KEMENTERIAN PENDIDIKAN TINGGI

JOJAPS

eISSN 2504-8457

POLITEKNIK MALAYSIA PORT DICKSON

Journal Online Jaringan COT POLIPD (JOJAPS)

Fluid Flow Analysis of Micro Gas Turbine Using Computational Fluid Dynamics (CFD)

Eko Prasetyo, Rudi Hermawan, Angger Liyundira Putra, D.L Zariatin

Mechanical Engineering Pancasila University, Jakarta, 16230, Indonesia

Abstract

The needs of micro-scale power plant for small and middle residential or industry are beginning to increase, especially for continuous power generation in remote area, emergency standby and peak shaving. There are several types of micro-scale power plant commonly used, such as micro-steam turbine, micro-hydro turbine, micro-gas turbine (MGT), etc. Among those micro-scale power plant, MGT has the highest performance. Therefore, it is a promising technology to develop. In MGT, an automotive turbocharger could be used as the gas generator or and the turbine because it has a compact structure, light weight and commercially available.

In order to design a micro-gas turbine, it is necessary to find the optimum design of compressors such as the outlet compressor diameter, the combustion chamber and turbine. This paper describes the analysis of fluid flow in a micro-gas turbine that used an automotive turbocharger as the compressor and the turbine of MGT to generate electric power by using Liquefied Petroleum Gas (LPG). Three models with different diameter of compressor outlet were developed and analyzed. The analysis shows that the compressor outlet diameter of 30 mm has a better velocity distribution, higher Mach number and higher turbulent intensity among other models. The combustion chamber and the turbine also analyzed by using Computational Fluid Dynamics (CFD). The combustion chamber analysis indicates that the combustion of C3H8 and the air were mixed perfectly with maximum chamber temperature of 905.56 K. Meanwhile the turbine analysis indicates that the turbine rotated with a speed of 4087.96 m/s.

Key-word: - Micro Gas Turbine (MGT), CFD, Turbocharger, LPG

1. Introduction

The needs of micro-scale power plant for small and middle residential or industry are beginning to increase, especially for continuous power generation in remote area, emergency standby and peak shaving. There are several types of micro-scale power plant commonly used, such as micro-steam turbine, micro-hydro turbine, micro-gas turbine (MGT), etc. Among those micro-scale power plant, MGT has the highest performance, in term of amount energy generated, low pollution and clean exhaust, etc. (do Nascimento, et al., 2013). MGT is a promising portable technology to generate power in a remote area.

In a micro-gas turbine, the kinetic energy is converted into mechanical energy that results in a rotation that can drive the turbine wheel so as to generate a power. Therefore, turbo machinery components, namely: the compressor and the turbine are very essential to the system performance, especially for supplying combustion air compressor (Darmawan, 2011).

On the other hand, an automotive turbocharger could be used as a turbine in MGT. Yamashita et al (Yamashita, Kuwabara, Tatsumi, & Nakabe, 2005) developed a MGT composed of two automobile turbocharger, the first turbocharger played a role as the gas generator of the system, while the other as the power generator turbine. A turbocharger typically consists of a compressor and a turbine coupled to a common shaft. With a compressor that produces exhaust gas to drive turbines, which generate power output of the engine is higher. Turbochargers are always produced by radial flow compressor type for the structure compact, lightweight and high efficiency (Zhu, Deng, & Liu, 2015).

Isomura et al (Isomura, et al., 2004) explained that gas turbine work under a closed Brayton cycle. Hence, it is important to specify the target cycle, clarify and analyze the required performance of each component in the design stage. CFD is one of simulation method tool used to analyzed or even optimize the MGT system performance. Han et al (Han, Seo, Park, Choi, & Do) used 3D CFD to design a 500 W ultra-micro gas turbine generator. Meanwhile Lie et al (Li, Yin, Li, & Zhang, 2013) used CFD to investigate the performance of the turbocharger centrifugal compressor when used as turbine in MGT.

The objective of this research is to analyze the flow in the turbocharger compressor when used as a micro gas turbine. Three variations of compressor outlet diameter, combustion chamber and turbine performance were simulated using CFD software.

