NUMERICAL MODELING OF SUPERCRITICAL CARBON DIOXIDE FLOW IN SEE-THROUGH LABYRINTH SEALS

Haomin Yuan, Sandeep Pidaparti, Mathew Wolf, John Edlebeck, Mark Anderson
University of Wisconsin - Madison
1500 Engineering Drive, Madison, WI, 53706
hyuan8@wisc.edu

ABSTRACT

This paper presents a numerical study of Supercritical Carbon Dioxide ($s\text{CO}_2$) flow in see-through labyrinth seals. The Computational Fluid Dynamic (CFD) simulation of this scenario is performed under the framework of OpenFOAM. Properties of $s\text{CO}_2$ are implemented into OpenFOAM with a user-defined interface. A test facility was constructed to measure the leakage rate and pressure drop of $s\text{CO}_2$ in see-through labyrinth seals. Various designs and conditions have been tested to study the flow characteristic and provide validation data for numerical model. The primary goal is to verify the model's capability to predict leakage rate, with a secondary goal focused on using the code to optimize the seal design for $s\text{CO}_2$. This research concludes with some guidelines for the see-through labyrinth seal optimization.

Keyword: OpenFOAM, Supercritical CO_2 , see-through labyrinth seal

Nomenclature

Latin Symbols

c	clearance between shaft and seal, mm
C_p	specific heat, J/kg-K
D	shaft diameter, mm
h	specific enthalpy, J/kg
H	height of cavity, mm
k	turbulent kinetic energy, J/kg
L_{cavity}	length of cavity, mm
L_{total}	total length of seal, mm
L_{tooth}	length of tooth, mm
P	pressure, MPa
Pr_t	turbulent Prandtl number,
T	temperature, $^{\circ}\text{C}$
U	velocity (vector), m/s

Greek Symbols

α	thermal diffusivity, m^2/s , or void fraction
ε	turbulent kinetic energy dissipation rate, J/s
K	thermal conductivity, $\text{W}/(\text{m}\cdot\text{K})$
v	specific volume, m^3/kg
μ	dynamic viscosity, $(\text{N}\cdot\text{s})/\text{m}^2$
ρ	density, kg/m^3

Subscripts

effective combination of thermal and turbulence properties

g	gas
inlet	condition at inlet
l	liquid
outlet	condition at outlet
thermal	thermal property
turbulence	turbulence property

Abbreviations

CFD	Computational Fluid Dynamic
CO ₂	Carbon Dioxide
HEM	Homogeneous Equilibrium Model
RANS	Reynolds Average Navier Stokes equation
sCO ₂	Supercritical Carbon Dioxide
SNL	Sandia National Laboratory
UW-Madison	University of Wisconsin at Madison

1. INTRODUCTION

Recent researches [1]–[3] show that with Supercritical Carbon Dioxide (sCO₂) Brayton Cycle can achieve high efficiency (~45%) by utilizing the unique properties change near the critical point with relative low turbine inlet temperature. Besides that, as fluid is in a supercritical state, two-phase condition is avoided in major components, thus simplify their designs and reduce total capital cost, as shown in Figure 1 and Figure 2. However, the unique fluid behavior of sCO₂ near its critical and pseudo-critical points challenges its numerical modeling. As a result, an accurate property database and a robust algorithm are needed.

Figure 1 Schematic of sCO₂ Simple Brayton cycle

Figure 2 T-S diagram of sCO₂ Simple Brayton cycle

Figure 3 SNL sCO₂ compressor and labyrinth seal [3]

Figure 4 Notations of see-through labyrinth seal

Labyrinth seal is widely used in turbomachinery. And it is also used as a part of more advanced seals, such as the dry gas seal. In this research, labyrinth seal is used as an example to demonstrate the capability of the proposed methodology. And, at the same time, provide some guidelines for the design of labyrinth seals for sCO₂. In the Sandia National Laboratory (SNL) sCO₂ Brayton cycle experiment loop [3], a stepped labyrinth seal is adopted for its compressor as shown in Figure 3. However, in this research, we focus on the see-through labyrinth seal, which does not have steps on its shaft. The stepped labyrinth seal will be studied in our future work. For convenience, we refer labyrinth seal for see-through labyrinth seal in the rest of this paper.

