Mean-line modeling and CFD analysis of a miniature radial turbine for distributed power generation systems

Kiyarash Rahbar, Saad Mahmoud and Raya K. Al-Dadah*
School of Mechanical Engineering, University of Birmingham, Birmingham B15-2TT, UK

Abstract
Distributed power generation (DPG) based on organic Rankine cycle offers potential in the effective use of energy from low grade heat sources up to 200°C. In this regard, developing an effective expander plays a major role in determining the overall cycle efficiency. In this work mean-line modeling and CFD techniques are employed to develop a small-scale radial turbine for DPG systems with a power output of ~5 kW. A parametric study is carried out using the mean-line approach to investigate the effects of key input parameters such as operating conditions, velocity ratio, rotational speed and rotor flow angles on the turbine rotor inlet diameter and overall performance. Results from the mean-line approach show that in order to achieve high power output, inlet total temperature, mass flow rate and pressure ratio should be increased. However, for reducing the rotor inlet diameter the velocity ratio should be decreased. CFD technique is then used to assess the flow field and to improve the blade loading by modification of blade angle distribution. CFD is also used to determine the minimum number of rotor blades and the results show that the value suggested by mean-line modeling overestimates this parameter. By using these two approaches a wide range of design configurations are explored and the most effective design is identified to be with specific diameter of 4.83 (rotor inlet diameter of 0.0787 m), specific speed of 0.433 (rotational speed of 55 000 rpm), 10 blades and output power of 4.662 kW.

Keywords: distributed power generation; organic Rankine cycle; radial turbine; mean line; CFD analysis

Received 1 March 2014; revised 2 July 2014; accepted 23 July 2014

1 INTRODUCTION
The accelerated world’s energy consumption has led to scarcity of fuel resources and severe environmental pollutions [1]. These issues require new solutions and alternatives. Distributed power generation (DPG) is a promising solution for supplying energy demands and reducing environmental problems. DPG is an electric power source connected directly to the distribution network or the customer site of the meter with the power ratings as classified in Table 1[2].

Steam-based Rankine cycle operates with steam at high temperatures and pressures which is not suitable for most distributed generation systems due to safety issues and the availability of the high temperature heat source [3–5]. Organic Rankine cycles (ORCs) are becoming one of the most reliable technologies suitable for efficient energy conversion of low-grade heat sources in the temperature range of up to 200°C. ORCs offer a number of advantages such as small size due to high fluid density, low complexity due to low pressure and temperature and low capital, and maintenance cost due to use of non-eroding and non-corrosive working fluids that cause low damage to the elements in the flow path [3–6]. Moreover, in contrast to steam turbines that inadequate insulation could cause rapid temperature changes in turbine shells and resulting in contact and damage to blades, this concern seems unnecessary for ORC expanders due to the low temperature difference between the expanders and the environment [7].

Expanders are the major component of the ORCs. Selection of the appropriate expander type is influential in determining the cycle performance. There are many types of expanders that can be used in ORCs such as scroll expanders, screw expanders, reciprocal piston expanders and turbo expanders [8]. Turbo expanders offer many advantages over other expanders such as compact structure, light weight and high efficiency which make...
them suitable for the system of interest. Most of the studies in the literature are devoted to the ORC’s thermodynamics [9–11] or to the selection of appropriate organic fluids [12–14] and there is little published literature on modelling the turbo-expander for these systems.

This article presents a methodology based on mean-line modeling and computational fluid dynamics to develop a radial turbine for distributed micro-power generation systems based on ORC characteristics. Mean-line modeling is used to investigate the effect of key design parameters such as operating conditions, velocity ratio and rotational speed on overall performance such as power, efficiency and rotor size. The mean-line approach is also used to determine the initial geometry of the expander based on the results of the parametric study. CFD analysis is then used to improve the blade profile by examining the flow field in terms of flow uniformity, formation of secondary flows and blade loading. CFD is also used to determine the minimum number of rotor blades. Using combination of these techniques, it is possible to explore a large design space of many different configurations and select the best geometry that satisfies the constraints of the application.

