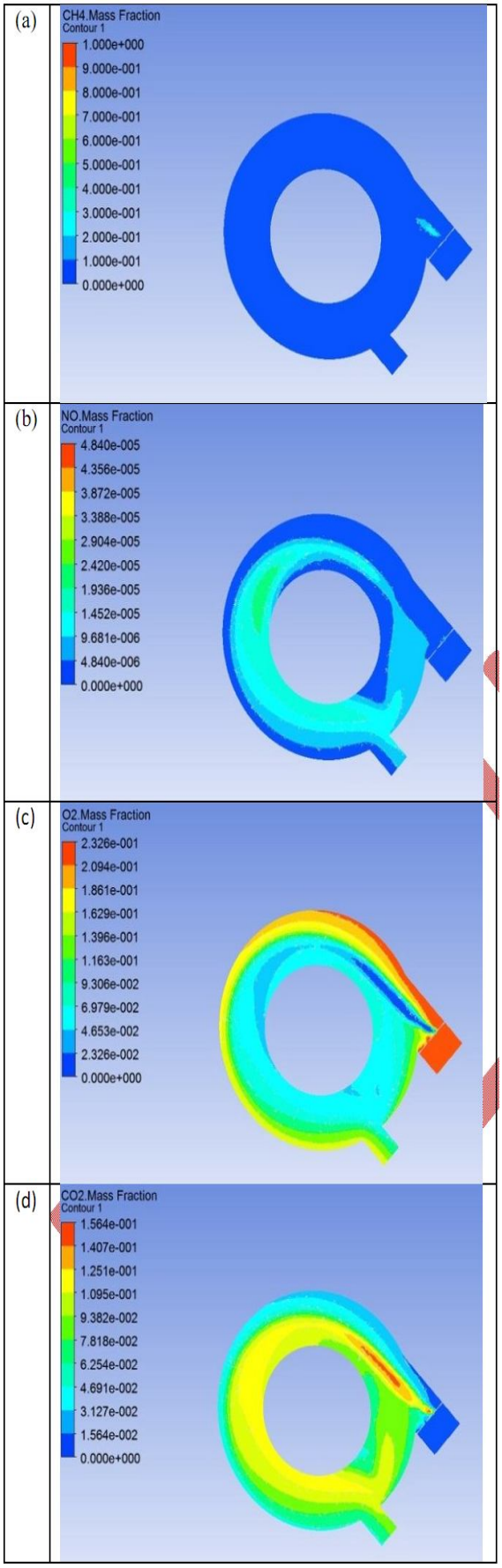


Figure 4.1 The composition of various substances in mass fraction for (a) methane, (b) nitrogen oxide, (c) oxygen, and (d) carbon dioxide.

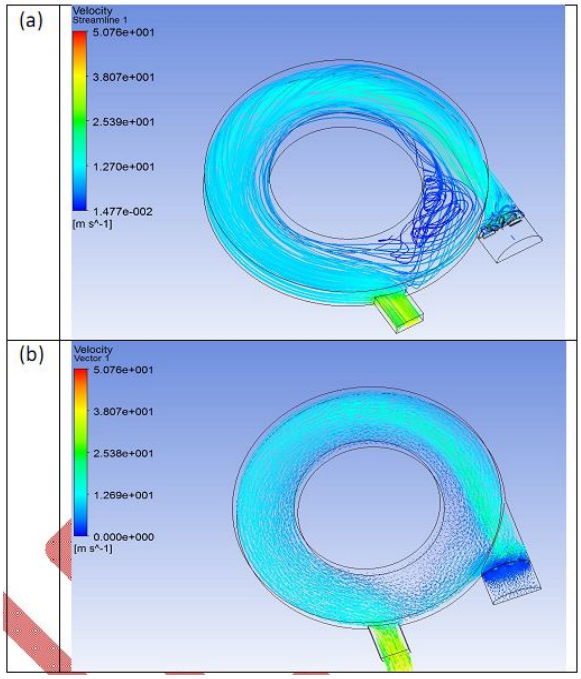


Figure 4.2 The flow pattern in the chamber illustrated by (a) streamline and (b) velocity vector.

