

Article

Numerical Investigation on Aerodynamic Performance of SCO₂ and Air Radial-Inflow Turbines with Different Solidity Structures

Yuqi Wang ¹, Jinxing Li ², Di Zhang ^{1,*}  and Yonghui Xie ²

¹ MOE Key Laboratory of Thermo-Fluid Science and Engineering, School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China; wyq_xjtu@163.com

² Shaanxi Engineering Laboratory of Turbomachinery and Power Equipment, School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China; xing837439166@stu.xjtu.edu.cn (J.L.); yhxie@mail.xjtu.edu.cn (Y.X.)

* Correspondence: zhang_di@mail.xjtu.edu.cn; Tel.: +86-29-8266-6559

Received: 24 February 2020; Accepted: 9 March 2020; Published: 19 March 2020



Featured Application: This work can provide the design reference of rotor solidity for SCO₂ and air radial-inflow turbines as well as a new splitter structure to improve the performance of low solidity case.

Abstract: Supercritical carbon dioxide (SCO₂) is of great use in miniature power systems. It obtains the characteristics of high density and low viscosity, which makes it possible to build a compact structure for turbomachinery. For a turbine design, an important issue is to figure out the appropriate solidity of the rotor. The objective of this research is to present the aerodynamic performance and provide the design reference for SCO₂ and air radial-inflow turbines considering different solidity structures. For the low solidity case of SCO₂ turbine, new splitter structures are proposed to improve its performance. The automatic design and simulation process are established by batch modes in MATLAB. The numerical investigation is based on a 3D viscous compressible N-S equation and the actual fluid property of SCO₂ and air. The distributions of flow parameters are first presented. Rotor blade load and aerodynamic force are then thoroughly analyzed and the aerodynamic performances of all cases are obtained. The SCO₂ turbine has larger power capacity and higher efficiency while the performance of the air turbine is less affected by rotor solidity. For both SCO₂ and air, small solidity can cause the unsatisfactory flow condition at the inlet and the shroud section of the rotor, while large solidity results in the aerodynamic loss at the trailing edge of rotor blade and the hub of rotor outlet. A suction side offset splitter can greatly improve the performance of the low solidity SCO₂ turbine.

Keywords: radial-inflow turbine; supercritical carbon dioxide; air; rotor solidity; aerodynamic performance

1. Introduction

In recent years, the study of supercritical carbon dioxide (SCO₂) Brayton cycle and its components has attracted lots of attention. Various heat sources including solar power [1–5], nuclear power [6,7] and waste-heat utilization [8] are employed. As the key component in a Brayton cycle, the design and characteristics of turbomachinery deserve to be investigated in depth. SCO₂ has a critical point around room temperature (7.38 MPa, 304.25 K). As a working fluid in a power cycle, SCO₂ has a large number of advantages. First of all, it is environmentally friendly and the critical point is easy to reach. Hence, it is safe and cheap in industrial applications. Additionally, the high density and low viscosity of SCO₂ can result in the high efficiency and compact mechanical structure of turbines and compressors in the

power system [9–11]. The rapid changes of density near the critical point can reduce the compressor work and lead to a higher heat transfer coefficient in heat regenerators and precoolers [12]. Finally, there is little phase change in a SCO₂ power cycle. Thus, it needs fewer valves and no condenser, which results in a concise cycle.

To realize this proposed technology, Sandia National Laboratory has developed several SCO₂ power cycles including various power class [13]. The radial turbine and compressor test rigs were established to investigate the key techniques. However, the outcomes of efficiency and rotation speed were far from the expectations. Hence, the design method and flow mechanism need to be further considered before the commercialization of this technique. A traditional design process of the turbomachinery is time-consuming. It needs several engineers to conduct multiple numerical calculation to obtain the case of best performance. Hence, it is vital to establish an automatic process in order to save the manpower and accelerate the design.

The flow conditions are rather complex in turbomachinery, which results in the difficulty of turbine design. In the past years, the research of design, influence of key parameters and optimization are widely covered [14–17]. Solidity, as a design parameter, plays an important role in turbine design. It was proposed by Zweifel [18] in 1945, in which he estimated the optimum solidity for turbines with large angular deflection. However, Horlock [19] pointed out that the prediction of Zweifel's optimum solidity correlation was limited to the outlet flow angle of the blade. In the past investigations, the influence of solidity on axial wind turbine performance occupied the vast majority. Chen et al. [20] investigated the effects of flanged diffusers on rotor performance of small wind turbines with different rotor solidities and wind speeds. Mohamed [21] studied the effect of the turbine solidity and the usage of hybrid system between drag and lift types on small wind turbine performance numerically and experimentally. Eboibi et al. [22] experimentally investigated the influence of solidity on the performance and flow field aerodynamics of vertical axis wind turbines at low Reynolds numbers. Gao et al. [23] thoroughly analyzed the effects of rotor solidity and leakage flow on the unsteady flow in axial turbine.

Some researches focus on the solidity of stator vane in radial-inflow turbines. Simpson et al. [24] conducted the numerical and experimental study of the performance effects of varying vaneless space and vane solidity in radial turbine stators. Pereiras et al. [25] concentrated on the influence of the guide vane solidity on the performance of a radial impulse turbine with pitch-controlled guide vanes. Dong et al. [26] evaluated the effects of outlet blade angle, solidity, blade height, expansion ratio, and surface roughness on the stator velocity coefficient.

To sum up, the stator solidity has been considered in radial-inflow turbines while the choice of rotor solidity is rarely investigated, especially its impact in a SCO₂ turbine. As the working fluid approaches the critical point at the outlet of a SCO₂ turbine and the value of solidity tends to decrease along the flow direction in a radial-inflow rotor, the concentration of rotor solidity is needed. We establish an automatic design process by batch modes in MATLAB 2019b of MathWorks (Natick, MA, USA) with an accurate numerical simulation method of radial-inflow turbines. Several solidity structures are considered and new splitter structures are proposed and analyzed to improve the performance of low solidity case. The distributions of flow parameters are first presented. Rotor blade load and aerodynamic force are then thoroughly analyzed and the aerodynamic performances are obtained. The design reference of rotor solidity for SCO₂ and air turbines are provided.

2. Modeling

2.1. Establishment of Automatic Design Process

To accelerate the design process, an automatic design and calculation process of the radial-inflow turbine is firstly established by calling batch modes in MATLAB. The flow chart is shown in Figure 1. The initial geometric model of the turbine is obtained based on design parameters, such as inlet pressure and temperature, outlet pressure, output power, rotation speed, etc. Then, the rotor solidity

and working fluid used in the calculation are determined. The macro files of batch mode are called by MATLAB to execute the process in Workbench BladeGen, Turbogrid, CFX, and CFD-Post. The modeling, discretization, numerical simulation and post-processing are then completed. Finally, the program repeats this procedure to obtain the results of the required different cases.

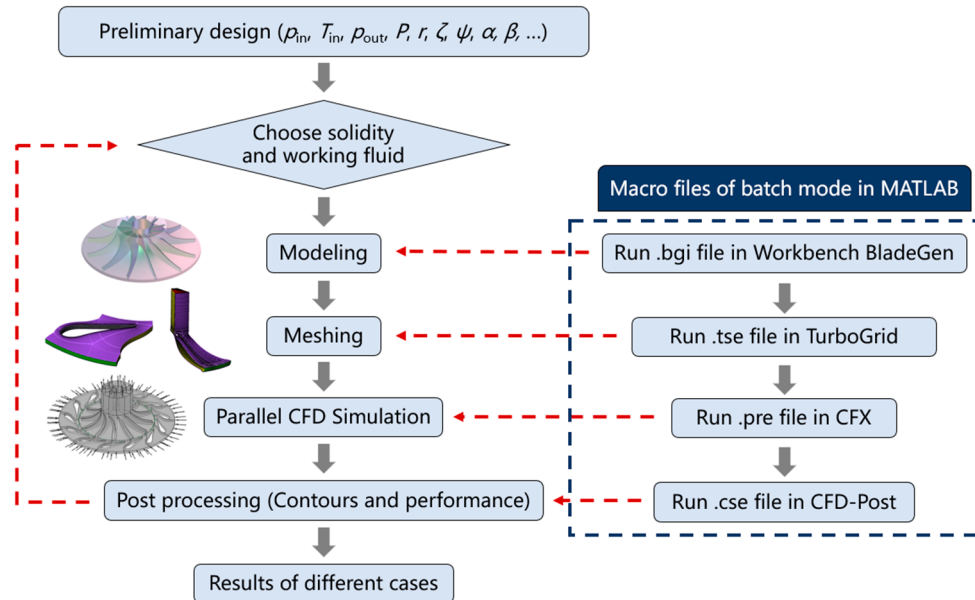


Figure 1. Flow chart of design and simulation.

