Dynamic Reed Valve in Rolling Piston Compressor: A 3-Dimensional Transient CFD Simulation

Nirav Chaudhari[1], Chinmoy K Mohapatra[2*], Hui Ding[3], Haiyang Gao[3]

[1]Simerics Inc.,
Bangalore, KA, IN
[2]Simerics India Ltd.,
Novi, MI, USA
248-513-3200
[3]Simerics Inc.,
Bellevue, WA, USA
425-502-9978
* Corresponding Author

ABSTRACT

This study focuses on a 3-D transient Computational Fluid Dynamics (CFD) model for a rolling piston compressor with a dynamic reed valve, a crucial element in refrigeration and air-conditioning systems known for its energy-intensive operation. The growing popularity of rolling piston compressors stems from their favorable characteristics, including silent and smooth operation, as well as high reliability and efficiency. Efficient CFD modeling of such compressors necessitates a thorough understanding of the flow dynamics within the suction chamber, compression chamber, and the thin leakage volume. This modeling must be coupled with considerations for the rotating piston and the dynamic behavior of the reed valve, which plays a pivotal role in discharging compressed fluid at prescribed pressures. Analyzing the discharge valve system is crucial for addressing over-compression loss and reliability issues. The presented work employs a detailed 3D transient CFD model, utilizing the Simerics-MP+ internal rolling piston template to generate a mesh for the entire system. The template incorporates a rotational dynamics module to accurately capture the movement and deformation of the mesh. Additionally, a rotational ordinary differential equation, based on cantilever beam theory, is formulated to account for the bending motion of the valve reed near the discharge port. A test simulation of a generic rolling compressor, incorporating a discharger reed valve, demonstrates the robustness, speed, and user-friendly nature of the algorithms and implementations. The findings suggest the potential applicability of these methods to rolling piston compressor systems, effectively capturing various physical phenomena.

1. INTRODUCTION

Positive displacement (PD) compressors compress fluids by mechanically changing the volume of the working chamber, trapping and squeezing the fluid until a designated pressure is reached. Positive displacement compressors can be reciprocating or rotary, depending on the motion of the displacing elements. Reciprocating compressors use pistons sliding in a linear motion, providing excellent sealing and adaptability to varying working conditions (Stouffs, 2001). Rotary compressors, with rotors in circular or near-circular motion, tend to have fewer components, simplifying the design and reducing the weight-to-displacement ratio (Spark,2019). Due to their efficiency, versatility, and relatively simple design, PD compressors are popular in various industries, particularly in refrigeration systems, where rolling piston compressors are commonly used for their compact size, affordability, and high performance.

Valves play a crucial role in PD compressors by regulating fluid flow during the suction and discharge phases. Unlike reciprocating compressors, rotary compressors typically don't require suction valves because the rotor separates the inlet and outlet ports. However, the discharge valve in rotary compressors is vital for maintaining pressure and preventing backflow. Reed valves, a type of check valve, are commonly used in PD compressors to control fluid flow direction. Made from thin metal or plastic layers, they open and close in response to fluid pressure, enabling the correct timing of fluid flow during the compression cycle. Studies have shown that reed valves can exhibit fluid-structure interaction (FSI), leading to phenomena like valve flutter (intense vibrations during operation) which has been observed by various researchers (Bhkta,2012, Nagy,2008 and Burgstaller, 2008). To ensure reliability and reduce pressure pulsations, research by Huang and Xie (Huang,2008) highlighted the importance of proper retainer design, while Yu et al. (Yu, 2018) examined the effects of torsional movement on valve operation.

Experimental measurements in compressor systems typically yield macroscopic data like discharge pressure, temperature, enthalpy differences, and work done. However, finer details, such as leakage from gaps in compressor components, sliding blade motions, reed valve responses, and internal flow patterns, are challenging to capture with traditional experimental methods. This is where Computational Fluid Dynamics (CFD) becomes invaluable, as it allows for detailed analysis of flow structures within the compression chamber and aids in investigating critical parameters affecting compressor efficiency, such as compression volume, inlet angle, reed valve dynamics, and leakage (Liang,2010, Ba, 2016).

CFD can precisely simulate cooling capacity and input power, providing insights into the intricate dynamics of compressor-valve interactions, a critical factor in compressor performance and efficiency. The dynamic motion of valves can influence noise levels and durability, presenting challenges in design due to the complex coupling between compressor and valve behaviors. CFD simulations offer a pathway to understanding these interactions and resolving issues like pressure pulsation and leakage, which are key to optimizing compressor design. By simulating both compressor and valve systems, engineers can gain a comprehensive view of the factors driving efficiency, reliability, and durability in these critical components.

The evolution of CFD simulations for rolling piston compressors spans several decades, with notable advancements in modeling accuracy and complexity. Early studies, such as those by Lenz and Cooksey (Lenz,1994) focused on optimizing discharge port geometry using steady-state CFD solutions. Subsequent research (Geng,2004, Liang,2010) introduced more sophisticated transient moving mesh approaches to improve accuracy. However, these models simplified reed valve behavior, overlooking complex valve dynamics under fluid forces. To address this limitation, researchers like Machu et al. (Machu,2004) employed finite element methods to model detailed valve deformation, while Silva et al. (Silva,2012) utilized ordinary differential equations (ODEs) within a 1D fluid model. Kinjo et al. (Kinjo,2010) and Prasad (Prasad,2004) highlighted the challenges of incorporating moving meshes and valve dynamics into CFD simulations, emphasizing the difficulty of accurately modeling these components. Tan et al. (Tan,2014) further investigated the effects of effective flow and force areas of reed valves on rotary compressor performance using fluid-structure interaction (FSI) models, while Brancher and Deschamps (Brancher,2014) developed a prediction method for effective flow and force area coefficients in rolling piston compressors using ANSYS CFX. These advancements underscore the ongoing efforts to enhance the fidelity and predictive capabilities of CFD simulations for compressor systems, addressing challenges such as valve dynamics and leakage flow between chambers.

