Journal of Engineering and Applied Sciences 11 (9): 2066-2071, 2016
ISSN: 1816-949X


# CFD Simulation in Annular Combustion Chamber of Proto X-3 Bioenergy Micro Gas Turbine

[1,2]Wilfredo Erenio Yutu Asyari Daryus, [1]Ahmad Indra Siswantara, [1,3]Gun Gun R. Gunadi, [1,4]Steven Darmawan and [1]Rovida Camalia
[1]Department of Mechanical Engineering, Universitas Indonesia, Depok, Indonesia
[2]Department of Mechanical Engineering. Universitas Darma Persada, Jakarta, Indonesia
[3]Department of Mechanical Engineering, Politeknik Negeri Jakarta, Depok, Indonesia
[4]Department of Mechanical Engineering, Universitas Tarumanagara, Jakarta, Indonesia

**Abustract:** Numerical investigation of the combustion in annular combustion chamber of micro gas turbine using Computational Fluid Dynamics (CFD) simulation with CFDSOF® solver is presented in this study. The aim is to investigate the phenomenon of combustion to determine the optimum operating conditions of the combustion chamber that meet the requirements of micro gas turbine and the quality of the emission into the environment. In this study, the mathematical models used for the simulations were finite rate and eddy dissipation for combustion and standard k-ε for turbulent flow. The Compressed Natural Gas (CNG) fuel was used for the simulations and assumed to be 100 kW and 100% methane ($CH_4$). Four different air mass flow rates were tested: 0.1, 0.12, 0.16 and 0.2 kg sec$^{-1}$. The simulations showed that the hottest zone in the combustion chamber was found right in front the dilution zone and it confirmed the results of various experiments and simulations of combustion of methane. The air mass flow rate of 0.12 kg sec$^{-1}$ gave the optimum result for the system because it produced gas with temperature of 1089 K that meets the requirement of the micro gas turbine while the other rates gave higher or lower value. The emissions of the rate of 0.12 kg sec$^{-1}$, in term of $NO_x$, $CO_2$ and $H_2O$, also gave good results.

**Key words:** Micro gas turbine, annular combustion chamber, CFD, turbulence model, LPG

## INTRODUCTION

The Micro Gas Turbine system (MGT) is another form of gas turbine system. It produces power between 25-500 kW (Basrawi *et al*., 2013; Bhalerao and Pawar, 2012; Bicsak *et al*., 2012; Bulat *et al*., 2011). The MGT has some advantages compared to the other prime mover such as high power density, low operation and maintenance cost, low emission, reliability and the flexibility to many kind of fuels (liquid, gaseous, renewable energy fuels and bio-fuels) (Basrawi *et al*., 2013; Chaudhari *et al*., 2012; Chiaramonti *et al*., 2013). The possibility of using renewable energy resources make the MGT suitable as prime mover for green building application (Steven *et al*., 2015; Gokalp and Etienne, 2004; Huicochea *et al*., 2011).

Generally, the gas turbine system consists of a compressor, a combustion chamber or combustor and a turbine. The combustor has the important role to make the system work as expected. It has a heavy task of burning a large quantity of fuels, supplied by a nozzle with extensive volumes of air from the compressor and release the hot air to fulfil the turbine needs. The air fuel ratio for combustion is generally between 50:1 and 200:1 to produce the efficient combustion and to keep the temperature of air coming in to the turbine below the allowable limits (Mare *et al*., 2004). The inlet turbine temperature for micro gas turbine is usually limited to 850-1700°C depends on the blades and nozzles material properties of turbine (Paepe *et al*., 2014). Understanding the flow phenomena in combustion chamber can help engineers to design the efficient combustion chamber and the right operations. The turbulent premixed flame found in combustion chamber is difficult to be investigated because of the complex chemical processes, heavy turbulence and high temperature and pressure (Pathan *et al*., 2012) and furthermore, the observations will need the complex and expensive laser diagnostics and the modified combustion chamber that can be accessed by the optical tools (Praveen and Yadev, 2015). Using Computational Fluid Dynamics (CFD) method, the investigation of flow phenomena can be calculated and simulated that in turn saves money and time. Many

