2016

Visualization and Simulation of Oil Flow in a Scroll Compressor Plenum

Jiu Xu
Air Conditioning and Refrigeration Center, University of Illinois at Urbana-Champaign, jiuxu2@illinois.edu

Pega Hrnjak
Air Conditioning and Refrigeration Center, University of Illinois at Urbana-Champaign/ Creative Thermal Solutions Inc., Urbana, Illinois, pega@illinois.edu

Visualization and Simulation of Oil Flow in a Scroll Compressor Plenum

Jiu XU[1], Pega HRNJAK[1,2,*]
[1]University of Illinois at Urbana-Champaign, Mechanical Science and Engineering,
Urbana, IL, USA
[2]CTS, Creative Thermal Solutions, Urbana, IL, USA
jiuxu2@illinois.edu, pega@illinois.edu

[*]Corresponding author

ABSTRACT

Discharge valves located in the compressor plenum are the gateway for the lubrication oil to leave the compressor. The compressor plenum space has the potential to separate the oil droplets and decrease the oil circulation ratio. In this paper, a video processing method is used to capture the oil droplet flow near the reed valve and in the discharge tube in a modified scroll compressor. Also, discrete phase model in computational fluid dynamics is used as the numerical approach to study the oil droplet trajectory inside the compressor plenum. In order validate the CFD simulation, size distribution from video processing at the valve is used as the input of the calculation and the calculation results are compared to size distribution in the discharge tube. Calculation shows how oil droplet size distribution affects the oil flow in the plenum.

1. INTRODUCTION

Oil existence in the compressor is necessary while oil circulating in vapor compression system is known to have adverse effects on system heat transfer and pressure drop. Therefore, it is preferable to reduce the oil circulation ratio (OCR) in the system and keep the oil in the compressor.

Separating the oil from the refrigerant vapor and draining it back to the compressor is a feasible way to achieve low OCR. For a scroll compressor, it has the space to integrate oil separation structure into the discharge plenum. Oil flow parameters, like oil droplet size and velocity, are very important for oil separator design. This justifies a closer look at the oil flow inside the compressor plenum as well as the discharge tube.

Many researchers have studied the oil flow and distribution in the compressor by visualization or numerical simulation. Toyama et al. (2006) 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.

Yokoyama et al. (2012) analyzed the oil flow in a CO_2 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.

Wujek and Hrnjak (2011) developed visualization techniques to quantify the annular mist flow at compressor discharge tube. The size and velocity of each oil droplet captured by the high speed camera can be determined by video processing. It shows that the droplet population periodicity at the discharge of a swash plate compressor with R134a and PAG46 oil.

Zimmermann and Hrnjak (2014a, 2015) 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 (2016) measure the OCR in a non-invasive way based on the oil annular mist flow visualization in the compressor discharge tube. Camera focus effect is considered into the video processing which gives a better estimation of oil droplet amount in a specific control volume.

Therefore, previous research has shown 1) the origin source of oil droplets in the scroll compressor; 2) the flow pattern of refrigerant-oil mixture at the compressor discharge 3) the visualization used to quantify the oil flow. 4) Numerical model used to simulate the oil flow.

The motivation of this research is to link the visualization with simulation in order to study the oil droplet flow in a scroll compressor plenum. Discrete phase model in CFD is used to predict the oil droplet trajectory in the plenum. Also the simulation results will be validated with experimental results from flow visualization. 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. OIL FLOW VISUALIZATION

2.1 Compressor modification

The oil flow visualization is realized inside the compressor plenum based on the video captured by a high speed camera. In order to get clear images of the oil droplets inside the plenum, side glasses and a transparent ring are added to the upper part of the compressor, as it is shown in Figure 1.

Figure 1: Modification of a scroll compressor plenum for visualization

The transparent ring is made of polycarbonate which can hold the high side pressure of the compressor. The transmittance of the transparent ring is highly dependent on the surface finishing and polishing process. A flange structure, together with O-ring and bolts is used to seal the chamber and make sure it's air tight under high pressure.