2. System Description

Figure 1 shows a simple schematic of a gas turbine. It shows that before entering the combustion chamber, the air is compressed by the compressor. In the combustion chamber, the air temperature is increased by using energy from the fuel, such as LPG. Leaving the combustion chamber, the high temperature working fluid is directed to the turbine, where it is expanded by supplying power to the compressor and for rotating the electric generator or other equipment available.

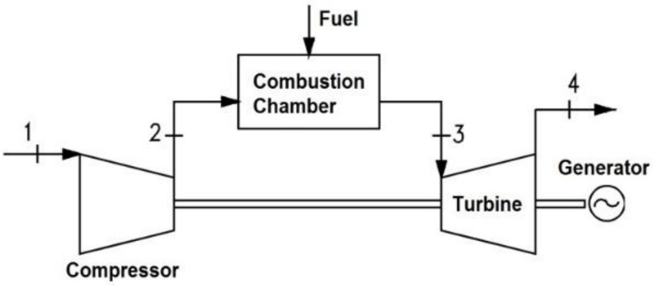


Figure 1 Gas turbine system scheme of a simple cycle (do Nascimento, et al., 2013)

In this research, an RHF series (ball bearing) turbocharger, as shown in Figure 2, was used as the compressor of MGT. The application of turbocharger compressor is to increase the pressure and the flow rate of the air that enter the combustion chamber. In centrifugal compressors, air enters the blade axially. Furthermore, the air will flow in accordance with the profile of the blade toward the rear of the impeller, so that the air will leave the blade radially. The air leaving the blades through the diffuser to the volute, with increased pressure.

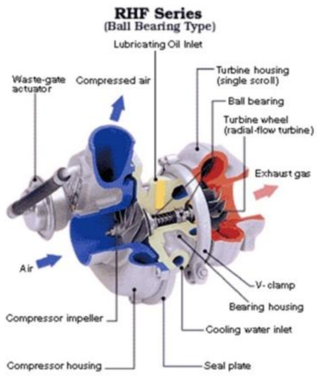
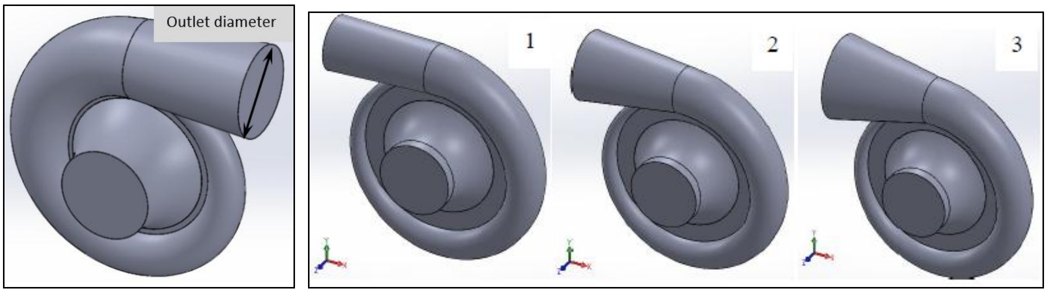


Figure 2 Unit RHF Turbocharger (IHI, 1999-2014)

Figure 3 shows three models with three different outlet diameter of the compressor before entering the combustion chamber. The compressor outlet diameter is listed in Table 1. Meanwhile Figure 4 shows the model of the combustion chamber.



(i) (ii)

Figure 3 Model of the compressors, (i) the outlet diameter, (ii) three compressor models with different outlet diameter