Figure 4 shows the notations used for a see-through labyrinth seal. The fluid of sCO₂ enters the seal from the left and leaks out to the right. To fully define a see-through labyrinth seal, we need the six parameters in Figure 4 plus the tooth number. However, an inherent correlation $L_{total} = nL_{tooth} + (n-1)L_{cavity}$ exists for L_{total} , L_{tooth} , L_{cavity} , and tooth number, n . These notations are used for the rest of this paper.

The idea of using labyrinth seals for turbomachinery has been around for a long time. From Sneek's [4] review paper, Parsons [5], [6] first introduced labyrinth seals. The idea was to introduce a tortuous flow path between high and low pressure regions by means of a series of non-contacting restrictors and separating cavities [4]. By this method, the pressure head is converted into kinetic energy and dissipated by the recirculation in these cavities. After Parsons's original designs, various researchers have performed studies on labyrinth seals from different perspectives.

In the pioneering paper by Martin [7], labyrinth seals were considered to be a series of discrete throttling processes. Egli [8] made a modification of Martin's labyrinth seal model by introducing a kinetic energy carry-over coefficient, which is determined empirically in his work. The carry-over coefficient represents the portion of kinetic energy carried from one cavity to the next. Egli also noticed that since the pressure drop across each restrictor increases, the last restrictor is the first to reach the choked flow pressure ratio. Hodkinson [9] assumed a conically shaped stream in each cavity from a fluid mechanical point of view. The expansion angle of this conically shaped stream determines the carry-over coefficient in Egli's model.

Eldin [10] conducted the most recent and significant work regarding to labyrinth seals. His research used air as the working fluid. Some of his conclusions are observed in this research. He mentioned that increasing the tooth length (L_{tooth}) in Figure 4 reduces leakage by the largest amount. The effect of cavity height (H) in Figure 4 was minor compared tooth length although there does exist an optimum cavity height which results in a minimum leakage rate. Increased eccentricity leads to increased leakage rate, but this effect is only significant at low Reynolds number. In this research, the Reynolds number is much larger than the work of Eldin, therefore, the issue of eccentricity does not influence leakage rate. Moreover, in our previous research by Wolf [11], there is no significant difference in leakage rate for maximum and minimum eccentricity. As a result, we exclude the study of eccentricity in this paper.

Regarding CFD simulations of labyrinth seals, Wittig [12], Rhode [13], and Schramm [14] did extensive works in this area. They used CFD tools to perform optimization for different kinds of labyrinth seals, including staggered, stepped, and see-through. However, they used ideal gas in their simulation. Suryanarayanan [15], also used ideal gas, on the other hand presented a new model to calculate the carry-over coefficient in Hodkinson's model [9]. A

recent study by Jiang [16] presents a numerical study on vane, gland, and shaft seals for supercritical water. They used finite element method to solve fluid dynamic equations to find the value of rotordynamic coefficients for these seals. In their research, they focused on the steam forcing induced by the leakage, and the effect from different parameters like the tooth number, seal clearance, etc. However, leakage rate prediction and two phase capability were not mentioned in their paper.

In our previous research [17], we used the proposed methodology to simulate sCO₂ flow through circular and annular orifices. And the numerical data show a very good agreement with experimental results. In this paper, the same method was used on labyrinth seals to provide more credential. The open source CFD code OpenFOAM was utilized to simulate this scenario. As sCO₂ flows through a labyrinth seal, two-phase scenario may appear. In order to handle this situation, we used the homogeneous equilibrium model (HEM) to simulate two-phase flow. In our previous study [17], the applicability of HEM to a similar flow condition has been discussed. As a result, this discussion will be skipped here. A test facility was constructed at UW-Madison to provide validation data for numerical modeling. More information about this test facility can be found in [11], [18].