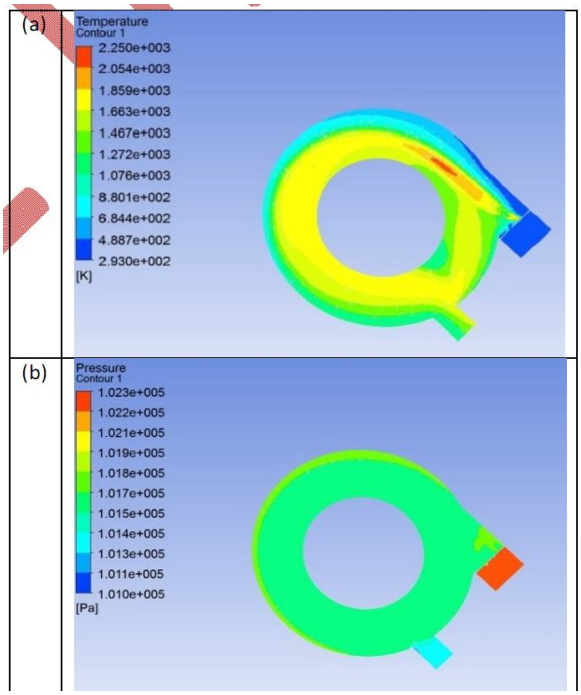


Figure 4.3 The (a) temperature and (b) pressure contour at the steady state combustion.

CONCLUSION AND RECOMMENDATION

The numerical model shows that the composition of nitrogen oxide remains low throughout the combustion. The finding is agreed with the nitrogen oxide composition measured from the gas analyzer. The new design combustion chamber induce large amount of recirculation and hence suppress the thermal NOx formation by achieving flameless oxidation.

However, in the flameless mode, the flame structure is not able to visualize since there is no significant radiation in the visible spectrum, as the flameless combustion or oxidation produce invisible and non-audible flame. It is recommended that an IR camera can be used to observe the flame and produce the details of temperature distribution.

ACKNOWLEDGEMENT

The authors would like to thanks Solution Engineering Holdings Berhadof supplying complementary biodiesel for the combustion experiments. The research is support by the FRGS research grant from Ministry of Higher Education Malaysia.

REFERENCES

[1] J.A. Wünning, J.G. Wünning (1997). Flameless oxidation to reduce thermal NO-formation, Prog. Energy Combust. Sci. 23, 81-94.

[2] Y. D. Wang, D. McIlveen-Wright, Y. Huang, N. Hewitt, P.Eames, S. Rezvani, J. McMullan, A. P. Roskilly (2007). The application of FLOX/COSTAIR technologies to reduce NOx emissions from coal/biomass fired power plant: A technical assessment based on computational simulation, Fuel 86, 2101-2108.

[3] Michael Flamme (2004). New combustion systems for gas turbine (NGT), Applied Thermal Engineering 24, 1551-1559.

[4] Y. D. Wang, Y. Huang, D. McIlveen-Wright, J. McMullan, N. Hewitt, P.Eames, S. Rezvani (2006). A techno-economic analysis of the application of continuous staged-combustion and flameless oxidation to the combustor design in gas turbine. Fuel Processing Technology 87, 727-736.

[5] ANSYS user guide.

[6] Saroj Kumar Jha, Sandun Fernando, S.D. Filip To (2008). Flame temperature analysis of biodiesel blends and components, Fuel 87, 1982–1988.

[7] MariuszZieba, Anders Brink, Anja Schuster, Mikko Hupa, Günter Scheffknecht (2009). Ammonia chemistry in a flameless jet, Combustion and Flame 156, 1950-1956.

[8] C. Galletti, A. Parente, M. Derudi, R. Rota, L. Tognotti (2009). Numerical and experimental analysis of NO emissions from a lab-scale buner fed with hydrogen-enriched fuels and operating in MILD combustion, International Journal of Hydrogen Energy 34, 8339-8351.

[9] Nozomu Hashimoto, Yasushi Ozawa, Noriyuki Mori, Isao Yuri, TohruHisamatsu (2008). Fundamental combustion characteristics of palm methyl ester (PME) as alternative fuel for gas turbine, Fuel 87, 3373-3378.

[10] Gustavo Rodrigues de Souza, Antonio Moreira dos Santos, Sergio Lucas Ferreira, Keyll Carlos Ribeiro Martins, Delson Luiz Modolo (2009). Evaluation of the performance of biodiesel from waste vegetable oil in a flame tube furnace, Applied Thermal Engineering 29, 2562-2566.

[11] Mohamed Harimia, M. M. H. Megat Ahmad, S.M. Sapuan, Azni Idris (2005). Numerical analysis of emission component from incineration of palm oil wastes, Biomass and Bioenergy 28, 339–345.

★★★