

PAPER • OPEN ACCESS

## Design and performance analysis of 1KW ORC turboexpander with R245fa as working fluid

To cite this article: Asybel Bonar *et al* 2021 *IOP Conf. Ser.: Mater. Sci. Eng.* **1109** 012049

View the [article online](#) for updates and enhancements.



The banner features a decorative top border with a red, white, and blue striped pattern. On the left, the ECS logo is displayed in a blue circle, followed by the text 'The Electrochemical Society' and the tagline 'Advancing solid state & electrochemical science & technology'. To the right of this text is a logo for the 18th International Meeting of the Solid State Ionics Society (IMCS18). The main text of the banner reads '239th ECS Meeting with IMCS18' in a large, bold, dark blue font, followed by 'DIGITAL MEETING • May 30-June 3, 2021' and 'Live events daily • Free to register' in a smaller, dark blue font. On the right side, there is a red button with the text 'Register now!' in white. The background of the banner is a collage of images, including a person's face, a laptop, and abstract digital patterns.

**ECS** The Electrochemical Society  
Advancing solid state & electrochemical science & technology

**239th ECS Meeting with IMCS18**  
DIGITAL MEETING • May 30-June 3, 2021  
Live events daily • Free to register

**Register now!**

# Design and performance analysis of 1KW ORC turboexpander with R245fa as working fluid

Asybel Bonar\*, Ari Darmawan Pasek, Harmen Burhannudin

Faculty of Mechanical and Aerospace Engineering, Institut Teknologi Bandung, 40132, Bandung, Indonesia

Email: asybel.sidabutar@gmail.com

**Abstract.** Organic Rankine Cycle (ORC) is a reliable technology to utilize low and medium thermal energy and has been in development since the beginning of the 19th century and has rapid development in the last few decades. There are a couple of ways to optimize ORC. One of them is to develop an optimized turboexpander. This paper will explain in detail the preliminary and detailed design approach used to design turboexpander for 1kW ORC with R245fa as working fluid. The numerical simulation was performed using ANSYS CFX with turbulence model of Eddy Viscosity Reynold Shear Stress and Aungier Redlich-Kwong Equation of state to predict the material properties. A couple of adjustments had been made when deciding the rotor thickness and blade number, as the proposed method was not able to produce reasonable geometry. The CFD result of the preliminary design has 4.79% less efficiency and 22.7% less power produced than the OD design. The off-design parameters of 16 Blade number, 20000 rpm, and mass flow of 0.32 kg/s able to produce power 1.38 kW with the efficiency of 76.44%

**Keywords:** Organic Rankine cycle, Radial inflow Turbine, CFD Analysis

## 1. Introduction

Organic Rankine Cycle (ORC) is an effective technology to utilize low and medium thermal energy sources such as geothermal, waste heat recovery, and solar power into electrical power since it has high reliability and simple configuration [1]. ORC has been in development since the early 19th century, and research has been growing rapidly over the last decade. A total of 2701 MW over 1754 ORC unit has been installed worldwide as of 31st December 2016, with 74,8% of the unit were installed for power generation from geothermal brines [2].

A microscale ORC has great potential to fulfill building energy needs or to improve engine efficiency by utilizing its heat waste using Combined Heat and Power system. A way to optimizing ORC technology is to develop the design of ORC turboexpander. Li et al. [3] studied the blade number and blade angle effect on radial inflow turbine performance. Most of the studies of Radial turbine design are for larger power generation (more than 50kWe). A micro-scale ORC with power generation of less than 10kWe has not shown its applicability at the commercial level. When building microscale ORCs, researchers mostly focus on utilizing scroll expanders, which converted from a compressor. However, the potential is limited in its efficiency (usually less than 60%) and several issues such as leakage, friction, and heat transfer losses. A radial turbine has larger efficiency compared to the scroll expander. However, designing a micro radial turbine for ORC applications is limited since most of the equations present made for larger size turbines. This study will use a previously developed OD design method for

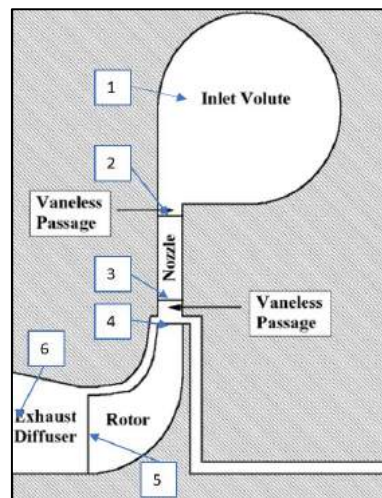


radial ORC expanders made for 200kW ORC [5] and analyze the geometry made using ANSYS CFX tools.

