Oil flow at the scroll compressor discharge: visualization and CFD simulation

Jiu Xu¹
Pega Hrnjak¹,²,*

¹ University of Illinois at Urbana-Champaign, Mechanical Science and Engineering, Urbana, IL, USA
² CTS, Creative Thermal Solutions, Urbana, IL, USA
pegah@illinois.edu

Abstract. Oil is important to the compressor but has other side effect on the refrigeration system performance. Discharge valves located in the compressor plenum are the gateway for the oil when leaving the compressor and circulate in the system. The space in between: the compressor discharge plenum has the potential to separate the oil mist and reduce the oil circulation ratio (OCR) in the system. In order to provide information for building incorporated separation feature for the oil flow near the compressor discharge, video processing method is used to quantify the oil droplets movement and distribution. Also, CFD discrete phase model gives the numerical approach to study the oil flow inside compressor plenum. Oil droplet size distributions are given by visualization and simulation and the results show a good agreement. The mass balance and spatial distribution are also discussed and compared with experimental results. The verification shows that discrete phase model has the potential to simulate the oil droplet flow inside the compressor.

1. Introduction

Oil plays an important role in cooling, lubrication and sealing of the compressors along with noise reduction. Oil is necessary for conventional compressors while oil circulating in vapor compression cycle has adverse effects on heat transfer and pressure drop. Even application of magnetic bearings and new material that reduce the need for lubrication to the level that compressors are oil free resulted in increasing number of non-lubricated machines, their number is still very small. Therefore, engineering attempts in design of vastly dominant lubricated compressors are made to keep the oil in the compressor and reduce the oil circulation ratio (OCR) in the system. Separating the oil from the high pressure refrigerant vapor and draining it back to the compressor is a feasible way to achieve this goal. For oil separator design, it is important to know the key oil flow parameter at the compressor discharge, including oil droplet size distribution, oil droplet velocity and vapor velocity. Furthermore, considering the compactness requirement of the compressor, oil separation structure integrated inside the compressor housing is preferred. These factors justify a closer look at the oil flow inside the compressor plenum.

For oil flow inside the compressor, flow visualization is a useful experimental method to understand the realistic condition. Toyama et al. [1] described methods to quantify oil droplet behavior using high-speed photography in the scroll compressor shell via sight glasses. Besides the size distribution of oil droplets, it is concluded that the mean diameter of oil droplets decreases as the flow speed of the refrigerant gas increases. Wujek and Hrnjak [2] developed visualization techniques to quantify the annular mist flow at compressor discharge pipe. The size and velocity of each oil droplet captured by
the high speed camera can be determined by video processing. It shows that the annular-mist flow is developing along the discharge pipe of a swash-plate compressor with R134a and PAG46 oil. Zimmermann and Hrnjak [3-4], showed that the most significant source of oil droplets in a scroll compressor is the break-up process of oil film between valve and valve seat during the opening and closing. The visualization of the reed valve in the scroll compressor shows the valve periodical movement and oil atomization process. Xu and Hrnjak [5-6] measured the OCR in a non-invasive way based on the oil annular mist flow visualization in the compressor discharge tube. Effect of focal depth is considered during the video processing to provide good estimation of number of oil droplets to provide amount of oil in that form in a specific control volume. Although flow visualization provides good image of what happens inside the compressor shell, the method is restricted by narrow vision and inaccuracy of modified space. Videos provide elements for reasonable assessment of droplet flow velocity in two dimensional case, but it is more difficult to say more about third dimension as well as velocities of vapor phase.

Computational Fluid Dynamics (CFD) is a strong tool to numerically simulate even the complicated two-phase flow when number of droplets is small and they are well dispersed. Yokoyama et al. [7] analyzed the oil flow in a CO₂ rotary compressor shell by CFD. Numerical simulations shows that flattening the lower balance weight or installing a rotating disk over upper balance weight can reduce the OCR and the results are verified by system experimental results. Gao et al. [8] conducted a simulation to get the dynamic trajectories and separation performance of oil droplets in oil-gas cyclone separator. Noh et al. [8] realized high-frequency visualization of the oil droplets in the compressor shell and simulated the separation efficiency of vane type separators. Xu and Hrnjak [9] validate the CFD simulation model using visualization results for oil droplet flow inside the scroll compressor plenum.