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 compressor has three discharge valves but in this work, two auxiliary valves are forced to close intentionally for easier visualization.

2.2 Videos of oil droplets

Videos are taken by a Phantom camera with different lens at three locations: the reed valve, the valve vicinity and the discharge tube, as shown in Figure 2.

The video of the reed valve shows the valve opening sequence and how oil film between valve and valve seat breaks up. A cloud of small oil droplets flows from the valve edge as the valve opens and the oil film breaks. Zimmermann and Hrnjak (2014b) gave a method to quantify the initial velocity of oil droplets injected by the film break-up process.

The video near the valve is processed to give the information of oil droplet size distribution. 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 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.

The video in the discharge tube is taken at a section of transparent Polytetrafluoroethylene observation tube. Compared to the video near the valve, it is easier to process to get the droplet velocity since the axial velocity along the tube is much larger than the radical one. The effect of camera depth of focus is also considered to estimate the oil droplet size.

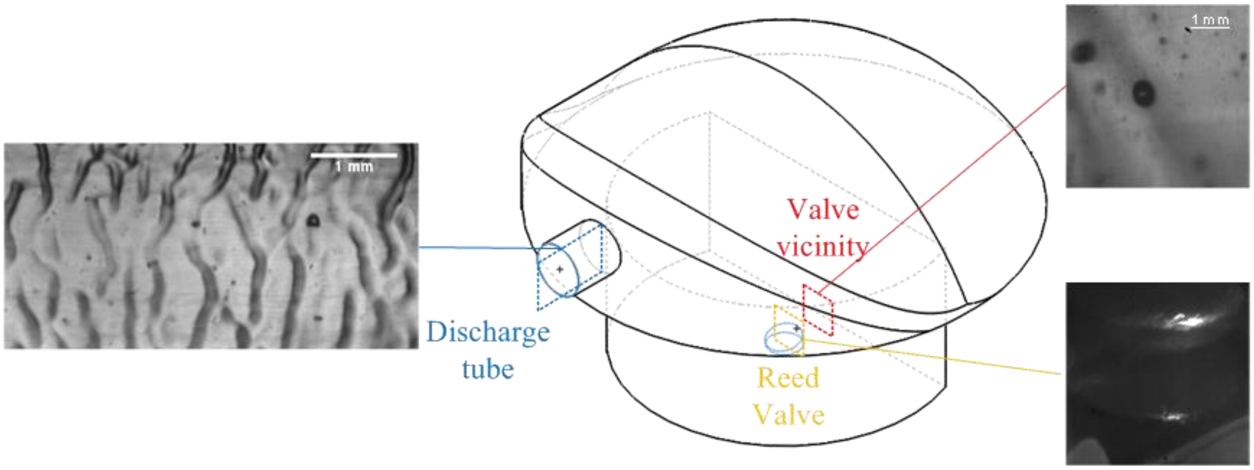

Figure 2: Locations of video taken in the modified compressor

3. OIL FLOW 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 our case, 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 Geometry and mesh

The research object of CFD simulation in this research is limited to the modified compressor plenum and discharge tube. The surface swept by the edge of the reed valve is defined as the inlet of refrigerant vapor flow and oil droplet injections while the discharge tube is defined as the outlet of the continuous fluids. Figure 3 shows the geometry and mesh of the modified compressor plenum used for calculation. Besides the original plenum space, the additional space surrounded by transparent polycarbonate ring and transparent screen is also included to make sure the simulation results can be comparable with the experimental results. Half of the geometry is actually used for CFD simulation in order to save the calculation time and the boundary of the center plane is set as “symmetry”.

