AERODYNAMIC DESIGN OPTIMIZATION OF A 200 KW-CLASS RADIAL INFLOW SUPERCRITICAL CARBON DIOXIDE TURBINE

Can Ma
Wuhan Second Ship Design and Research Institute
macan1234@sina.com
Wuhan, Hubei, China

Jun Wu
Wuhan Second Ship Design and Research Institute
wu_jun1986@126.com
Wuhan, Hubei, China

Zhouyang Liu
Wuhan Second Ship Design and Research Institute
zhouyangsea@126.com
Wuhan, Hubei, China

Yuansheng Lin
Wuhan Second Ship Design and Research Institute
lin_ysh@163.com
Wuhan, Hubei, China

ABSTRACT
The supercritical carbon dioxide-based power cycle is very promising for its potentially higher efficiency and compactness compared to steam-based power cycle. Turbine is the critical component in the supercritical carbon dioxide-based cycle which delivers the power. Due to the unique fluid properties of the supercritical carbon dioxide, the design parameters of the turbine in the power system show a number of differences to the conventional steam turbine or gas turbine. Few design experience with the supercritical carbon dioxide turbine has been gained up to now. In this work, a 200 kW-class radial inflow supercritical carbon dioxide turbine is designed as a component of a small supercritical carbon dioxide-based power cycle. The aerodynamic optimization is carried out based on computational fluid dynamics (CFD) to improve the performance of the turbine. The flow field of the optimal design is compared to the original design and the variations in the performance parameters are discussed.

INTRODUCTION
The supercritical carbon dioxide-based Brayton cycle is a promising option for future propulsion and power. Due to the unique fluid properties of the supercritical carbon dioxide, the design parameters of the turbine in the power system show a number of differences to the conventional steam turbine or gas turbine, with the size of the turbine considerably reduced, leading to a more compact power system. In recent years, more and more research on the supercritical carbon dioxide power cycle are carried out (Dostal et al., 2006; Reyes-Belmonte et al., 2016; Yari, 2012). However, few works among them focus on the turbine component. The computational flow simulation of the supercritical carbon dioxide turbine has been carried out and the importance of using accurate fluid properties is pointed out (Jeong et al., 2008; Odabaee et al., 2016). Although some research discusses the design of the supercritical carbon dioxide turbine (Qi et al., 2016; Shi et al., 2015; Zhang et al., 2015), there has been little work on the aerodynamic optimization of the turbine.

In this work, the aerodynamic design optimization of a 200 kW-class radial inflow supercritical carbon dioxide turbine is carried out. The optimization is carried out based on three-dimensional CFD to improve the performance of the turbine. The flow field of the optimal design is compared to the original design and the variations in the performance parameters are discussed.

DESIGN OBJECTIVE
The basic design of the radial turbine is characterized by low specific speed, with the geometric model shown in Fig 1. The nozzle blade number is 9 while the impeller blade number is 12. The aerodynamic design parameters of the turbine are shown in Table 1. The inlet blade height of the impeller is only 1.8 mm and a uniform impeller tip clearance...
of 0.2 mm exists. In this work, the impeller of the radial turbine is re-designed for better performance.

![Figure 1 Original Design of Radial Turbine](image1)

Table 1 Design parameters of the radial turbine

<table>
<thead>
<tr>
<th>Design parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$p_0$ [MPa]</td>
<td>13.85</td>
</tr>
<tr>
<td>$T_0$ [K]</td>
<td>773</td>
</tr>
<tr>
<td>$p_2$ [MPa]</td>
<td>7.84</td>
</tr>
<tr>
<td>$N$ [rpm]</td>
<td>$4 \times 10^4$</td>
</tr>
<tr>
<td>Specific speed $N_s$ [-]</td>
<td>0.22</td>
</tr>
<tr>
<td>Velocity ratio $v$ [-]</td>
<td>0.64</td>
</tr>
</tbody>
</table>

NUMERICAL METHODS

Geometric modelling

The geometric modelling is carried out using the commercial software NUMECA/AutoBlade. The impeller blade is straight with no bend, which results in two controlling blade profiles for the three-dimensional blade geometry. The impeller blade profile is designed with uniform blade thickness which is determined by the manufacturing requirement. Therefore, only the blade camber line needs to be parameterized for the purpose of optimization. The camber line of the blade profile is parameterized with B-spline with four parameters, as shown in Fig 2. In the optimization, the first parameter (camber_h1) is fixed to ensure that the blade maintains a radial inlet blade angle in the optimization, which minimizes the mechanical stress. As a result, there are three free parameters for each blade profile.