Based on previous research we can say: 1) The source of the oil droplets is the reed valve (if the compressor has it); 2) The flow pattern of refrigerant-oil mixture at the compressor discharge pipe is annular-mist flow; 3) Visualization techniques can be used to quantify the oil flow inside the compressor housing and in the compressor discharge tube; 4) CFD simulation has the potential to simulate the misty oil flow accurately in the compressor or the oil separator.

However, few literatures have combined the visualization result and CFD results together to give validated conclusions. So, our objective is to analyze the oil droplet flow in the scroll compressor by both visualization and simulation. Discrete phase model in CFD is used to predict the oil droplet trajectory in the scroll compressor plenum. The simulation results are validated with visualization results from high speed camera and mass balance. Furthermore, based on the validated simulation, it is meaningful to explore the effect of oil droplet size and potential oil separation structures that can be integrated into scroll compressor plenum.

2. Visualization

2.1. Experimental facilities
The oil flow inside the compressor plenum is visualized by high speed camera. In order to get clear images of the oil droplets inside the plenum, some modifications are made to the upper part of the scroll compressor plenum. As shown in Figure 1(a), initially mounted four side glasses were not sufficient and the transparent cylindrical ring was added between the original compressor housing. The transparent ring can hold around 1 MPa pressure at the compressor discharge. The visibility through the transparent ring is highly dependent on the surface finish. Two flanges with O-rings and bolts are used to seal the chamber. As shown in Figure 1(b), a plastic screen is placed in front of the compressor valve in order to eliminate the misty flow in front of the valves so that a clear vision can be achieved when capturing the video of oil near the reed valve. The particular compressor used in this project has three discharge valves, two of which are closed intentionally for easier visualization. Modification mentioned above makes some change to the real geometry and operating condition of the scroll compressor, but it is necessary cost which makes visualization possible. Videos are taken by the high speed camera with different lenses. Strong background light source is needed for enough brightness for a clear image. In- front light is needed when capturing the movement
of the reed valve. Camera frame rate needs to be higher than 1000 fps in order to capture the movement of oil droplets.

![Image of a scroll compressor plenum]

**Figure 1.** Modification of a scroll compressor plenum for visualization: (a) outer view; (b) inner view

### 2.2. Visualization of the oil flow

Videos of the oil flow are taken at three locations around the compressor: the center reed valve, the vicinity area in front of the valve and the discharge pipe. The locations are marked with frames in Figure 4 (geometry for CFD simulation). The shape of the plenum after modification is not exact the same as the original shape but the flow condition is expected to be similar under real conditions.

The video of the reed valve is taken by putting the focus directly on the valve tip. As shown in Figure 2(a), the valve opening process can be observed from about 45°.

Zimmermann and Hrnjak [4] have shown that the break-up process of the oil film between valve and valve seat generates oil droplets. A cloud of oil droplets injected by the high pressure of the compressed refrigerant as the valve opens. Video processing method is used to quantify the initial velocity of oil droplets injected to the plenum during the oil film break-up.

Figure 2(b) shows a sample frame from a video taken at the valve vicinity by focusing on the transparent screen in front of the valve. It is processed to estimate the size distribution of the oil
droplets injected in the valve opening. In the plenum chamber, the oil droplets are moving in all
directions at a relatively low velocity. The droplet size can be better estimated than the droplet velocity
because it is difficult to give the three-dimensional velocity by a series of planar images. Therefore, oil
droplet size distribution is the key parameter that is used to compare visualization results and
simulation results.

Figure 2(c), shows the frame taken in transparent discharge pipe located around 20 centimetres
from the compressor discharge port. The view in a raw form presents droplets (dark spots) behind the
film with waves. During quantification and analysis several filtering procedures are conducted to
better capture and measure the size, number and velocity of droplets. Since the flow is strongly one
dimensional droplet velocities determined are closer to the actual values. When estimating the total
number of droplets in the tube authors have first determined the depth of view, then number of
droplets in the view volume and their diameter and its distribution and then the total mass in the
droplet form. Details were presented by Xu and Hrnjak’s recent published paper [10].