The thermodynamic design is based on the conservation of momentum, mass and energy equations. The fluid properties of SCO_2 and air are obtained from NIST REFPROP. The basic principal for the thermodynamic design is to ensure the highest efficiency at a given rotation speed [27]. Due to space limits, the following formulas are some of the calculations in the thermodynamic design program. The influence of friction loss at the wheel back and the flow loss in the volute are ignored. The loss producing by nozzle and impeller are considered. Hence, the isentropic efficiency can be estimated as Equation (1):

$$\eta_{is} = 1 - (1 - \alpha)(1 - \zeta^2) - \left(\frac{\omega_{out}}{C_{is}}\right)^2 \left(\frac{1}{\psi^2} - 1\right) \quad (1)$$

where α is the degree of reaction. ζ and ψ are the velocity coefficient of nozzle and impeller respectively, which represents the loss in nozzle and impeller. We adopt 0.96 for ζ and 0.84 for ψ . ω_{out} and C_{is} are, respectively, the relative velocity at turbine outlet and the isentropic ideal velocity.

To gain the best isentropic efficiency in the thermodynamic design, we can correspondingly acquire the degree of reaction, the velocity ratio and etc. For example, the degree of reaction is chosen as 0.47 and the velocity ratio is 0.67. After the determination of design parameters, we can calculate the geometry parameters. For example, the impeller diameter is acquired by Equation (2):

$$D = \frac{60v_{in}}{\pi r} \quad (2)$$

where r is the rotating speed, which is 50,000 rpm in this case. v_{in} is the linear velocity at the inlet of the impeller.

The blade height at the turbine inlet can be calculated as Equation (3):

$$l_{in} = \frac{\dot{m}}{\pi D \rho_{in} \omega_{in}} \quad (3)$$

where $\dot{m}$ is the mass flow rate, ρ_{in} and ω_{in} are, respectively, the density and the relative velocity at the rotor inlet.

The area of turbine outlet is calculated in Equation (4):

$$A_{out} = \frac{\dot{m}}{\rho_{out}\omega_{out} \sin \beta_{out}} \quad (4)$$

where β_{out} is the flow angle at turbine outlet, ρ_{out} and ω_{out} are, respectively, the density and the relative velocity at rotor outlet. With the empirical coefficients regarding to impeller diameter, the average outlet diameter and axial length are determined. Then we can obtain the geometric parameters of meridian plane for the rotor.

The geometric parameters are calculated by estimating the velocity at both inlet and outlet of the stator and the rotor. To compare the difference of air and SCO_2 , we firstly conduct the thermodynamic design using SCO_2 and then, estimate the working condition with air to conduct the numerical simulation with the same radial-inflow turbine model.

2.2. Research Model

The working fluid first flows into the nozzle flow passage. In the nozzle, the pressure gradually declines and accordingly, the velocity increases. Figure 2 presents the geometry model of nozzle. A straight blade with uniform geometry angles from hub to shroud direction is adopted due to the low blade height. The geometry of nozzle is identical in all conditions with different solidity structures. The red part represents high temperature and pressure while the blue part stands for lower temperature and pressure. Detailed geometric parameters, i.e., the diameter of inlet and outlet, the blade height, and the geometry angle of the inlet and outlet are given in Figure 2.

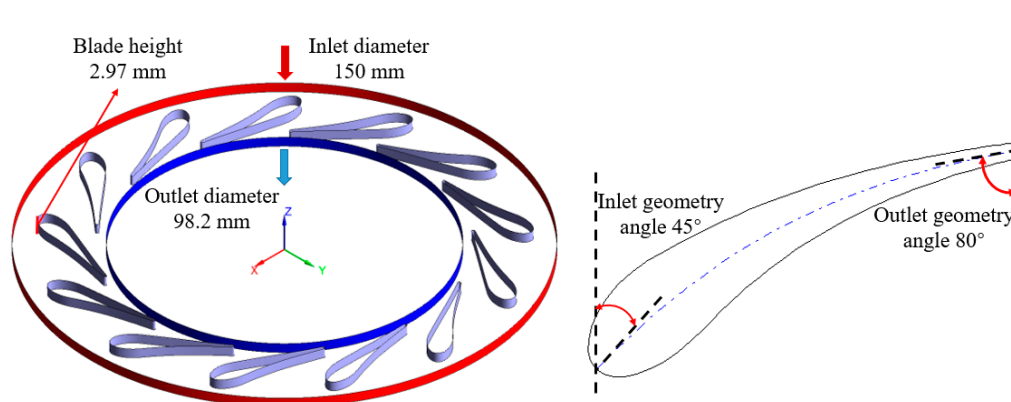


Figure 2. Geometry model of nozzle.

After the process in the nozzle, the working fluid enters from a radial direction into the impeller. In this process, SCO_2 expands in the impeller and exits from axial direction. The backward bent vane is adopted to guarantee the designed velocity triangle and a shrouded impeller is used to promote the aerodynamic efficiency. Figure 3 shows the geometry model of impeller, taking blade number 12, as an example. After the calculation of impeller diameter, blade height of inlet and outlet, external diameter of outlet and axial length, the arc to connect each point is determined by design experience. In Figure 3, the red arrow stands for high temperature and pressure while the blue arrow represents relatively low temperature and pressure. Brennen [28] gave the definition of solidity for radial pumps. Likewise, we define the solidity with cord length and pitch in a radial-inflow turbine. These parameters are obtained in Workbench BladeGen while modelling. In this research, the rotor solidity is defined by c/s , where c is the cord length of the rotor blade and s is the trailing edge pitch at the outlet of the impeller, as shown in Figure 3.

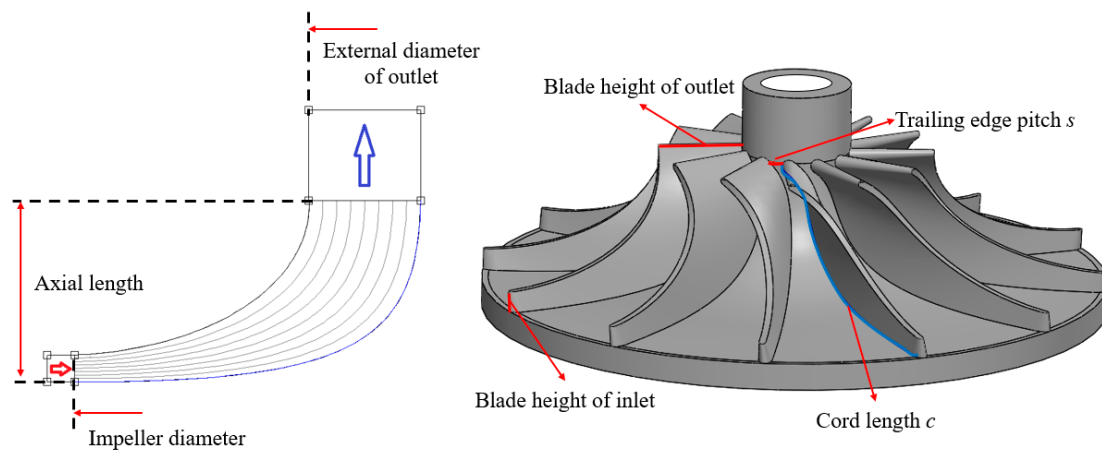


Figure 3. Geometry model of the impeller.

The concrete geometric parameters of impeller are shown in Table 1. Generally, the number of nozzle blades and the number of impeller blades should be relatively prime for the consideration of reducing the exciting force of turbomachinery. Hence, while the nozzle blade number is designed as 13, we choose five kinds of rotor solidity, which are realized by changing the rotor blade numbers. They are even from 8 to 16. The large range of rotor solidity can present its effect on aerodynamic performance.

Table 1. Geometric parameters of the impeller.

Parameter	A	B	C	D	E
Blade number	8	10	12	14	16
Impeller diameter/mm			92.2		
External diameter of outlet/mm			40.6		
Axial length/mm			20		
Blade height of inlet/mm			2.97		
Blade height of outlet/mm			24.6		
Inlet geometry angle/°			90		
Outlet geometry angle/°			27		
Chord length (c)/mm			50		
Pitch at trailing edge (s)/mm	6.28	5.03	4.19	3.59	3.14
Solidity (c/s)	7.96	9.95	11.94	13.93	15.92

High solidity usually results in large weight and axial thrust of the wheel. In some application environments such as aerospace and warships, the weight of the wheel is limited. The overlarge axial thrust and the trans-critical problem of the high solidity case also need consideration in the turbine design. Hence, for the low solidity case of SCO_2 turbine mentioned above, we try to establish a new structure to improve the flow condition at the leading edge of the rotor blade. A splitter, which is normally used in centrifugal compressors, is introduced here. We establish a pressure side (ps) offset splitter and a suction side (ss) offset splitter to compare with the normal one, as shown in Figure 4. The offset value is 20% of the local pitch, that is, the distance between the splitter blade the closest main blade is 30% of the local pitch.

