The Study of Converging-Diverging Nozzle for Improving the Impulse Momentum of Cross Flow Turbine in a Bio-Micro Power Plant

H Pujowidodo[1,a), A I Siswantara[2,b), G G R Gunadi[3], A Daryus[4]

[1]*Center for Thermodynamics, Engine, and Propulsion, BPPT, Serpong 15314, Indonesia*
[2]*Department of Mechanical Engineering, University of Indonesia, Depok 16424, Indonesia*
[3]*Mechanical Engineering Department, State Polytechnic of Jakarta, Depok 16424 Indonesia*
[4]*Mechanical Engineering Department, University of Darma Persada, Jakarta, Indonesia*
[a)]Corresponding author: hariyotejo.pujowidodo@bppt.go.id
[b)]Corresponding author: a_indra@eng.ui.ac.id

Abstract. Nozzle is the one important component in a power cycle plant that transfers heat enthalpy to kinetic energy for rotating power turbine. The biomass has specific characteristics heating value comparing to conventional fuel, and it is one of the renewable energy. The flow characteristics flow trough cross area nozzle plays important role that absorbs maximum drop enthalpy and momentum flux to drive turbine blade. The study of converging-diverging steam nozzle design was conducted using CFD modelling for improving a micro power bio-energy cross-flow turbine model. The objective of this work was to analyze the improvement of momentum flux by simulating a converging-diverging nozzle. A mathematical modelling of compressible flow using EES® tools was developed as well, to calculate the suitable dimension of inlet, throat, and outlet as computational domain. The flow characteristic parameters such as distribution of pressure, temperature, and velocity were compared analytically to find the good approximation of momentum flux for turbine demand. For pressure ratio 0.5 and temperature 200 ^{o}C of steam fluid, the maximum velocity of 1.3 Ma and mass flow 0.978 kg/s (3.52 ton/hour) were occurred. Flux could be increased by making larger cross sectional area.

1. Introduction

Nozzle is the one important component in a power cycle plant that transfers heat enthalpy to kinetic energy for rotating power turbine. Consequently the velocity will increase as the compressible fluid flows through the reduced sectional cross area. This magnitude corresponds to the square root of enthalpy difference. To increase the velocity magnitude for momentum source in rotating power turbine, a compressible fluid e.g. steam could reach the supersonic flow through the convergent-divergent nozzle (CD-nozzle). A throat section, between inlet and outlet sections, controls the outlet Mach when velocity in sonic condition.

Rotating power turbine converts the thermal energy (enthalpy) to kinetic energy, as the gradual flow momentum occurs along the runner blades (rotor). Momentum flux flows on the blade surfaces will determine the shaft mechanical power. Momentum variables are mass and velocity would be the important variables to obtain the suitable required turbine power design. Accordingly, the nozzle design plays important role to give the optimum momentum flux. Nozzle geometry and operating parameters are the main factors in convergent-divergent nozzle design, which affect the flow characteristics and properties.

In thermodynamics principles subject to enthalpy conversion in nozzle device, there is critical pressure ratio to analyze the supersonic velocity, which refers to ratio of actual pressure to stagnation inlet pressure and the sonic velocity in throat section. The velocity calculated analytically by the correlation of specific heat ratio for specific working fluids (compressible), based on the adiabatic irreversible process.

Many studies on convergent-divergent nozzles utilized the computational method to investigate the flow characteristic and properties. Geometry and operating parameters are the main concern to analyze hydrodynamics of internal flow and optimize the momentum flux. It was also been reviewed that the potential head conversion converted in nozzle was the one of main design parameters of high efficiency turbines [1]. In application of rocket, is found that the proper geometrical design effecting the maximum effective velocity from enthalpy drop of combustion gas [2]. The study of mesh discretization for rocket nozzle models analyzed using the contour features of pressure, velocity, Mach number, cell Reynolds Number and Cell Equiangle Skew [3]. The geometry shape of cross sectional area in convergent-divergent nozzle has been analyzed and been concluded that fluid properties like velocity, pressure, and temperature had been the flow cross sectional dependency and the extent of expansion condition [4]. Two dimensional axi-symmetric conical nozzle models with varying divergence degree of angle were studied to get optimum geometry design for Mach 3 [5] Investigated the back pressure and area ratio for supersonic converging-diverging also reported to obtain the optimum Mach nozzle [6]. The particular geometry and inlet condition of converging-diverging nozzle in steam jet ejector investigation done using CFD analysis [7].