![Figure 2 Parameterization of Impeller Blade Camber Line](image2)

Grid generation

In addition to the design variables controlling the blade camber line, the chord length of the blade profile is also allowed to vary within specific range. The variations of the chord length of the two controlling blade profiles are equal, which adds one additional free parameter for the impeller blade. There are seven free design parameters in total for the three-dimensional impeller blade.

![Figure 3 Multi-Block Structured Grid Used for Performance Evaluation](image3)
Flow solver
The three-dimensional steady Reynolds Averaged Navier-Stokes (RANS) equations are solved using the commercial software NUMECA/Fine-Turbo. The fluid property of the supercritical carbon dioxide is based on the NIST database. The 2nd order center scheme is used for the spatial discretization. The two-equation k-ω SST model is used for the turbulence modelling, which is suggested for supercritical carbon dioxide turbomachinery flow in previous work. The four-step Runge-Kutta explicit method is used for the time marching solution of the steady equations, with the full multigrid method adopted for convergence acceleration. As single blade passage is modelled for both the stator and rotor, the mixing-plane model is used for the data exchange on the rotor-stator interface, where the circumferentially averaged flow variables are specified as the boundary condition on the other side.

Optimizer
The aerodynamic optimization is carried out using the commercial software NUMECA/Fine-Design3D. A two-layer artificial neural network is used to approximate the performance parameters based on the geometric design variables. A database consisting of 15 samples with difference geometries generated in random is used for training. The genetic algorithm is used for optimization, with 25 reproduction cycles. For each design iteration, a neural network learning is performed, followed by the genetic algorithm optimization and the CFD flow simulation.

RESULTS AND DISCUSSION

Design optimization
The optimization target is to maximized the isentropic efficiency of the turbine stage. The constraint is that the variation of the mass flow rate less than 0.1%. Therefore, the objective function $F$ is a combination of the isentropic efficiency $\eta$ and the mass flow rate $m$

$$F = w_1 (\eta - 1)^2 + w_2 \left( \frac{m - m_0}{m_0} \right)^2$$

where $m_0$ is the mass flow rate of the original design and $w_1$ is the weight factor.

Thirty design iterations are performed and the design history is shown in Fig 4. The optimal design is obtained at the 25th design iteration with the minimal objective function.

The performance parameters of the original and the optimal designs are shown in Table 2, with the results obtained using the coarse grid CFD solution. The shaft power is increased by 0.61% and the isentropic efficiency is increased by 0.55% after optimization. The mass flow rate is decreased by 0.1%, which satisfies the constraint.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Original</th>
<th>Optimal</th>
<th>Variation</th>
</tr>
</thead>
<tbody>
<tr>
<td>$m$ [kg/s]</td>
<td>3.463</td>
<td>3.460</td>
<td>-0.10%</td>
</tr>
<tr>
<td>$P$ [kW]</td>
<td>215.7</td>
<td>217.1</td>
<td>+0.61%</td>
</tr>
<tr>
<td>$\eta$ [-]</td>
<td>78.12%</td>
<td>78.67%</td>
<td>+0.55%</td>
</tr>
</tbody>
</table>

The meridional shape of the optimal design is compared to the original design in Fig 5. The optimal design shows larger chord length. The allowable variation magnitude of the trailing edge position is set to 5 mm in the optimization and the trailing edge position moves about 3 mm downstream after optimization. The hub and shroud are parameterized with B-spline and the parameters are fixed. Therefore, the hub and shroud shapes change accordingly as the trailing edge position changes.
The three-dimensional geometric model of the original and optimal designs are shown in Fig 6. In addition to the variation in the meridional shape, the blade profile also shows obvious variation. The impact of the geometric variation on the flow field and the performance of the turbine are discussed in the following subsection.

**Figure 6 Geometric Model of Turbine: Original (left) and Optimal (right) Designs**

**Performance evaluation**

After optimization, the flow simulation is carried out with the original grid with 1.2M points for both the original and optimal designs.

