2D transient analysis of suction process for coupled vane compressor

W C Poh, B Elhadidi, K T Ooi
Nanyang Technological University, Singapore
waichang001@e.ntu.edu.sg

Abstract. Although CFD has been widely applied to the study of positive displacement compressors, the primary challenge remains in the generation of solution domain of complicated geometry and accurate handling of the deformation of the working chamber during the compressor operation. In this paper, a 2D transient CFD simulation employing the overset mesh method is performed on the Coupled Vane Compressor (CVC) with air as the working fluid. Initial results revealed the presence of flow reversal and circulation zones at the suction channel resulting in the reduction of effective suction flow area. This has significantly reduced the mass intake. Proposed modification to the suction channel design has resulted in an increase of mass flow rate of 6.97% when operating at 1500rpm. Simulation approach, results and design modification are presented and discussed in this paper.

1. Introduction
The advances in computer technologies had boosted the use of CFD in analyzing many engineering problems, including the performance and design improvement studies of compressors [1, 2, 3]. However, the primary challenges of implementing CFD studies are the difficulties facing the ability to cope with the generation of complicated geometry and accurate handling of the deformation of the working chamber during the compressor operation. Various existing modelling approaches such as the dynamic mesh with remeshing method was presented for rolling piston compressor [4], innovative solution domain generations are also available in the literature for screw compressor [5] and sliding vane compressor [6]. Nevertheless, these methods could only cater to the relatively uniform deformations of the working chamber geometry of widely known compressors. Although CFD was shown to be employed successfully in the modelling of compressors, a few numbers were focused on the flow at the discharge port [7] and the discharge valve [8] of rolling piston compressors.

In this paper, the transient 2D simulation was employed to model the working processes of the Coupled Vane Compressor (CVC) [9] using the overset mesh methodology to demonstrate its capability in obtaining the solution domain without the need to remesh or generate mesh files at every angular position. Further overset mesh guidelines are available in the literature [10] and its successful application in modelling a single screw expander [11]. Flow analyses were then conducted based on the results obtained at the suction chamber to improve its design and to enhance its performance.
2. Simulation setups

2.1. Solution domain

Figure 1(a) illustrates the 3D schematic of CVC showing the suction, discharge and working chambers. The midplane representation of CVC as a 2D schematic is depicted in Figure 1(b) with the geometry of the coupled vane shown in Figure 1(c). The main dimensions of CVC are listed in Table 1 with the designed ideal volumetric displacement of 160 cm³/rev.

![Diagram](image)

**Figure 1.** (a) Isometric view of CVC, (b) 2D solution domain of CVC, (c) Coupled vane.

| Nomenclature          | Description                      | Dimension   |
|-----------------------|----------------------------------|-------------|
| \( C_r \)             | Centre axis of rotor             | –           |
| \( C_c \)             | Centre axis of cylinder          | –           |
| \( C_{vt} \)          | Centre axis of vane tip          | –           |
| \( R_r \)             | Radius of rotor                  | 20.25 mm    |
| \( R_c \)             | Internal radius of cylinder      | 32.50 mm    |
| \( D_{disc, port} \)  | Diameter of discharge port       | 7.00 mm     |
| \( b \)               | Distance between \( C_r \) and \( C_c \) | 12.50 mm    |
| \( R_{vt} \)          | Radius of vane tip               | 5.00 mm     |
| \( l_{vt} \)          | Length of vane from \( C_{vt} \) | 34.00 mm    |
| \( l_{vt} \)          | Length of vane tip from \( C_{vt} \)| 8.00 mm    |

Table 1. Main dimensions of CVC.

In the overset mesh approach, the component meshes are generated individually, as illustrated in Figure 2(a) while Figure 2(b) shows the assembly of all the components to form the complete solution domain with the arbitrary constant clearances of 50 µm between components. Different values of clearances (100 and 50 µm) were tested and the clearances can be reduced further with the overset method as needed. At least four elements across the clearances between each component are required such that the overset interface can be applied. Mesh regeneration is not required since each component mesh is displaced relative to the center of rotation based on the user-defined functions for the motion of the components while preserving the mesh quality and the specified clearance value. Thus, design modifications can be made easily by changing either one of the component mesh.
2.2. Simulation parameters
ANSYS Fluent 2019 R3 computational code was used to perform analyses with air as the working fluid, modelled as an ideal gas under adiabatic consideration. The $k - \omega$ SST turbulence model was used due to the presence of adverse pressure gradient at the clearances and better prediction for the near-wall flow conditions. Three operating speeds, i.e. 1500 rpm, 3000 rpm and 4800 rpm were simulated with the boundary conditions listed in Table 2. Simulations were carried out at angular steps of 0.009° with up to 30 iterations per time step to ensure numerical convergence.