The objective of this study was to analyze the converging-diverging nozzle for cross flow power turbine in a bio micro power plant using thermodynamic analytical model and computational model. Momentum flux and flow properties were the main parameters to obtain the suitable requirement design.

2. Materials and Methods

2.1 Thermodynamics Model Analysis

A conceptual design of bio micro power plant utilizing solid fuels consist of steam generation, nozzle, cross flow power turbine as described in Figure 1. Steam flows through converging-diverging nozzle (CD Nozzle) will increase to supersonic velocity to rotate the cross flow power turbine (CF turbine). The condensate will be re-circulated by pump to be heated in a circulating fluidized bed boiler (CFB Boiler). The mechanical power turbine depends on the quantity of momentum flux exhausted from nozzle. Nozzle converts the hot pressurized gas thermal energy (steam enthalpy) to kinetic energy, thus pressure and temperature are the main thermodynamic parameters that have to be considered and calculated. Momentum variable like mass flow plays an important consideration and affected by nozzle geometry design (cross sectional area).

IOP Conf. Series: Earth and Environmental Science 209 (2018) 012054 doi:10.1088/1755-1315/209/1/012054

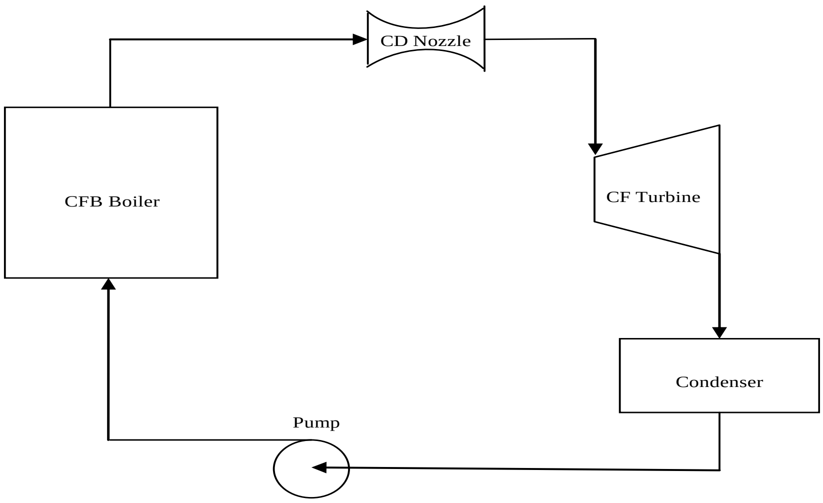


Figure 1. Schematic Diagram (conceptual design) of Bio-Micro Power Plant

As the pressure decreases the velocity will increase and the density will change to reduce. The analysis of flow properties refers to gas ideal equation state that the pressure has linearly correlation to density and temperature. The supersonic condition at outlet depends on the back pressure and throat size to reach the sonic velocity which effected by specific heat ratio of working fluid (k=1.3 for steam).

$$P = \rho.R.T \tag{1}$$

where : P is gas pressure, = density , R is universal gas constant, and T is temperature.

