A GEOMETRIC DESIGN METHOD OF RADIAL INFLOW TURBINE FROM 0D TO 3D FOR ORGANIC RANKINE CYCLE MICRO POWER GENERATION

Ari Darmawan Paseka, Prihadi Prasetyo, Asybel Bonara, Maulana Arifin

aFaculty of Mechanical and Aerospace Engineering, Institut Teknologi Bandung, Jl. Ganesha no.10, Bandung (40132), Indonesia
bResearch Centre for Electrical Power and Mechatronics, Indonesian Institute of Sciences – LIPI, Indonesia

Abstract

Radial turbine is an essential component of Organic Rankine Cycle system and requires a medium to high specific speed turbine. Radial turbine has a compact structure that can easily be made with current additive manufacturing technology if the 3D geometry of turbine components is known. Current researches only conduct 2D geometry design then import it into third-party software to construct the 3D geometry. This paper will explain design methodology to design radial inflow turbines from 0D until 3D using simple tools. The methods used to determine the geometry were based on Aungier, with modification in determining value of a, b, and c in nozzle design and A1 in Volute design to simplify the design process. The tools used in design were MS Excel and Autodesk Inventor. Rotor design starts with determining the two-dimensional parameters. All parameters are calculated based on the angle and velocities occurring in the velocity triangle at the inlet and outlet of the rotor using equations proposed by Aungier. Then, the straight, radial and quasi-normal lines of the blades are drawn based on governing equations. The transformation from 2D to 3D blade coordinates is done by using vector equations. The nozzle is designed by drawing the camber line profile and calculating the nozzle thickness to get the profile based on the governing equations given by Aungier. The volute dimensions are obtained by calculating the area of volute inlet passage and mean radius from mass and momentum conservation equations. A case study is shown in this paper with R134a as working fluid with the following range inlet conditions: mass flow rate at 1–2 kg/s, inlet pressure at 1.5 to 5 bar, inlet temperature at 80 to 130 °C, and power output target between 20 to 25 kW. The CFD results show that the designed turbine performs well with slight wake flow at the pressure side on the rotor inlet. A further study needs to be done in order to check the validity of this method by conducting analysis through experimental.

Keywords: Radial Inflow Turbine, Organic Rankine Cycle, Power Generations, Design Method, CFD Simulation

© 2022 Penerbit UTM Press. All rights reserved

1.0 INTRODUCTION

New and renewable energy contribute significantly to the national energy supply in developed countries [1] as well as in developing countries[2]. The Organic Rankine Cycle (ORC) system is suitable for utilizing low temperature heat [3] for power generation, such as heat from geofluid, waste heat, solar heat and biomass combustion [4]. The system uses organic fluids such as halocarbons and hydrocarbons as working fluid [5]. ORC can supply additional power even with low thermal efficiency [6]. System efficiency also depends on turbine geometry [7]. Hence, it is important to develop a design method to determine the ORC turbine geometry. Several authors have studied methods to design micro radial turbines. The overview of some turbines builds for ORC shown in Table 1.
Table 1 Review of works in radial turbine design for ORC for micro power generation application [6,8–11]

<table>
<thead>
<tr>
<th>Author</th>
<th>Output (kW)</th>
<th>Efficiency (%)</th>
<th>Working Fluid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alshammari et al.</td>
<td>20-25</td>
<td>30-35.2</td>
<td>R-134a</td>
</tr>
<tr>
<td>Fiaschi, et al.</td>
<td>5</td>
<td>69</td>
<td>R-134a</td>
</tr>
<tr>
<td>Kang, S.H.</td>
<td>190</td>
<td>84</td>
<td>R-245fa</td>
</tr>
<tr>
<td>Jubori, A.M, et al.</td>
<td>13.6</td>
<td>79.05</td>
<td>n-Pentane</td>
</tr>
<tr>
<td>Costall, A.W, et al.</td>
<td>45.6</td>
<td>56.1</td>
<td>Toluene</td>
</tr>
<tr>
<td>Costall, A.W, et al.</td>
<td>24</td>
<td>79</td>
<td>R-245fa</td>
</tr>
</tbody>
</table>

The design method was first introduced in [12]. The equations used in the design were developed by Aungier [13]. A general method for designing radial turbines has been introduced in [14], while a preliminary design has been proposed in [15]. However, both papers only discuss the design of 2D parameters. Two other papers [15,16] reported a numerical simulation to show the influence of important geometric parameters on turbine performance and in [17] a thermodynamic consideration on the radial inflow turbine design is discussed.