## 2 DEVELOPMENT METHODOLOGY

The development methodology aims at maximizing the efficiency and minimizing the losses by considering the geometrical and performance constraints. This is achieved by systematic combination of two approaches: the mean-line modeling and CFD analysis. Mean-line modeling determines the overall characteristics and the performance levels of the radial turbine for specific input parameters. The overall characteristics include the rotor inlet and exit velocity triangles, rotor inlet tip radius, inlet blade height and exit hub and tip radii. Mean-line modeling is a highly iterative process since it requires comprehensive studies in terms of varying input parameters such as operating conditions, velocity ratio, rotational speed and flow angles to achieve the desired outcome.

### 2.1 Mean-line modeling

Mean-line modeling is based on a one-dimensional assumption that there is a mean streamline through the stage, such that conditions on the mean streamline are an average of the passage conditions [15]. A code was developed for the mean-line approach based on the principals of radial turbine design outlined in refs [16–18]. Figure 1 shows the flowchart for the code. In this approach it is feasible to explore a wide range of design configurations by performing comprehensive studies in terms of varying input parameters such as operating conditions, efficiency ratio, rotational speed and flow angles.

Figure 2 outlines the flowchart of the algorithm used to perform such parametric study by employing the code developed in the Engineering Equation Solver (EES) platform. This algorithm is based on varying two input parameters at a time while keeping the other inputs as constants. At each run two input parameters are varied simultaneously in a defined range and their effects on the critical performance, such as power, efficiency and on rotor inlet diameter, are plotted. Such plots will be examined to determine the values of these two parameters that give the best performance and these values will be fixed in the next run where two other variables will be changed. This procedure will be repeated for the rest of input parameters until the best design values will be determined. The variation ranges of the input parameters of the mean-line code are shown in Table 2. The ranges for inlet temperature and pressure are selected based on the low-grade heat source similar to refs [3–10]. For the rest of input parameters the values are close to the typical range for radial turbines as discussed in refs [18, 20–23].

### 2.2 Mean-line solving method

The flowchart as shown in Figure 1 has been implemented into a code written in the EES software to make use of its powerful iterative sparse-matrix solving technique and the extensive built-in thermodynamic property functions. Figure 3 presents the enthalpy–entropy diagram of the radial turbine stage detailing the expansion process together with the schematic of rotor blade and corresponding velocity triangles. Equations (1)–(12) describe the most important equations that are employed in the mean-line code using air as the working fluid.

Spouting velocity \( C_s \), rotor inlet wheel velocity \( U_4 \), rotor inlet radius \( r_4 \) and actual enthalpy drop \( \Delta h_{actual} \) are calculated from the following equations:

\[
C_s = \sqrt{2c_p T_{T,1} \left(1 - \left(\frac{1}{P}R_{ts}\right)\right)^{\gamma-1}}
\]

\[ U_4 = C_s \cdot v_{ts} \]

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

\[ \Delta h_{actual} = 0.5 C_s^2 \eta_{stage,ts} \]
where \( c_p \) is the specific heat at constant pressure (J/kg K); PR, the pressure ratio (-); \( \gamma \), the specific heat ratio (-); \( T \), the temperature (K); \( v \), the total to static velocity ratio (-); \( U \), the rotor wheel velocity (m/s); \( r \), radius (m); \( \omega \), the rotational velocity (RPM); \( \Delta h \), the change in enthalpy; \( \eta_o \), the total to static efficiency (-).

The stagnation thermodynamic properties \( (P_{t,4}, T_{t,4}, T_{t,5}) \) and the rotor outlet area \( (A_5) \) are obtained from the following equations (with adiabatic assumption).

\[
P_{t,4} = P_{t,1} - \left( \rho_{t,1} c_p^2 \cdot \frac{1 - \eta_{stage,t}}{8} \right) \tag{5}
\]

\[
T_{t,4} = T_{t,1} \tag{6}
\]
where $A$ is the area ($m^2$) and the subscript 1 refers to the turbine inlet.

Euler turbomachinery is shown in Equation (9). This equation is essential for calculating the velocity triangles and static thermodynamic properties both at inlet and outlet of the rotor blades.

$$\Delta h_{\text{actual}} = U_4 C_{\theta 4} - U_5 C_{\theta 5}$$  \hspace{1cm} (9)