The velocity of ideal gas or acoustic velocity is independent of pressure, and it could be stated that the sound velocity of gas depends on the temperature.

$$c = \sqrt{k.R.T} \tag{2}$$

So the Mach number is able be calculated using the next formula, that is the ratio of gas velocity to sound velocity.

$$M = \frac{v}{c} = \frac{v}{\sqrt{k.R.T}} \tag{3}$$

Gas velocity is obtained from the mechanical energy equilibrium of energy conservation where negligible the gravitational potential energy,

$$v = \sqrt{2000.(h_i - h_0)} \tag{4}$$

and h_o is outlet enthalpy at exit side and h_i is inlet enthalpy (all units in kJ/kg) thus velocity in m/s.

Meanwhile for flow properties of sonic characteristic determined by specific heat ratio (k) of fluid and every parameters divided by stagnation properties where velocity is zero (rest state), and usually taken at inlet or throat section for a M at exit/outlet. The critical pressure ratio (CPR) defined as the division of operating pressure at any section to stagnation point. This value would be a basic reference to analyze sonic condition at throat and outlet, compared to actual pressure ratio (APR). When the value of APR lower than CPR, the CD nozzle would be in sonic condition and supersonic flow.

$$CPR = \left[\frac{2}{k+1}\right]^{k/k-1} \tag{5}$$

$$APR = \frac{P_e}{P_0} \tag{6}$$

where P_e is exit pressure, and P_0 is stagnation pressure at inlet.

And then the operating pressure at any cross sectional area (P) could be found using the following formula.

$$P = APR.P_0 \tag{7}$$

For operating temperature at any section also could be calculated using the ideal gas correlation and isentropic process ($P.v^k$ = constant), and thus the formula would be as stated below.

$$T = T_0 \cdot \left(\frac{P}{P_0}\right)^{k-1/k} \tag{8}$$

The density is inversely proportional with specific volume (v) and therefore be analyzed as well by substituting it.

$$\rho = \rho_0 \cdot \left(\frac{P}{P_0}\right)^{1/k} \quad \text{or} \quad \rho = \rho_0 \cdot \left(\frac{T}{T_0}\right)^{1/(k-1)} \tag{9}$$

2.2 CFD Model Analysis

Computational modelling was begun from geometry and grid/mesh definitions, where geometry model built 2 dimensionally from the dimension obtained in the previous analysis based on the constraint data (flow properties). The available nozzle used in bio-micro power plant has diameter of 2 mm at throat and 2.4 mm at exit side. The boundary conditions determined at inlet and outlet in domain computation using operating parameters such as pressure and temperature inlet steam, and outlet temperature.

The output features from the post-processing like contour or vector were choosen as the analysis parameters e.g. pressure, temperature, velocity, Ma number. As the additional is the flux analysis like mass flow or momentum to verify the result. Simulation model was performed using the tools of CFDPRE© and CFDSOF©.

3. Results and Discussions

Thermodynamics calculation conducted by the EES™ tools was performed to find the flow properties along cross sectional area for a Mach number. When the operating values are inlet pressure of 2 bar. G and inlet temperature 200 ^{o}C, the range of Mach was from 0 up to 3 the properties ratio shown in Figure 2.