Table 1. Compressor Outlet Diameter

	1st Model	2nd Model	3rd Model
Outlet Compressor Diameter (mm)	30	40	50

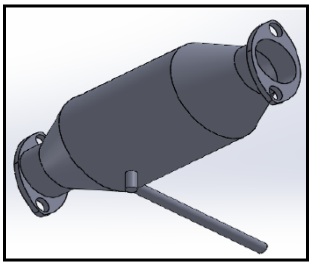


Figure 4 Combustion Chamber Model

3. CFD Analysis

3.1 The Compressor Analysis

By assuming that the turbocharger acted at its maximum design condition based on the manufacturer specification, which is 0.171 kg/s of mass flow rate, pressure ratio of 2.7 ATM, maximum speed of 190,000 RPM and temperature of 298.15 K, the flow analysis was performed. Due to the difficulty and complexity in analyzing the compressor, only the steady flows were analyzed. The analysis was performed using the discretization scheme (pressure-based implicit solver), and the air was set as a compressible flow of ideal gas. Meanwhile the equation of momentum and energy were used the first-order scheme. The turbulent kinetic energy and dissipation rate was solved by using the equation of power law differencing scheme (PLDs). To analyze the velocity-pressure, the algorithm equation Semi-Implicit pressure Linked equation (SIMPLE) were used.

The pressure relaxation factor was 0.8. The momentum, energy turbulence k-epsilon was 0.5. The convergent criteria was 0.0001. Table 2 shows the computation data for the compressor simulation.

Table 2 Computation Data for Turbocharger Compressor

Condition Data			**Reference**
Model setting		3D Steady	
Fluid		Air	
Fluid Properties	Density	Ideal Gas	Fluent Data
	Viscosity	1.7894e-05 kg/m-s	
	Cp	1006.43 J/kg-K	
	Thermal Conductivity	0.0242 W/m-K	
Boundary Condition	Mass Flow Inlet	0.171 kg/s	Input
	Temperature	298.15 K (ref Fluent)	
	Pressure Outlet	0 Pascal	

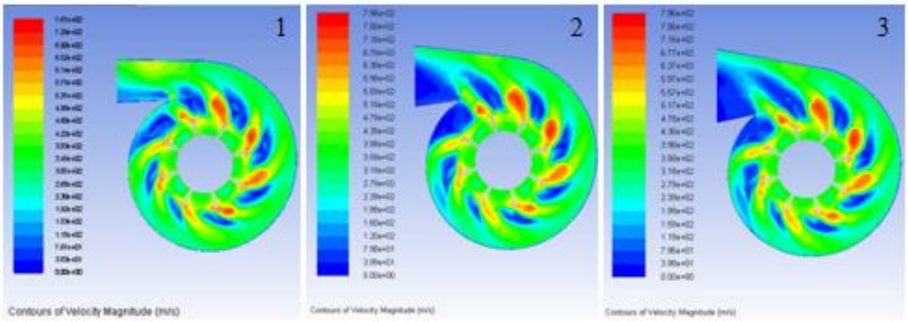


Figure 5 Contour of velocity of compressor outlet 1st, 2nd, and 3rd model

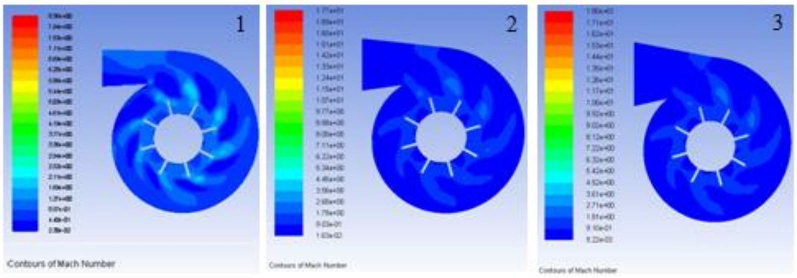


Figure 6 Contour Mach number of compressor outlet 1st, 2nd, and 3rd model

Table 3 Result of the CFD analysis

	1st Model	2nd Model	3rd Model
Velocity (m/s)	383.44	319.15	278.57
Mach Number	1.27	0.91	0.91
Static Temperature (K)	462.09	450.87	435.91

Table 3 indicates that the exit speed on the compressor outlet for the 1st variant is higher compared to others. It has a speed of 383.44 m/s and Mach number of 1.27. It can be concluded that the 1st variant has a better fluid flow distribution when entering the combustion chamber based on the speed and Mach number. Fig 5 shows the contour of velocity of compressor outlet 1, 2 and 3. The air velocity entering the combustion chamber for the 1st model is distributed evenly, indicated by the green color in the compressor outlet as shown in Fig 6 (1). Meanwhile the air velocity entering the combustion chamber for the 2nd and the 3rd model is distributed unevenly. It is indicated by the blue among the green color in the compressor outlet, as shown in Figure 6 (2) and (3).

