International Journal of Technology (2016) 2: 204-211
ISSN 2086-9614

CFD SIMULATION OF TURBULENT FLOWS IN PROTO X-3 BIOENERGY MICRO GAS TURBINE COMBUSTOR USING STD *k-ε* AND RNG *k-ε* MODEL FOR GREEN BUILDING APPLICATION

Asyari Daryus[1*], Ahmad Indra Siswantara[1], Steven Darmawan[1,3], Gun Gun R. Gunadi[1,4], Rovida Camalia[1]

[1]Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Kampus UI Depok, Depok 16424, Indonesia
[2]Department of Mechanical Engineering, Universitas Darma Persada, Jakarta Timur 13450, Indonesia
[3]Department of Mechanical Engineering, Universitas Tarumanagara, Jakarta 11440, Indonesia
[4]Department of Mechanical Engineering, Jakarta State Polytechnic, Depok, West Java, Indonesia

(Received: October 2015 / Revised: December 2015 / Accepted: January 2016)

ABSTRACT

This paper presents a numerical analysis of gas flow in the annular combustion chamber of a Proto X-3 Bioenergy micro gas turbine for green building applications. The computational fluids dynamics (CFD) simulation was conducted in two dimensions, turbulent flow and gas phase combustion, with the goal of comparing the effects of different models in real conditions. Two different turbulence models, standard (STD) *k-ε* and renormalization group (RNG) *k-ε,* were applied for simulations. The fuel used was biogas produced from animal waste. Fuel consumption was assumed to be 100 kJ/s for simulations. The results of the simulations were analyzed and compared for reference. The temperature and the mass fraction of CH_4, H_2, O_2, and CO_2 distributions gave almost the same results for both models; therefore, both models (STD *k-ε* and RNG *k-ε*) could be used to represent the combustion process phenomenon without many significant differences.

Keywords: Biogas; CFD simulation; Gas turbine combustor; Green building; Proto X-3 Bioenergy; Turbulent flow

1. INTRODUCTION

A micro gas turbine (MGT) system usually produces power between 25–500 kW (Paepe et al., 2014; Renzi et al., 2014). Besides its high power density, low impact on the environment, and minimal operational and maintenance cost, the micro gas turbine system can also be operated with various kinds of fuel such as solar, ethanol, biomass, CNG, biogas, and so on (Basrawi et al., 2013; Chiaramonti et al., 2013; Paepe et al., 2014; Renzi et al., 2014; Siswantara et al., 2013). This is one of many factors that attracts researchers or manufacturers to develop MGTs, especially for renewable energy fuel types.

A micro combustor is one of the main components of the MGT that burns and converts the chemical energy of fuel to heat energy, and finally drives the gas turbine (Cao & Xu, 2007). A complex chemical reaction takes place in the combustor between fuel and air; its characteristics, such as temperature and pressure, are influenced by the type of fuel used.

In addition, the utilization of renewable energy sources is now of great interest to many

---

[*] Corresponding author's email: asyari@ui.ac.id, Tel. +62-21-7270032, Fax. +62-21-7270033

Permalink/DOI: http://dx.doi.org/10.14716/ijtech.v7i2.2978

researchers and industries, since it has the potential to resolve the fossil fuel crisis. One such energy source is biogas, which comes from organic substances such as agricultural or municipal waste that has been processed in an aerobic digester (Alexopoulos, 2012; Coughtrie et al., 2013).

It could be used as fuel for a micro gas turbine (Alexopoulos, 2012); it also does not cause pollution to the environment (Alexopoulos, 2012), so it is suitable for use in urban areas.

Understanding the fluid flow phenomena in a gas turbine combustor can help designers to create a better machine. These phenomena involve very complex mathematical equations; modeling these equations could make the calculation easier. The turbulent models are used to describe the turbulence flow in fluids; the k-ε model is one of those that express the turbulent viscosity by two transport parameters, namely turbulent kinetic energy, k and turbulent dissipation rate, ε. There are some models for k-ε; two of these are the STD k-ε model (standard k-ε model) and the RNG k-ε model (renormalization group k-ε model). These will be used in this paper.