## 2. Radial Turbine Design

### 2.1. Preliminary design

The basic geometry of the inflow radial turbine was shown in figure 1. The component is divided into several sections. The blade rotor of an inflow radial turbine is schematically shown in Figure 1, divided into several radial and axial sections. Station 1 indicates the inlet volute, station 2 indicates the inlet to the nozzle, station 3 indicates the nozzle's outlet, stations 4 and 5 indicate the inlet and outlet of the blades, and station 6 indicates the outlet of the exhaust diffuser.



**Figure 1.** Basic radial turbine configuration [6]

The design process was accomplished based on the predefined parameters, consist of inlet and outlet total temperature and pressure and turbine rotational speed. The relation of specific speed and rotational speed were shown in equation 1. The parameters used to design in this research are presented in table 1.

**Table 1.** Input Data of 1kW turboexpander

Parameter	Unit	Value
Fluid	-	R-245fa
Inlet pressure	KPa	730
Inlet temperature	K	368,87
Outlet pressure	KPa	560
Mass flow rate	kg/s	0.3
Power output target	kW	1
Rotational speed	rpm	20000
Efficiency	%	83.3
Expected Power	kW	1.292

ORC turbine required high-speed generator, especially in small geometry turbine. However, due to the limitation of available bearing system in the market, the turbine was designed to work around 20000 rpm. The results of preliminary design parameter are shown in table 2.

**Table 2.** 0D Design results

Parameter	Symbol	Value	Unit
Rotational Speed	N	20.000	rpm
Mass flow	$\dot{m}$	0,3	kg/s
Rotor discharge flow rate	$Q_5$	0,01	$m^3/s$
Rotor Outlet Pressure	$\Delta H_{id}$	5,17	$kJ/kg$
Specific Speed	$n_s$	0,35	-
Velocity Ratio $U_4/C_{0s}$	$v_s$	0,599	-
Spouting velocity	$C_{0s}$	101,69	m/s
Inlet Tangential Speed	$U_4$	60,90	m/s
Rotor inlet radius	$r_4$	29,08	mm
Rotor inlet pressure	$P_{t4}$	722	kPa
Inlet tangential absolute velocity	$C_{\theta 4}$	70,70	m/s
Absolute inlet velocity angle	$\alpha_4$	77,42	°
Inlet meridional absolute velocity	$C_{m_4}$	15,78	m/s
Relative inlet velocity angle	$\beta_4$		°
Inlet passage width	$b_4$	3,25	mm
Rotor axial length	$\Delta Z_R$	10,47	mm
Rotor outlet radius	$r_5$	15,12	mm
Outlet passage width	$b_5$	6,98	mm
Outlet meridional absolute velocity	$C_{m_5}$	16,04	m/s
Relative outlet velocity angle	$\beta_5$	26,86	°

## 2.2. Detailed Rotor Design

The detailed rotor design was carried out using 2D and 3D model construction [6], [7]. The 2D model construction was carried out by drawing shroud contour using polynomial curve equation below [6], and hub contour by making a quarter circle with a radius of  $\Delta Zr$  with a center that located parallel to outlet station and adding straight line segment to the inlet. The quasi-normal line was constructed by splitting the curve of shroud and hub into the same size. These actions could be done easily with CAD software.

$$r = r_{s5} + (r_4 - r_{s5})\xi^n; 2 \leq n \leq 9 \quad (1)$$

$$\xi = \frac{(z - z_5)}{\Delta Z_R - b_4} \quad (2)$$

The first step to constructing a 3D model was by creating a meridional contour of the blade. The meridional of the rotor hub and shroud could be constructed from the following equations:

$$x_{s,i} = r_{s,i} \sin \theta_{s,i} \quad (3)$$

$$y_{s,i} = r_{s,i} \cos \theta_{s,i} \quad (4)$$

$$x_{h,i} = r_{h,i} \sin \theta_{h,i} \quad (5)$$

$$y_{h,i} = r_{h,i} \cos \theta_{h,i} \quad (6)$$

The thickness distribution along the blade was carried out using space vector, mentioned in the equation below, and the coordinates of blade surface were calculated using equation 39.