2. NUMERICAL METHODOLOGY

This section introduces the numerical procedure used in this research. We start with fundamental Navier-Stokes equations and solver algorithm. Then the property module of sCO₂ and two-phase model are introduced. After that, numerical domain and boundary conditions are discussed. Finally, this section ends with a discussion of turbulence modeling and meshing.

2.1 Solver Algorithm

The *rhoSimplecFoam* solver in OpenFOAM is used as the prototype for the modified solver. This solver uses the SIMPLEC[19] algorithm to solve the Navier-Stokes equations for steady-state single-phase compressible flow. As HEM is used, these equations also govern the flow in this problem and can be seen in Equation (1) (2) (3).

Continuity equation:

$$\nabla \cdot (\rho \cdot U) = 0 \quad (1)$$

Momentum equation:

$$\nabla \cdot (\rho \cdot U \cdot U) - \nabla \cdot (\mu_{\text{effective}} \cdot \nabla U) = -\nabla P \quad (2)$$

Energy equation:

$$\nabla \cdot (\rho U h) + \nabla \cdot \left(\rho U \frac{U^2}{2} \right) - \nabla \cdot (\alpha_{\text{turbulence}} \nabla h) = \nabla \cdot (k \nabla T) \quad (3)$$

Where

$$\mu_{\text{effective}} = \mu_{\text{thermal}} + \mu_{\text{turbulence}} \quad (4)$$

2.2 sCO₂ Properties Module

The property change of sCO₂ around its pseudo-critical point at 8 MPa is shown in Figure 5. As we can see, there is a huge density and specific heat variation. The variation gets more dramatic as pressure is closer to the critical point. The variation of properties challenges numerical simulation causing numerical exemption easily. This requires we reducing the outlet pressure gradually to its target value and using relaxation factors in each iteration.

Figure 5 Property change of CO₂ at pseudo-critical point at 8 MPa

Initially, REFPROP [20] database was used to provide CO₂ properties. However, we found that using REFPROP results running simulation extremely slow. Alternatively, the fluid properties are calculated using FIT [21] software library which provides an interpolated representation of the equation of state presented by Span and Wagner [22]. However, the underlying property data of FIT is still obtained from REFPROP. In this way FIT generates fluid properties much faster than REFPROP within acceptable errors. Figure 6 shows the comparison of FIT and REFPROP property calculation for the specific heat of sCO₂. The data in Figure 6 is based on 1 million points, distributed evenly from 240 K to 1500 K and from 1 kPa to 100 MPa. As we can see, the maximum relative difference is less than 1e-5 and even less than the uncertainty of equation of state. Other properties' comparisons show the same degree of agreement.

Figure 6 Comparisons of FIT and REFPROP for Cp of CO₂

2.3 Homogeneous Equilibrium Model

In HEM, all phases are assumed to have the same velocity, temperature and pressure [23]. With the HEM assumption, the state of two-phase can be determined by pressure and enthalpy uniquely as shown in Equation (5) and (6).

$$v_{two-phase} = x \cdot v_{gas} + (1-x) \cdot v_{liquid} \quad (5)$$

$$h_{two-phase} = x \cdot h_{gas} + (1-x) \cdot h_{liquid} \quad (6)$$

For transport properties (e.g., viscosity and conductivity), the average value for a two-phase state can be based on area, mass or volume. There is no evidence to show that which one is supreme better than the others. Therefore, in the current research, we obtain two-phase transport properties based on the mass average using equation (7) and (8). However, as the Re number is around 200,000, which makes turbulent viscosity and diffusivity much larger than their molecular counterparts. As a result, different averaging methods for transport properties do not affect simulation a lot.

$$\mu_{two-phase} = x \cdot \mu_{gas} + (1-x) \cdot \mu_{liquid} \quad (7)$$

$$\kappa_{two-phase} = x \cdot \kappa_{gas} + (1-x) \cdot \kappa_{liquid} \quad (8)$$

2.4 Geometry and boundary condition

Since labyrinth seal geometry is axisymmetric, 2D axisymmetric geometry method was used to save computational time. However, Navier-Stokes equations in OpenFOAM are designed for Cartesian system [24], [25]. Therefore, the OpenFOAM community uses a pseudo-axisymmetric method instead [24], [25]. The computational domain of a

two-tooth labyrinth seal is shown in Figure 7 and Figure 8. However, these figures are not created in scale. A two-tooth seal labyrinth has the same dimensions in Figure 7 is also tested in experiment for validation.