This paper will clearly describe the method to design rotor, nozzle, and volute geometry for radial inflow turbines. The equations were introduced by Aungier, arranged in a more systematic order with some modification. The modifications were when designing preliminary parameters, where the results were evaluated at 5 conditions chosen from several resources. Calculation values of a, b, and c when designing nozzle and A1 when designing Volute were taken iteratively. Also, the coordinate of meridional lines was determined with the help of 3D Drawing software to simplify the process. The design process of radial turbine was shown in figure 1.

2.0 BLADE ROTOR GEOMETRY

The blade rotor of an inflow radial turbine is schematically shown in Figure 2, divided into several radial and axial sections. Station 1 indicates the inlet volute station, station 2 indicates the inlet to the nozzle, station 3 indicates the outlet of the nozzle, stations 4 and 5 indicate the inlet and outlet of the blades, and station 6 indicates the outlet of the exhaust diffuser. The design started by selecting its operating condition: total temperature at inlet volute, $T_{t1}$; the inlet total pressure $P_{t1}$; the fluid mass flow rate, $\dot{m}$; and the pressure ratio, $P_{t1}/P_{t5}$, followed by selecting a working fluid. Then, rotational speed $n_s$ can be calculated with:

$$n_s = \frac{\omega \sqrt{Q_5}}{(\Delta H_{id})^{0.75}}$$

(1)

where $\omega$ is the angular velocity in rad/s, which can be derived from rotational speed $n_s$. $Q_5$ is the volumetric rate in m$^3$/s, which can be calculated from the mass flow rate $\dot{m}$, with density is taken at station 5 ($\rho_5$). $\Delta H_{id}$ is the isentropic enthalpy difference of the working fluid. Then, $v_s$ can be calculated from [17]

$$v_s = \frac{U_4}{C_{0s}} = 0.737n_s^{0.2}$$

(2)

The jet velocity $C_{0s}$ and the tangential velocity at rotor inlet, $U_4$, can be calculated from [17] as shown in equation (3)

$$C_{0s} = \sqrt{2\Delta H_{id}} U_4 = v_s C_{0s}$$

(3)
The total-static efficiency was calculated by

\[ \eta_s = 0.87 - 1.07(n_s - 0.55)^2 - 0.5(n_s - 0.55)^3 \]  

Equation (4)

Figure 2 shows the \( U_4 \) and other velocities in the inlet and outlet of the blade rotor, and Figure 3 (b) shows some rotor dimensions that must be determined. Referring to these figures, the inlet rotor radius \( r_4 \) is calculated from

\[ r_4 = \frac{U_4}{\omega} \]  

Equation (5)

Meanwhile, the inlet rotor passage is determined by using the following equations [17]:

\[ b_4 = \frac{m}{2\pi r_4 \rho_4 C_{m4}} \]  

Equation (6)

\[ C_{m4} = C_{\theta4} \tan \tan \alpha_4 \]  

Equation (7)

\[ C_{\theta4} = \frac{U_4 \eta_s}{2\nu_s^2} \]  

Equation (8)

\[ \alpha_4 = 10.8 + 14.2 n_s^2 \]  

Equation (9)

The density at the rotor inlet, \( \rho_4 \), is determined by iteratively using the following equation [17]:

\[ h_4 = H_4 - \frac{1}{2} \left( C_{m4}^2 + C_{\theta4}^2 \right) P_4 = P_{t4} - \frac{1}{2} \rho_4 C_4^2 C_4 \]  

Equation (10)

\[ = P_{t4} - \rho_4 \Delta H_{ad}(1 - \eta_s) \]  

It is assumed that theoretically the total enthalpy at the rotor inlet (\( H_4 \)) is equal to the total enthalpy at the inlet volute (\( H_1 \)). Since the velocity at the inlet volute (station 1) is small compared to that in the other stations, it can be assumed that the total pressure in that station (\( P_1 \)) is equal to the static pressure (\( P_t \)). The equation of state that relates the density (\( \rho_4 \)), static enthalpy (\( h_4 \)), and static pressure (\( P_4 \)) at the inlet of the blade rotor is needed. For this purpose, Computer-Aided Thermodynamics Tables 3 (CATT3) are used [20,21]. The iteration is carried out until the \( r_4 \) and \( h_4 \) reach their convergent values.