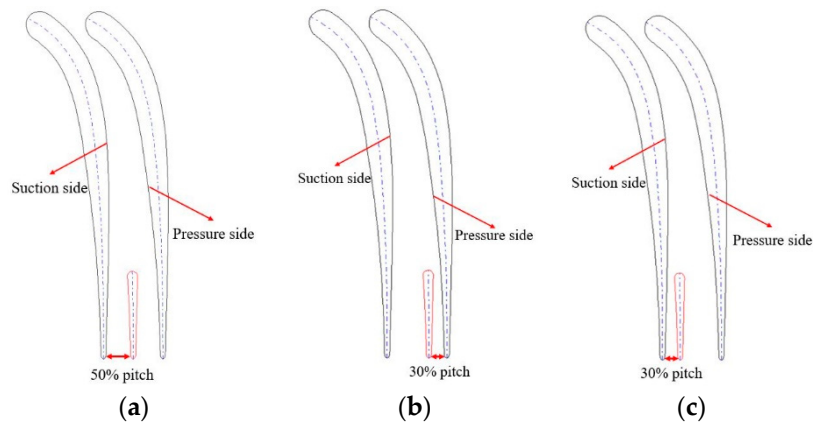


Figure 4. Splitter structures for the lowest solidity case: (a) normal; (b) ps offset; (c) ss offset.

3. Methodology

A 3D viscous compressible CFD simulation is carried out in commercially-available software CFX 18.2. In this research, the computing resource of an i7-3770 CPU and eight threads is used. In order to obtain the specific characteristics of SCO₂ and air radial-inflow turbines with different solidity structures, an accurate numerical simulation model needs to be established. The following sections introduce the validation of turbulence model, working fluids as well as the discretization method used in numerical calculations.

3.1. Validation of Computational Method

In CFD simulation, the conservation equation of mass, energy and momentum are solved by Reynolds-Averaged Navier–Stokes (RANS) method. In this investigation, a compressible and steady turbulent flow is simulated. The equations can be expressed as below:

$$\frac{\partial(\rho u_i)}{\partial x_j} = 0 \quad (5)$$

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial\left(\mu \frac{\partial x_i}{\partial x_j} - \rho \overline{u'_i u'_j}\right)}{\partial x_j} + S_i \quad (6)$$

$$\frac{\partial(\rho u_i T)}{\partial x_j} = -\frac{\partial\left(\Gamma \frac{\partial T}{\partial x_j} - \rho \overline{u'_i T'}\right)}{\partial x_j} + S \quad (7)$$

where u stands for velocity components, x is displacement term, S_i and S are, respectively, the source term of momentum and inner heat. To handle the governing equations of turbulent flow, the RANS method is adopted. Hence, a turbulence model is needed for the calculation of Reynolds stress.

To discretize the governing equations, an element-based finite volume method is adopted. TurboGrid 18.2 is employed to generate the hexahedral meshes in all fluid domains, which correspond to lower truncation error. To obtain the best accuracy and the influence of the wake flow, a full cycle of the impeller and the nozzle is adopted in the investigation. Figure 5 shows the mesh of the fluid domain including the partially enlarged view of the leading edge of the nozzle, the rotor-stator interface and the trailing edge of the rotor. O-type mesh is applied around the rotor blades and nozzle blades. The mesh of the boundary layers has been densely generated to adapt the turbulence model. Concretely, the discretization in the boundary layer area ensures the averaged solver y^+ is set around 1 as recommended in the CFX User Guide.

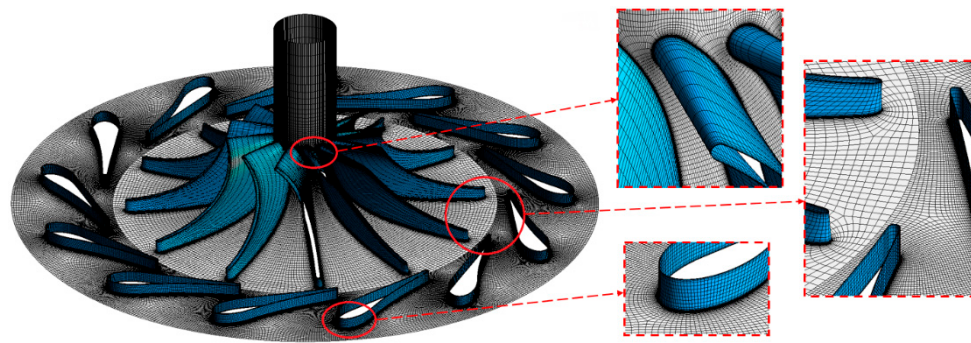


Figure 5. Mesh of the computational field.

For the determination of near-wall velocities, the automatic wall function is chosen. Total energy equation including the viscous work term is used for energy conservation. A high-resolution advection scheme of CFX is used. The boundary conditions employed in this research are listed in Table 2. We use total pressure and temperature as inlet boundary conditions. To simulate the inlet volute, the flow angle at the nozzle inlet is set as 45° . The static pressure is used for outlet boundary condition while the rotation speed is set as 50,000 rpm for the rotor. A frozen-rotor method is adopted to handle the interface between stator and rotor. An automatic domain initialization method is used.

Table 2. Boundary conditions of the calculated cases.

Working Fluid	Total Pressure of Inlet/MPa	Total Temperature of Inlet/K	Static Pressure of Outlet/MPa	Rotation Speed/rpm
SCO ₂	15	550	8	50,000
Air	0.5	550	0.3	50,000

The computation is considered converged when RMS residuals are below 10^{-5} . In the meantime, we evaluate the mass flow rate of the turbine inlet and outlet. If they are equal and slightly fluctuate with each iteration step, the case reaches convergence.

In order to validate the method of CFD simulation, we choose a similar radial-inflow turbine working with air from Deng et al. [29], which possesses experimental results. A CFD simulation with Deng's model and boundary conditions under design point is conducted. Four common turbulence models for turbomachinery simulation including $k-\varepsilon$, RNG $k-\varepsilon$, $k-\omega$, and SST $k-\omega$ are adopted and the validation results are given in Figure 6. It can be seen from Figure 6 that compared with the results in [29], the mass flow rate and efficiency calculated by $k-\varepsilon$ models are higher. Compared with the experiment, we adopt a shrouded impeller in the design. The leakage flow derived from tip clearance is not taken into account. Besides, due to the simplification of the research model, the leakage flow between the nozzle and impeller is not considered in the numerical simulation. These cause the simulated mass flow rate higher than Deng's experiment. Comparing four turbulence models, SST $k-\omega$ turbulence model has the smallest error. The relative errors of the mass flow and efficiency are, respectively, 3.9% and -0.12% . Therefore, we use the SST $k-\omega$ turbulence model in the CFD analysis. The turbulence model was first proposed by Menter [30] and it was based on the $k-\varepsilon$ and $k-\omega$ turbulence models. This model can accurately simulate the wall region and does not acquire high mesh quality as in the $k-\omega$ turbulence model.

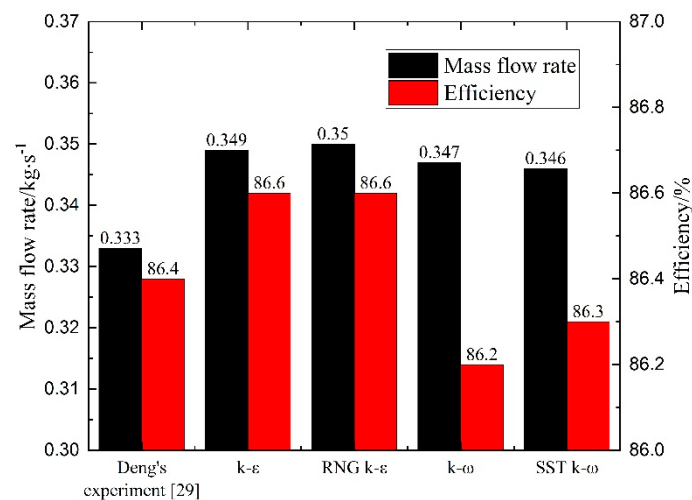


Figure 6. Validation of different turbulence models.

To guarantee the accuracy of the numerical simulation, a mesh independence test has been conducted under the condition of solidity 11.94. Grid convergence index (GCI) measurement is employed, which was proposed by Roache [31] to provide an objective asymptotic approach to quantification of uncertainty of grid convergence. It is based on the theory of generalized Richardson Extrapolation and involves the comparison of discrete solutions at different grid resolutions. The method of GCI is commonly employed in the grid refinement study [32].