**Corresponding Author:** Wilfredo Erenio Yutu Asyari Daryus, Department of Mechanical Engineering, Universitas Indonesia, Depok, Indonesia

mathematical models have been developed in various fields, especially in fluid dynamics and with the CFD, many fluid problems can be solved, including the combustions. Pathan *et al.* (2004) had carried out the numerical simulation of combustion of methane air mixture in can combustor of gas turbine. The aim of the works was to investigate the phenomena of combustion and its emission. In the study, various parameters like air-fuel ratio, swirler angle of primary air inlet, the change of axial position of dilution holes were investigated to see the effects to the combustion chamber performances and the emissions. From the three different tested swirler angle, 30°, 45° and 60°, the 60° swirler angle gave less NO emission. The increment of temperature at reaction zone due to the combustion of methane and the decrements in temperature downstream of dilution holes due to the air entering the dilution holes had increased the NO mass fraction, whereas the shifting of dilution holes have no much variation in temperature and NO emission. Bhalerao and Pawar (2012) and Renzi *et al.* (2014) had investigated the thermal mapping of a can type gas turbine combustion chamber using CFD. The simulations had been done for kerosene fuels which the evaporating species was $CH_4$. The Non-Premixed Combustion model working on the principle of mixture fraction was used for simulating combustion while the realizable k-å model was used for swirling flow. The results proved that simulations using CFD were efficient and reliable to understand the fluid flow behaviours through complex geometries. Chaudhari *et al.* (2012) and Sim *et al.* (2013) had included the CFD Method in designing the annular combustion chamber for 20 kW annular gas turbine power generation with kerosene.

The k-ε turbulence model was used for analysis. From the simulation results found that the streamlines from the wall cooling holes did not cool the gas properly, suggest redesigning the holes. The temperature of gas at the exit and near all the fuel injectors in the primary zone were not uniform that suggests doubling the number of injectors. The numerical simulation showed that the flame touched the chamber liner decreased the durability of liner materials. Mare *et al.* (2014) used the Large Eddy Simulation (LES) to predict temperature and species concentrations in a can type gas turbine combustor. The subgrid scale stresses were modeled by adopting the standard Smagorinsky-Lilly Model whilst combustion was analyzed by using conserved scalar approach. The results showed that an accurate assumption of the inlet section of combustor influenced the temperature distribution in the primary zone and the injector position. Bulat *et al.* (2011) and Pathan *et al.* (2012) simulated the combustion processes in a 3 bar industrial gas turbine combustor. The simulations were done using the sgs-pdf evolution approach in the context of LES. The chemistry processes used an ARM reduced GRI 3.0 mechanism with 15 reactions and 19 species. The results showed the good agreement between simulations and the experimental data in the flame region. It concluded that the Eulerian stochastic field method was capable of predicting the temperatures and species; the LES could be used to calculate the combustion processes in an industrial burner. From previous researchs, it has succeeded to run the micro gas turbine using solar fuel, but when attempted to run using the gas fuel with the same Air Fuel Ratio (AFR), the system failed to run independently. The inappropriate air supply or mass air fuel ratio was suspected to cause the problem, so the further investigations need to be done to solve the problem. The aim of this research is to investigate the combustion phenomena for determining the optimum operating conditions of combustion chamber that meet the requirements of micro gas turbine prototype called “Proto X-3 Bioenergy Micro Gas Turbine” and the quality of emission to the environtment. The investigations used the CFD simulation using finite rate and eddy dissipation model for combustion and standard k-ε (STD k-ε) turbulence model for gas flows.

## MATERIALS AND METHODS

**Finite rate and eddy dissipation combustion model:** The simulations used the Finite Rate and Eddy Dissipation Combustion Model for combustion process. This model assumes that the chemical reaction is happening quickly in the molecular level relative to the transport process in the flow when the reactants mix (Mare *et al.*, 2004; Paepe *et al.*, 2014). The reaction rate is proportional to the mixing time of reactants. In the turbulent flow, the mixing time is influenced by eddy properties and the reaction rate is then proportional to the mixing time of turbulent kinetic energy and the dissipation.