Once the inlet rotor radius \( r_4 \) is known, the inlet and outlet blade thickness and hub radius can be obtained from these following equation [17]:

\[ t_{b4} = 0.04r_4 \]  

\[ t_{b5} = 0.02r_4 \]  

\[ r_{h5} = 0.185r_4 \]  

Equation (11)

The outlet shroud radius \( r_{s5} \) is calculated from and the rotor axial length (\( \Delta Z_R \)) is calculated from

\[ \frac{r_{s5}}{r_4} \leq 0.78 \]  

Equation (12)

\[ \Delta Z_R = 1.5(r_{s5} - r_{h5}) \]  

Equation (13)

The number of blades and the inlet blade angle can be calculated from

\[ N_R = \frac{\pi(110 - \alpha_4) \tan \alpha_4}{30} \]  

Equation (14)

\[ \beta_4 = \left( \frac{C_{\theta4} - U_4}{C_{m4}} \right) \]  

Equation (15)

Rotor outlet radius \( r_5 \) and outlet passage width \( b_5 \) are obtained from following equations [17]

\[ r_5 = r_{s5} + r_{h5} \]  

Equation (16)

\[ b_5 = r_{s5} - r_{h5} \]  

Equation (17)

The pitch distance between each blade at rotor outlet \( s_5 \) is obtained from following equation:

\[ s_5 = \frac{2\pi r_5}{N_R} \]  

Equation (18)

If the Mach number at the rotor outlet is less than 1, then the throat width, \( o_5 \), is calculated from

\[ o_5 = \frac{s_5 C_{m5}}{W_5} \]  

Equation (19)

\[ C_{m5} = \frac{m}{2\pi r_5 \rho_5 C_{m5}} \]  

Equation (20)

The density of fluid \( r_5 \) is calculated iteratively using Computer-Aided Thermodynamics Tables 3 (CATT3) [22, 23] and the enthalpy equation

\[ h_5 = H_5 - \frac{1}{2} C_{m5} \]  

Equation (21)

Iterative calculation is done until \( C_{m5} \) reaches its convergent value. Then the relative velocity at the rotor outlet, \( W_5 \), is calculated from
Some dimensions should be checked so that the dimensions are within practical limits. The rotor axial length should be in the range \[0.2 \leq \frac{C_m5}{u_4} \leq 0.4\] [17]. It is also important to keep the \(\frac{C_m5}{C_m4}\) between 0 and 1 to avoid choking.

The ratio of rotor outlet meridional velocity to rotor inlet absolute velocity and ratio of outlet to inlet meridional velocity are given by [22,23]:

\[
0.2 \leq \frac{C_m5}{u_4} \leq 0.4 \quad 1 \leq \frac{C_m5}{C_m4} \leq 1.5
\] (25)

The shroud outline is given by \[\Delta Z_R \geq 1.5b_4\] (24)

The ratios of rotor outlet meridional velocity to rotor inlet absolute and ratio of outlet to inlet meridional velocity are given by [22,23]:

\[
0.45 \leq R \leq 0.65
\] (26)

Where
\[
R = \frac{h_4 - h_5}{H_1 - H_5}
\] (27)

The shroud outline is drawn using the following equation:

\[
r = r_{55} + (r_4 - r_{55})\xi^n, 2 \leq n \leq 9
\] (28)

\[
\xi = \frac{(z - z_b)}{\Delta Z_R - b_4}
\] (29)

Where \(z\) is a step size value between 0 and \(\Delta Z_R - b_4\). The step size is determined arbitrarily. The hub outline is made by making a quarter circle with radius \(r_4 - r_5\) and its center located on a point that is parallel to the outlet station and adding a straight-line segment to the exit or creating a quarter circle with radius \(\Delta Z_R\) and its center located on a point that is parallel to the outlet station and adding a straight-line segment to the inlet. After the \(r\) and \(z\) values were determined, they were imported to Autodesk Inventor (AI) to generate the shroud and hub outline coordinates for the further design process and drawing creating 2D blade geometry. The illustration of this process both by two available methods was shown by Figure 4.