The current research presents a 3D transient CFD model of a rolling piston compressor with a discharge reed valve. Using Simerics-MP+'s rolling piston template, the model automates setup and mesh generation, applying a moving mesh approach to represent the compressor's fluid volume. Reed valve bending is simulated with a simple cantilever beam theory, while the surrounding fluid volume and clearance between the port and the valve are modeled with moving and deforming meshes, respectively. The valve's motion is driven by a 1D rotational ordinary differential equation (ODE), capturing the fluid-structure interaction (FSI) as the valve bends and adjusts during the strokes. This detailed model aims to improve understanding of the compressor's internal dynamics and the FSI effects of the reed valve.

2. ROLLING PISTON MODEL

2.1 Working Principle

A rolling piston, depicted in Figure 1, features a cylindrical rotor mounted eccentrically on a driving shaft within a concentric cylinder chamber. With a moving vane, the space between the rotor and chamber is split into suction and compression chambers. The contact points between the vane and rotor, and between the rotor and chamber wall, define the boundaries separating these two chambers.

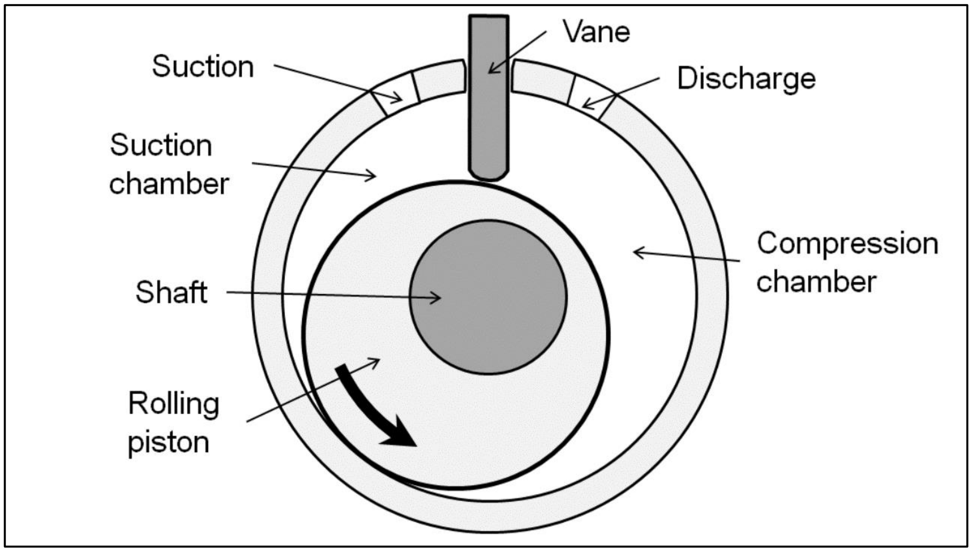


Figure 1: Schematic of a rolling piston compressor

In operation, as the contact point between the piston and the chamber wall rotates past the inlet port and before the discharge valve opens, the compression chamber becomes fully sealed. As the piston continues to rotate, the chamber's volume decreases, compressing the trapped gas. Once the pressure in the compression chamber reaches the discharge pressure, the discharge valve opens, allowing the compressed gas to escape. Figures 2.1 through 2.5 illustrate the entire compression cycle of a rolling piston compressor.

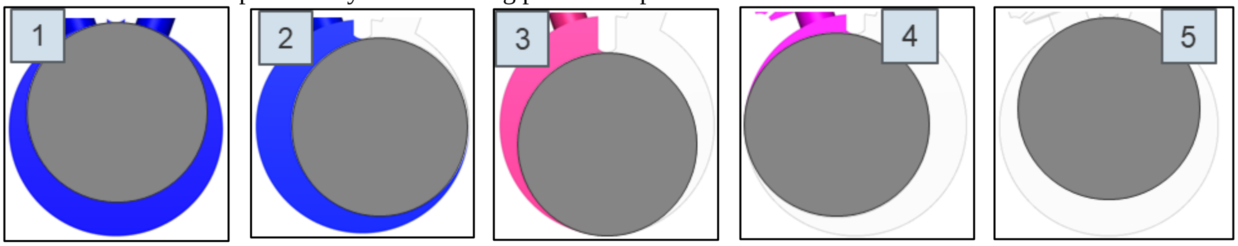


Figure 2: Rolling piston compression process

2.2 Mesh Details

The rolling piston fluid domain has two key parts: the main chamber and the fluid gap at the vane tip (Figure 3). The main chamber, which excludes the vane and its tip gap, occupies the space between two eccentric circles. Its shape is determined by the crankshaft angle and filled with a continuous structured mesh, including the narrow leakage gap near the contact point between the piston and the chamber wall (Figure 4). The fluid gap at the vane tip is meshed separately to model leakage flow through this gap (Figure 5), though this part can be optional if leakage is not a concern.