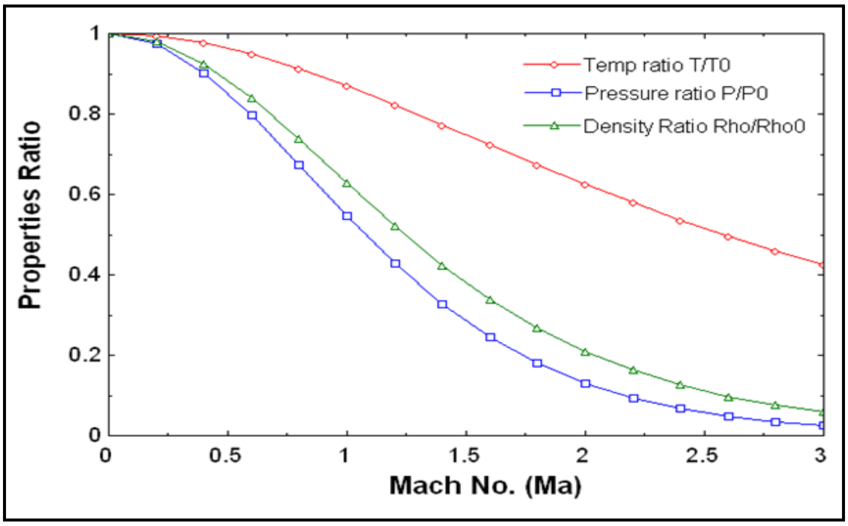


Figure 2. Flow Properties change according to Mach number for steam CD Nozzle (solved by EES™)

Figure 2 shows that pressure and density properties almost have the similar shape, therefore when velocity is more supersonic, its properties decrease more. The Ma number was very strongly affected by the pressure ratio between back pressure and inlet pressure. These trends would be verified using CFD model to know the characteristics of available CD-Nozzle outlet diam. 2.4 mm and throat diam. 2 mm.

The CD-Nozzle model geometry has the dimension of diameter at exit position and throat position, as mentioned in the previous section. This model and its discretization (38x74 cells) shown in Figure 3.

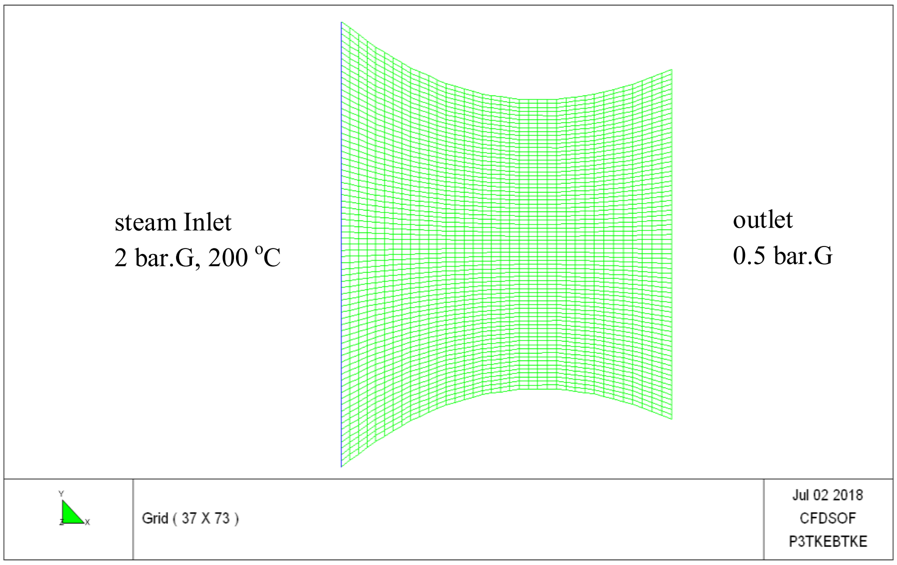
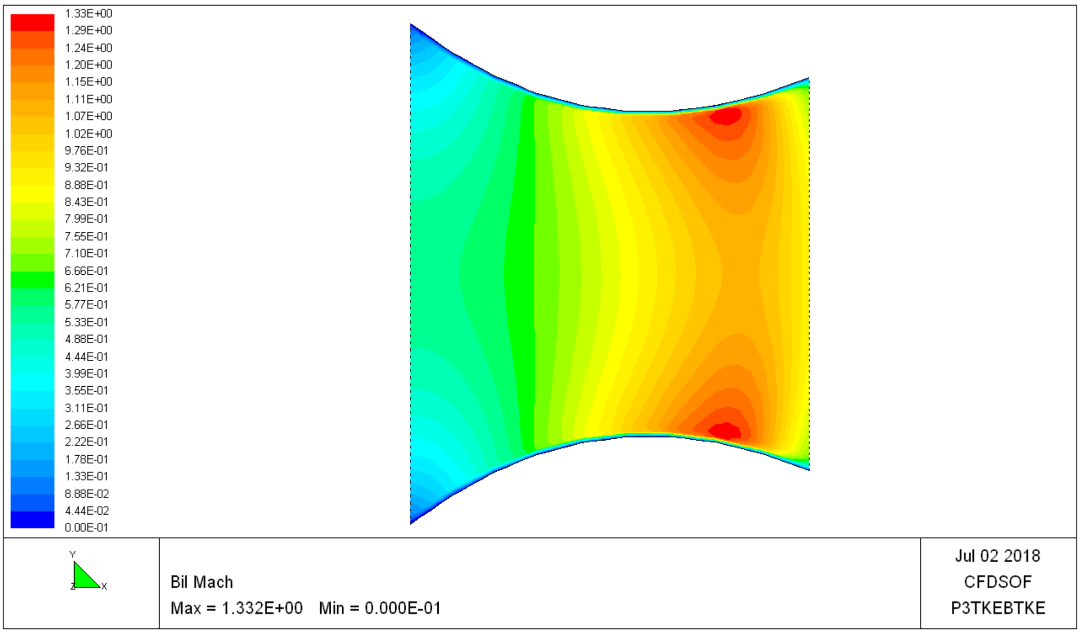