To calculate the GCI, three mesh sizes are used with a constant refinement ratio $r = 1.2$. The value of F_s is chosen 1.25 as suggested by Roache [31]. The results are presented in Table 3. The convergence condition is evaluated by the convergence ratio $R = (f_3 - f_2)/(f_2 - f_1)$, which is 0.625 considering mass flow rate and 0.333 considering output power. Hence, it corresponds to monotonic convergence in this case. The small value of GCI indicates the reduced dependency of results on mesh size. When the total element number is 3,354,000, GCI_{12} for mass flow rate and output power are below 1%. Hence, the fine mesh scheme (f_1) is accurate to conduct the numerical simulation of all cases.

Table 3. Result of grid convergence index.

Mesh	Total Element Number/ 10^4	Mass Flow Rate/kg·s ⁻¹	GCI/%	Output Power/kW	GCI/%
Coarse(f_3)	113.2	6.581	$GCI_{23} = 1.02$ $GCI_{12} = 0.64$	343.03	$GCI_{23} = 0.16$ $GCI_{12} = 0.05$
Medium(f_2)	193.6	6.549		342.16	
Fine(f_1)	335.4	6.529		341.87	

3.2. Validation of Fluid Property

For the working fluid air, the fixed composition mixture is set up in CFX 18.2 with the state equation of Peng Robinson. This option is commonly used in CFX to model the real gas properties for gas density, enthalpy, etc.

In the simulation of a SCO_2 turbine, it is of great importance to precisely obtain the fluid property of SCO_2 . For the simulated condition, the state of working fluid lies in the supercritical region. To simulate the fluid property precisely, we use the real gas property (RGP) format table to implement the sharply variable density and specific heat capacity near the critical point. The tabulated pressure and temperature region are, respectively, 7 MPa to 16 MPa and 300 K to 600 K in order to simulate the fluctuated state parameter during the converging process.

The parameters for the RGP table are obtained from NIST REFPROP, which is widely referred to as a fluid property database [33]. MATLAB is used to generate the RGP format file with a predefined

temperature and pressure range. Figure 7 gives the density variation with the temperature and pressure range for the SCO₂ turbine.

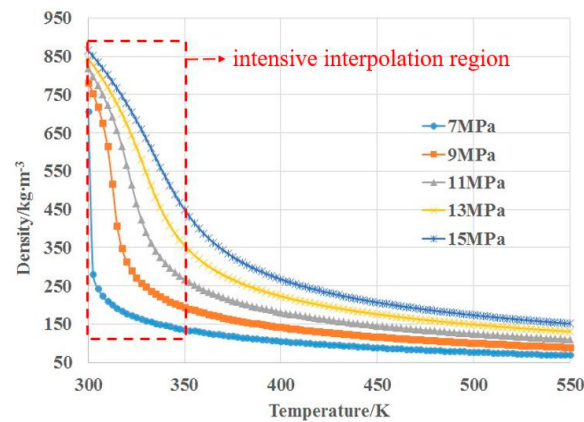


Figure 7. Density variation with temperature and pressure of SCO₂.

As shown in Figure 7, the density of the working fluid changes sharply around the critical point. The CFX solver calculates the SCO₂ property by bilinear interpolation between parameter points. Hence, we choose to densely generate the RGP table near the critical point for the balance of accuracy and computation time. The region is shown in the dashed box in Figure 7. For the temperature range, we adopt a gradient of 2 K for 300–350 K while for 350–600 K, we use a gradient of 5 K. For the pressure range, we utilize 0.1 MPa as a gradient between 7–8 MPa while 0.2 MPa as a gradient is employed between 8–16 MPa. Compared with 2 K for all temperature range and 0.1 MPa for all pressure range, the computation time is effectively reduced by 20.4%.

The accuracy of RGP format table using in ANSYS CFX has been validated by Odabae et al. [34] and Ameli et al. [35]. We conduct a validation in the selected range of this research. Three points are chosen to validate the precision of RGP file, which are, respectively, the inlet, outlet and the interface between stator and rotor. We consider four parameters to test the relative error between CFX calculation and data from NIST, which are density, enthalpy, C_p and dynamic viscosity. As shown in Table 4, the maximum relative error is 0.009%. Hence, the RGP table is sufficiently accurate for the simulation of SCO₂ working fluid.

Table 4. Validation of SCO₂ fluid property.

Parameter	Density/kg·m ⁻³	Enthalpy/kJ·kg ⁻¹	C_p /J·kg ⁻¹ ·K ⁻¹	Dynamic Viscosity/Pa·s
Inlet				
CFX	173.19	648.10	1243.97	2.6859×10^{-5}
NIST	173.19	648.10	1244.00	2.6859×10^{-5}
Relative error/%	0	0	0.002	0
Interface between stator and rotor				
CFX	144.15	630.05	1211.87	2.5113×10^{-5}
NIST	144.14	630.05	1211.80	2.5111×10^{-5}
Relative error/%	0.006	0	0.006	0.008
Outlet				
CFX	106.35	601.88	1159.11	2.2715×10^{-5}
NIST	106.35	601.88	1159.10	2.2717×10^{-5}
Relative error/%	0	0	0.001	0.009

4. Results and Discussion

After the automatic design calculation by batch modes in MATLAB, we obtain the aerodynamic performance of cases with different solidity structures and working fluid. In this chapter, we first present the flow parameters to understand the flow mechanism of different solidity structures and working fluid. Then the rotor blade load and aerodynamic force are compared and analyzed. Finally, the aerodynamic performance is presented to estimate the best solidity and the effectiveness of new splitter structures are tested for low solidity case.

4.1. Distributions of Flow Parameters

4.1.1. Pressure Distribution

The pressure distribution reveals the power capacity of the turbine. In this section, the pressure distribution of the SCO_2 and air turbine are presented and compared. Figure 8 shows the pressure distribution at the mid-span of the SCO_2 radial-inflow turbine. In a SCO_2 radial-inflow turbine, the rotor solidity gradually increases along the flow direction as the pitch at leading edge is much wider than that of trailing edge. As shown in Figure 8, when the rotor solidity is the minimum, the suction side of the rotor blade exists an apparent low-pressure region and the pressure distribution is non-uniform. With the rotor solidity increasing, the low-pressure region tends to decrease and the working capacity for each blade decreases. The diminishing space of rotor flow passage restricts the flow of SCO_2 fluid. Hence, the expansion of working fluid tends to be light in (d) and (e). Condition (c) has a most uniform pressure distribution and the working capacity of blades is satisfying. In condition (a) and (b), the large space at the inlet of the rotor causes the disorder of the pressure distribution.

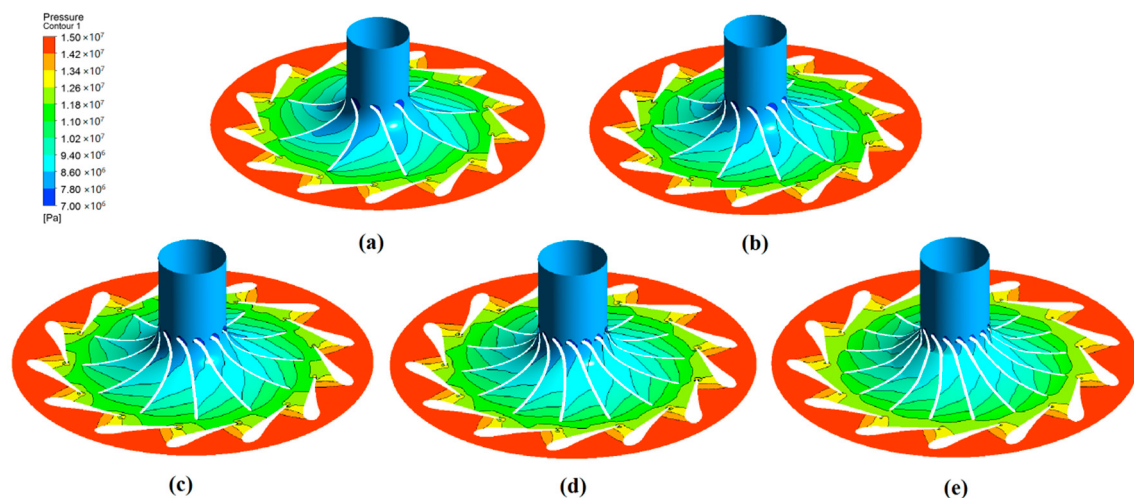


Figure 8. Pressure distribution at mid-span of SCO_2 : (a) solidity 7.96; (b) solidity 9.95; (c) solidity 11.94; (d) solidity 13.93; (e) solidity 15.92.

As is known to all, when the temperature or pressure is around critical point, the physical property of SCO_2 can be sharply variable, especially for density and specific heat capacity. In the turbomachinery with working fluid SCO_2 , it is a priority problem to minimize the region where SCO_2 transforms from a supercritical state to a subcritical state to obtain stable operation. We term the transformation from the supercritical state to the subcritical state as a trans-critical phenomenon. Hence, Figure 9 presents the trans-critical phenomenon at the trailing edge. It is a vital position in radial-inflow turbine to analyze.