![Image](image_url)

**Figure 2.** Sample video frames taken in different locations inside the compressor: (a) center reed
valve; (b) valve front; (c) discharge pipe

### 2.3. Results from video processing

The videos at different locations in the plenum were processed following the procedure developed by
Wujek and Hrnjak [3]. The fundamental principles are generally described as following:

- Eliminate the background/foreground wavy pattern (seen in Fig. 2c) by subtracting the average pixel
  value in period of time;
- Estimate the droplet size based on steepest grayscale gradient method, which detects the boundary
  between dark droplet image and the light background. The size of the droplets that are out-of-focus
  is corrected based on estimated depth of the view field;
- Compare two consecutive images in small sections to interrogate droplet displacement;
- Screen out the velocity vector using algorithm as in Particle Image Velocimetry (PIV). Numerous
  erroneous velocity vectors are generated throughout this process and the two-dimensional cross-
correlation is used to screen out true velocity vectors.

The oil size distribution at the valve front and discharge pipe for the same compressor at different
motor speed are shown in Figure 3. The curve shows the diameter-specific mass flow as a function of
oil droplet diameter. The integration of each curve is equal to the mass flow rate captured by the camera
under one specific working condition.

Comparing the curves under different compressor speeds (from 30 Hz to 60 Hz), it is intuitive to
conclude that higher compressor speed leads to more oil mass flow rate. Also, the average diameter of
the oil droplets at the valve is larger than that of the oil droplets in the discharge pipe. The droplet size
distribution results from the visualization will be discussed together with the CFD simulation results in
“3.7 Validation of the model”.

4
Figure 3. Oil size distribution by video at the vicinity of the valve and at discharge tube 20 cm downstream of the discharge under different compressor speed

3. CFD simulation

3.1. Discrete phase model
Discrete phase model (DPM) in computational fluid dynamics is usually used to simulate one fluid phase dispersed in another continuous fluid phase in a Lagrangian frame of reference. Discrete phase model is commonly used to simulate the liquid/solid particle motion in a continuous fluid. An important preliminary assumption made in the discrete phase model is that the dispersed phase occupies a low volume fraction. In this research, oil flow inside compressor discharge chamber is considered to be misty and oil volume ratio is always less than 5%, which is consistent with the DPM assumptions.

In this research, we use the Euler-Lagrange approach. The fluid phase is treated as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of droplets through the calculated flow field. The dispersed phase can exchange momentum with the fluid phase.

3.2. Simulation geometry and mesh
The flow region of the modified compressor plenum is depicted in Figure 4. Due to the visualization requirement, extra space is added to the original plenum, including two sight glasses and a cylindrical space surrounded by the polycarbonate ring.

The inlet of the flow region is set as the surface between the center reed valve and the valve base. Droplet injection is applied to the inlet surface and the details will be discussed in “3.3 Droplet injection”. The outlet of the flow region is the cross section of the discharge pipe.

Mesh independence is also carried out in order to achieve a balance between discretization error and calculation time. As shown in Figure 5, six different cases with different mesh numbers are tested and the average velocity in the discharge pipe is used as a reference to check the mesh independence. In the final simulation, the mesh number is around 1 million and the mesh type is tetrahedron with acceptable skewness and mesh quality.
3.3. Droplet injection
For discrete phase model, discrete phase is introduced in the simulation by define injection. As shown by visualization, oil droplets are generated by the valve opening process. Droplet pulsating injection instead of dynamic mesh is used for simulation because the plenum space is the main research object and the simplification of the opening of the valve can save a lot of calculation resources. Droplet size distribution of the injection is obtained from the video processing results at the valve vicinity as shown in Figure 3. Droplet injection direction and vapor phase velocity is got from simplified reed valve model from Zimmermann and Hrnjak [4].

3.4. Boundary conditions
In the discrete phase model, injection function set the source of discrete phase (usually droplets or particles). The injection is released from the inlet surface of discharge valve in the direction normal to

---

**Figure 4.** Geometry of flow region used for CFD simulation and locations of video capture

**Figure 5.** Mesh independence validation using average velocity in the discharge pipe as a reference
the inlet surface. The distribution of injection velocity and size distribution can be decided by the results from video processing.

