Observation of Oil Flow Characteristics in Rolling Piston Rotary Compressor for Reducing Oil Circulation Rate

S j Song\textsuperscript{1}, K Y Noh\textsuperscript{1}, B C Min\textsuperscript{1}, J S Yang\textsuperscript{2}, G M Choi\textsuperscript{3}, D J Kim\textsuperscript{3}

\textsuperscript{1}Graduate School of Mechanical Engineering, Pusan National University, San 30, Jangjeon-dong, Geumjeong-gu, Busan 609-735, Korea
\textsuperscript{2}Rolls-Royce University Technology Centre, Pusan National University, San 30, Jangjeon-dong, Geumjeong-gu, Busan 609-735, Korea
\textsuperscript{3}Department of Mechanical Engineering, Pusan National University, San 30, Jangjeon-dong, Geumjeong-gu, Busan 609-735, Korea

E-mail : Choigm@pusan.ac.kr

Abstract. The oil circulation rate (OCR) of the rolling piston rotary compressor is a significant factor which affects the performance of refrigeration system. The increase of oil discharge causes decreasing of the heat transfer efficiency in the heat exchanger, pressure drop and lack of oil in lubricate part in compressor. In this study, the internal flow of compressor was visualized to figure out the oil droplet flow characteristics. The experiments and Computational Fluid Dynamics (CFD) simulations were conducted in various frequency of compressor to observe the effect of operation frequency on oil droplet flow characteristics for reducing OCR. In situ, measurement of oil droplet diameter and velocity were conducted by using high speed image visualization and Particle Image Velocimetry (PIV). The flow paths were dominated by copper wire parts driving the motor which was inserted in compressor. In order to verify the reliability of CFD simulation, the tendency of oil flow characteristics in each flow path and the compressor operating conditions were applied in CFD simulation. For reducing OCR, the structure such as vane, disk and ring is installed in the compressor to restrict the main flow path of oil particle. The effect of additional structure for reducing OCR was evaluated using CFD simulation and the results were discussed in detail.

1. Introduction

In rolling piston rotary compressor the oil is essential for lubricating the sliding parts and filled in the sealed parts to reduce refrigerant leakage. The oil circulation rate called OCR is a significant factor which affects the performance of refrigeration system. Increase of the amount of oil discharge causes decrease of the heat transfer efficiency in the heat exchanger, pressure drop and lack of oil in lubrication part in compressor. Therefore, OCR reduction is necessary to improve the performance of refrigeration system. The rolling piston rotary compressor is divided in two types such as constant velocity type and inverter type. The inverter type rolling piston rotary compressor is using Brushless DC (BLDC) motor and it has high efficiency compared with constant velocity type. Moreover, by using the BLDC motor, the flow paths are formed inside of the compressor and these are changed by different muffler type.

Kamal sharma et al. (2010) carried out the CFD simulation of rolling piston rotary compressor for reducing OCR. This paper represents the importance of the gas velocities and turbulence in the rolling
piston rotary compressor for the reducing OCR. Yong Jae Kim et al. (2004) conducted the visualization inside the rotary compressor to observe the oil movement when the compressor was operating. Liyang Deng et al. (2012) studied the oil distribution in the rotary compressor and compared the influence of oil discharge rate in different operation condition.

In this study, to find the way to reduce OCR, the oil characteristics were observed by experiments and the experimental results were applied to CFD simulation. The rolling piston rotary compressor was visualized by using a high speed camera and particle image velocimetry (PIV) to investigate the flow characteristics. Also, for reducing OCR, the research on the effect of the structure installed in the compressor was carried out using CFD simulation.

2. Experimental Setup

To visualize the oil behavior inside the rolling piston rotary compressor, the sight glasses were installed at the top and side of the compressor. Figure 1 shows the schematic diagram of the experimental setup and continuous image capturing system. In this case, the laser beam was inserted vertically to the sight glass at the top of the compressor by mirror which was placed above the compressor. The optical setup was adjusted to visualize the flow path in the compressor. In addition, to obtain the position of main flow path at the top of the compressor, the install position of high speed camera and laser were changed. The specification of the high speed camera and laser is shown in Table 1. Figure 1(b) shows the schematic diagram of PIV system. The specification of PIV is in Table 2.

For a test model, the experimental condition was \( P_s : 10.15 \text{kgf/cm}^2, P_d : 23.37 \text{kgf/cm}^2, T_s : 18.3^\circ \text{C}, T_{\text{cond}} : 37.8^\circ \text{C}, T_{\text{eva}} : 7.2^\circ \text{C} \) and operating frequency was 40, 50, 60 Hz. Where, \( P_s \) is the suction pressure measured at the suction port at the compressor, \( P_d \) is the discharge pressure measured at the discharge port at the compressor, \( T_s \) is the suction temperature measured at the suction port at the compressor, \( T_{\text{cond}} \) is the suction temperature measured at the suction port at the compressor, \( T_{\text{eva}} \) is the condensing temperature and \( T_{\text{eva}} \) is the evaporating temperature.