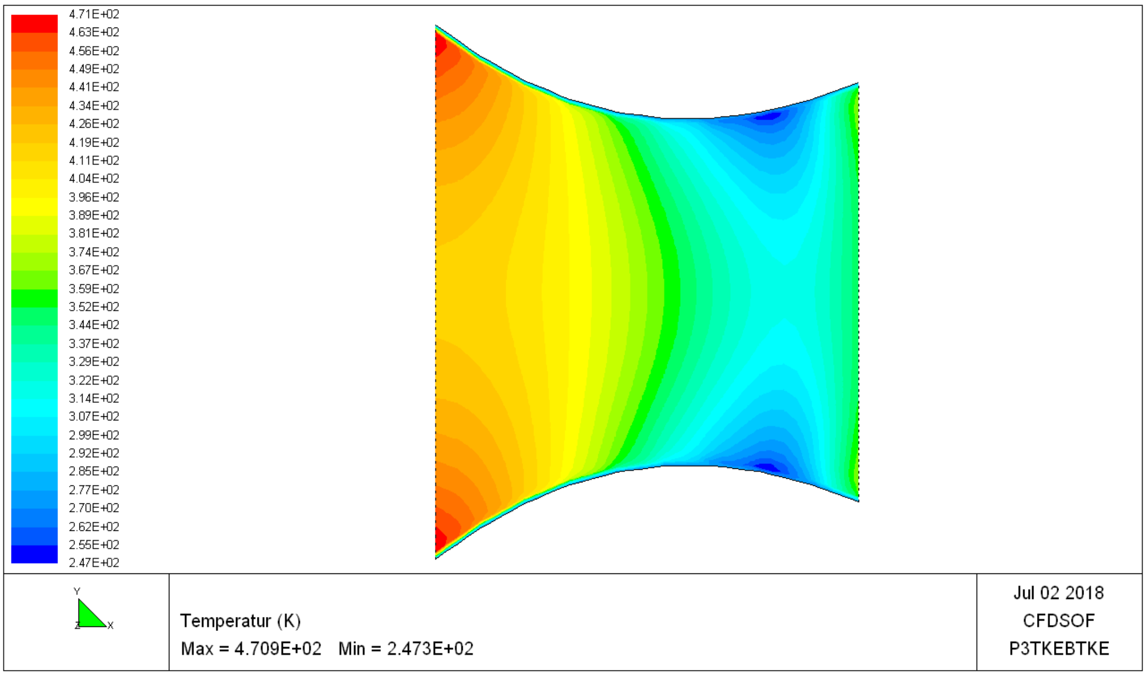
Figure 3. Grid size of 2 dimensional steam CD-Nozzle (solved by CFDPRE©).