2D blade geometry was transformed into 3D by dividing streamlines into equal parts. The numbers of streamlines and quasi normal lines are taken arbitrarily. The intersection between the quasi-normal lines with the shroud, hub and streamlines create the coordinate points of the blade. For each point, the twist angle or polar angle \((\phi_j,i)\), the blade angle \((\beta_j,i)\) and the tangential angle of the lines to the axial direction \((\phi_j,i)\) are then calculated. Referring to Figure 5, \(i = 1,2,3,\ldots,n\) and \(j = h, a, b, c, s\).

Streamline was constructed graphically but can also be made by GAMBIT [21]. The quasi-normal line was constructed by selecting several \(z\) values in the shroud, after which the \(r\) ordinates were taken from the Autodesk Inventor (AI) drawing. The polar angles for each point along the shroud line \((\theta_{s,i})\) were calculated from [17]:

\[
\theta_{s,i}(m_{s,i}) = A m_{s,i} + B m_{s,i}^2 + C m_{s,i}^3 + D m_{s,i}^4
\] (30)

\[
dm_{s,i} = \sqrt{(dz_{s,i})^2 + (dr_{s,i})^2}
\] (31)

\[
A = \frac{1}{m_4} \left[ \cot \beta_{s5} \right]
\] (32)

\[
B = \frac{1}{r_4} \left[ \cot \beta_{s4} - \cot \beta_{s5} \right]
\] (33)

\[
C = -\frac{B}{2m_4}
\] (34)

\[
r_{55} \tan \beta_{s5} = r_5 \tan \beta_{s5} = r_{5h} \tan \beta_{s5}
\] (35)

Where \(i\) and \(m_{h,i}\) are calculated similarly as \(m_{s,i}\)

\[
D = \frac{\cot \beta_{s4}}{m_4}
\] (36)

The blade angle \((\beta_{j,i})\) and the tangential angle \((\phi_{j,i})\) of each point along the shroud, hub and every intersection point of the streamlines and the quasi-normal lines can be determined from

\[
cot \cot \beta_{j,i} = \frac{r_{j,i}}{dm_{j,i}} \sin \sin \phi_{j,i} = \frac{dr_{j,i}}{dm_{j,i}}
\] (37)

Referring to Figure 5, the coordinates of the points in the shroud and hub lines can be calculated from the following equations:
\[ \tau_{ji} = \frac{\phi_{j+1,i} + \phi_{j-1,i}}{2} \]  
(37)

\[ x_{s,i} = r_{xi} \sin \sin \theta_{s,i} \quad y_{s,i} = r_{xi} \cos \cos \theta_{s,i} \]  
(38)

\[ x_{h,i} = r_{hi} \sin \sin \theta_{h,i} \quad y_{h,i} = r_{hi} \cos \cos \theta_{h,i} \]  
(39)

The coordinates of the streamline and quasi normal line intersections can be obtained from:

\[ x_{h,i} = x_{s,i} + (x_{s,i} - x_{h,i})\varphi_{j,i} \quad y_{h,i} = y_{s,i} + (y_{s,i} - y_{h,i})\varphi_{j,i} \]  
(40)

\( \varphi_{j,i} \) is the fraction length of the quasi normal line from the intersection point to the hub. \( \varphi_{j,i} = 0 \) and \( \varphi_{j,i} = 1 \) indicates the intersection point at the hub line shroud respectively. The ordinates \( r \) and \( z \) of the points are obtained from the AI drawing. Figure 6 shows the blade parameters and coordinates.