**Turbulence Model:** The STD k-ε Model is used for modelling turbulent flow. It is one of the most common turbulence models and two equations model type which means that there are two extra transport equations included to present the turbulent properties of flow: transport equation of kinetic energy, k and transport equation of dissipation, ε. The transport equation of k is (Tomczak *et al.*, 2002):

$$\frac{\partial(\rho k)}{\partial t} + \text{div}\ (\rho\ k\mathbf{U}) = \text{div}\left[\frac{\mu_t}{\sigma_k}\text{grad}\ k\right] + 2\mu_t E_{ij}.E_{ij} - \rho\varepsilon \quad (1)$$

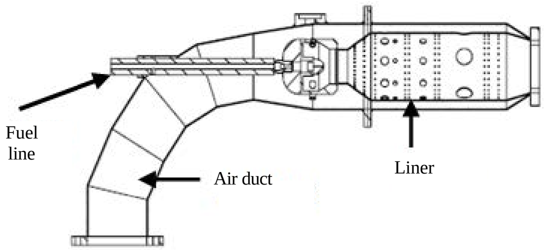


Fig. 1: Combustion chamber

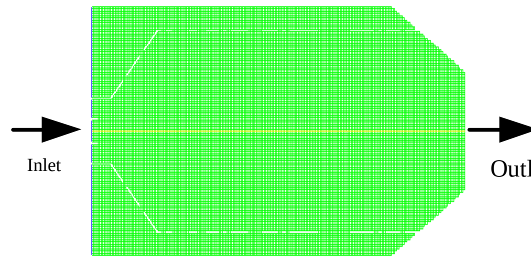


Fig. 2: 2D mesh model of combustion chamber

And the transport equation of ε is (Versteeg and Malalasekara, 2007):

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \text{div}\ (\rho\varepsilon\ \mathbf{U}) = \text{div}\ \left[\frac{\mu_t}{\sigma_\varepsilon}\text{grad}\ \varepsilon\right] + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t\ E_{ij}.E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

And:

$$\mu_t = \rho\ C_\mu \frac{k^2}{\varepsilon}$$

Where:

$\rho$ = A density
U = A velocity vector
$\mu_t$ = A eddy viscosity
$E_{ij}$ = A mean rate of deformation, if i or j = 1 corresponds to the x-direction, i or j = 2 the y-direction and i or j = 3 the z-direction, $C_\mu$, $\sigma_k$, $\sigma_\varepsilon$, $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are contants

**Geometry:** Figure 1 shows the longitudinal view of the combustion chamber. The combustion chamber is annular type with 253 mm total length and 112 mm diameter. The primary air inlet pipe diameter is 30 mm with the 10 mm diameter of fuel injector pipe inside. The fuel injector has only one hole in the end. The secondary air is injected using 3 rows of holes with 9 holes in a row and diameter of 10, 5 and 8 mm for each row sequentially. There are 5 dilution air holes with diameter of 20 mm in the liner.

**Meshing:** The simulations have been done using 2- dimension model because the purpose of the study not to find the precision results but to compare between various mass air flow rates, so the processes could use the medium specification of computational devices and faster. Mesh generation has been done by the commercial CFD Software CFDSOF®, the same software used for the calculations. The mesh was 2-dimension Cartesian type and consisted of 13794 cells. Figure 2 shows the meshed geometry of the combustion chamber.

**Boundary conditions:** The results from previous experiments of this micro gas turbine are used as references for boundary conditions. The fuel used in these simulations is CNG that is mainly composed of methane ($CH_4$). The boundary conditions of fuel are: the mass flow rate is 0.002 kg sec$^{-1}$ equivalent to 100 kW of energy, the inlet temperature is 297 K, the turbulence intensity is 10% and $CH_4$ mass fraction is 1. The boundary conditions of inlet air are: the inlet temperature is 322 K and the turbulence intensity is 10%. The boundary condition of the outlet of combustion chamber is defined by providing relative pressure value which is zero Pascal. The air mass flow rate will be varied to find the best optimum performance of combustion chamber.