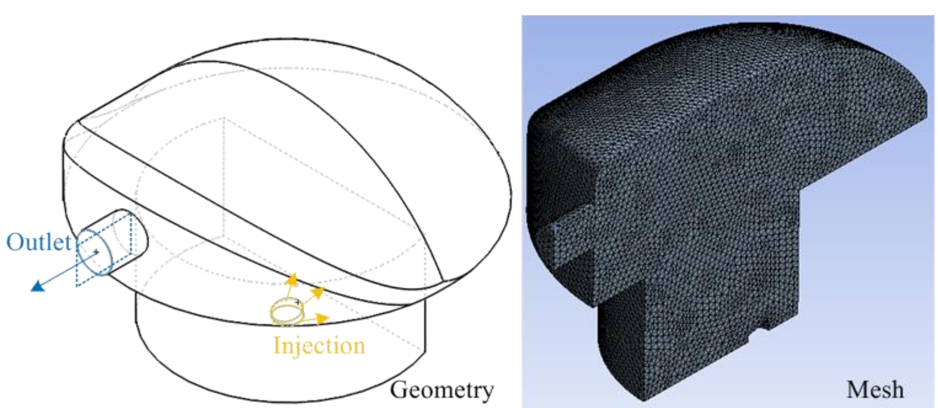

Figure 3: Geometry of the modified compressor plenum and mesh of the half of the plenum

For meshing, the max face size is set as 1 mm for the 3-D model and the elements number is around one million. The mesh around the valve is refined because of finer geometry details. Overall mesh quality is checked before running the CFD calculation.

3.3 Boundary Condition

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 the inlet surface. The distribution of injection velocity and size distribution can be decided by the results from video processing.

The physics of oil droplet interaction with solid wall is complicated. In this research, the boundary condition is simplified to four different cases: trap, escape, reflect and wall film. Trap condition means that the droplet will be absorbed by the wall while escape condition allows the droplet to 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. Table 1 shows the boundary conditions for continuous phase and discrete phase that are used in the simulation.

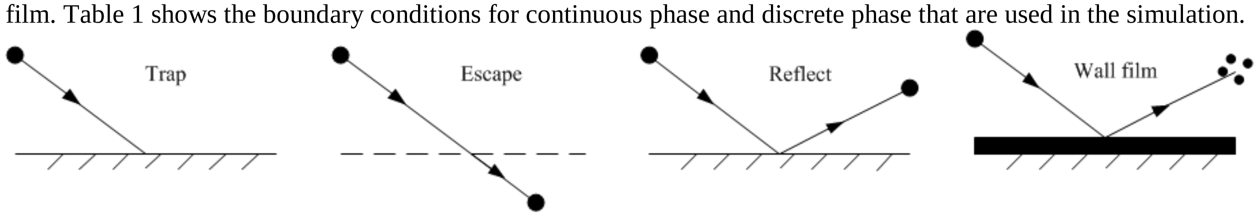

Figure 4: Mechanisms for DPM boundary conditions

Surface	Continuous phase boundary condition	Discrete phase boundary condition
Inlet	Velocity inlet	Injection
Valve	No-slip wall	Reflect
Wall	No-slip wall	Wall-film
Bottom	No-slip wall	Wall-film
Outlet	Outflow	Escape

Table 1: Boundary conditions defined for each

3.4 Droplet Injection

As mentioned in 3.3, injection is the discrete phase boundary condition for the inlet. The information of the droplet injection from video can be specified in the CFD calculation. The distribution of droplet diameter is fitted by Weibull distribution to be used as the injection parameter at the inlet. A typical droplet size distribution curve can be seen in Figure 8.

For the real case, the geometry should be a moving mesh because valve opens and closes from time to time. However, since the volume of valve moving space is relatively small compared to the whole plenum space, the valve is set to be open all the time. Instead of the moving mesh, pulsating flow at the inlet is used to simulate the dynamic mass flow discharged by the reed valve. The valve opening and closing period is decided by video taken at the reed valve and the mass flow rate is given by the mass flow meter in the system. As shown in Figure 5, the pulsating flow of the vapor injected by the discharge valve is simplified to a square wave and the vapor velocity and valve opening displacement are linked with mass flow rate conservation.