![Figure 1](image)

**Figure 1. Schematic of experimental setup : (a) high speed camera (b) PIV**

| Table 1. Specification of high speed camera and laser |
|-----------------------------------------------|
| **High speed camera** | **Laser** |
| Resolution | 1024x1024, 128(W)X16(H) |
| Frame rate | 1000fps@1024X1024 |
| Shutter | 1.5μs (Min.) |
| Exposure time | 940μs |
| Lens | 50mm, f1.4 |
| Output Power | 0.8-5W |
| Line thickness | 60μm |

| Table 2. Specification of PIV and laser |
|---------------------------------------|
| **PIV** | **Laser** |
| Resolution | 2048 x 2048 |
| Frame rate | over 15frame/second with no binning |
| Lens | AF MICRO 105mm 1:2.8 D |
| Laser | 2 x 120mJ@532nm 15Hz Repetition Rate |
3. Numerical Analysis

To improve the reliability of the results, the CFD simulation was divided in two parts. The first case is the lower part of the motor and stator with muffler and the other is the whole parts which are placed above the muffler. Figure 2(a) and 2(b) show the mesh model of the muffler and flow field, and the mesh model of the whole part except muffler, respectively. For the boundary condition, the pressure inlet and pressure outlet were used, and the oil distribution at the inlet is almost uniform because the mesh is uniform at the inlet boundary condition. Numerical simulations were calculated by the discharge pressure when the valve is maximum opened.

![Figure 2. Mesh model: (a) Muffler mesh model (b) Whole mesh model](image)

In rolling piston rotary compressor, the flow paths are formed to A, B, C and D zones in the shell as shown figure 3(a). The position A is called air cut and B, C, D is copper cut, air gap, vent hole respectively. Also the location of holes at the muffler affects the refrigerant and oil flows in the compressor. In test model, the outlets at the muffler were placed as shown in figure 3(a). Also to observe the effect of the flow characteristics on the location of muffler outlet, the muffler holes were shifted as shown in figure 3(b). In this paper, the CFD simulation results of figure 3(a) and 3(b) is called Case 1 and Case 2, respectively.

![Figure 3. Outlet position at muffler: (a) Parallel with copper cut (b) Shifted with copper cut](image)

To observe the effect of additional structure on OCR, the ring and the disk were installed as shown in figure 4(a) and 4(b), respectively.

The CFD simulation was carried out at the compressor frequency of 60Hz and validated with the experimental results, the particle size and flow characteristics. In this simulation results, the steady state analysis was conducted and the operating fluid was single phase R410a. In addition, to simulate the rotation inside compressor, the Multiple Reference Frame (MRF) model was used. The specification of the simulation conditions are in Table 3. The commercial CFD tool ANSYS Fluent 14v was used for simulation.
4. Results and Discussion

4.1 Experimental Results

Figure 5(a) shows the experimental test model and Figure 5(b) and (c) show the top view of the high speed images at 40Hz and 60Hz respectively. As shown in Figure 5(b) and 5(c), the left side and right side are the copper wire and the bottom side is the rotor. Most of oil was discharged at the copper cut from the results of figure 5(b) and (c). In addition, comparing the results of 40Hz and 60Hz, the oil discharge amount of 60Hz was larger than that of 40Hz.
Figure 6 shows the velocity field obtained by PIV. As shown in figure 6 the oil flow direction is the upper right corner of the diagonal direction. Also the average velocity was about 3m/s.

![Vector field of oil droplet](image)

**Figure 6. Vector field of oil droplet**

### 4.2 Analysis of Numerical Simulation

#### 4.2.1 The Flow Characteristic at Muffler

Figure 7 shows the vector field of the lower cavity and inside of the muffler. When the refrigerant and oil are discharged from muffler, the direction is not vertically to the outlet surface. To apply the flow and particle direction to whole simulation condition, the part of muffler was simulated. At muffler, the 4 discharge holes were placed. As shown in figure 7, the refrigerant was discharged from muffler to outer region of the compressor.

![Vector field of the lower part with muffler](image)

**Figure 7. Vector field of the lower part with muffler**

#### 4.2.2 The Flow Characteristic with Inlet position Change

Figure 8 (a) and (b) show the vector field of case 1 and case 2, respectively. The flow direction of simulation result is similar with the experimental result. In addition, the average velocity at the copper cut has the same order as the average velocity of experimental result. In Figure 8 (a), the most of refrigerant flow passes through the copper cut. However, in Figure 8 (b), the main path of flow is different from case 1, because of the location change of muffler outlet. The reason of this result is because the refrigerant flow run into the weight balance of lower cavity and the direction of flow is changed to outer region of the lower cavity.