As Figure 9 presents, the trans-critical phenomenon is obvious at the trailing edge of the SCO_2 radial-inflow turbine. In condition (a) and (b), the region area is rather small and it concentrates on the tip of the rotor blade. As the rotor solidity increases, the pitch at trailing edge decreases. Hence, the flow area of SCO_2 reduces, which distinctly extends the area of trans-critical region. As we can see,

in condition (c), the minimum pressure along the fluid domain is below 6.5 MPa. With the further decrease of flow area, trans-critical region also appears at the root of rotor blade. In (d) and (e), there exists a local constriction at the trailing edge of rotor blade, which causes an entire trans-critical region at the pressure side of trailing edge.

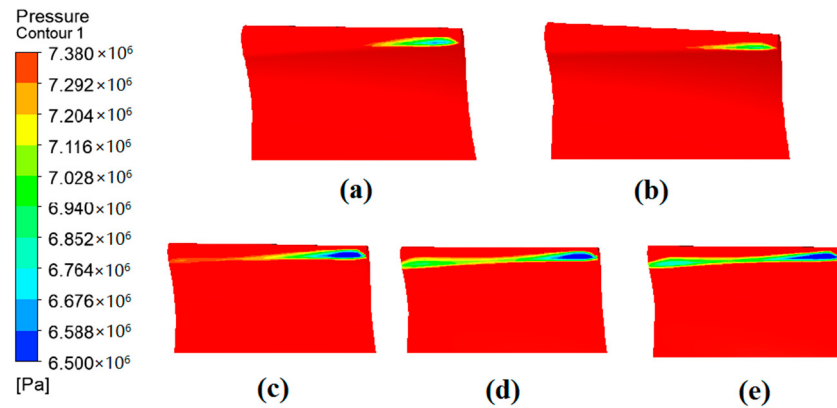


Figure 9. Trans-critical phenomenon at the trailing edge: (a) solidity 7.96; (b) solidity 9.95; (c) solidity 11.94; (d) solidity 13.93; (e) solidity 15.92.

For comparison, Figure 10 shows the pressure distribution at the mid-span with air as working fluid. Likewise, when the rotor solidity is the minimum, the pressure gradient is not uniform in the middle of the flow passage. A low-pressure region presents at the suction side of rotor blade in Figure 10a,b. In (c) and (d), the pressure distribution is uniform while in (e), the pressure difference between suction side and pressure side of the blade decreases. It is worth mentioning that in an air turbine, the low-pressure region at the suction side of rotor trailing edge vanishes. There is no trans-critical phenomenon in the air turbine.

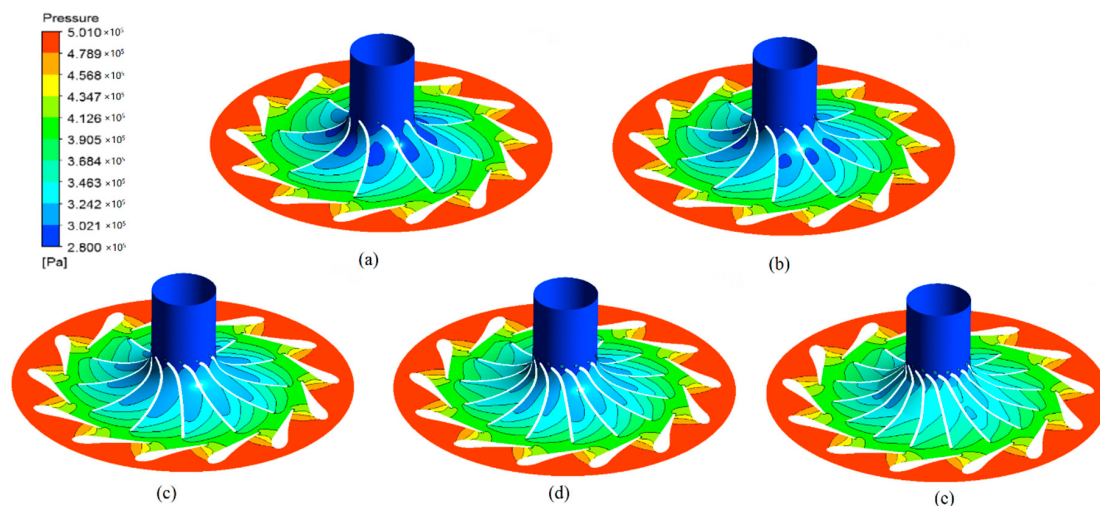


Figure 10. Pressure distribution at mid-span of air: (a) solidity 7.96; (b) solidity 9.95; (c) solidity 11.94; (d) solidity 13.93; (e) solidity 15.92.

The averaged pressure along spanwise location at turbine outlet are presented in Figure 11, where (a) stands for SCO_2 and (b) is air. When the value of spanwise location is 1, it corresponds to the shroud while value zero corresponds to the hub. As Figure 11a presents, the distribution of the averaged pressure along spanwise location at turbine outlet is quite different with the change of rotor solidity in a SCO_2 turbine. Especially when the solidity is 15.92, the power capacity at the different cross-section of the blade varies a great deal. The local constriction at the hub results in more pressure loss and it

corresponds to the lowest pressure. While at the shroud, the expansion of working fluid is incomplete, which results in the highest pressure. In general, the cross-section of the hub has the highest power ability. Comparing five kinds of solidity, the pressure along spanwise tends to be more uniform as the solidity decreases. This is because higher solidity greatly disturbs the flow between blades, which is not beneficial for the safe operation of the turbine. In Figure 11b, the tendency is similar in an air turbine except that the pressure along spanwise is less affected by solidity. The working fluid air has no sharp change of fluid property at the outlet when the solidity is high.

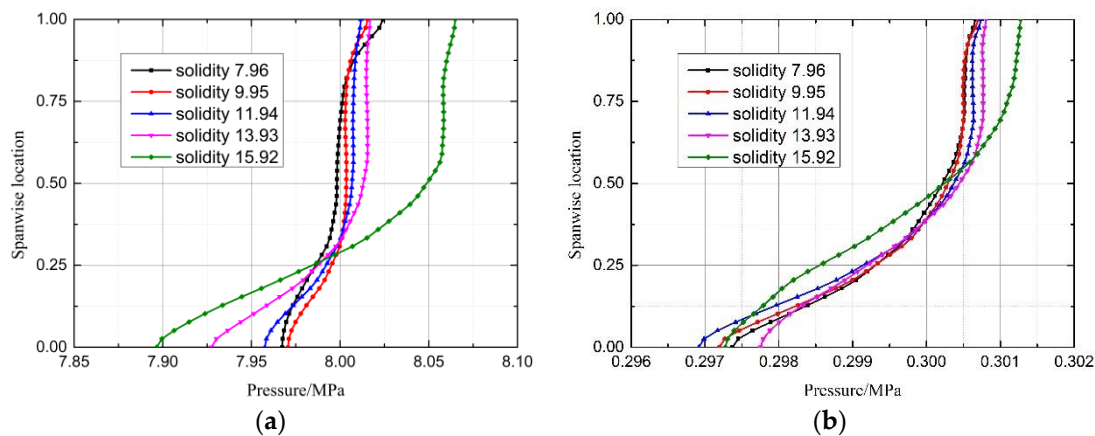


Figure 11. Averaged pressure along spanwise location at turbine outlet: (a) SCO_2 ; (b) air.

4.1.2. Entropy Distribution

The static entropy distribution can reflect the aerodynamic loss in the flow passage. For different rotor solidity, the difference of aerodynamic loss mainly concentrates on the outlet. In this section, the static entropy at the outlet of SCO_2 and air are presented and compared.

The averaged static entropy along spanwise location at turbine outlet are presented in Figure 12. Spanwise location 1 and 0 correspond to the shroud and hub, respectively. The green plane is the contour location. It is acknowledged that higher entropy corresponds to higher aerodynamic loss. For both SCO_2 and air, the aerodynamic loss mainly concentrates on the hub and shroud area due to the boundary layer effect of the impeller. The distribution is quite different with the change of rotor solidity. At the shroud, the highest static entropy appears due to the relatively vacant flow passage when solidity is 7.96 in the SCO_2 turbine. For air, it locates at solidity 9.95, which means the decrease of solidity less affects the aerodynamic loss at the shroud. While at the hub, the highest static entropy presents when solidity is 15.92 on account of the limited flow area for both two working fluids.