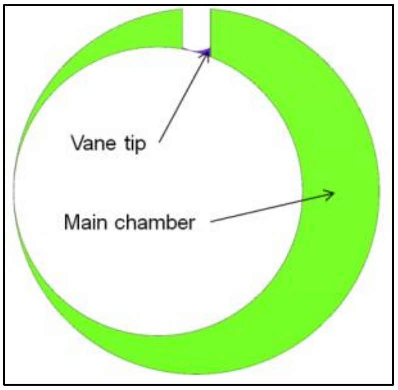


Figure 3: Rolling piston fluid domain

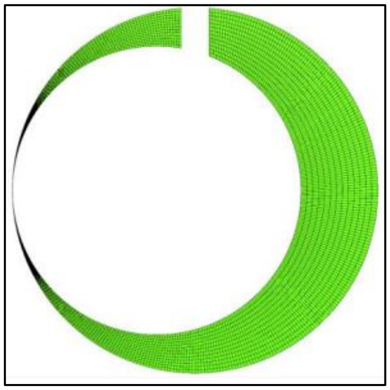


Figure 4: Main chamber mesh

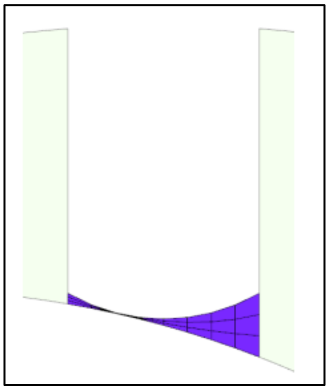


Figure 5: Vane tip mesh

The vane tip volume is bounded by two surfaces, the vane tip, and a section of the rotor cylinder. Its position is set by checking the contact condition with the rotor at its current location, adding a predefined clearance for leakage. Once set, a structured mesh is applied to this volume (Figure 5). The meshes for both the main chamber and the vane tip are then linked using the Mis-Matched Grid Interface (MGI) function in Simerics-MP+. This feature allows for seamless integration between different mesh types, providing a complete fluid domain for the rotor section.

The Rolling Piston template in Simerics-MP+ automates the mesh creation process explained above, based on user-defined rotor, chamber, and vane tip geometry. The template also controls rotor movement according to user-defined operating conditions, generating a new mesh at each time step for transient simulations.

3. REED VALVE MODEL

3.1 Working Principle

Figure 6 depicts a typical reed valve, comprising a valve reed, a valve seat, and a backing plate. The valve reed is a thin, flexible sheet of metal or composite material, clamped at one end with a bolt, while the other end is free to move. This design allows the reed to bend and open under fluid pressure to permit flow, then close against the valve seat to stop reverse flow when the pressure changes. The backing plate limits how much the reed valve can bend.

In this paper, two approaches to valve modeling are discussed. The first approach models valve movement as rotation around its clamped end, while the second approach calculates valve movement based on the assumption that the valve reed is a cantilevered beam.

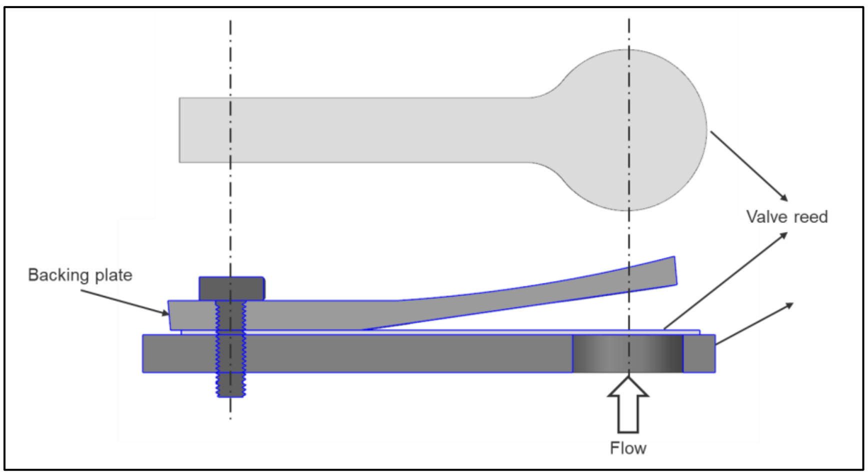


Figure 6: Reed valve image

3.2 Valve Dynamics

The first approach approximates the real valve bending compared with a 1D linear valve motion model, without coupling to a complex strain-stress analysis. As depicted in Figure 8**,** this approach assumes the angular position of the valve to be similar to the bending of the valve at the centerline of the flow channel. The detailed explanation on this simplified approach are included in (Ding, 2014).

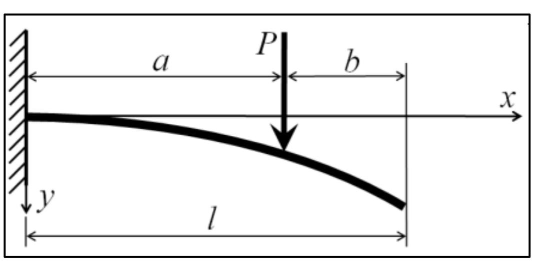


Figure 7: Bending of cantilever beam

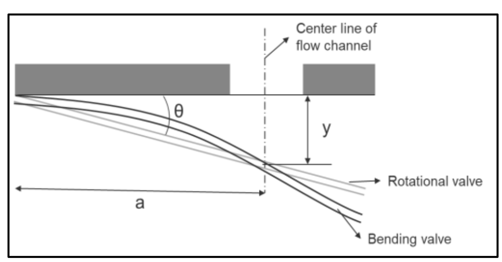


Figure 8: Rotation vs Bending

However, this paper focuses on the second approach, where the actual bending of the valve is modeled based on the forces acting upon it. In this method, the valve reed is assumed to move in only one direction: the direction of fluid flow through the channel. Consider a cantilever beam (Figure 7) with a length l and a concentrated load P at a point located at a distance a from the fixed end. Using a concentrated load is a suitable assumption for a discharge reed valve when it's slightly open. This is because the primary fluid force is exerted on the valve by high-pressure fluid within a relatively small flow channel connected to the discharge valve. The deflection y at any location x can be described as:

$$y = \frac{Px^2}{6EI}(3a - x) \quad for\ \ 0 < x < a \tag{1a}$$

$$y = \frac{Pa^2}{6EI}(3x - a) \quad for\ \ a < x < l \tag{1b}$$

Where, E is the modulus of elasticity, and I is the area moment of inertia. This deflection profile is applied to valve reed to bend the valve based on the force P which is a function of the rotor position.

3.3 Mesh Details

Two fluid volumes shaped like the reed valve are isolated from the outlet fluid volume, one above and one below the valve. The moving mesh algorithm is then applied to these volumes to simulate the bending of the reed valve. During valve closure, the mesh on the valve seat side compresses while the mesh on the opposite side stretches; conversely, during valve opening, the mesh on the valve seat side stretches while the mesh on the opposite side compresses. Figure 9 illustrates a comparison of mesh deformation between two modeling approaches at various valve opening positions in a simplified model.

(a).

(b).

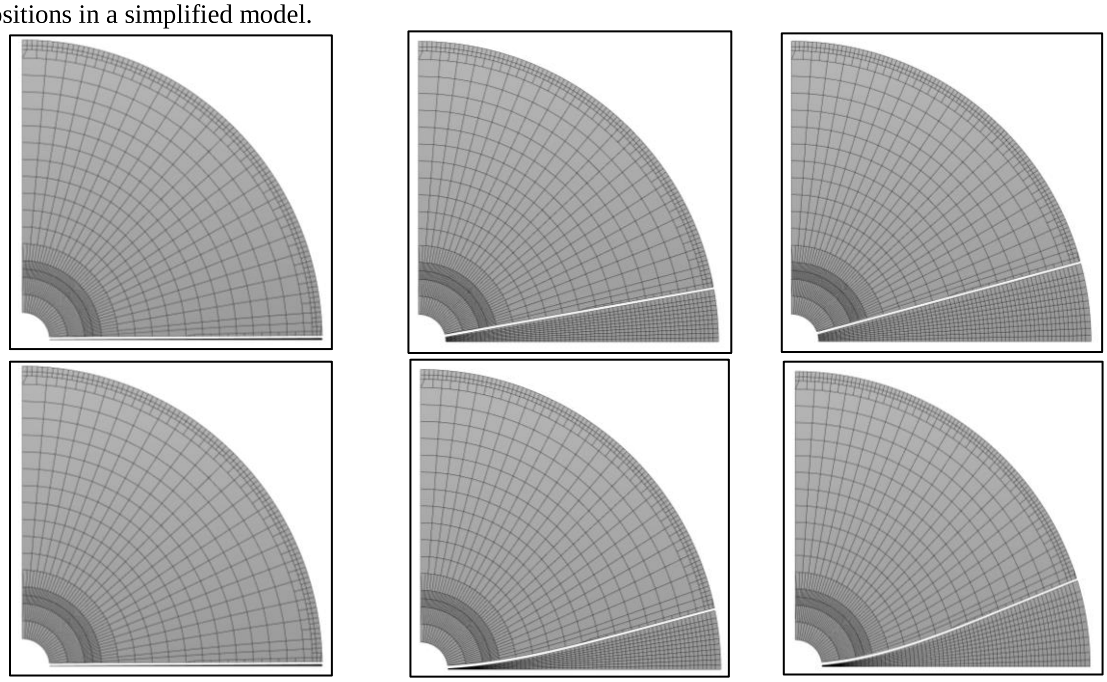


Figure 9: Comparison between mesh deformation in two modelling approaches at different valve opening positions. Row-a: Rotating valve, Row-b: Bending valve

4. CFD MODEL DETAILS

The CFD solver in Simerics-MP+ uses a finite volume approach to solve conservation equations for mass, momentum, and energy in a compressible fluid (Ding, 2014). The solver also incorporates the standard k-ε two-equation model (Launder and Spalding, 1974) to address turbulence. Fluid properties are modeled using an equation of state, with detailed governing equations provided by Ding et al. (Ding, 2014). This framework includes a closure model to simulate valve closure. Unlike other closure models (Mohapatra, 2017), Simerics-MP+ creates artificial walls in the computational domain to block fluid flow when the valve reaches its minimum position. The current solver implementation has been validated for a rolling piston compressor by Hsu et al. (Hsu, 2020). Simerics-MP+ has been used in CFD simulations for various compressor types, including centrifugal compressors, lobe compressors, twin screw compressors (Kovacevic, 2014), and scroll compressors (Gao and Jiang, 2014).