The use of numerical simulation techniques to investigate the phenomena in fluids can save time and money since they do not require expensive, complex, and difficult experiments to collect data, but rather a set of computational equipment using numerical methods (Bicsak et al., 2012; Bulat et al., 2011; Mare et al., 2004). CFD (Computational Fluid Dynamics) is a computational numerical method that can investigate the fluid flow phenomena and simulate it with CFD software.

The characteristics of fluid flow in specific combustors are unique; as each combustor has different attributes, then a particular turbulent model might be more appropriate than others. From the two models mentioned above, it is worthwhile to know which one is more appropriate to describe the flow phenomena in a Micro Gas Turbine Bioenergy Proto X-3 combustor using biogas fuel.

The Micro Gas Turbine Bioenergy Proto X-3 is a prototype turbine designed to be able to operate on any kind of fuel, especially the renewable energy types, and is applicable in green building concepts. This prototype was developed to use several types of fuel, such as solar, bio-ethanol, LPG, and biogas; to take various loads, such as heat exchanger and small generator; and to operate in various configurations.

The aim of this paper is to find the most appropriate model of turbulent flow to investigate the flow phenomena in the combustor of the Micro Gas Turbine Proto X-3 Bioenergy, from the STD k-ε and RNG k-ε models, using the CFD method. The simulation results will be compared to references and discussed

2. COMPUTATIONAL MODELS AND METHODS

Two turbulence models were used to simulate the biogas combustion flow in the combustor: the STD k-ε model and the RNG k-ε model. Turbulence intensity was assumed to be 10% for each model. Meanwhile, the finite rate eddy dissipation combustion model was used for combustion.

CFD was used to calculate the flow of fluids in the combustor. The simulation procedure using CFD is divided into three steps: pre-processing, processing, and post-processing. Pre-processing consists of geometry and mesh creation and boundary setup, while processing consists of setting up conditions and finding a solution. Post processing entails obtaining results (graphics, vector, and contours) and analyzing them.

CFDSOF® software was used for all three steps of the simulation. The CAD model of combustor is presented in Figure 1. The combustor was an annular type, 180 mm long and 100 mm in diameter. A 2D Cartesian-type mesh of 12,740 nodes was designed for the simulation (Figure 2). A 2D geometry model was used instead of 3D for simplicity and to reduce the calculation time; a 2D model was also able to provide sufficient description of what was happening in the combustor.

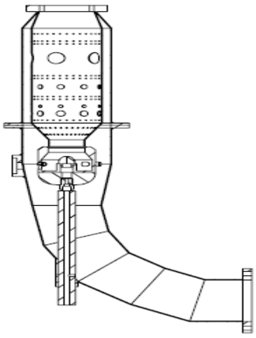

Figure 1 CAD model of combustor

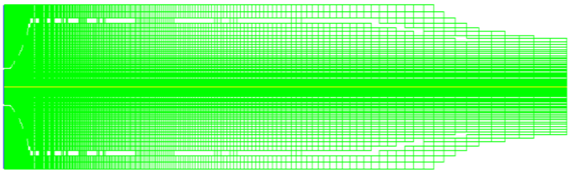

Figure 2 Meshed geometry

The experimental data for the simulation, such as flow rate of fuel and air, working pressure, and temperatures, was collected. The experimental setup consisted of a micro gas turbine, a compressor (both constructed from an automobile turbocharger), an annular-type combustor, and a cross-flow-type turbine as a second stage turbine that connected to the generator to produce electricity (Figure 3).