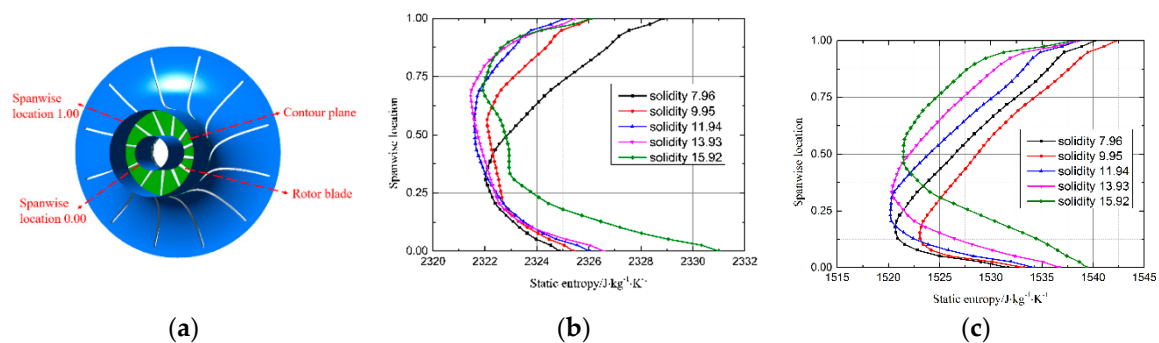


Figure 12. Averaged static entropy along spanwise location at turbine outlet: (a) location of graphical results; (b) SCO_2 ; (c) air.

Figure 13 shows the entropy distribution at outlet with surface streamlines of the SCO_2 radial-inflow turbine. When the rotor solidity is the minimum, the local entropy production in the flow passage near

the shroud is apparent. Due to the low solidity, the disordered flow near the shroud results in several vortices, thus causing local entropy concentration. With the rotor solidity increases, the vortices gradually disappear and the increase of local entropy near the shroud completely vanishes in condition (d), i.e., solidity 13.93. The location of entropy concentration changes to the flow passage near the hub. When rotor solidity is the minimum, the flow near the shroud is disordered while solidity is the highest, the diminishing space of the rotor flow passage restricts the flow of SCO₂ fluid. These reasons cause the local enlargement of entropy as well as aerodynamic loss.

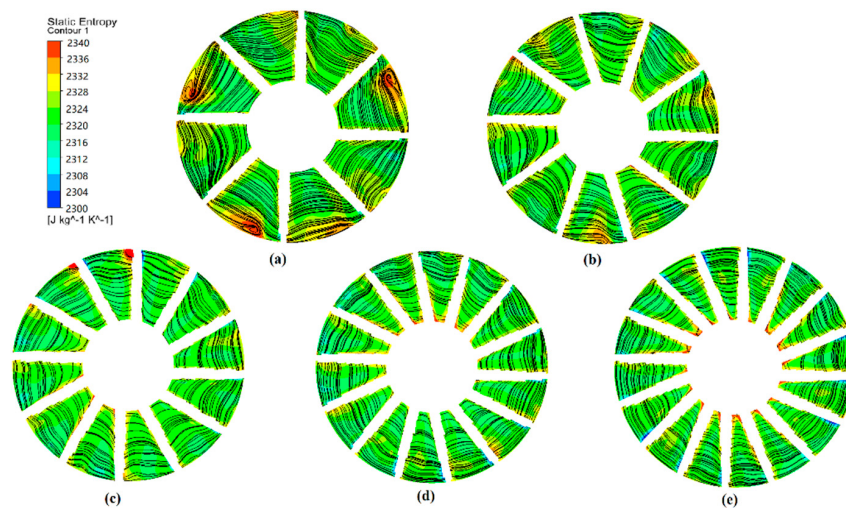


Figure 13. Entropy distribution at outlet with surface streamlines of SCO₂: (a) solidity 7.96; (b) solidity 9.95; (c) solidity 11.94; (d) solidity 13.93; (e) solidity 15.92.

Comparing to Figures 13 and 14 shows the entropy distribution at outlet with surface streamlines of the radial-inflow turbine working with air. The static entropy of the air turbine is apparently lower. In Figure 14a,b, the local vortex at the shroud causes local entropy production while in the SCO₂ turbine, the vortex nearly disappears when solidity is 9.95. With the solidity further increasing, the entropy production still concentrates on the shroud side even when the solidity becomes the highest. Hence, in the air turbine, the diminishing space of rotor flow passage does not cause the local enlargement of aerodynamic loss at the hub.

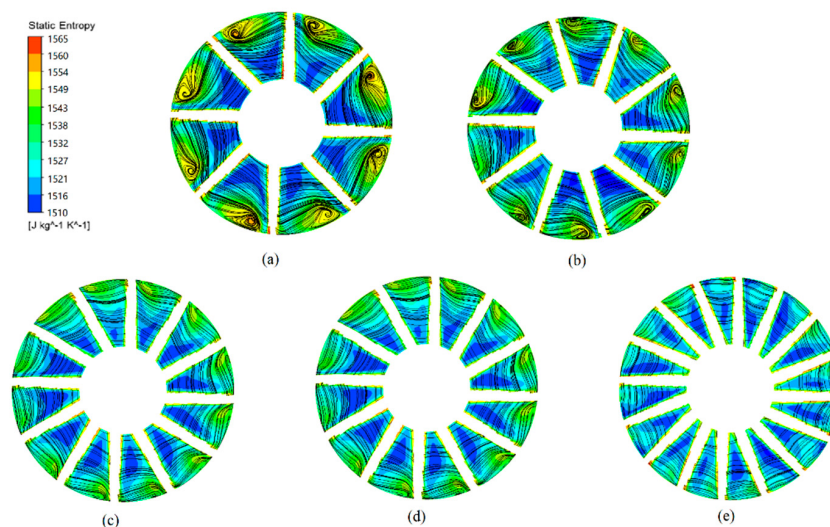


Figure 14. Entropy distribution at outlet with surface streamlines of air: (a) solidity 7.96; (b) solidity 9.95; (c) solidity 11.94; (d) solidity 13.93; (e) solidity 15.92.

4.2. Rotor Blade Load and Aerodynamic Force

The power capability of the radial-inflow turbine depends on rotor blade load and it can be further studied by rotor blade pressure distributions. Figure 15 presents the pressure distribution at the mid-span and its partially enlarged view, i.e., the rotor blade load along the streamwise location of different solidity and working fluid. As Figure 15a indicates, when solidity is higher than 11.94, the pressure below critical point appears. The trans-critical phenomenon arises at the trailing edge of the blade. In both Figure 15a,b, when solidity is 7.96, the pressure difference between suction side and pressure side is the largest. Hence, the blade can obtain the highest power capacity when the solidity is the lowest. Additionally, the capacity of the blade concentrates on the front and middle of the streamwise location, especially for solidity 15.92, which has little power output at the latter half of the blade. This phenomenon derives from the diminishing pitch of the blade along the flow direction. Moreover, for both SCO_2 and air, the solidity mainly affects the pressure distribution on the suction side and the pressure on the pressure side is of minor variation.

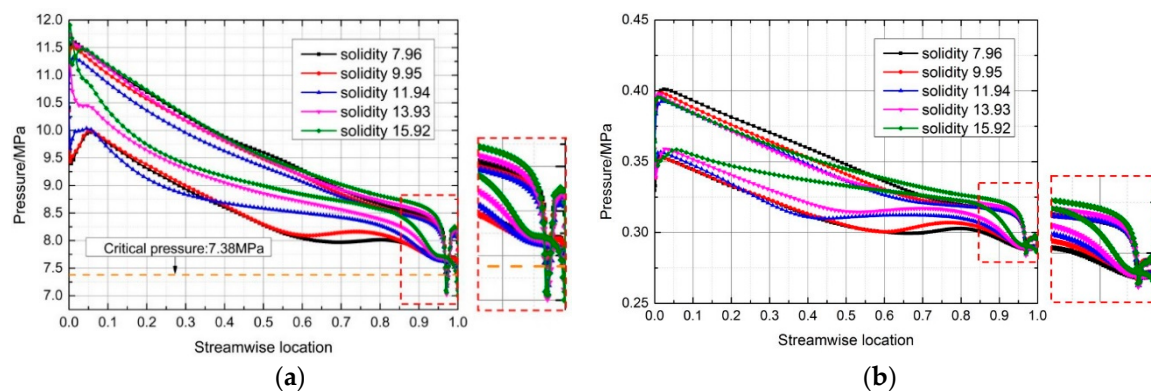


Figure 15. Rotor blade load along streamwise location of different solidity: (a) SCO_2 ; (b) air.

The axial force of a single rotor blade with different solidity and working fluid are presented in Figure 16. The direction of the force is shown by the red arrow. The axial force of the rotor is not beneficial to the safe operation and requires high-level balance method of axial thrust as well as a reliable bearing. From Figure 16, we can conclude that with the solidity increasing, in a SCO_2 radial-inflow turbine, the axial force of a single rotor blade increases. However, with the working fluid air, the axial force first increases and when the solidity is greater than 13.93, it slightly decreases. In general, the axial thrust of the rotor increases considering the rotor blade number. The effect of rotor solidity on axial force is greater for SCO_2 turbine than it is for air turbine. The high density of SCO_2 working fluid causes much higher axial thrust. With the augmented axial force, the safe operation of the turbine is affected and the load of the bearing enlarges.