$$\begin{bmatrix} \vec{T}_x \\ \vec{T}_y \\ \vec{T}_z \end{bmatrix} = \vec{S} \times \vec{B} = \begin{vmatrix} i & j & k \\ \sin \theta_{j,i} \sin \phi_{j,i} \sin \beta_{j,i} & \cos \theta_{j,i} \sin \phi_{j,i} \sin \beta_{j,i} & \sin \phi_{j,i} \sin \beta_{j,i} \\ + \cos \theta_{j,i} \cos \beta_{j,i} & - \sin \theta_{j,i} \cos \beta_{j,i} & \\ \frac{x_{s,i} - x_{h,i}}{L} & \frac{y_{s,i} - y_{h,i}}{L} & \frac{z_{s,i} - z_{h,i}}{L} \end{vmatrix} \quad (7)$$

$$x_{j,i}^{\pm} = x_{j,i} \pm \frac{1}{2} t_b T_{x,j,i} \quad (8)$$

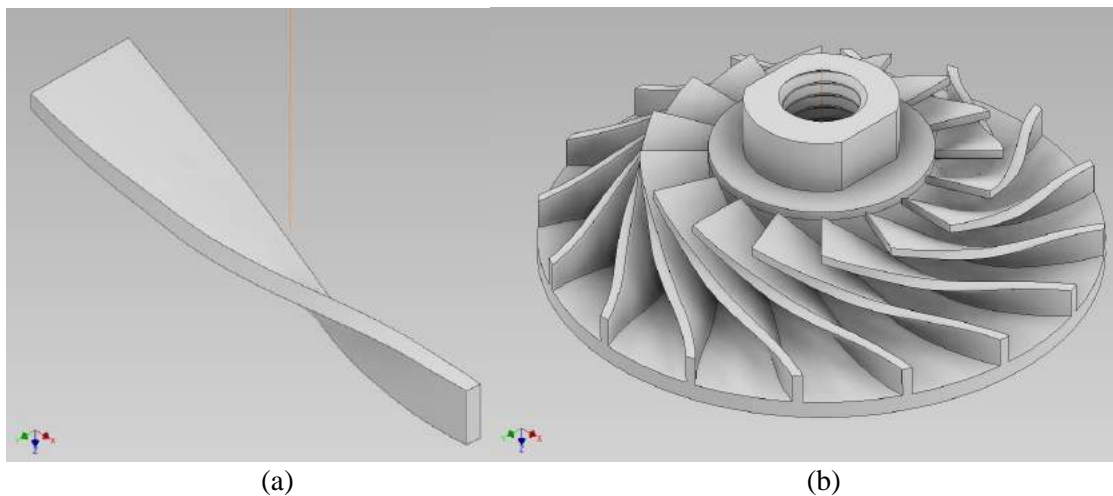
$$y_{j,i}^{\pm} = y_{j,i} \pm \frac{1}{2} t_b T_{y,j,i} \quad (9)$$

$$z_{j,i}^{\pm} = z_{j,i} \pm \frac{1}{2} t_b T_{z,j,i} \quad (10)$$

In this research, the thickness of the blade was considered similar from leading until trailing edge. This action was taken because the thickness of the blade was calculated using equations proposed by Aungier [6], resulting in a very small dimension size. These happened because the equation was generally made for a larger sized turbine, with air as its working fluid [5]. The blade number was calculated using the equation proposed by Glassman [8]; however, a couple of research noted that the optimum blade number, especially in micro radial turbine usually different from equations as these equations also generally made for larger size turbine. Thus, the turbine off-design blade number was analyzed in CFD sections.

**Table 3.** Rotor design result

Parameter	Unit	Value
Rotor inlet radius	mm	29,08
Rotor blade thickness	mm	1
Rotor blade number	-	16
Rotor outlet passage width	mm	6,98
Rotor inlet passage width	mm	3,25