5. ROLLING PISTON COMPRESSOR WITH REED VALVE TEST CASE

The proposed approach is demonstrated using a generic rolling piston model, comprising a suction port, rotor, and discharge port with a reed valve connecting the rotor to the discharge port. Aside from the rotor and valve volumes, all other fluid volumes are meshed using a binary tree unstructured mesh. The different fluid volumes are connected using a Mismatched Grid Interface (MGI), which allows seamless integration between non-conformal meshed regions. The total mesh count is approximately 0.46 million cells. Figure 10 shows the fully meshed fluid domain, while Figure 11 depicts the mesh for the reed valve on a cross-section plane. This model also includes three adjustable leakage channels: at the vane tip, at the contact point between the piston and the chamber wall, and in a narrow gap between the valve reed and the valve seat.

The inlet and outlet boundaries are maintained at constant pressures and temperatures for the simulation, with air used as the fluid, modeled according to the ideal gas law. This provides a realistic depiction of fluid behavior under changing pressure and temperature conditions. The rotational speed of the compressor is set at 3600 RPM.

Starting with a well-prepared CAD model, the entire process of meshing and setting up the simulation is efficient, taking less than an hour using the Rolling Piston and Circumferential Valve templates. These specialized templates simplify the setup with pre-configured settings optimized for rolling piston machines. The simulation takes approximately 40 minutes to simulate one complete revolution on a standard PC equipped with a quad-core Intel Xeon CPU operating at 2.30 GHz. This quick simulation time allows for rapid iteration and testing of different configurations, aiding in the development and optimization of the system.

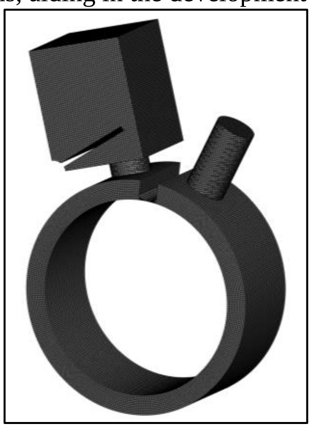


Figure 10: Mesh for all fluid volumes

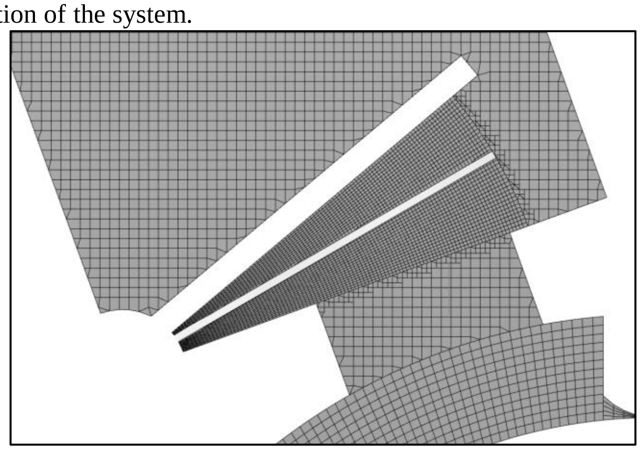
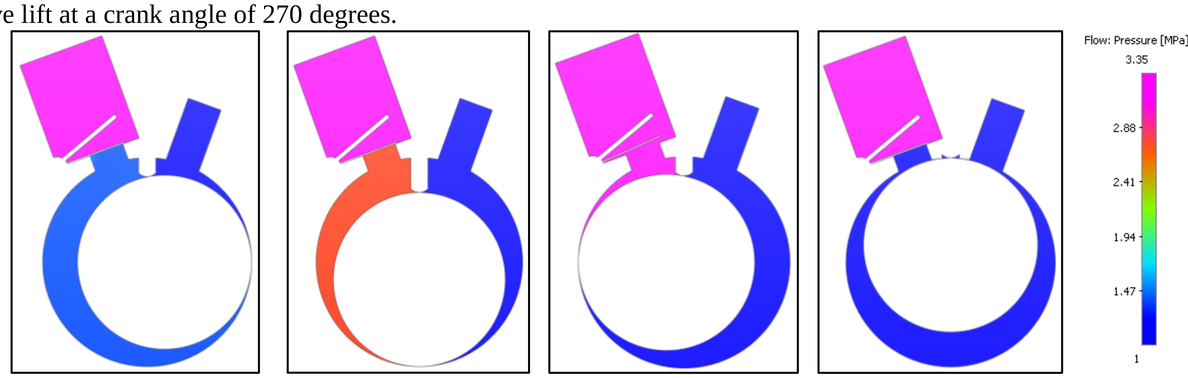


Figure 11: Mesh for reed valve

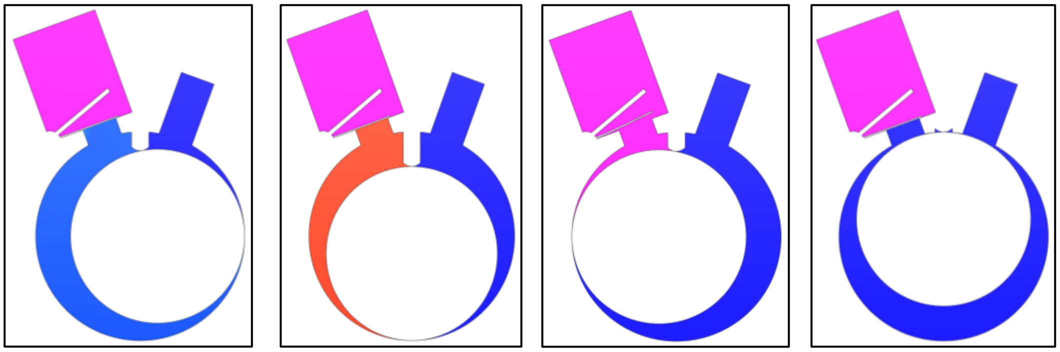
6. RESULTS AND DISCUSSION

To compare the two approaches, the inlet pressure was set to 1 MPa and the outlet pressure to 3.35 MPa, with 400 time steps per revolution. Both simulations reached a periodic state after three revolutions. Figure 12 presents a comparison of pressure contours for the two approaches at various crank angle positions. The results from both approaches are similar, with the primary difference being the valve lift. Figure 13 clearly illustrates the differences in valve lift at a crank angle of 270 degrees.