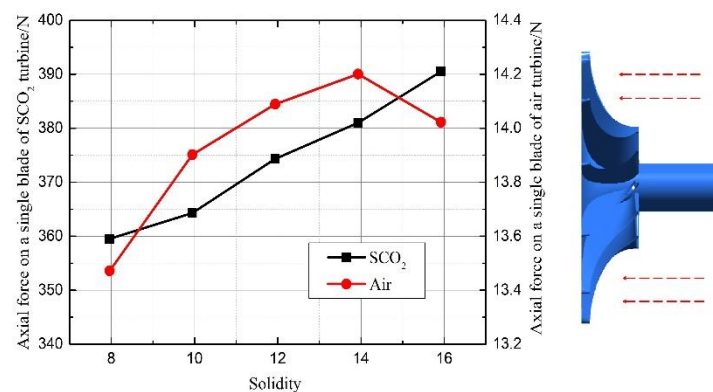


Figure 16. Axial force of a single rotor blade with different solidity and working fluid.

4.3. Aerodynamic Performance

Output power, isentropic efficiency and mass flow rate are vital parameters in the assessment of the turbine aerodynamic performance. The isentropic efficiency discussed in this research is defined as Equation (8):

$$\eta_{is} = \frac{2T_z \times 2\pi \times r}{60 \times (\dot{m}_{in} + \dot{m}_{out}) \Delta h_{is}} \quad (8)$$

where T stands for torque, r represents the rotation speed, h is enthalpy and $\dot{m}$ is mass flow rate. As for the subscripts, is stands for isentropic, in and out represent the inlet and outlet of the turbine respectively, z corresponds to axial direction. The mass flow rate of inlet and outlet are slightly different due to the random error in the numerical simulation. To be precise, we adopt the average to calculate the efficiency.

High solidity usually results in large weight and axial thrust of the wheel. The working fluid SCO_2 has more special application scenarios (such as aerospace and warships) than air due to its small size. In these cases, the weight of the wheel is limited. The problems of overlarge axial thrust and trans-critical phenomenon need to be solved. Hence, we concentrate on the improvement of SCO_2 turbine. Three new splitters structures with and without offsetting are introduced. All solidity structures are calculated and compared in this section to test the effectiveness in the SCO_2 radial-inflow turbine. Figure 17 gives the variation of output power and isentropic efficiency with different solidity.

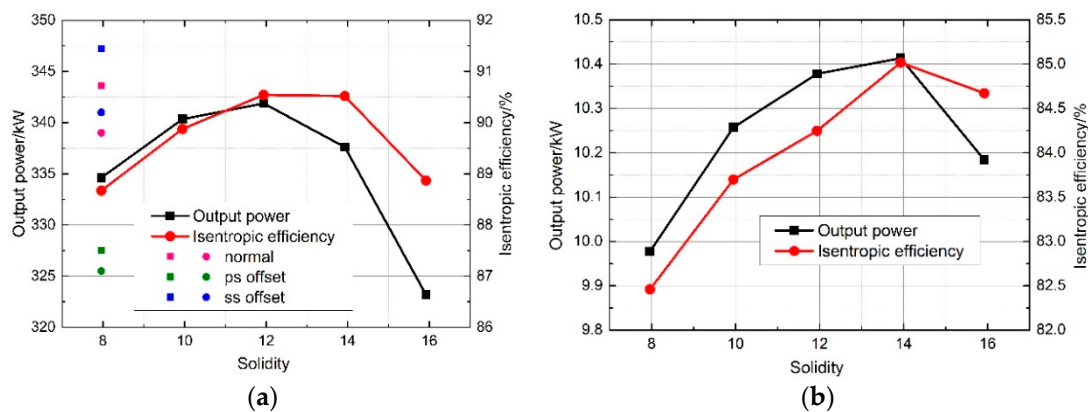


Figure 17. Output power and isentropic efficiency of different solidity and splitters: (a) SCO_2 ; (b) air.

For both SCO_2 and air, the output power of the turbine first increases and then decreases with the augment of solidity. For SCO_2 , the maximum output power is 341.87 kW and it is realized when solidity is 11.94. While in Figure 17b, the maximum output power is 10.41 kW and it happens when solidity is 13.93 for the air turbine. Comparing (a) and (b), the radial-inflow turbine working with air has apparently lower power capacity, which results from a lower power density. With the augment of solidity, the output power descends more rapidly after the maximum in SCO_2 turbine than that in air turbine. The decline of output power is owing to the blocking effect of turbine outlet when solidity reaches a certain extent. Obviously, it has a greater influence in a SCO_2 turbine due to the fluid property similar to liquid.

As for the isentropic efficiency, the tendency is similar for SCO_2 and air. In general, the isentropic efficiency first increases and then decreases with the augment of solidity and the efficiency of SCO_2 turbine is higher in this power scale. In a SCO_2 turbine, when solidity is 11.94, the isentropic efficiency reaches the maximum 90.54%. When solidity is 13.93, the efficiency is 90.52%. Although the highest solidity corresponds to the lowest output power, its efficiency is acceptable due to the lowest mass flow rate. The solidity 7.96 has the minimum efficiency owing to the unsatisfactory flow condition at the inlet of impeller, which is caused by the large pitch at leading edge. In the air turbine, the highest efficiency is 85.02% when solidity is 13.93. Likewise, the isentropic efficiency is the lowest 82.46% when

solidity is 7.96. The efficiency declines slower than that in SCO₂ turbine after the solidity reaches the maximum. The blocking effect is smaller and the large pitch at the leading edge affects the efficiency in a large scale.

The ss offset splitter can increase the output power and efficiency to the greatest extent. The power is increased to 347.22 kW, and the efficiency is increased to 90.18%. The normal splitter can improve the performance to a small extent while the ps offset splitter is ineffective. Overall, in the preliminary design of a SCO₂ radial-inflow turbine, the best solidity is around 12 and for an air turbine, it is around 14. If the weight, axial thrust and trans-critical problem is concerned, arranging the ss offset splitter is a choice to improve the flow condition and the aerodynamic performance of the low solidity case.

5. Conclusions

In this research, an automatic design and simulation process of the radial-inflow turbine is established in MATLAB. The design reference of rotor solidity for both SCO₂ and air are provided. A new splitter structure to improve the aerodynamic performance of low solidity case is proposed. Concretely:

1. For both SCO₂ and air, with the increase of rotor solidity, the working capacity for each blade declines. The cross section of hub has the highest power ability. Comparing five kinds of solidity, the pressure along spanwise tends to be more uniform as the solidity decreases. Small rotor solidity can result in the chaotic flow at the inlet of impeller, which further leads to the nonuniform pressure distribution. Large solidity disturbs the flow at the outlet and the blocking flow increases the trans-critical area in SCO₂ turbine. Comparing two working fluids, the air turbine has no trans-critical area and the low-pressure region at the suction side of the rotor blade trailing edge vanishes.
2. The aerodynamic loss of the SCO₂ and air turbine mainly concentrates on the hub and shroud area due to the boundary layer effect of the impeller. For SCO₂, when solidity is 7.96, the highest static entropy locates at the shroud due to the vacant flow passage. When solidity is 15.92, the highest static entropy locates at the hub on account of the limited flow area. For air, the increase of solidity less affects the aerodynamic loss and the entropy production mainly concentrates on the shroud.
3. For both SCO₂ and air, the solidity mainly affects the pressure distribution on the suction side and the pressure on the pressure side is of minor variation. In the SCO₂ turbine, large solidity corresponds to lower power capacity and greater trans-critical area of each blade. The high density of SCO₂ working fluid causes much higher axial thrust than air.
4. For both SCO₂ and air, the output power and isentropic efficiency of the turbine first increases and then decreases with the augment of solidity. In the SCO₂ turbine, the isentropic efficiency reaches the maximum 90.54% and the maximum output power is 341.87 kW when solidity is 11.94. While in the air turbine, the maximum isentropic efficiency is 85.02% and the maximum output power is 10.41 kW when solidity is 13.93. In the preliminary design of a SCO₂ radial-inflow turbine, the best solidity is around 12 and for an air turbine, it is around 14. Arranging the ss offset splitter can improve the flow condition and the aerodynamic performance of the low solidity case.