**Figure 2.** 3D construction of rotor (a)Single Rotor blade (b) Final rotor geometry

### 2.3 Nozzle Design

The nozzle design input parameters were  $b_4$ ,  $\alpha_4$ ,  $r_4$ ,  $C_{\theta 4}$ ,  $\rho_4$ ,  $Nr$ , and  $\dot{m}$ . The design was started by building its meanline contour then constructing the top and bottom contour using thickness distribution along the meanline. The meanline contour and thickness were constructed using the equation below:

$$y_c = \frac{x_c(c - x_c)}{\left[ \frac{(c - 2a)^2}{4b^2} + \frac{c - 2a}{b} x_c - \frac{c^2 - 4ac}{4b} \right]} \quad (11)$$

$$\frac{t_2}{c} = 0.03 \quad (12)$$

$$\frac{t_3}{c} = 0.015 \quad (13)$$

$$\frac{t_{max}}{c} = 0.06 \quad (14)$$

$$\frac{d}{c} = 0.4 \quad (15)$$

Then, the coordinate for top and bottom contour were constructed using:

$$x = x_c \pm 0.5t \sin \mathcal{K} \quad (16)$$

$$y = y_c \pm 0.5t \sin \mathcal{K} \quad (17)$$

Where

$$\mathcal{K} = \tan^{-1} \left( \frac{\partial y_c}{\partial x_c} \right) \quad (18)$$

**Table 4.** Nozzle design result

Parameter	Unit	Value
Nozzle inlet radius	mm	36,59
Nozzle outlet radius	mm	30,49
Nozzle maximum thickness	mm	1,56
Nozzle passage width	mm	3,25
Nozzle blade number	-	21
Nozzle chord length	mm	25,97

### 2.4 Volute Design

The volute type chosen in this research was the external volute type. The input parameters for volute design were  $c_{m2}$  and  $c_{\theta 2}$ . The distribution of volute cross section area was assumed linear starting from largest at position angle of  $360^\circ$  and smallest at  $0^\circ$ . These considerations were taken to simplify the design and manufacture process.

**Table 5.** Volute design result

Position angle (°)	A <sub>c</sub> (mm <sup>2</sup> )	r <sub>max</sub> (mm)	B (mm)	A (mm)
0	0,00	38,59	0,00	0,00
30	28,42	40,08	0,74	0,89
60	56,84	41,57	1,49	1,79
90	85,25	43,06	2,23	2,68
120	113,67	44,55	2,98	3,57
150	142,09	46,04	3,72	4,47
180	170,51	47,53	4,47	5,36
210	198,92	49,02	5,21	6,25
240	227,34	50,50	5,96	7,15
270	255,76	51,99	6,70	8,04
300	284,18	53,48	7,44	8,93
330	312,59	54,97	8,19	9,83
360	341,01	56,46	8,93	10,72

#### 2.4. Diffuser Design

The diffuser were designed based on method proposed by Aungier [6], with a couple of design value were recommended by Merandy et.al [9]. to give optimal diffuser size. Those value were  $2\theta_c = 11^\circ$ ,  $AR = 2.55$ ,  $L/b_6 = 8$ . The geometry of the diffuser could be calculated with equations 68 and 69.

$$2\theta_c = 2 \tan^{-1} \left[ \frac{(AR - 1)b_6}{2L} \right] \quad (19)$$

$$AR = \frac{b_7}{b_6} \quad (20)$$

**Table 6.** Diffuser design result

Parameter	Unit	Value
Inlet diameter	mm	37,22
Outlet diameter	mm	94,91
Length	mm	297,76

### 3. Numerical Analysis

Three-dimensional modeling of rotor, nozzle, and volute are completed using Autodesk Inventor. The grid generated using ANSYS design modeler mesh. The numerical study was carried out from volute to rotor. ANSYS-CFX solver was chosen to solve the three-dimensional fluid problem.

The steady-state analysis was conducted using SST K- $\omega$  turbulence model. The model boundary conditions were set for pressure and temperature inlet and outlet based on thermodynamic calculations with mixing plane interface set at rotor-nozzle interface. The material thermodynamic properties were calculated using Aungier Redlich Kwong EoS with input properties taken from NIST REFPROP 9.1 [10]. The fourth-order polynomial of specific heat capacity at zero pressure was shown in equation 70. The analysis was conducted at several blade numbers and rotational speed.

$$\frac{C_p}{R} = 5.327 + 1.317 \times 10^{-2}T + 1.223 \times 10^{-4}T^2 - 3.132 \times 10^{-7}T^3 + 2.585 \times 10^{-10}T^4 \quad (21)$$