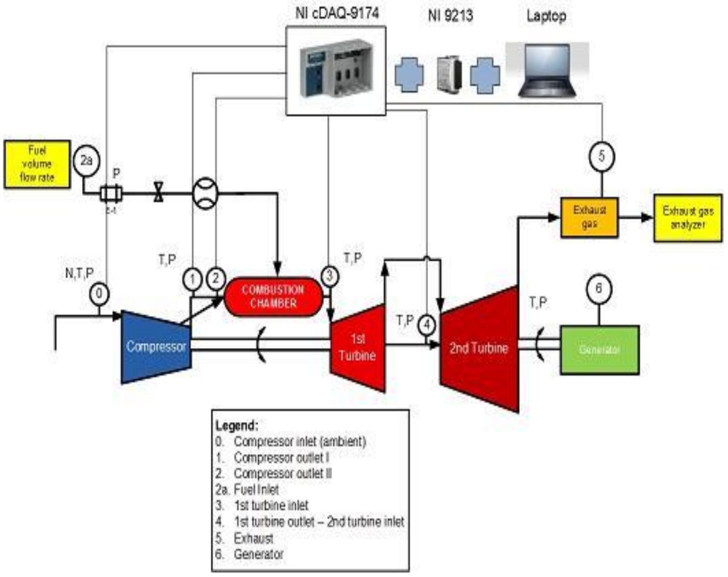

Figure 3 Experimental setup

LPG fuel experiments data was used, on the assumption that its condition parameters do not differ much from biogas. Air entered the combustor at 322 K temperature, under atmospheric pressure, and at a rate of 0.75 kg/s. The fuel entered through the small pipe in the middle, whereas air came from the surroundings. To reach a turbine speed of around 80,000 rpm, 100 kW energy of biogas was needed, or around 0.005 kg/s. Fuel was injected at 297 K. Biogas was assumed to be CH_4 (60% mass wt.), CO_2 (30% mass wt.), and H_2 (10% mass wt.); the other species were negligible due to their very small quantities.

The results of the simulation for both models were compared to Bhalerao and Pawar's works (2012) which conducted 3D simulations of CH_4 in a gas turbine combustion chamber using an LES (large eddies simulation) model. This model is assumed to give more precise results than the *k*-*ε* model, but involves more complex equations and needs more powerful computation tools to solve.

3. RESULTS AND DISCUSSION

Figures 4 and 5 show the temperature distribution along the length of the combustor. The highest temperature was around 1916 K, found in the STD *k*-*ε* model, while in the RNG *k*-*ε* model it was a little lower at around 1874 K; this was the case in the diluted zone. The temperature at the outlet was 1160 K for the STD *k*-*ε* model and 1250 K for the RNG *k*-*ε* model. The cross section from the nozzle to the tip of the flame shows that the maximum temperature was at the center and was reduced towards the wall; this was the same for both models. The wall temperature rose from the primary zone to the secondary zone, and fell in the tertiary zone. The temperature distributions between the two models were almost identical, both in length and radial direction. These results are close to those of Bhalerao and Pawar.

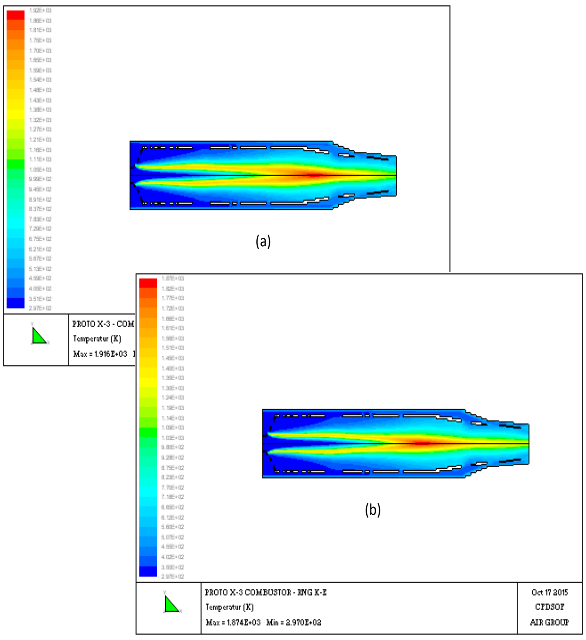

Figure 4 Temperature distribution: (a) STD *k*-*ε* model; (b) RNG *k*-*ε* model

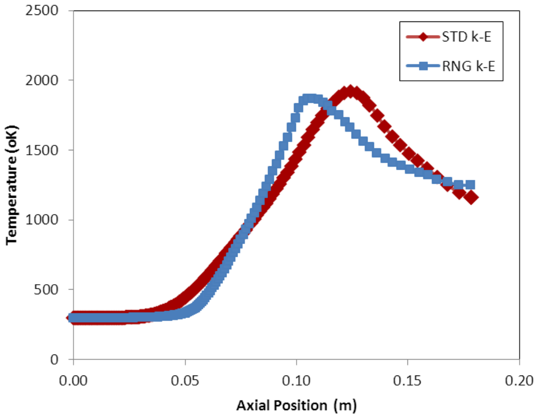

Figure 5 Temperature distribution along axis of combustor

Figures 6 and 7 show the contour and plot of the mass fraction of CH_4. It can be seen that all the methane was burnt in the primary zone. Both models, as well as Bhalerao and Pawar's work, showed similar results.