Figure 6 shows the contour Mach number of the compressor outlet for 1st, 2nd, 3rd model. It shows that the higher Mach number is occurred in the 1st model, indicated by the light blue color in the compressor outlet. The same analysis was also performed for contour turbulent intensity and the static temperature of the compressor outlet, as shown in Figure 7 and 8, respectively.

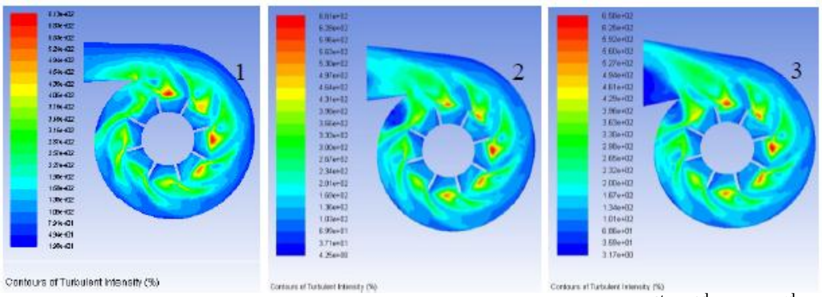


Figure 7 Contour of turbulent intensity of compressor outlet for 1st, 2nd, and 3rd model

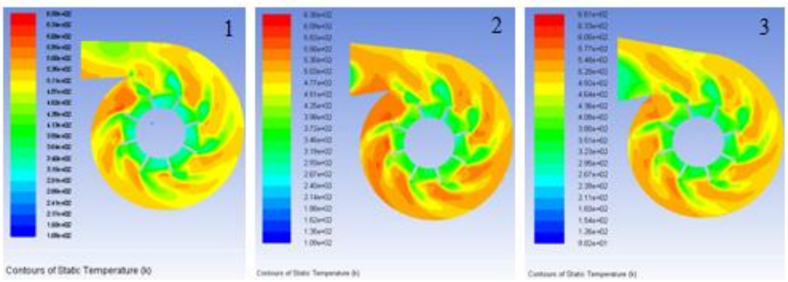


Fig 8. Contour static temperature of compressor outlet for 1st, 2nd, and 3rd model

3.2 The Combustion Chamber Analysis

Table 4 shows the computational data analysis for the combustion chamber. The relaxation factor is 0.2. The momentum, energy and k-epsilon turbulence mode is 0.5. The convergent criteria is 0.0001 for all simulation analysis. The combustion analysis model used was non-premixed combustion-adiabatic with PDF transport species for C_3H_8 (propane).

Table 4 Computation Data in Combustor

Condition Data			**Reference**
Model setting		3D Steady	
Fluid		Air	
Fluid Properties	Density	Ideal Gas	Fluent Data
	Viscosity	1.7894e-05 kg/m-s	
	Cp	1006.43 J/kg-K	
	Thermal Conductivity	0.0242 W/m-K	
Boundary Condition	Pressure Inlet Air	273577.5 Pascal	Input
	Pressure Inlet Gas	883000 Pascal	
	Pressure Outlet	0 Pa	

Figure 9 shows the result of CFD analysis for contour of velocity, static temperature, mass fraction C_3H_8 and mass fraction O_2. Figure 9 (c) and (d) shows that the mixture of C_3H_8 and air is appropriate. The simulation shows that the combustion would be perfect. The maximum temperature at the combustion chamber is 905.56 K with fuel-air mixture velocity of 947.24 m/s.