a

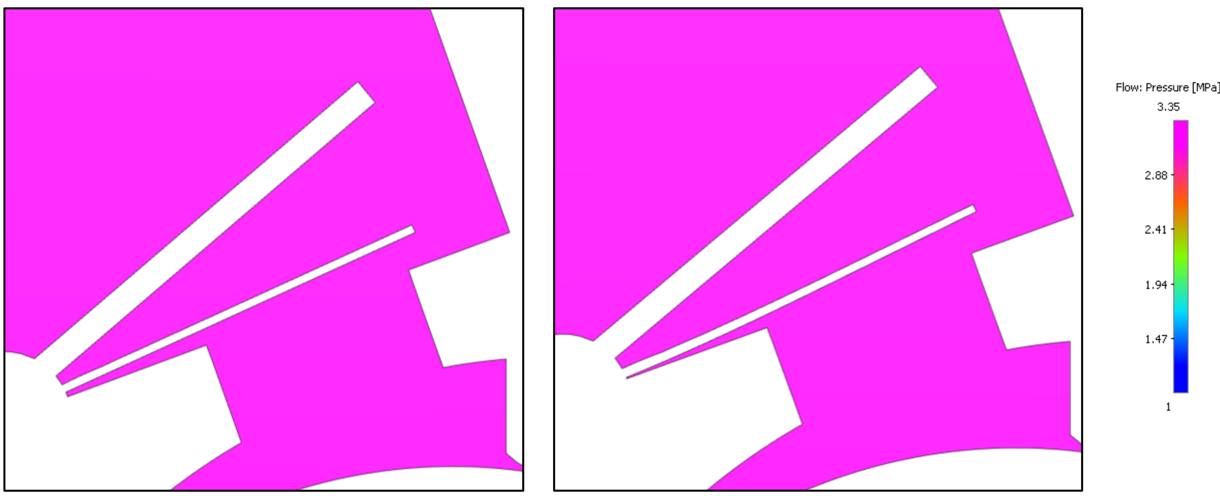


b



90 Deg 180 Deg 270 Deg 360 Deg

Figure 12: Comparison between pressure contour at different crank angle position for two different approaches, Row-a: Rotating valve, Row-b: Bending valve



1. Rotating valve
2. Bending valve

Figure 13: Comparison between valve lift for both approaches at crank angle of 270-Degree

Although both approaches produce comparable overall outcomes, they diverge in their initial valve lift profiles, as illustrated in Figure 14, which shows the valve lift at the valve tip relative to the compressor's crank angle. Initially, both methods exhibit a rapid valve lift; however, the bending approach demonstrates a slightly faster and higher lift, with a maximum lift difference of 10% (0.22 mm). Following this initial phase, both valves return to the closed position at a similar rate. The earlier lift observed in the bending approach influences the flow dynamics through the port. Figure 15 displays a plot of the mass flow rate from the port relative to the compressor's crank angle. In the bending valve approach, the mass flow rate increases more rapidly earlier compared to the rotating valve approach. Additionally, the overall mass flow rate from the port is 12% higher in the bending valve approach. The mass flow rate plot also indicates a marginally earlier opening of the valve with the bending method.

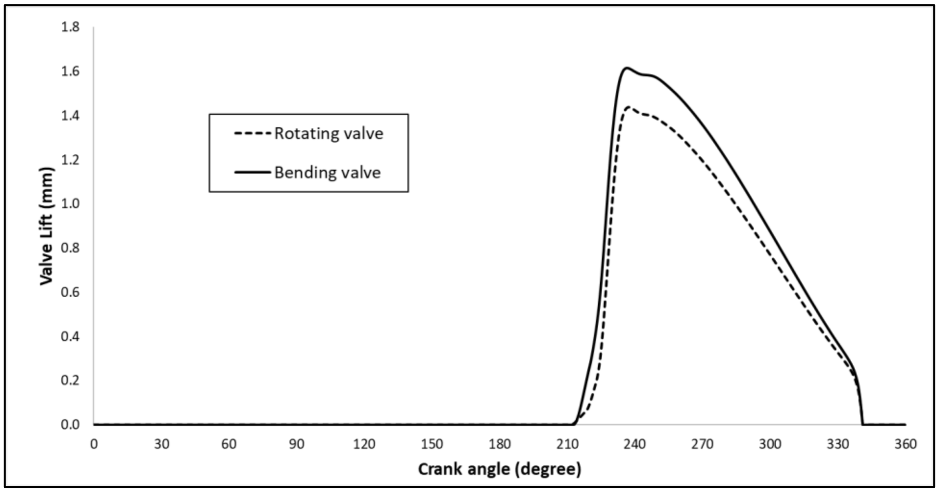


Figure 14: Comparison of valve lift at the tip of the valve between two modelling approaches