The contour of dynamics flow was analyzed using k-ε turbulence model to know the distribution of pressure, velocity, and density for given constraints. The velocity was more supersonic which gradually changes to be faster flowing out from nozzle. The complete results of simulation are given in Figure 4.

IOP Conf. Series: Earth and Environmental Science 209 (2018) 012054 doi:10.1088/1755-1315/209/1/012054

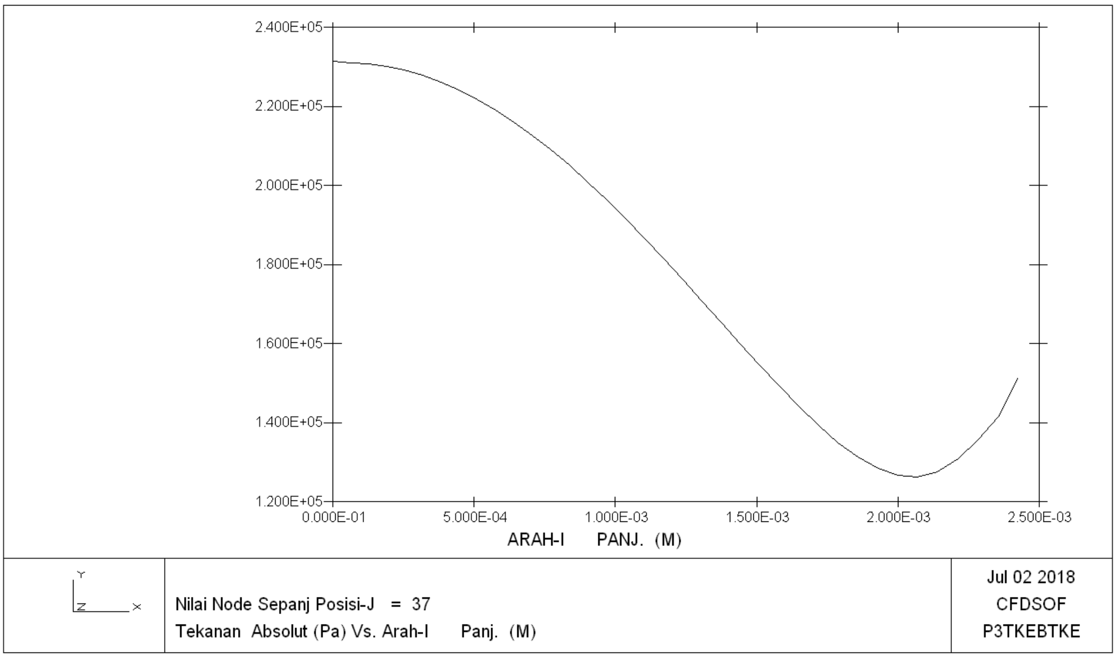


(a)