The boundary condition is simplified to four different cases presented in Figure 6: trap, escape, reflect and wall film. Trap condition means that the droplet will be absorbed by the wall while escape condition allows the droplet pass through when they hit the boundary. A reflect wall will give an opposite velocity vector to the incoming droplet based on momentum balance. Wall-film model is most realistic case which allows a single component liquid drop to impinge upon a boundary surface and form a thin film.

For the CFD simulation carried out in this study, discrete phase boundary condition of all the inner wall of the compressor plenum is set as “wall film”. Inlet is set as injection as described in section 3.3 while the outlet of the discharge pipe is set as “escape”.

![Figure 6. Mechanisms for DPM boundary conditions](image)

3.5. **Solver settings**

The effect of turbulence is considered and k-ε model is used in the flow calculation. The simulation takes SIMPLE as the scheme and uses second order for pressure discretization and second order upwind for momentum discretization. The time step for transient calculation should be as small as possible but here it is set as 0.001s for reasonable computational time.

3.6. **Results**

The trajectories of oil droplets can be calculated by discrete phase model using ANSYS Fluent. When the oil droplets hit the boundary, they will either bounce back or splash into more droplets or be absorbed by the wall (presenting absorption in the film). Some of the oil droplets can and will leave the plenum and are recorded by the monitor at the compressor discharge tube. Figure 7 shows the trajectories in the plenum space of a small sample of tracked oil droplets, colored by droplet diameter. It can be seen that larger droplets tend to settle down in the film at the bottom of the compressor while smaller droplets tend to flow along with the vapor stream.

![Figure 7. Droplet trajectories colored by droplet diameter in the plenum space](image)
Figure 8 presents the mass balance of the oil in droplet form in the plenum in time. The solid line shows the inlet of oil droplets from the valve opening. The influx curve looks linear but it is actually periodic according the compressor frequency. The dashed line presents the flow out (Escaped through the discharge line). Equal gradients of solid and dash line indicate the steady state in the mass of droplets in the plenum. The dash dot line with its constant value shows the time when flow out became equal to the inlet to the plenum. Constant value of the dot line indicates that as much of the oil is absorbed in the film is removed from it by entrainment. There are three fates that a droplet will meet: flow with the refrigerant vapor, accumulate in the wall film, or escape from the discharge tube. The mass conservation holds when all the oil droplets injected into the domain will either stay in the domain or escape from it. The curve of the droplets of different fates in Figure 8 has shown the mass conservation generally holds.

In principle we would like to see accumulation to grow in time (separation) and to be taken out back to compressor suction.

3.7. Validation of the model
In order to validate the CFD simulation, besides mass balance oil droplet size distribution is chosen as the key parameter to compare visualization results and CFD results. This is mainly because droplet size distribution is the most reliable and statistically representative parameter that we can get from both approaches and it is important for oil separator design. The logic and process of comparison and validation are shown graphically in Figure 9. Video taken near the valve is used to give the as the input parameters for injection, like oil droplet size and velocity. The vapor velocity is given by the pulsating flow model mentioned in 3.3. Based on the input parameters, transient CFD calculation can monitor the oil droplets that escaping out of the compressor plenum and get a droplet size distribution. On the other side, the video taken at the discharge tube is processed to give a droplet size distribution. These two distributions are compared to see if the CFD results can agree with the video results.
Figure 9. The process of validation of CFD results based on droplet size distribution from visualization

Both experiments and simulation are carried out at different compressor speed in steady state. Figure 10 shows the comparison of droplet size distribution between CFD results and video results. The curve represents the injected oil droplet size distribution, which is set as the input information of the injection in the simulation. What we want to compare is the distribution curve by CFD (dash line) and the distribution curve by video (solid line) in each plots. By comparison, it shows the CFD results can match well with video results at 30, 40 and 50 Hz. The discrepancy of the two results at 60 Hz may be explained by high OCR and high mass flow rate of oil injected by valve frequent opening will lead to a high possibility of collision between droplets. More droplets collision will make smaller droplets coalesce into bigger ones. Since the interaction between droplets are not considered in the simulation, it is possible that the simulation underestimate the amount of large droplets and overestimate the amount of smaller droplets at the compressor discharge.