All hydrogen was also burnt in the primary zone, as can be seen in Figures 8 and 9, but the STD *k*-*ε* model had a longer path in the length direction than that of the RNG *k*-*ε* model. It can be concluded that all reactant materials of a biogas reaction are burnt completely in a combustor; therefore, emissions might be free of hydrocarbon compounds. Both models showed similar results.

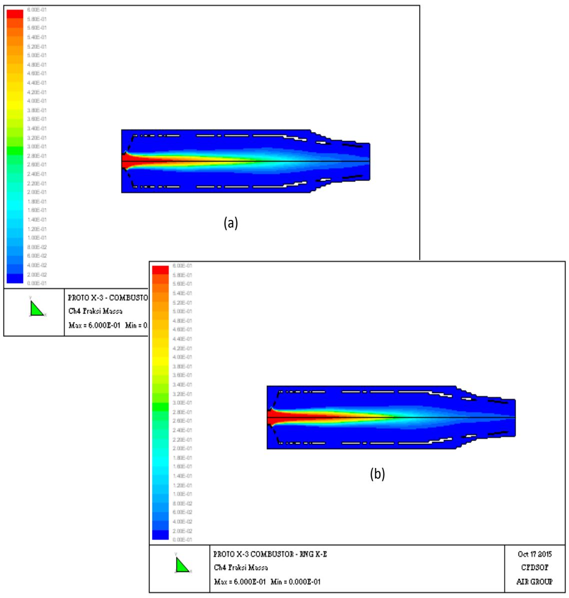

Figure 6 Contour of mass fraction of CH_4: (a) STD k-ε model; (b) RNG k-ε model

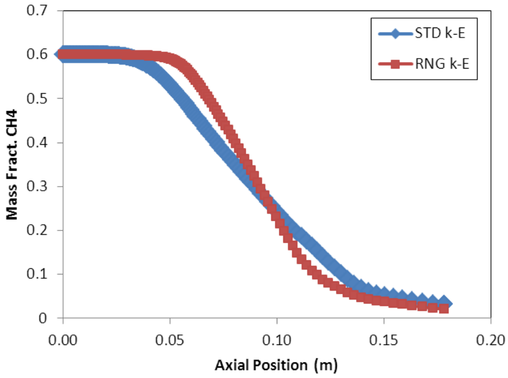

Figure 7 Mass fraction of methane along axis of combustor

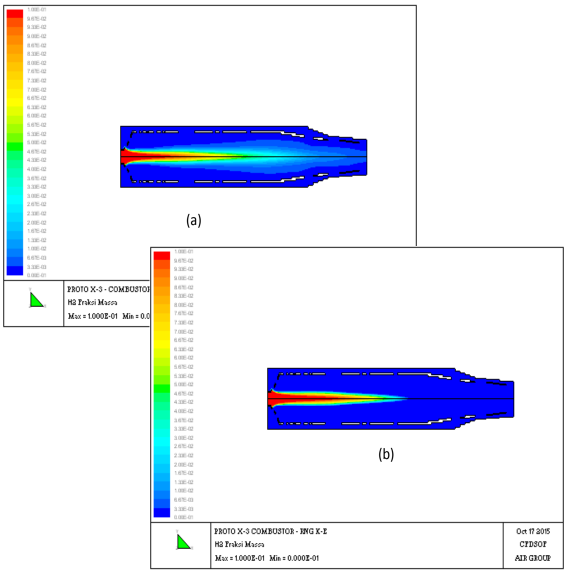

Figure 8 Contour of mass fraction of H_2: (a) STD k-ε model; (b) RNG k-ε model

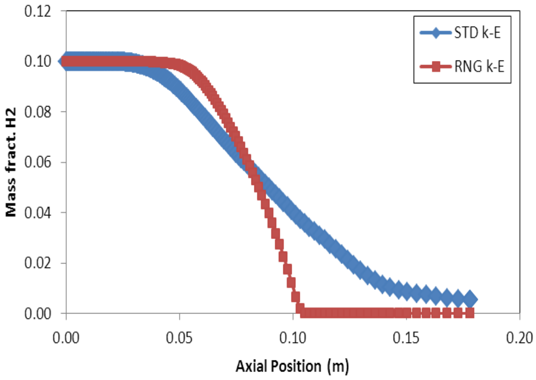

Figure 9 Mass fraction of H_2 along axis of combustor

Based on the oxygen distribution graph in Figures 10 and 11, the oxygen was mostly used in the axis of the combustor, as signified by the lower mass fraction in this area. Again, similar results were shown in Bhalerao and Pawar's works. There were no significant differences between the models.