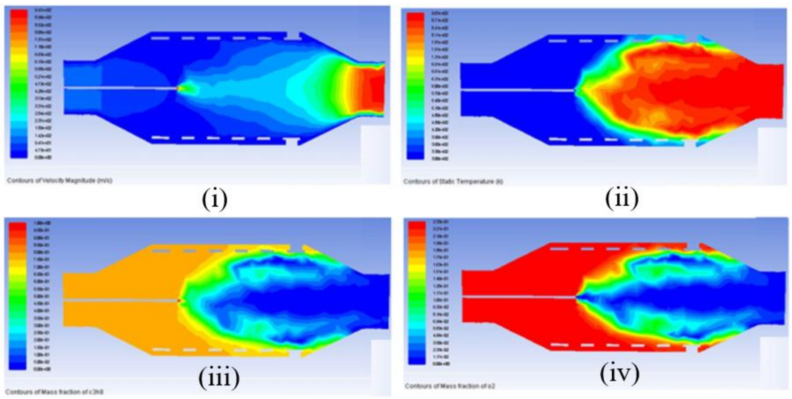


Figure 9 Contour of (i) velocity, (ii) static temperature, (iii) mass fraction of C_3H_8 and (iv)mass fraction of O_2.

3.3 The Turbine Analysis

In Micro Turbine Generator, the hot gases coming from the combustion chamber is accelerated and directed to the guide vanes, and entered into a series of turbines that generate power to drive the compressor and generator. In this research, there are two turbine condition analyzed. Table 5 and 6 shows the computational data for 1st and 2nd turbine model, respectively.

Table 5 Computation Data for 1st Turbine Model

Condition Data			Reference
Model setting		3D Steady	
Fluid		Air	
Fluid Properties	Density	1.225 kg/m	Fluent Data
	Viscosity	1.7894e-05 kg/m-s	
	Cp	1006.43 J/kg-K	
	Thermal Conductivity	0.0242 W/m-K	
Boundary Condition	Velocity inlet	883 m/s	Input
	Temperature	1000 K	
	Pressure Outlet	0 Pascal	

Table 6 Computation Data for 2nd Turbine Model

Condition Data			Reference
Model setting		3D Steady	
Fluid		Air	
Fluid Properties	Density	1.225 kg/m	Fluent Data
	Viscosity	1.7894e-05 kg/m-s	
	Cp	1006.43 J/kg-K	
	Thermal Conductivity	0.0242 W/m-K	
Boundary Condition	Velocity inlet	1132 m/s	Input
	Temperature	1000 K	
	Pressure Outlet	0 Pascal	

The CFD analysis was performed by using the finite volume method and the discretization scheme (pressure-based implicit solver). The parameter of the air is set as a compressible flow as an ideal gas. The momentum equation and energy was used the second order upwind. Meanwhile the turbulent kinetic energy and dissipation rate equations was solved using first order upwind. The analysis of velocity-pressure equation algorithm was perform by The Semi-Implicit pressure Linked equation (SIMPLE). The relaxation factor for the pressure is 0.2. The momentum, energy and k-epsilon turbulence models is 0.2. Meanwhile the convergent criteria used is 0.0001 for all formulations of the equation settlement. The gas produced by the combustion chamber flew directly into the turbine nozzle. Assuming that there is no heat loses during the trip to the gas turbine, the turbine inlet temperature was estimated at 950 ° C or 1223 K.

Fig 10 shows the contour of velocity and static pressure for Turbine 1. The figure shows that the mixture gas comes out from combustion chamber able to drive the turbine. The maximum velocity of gas that comes out from the Turbine 1 reaches 1430.12 m/s. The maximum high pressure output is 1459639.4 Pa. It reached when the turbine rotates on its optimum conditions.

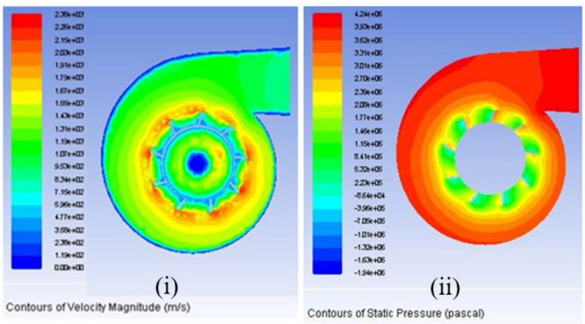


Figure 10 Contour of (i) Velocity and (ii) static pressure of 1st turbine model.