As recommended in refs [19–21] the rotor exit swirl angle ($\alpha_5$) is assumed to be 0 for minimizing the rotor exit kinetic losses and then Equation (9) can be readily solved for $C_{\theta 4}$. Subsequently, the solution for velocity triangles and static thermodynamic properties is achieved by applying the known rotor inlet absolute flow angle and ideal gas laws. However, in some applications it is required to have a positive swirl ($\alpha_5 > 0$) in order to compromise between the passage losses and exit kinetic losses. Also in some cases a negative exit swirl ($\alpha_5 < 0$) is required for increasing the power output. These cases should be treated differently and their solving method is based on the algorithm shown in Figure 4 that is implemented into the code written in EES. Using this technique, velocity triangles and static thermodynamic properties can be determined accurately with any value of $\alpha_5$ (i.e. positive, zero or negative). Power is obtained from the following equation.

$$\text{Power} = \Delta h_{\text{actual}} \times \dot{m}$$  \hspace{1cm} (10)

where $\dot{m}$ is the mass flow rate (kg/s).
As shown in Figure 1, it is essential to update the estimate of turbine performance to more realistic value. This is performed by implementing the rotor losses into the code and calculating them based on the obtained rotor geometry. The losses in the rotor are mainly categorized as passage, incidence, clearance and exit losses which are calculated using the correlations outlined in refs [18, 21, 22]. Then the new estimate of turbine efficiency is obtained based on total specific enthalpy drop using the following Equation.

\[
\eta_{\text{stage,IS}}^{\text{new}} = \frac{\Delta h_{\text{actual}}}{\Delta h_{\text{actual}} + \sum_{\text{Losses}} (\Delta h_{\text{passage}} + \Delta h_{\text{incidence}} + \Delta h_{\text{clearance}} + \Delta h_{\text{exit}})}
\]  

Equation (11)

Glassman empirical correlation [23], Equation (12), is used to determine the minimum number of rotor blades required for calculating the incidence and the passage losses.

\[
Z_{\text{min}} = \frac{\pi}{30} \left(110 - \alpha_4\right) \tan(\alpha_4)
\]

Equation (12)

2.3 CFD analysis

Figure 5 shows the block diagram used for CFD analysis. The design output from the mean-line approach was used as input for CFD analysis using the ANSYS 15 software. Within ANSYS, the initial blade geometry was generated using the inlet tip diameter, inlet blade height, exit tip and hub radii and exit blade angles. Next the grid was generated using structured hexagonal mesh. The selected boundary conditions from the mean-line modeling such as inlet total pressure and temperature, mass flow rate and rotational speed were used to perform the CFD analysis.
ANSYS CFX-15 was used to perform such analysis by solving the full three-dimensional Reynolds-average Navier–Stokes equations with shear stress transport (SST) turbulent flow model. The advantage of using an SST turbulent flow model compared with others is the capability of having automatic near-wall treatment for locating the first node away from the wall. The flow characteristics from the CFD analysis in terms of velocity vectors, secondary flows and blade loading were examined and the blade profile was modified in terms of blade angle distribution (blade profile). This process was repeated until the best geometry was achieved. The minimum number of rotor blades is a critical parameter that determines the blade loading and the rotating inertia of the turbine. Despite a number of empirical correlations such as the one shown in Equation (12), accurate estimation of the minimum number of rotor blades that leads to appropriate blade loading is obtained reliably with CFD analysis. The flow field behaviour, blade loading and the power output were examined for different number of rotor blades to select the appropriate value for this parameter.

3 RESULTS AND DISCUSSION

3.1 Mean-line approach

Following the procedure outlined in Figure 2, the inlet total pressure and total temperature were varied to investigate their effect on power, size and efficiency and the best values were identified and used as constants for the following runs. The same procedure was performed for the rest of input parameters of Table 2 and their results are presented in Figures 6–9. As shown in Figure 6, power, rotor efficiency and rotor diameter were plotted versus inlet total pressures (150–400 kPa) at various inlet total temperatures (333–473 K). It is clear from Figure 6a and c that the effect of the inlet total temperature is
Figure 7. Effect of mass flow rate and pressure ratio on the power, size and efficiency.