## RESULTS AND DISCUSSION

From the previous experiments with gas fuel, where air mass flow rate adjusted from 0.2-0.7 kg sec$^{-1}$, the micro gas turbine system could not run independently. To figure out the problems and to find the solutions, the simulations of combustion processes in the combustion chamber had been conducted and the results had been analysed. From the simulations, the temperature distribution in the combustion chamber using CNG fuel with the air mass flow rate of 0.2 kg sec$^{-1}$ is shown in Fig. 2d, the maximum temperature of combustion is 596 K; this result is not many different with the experiments. This temperature is so low to drive the gas turbine. The other simulations done with higher air mass flow rate did not give the better results, even the temperature decreased.

Figure 3 shows the temperature distributions along the axial length of liner for various air mass flow rates. From these distributions it is observed that in 0.1 kg sec$^{-1}$ air mass flow rate or mass Air Fuel Ratio (AFR) 50, the highest temperature found is 1450 K (Fig. 3a). For the air mass flow rate of 0.12 kg sec$^{-1}$ (AFR = 60), 0.16 kg sec$^{-1}$ (AFR = 80) and 0.2 kg sec$^{-1}$ (AFR = 100), the maximal temperature are 1089, 769 and 596 K, respectively. The pictures also give that the high temperature area exists in

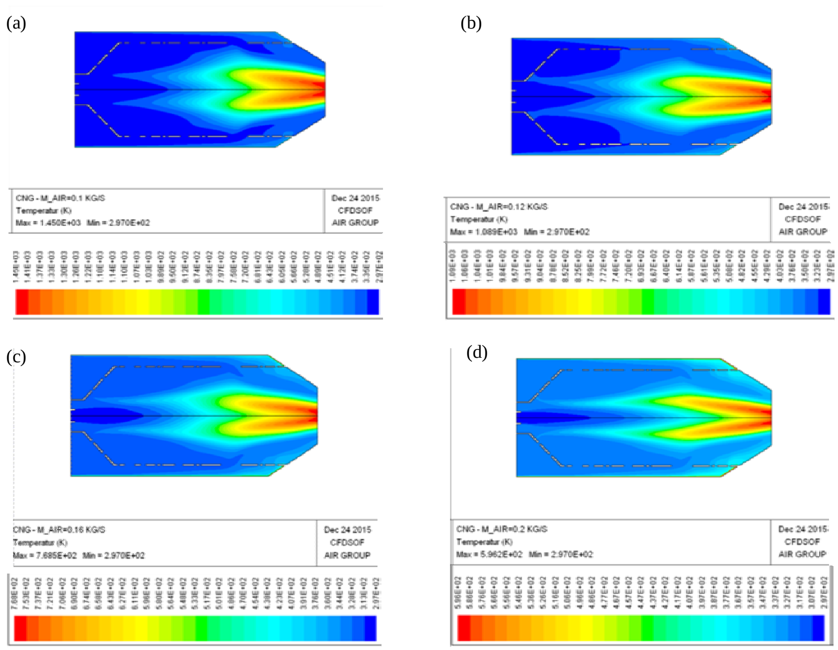


Fig. 3: Temperature distribution along the length of liner. a) air mass flow rate is 0.1 kg sec$^{-1}$ (AFR = 50), b) air mass flow rate is 0.12 kg sec$^{-1}$ (AFR = 60), c) air mass flow rate is 0.16 kg sec$^{-1}$ (AFR = 80) and d) air mass flow rate is 0.2 kg sec$^{-1}$ (AFR = 100)

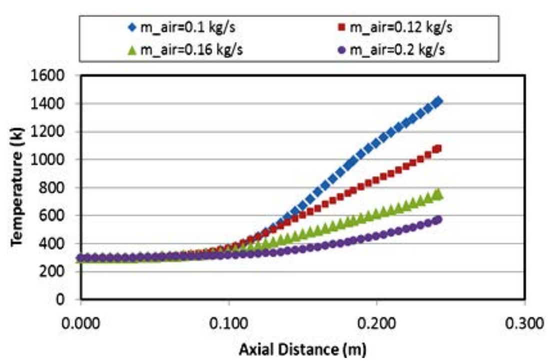


Fig. 4: Profile of temperature along the center line

the dilute zone, near the exit of the combustion chamber. The consequence is that the exit gas temperature to the turbine is still high.