**Table 7.** Parameter used to calculate R245fa Properties

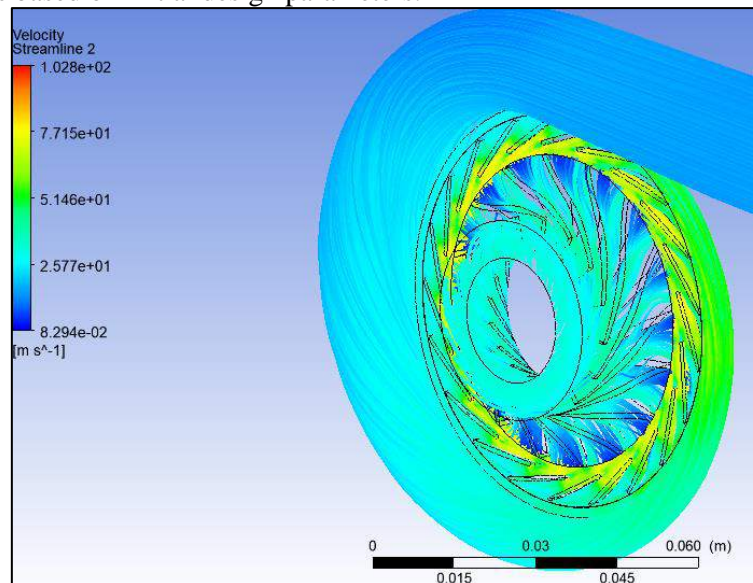
Parameter	Nilai	Satuan
Real gas model	Aungier Redlich Kwong	-
Molar mass	134,05	g/mol
Critical temperature	427,16	K
Critical Pressure	3651	kPa
Critical volute	$0,260 \times 10^{-3}$	$m^3/mol$
Acentric factor	0,378	-
Boiling temperature	288,29	K

#### 4. Simulation Results

The purpose of simulations was to evaluate the purposed method of geometry design method to generate a high efficiency turbine with producible geometry.

##### 4.1. Initial Design Results

The initial design analysis was done in condition mentioned in table 1. The following results resulted from CFD analysis based on initial design parameters.

**Figure 3.** Initial design velocity streamline

1. The attempted parameter's power output was 0.998 kW, while the expected power produced was 1.292 kW, 22.7 % lower than the expected power. The working fluid mass flow usage for analysis results was lower than the preliminary design caused by a decreasing fluid passage area due to blade thickness modification, affecting the fluid's guidance by the blade.
2. Increasing velocity was observed in volute with small area passage, while an ideal volute design would not have this phenomenon and could affect the overall turbine efficiency.

The maximum Mach number from the design was observed at 0.69. Thus, the shockwave phenomenon did not occur in this analysis.