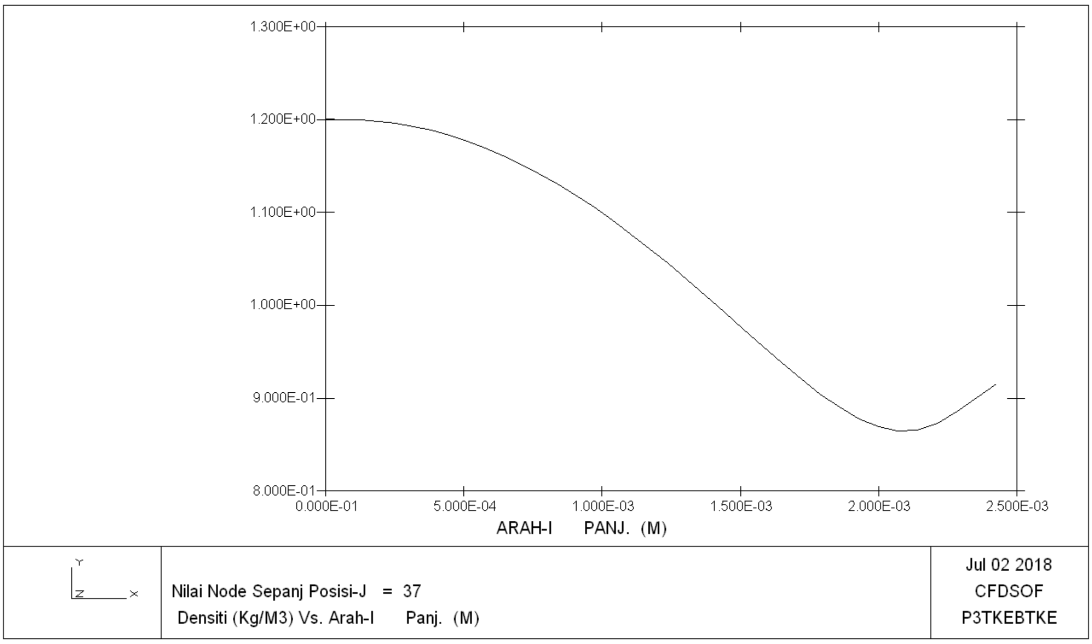


(b)

Figure 4. Contours of Ma Number (a) and Temperature (b) for steam CD-Nozzle (solved by CFDSOF©).



(a)



(b)

Figure 5. Distribution of Static Pressure and Density along centreline on CD-Nozzle (solved by CFDSOF©)

Figure 4 and 5 show that along the flows pass through geometry profile, the velocity increase corresponds to the pressure and density decrease. When pass trough throat section, pressure decrease and will increase at outlet. This flow properties change patterns are in the same with the study had been investigated by measurement validation [6]. This profile change showed that the velocity factor of momentum flux determined from the choosing on the operating condition such as pressure at outlet and inlet side.

At the outlet side using the integration from post-menu could be calculated that the total mass flow magnitude is 0.978 kg/s. This the integration all values located in every cell in computational domain.

This magnitude depends on the cross sectional area, when larger sectional area, the mass flow will increase thus the momentum flux will be higher. The cross sectional area of exit side will determine the amount of steam mass flow delivered to turbine's blade.

4. Conclusions

The analyze of momentum flux for rotating power turbine using integration thermodynamics and CFD models has shown that the smallest pressure ratio will increase the supersonic flow pass through the CD-nozzle with throat diameter 2 mm and outlet diameter 2.4 mm. For pressure ratio 0.5 and temperature 200 ^{o}C of steam fluid, the maximum velocity that occurred was 1.3 Ma and mass flow was 0.978 kg/s (3.52 ton/hour). Flux could be increased by making larger cross sectional area.

5. References

[1] Adhikari R and Wood D 2018 11(2):267. PubMed PMID: doi:10.3390/en11020267.
[2] Belega BA and Nguyen TD editors 2015 Proceedings of International Conference of Scientific Paper AFASES Brasov, Romania.
[3] Narayana KPSS and Reddy KS 2016 *Simulation of Convergent Divergent Rocket Nozzle using CFD Analysis* 58-65 p.
[4] Satyanarayana G et.al 2013 Acta Technica Corviniensis-Bulletin of Engineering 6(3):139.
[5] Pandey K and Singh A 2010 International Journal of Chemical Engineering and Applications. ;1(2):179.
[6] Shariatzadeh OJ et.al 2015 J Clean Energy Technol ;3(3):220-5.
[7] Singhal A Chitkara T K and Ameenuddin M 2013 National Convention of Aerospace Engineers (27th NCAE), November 08-09, Dehradun, India

6. Acknowledgements

The authors would like to thanks, DRPM Universitas Indonesia for funding this research through "Hibah Publikasi Internasional Terindeks untuk Tugas Akhir Mahasiswa UI 2017" and to PT. CCIT Group Indonesia for CFD Package and EES software license.