This is the characteristic of burning methane as investigated by Tomczak *et al*. (2002) and Versteeg and Malalasekara (2007) that the hottest zone of burning natural gas was right in front of the dilution zone. This results confirm the previous similar simulations or experiments.

The higher inlet gas temperature of turbine will give the better efficiency but there is limitation from the blade material. Recommended inlet gas temperature to this micro gas turbine is not more than 1300 K. From the outlet gas temperature seen in Fig. 3, the suitable outlet gas temperature is 1080 K from the 0.12 kg sec$^{-1}$ air mass flow rate, meanwhile the lower air mass flow rate gives the higher temperatures and vice versa. From the charts in Fig. 4 it is observed that the exit gas temperature is highest in 0.1 kg sec$^{-1}$ air mass flow rate, respectively followed by 0.12, 0.16 and 0.2 kg sec$^{-1}$. The $NO_x$ emission is directly proportional with the temperature, the higher the exit gas temperature the more the NO emission (Mare *et al*., 2004). So, the case of 0.2 kg sec$^{-1}$ air mass flow rate is the best compare to the others but the conditions applicable to the system are only the first two of low air mass flow rate and between these two, the 0.12 kg sec$^{-1}$ air mass flow rate gives better NO emission. The oxygen profile can be seen in Fig. 5, where the fastest air mass flow rate also has the highest mass fraction at outlet. High mass fraction of oxygen at outlet indicates that more air is fed to the combustion chamber. The products of combustion are $CO_2$ and $H_2O$. The $CO_2$ profile along the centre line of combustion chamber is shown in Fig. 6. It is observed from the profile that the mass fraction of $CO_2$ is increasing toward the outlet. It is caused that the combustion mostly takes place right in front the dilution zone. At outlet position, less mass fraction of $CO_2$ is found at larger air mass flow rate. The air mass flow rate of 0.12 kg sec$^{-1}$ has less mass fraction of $CO_2$ than the air mass flow rate of 0.1 kg sec$^{-1}$ but more than the 0.16 and 0.2 kg sec$^{-1}$. Since, the air mass flow rates that applicable to the system are only the 0.1 and 0.12 kg sec$^{-1}$, the $CO_2$