3.5 Numerical 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.

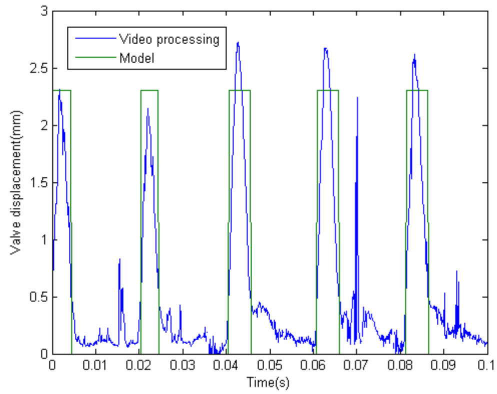

Figure 5: Pulsating flow to simulate reed valve opening and closing at 50 Hz

4. VALIDATE SIMULATION WITH VISUALIZATION RESULTS

4.1 The effect of oil droplet size

The trajectories of oil droplets can be calculated by DPM model in Fluent. When the oil droplets hit the boundary, they will either bounce back or splash into more droplets or be absorbed by the wall. Some of the oil droplets can leave the plenum and recorded by the monitor at the compressor discharge tube. Figure 6 shows the droplets in the free stream and bottom wall-film, colored by droplet diameter. It can be concluded that larger droplets tend to settle down in the wall film at the bottom of the compressor while smaller droplets tend to flow along with the vapor free stream.

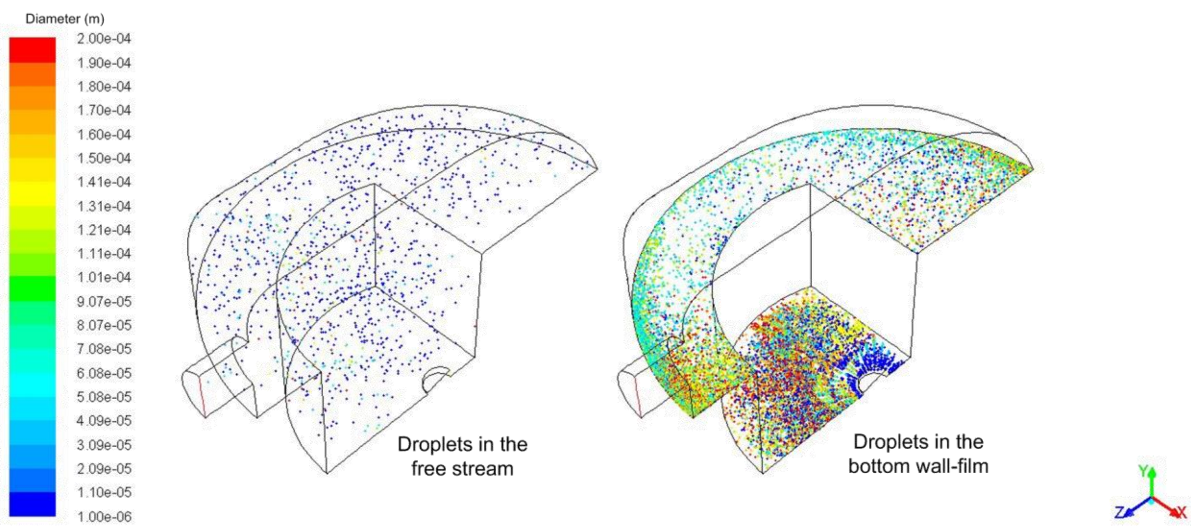

Figure 6: Tracked droplets position at t =1s, colored by droplet diameter

4.2 Oil droplet size distribution under different compressor speed

In order to validate the CFD simulation, 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 3.4. 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. The process of validation is shown graphically in Figure 7.