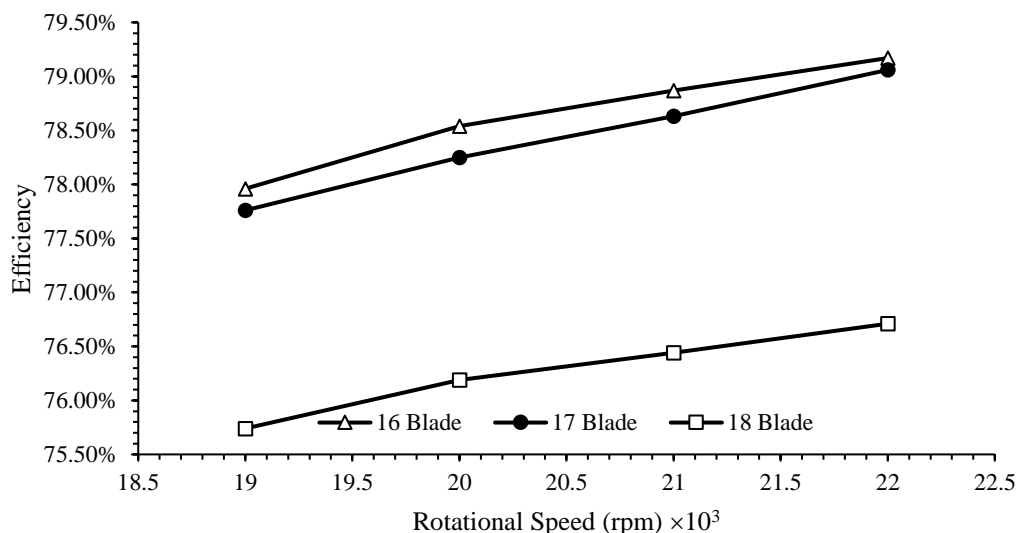


**Table 8.** Preliminary design results

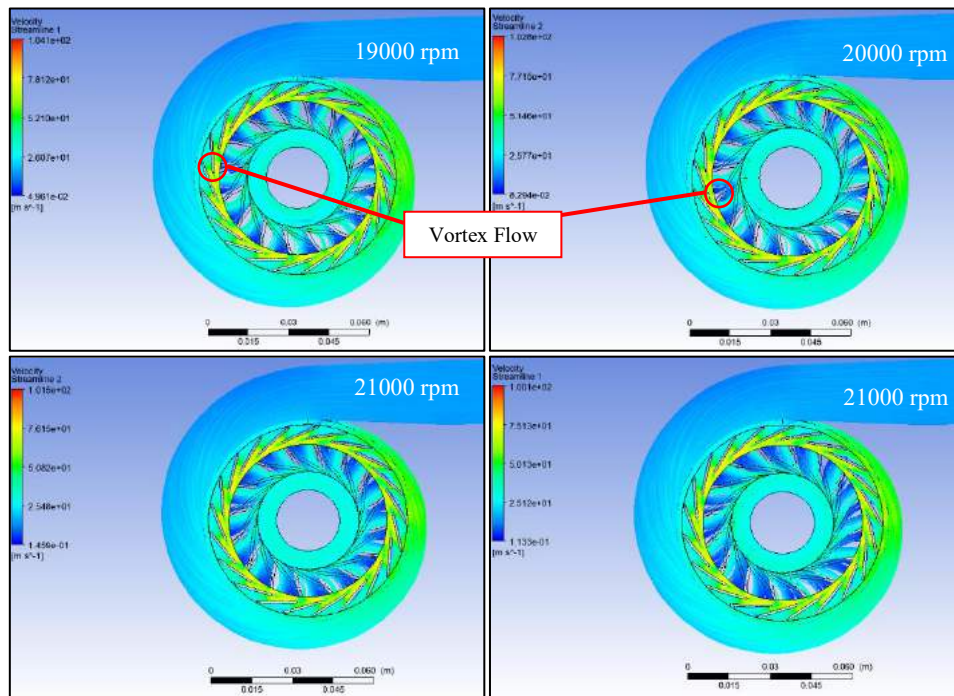
Rotational Speed (rpm)	Massflow (kg/s)	Efficiency	Total Power (W)	Torque (N.m)	Maximum Mach Number
20000	0,2761	78,54%	998,54	0,4768	0,6896

#### 4.2. Off Design Parameters

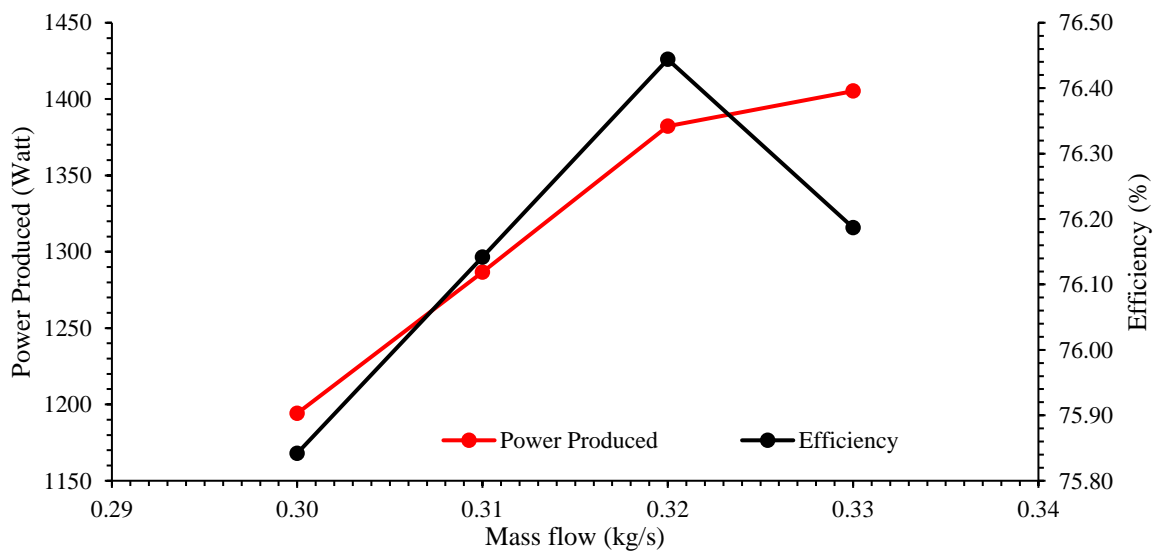
Due to unsatisfaction results of initial design parameters, some modifications of input parameters were considered. Rotor blade number plays a vital role in fluid guidance by the blade. This research use equation proposed by Glassman [8] to determine the rotor blade number. Several equations were also tested to calculate blade number, but they resulted in an excessive number of blades, resulting in unproducible rotor geometry. These results happened because equations to determine rotor geometry usually made for larger sized turbine [5]. Thus, the off-design blade number analysis was evaluated from 16 to 18 number of blades. Another essential aspect is the working fluid mass flow rate. Thus, the off-design mass flow rate parameter was also analyzed. The design was evaluated at the mass flow of working fluid at 0.3 to 0.33 kg/s at 20000 rpm.