![Figure 6](image)

Figure 6 (a) Sketch of quasi normal line position on 2D meridional component (b) 3D Rotor meridional geometry component

The points discussed above are then transformed into three-dimensional coordinates by using vector operation. Three kinds of vectors are defined, i.e. [17]: the vector tangential to the blade profile along the meridional direction (\( \vec{S}_{ji} \)), the vector along a quasi-normal line (\( \vec{B}_{ji} \)), and the vector that is perpendicular to the blade profile (\( \vec{T}_{ji} \)) and defined as equation below:

\[ \vec{S}_{ji} = S_{x,j,i} + S_{y,j,i} \hat{x} + S_{z,j,i} \hat{z} \]  
(41)

\[ \vec{B}_{ji} = B_{x,j,i} \hat{x} + B_{y,j,i} \hat{y} + B_{z,j,i} \hat{z} \]  
(42)

\[ \vec{T}_{ji} = T_{x,j,i} \hat{x} + T_{y,j,i} \hat{y} + T_{z,j,i} \hat{z} \]  
(43)

The vectors are calculated for each intersection point in the shroud, hub and other intersections of the streamlines and the quasi-normal lines. The components of vector \( \vec{S}_{ji} \) can be calculated from equations below:

\[ S_{x,j,i} = \sin \sin \theta_{j,i} \sin \sin \phi_{j,i} \sin \sin \beta_{j,i} + \cos \cos \theta_{j,i} \cos \cos \beta_{j,i} \]  
(44)

\[ S_{y,j,i} = \sin \sin \phi_{j,i} \sin \sin \beta_{j,i} - \sin \sin \theta_{j,i} \cos \cos \beta_{j,i} \]  
(45)

\[ S_{z,j,i} = \sin \sin \beta_{j,i} \]  
(46)

The component vector \( \vec{B}_{ji} \) for the points in the shroud and the hub are calculated from:

\[ B_{x,j,i} = B_{x,h,i} = \frac{x_{s,i} - x_{h,i}}{L} B_{x,s,i} \]  
(47)

\[ B_{y,j,i} = \frac{y_{j,i} - y_{h,i}}{L} B_{y,s,i} \]  
(48)

\[ B_{z,j,i} = \frac{z_{j,i} - z_{h,i}}{L} B_{z,s,i} \]  
(49)

Then, the component vector \( \vec{B}_{ji} \) and \( \vec{T}_{ji} \) for the other intersection points are calculated from:

\[ B_{x,j,i} = \sin \sin \theta_{j,i} \quad B_{y,j,i} = \cos \cos \theta_{j,i} \quad B_{z,j,i} = 0 \]  
(50)

\[ T_{x,j,i} = S_{x,j,i} B_{x,j,i} - S_{y,j,i} B_{y,j,i} \]  
(51)

\[ T_{y,j,i} = S_{x,j,i} B_{y,j,i} + S_{y,j,i} B_{x,j,i} \]  
(52)

\[ T_{z,j,i} = S_{x,j,i} B_{z,j,i} - S_{y,j,i} B_{z,j,i} \]  
(53)

The coordinates of points on the blade \( x_{j,i}, y_{j,i}, z_{j,i} \) are the middle points. Vector transformation above will form a cross section of the blade at quasi normal lines by connecting the points calculated by the Equations below with straight lines. Figure 7 shows the blade transformation from 2D to 3D geometry:

\[ x_{j,i} \pm y_{j,i} \pm z_{j,i} = \pm \frac{1}{2} t_{b5} [T_{x,j,i} \quad T_{y,j,i} \quad T_{z,j,i}] \]  
(54)

Where \( t_{b5} \) is calculated from Equations (12) and changed linearly from \( t_{b4} \) to \( t_{b5} \). The coordinates of the points that were calculated from Equations (48) were plotted in AI. The results are shown in Figure 7. Using the Loft Surface-Patch Menu in AI and taking the Edge option, the 3D blade image as shown in Figure 7(b, a) was created. With selection of Stitch Surface to Solid and Revolve Menu in AI, the image shown in Figure 7(b, b) was generated. Then using the Circular Pattern Menu, resulted in blade rotor as shown in Figure 7(b, c).