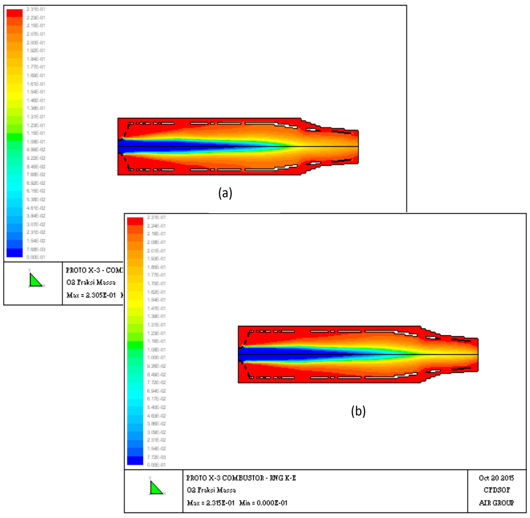

Figure 10 Contour of mass fraction of O_2: (a) STD *k*-*ε* model; (b) RNG *k*-*ε* model

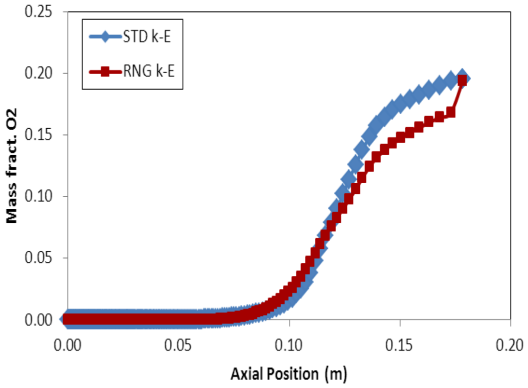

Figure 11 Mass fraction of O_2 along axis of combustor

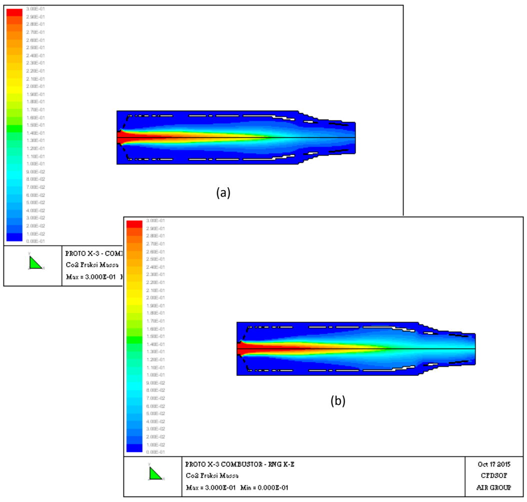

Figure 12 Contour of mass fraction of CO_2: (a) STD *k*-*ε* model; (b) RNG *k*-*ε* model

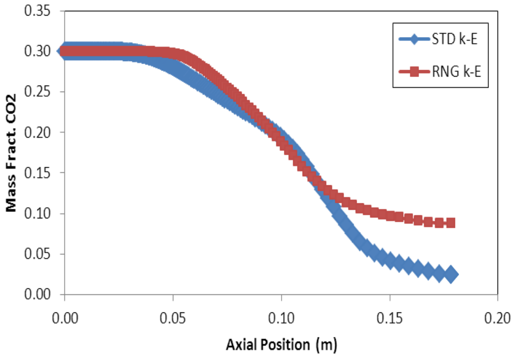

Figure 13 Mass fraction of CO_2 along axis of combustor

Carbon dioxide and water are products of combustion. There was also no significantly different result for CO_2 and H_2O distribution between models, as shown in Figures 12 to 15. A small difference was seen in the CO_2 mass fraction at the tertiary zone or near the outlet, where the RNG *k*-*ε* model had a higher concentration than that of the STD *k*-*ε* model; 0.08 vs. 0.02.