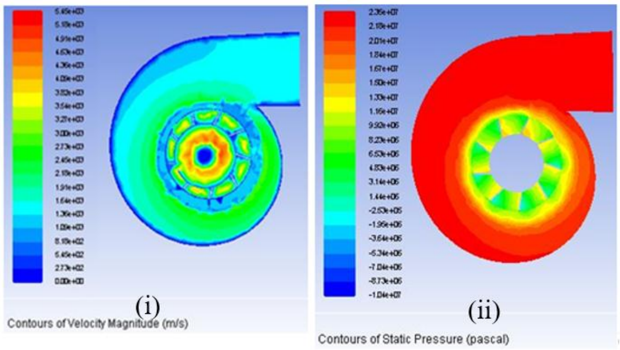


Figure 11 Contour of (i) Velocity and (ii) static pressure for 2nd turbine model.

Figure 11 shows the contour of velocity and static pressure for 1st turbine model. The figure shows that the mixture gas comes out from combustion chamber able to drive the turbine. The maximum velocity of gas that comes out from the 2nd turbine model reaches 1587.96 m/s. The maximum high pressure output is 4378918.2 Pa. It reached when the turbine rotates on its optimum conditions. Comparison of 1st and 2nd turbine model shows that 2nd turbine model condition generates higher velocity and pressure than 1st turbine model.

4. CONCLUSION

Analyses of the results that have been done using CFD methods, it can be concluded as follows:

1. Three different diameter of outlet compressor of 30 mm, 40 mm, and 50 mm were analyzed. The compressor analysis shows that compressor outlet diameter of 30 mm produces better flow distribution based on the contour of velocity, Mach number and turbulent intensity. The velocity, Mach number and turbulent intensity for compressor outlet diameter of 30 mm is 383.44 m/s, 1.27,
2. The combustion chamber analysis indicates that the combustion of C_3H_8 and the air in the chamber will happen perfectly.
3. The turbine analysis for two condition of 1st and 2nd turbine model shows that 2nd turbine model generates higher velocity than 1st turbine model. The 2nd turbine model generate 1587.96 m/s of speed and 4378918.2 Pa of pressure, enough to drive the composer and generator.

Acknowledgements

This research is fund by The Ministry of Research Technology and Higher Education of Indonesia in a grant scheme of “Penelitian Produk Terapan”.

References

Darmawan, S. (2011). *Thesis - Flow analysis on centrifugal compressor blade of micro-gas turbine Proto X-1 (Analisis Aliran Pada Sudu Kompresor Sentrifugal Turbin Gas Mikro Proto X-1).* Depok, Indonesia: Mechanical Engineering Department, Indonesia University.

do Nascimento, M. A., Rodrigues, L. d., Santos, E. C., Gomes, E. E., Dias, F. L., & Carrilo, R. A. (2013). Micro Gas Turbine Engine : A Review. In *Progress in Gas Turbine Performance* (pp. 107-141). InTech.

Han, S., Seo, J., Park, J.-Y., Choi, B.-S., & Do, K. H. (n.d.). Design and simulation of 500W ultra-micro gas turbine generator.

IHI Relize your dream. (1999-2014). (IHI Turbo America.) Retrieved August 6, 2017, from http://www.ihi-turbo.com/product.htm

Isomura, K., Tanaka, S., Togo, S., Kanebako, H., Murayama, M., Saji, N., . . . Esasi, M. (2004). Development of Micromachine Gas Turbine for Portable Power Generation. *JSME International Journal, 47*(3), 459-464.

Li, J., Yin, Y., Li, S., & Zhang, J. (2013). Numerical simulation investigation on centrifugal compressor performance of turbocharger. *Journal of Mechanical Science and Technology, 27*(16), pp 1597-1601.

Yamashita, D., Kuwabara, K., Tatsumi, K., & Nakabe, K. (2005). Experimental evaluation on low-heating value fuel acceptability of micro gas turbine system operation. *16th International Symposium on Transport Phenomena.* Prague.

Zhu, S., Deng, K., & Liu, S. (2015). Modeling and extrapolating mass flow characteristics of a radial turbocharger turbine. *Energy, 87*, 628-637.