Author Contributions: Conceptualization: Y.W. and D.Z.; Methodology: D.Z.; Software: Y.X.; Validation: Y.W. and J.L.; Formal analysis: Y.W.; Investigation: Y.W. and J.L.; Resources: Y.X.; Data curation: D.Z.; Writing—original draft preparation: Y.W.; Writing—review and editing: J.L.; Visualization: J.L.; Supervision: D.Z.; Project administration: Y.X.; funding acquisition: Y.X. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by 111 project, grant number B16038.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Garg, P.; Kumar, P.; Srinivasan, K. Supercritical carbon dioxide Brayton cycle for concentrated solar power. *J. Supercrit. Fluids* **2013**, *76*, 54–60. [\[CrossRef\]](#)
2. Chen, M.-F.; Yamaguchi, H.; Zhang, X.-W.; Niu, X.-D. Performance analyses of a particularly designed turbine for a supercritical CO₂-based solar Rankine cycle system. *Int. J. Energy Res.* **2015**, *39*, 1819–1827. [\[CrossRef\]](#)
3. Osorio, J.D.; Hovsapien, R.; Ordonez, J.C. Dynamic analysis of concentrated solar supercritical CO₂-based power generation closed-loop cycle. *Appl. Therm. Eng.* **2016**, *93*, 920–934. [\[CrossRef\]](#)
4. Chacartegui, R.; de Escalona, J.M.M.; Sanchez, D.; Monje, B.; Sanchez, T. Alternative cycles based on carbon dioxide for central receiver solar power plants. *Appl. Therm. Eng.* **2011**, *31*, 872–879. [\[CrossRef\]](#)
5. Valdes, M.; Abbas, R.; Rovira, A.; Martin-Aragon, J. Thermal efficiency of direct, inverse and sCO₂ gas turbine cycles intended for small power plants. *Energy* **2016**, *100*, 66–72. [\[CrossRef\]](#)
6. Dostal, V.; Hejzlar, P.; Driscoll, M.J. High-performance supercritical carbon dioxide cycle for next-generation nuclear reactors. *Nucl. Technol.* **2006**, *154*, 265–282. [\[CrossRef\]](#)
7. Ahn, Y.; Bae, S.J.; Kim, M.; Cho, S.K.; Baik, S.; Lee, J.I.; Cha, J.E. Review of supercritical CO₂ power cycle technology and current status of research and development. *Nucl. Eng. Technol.* **2015**, *47*, 647–661. [\[CrossRef\]](#)
8. Mecheri, M.; Le Moullec, Y. Supercritical CO₂ Brayton cycles for coal-fired power plants. *Energy* **2016**, *103*, 758–771. [\[CrossRef\]](#)
9. Monje, B.; Sanchez, D.; Savill, M.; Pilidis, P.; Sanchez, T.; ASME. *A Design Strategy for Supercritical CO₂ Compressors*; America Society Mechanical Engineers: New York, NY, USA, 2014.
10. Zhang, H.; Zhao, H.; Deng, Q.; Feng, Z. *Aerothermodynamic Design and Numerical Investigation of Supercritical Carbon Dioxide Turbine*; America Society Mechanical Engineers: New York, NY, USA, 2015.
11. Zhang, D.; Wang, Y.; Xie, Y. Investigation into Off-Design Performance of a S-CO₂ Turbine Based on Concentrated Solar Power. *Energies* **2018**, *11*, 3014. [\[CrossRef\]](#)
12. Baltadjiev, N.D.; Lettieri, C.; Spakovszky, Z.S. An Investigation of Real Gas Effects in Supercritical CO₂ Centrifugal Compressors. *J. Turbomach. Trans. ASME* **2015**, *137*. [\[CrossRef\]](#)
13. Conboy, T.; Wright, S.; Pasch, J.; Fleming, D.; Rochau, G.; Fuller, R. Performance Characteristics of an operating Supercritical CO₂ Brayton Cycle. *J. Eng. Gas Turbines Power* **2012**, *134*, 941–952. [\[CrossRef\]](#)
14. Lettieri, C.; Baltadjiev, N.; Casey, M.; Spakovszky, Z. Low-Flow-Coefficient Centrifugal Compressor Design for Supercritical CO₂. *J. Turbomach. Trans. ASME* **2014**, *136*. [\[CrossRef\]](#)
15. Lopez, A.; Monje, B.; Sanchez, D.; Chacartegui, R.; Sanchez, T. *Effect of Turbulence and Flow Distortion on the Performance of Conical Diffusers Operating on Supercritical Carbon Dioxide*; America Society Mechanical Engineers: New York, NY, USA, 2013.
16. Noughabi, A.K.; Sammak, S. Detailed Design and Aerodynamic Performance Analysis of a Radial-Inflow Turbine. *Appl. Sci.* **2018**, *8*, 2207. [\[CrossRef\]](#)
17. Deng, Q.; Shao, S.; Fu, L.; Luan, H.; Feng, Z. An Integrated Design and Optimization Approach for Radial Inflow Turbines-Part II: Multidisciplinary Optimization Design. *Appl. Sci.* **2018**, *8*, 2030. [\[CrossRef\]](#)
18. Zweifel, O. The Spacing of Turbo-Machine Blading, Especially with Large Angular Deflection. *Brown Boveri Rev.* **1945**, *32*, 436–444.
19. Horlock, J.H. *Axial Flow Turbines*; Butterworth-Heinemann: Oxford, UK, 1966.
20. Chen, T.Y.; Liao, Y.T.; Cheng, C.C. Development of small wind turbines for moving vehicles: Effects of flanged diffusers on rotor performance. *Exp. Therm. Fluid Sci.* **2012**, *42*, 136–142. [\[CrossRef\]](#)
21. Mohamed, M.H. Impacts of solidity and hybrid system in small wind turbines performance. *Energy* **2013**, *57*, 495–504. [\[CrossRef\]](#)
22. Eboibi, O.; Danao, L.A.M.; Howell, R.J. Experimental investigation of the influence of solidity on the performance and flow field aerodynamics of vertical axis wind turbines at low Reynolds numbers. *Renew. Energy* **2016**, *92*, 474–483. [\[CrossRef\]](#)
23. Gao, K.; Xie, Y.; Zhang, D. Effects of rotor solidity and leakage flow on the unsteady flow in axial turbine. *Appl. Therm. Eng.* **2018**, *128*, 926–939. [\[CrossRef\]](#)
24. Simpson, A.T.; Spence, S.W.T.; Watterson, J.K. Numerical and Experimental Study of the Performance Effects of Varying Vaneless Space and Vane Solidity in Radial Turbine Stators. *J. Turbomach. Trans. ASME* **2013**, *135*. [\[CrossRef\]](#)

25. Pereiras, B.; Takao, M.; Garcia, F.; Castro, F. *Influence of the Guide Vanes Solidity on the Performance of a Radial Impulse Turbine with Pitch-Controlled Guide Vanes*; America Society Mechanical Engineers: New York, NY, USA, 2011.
26. Dong, B.S.; Xu, G.Q.; Li, T.T.; Quan, Y.K.; Zhai, L.J.; Wen, J. Numerical prediction of velocity coefficient for a radial-inflow turbine stator using R123 as working fluid. *Appl. Therm. Eng.* **2018**, *130*, 1256–1265. [[CrossRef](#)]
27. Ji, G.H. *Turbo-Expander*, 1st ed.; China Machine Press: Beijing, China, 1982.
28. Brennen, C.E. *Hydrodynamics of Pumps*; Cambridge University Press: Cambridge, UK, 2011.
29. Deng, Q.; Shao, S.; Fu, L.; Luan, H.; Feng, Z. An Integrated Design and Optimization Approach for Radial Inflow Turbines-Part I: Automated Preliminary Design. *Appl. Sci.* **2018**, *8*, 2038. [[CrossRef](#)]
30. Menter, F.R. 2-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA J.* **1994**, *32*, 1598–1605. [[CrossRef](#)]
31. Roache, P.J. Perspective—A Method for Uniform Reporting of Grid Refinement Studies. *J. Fluids Eng. -Trans. ASME* **1994**, *116*, 405–413. [[CrossRef](#)]
32. Paudel, S.; Saenger, N. Grid refinement study for three dimensional CFD model involving incompressible free surface flow and rotating object. *Comput. Fluids* **2017**, *143*, 134–140. [[CrossRef](#)]
33. Kim, S.G.; Lee, J.; Ahn, Y.; Lee, J.I.; Addad, Y.; Ko, B. CFD investigation of a centrifugal compressor derived from pump technology for supercritical carbon dioxide as a working fluid. *J. Supercrit. Fluids* **2014**, *86*, 160–171. [[CrossRef](#)]
34. Odabae, M.; Sauret, E.; Hooman, K. CFD Simulation of a Supercritical Carbon Dioxide Radial-Inflow Turbine, Comparing the Results of Using Real Gas Equation of State and Real Gas Property File. *Appl. Mech. Mater.* **2016**, *846*, 85–90. [[CrossRef](#)]
35. Ameli, A.; Uusitalo, A.; Turunen-Saaresti, T.; Backman, J. Numerical Sensitivity Analysis for Supercritical CO₂ Radial Turbine Performance and Flow Field. In Proceedings of the 4th International Seminar on Orc Power Systems, Milan, Italy, 13–15 September 2017; Volume 129, pp. 1117–1124.



© 2020 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<http://creativecommons.org/licenses/by/4.0/>).