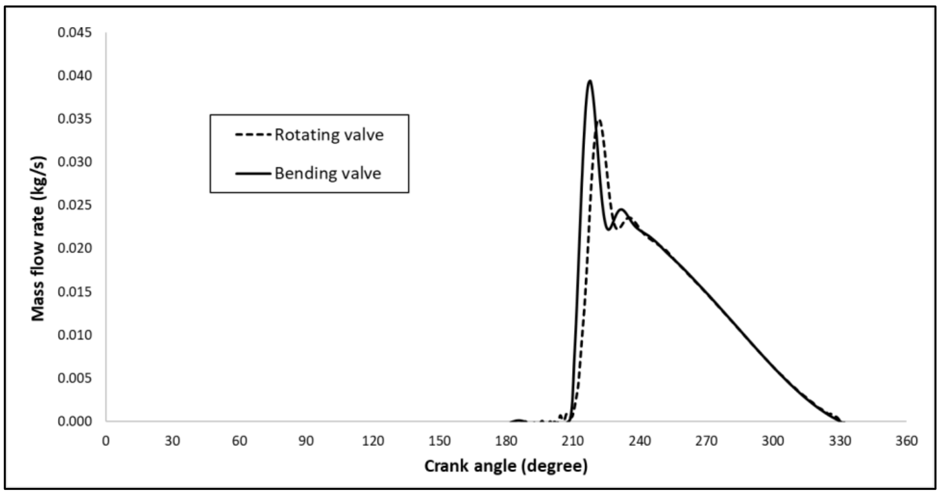


Figure 15: Comparison of mass flow rate from the port between two modelling approaches

Figure 16 shows the leakage flow in the contact region at vane tip, and the contact region in between the piston and chamber wall at the 270-degree crankshaft angle in a cutting plane.

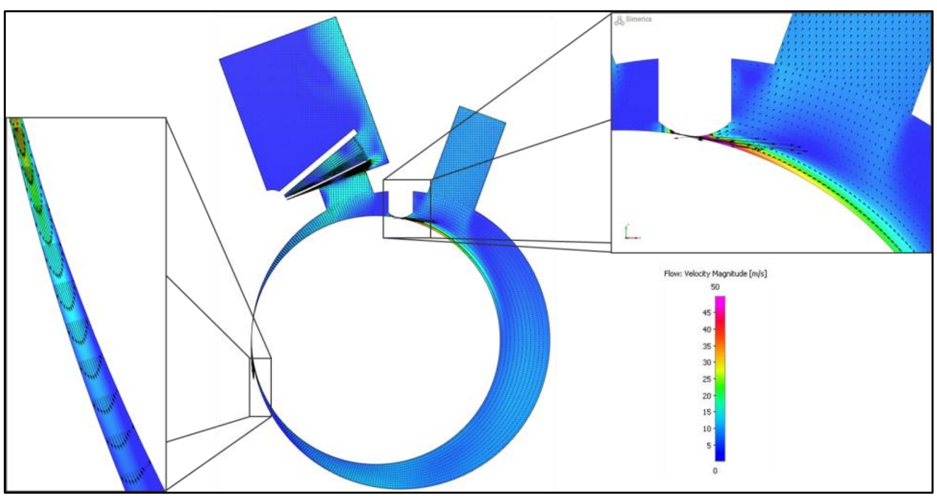
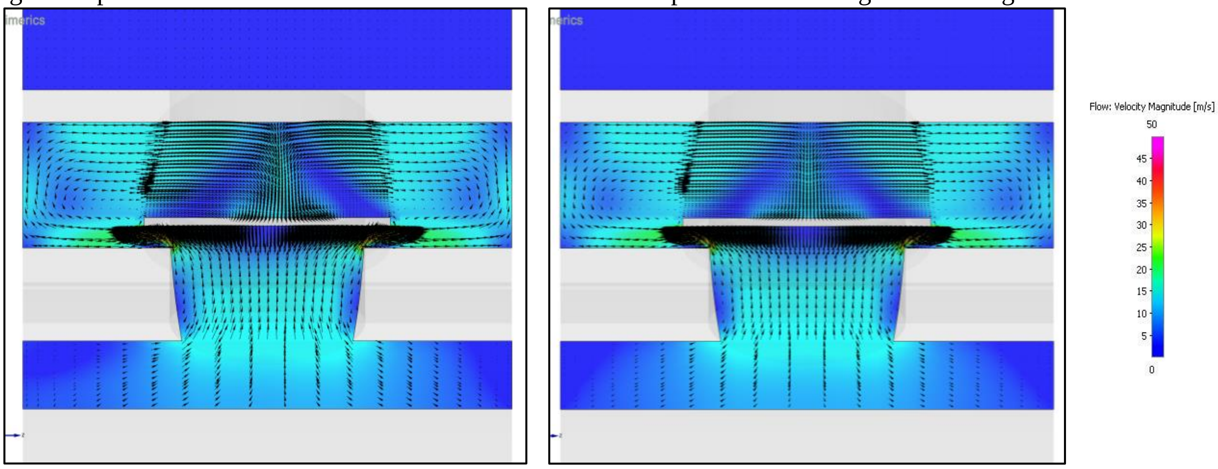


Figure 16: Compressor leakage flow

Figure 17 plots the flow field around the valve when the valve opened at crank angle of 270-degree.