Figure 8. Effect of velocity ratio and rotational speed on the power, size and efficiency.
more significant on the power output and the rotor inlet diameter than the inlet total pressure. Figure 6b shows that the effects of both the inlet total temperature and pressure on the rotor total to static efficiency are limited. Figure 7 shows that power and efficiency are increasing as the mass flow rates increase. This is directly related to the relation between the mass flow rate and power as shown in Equation (10) with constant enthalpy drop. Moreover, at higher mass flow rates the temperature drop across the turbine is increasing leading to higher efficiency levels with constant pressure ratio as illustrated in Equation (7). However, the rotor inlet diameter is independent of this variation. It is only for the rotor total to static efficiency and at a lower pressure ratio of 1.5 that the effect of increasing the flow rate is limited. As Figure 7a shows, power is increasing as the pressure ratio increases; however, a higher pressure ratio of 3 will have the adverse effect of lowering the rotor total to static efficiency (Figure 7b) and increasing the rotor diameter (Figure 7c). Figure 8a and b shows the effect of velocity ratio and rotational speed on the power output and rotor total to static efficiency. There is an optimum condition that yields to highest power and efficiency at a velocity ratio of \( \sim 0.7 \). Figure 8c shows the effects of these parameters on the rotor.

Table 3. Rotor mean-line modeling results.

| Parameter                                      | Unit | Value  |
|------------------------------------------------|------|--------|
| Inlet total pressure \([P_t,4]\)                 | kPa  | 193.15 |
| Inlet total temperature \([T_t,4]\)             | K    | 333    |
| Inlet static pressure \([P_s]\)                 | kPa  | 151.29 |
| Inlet static temperature \([T_s]\)              | K    | 310.5  |
| Exit total pressure \([P_t,5]\)                | kPa  | 100    |
| Exit total temperature \([T_t,5]\)             | K    | 286.2  |
| Exit static pressure \([P_s]\)                 | kPa  | 98.39  |
| Exit static temperature \([T_s]\)              | K    | 284.9  |
| Inlet diameter \([d_4]\)                       | m    | 0.0787 |
| Inlet blade height \([b_4]\)                   | m    | 0.00412|
| Inlet area \([A_4]\)                          | \(\text{m}^2\) | 0.001019|
| Exit tip diameter \([d_{5t}]\)                | m    | 0.0512 |
| Exit hub diameter \([d_{5h}]\)                | m    | 0.0236 |
| Exit area \([A_5]\)                           | \(\text{m}^2\) | 0.001620|
| Rotor blade number \([Z]\)                     | –    | 14     |
| Inlet wheel velocity \([U_4]\)                 | m/s  | 226.7  |
| Inlet relative flow angle \([\beta_4]\)       | Degree | 21.17  |
| Inlet absolute velocity \([C_4]\)              | m/s  | 212.4  |
| Inlet relative velocity \([W_4]\)              | m/s  | 61.97  |
| Exit wheel velocity at tip \([U_{5t}]\)       | m/s  | 147.4  |
| Exit relative flow angle at tip \([\beta_{5t}]\)| Degree | 71.34  |
| Exit absolute velocity \([C_5]\)               | m/s  | 51.48  |
| Exit relative velocity at tip \([W_{5t}]\)     | m/s  | 160.3  |
| Actual enthalpy drop \([\Delta h_{\text{actual}}]\) | kJ/kg | 46.98  |
| Power                                          | kW   | 4.703  |
| Total to static efficiency \([\eta_{\text{rotor,ts}}]\) | %    | 81.3   |

Figure 9. Effect of rotor inlet and exit absolute flow angles on the power, size and efficiency.

Figure 9. Effect of rotor inlet and exit absolute flow angles on the power, size and efficiency.
inlet diameter are significant and as the RPM increases, the rotor diameter decreases, while as the velocity ratio increases, the rotor inlet diameter increases. Figure 9a and b show that the effects of varying the rotor inlet absolute flow angle ($\alpha_4$) are more significant than the rotor exit absolute flow angle ($\alpha_4$) while the rotor diameter is completely independent of these variations. The best values for all of the investigated input parameters are shown in Table 2. Table 3 shows the rotor mean-line results obtained by running the code using the selected values of Table 2.

3.2 CFD analysis

The candidate design with the properties shown in Tables 2 and 3 was investigated in greater details using CFD analysis and the results are shown in Figures 10–12. These CFD analyses were performed based on a mesh with resolution of 150,000 nodes. This value was obtained based on experience and mesh independence analyses to compromise between computational time and quality of results as shown in Table 4. The second column is the number of nodes used for CFD analyses. The last two columns are the relative percentage change in the critical outputs as total to static efficiency and power between two consecutive rows. Considering the solving time and the relative change in outputs, row 3 was considered as the appropriate choice for performing the CFD analyses based on that.

Three different blade profiles were investigated with the aim of achieving appropriate blade loading and uniform flow.