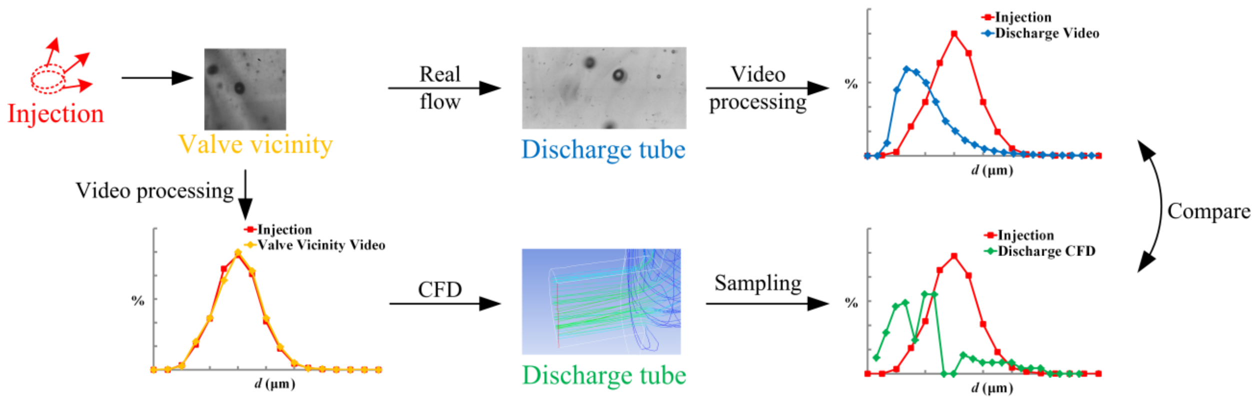

Figure 7: Comparison of visualization results and simulation results under different compressor speed

Both experiments and simulation are carried out at different compressor speed in steady state. Figure 8 to Figure 10 show the comparison of droplet size distribution between CFD results and video results. The red curve represents the injected oil droplet size distribution, which is the input information to the model. Experimental flow visualization gives the green dash line while the simulation gives the blue dot line. Both of them represent the escaped oil droplet size distribution.

The results show that CFD simulation can give a qualitatively prediction about 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.

It is also interesting to see higher compressor speed generates finer oil droplets near the valve when we compare between the red curve in Figure 8, 9, and 10. This trend also agrees with the conclusion given by other researchers (Zimmerman and Hrnjak, 2014a).

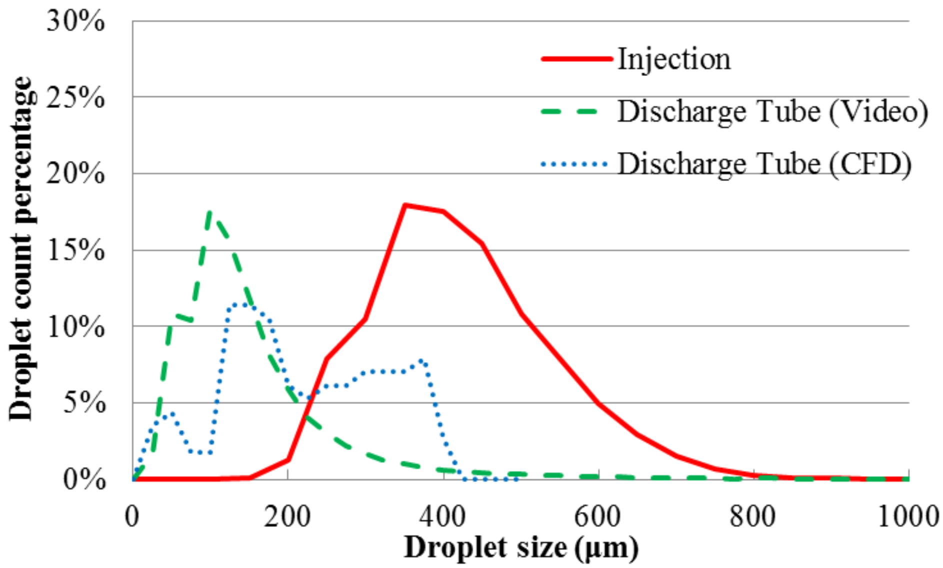

Figure 8: Comparison of oil droplet size distribution by video or CFD for compressor running at 40 Hz