**Figure 4.** Rotational speed vs efficiency of different blade number

The highest efficiency was achieved at 16 blade configurations, while off design parameters at 17 and 18 had lower efficiency than the initial design. The turbine tends to have more vortex flow around the nozzle outlet at lower rpm, shown in figure 10. Based on figure 4, the turbine produced higher efficiency at higher rpm, similar to every blade number configuration. The increasing velocity across the volute small passage area is also noticed in every blade and rpm configuration. Maximum Mach number occurred at 16 blade passage, at Mach 0.698.



**Figure 5.** Velocity streamline with different rotational speed



**Figure 6.** Power Produced vs Mass flow variation of working fluid

## 5. Conclusion

In this paper, the CFD analysis of the ORC radial inflow turbine for 1kW power production was carried out. The system was simulated using ANSYS CFX with fluid properties based on REFPROP 9.1 and calculated using Aungier Redlich Kwong EoS. From this study, the following conclusions were drawn.

- The preliminary design's efficiency drop was 4.79% from the CFD result to the initial design result, while the power produced is 22.7% lower. Thus, a couple of parameter adjustments are needed so that the turbine produces the expected power.

- Based on geometry results, the proposed design method could build a micro radial turbine with some adjustment when calculating rotor blade thickness and blade number, as the proposed method from the previous study was not capable of producing a reasonable micro turbine geometry.
- Based on the CFD simulation, the proposed method of geometry design could build a radial turbine for purposed power of 1.38 kW with an efficiency of 76,44% at off-design parameters. The parameters are 16 blade numbers, with rotational speed of 20000 rpm, and mass flow rate of 0.32 kg/s. The 20000 rpm parameters were taken even though having lower efficiency than higher rpm because of the limited available component such as bearing and generator in the market for desirable design. So, the turbine was designed as such to work on 20000 rpm.

## References

- [1] Bao J and Zhao L 2013 A review of working fluid and expander selections for organic rankine cycle *Renew. Sustain. Energy Rev* **24** pp 325–342
- [2] Tartière T and Astolfi M 2017 A world overview of the organic rankine cycle market *Energy Procedia*
- [3] Li P, Han Z, Jia X, Mei Z, and Han X 2018 Analysis of the effects of blade installation angle and blade number on radial-inflow turbine stator flow performance *Energies*
- [4] Han S, Seo J B, and Cho B S 2014 Development of a 200 kW ORC radial turbine for waste heat recovery *J. Mech. Sci. Technol.* **28** 12 pp 5231–5241
- [5] Fiaschi D, Innocenti G, Manfrida G, and Maraschiello F 2015 Design of micro radial turboexpanders for ORC power cycles: From 0D to 3D *Appl. Therm. Eng.* **99** pp 402–410
- [6] Aungier R H 2006 *Turbine aerodynamics: axial-flow and radial-inflow turbine design and analysis* (New York: ASME Press)
- [7] Putri R P 2019 Studi perancangan turbin radial aliran masuk 250 kw pada siklus rankine organik dengan fluida kerja propana (Institut Teknologi Bandung)
- [8] Glassman A J et al. 1994 *Turbine Design and Application* (Nasa Spec. Publ.)
- [9] Pala'biran M 2020 *Pengembangan Lapangan Gas Raksasa East Natuna Dengan Metode Carbon Capture Storage (CCS) dan Memanfaatkan Rancangan Turboexpander Fluida Kerja Superkritis Untuk Meningkatkan Keekonomisan Lapangan* (Institut Teknologi Bandung)
- [10] Lemmon E W, Huber M L, and McLinden M O 2019 *NIST Standard Reference Database 23: Reference Fluid Thermodynamic and Transport Properties-REFPROP, Version 9.1*