The boundary conditions used in simulation are also presented in Figure 7 and Figure 8. The values of pressure are fixed at inlet and outlet. As velocity is driven by pressure gradient, zero-gradient condition is used at inlet and outlet for velocity in flow direction. In order to eliminate the influence of boundary conditions on inlet and outlet, the numerical domain has large volumes before and after the inspected seal, which are not represented properly in Figure 7 as it is not created in scale. The 'wall' boundary condition means non-slip and non-penetration for velocity, and zero-gradient in wall-normal direction for pressure. Due to the adiabatic assumption in this process, we also assume zero heat flux at wall, which applies zero-gradient of temperature. This is achieved by the combination of zero-gradient for both pressure and enthalpy as shown in equation (11). The 'wedge' boundary condition is recommended the OpenFOAM User Guide [24] for pseudo-axisymmetric geometry. It assumes the adjacent cell is the cell itself. The 'wedge' boundary condition is actually a specialized periodic boundary condition for pseudo-axisymmetric geometry.

$$\frac{\partial T}{\partial y} = \frac{\partial T}{\partial P} \frac{\partial P}{\partial y} + \frac{\partial T}{\partial h} \frac{\partial h}{\partial y} \quad (9)$$

Figure 7 Computational domain of a two-tooth labyrinth seal (front view)

Figure 8 Computational domain of a two-tooth labyrinth seal (axial view)

2.5 Turbulence Model and Meshing

The solver is designed for RANS turbulence model. He [26] and Yoo [27] discussed the choice of turbulence models for CFD simulation for supercritical fluid. However their discussion is based on relatively low Re situations, where heat transfer is one of the major concerns. In this research, all walls are assumed to be adiabatic. And, only leakage rate through seals are measured in experiment and extracted from simulation. Moreover, the Re number is about 200,000, which is much larger than the situation discussed in [26], [27]. It is theoretically unclear what turbulence model should be used for this problem. Therefore, the standard k-epsilon model [28] which is designed for high Re situation is apparently our best choice. In our previous study [17], the k-omega SST model was also used, and the predicted mass flow rate for a similar flow condition shows no difference compare with k-epsilon model. Different turbulent Pr numbers are also tested, and do not predict different mass flow rate.

In standard k-epsilon model:

$$\frac{D\rho k}{Dt} = \nabla \cdot \left(\left(\frac{\mu_t}{\sigma_k} + \mu \right) \nabla k \right) + \rho P_k - \frac{2}{3} \rho k \nabla \cdot \mathbf{U} - \rho \varepsilon \quad (10)$$

$$\frac{D\rho \varepsilon}{Dt} = \nabla \cdot \left(\left(\frac{\mu_t}{\sigma_\varepsilon} + \mu \right) \nabla \varepsilon \right) + C_{\varepsilon 1} \frac{\rho P_k \varepsilon}{k} - \left(\frac{2}{3} C_{\varepsilon 1} + C_{\varepsilon 3} \right) \rho \varepsilon \nabla \cdot \mathbf{U} - C_{\varepsilon 2} \frac{\rho \varepsilon^2}{k} \quad (11)$$

$$\mu_{turbulence} = C_\mu \frac{\rho k^2}{\varepsilon} \quad (12)$$

$$\alpha_{turbulence} = \frac{\mu_{turbulence}}{Pr_t} \quad (13)$$

Where

Table I Coefficients for standard k-epsilon model in OpenFOAM

C_μ	$C_{\epsilon 1}$	$C_{\epsilon 2}$	$C_{\epsilon 3}$	σ_k	σ_ϵ	Pr_t
0.09	1.44	1.92	-0.33	1.0	1.3	1.0

Grid study has also been perform in our previous study [17] for a similar flow condition. In the standard k- ϵ model, the wall function requires the boundary cells to be in the log-arithmetic layer. As a consequence, the boundary cells has y^+ around 30-50 for all simulated cases.