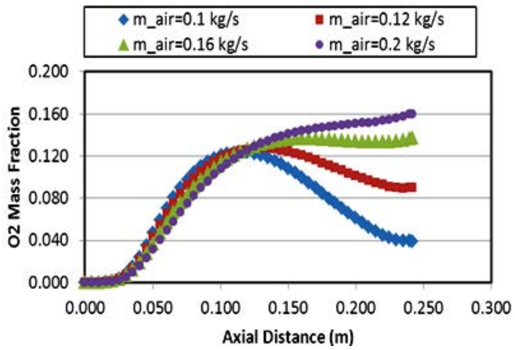


Fig. 5: Profile of $O_2$ mass fraction along the center line

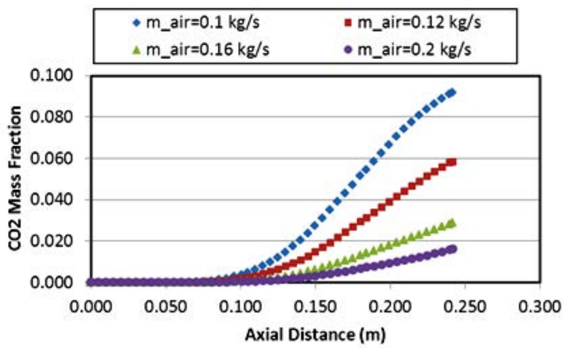


Fig. 6: Profile of $CO_2$ mass fraction along the center line

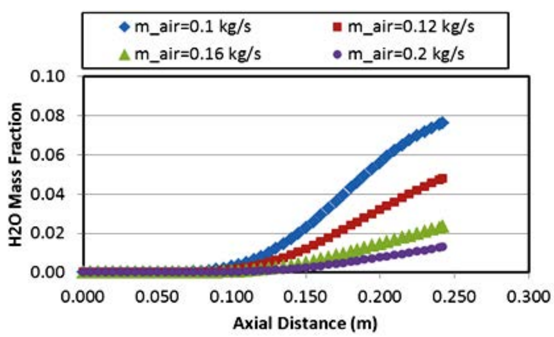


Fig. 7: Profile of $H_2O$ mass fraction along the center line

mass fraction of 0.12 kg $sec^{-1}$ is still better. The similar results are found in mass fraction of $H_2O$ where the mass fraction of $H_2O$ of air mass flow rate of 0.12 kg $sec^{-1}$ is better than the 0.1 kg $sec^{-1}$ (Fig. 3). Temperature distribution along the length of liner. Air mass flow rate is 0.1 kg $sec^{-1}$ (AFR = 50), air mass flow rate is 0.12 kg $sec^{-1}$ (AFR = 60), air mass flow rate is 0.16 kg $sec^{-1}$ (AFR = 80) and air mass flow rate is 0.2 kg $sec^{-1}$ (AFR = 100) (Fig. 7).

## CONCLUSION

The CFD simulations were done to the annular combustion chamber of Proto X-3 Bioenergy Micro Gas Turbine using finite rate and eddy dissipation combustion model and standard k-ε turbulence model, 100 kW CNG fuel consisted of 100% $CH_4$ and four different air mass flow rates to investigate the phenomenon of the combustion. The hottest zone was found right in front of the dilution zone and confirmed the results of various experiments and simulations of combustion of methane from various researches. From four different air mass flow rates flowing into the combustion chamber: 0.1, 0.12, 0.16 and 0.2 kg $sec^{-1}$, the optimum outlet gas temperature for micro gas turbine was 1089 K at a rate of 0.12 kg $sec^{-1}$ meanwhile the gas temperature of 1450 K resulted from the rate of 0.1 kg $sec^{-1}$ was too high and the gas temperature of 769 and 596 K from the rate of 0.16 and 0.2 kg $sec^{-1}$, respectively were low that did not meet the requirements of micro gas turbine. The $NO_x$ emission is directly proportional with the temperature, so the $NO_x$ emission of rate of 0.12 kg $sec^{-1}$ would be better than that of 0.1 kg $sec^{-1}$ but worse than that of 0.16 and 0.20 kg $sec^{-1}$.

From the $CO_2$ emission, the minimum value was found in the rate of 0.2 kg $sec^{-1}$ and the maximum in the rate of 0.1 kg $sec^{-1}$, the same condition exists for $H_2O$. Both emissions of the rate of 0.12 kg $sec^{-1}$ existed in the medium level. From all the results, it can be concluded that the air mass flow rate of 0.12 kg $sec^{-1}$ will give the optimum performances of micro gas turbine system.

## ACKNOWLEDGEMENTS

The researchers would like to thanks Direktorat Riset dan Pengabdian kepada Masyarakat (DRPM) Universitas Indonesia for funding this research through “Hibah Pasca Sarjana 2015”.

## NOMENCLATURE

- MGT: Micro Gas Turbine system
- LES: Large Eddy Simulation
- k: Transport equation of kinetic energy
- ε: transport equation of dissipation
- $CO_2$: Carbon dioxide
- $H_2O$: Water
- ρ: A density
- U: a velocity vector
- $\mu_t$: A eddy viscosity
- $E_{ij}$: A mean rate of deformation, if i or j = 1 corresponds to the x-direction, i or j = 2, the y-direction and i or j = 3 the z-direction
- $C_\mu$: Contants
- $\sigma_k$: Contants
- $\sigma_\varepsilon$: Contants
- $C_{1\varepsilon}$: Contants
- $C_{2\varepsilon}$: Contants
- $\mu_t = \rho C_\mu(k^2/\varepsilon)$: A eddy viscosity

## REFERENCES

Basrawi, F., T. Yamada and S.Y. Obara, 2013. Theoretical analysis of performance of a micro gas turbine co-trigeneration system for residential buildings in a tropical region. Energy Build., 67: 108-117.