| Table 2. Boundary conditions.                      |
|---------------------------------------------------|
| Type | Suction Chamber | Discharge Chamber |
|------|-----------------|-------------------|
| Type | Pressure Inlet | Pressure Outlet   |
| Absolute Pressure | 101,325 Pa | 506,625 Pa |
| Total Temperature | 300 K | 475 K |
3. Results and discussions

3.1. Basic simulation results
Mesh independence test shows the relative difference in mass flow rate per unit length from Mesh 3 is 1.92% as compared to Mesh 2, as shown in Table 3. Hence, Mesh 3 is used for further simulations.

Table 3. Mesh independence test.

| Mesh | No. of Elements | Mass Flow Rate per Unit Length [kg·s⁻¹/m] | Relative Difference |
|------|-----------------|--------------------------------------------|---------------------|
| 1    | 345,329         | 0.038478                                   |                     |
| 2    | 481,034         | 0.040122                                   | 4.27%               |
| 3    | 571,952         | 0.040893                                   | 1.92%               |

Figure 3. Variation of working chamber pressure from Shakya [9] and CFD at 1500 rpm.

Comparison of the working chamber pressure from CFD and that of the lumped parameters model from Shakya [9] is illustrated in Figure 3 along with the variation of working chamber volume. The results from both methods are comparable in predicting the working chamber pressure. Meanwhile, the deviation at the discharge process is due to the non-uniform reed valve model used in the lumped approach [9] as compared to the 1D torsional spring model used in this study. Furthermore, the effect of internal leakage that caused the reversed flow during the discharge process is present in the CFD result whereas the lumped parameters approach assumed the working fluid is discharged completely in a single compression cycle.

Table 4 presents the mass flow rate per unit length (which only represents the 2D midplane of CVC) whereas the variations of the working chamber pressure at the three operating speeds show that the size of the suction port chosen is able to cater for these speeds. However, for the discharge process, higher overpressure was observed at higher operating speed, which may be caused by the inability of discharge port size and/or reed valve to respond to higher operating speeds, as shown in Figure 4. Furthermore, the internal leakage effect diminishes with the increased operating speed.

Table 4. Mass flow rate per unit length at different operating speeds.

| Operating Speed [rpm] | Mass Flow Rate per Unit Length [kg·s⁻¹/m] |
|-----------------------|-------------------------------------------|
| 1500                  | 0.040893                                  |
| 3000                  | 0.103303                                  |
| 4800                  | 0.217469                                  |
Figure 4. Variation of working chamber pressure at different operating speeds.

Figure 5 shows the typical velocity flow fields at every 30° rotor rotational angle for the operating speeds of 1500 rpm, 3000 rpm and 4800 rpm. There are circulations and backflows during the suction process up to 90° rotor rotational angle due to suction port design. This has resulted in the reduction in the effective suction flow area throughout the entire suction process.
3.2. Effect of suction valve

To improve the suction process by reducing the reversed flow, a non-return suction valve has been added and the results of the typical velocity flow fields are shown in Figure 6. Notable differences in the flow field can be observed from rotor rotational angle of 30° to 90° where the backflow is trapped in the suction channel instead of flowing out of it as seen previously in Figure 5.

![Figure 6. Typical velocity flow fields with suction valve.](image)

![Figure 7. Variation of working chamber pressure with suction valve at 1500 rpm.](image)
Figure 7 shows the working chamber pressure between the default and modified designs at 1500 rpm. The accumulated working fluid in the suction chamber causes the rise in working chamber pressure and peaked around 100° rotor rotational angle due to the additional mass flow caused by the inclusion of the one-way flow suction valve. The pressure then decreases due to the expansion of the working chamber volume during the suction process. Consequently, the compression process occurs earlier at 225° rotor rotational angle from the extra mass leftover from the leakages. Similar trends also occur at 3000 rpm and 4800 rpm. The mass flow rate per unit length (which only represents the 2D midplane of CVC) for the three operating speeds improved, but the increment diminishes with the rise in operating speed, as presented in Table 5. 3D investigations of the current improvement are being carried out to account for the 3D nature of the suction flow entering the working chamber and the discharge flows effects through the discharge ports into the discharge chamber.