3. RESULTS AND DISCUSSION

In this section, numerical data are presented and discussed. First, a two-tooth labyrinth seal is used to validate numerical model. Leakage rate predictions by the proposed model are compared with experimental data. Then some details of the flow, including mass fraction distribution, pressure distribution, and stream line plot, are also presented and discussed. After that, several parametric studies are performed to study the characteristic of sCO₂ flow through labyrinth seal. The selected parameters are the radial clearance, cavity length, cavity height, and tooth number. Some parametric studies are presented with both experimental and numerical data. Others are discussed only with numerical data. The total length is not chosen, as the total length is usually determined by the available space in a turbomachinery component.

3.1 Validation with experimental data

Experiments of a two-tooth labyrinth seal that has the same dimensions in Figure 7 were conducted to provide validation data. Two upstream conditions were tested in the experiment, they are (10 MPa, 325 kg/m³) and (10 MPa, 475 kg/m³), however, only one set (10 MPa, 475 kg/m³) of data are presented here in Figure 9. In Figure 9, we fixed the upstream condition and gradually reduced the downstream pressure. The leakage rate increases initially but remain constant as pressure ratio is lower than choking pressure ratio. The other set of data shows the similar agreement. Figure 10 shows the trajectory line of discharge for data in Figure 9. As we can see, when the downstream condition is inside the two-phase domain, two-phase scenario appears. That is the reason HEM was implemented to handle two-phase scenario. Figure 11 shows the mass fraction distribution at the choked outlet pressure of 6 MPa. In Figure 11, mass fraction is defined as one for supercritical state. Figure 12 presents the pressure distribution, and Figure 13 shows the stream line plot. As can seen in Figure 11, flashing occurs at the second tooth, where the pressure drops into two-phase region in Figure 12. In Figure 13, recirculation in cavity and expansion/contraction of main stream are observed. This flow structure is expected for the see-through labyrinth seal. We will discuss more detail of the flow structure when discussing parametric study results.

From the discussion above, we can see that the simulation prediction matches experimental data in a very good agreement. Through this observation, we can gain sufficient confidence that the proposed numerical method is capable to predict labyrinth seal leakage rate.

Figure 9 Two-tooth labyrinth seal experiment and simulation comparison for upstream condition of (10 MPa, 475 kg/m³),

Figure 10 Trajectory line of discharge for two teeth labyrinth seal

Figure 11 Mass fraction distribution at choked outlet pressure (6 MPa)

Figure 12 Pressure distribution at choked outlet pressure (6 MPa)

Figure 13 Stream line plot at choked outlet pressure (6 MPa)

3.2 Radial clearance

A three-tooth labyrinth seal is used to investigate the effect of radial clearance on the leakage rate. The dimensions for this seal are shown in Figure 14. The data in Figure 15 shows that, the leakage rate increases with increase in radial clearance. This is a fairly obvious result as an increase in clearance area allows more fluid to be forced underneath the tooth. It should be noted that the inlet operating condition is fixed at (10 MPa, 498 kg/m³). The outlet

pressure is 5 MPa. Although the presented result is for three teeth seals, this conclusion should also be valid for any tooth number and different operating conditions as well. However, from manufacture and engineering perspectives, the value of radial clearance does not have too much freedom. It is constrained by manufacture precision, shaft oscillation, etc. As a result, combined with our previous discussion, radial clearance should be as small as possible.

Figure 14 Dimension of labyrinth seal for radial clearance parametric study

Figure 15 Effect of radial clearance on mass flow rate

3.3 Cavity length

The two-tooth labyrinth seal geometry is used to investigate the effect of cavity length. The cavity length is changed to investigate the response of the leakage rate. During which, the total length is kept constant, which means when increasing cavity length, the tooth length is reduced accordingly. In this study the cavity height is fixed to be 0.88 mm, while the cavity length is sampled evenly between 1.27 mm to 3 mm. Figure 16 shows the dimensions used in

parametric study. All the data points in Figure 17 have the same inlet condition of (9 MPa, 498 kg/m³) and the same outlet pressure of 5 MPa. As can be seen, the leakage rate decreases as the cavity length increase.