Figure 9 shows the particle tracks of case 1 and case 2. Figure 9 (a) and (b) are side view and figure 9 (c) and (d) are top view of the particle track. In case 1, the most of particle passes through the copper cut as same result with the refrigerant flow. The particles passed through the copper cut, rotate at the center of the upper cavity as shown in Figure 9(c) and these were discharged to outlet. However, in case 2, the particles were run into the weight balance as same of refrigerant and these are passed through the both air cut and copper cut. The particles which were passed to air cut, rotate outer of the shell as shown in Figure 9(d) and it affects the amount of discharged particle through outlet reduce.
4.2.3 The Flow Characteristics with Additional Structure Installed

Figure 10 shows the vector field when the structures, such as the ring and the disk, were installed. To observe the flow characteristic change more efficiently when the structure was installed, the case 1 with disk and case 2 with ring were compared. As shown in figure 10 (a), the flow is bumped into the disk and it makes the recirculation zone near the disk. Figure 10 (b) and (c) show the model with the ring which was simulated at the case 2. As shown in figure (b) and (c) the flow which passes through the air cut is interrupted by the ring and it circulates at the bottom of the ring.

Figure 11 shows the particle track of case 1 and case 2 with structure. Figure11 (a) and (b) are side view and figure11 (c) and (d) are top view of the particle track. Comparing the particle track of case 1 and case 1 with disk, the particle of case 1 with disk rotated outer of the upper cavity because the disk is placed at the center of the upper cavity and the particles pass through the outer of the disk. Also comparing the particle track of case 2 and case 2 with ring, the number of particles which is rotated at
upper cavity is reduced because the particle is bumped into the ring and the direction is changed to downward.

![Figure 10. Vector field of additional structure case: (a) Disk with case 1 (b) Ring with case 2 (c) Detailed view of figure 10 (b).](image)

![Figure 11. Particle track of additional structure case: (a) Side view of disk with case 1 (b) Side view of ring with case 2 (c) Top view of disk with case 1 (d) Top view of ring with case 2.](image)

**4.2.4 Particle and Mass Flow Rate**

In this study, the mass of particle is unconsidered and the discharged particle rate in each case was considered as shown in Figure 12. Comparison of the case 1 and disk with case 1, the discharged particle rate reduces about 4% because the disk interrupts the particle discharge to the center of the upper cavity. In addition, comparison of the case 2 and ring with case 2, the discharged particle rate reduces about 2%.

Figure 13(a) shows the mass flow rate in each case. The positions to measure the average mass flow rate are shown in Figure 13(b). Each position has the distance of 10mm from position 1 to upper cavity in figure 13(b). As shown in figure 13, the mass flow rate of all cases is similar. It means that the installation of disk and ring is influenced a little in COP. In figure 12 and 13, the mass flow rate of
refrigerant is similar but the oil reduces when the disk and ring was installed. The OCR is the function
of the refrigerant and oil. Therefore, the installation of the ring and disk is effective for reducing OCR.

![Graph showing percentage of discharged particle with variable cases](image)

Figure 12. Percentage of discharged particle with variable cases

![Graph showing mass flow rate in each position](image)

Figure 13. : (a) Mass flow rate in each position (b) Position measuring at the upper cavity

5. Conclusion

In this paper, the oil paths in the rolling piston rotary compressor were visualized and simulated
using high speed camera and PIV to observe the effect of additional structure in compressor. The
following conclusions are obtained from the experiment and CFD results.

1. Most of oil passes through the copper cut and the amount of discharged oil was higher with
increase of the compressor frequency from the experimental results.

2. The position of muffler outlet is influenced to the main refrigerant and oil paths which were
formed by stator and rotor.

3. When the oil rotates outer region of the upper cavity shell, the amount of discharged oil reduces
to compare when the oil rotates center of the upper cavity.

4. The installation of structures such as ring and disk is significantly influenced in reduction of OCR.
Reference

[1] Toyama T, Matsuura H, and Yoshida Y 2006 Visual Techniques to Quantify Behavior of Oil Droplets in a Scroll Compressor International Compressor Engineering Conference p 1788.

[2] Deng L, Liang S, Liu Q, Wu J and Xu J 2012 CFD Analysis of Oil Discharge Rate in Rotary Compressor International Compressor Engineering Conference p 2067.

[3] Sharma K, Rao K, V Kiran M. N. S and Gopinathan A 2010 CFD Analysis of Discharge Gas flow in Rotary Compressor for OCR reduction International Compressor Engineering Conference p1965.

[4] Kim Y J, Lee S K, Youn Y and Ko H S 2004 Experimental Study of Refrigerant-Oil Behavior Inside Rotary Compressor International Compressor Engineering Conference p 1698.

[5] Min K and Hwang, I 2000 Oil Circulation Rate in Rotary Compressor: Its Measurement and Factors Affecting the Rate International Compressor Engineering Conference. p 1395.

[6] ANSYS Fluent manual