Figure 10 shows the geometry of these blades in terms of blade angle versus the normalized meridional distance. Blade profile ‘a’ has a small turning angle in the inlet part of passage and sharp turning around the exit, blade profile ‘b’ has significant turning in the first part of passage and small turning at exit, while blade profile ‘c’ has a modest blade turning angle along the whole length of passage. Figure 11 shows the blade loading and velocity vectors at half span for the three different blade profiles investigated. All three configurations have 14 rotor blades as suggested by Equation (12). As shown in Figure 11, blade profile ‘a’ suffers from flow separation on suction surface in the middle section of passage and is unable to accelerate the flow relative velocity to high levels compared with the other two blade profiles. In contrast, both blade profiles ‘b’ and ‘c’ provide uniform velocity vectors and prevent formation of secondary flows. In addition, blade loading charts of Figure 11 show that both blade profiles ‘b’ and ‘c’ are able to accelerate flow relative velocity to higher values so that the enclosed area by the corresponding pressure and suction lines are larger than blade profile ‘a’ and this leads to higher output power for profiles ‘b’ and ‘c’. Profile ‘b’ was selected as the best blade profile since it provides rapid and more uniform expansion on the pressure surface and higher power output compared with profile ‘c’. Profile ‘b’ was then investigated using lower number of blades ($Z = 8$, $Z = 10$ and $Z = 12$) for preserving the output power. Figure 12 shows the blade loading and velocity vectors at half span for rotor blade numbers 8, 10 and 12. Case ‘$Z = 8$’ suffers from strong secondary flow on the pressure surface near the leading edge and also the flow is separated on the suction surface in early part of the passage. These issues led to poor performance and excessive diffusion levels as shown in the blade loading chart of ‘$Z = 8$’. Comparing the results of ‘$Z = 10$’ and ‘$Z = 12$’ indicates that these two cases provide a uniform flow with appropriate blade loading, though, there are some weak secondary flows near the leading edge of ‘$Z = 10$’ which cause a slight level of diffusion. By comparing the power output and blade loading of ‘$Z = 10$’ and ‘$Z = 12$’ with that of ‘$Z = 14$’ in Figure 11, it is obvious that the last two cases are not superior compared with ‘$Z = 10$’. Hence, ‘$Z = 10$’ is selected for the minimum number of rotor blades since it provides appropriate expansion on both suction and pressure surface with uniform velocity vectors and almost constant output power. It is clear from the above results that CFD analysis can be used to predict a more realistic rotor design in terms of blade profiles and number of blades.

Figure 13 shows the three-dimensional geometry of the final radial turbine design using the proposed methodology of mean-line modeling and CFD simulation. It can generate a power output of 4.662 kW with specific diameter of 4.83 (rotor inlet diameter of 0.0787 m) and specific speed of 0.433 (rotational speed of 55,000 rpm).

Figure 10. Blade angle distribution at half span for three different blade profiles.
Figure 11. Blade loading and velocity vectors at half span for three different blade profiles.
Figure 12. Blade loading and velocity vectors at half span for various rotor blade numbers.
4 CONCLUSIONS

There is a need for designing an efficient small-scale radial turbo expander for DPG systems based on ORC. For this purpose a methodology was developed that combines two techniques namely mean-line modeling and CFD analysis. This methodology was effective in investigating the effects of a wide range of key design parameters such as velocity ratio, flow angles, pressure ratio, mass flow rate, rotational speed, blade profile, rotor diameter and number of rotor blades on the overall turbine performance in terms of power output and efficiency. Initial configurations were generated using the mean-line approach and the flow fields were obtained using the computational fluid dynamics simulation and used to refine the configurations. One major parameter is the number of rotor blades where the CFD analysis predicted that a similar turbine performance can be achieved using lower number of blades than those predicted by the mean-line approach. Using this methodology, a small radial turbine with specific diameter of 4.83 (rotor inlet diameter of 0.0787 m), specific speed of 0.433 (rotational speed of 55 000 rpm) and 10 blades was shown to produce a power output of 4.662 kW and total to static efficiency of 80.16%.