As discussed previously, after the main stream enters the cavity, it will spread as a small angle (called expansion angle by Hodkinson [9]), and then contract again when it meets the next tooth, as shown in Figure 13. During this process, the main stream interacts with the recirculation inside the cavity, and loses part of its kinetic energy. When the main stream contracts at the next tooth, there is a contraction pressure loss. This pressure loss is proportional to the main stream area change and contributes as a major pressure loss in labyrinth seals. The expansion angle was assumed to be constant by Hodkinson [9]. However, Suryanarayanan [15] disagreed with this assumption. In Figure 18, the streamline plot from numerical simulation indicated that an increased cavity length results in an increased main stream area change at the next tooth, which produces an increased pressure loss and a decreased leakage. This phenomenon is also observed for air in Eldin's work [10]. From the above discussion, we can conclude that labyrinth seals should have large cavity length. However, increasing the cavity length reduces the space of teeth, when the total length is limited. The limitation of tooth length comes from the mechanical and manufacture perspective, as each tooth should be able to hold the pressure difference across it and the length of tooth is available and economical in manufacture.

Figure 16 Dimension of labyrinth seal for cavity length parametric study

Figure 17 Mass flow rate changes with cavity length

Figure 18 Flow patterns of different cavity lengths

3.4 Cavity height

In order to inspect the effect of cavity height on the leakage rate, the cavity length is fixed and the cavity height is varied. Figure 19 presents the dimensions in simulation. The data in Figure 20 is based on the simulations of the two-tooth labyrinth seal geometry in previous subsection. All the data points in Figure 20 have the same inlet condition of (9 MPa, 498 kg/m³), and the outlet pressure of 5 MPa. Figure 20 concludes that there is an optimum point for the cavity height that results in a minimum leakage rate. This phenomenon is also observed in Eldin [10]'s experiment. Another set of data with different cavity length also shows the existence of an optimum cavity height. Further investigation reveals that this observation can also be explained by the main stream contraction in the cavity. The streamline plot shows that when the cavity height is too small, it limits the expansion of the main stream. When the cavity height is large enough, then the main stream is fully developed. However, there exists a medium value of cavity height that results in a maximum main stream expansion, which leads to a maximum contraction pressure loss and minimum leakage. Figure 21 is obtained from numerical results; and shows the trend in previous discussion. Combine the findings from previous and this section, after cavity length, tooth length, and tooth number are determined, the cavity height can be optimized to bring a few percent improvement.

Figure 19 Dimension of labyrinth seal for cavity height parametric study

Figure 20 Mass flow rate changes with cavity heights

Figure 21 Flow pattern in different cavity heights

3.5 Tooth Number

In this section, the tooth number is varied. In order to compare designs for different tooth number, the total length is fixed. From previous section we know that the larger cavity length leads to lower leakage. This means the minimum tooth length results in the lowest leakage rate. As a result, in this part, we are using a fixed tooth length for all teeth and assuming this is the minimum tooth length which is allowable in manufacture. By inserting more teeth into the seal, the leakage rate initially decreases. However, after a certain number of teeth are inserted, the leakage rate increases. When inserting more teeth, the position of each tooth is tuned to make all teeth evenly distributed. It seems there is an optimum tooth number leads to a minimum leakage rate. Figure 22 shows the design for different teeth number with the same total length. The designs in Figure 22 have the same total length of 11.43 mm, and the same tooth length of 1.27 mm. Figure 23 shows the leakage rate for these designs at a fixed upstream condition of (10 MPa, 325 kg/m³) and downstream pressure of 5 MPa. As we can see in Figure 23, the design with three teeth produces the minimum leakage rate. We have also preformed the experiments for 2, 3, and 5 teeth seals under the same condition to provide validation data. Another set of data from upstream condition of (10 MPa, 475 kg/m³) presents the same conclusion. This observation can be explained as following. From the previous sections, we know that the increasing the cavity length reduces leakage. In this section, as the teeth are inserted into the seal, the cavity length decreases. As a result, at a certain point, inserting more teeth cannot bring more benefits to decrease leakage as cavity length is too small. The above discussion assumes we utilize all the available space for seals. And all teeth have a minimum thickness that is manufacture allowable.