![Figure 7](image)

Figure 7 (a) Transformation to 3D coordinates (b) 3D Rotor Figures Results

### 3.0 NOZZLE GEOMETRY

The nozzle geometry parameters are shown in Figures 8 (a) and 8(b). The input parameters of the calculation are \( b_n, a_n, r_a, C_o, p_o, N_o \) and the mass flow rate is \( m \). The aforementioned parameters are obtained from Equations (6), (7), (9) to (12) and (20) above.

![Figure 8](image)

Figure 8 (a) Nozzle blade profile (b) Parameters of guide nozzle
There is a gap between the blade rotor and the stator nozzle called the vaneless passage. The relation between \( r_3 \) and \( r_4 \) is expressed as [17]

\[
\frac{r_3}{r_4} = 1 + \frac{K_1 c_3 \sin \alpha_4}{r_4}
\]  

(48)

\( K_1 = 2 \) for optimum performance [24]. Then, the blade outlet angle \( \alpha_3 \) can be calculated from

\[
\alpha_3 = \frac{C_{m3}}{C_{b3}}
\]  

(49)

\[
C_{m3} = \frac{\dot{m}}{2\pi r_3^3 \sin \alpha_4}
\]  

(50)

\[
C_{b3} = \frac{C_{a4} r_4}{r_3}
\]  

(51)

The outlet pitch of the nozzle \( s_3 \) and number nozzle \( N_N \) and selection range of \( r_2 \) is calculated from [17].

\[
s_3 = \frac{2\pi r_3}{N_N}
\]  

(52)

\[
1.1 \leq \frac{r_2}{r_3} \leq 1.7
\]  

(53)

To develop the nozzle blade profile, the camber line is used. The camber line is drawn using the following equation [17]:

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

(54)

Referring to Figure 8, \( a \) and \( b \) are designated for location of maximum camber, and \( c \) is the nozzle blade chord length. The \( a, b, \) and \( c \) are determined iteratively using the following equations:

\[
0.25 \leq \frac{a}{c} \leq 0.75
\]  

(55)

\[
b = \frac{1}{4} \tan \tan \eta
\]  

(56)

where the camber angle of nozzle \( \eta \) is calculated:

\[
\eta = X_2 + X_3 \tan \tan X_2
\]  

(57)

\[
X_2 = \frac{4a - c}{3c - 4b}
\]  

\[
X_3 = \tan \tan X_3
\]  

blade angle at the inlet \( \alpha_1 \) and the fluid inlet angle \( \beta_2 \) are determined iteratively from:

\[
\frac{4a_3 \sin \beta_2 - a_3}{c \sin \beta_2 \left[ 1 + 4 \frac{\sin \alpha_1}{r_2 \sin \alpha_2} \right]} \leq 1
\]  

(58)

\[
\beta_2 = \gamma_2 - X_2
\]  

(59)

\[
r_2 \cos \gamma_2 = r_3 \cos \gamma_3
\]  

(60)

where \( g \) is the nozzle setting angle and the following equation for the length of the nozzle chord \( c \) is also used:

\[
c = \frac{r_2 - r_3}{\sin \gamma_3}
\]  

(61)

Once the camber line has been constructed, the nozzle profile line coordinates can be drawn using the following equations:

\[
x = x_c \pm 0.5t \sin K \quad y = y_c \pm 0.5t \sin K
\]  

(62)

\[
t = t_{\text{ref}} + (t_{\text{max}} - t_{\text{ref}}) \xi^e
\]  

(63)

\[
t_{\text{ref}} = t_2 + [t_3 - t_2] \left( \frac{x_c}{d} \right)
\]  

(64)

\[
\xi = \frac{x_c}{d} \text{ for } x \leq d
\]  

(65)

\[
e = \sqrt{0.4d \left( 0.95 \left( 1 - \frac{x_c}{c} \right) (1 - \xi) + 0.05 \right)}
\]  

(66)

\[
K = \frac{\partial y_c}{\partial x_c}
\]  

(67)

To solve some of the equations above, there are several values proposed by Aungier [17], i.e.:

\[
t_2 = 0.03; \quad t_3 = 0.015; \quad t_{\text{max}} = 0.06; \quad \frac{d}{c} = 0.4
\]  