The relative Mach number distributions at midspan are shown in Fig 7. The flow field in the impeller passage shows obvious variation, especially in the low Mach number region near the suction surface. To further examine the flow detail in this region, the tangential relative velocity vector in the low Mach number region is shown in Fig 8. For the original design, a small reverse flow region exists near the suction surface, which disappear after the optimization. The elimination of the reverse flow region clearly illustrates the improvement in the impeller flow field after the optimization.

**Figure 7 Relative Mach Number Distributions at Midspan: Original (left) and Optimal (right) Designs**

**Figure 8 Tangential Velocity Vector at Midspan: Original (upper) and Optimal (lower) Designs**

The relative Mach number distributions at 90%, near the impeller tip, are shown in Fig 9. A low Mach number region exist on the suction side from the middle of the passage to the trailing edge, which corresponds to the impeller tip leakage flow. For the original design, the tip leakage flow is directed to the trailing edge. After optimization, as the trailing edge position moves downstream, the tip leakage flow is directed to the pressure surface near the trailing edge. This results in the variation of the impeller blade load near the tip.

The performance parameters of the original and optimal designs obtained from the flow simulation using the fine grid are shown in Table 3. Note that with the fine grid, the performance parameters show slight change compared to the results using the coarse grid as shown in Table 2. The shaft power is increased by 2.2% and the isentropic efficiency is increased by 0.98% after optimization. The mass flow rate is decreased by 0.94%. The performance of the optimal design is obviously improved compared to the original design.

**Figure 9 Relative Mach Number Distributions at 90%: Original (left) and Optimal (right) Designs**

The shaft power is increased by 2.2% and the isentropic efficiency is increased by 0.98% after optimization. The mass flow rate is decreased by 0.94%. The performance of the optimal design is obviously improved compared to the original design.
The impeller design optimization is carried out to maximize the isentropic efficiency with constraint on the mass flow rate variation magnitude. After optimization, the flow field in the impeller is improved, with the reverse flow region near the impeller blade suction surface at midspan eliminated. The shaft power is increased by 2.2% and the isentropic efficiency is increased by about 1% after optimization. The performance of the radial turbine stage is obviously improved after optimization, which demonstrates the effectiveness of the optimization strategy adopted in this work. Although only seven design variables controlling the impeller blade camber line and the chord length are chosen as the design variables in this work, more geometric parameters can be added as the design variables and the turbine performance can be potentially further improved.

CONCLUSIONS

The aerodynamic design optimization of a 200 kW-class radial inflow supercritical carbon dioxide turbine is carried out in this work. The original design is characterized by low specific speed and the impeller inlet blade height is only 1.8 mm, which results in relatively large friction loss and an isentropic efficiency of 78%.

The impeller design optimization is carried out to maximize the isentropic efficiency with constraint on the mass flow rate variation magnitude. After optimization, the flow field in the impeller is improved, with the reverse flow region near the impeller blade suction surface at midspan eliminated. The shaft power is increased by 2.2% and the isentropic efficiency is increased by about 1% after optimization. The performance of the radial turbine stage is obviously improved after optimization, which demonstrates the effectiveness of the optimization strategy adopted in this work. Although only seven design variables controlling the impeller blade camber line and the chord length are chosen as the design variables in this work, more geometric parameters can be added as the design variables and the turbine performance can be potentially further improved.

NOMENCLATURE

Symbols
\( m \) Mass flow rate
\( N \) Rotating speed
\( N_s \) Specific speed
\( P \) Shaft power
\( p \) Pressure
\( p_0 \) Total pressure
\( T_0 \) Total temperature
\( v \) Velocity ratio
\( \eta \) Isentropic efficiency based on shaft power

Subscripts
0 Turbine nozzle inlet
2 Turbine impeller outlet

Abbreviations
CFD Computational Fluid Dynamics
NIST National Institute of Standards and Technology
RANS Reynolds Averaged Navier-Stokes
SST Shear Stress Transport

ACKNOWLEDGMENTS
This work is partly supported by a grant from the Hubei Provincial Natural Science Foundation of China [Project Number = 2016CFA019].

REFERENCES