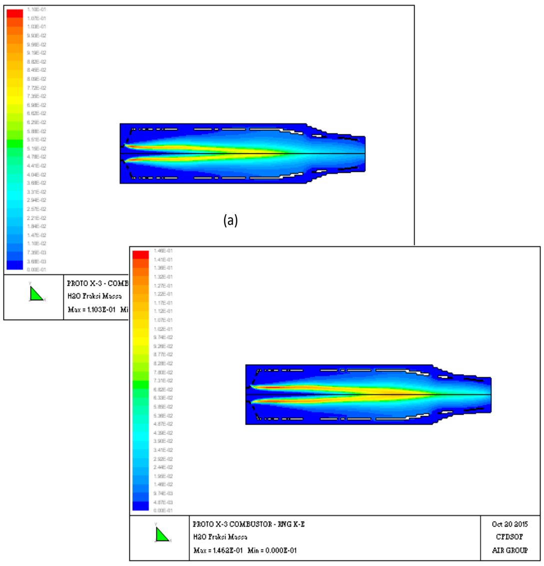

Figure 14 Contour of mass fraction of H_2O: (a) STD k-ε model; (b) RNG k-ε model

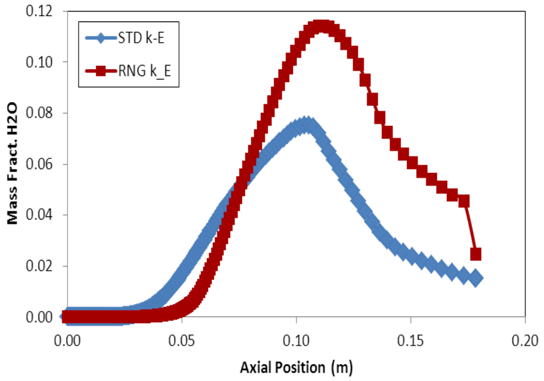

Figure 15 Mass fraction of H_2O along axis of combustor

A minor difference is also seen in the H_2O graph, where the RNG k-ε model had a higher maximum mass fraction than that of the STD k-ε model at the secondary zone; 0.11 vs. 0.07. The mass fraction of the H_2O graphs shows similar results to those of Bhalerao and Pawar. Overall, both models gave similar results.

The temperature and mass fraction of CH_4, H_2, O_2, CO_2, and H_2O distributions from the simulations showed that both models gave almost identical results. Some plots might be different, but were measured along the axis of the combustor; whereas for the entire area, there seem to be no significant differences.

From this study, it can be concluded that both models (STD k-ε and RNG k-ε) could be used to simulate the combustion process in this micro combustor without many major differences. To obtain better results, the RSM (Reynolds Stress Model) could be used, since it can solve complex strain fields or significant body forces, unlike the models used in this study (Versteeg & Malalasekera, 2007).

4. CONCLUSIONS

The turbulence model used in CFD simulations plays an important role in the precise prediction of actual flow conditions in a combustor, as not every model is appropriate for every condition. This paper presented the results of a CFD simulation in a Micro Gas Turbine Bioenergy Proto X-3 combustor to determine which model is best suited to its conditions out of two k-ε turbulence models: STD k-ε and RNG k-ε. The results are as the following; (1) the highest temperature was found in the secondary zone, at around 1916 K for the STD k-ε model and 1874 K for the RNG k-ε model. The outlet temperatures were 1160 K and 1250 K for the STD k-ε and RNG k-ε models, respectively. Both models gave broadly similar results; (2) methane and hydrogen were burnt completely in the primary zone. The STD k-ε model took longer to burn hydrogen than the RNG k-ε model; (3) the CO_2 mass fraction at the tertiary zone or near the outlet was found to be higher in the RNG k-ε model than in the STD k-ε model, i.e. 0.08 vs. 0.02. This was also the case for H_2O at the secondary zone, i.e. 0.11 vs. 0.07; (4) overall, the mass fraction of CH_4, H_2, O_2, CO_2, and H_2O was almost identical for both models. Therefore, any model could be used to model the combustion process in a micro gas turbine combustor; (5) more combustion cases are required to be conducted with these two turbulence models to decide the appropriate model; more advanced turbulence models, such as the RSM (Reynolds Stress Model), may be needed.