(68)

The upper profile was drawn by plotting the \( x, y \) coordinates while the lower profile was drawn using the \( x', y' \) coordinates. The coordinates were then imported to Al and the nozzle was positioned manually by placing the tip of the nozzle’s leading edge on the outer circle of the nozzle passage and the trailing edge on the innermost circle. Once the coordinates were well placed, the coordinates were connected using the Spline Menu and then Extrude with a value of \( b_z \). The tips of the nozzle were drawn with the Tangent Pattern Menu. The surrounding nozzles were constructed using the Circular Pattern Menu.

### 4.0 Volute Geometry

This study will use external type volute. The volute geometry is shown in Figures 9 (a) and 9(b) while the input parameters to calculate the volute parameters were: the density of the fluid at the volute inlet \( \rho_1 \), the fluid mass flow rate \( \dot{m} \), and the following equations:

\[
C_2 = \frac{c_m2}{\sin \sin \alpha_2}; \quad C_{b2} = \frac{c_m2}{\tan \tan \alpha_2}; \quad C_m2 = \frac{r_3 c_m3}{r_2}
\]  

(69)

Figure 9 (a) Volute geometry (b) External elliptical volute
Then, the aspect ratio $S/V$ is selected between 0.75 and 1.5. Inlet velocity $A_1$, $C_1$, $S$, $r_1$, and $V$ are calculated iteratively from equations below:

$$\rho_1 C_1 A_1 = m \dot{r}_1$$
$$C_1 = \frac{r_2 C_{\theta 1} A_1}{\sqrt{\frac{3\pi}{4} + 1}} S \times V r_1$$
$$= r_2 + V + \text{vaneless passage}$$

Once the volute parameters have been found, the cross-sectional area of the volute is calculated for various angles ($\phi$) along 360°:

$$A_c = \frac{\phi}{2\pi} A_1 V = \frac{A_c}{(\frac{3\pi}{4} + 1)} S r_{max} = r_1 + V$$

Figure 10 shows the formation of the volute. The cross sections are developed every 30°, then the 3D surface is developed, covering the cross-sections. At the end of the process, an extended pipe is attached to the volute.

5.0 DESIGN RESULTS AND CONCLUSION

The radial inflow turbine from the design is shown in this section. A case study is shown in this paper with R134a as working fluid with following conditions: mass flow rate at 1-2 kg/s, inlet pressure at 1.5 to 5 bar, inlet temperature at 80 to 130 °C, and power output target between 20 to 25 kW. The detailed input parameters for designing the turbine are shown in Table 2 with the geometry results for rotor, nozzle and volute are shown in Table 3, Table 4, and Table 5 respectively. Figure 11 show the complete assembly of turbine generated from the design process.

<table>
<thead>
<tr>
<th>Table 2 Input Parameters for Rotor Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
</tr>
<tr>
<td>Fluid</td>
</tr>
<tr>
<td>Inlet pressure</td>
</tr>
<tr>
<td>Inlet temperature</td>
</tr>
<tr>
<td>Mass flow rate</td>
</tr>
<tr>
<td>Power output target</td>
</tr>
<tr>
<td>Rotational speed</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 3 Calculation Results of Turbine Rotor Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
</tr>
<tr>
<td>Specific speed</td>
</tr>
<tr>
<td>Inlet Mach number</td>
</tr>
<tr>
<td>Inlet rotor radius</td>
</tr>
<tr>
<td>Inlet blade thickness</td>
</tr>
<tr>
<td>Inlet passage width</td>
</tr>
<tr>
<td>Outlet blade thickness</td>
</tr>
<tr>
<td>Outlet hub radius</td>
</tr>
<tr>
<td>Outlet shroud radius</td>
</tr>
<tr>
<td>Rotor axial length</td>
</tr>
<tr>
<td>Rotor outlet radius</td>
</tr>
<tr>
<td>Outlet passage width</td>
</tr>
<tr>
<td>Outlet mean blade pitch</td>
</tr>
<tr>
<td>Number of blades</td>
</tr>
<tr>
<td>Output rotor power</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 4 Input Parameters and Calculation Results for Nozzle Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Input Parameters</td>
</tr>
<tr>
<td>Parameter</td>
</tr>
<tr>
<td>Ratio of a/c</td>
</tr>
<tr>
<td>Inlet passage width</td>
</tr>
<tr>
<td>Inlet absolute flow angle</td>
</tr>
<tr>
<td>Inlet rotor radius</td>
</tr>
<tr>
<td>Mass flow rate</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Calculation Results</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
</tr>
<tr>
<td>Location of maximum camber line height</td>
</tr>
<tr>
<td>Maximum height of camber line</td>
</tr>
<tr>
<td>Camber line chord length</td>
</tr>
<tr>
<td>Nozzle inlet tangential angle</td>
</tr>
<tr>
<td>Nozzle outlet tangential angle</td>
</tr>
<tr>
<td>Camber line angle</td>
</tr>
<tr>
<td>Nozzle outlet radius</td>
</tr>
<tr>
<td>Nozzle inlet radius</td>
</tr>
<tr>
<td>Nozzle inlet angle</td>
</tr>
</tbody>
</table>
Table 5  Input Parameters and Calculation Results for Volute Geometry