Bhalerao, S. and A.N. Pawar, 2012. Thermal mapping of a can type gas turbine combustion chamber using CFD. Int. J. Emerging Trends Eng. Dev., 1: 103-110.

Bicsak, G., A. Hornyak and A. Veress, 2012. Numerical simulation of combustion processes in a gas turbine. Proceedings of the 9th International Conference on Mathematical Problems in Engineering, Aerospace and Sciences (ICNPAA), November 10-14, 2012, AIP Publishing, Vienna, Austria, pp: 140-148.

Bulat, G., W.P. Jones, A. Marquis, V. Sanderson and U. Stopper, 2011. Large eddy simulation of a gas turbine combustion chamber. Proceedings of the 7th Conference on Mediterranean Combustion Symposium, September 11-15, 2011, Chia Laguna, Sardinia, Italy, Cagliari, pp: 1-12.

Chaudhari, K.V., D.B. Kulshreshtha and S.A. Channiwala, 2012. Design and CFD simulation of annular combustion chamber with kerosene as fuel for 20 kW gas turbine engine. Int. J. Eng. Res. Appl., 2: 1641-1645.

Chiaramonti, D., A.M. Rizzo, A. Spadi, M. Prussi and G. Riccio *et al*., 2013. Exhaust emissions from liquid fuel micro gas turbine fed with diesel oil, biodiesel and vegetable oil. Appl. Energy, 101: 349-356.

Gokalp, I. And L. Etienne, 2004. Alternative fuels for industrial gas turbines (AFTUR). J. Appl. Thermal Eng., 24: 1655-1663.

Huicochea, A., W. Rivera, G.G. Urueta, J.C. Bruno and A. Coronas, 2011. Thermodynamic analysis of a trigeneration system consisting of a micro gas turbine and a double effect absorption chiller. Appl. Thermal Eng., 31: 3347-3353.

Mare, D.F., W.P. Jones and K.R. Menzies, 2004. Large eddy simulation of a model gas turbine combustor. Combust. Flame, 137: 278-294.

Paepe, D.W., F. Contino, F. Delattin, S. Bram and D.J. Ruyck, 2014. Optimal waste heat recovery in micro gas turbine cycles through liquid water injection. Appl. Thermal Eng., 70: 846-856.

Pathan, H., K. Partel and V. Tadvi, 2012. Numerical investigation of the combustion of methane air mixture in gas turbine can-type combustion chamber. Int. J. Sci. Eng. Res., 3: 1-7.

Praveen, C.U. and A.H.K. Yadav, 2015. Numerical simulation of gas turbine can combustor engine. Int. J. Eng. Res. General Sci., 3: 192-201.

Renzi, M., F. Caresana, L. Pelagalli and G. Comodi, 2014. Enhancing micro gas turbine performance through fogging technique: Experimental analysis. Appl. Energy, 135: 165-173.

Sim, K., B. Koo, C.H. Kim and T.H. Kim, 2013. Development and performance measurement of micro-power pack using micro-gas turbine driven automotive alternators. Appl. Energy, 102: 309-319.

Steven, D., I.S. Ahmad, Budiarso, A. Daryus and A.T. Gunawan *et al*., 2015. Turbulent flow analysis in auxiliary cross-flow runner of a proto X-3 bioenergy micro gas turbine using RNG k-e Turbulence Model. ARPN. J. Eng. Apl. Sci., 10: 7086-7091.

Tomczak, H.J., G. Benelli, L. Carrai and D. Cecchini, 2002. Investigation of a gas turbine combustion system fired with mixtures of natural gas and hydrogen. IFRF. Combust. J., 1: 1-19.

Versteeg, H.K. and W. Malalasekera, 2007. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. 2nd Edn., Pearson Education Limited, New York, USA., ISBN-13: 978-0131274983, Pages: 520.