The results show that CFD simulation is able to give a qualitatively prediction of the droplet size distribution at the compressor outlet. From the comparison between droplet size distribution of the injection and that of the discharge, it is clear that large droplets are more likely to hit the wall and splash on the wall film to generate more small droplets. Smaller droplets are less affected by gravity and inertia so that they can catch up with the vapor flow to escape the compressor and form the annular-mist flow in the discharge tube.

Figure 10. Comparison of oil droplet size distribution by video or CFD at the scroll compressor discharge under different compressor running speed

4. Summary
This paper presented the oil flow inside the discharge plenum of scroll (but could be any) compressor using both visualization technique and CFD simulation. Comparing visualization and simulation results, we can conclude that CFD model provided reasonably good description of the process inside the discharge plenum.

- Visualization of oil flow through discharge reed valve provides droplet size distribution and droplet velocity, which are used as the important input of the CFD simulation.
- Videos of oil droplet flow are taken in front of the valve and in the discharge tube. Video processing techniques can estimate droplet size and velocity based on the high speed video captured.
- Discrete phase model can be utilized to simulate the oil droplet trajectory inside compressor plenum. CFD simulation is validated by comparison of oil drop size distribution at the compressor discharge tube.
- CFD simulation also presents that mass conservation holds, which means oil mass flow rate at the discharge valve is the same as that at the compressor discharge.
- Higher compressor speed generates more oil droplets. Large droplets tend to retain in the plenum while small droplets are more likely follow the refrigerant vapor flow to escape. The discharge plenum space of a vertical scroll compressor has the potential to separate large oil droplets.

Acknowledgement

The authors would like to acknowledge the companies who sponsor of the Air Conditioning and Refrigeration Center for their technical and financial support to this work. The authors are also thankful for Augusto Zimmermann who made the compressor prototype and built a solid foundation of the visualization work.

References

[1] T. Toyama, H. Matsuura, and Y. Yoshida, “Visual Techniques to Quantify Behavior of Oil Droplets in a Scroll Compressor,” in International Compressor Engineering Conference. Paper 1788, 2006.
[2] S. Wujek and P. Hrnjak, “Mist to annular flow development quantified by novel video analysis. ACRC Report TR285,” Urbana, 2011.
[3] A. J. P. Zimmermann and P. S. Hrnjak, “Visualization of the Opening Process of a Discharge Reed Valve in the Presence of Oil,” in International Compressor Engineering Conference. Paper 2369, 2014.
[4] A. J. P. Zimmermann and P. S. Hrnjak, “Oil flow at discharge valve in a scroll compressor.,” Proc. Inst. Mech. Eng. Part E J. Process Mech. Eng., vol. 229, no. 2, pp. 104–113, 2015.
[5] J. Xu and P. Hrnjak, “Refrigerant-Oil Flow at the Compressor Discharge,” in SAE Technical Paper 2016-01-0247, 2016.
[6] J. Xu and P. Hrnjak, “Oil Flow Measurement at the Compressor Discharge,” in International Compressor Engineering Conference. Paper 2506, 2016.
[7] T. Yokoyama, K. Shingu, S. Sekiya, H. Maeyama, and N. Nishimura, “CFD Analysis inside a CO2 Rotary Compressor Shell to Improve Oil Separation and Reduce the Shell Size,” in International Compressor Engineering Conference. Paper 2074, 2012.
[8] X. Gao, J. Chen, J. Feng, and X. Peng, “Numerical and experimental investigations of the effects of the breakup of oil droplets on the performance of oil–gas cyclone separators in oil-injected compressor systems,” Int. J. Refrig., vol. 36, no. 7, pp. 1894–1904, 2013.
[9] J. Xu and P. Hrnjak, “Visualization and Simulation of Oil Flow in a Scroll Compressor Plenum Corresponding author,” Int. Compress. Eng. Conf. Pap. 2507, 2016.
[10] J. Xu and P. Hrnjak, “Quantification of flow and retention of oil in compressor discharge pipe,” Int. J. Refrig., 2017.