Table 5. Mass flow rate improvements from suction valve.

| Operating Speed [rpm] | Mass Flow Rate per Unit Length [kg·s⁻¹/m] | Improvement |
|-----------------------|------------------------------------------|-------------|
|                       | No Valve                   | With Valve  |              |
| 1500                  | 0.040893                   | 0.043745    | 6.97%       |
| 3000                  | 0.103303                   | 0.108973    | 5.49%       |
| 4800                  | 0.217469                   | 0.222244    | 2.20%       |

4. Conclusions
The transient 2D simulation was performed on the Coupled Vane Compressor (CVC) utilizing the overset mesh method. Basic simulation result shows that it is comparable to that from the lumped parameters approach. Furthermore, the flow field shows the occurrences of circulation zones and undesirable reversed flow which has reduced the effective suction flow area and resulted in the reduction of mass intake. A design modification has been carried out by adding a non-return valve at the suction has effectively mitigated the problem. This has resulted in an improvement of 6.97% in the mass flow rate at 1500 rpm and 2.20% at 4800 rpm. This exercise has shown that design modifications can be made swiftly for performance improvement. A more comprehensive transient 3D modelling using the same overset approach is currently underway.

Acknowledgements
The authors would like to express their gratitude and thanks to Nanyang Technological University, Singapore for its continuous support towards the research.

References
[1] H. Ding and H. Gao, "3-D Transient CFD Model For A Rolling Piston Compressor With A Dynamic Reed Valve," in International Compressor Engineering Conference, 2014.
[2] S. R. Rane, A. Kovačević and M. Kethidi, "CFD Modelling in Screw Compressors With Complex Multi Rotor Configurations," in International Compressor Engineering Conference, 2012.
[3] G. Bianchi, S. Rane, A. Kovačević, R. Cipollone, S. Murgia and G. Contaldi, "Grid generation methodology and CFD simulations in sliding vane compressors and expanders," in IOP Conference Series: Materials Science and Engineering, 2017.
[4] B. Farkas, V. Szente and J. M. Suda, "Dynamic meshing strategies to model fluid flow in rolling piston compressors," in The 16th International Conference on Fluid Flow Technologies, Budapest, Hungary, 2015.
[5] S. Rane, A. Kovačević, N. Stošić and M. Kethidi, "Grid deformation strategies for CFD analysis of screw compressors," International Journal of Refrigeration, vol. 36, no. 7, pp. 1883-1893, 2013.
[6] G. Bianchi, S. Rane, A. Kovačević and R. Cipollone, "Deforming grid generation for numerical simulations of fluid dynamics in sliding vane rotary machines," Advances in Engineering Software, vol. 112, pp. 180-191, 2017.
[7] W. Geng, C. H. Liu and Y. Z. Wang, "The Performance Optimization of Rolling Piston Compressors based on CFD Simulation," in *International Compressor Engineering Conference*, 2004.

[8] Q. Tan, S.-l. Pan, Q.-k. Feng, X.-l. Yu and Z.-l. Wang, "Fluid-structure interaction model of dynamic behavior of the discharge valve in a rotary compressor," *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, vol. 229, no. 4, pp. 280-289, 2015.

[9] P. Shakya and K. T. Ooi, "Introduction to Coupled Vane compressor: Mathematical modelling with validation," *International Journal of Refrigeration*, vol. 117, pp. 23-32, 2020.

[10] W. M. Chan, R. J. Gomez, S. E. Rogers and P. G. Buning, "Best practices in overset grid generation," in *32nd AIAA Fluid Dynamics Conference and Exhibit*, 2002.

[11] N. Casari, E. Fadiga, M. Pinelli, S. Randi, A. Suman and D. Ziviani, "Investigation of flow characteristics in a single screw expander: A numerical approach," *Energy*, vol. 213, p. 118730, 2020.