Figure 22 Labyrinth seals with different teeth numbers

Figure 23 Leakage rate for different teeth number

4. CONCLUSION

This research studied the $s\text{CO}_2$ flow in the see-through labyrinth seals using numerical simulation. The test facility constructed at UW-Madison was used to validate the numerical data and provide database for future research. We used OpenFOAM to perform numerical simulation, with a user-defined property module for $s\text{CO}_2$ properties.

Experimental and numerical results show a good agreement for a two-tooth labyrinth seal at two upstream conditions. This comparison validates the numerical methodology for this scenario. Four geometry parameters namely, radial clearance, cavity length, cavity height, and tooth number are chosen to inspect their effects on the leakage rate. Increasing the radial clearance increased the leakage rate, while increasing the cavity length reduced the leakage rate. However for the cavity height, if other parameters are fixed, there is a value results in a minimum leakage. The same holds true for the effect of tooth number while holding the total length fixed. As a result, in order to have a minimum leakage rate for a see-through labyrinth seal, we should have a minimum tooth length, a optimized tooth number, and if possible a optimized cavity height. Some of these conclusions are also observed in the research of Eldin [10] using air. However, the final optimization designs and the corresponding leakage rates would be different for air and $s\text{CO}_2$.

ACKNOWLEDGEMENT

This study was supported by the U.S Department of Energy Nuclear Energy University Program (No. 11-1625-Technical Development for $s\text{CO}_2$ Advance Energy Conversion). The authors would like to thank for their support for this research.

REFERENCE

- [1] V. Dostal, "A Supercritical Carbon Dioxide Cycle for Next Generation Nuclear Reactors," Doctoral dissertation, Massachusetts Institute of Technology, 2004.
- [2] V. Dostal, P. Hejzlar, and M. J. Driscoll, "The Supercritical Carbon Dioxide Power Cycle : Comparison to Other Advanced Power Cycles," *Nuclear technology*, vol. 154, no. 1, pp. 283–301, 2006.