Table 4. Mesh independence study results.

| Number of nodes | Solving CPU time (s) | Rotor total to static efficiency (%) | Power output (W) | Relative change in efficiency (%) | Relative change in power (%) |
|-----------------|----------------------|--------------------------------------|-----------------|-----------------------------------|-----------------------------|
| 1               | 57 281               | 85.01                                | 4894.2          | –                                 | –                           |
| 2               | 104 259              | 82.45                                | 4778.8          | 3.01                              | 2.35                        |
| 3               | 150 100              | 80.16                                | 4699            | 2.77                              | 1.67                        |
| 4               | 240 635              | 78.83                                | 4599.4          | 1.659                             | 2.12                        |

Figure 13. Three-dimensional geometry of the developed rotor with $Z = 10$.

REFERENCES

[1] Sun Z, Zhang XY. Advances on distributed generation technology. Energy Proc 2012;17:32–8.
[2] Ackermann T, Andersson G, Söder L. Distributed generation: a definition. Electr Pow Syst Res 2001;57:195–204.
[3] Kang SH. Design and experimental study of ORC (organic Rankine cycle) and radial turbine using R245fa working fluid. Energy 2012;41:514–24.
[4] Qui G, Shao Y, Li J, et al. Experimental investigation of a biomass-fired ORC-based micro-CHP for domestic applications. Fuel 2012;96:374–82.
[5] Pei G, Li J, Li Y, et al. Construction and dynamic test of a small-scale organic Rankine cycle. Energy 2011;36:3215–23.
[6] Liu H, Shao Y, Li J. A biomass-fired micro-scale CHP system with organic Rankine cycle (ORC)—thermodynamic modeling studies. Biomass Bioenergy 2011;35:3985–94.
[7] Li J, Pei G, Li Y, et al. Evaluation of external heat loss from a small-scale expander used in organic Rankine cycle. Appl Therm Eng 2011;31:2694–701.
[8] Qiu G, Liu H, Riffat S. Expanders for micro-CHP systems with organic Rankine cycle. Appl Therm Eng 2011;31:3301–7.
[9] Tempesti D, Manfrida G, Fiaschi D. Thermodynamic analysis of two micro CHP systems operating with geothermal and solar energy. Appl Energy 2012;97:609–17.
[10] Zhai H, Dai YJ, Wu JY, et al. Energy and exergy analyses on a novel hybrid solar heating, cooling and power generation system for remote areas. Appl Energy 2009;86:1395–404.
[11] Fiaschi D, Lilshitz A, Manfrida G, Fuel-assisted solar thermal power plant with supercritical ORC cycle. In: Proceeding of ECOS2010, Lausanne, 2010.
[12] Hung TC. Waste heat recovery of organic Rankine cycle using dry fluids. Energy Convers Manage 2001;42:539–53.
[13] Saleh B, Koglbauer G, Wendland M, et al. Working fluids for low-temperature organic Rankine cycles. Energy 2007;32:1210–21.
[14] Tchanche BE, Papadakis G, Lambrinos G, et al. Fluid selection for a low-temperature solar organic Rankine cycle. Appl Therm Eng 2009;29:2468–76.
[15] Baines NC. Radial turbines—an integrated design approach. In: 6th European conference on turbomachinery-fluid dynamics and thermodynamics, Lille, France, 2005.
[16] Whitefield A, Baines NC. Design of Radial Turbomachines. Longman, 1990.
[17] Dixon SL, Hall CA. Fluid Mechanics and Thermodynamics of Turbomachinery. 6th edn. Butterworth-Heinemann, Elsevier, 2010.
[18] Moustapha H, Zelesky MF, Baines NC, et al. Fluid-assisted solar thermal power plant with supercritical ORC cycle. In: Proceeding of ECOS2010, Lausanne, 2010.
[19] Fiaschi D, Lilshitz A, Manfrida G, Fuel-assisted solar thermal power plant with supercritical ORC cycle. In: Proceeding of ECOS2010, Lausanne, 2010.
[20] Hung TC. Waste heat recovery of organic Rankine cycle using dry fluids. Energy Convers Manage 2001;42:539–53.
[21] Saleh B, Koglbauer G, Wendland M, et al. Working fluids for low-temperature organic Rankine cycles. Energy 2007;32:1210–21.
[22] Tchanche BE, Papadakis G, Lambrinos G, et al. Fluid selection for a low-temperature solar organic Rankine cycle. Appl Therm Eng 2009;29:2468–76.
[23] Baines NC. Radial turbines—an integrated design approach. In: 6th European conference on turbomachinery-fluid dynamics and thermodynamics, Lille, France, 2005.