1. Rotating valve
2. Bending valve

Figure 17: Flow field comparison around reed valve

7. CONCLUSIONS

A CFD model of a rolling piston compressor with a reed valve is developed to compare two approaches to reed valve modeling. In the rotating valve approach, the valve is assumed to rotate around its clamped position, simulating its actual movement. The second approach mimics the real bending of the valve by treating the reed valve as a cantilever beam, with one end fixed and the other end freely moving. CFD results from both approaches are compared using the same mesh and boundary conditions. While both approaches offer similar insights into the complete system, there is a difference in the valve lift profile: the maximum valve lift is 10% higher in the bending approach compared to rotation. This disparity in valve lift leads to differences in outflow from the port, with the overall outflow rate per revolution being 12% higher with the bending approach than with the rotating valve approach.

NOMENCLATURE

CFD	Computational Fluid Dynamics
PD	Positive Displacement
MGI	Mismatched Grid Interface
FSI	Fluid Structure Interaction
ODE	Ordinary Differential Equation
P	Load
E	Modulus of Elasticity
I	Moment of Inertia
x	Coordinates
y	Deflections
a	Length

REFERENCES

Ba, D., Deng, W.-J., Che, S.-G., Li, Y., Guo, H.-X., Li, N., & Yue, X.-J. (2016, April). Gas dynamics analysis of a rotary compressor based on CFD. *Applied Thermal Engineering, 99*, 1263-1269. doi:10.1016/j.applthermaleng.2016.01.062

Bhakta, A., Dhar, S., Bahadur, V., Angadi, S., & Dey, S. (2012). A valve design methodology for improved reciprocating compressor performance. *International Compressor Engineering Conference*.

Brancher, R. D., & Deschamps, C. J. (2014). Modeling of rolling-piston compressors with special attention to the suction and discharge processes. *International Compressor Engineering Conference*.

Burgstaller, A., Nagy, D., Almbauer, R., & Lang, W. (2008, January). Influence of the Main Parameters of the Suction Valve on the Overall Performance of a Small Hermetic Reciprocating Compressor. *International Compressor Engineering Conference*.

Ding, H., & Gao, H. (2014). 3-D Transient CFD Model For A Rolling Piston Compressor With A Dynamic Reed Valve. *International Compressor Engineering Conference*.

Gao, H., and Jiang, Y., (2014), Numerical Simulation of Unsteady Flow in a Scroll Compressor, Int. Compressor Engrg. Conf. at Purdue.

Geng, W., Liu, C. H., & Wang, Y. Z. (2004). The performance optimization of rolling piston compressors based on CFD simulation. *International Compressor Engineering Conference*.

Hsu, L. C., Wong, G. W., Lu, P. J., Hsu, F. S., & Chen, Y. C. (2020). Numerical simulation for flow of rolling piston type of rotary compressor. Energies, 13(10), 2526.

Huang, B., & Xie, F. (2008). Dynamic analysis of the discharge valve of the rotary compressor. *International Compressor Engineering Conference*.

Kinjo, K., Nakano, A., Hikichi, T., & Morinishi, K. (2010). Study of CFD considering valve behavior in reciprocating compressor. *International Compressor Engineering Conference*.

Kovacevic1, A., Rane1, S., Stosic1, N., Jiang, Y., and Lowry, S., 2014, Influence of approaches in CFD Solvers on Performance Prediction in Screw Compressors, Int. Compressor Engrg. Conf. at Purdue.

Launder, B.E., and Spalding, D.B., (1974), The numerical computation of turbulent flows, Comput. Methods Appl. Mech. Eng., 3, pp. 269-289.

Lenz, J. R., & Cooksey, E. A. (1994). Application of computational fluid dynamics to compressor efficiency improvement. *International Compressor Engineering Conference*.

Liang, S., Xia, S., Kang, X., Zhou, P., Liu, Q., & Hu, Y. (2010). Investigation of refrigerant flow simulation and experiment of rolling piston. *International Compressor Engineering Conference*.

Machu, G., Albrecht, M., Bielmeier, O., Daxner, T., & Steinruck, P. (2004). A universal simulation tool for reed valve dynamics. *International Compressor Engineering Conference*.

Mohapatra, C., Jacobsohn, G., Baldwin, E., & Schmidt, D. (2017, July). Modeling Sealing in Transient Injector Simulations., *Volume 1A, Symposia: Keynotes; Advances in Numerical Modeling for Turbomachinery Flow Optimization; Fluid Machinery; Industrial and Environmental Applications of Fluid Mechanics; Pumping Machinery*, p. V01AT04A009. doi:10.1115/FEDSM2017-69309

Nagy, D., Almbauer, R. A., Lang, W., & Burgstaller, A. (2008). Valve lift measurement for the validation of a compressor simulation model. *International Compressor Engineering Conference*.

Prasad, B. G. (2004). CFD for positive displacement compressors. *International Compressor Engineering Conference*.

Silva, E., Deschamps, C. J., & Fancello, E. A. (2012). A procedure to optimize reed type valves considering efficiency and bending fatigue. *International Compressor Engineering Conference*.

Spark, I., & Lu, K. (2019, August). The orbital displacer: Implications and applications.

Tan, Q., Liu, Z., Cheng, J., & Feng, Q. (2014). Effective flow and force areas of discharge valve in a rotary compressor. *International Compressor Engineering Conference*.

Yu, X., Ren, Y., Tan, Q., Lu, Z., Jia, X., & Wang, X. (2018, June). Study on the torsional movement of a reed valve in a rotary compressor. *Advances in Mechanical Engineering, 10*, 168781401877840. doi:10.1177/1687814018778402