- [3] S. A. Wright, R. F. Radel, M. E. Vernon, G. E. Rochau, and P. S. Pickard, "Operation and Analysis of a Supercritical CO₂ Brayton Cycle," Albuquerque, New Mexico and Livermore, California, USA, 2010.
- [4] H. J. Sneek, "Labyrinth Seal Literature Survey," *Journal of Lubrication Technology*, vol. 96, no. 4, pp. 579–581, 1974.
- [5] C. Parsons, "The Labyrinth Packing," *Engineer*, 1938.
- [6] C. Parsons, "STANLEY SMITH COOK," *US Patent 1,374,520*, 1921.
- [7] H. Martin, "Labyrinth packings," *Engineering*, 1908.
- [8] A. Egli, "The Leakage of Steam Through Labyrinth Seals," *TRANS.ASME*, vol. 57, pp. 115–122, 1935.
- [9] B. B. Hodgkinson and A. M. E. Mech, "ESTIMATION OF THE LEAKAGE THROUGH A LABYRINTH GLAND," in *Proceeding of the Institution of Mechanical Engineers*, 1939.
- [10] A. M. Eldin, "LEAKAGE AND ROTORDYNAMIC EFFECTS OF POCKET DAMPER SEALS AND SEE-THROUGH LABYRINTH SEALS LEAKAGE AND ROTORDYNAMIC EFFECTS OF POCKET DAMPER SEALS AND SEE-THROUGH LABYRINTH SEALS," Texas A&M University, 2007.
- [11] M. P. Wolf, "Flow of Supercritical Carbon Dioxide Through Annuli And Labyrinth Seals," Master Thesis, University of Wisconsin-madison, 2014.
- [12] S. Wittig, U. Schelling, S. Kim, and K. Jacobsen, "Numerical predictions and measurements of discharge coefficients in labyrinth seals," ... *Gas Turbine Conference and Exhibition*, 32 ..., 1987.
- [13] D. Rhode, S. Ko, and G. Morrison, "Leakage optimization of labyrinth seals using a Navier-Stokes code," *Tribology transactions*, 1994.
- [14] V. Schramm, J. Denecke, S. Kim, and S. Wittig, "Shape Optimization of a Labyrinth Seal Applying the Simulated Annealing Method," *The International Journal of Rotating Machinery*, vol. 10, no. 5, pp. 365–371, Jan. 2004.
- [15] S. Suryanarayanan, "LABYRINTH SEAL LEAKAGE EQUATION," Texas A&M University, 2009.
- [16] P. N. Jiang, W. Z. Wang, Y. Z. Liu, and G. Meng, "Influence of steam leakage through vane, gland, and shaft seals on rotordynamics of high-pressure rotor of a 1,000 MW ultra-supercritical steam turbine," *Archive of Applied Mechanics*, vol. 82, no. 2, pp. 177–189, May 2011.
- [17] H. Yuan, J. Edlebeck, M. Wolf, M. Anderson, M. Corradini, S. Klein, and G. Nellis, "Simulation of Supercritical CO₂ Flow through Circular and Annular Orifice," *Journal of Nuclear Engineering and Radiation Science*, 2014.
- [18] J. P. Edlebeck, "Measurements And Modeling of The Flow of Supercritical Carbon Dioxide," Master Thesis, University of Wisconsin-madison, 2013.
- [19] J. P. Van Doormaal and G. D. Raithby, "Enhancements of The Simple Method for Predicting Incompressible Fluid Flows," *Numerical Heat Transfer*, vol. 7, no. 2, pp. 147–163, Apr. 1984.
- [20] E. W. Lemmon, M. L. Huber, and M. O. McLinden, "NIST Reference Fluid Thermodynamic and Transport Properties-REFPROP," Boulder, Colorado, 2007.

- [21] Northland Numerics, "FIT," 2012. [Online]. Available: <http://www.northlandnumerics.com/>.
- [22] R. SPAN and W. WAGNER, "A NEW EQUATION OF STATE FOR CARBON DIOXIDE COVERING THE FLUID REGION FROM THE TRIPLE-POINT TEMPERATURE TO 1100 K AT PRESSURES UP TO 800 MPa," *JOURNAL OF PHYSICAL AND CHEMICAL REFERENCE*, vol. 25, no. 6, pp. 1509–1596, 1996.
- [23] M. L. Corraini, "Fundamentals of Multiphase Flow," 1997. [Online]. Available: <http://wins.engr.wisc.edu/teaching/mpfBook/>. [Accessed: 04-Jul-2013].
- [24] OpenCFD Limited, "OpenFOAM User Guide," 2011. [Online]. Available: <http://www.openfoam.org/docs/user/>.
- [25] OpenFOAM Foundation, "Open FOAM Programmer's Guide," 2011. [Online]. Available: <http://www.foamcf.org/Nabla/guides/ProgrammersGuide.html>.
- [26] S. He, W. S. Kim, and J. H. Bae, "Assessment of performance of turbulence models in predicting supercritical pressure heat transfer in a vertical tube," *International Journal of Heat and Mass Transfer*, vol. 51, no. 19–20, pp. 4659–4675, Sep. 2008.
- [27] J. Y. Yoo, "The Turbulent Flows of Supercritical Fluids with Heat Transfer," *Annual Review of Fluid Mechanics*, vol. 45, no. 1, pp. 495–525, Jan. 2013.
- [28] W. Jones and B. Launder, "The prediction of laminarization with a two-equation model of turbulence," *International journal of Heat Mass Transfer*, vol. 15, no. 2, pp. 301–304, 1972.

Haomin Yuan is currently a graduate student major in Nuclear Engineering at University of Wisconsin-madison. He got his bachelor degree in Tsinghua University, Beijing, also in Nuclear Engineering. In his current study, he focuses on CFD simulation of supercritical carbon dioxide flow in labyrinth seals and valves.