5. ACKNOWLEDGEMENT

The authors would like to thank the Ministry of Research, Technology, and Higher Education, Republic of Indonesia through "Hibah Insentif SINas 2015", as well as PT. CCIT Group Indonesia for the CFDSOF® software license.

6. REFERENCES

Alexopoulos, S., 2012. Biogas Systems: Basics, Biogas Multifunction, Principles of Fermentation and Hybrid Application with a Solar Tower for Treatment of Waste Animal Manure. *Journal of Engineering Science and Technology Review*, Volume 5(4), pp. 48–55

Basrawi, F., Yamada, T., Obara, S., 2013. Theoretical Analysis of Performance of a Micro Gas Turbine Co/Trigeneration System for Residential Buildings in a Tropical Region. *Journal of Energy and Buildings*, Volume 67, pp. 108–117

Bhalerao, S., Pawar, A.N., 2012. Thermal Mapping of a Can Type Gas Turbine Combustion Chamber using CFD. *International Journal of Emerging Trends in Engineering and Development*, Volume 1(2), pp. 103–110

Bicsak, G., Hornyak, A., Veress, A., 2012. Numerical Simulation of Combustion Processes in a Gas Turbine. In: *the Proceedings of the 9th International Conference on Mathematical Problems in Engineering*, Aerospace Sciences, Vienna, Austria

Bulat, G., Jones, W.P., Marquis, A., Sanderson, V., Stopper, U., 2011. Large Eddy Simulation of a Gas Turbine Combustion Chamber. In: *the Proceedings of the 7th Mediterranean Combustion Symposium*, Chia Laguna, Cagliari, Sardinia, Italy

Cao, H.L., Xu, J.L., 2007. Thermal Performance of a Micro-combustor for Micro-gas Turbine System. *Journal of Energy Conversion and Management*, Volume 48, pp. 1569–1578

Chiaramonti, D., Rizzo, A.M., Spadi, A., Prussi, M., Riccio, G., Martelli, F., 2013. Exhaust Emissions from Liquid Fuel Micro Gas Turbine Fed with Diesel Oil, Biodiesel and Vegetable Oil. *Journal of Applied Energy*, Volume 101, pp. 349–356

Coughtrie, A.R, Borman, D.J, Sleigh, P.A., 2013. Effects of Turbulence Modelling on Prediction of Flow Characteristics in a Bench-scale Anaerobic Gas-lift Digester. *Journal of Bioresource Technology*, Volume 138, pp. 297–306

de Paepe, W., Contino, F., Delattin, F., Bram, S., 2014. Optimal Waste Heat Recovery in Micro Gas Turbine Cycles through Liquid Water Injection. *Journal of Applied Thermal Engineering*, Volume 70, pp. 846–856

di Mare, F., Jones, W.P., Menzies, K.R., 2004. Large Eddy Simulation of a Model Gas Turbine Combustor. *Journal of Combustion and Flame*, Volume 137, pp. 278–294

Renzi, M., Caresana, F., Pelagalli, L., Comodi, G., 2014. Enhancing Micro Gas Turbine Performance through Fogging Technique: Experimental Analysis. *Journal of Applied Energy*, Volume 135, pp. 165–173

Siswantara, A.I., Darmawan, S., Purba, O., 2013. Combustion Analysis of Proto X-2 Bioenergy Micro Gas Turbine with Diesel - Bioethanol Blends. In: *the Proceedings of the 13th International Conference on QIR (Quality on Research)*, Yogyakarta, Indonesia

Versteeg, H., Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics, The Finite Volume Method*, 2nd ed., Pearson Educational Ltd., Essex, England