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flow rate</td>
<td>( \dot{m} )</td>
<td>1.6</td>
<td>kg/s</td>
</tr>
<tr>
<td>Nozzle inlet radius</td>
<td>( r_2 )</td>
<td>0.125</td>
<td>m</td>
</tr>
<tr>
<td>Aspect Ratio</td>
<td>( S/V )</td>
<td>1.1</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Volute inlet passage</td>
<td>( A_1 )</td>
<td>0.003</td>
<td>m²</td>
</tr>
<tr>
<td>Center point radius</td>
<td>( r_1 )</td>
<td>0.154</td>
<td>m</td>
</tr>
<tr>
<td>Volute outlet radius</td>
<td>( r_2 )</td>
<td>0.125</td>
<td>m</td>
</tr>
<tr>
<td>Volute maximum vertical length</td>
<td>( S )</td>
<td>0.032</td>
<td>m</td>
</tr>
<tr>
<td>Volute maximum horizontal length</td>
<td>( V )</td>
<td>0.029</td>
<td>m</td>
</tr>
<tr>
<td>Volute maximum radius</td>
<td>( r_{\text{max}} )</td>
<td>0.154</td>
<td>m</td>
</tr>
</tbody>
</table>

ANSYS CFX was selected to perform the simulations. A turbulence model of \( k-\varepsilon \) was used for the turbulence model. For boundary conditions, the simulation used a total pressure inlet and static pressure outlet. The specific boundary was set into two conditions to improve the validation of this analysis. Condition A with rotor rated speed of 20,000 rpm, pressure inlet at volute inlet at 3.85 bar, an inlet temperature of 100°C, and pressure outlet at rotor outlet at 1.81 bar. Then, condition B was set with mass flow at Volute inlet while keeping the other condition the same as condition A. In this analysis, one dynamic interface was located between nozzle and rotor, defined as Mixing Plane model.

Figure 13 and 14 shows streamline distribution of radial turbine and streamline distribution at midspan of blade. From the figure, it can be seen that the flow to the nozzle is relatively smooth with relatively uniform distribution. The nozzle shows significant velocity changes located near the end of the nozzle blade, peaking at the nozzle exit. Slight reverse flow occurred in rotor entrance around pressure area and will only cause slight flow loss.

Table 6 showing the comparison of design results and CFD results. The result shows that the design has good efficiency and similar condition results compared to theoretical design, thus validating the method used in this paper.
Although the calculation methods discussed above provide a simple way to determine the radial inflow turbine geometry, further study is required to see the sensitivity of the assumed parameters. The coordinates of the turbine geometry can be easily imported into a 3D printing or additive machine for turbine production. To prove the validity of the method, turbine production and assembly engineering has to be done before a real test in an ORC system can be conducted to verify the performance of the turbine in the future.

Acknowledgement

The authors would like to thank Institut Teknologi Bandung for providing financial support for this research.

References