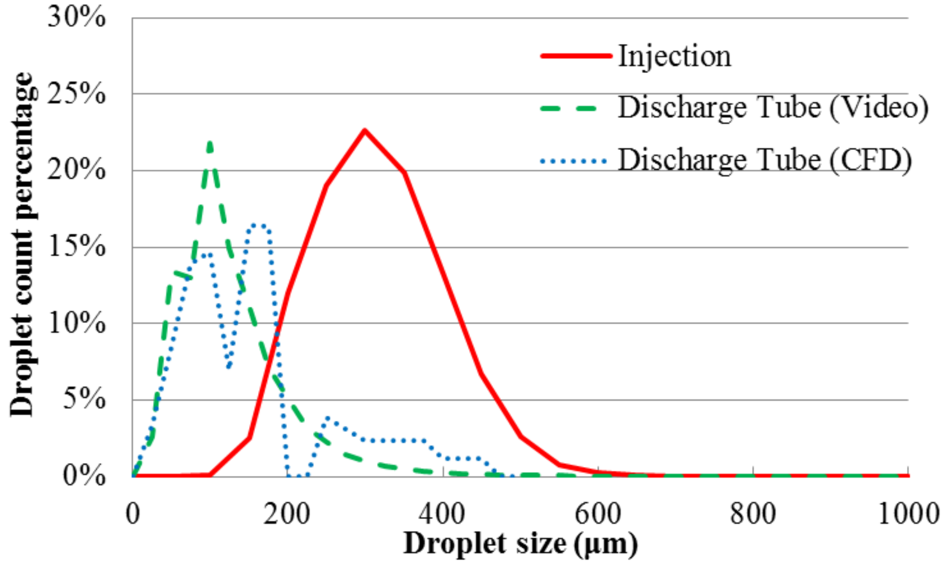

Figure 9: Comparison of oil droplet size distribution by video or CFD for compressor running at 50 Hz

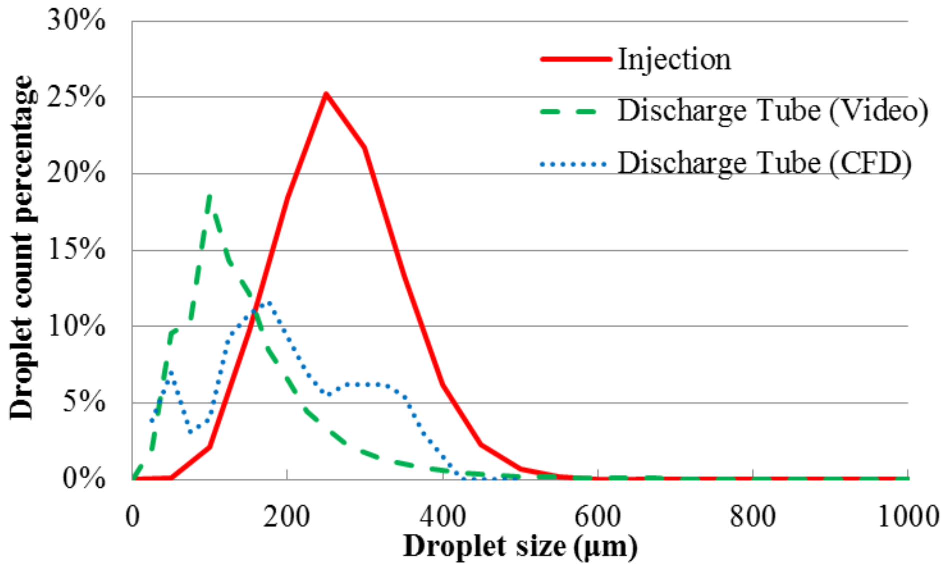

Figure 10: Comparison of oil droplet size distribution by video or CFD for compressor running at 60 Hz

This research is limited to the top plenum of the high side of a scroll compressor. For the start-up process of a compressor, oil droplets are generated by the opening and closing process of the discharge valve and some of the oil will retain in the plenum while the other will escape from the discharge tube. With more and more oil retained in the plenum, the re-entrainment effect becomes more and more significant and finally the oil mass flow rate entering the plenum will equal to the oil mass flow rate leaving the plenum. Therefore, at steady state, the oil flow rate at the valve is the same as the oil flow rate in the tube. The difference is oil droplet size distribution. The results shown in this research also verifies the idea that oil separating structure can be designed in the compressor plenum for easier separation where most of the oil exists in larger size.

5. CONCLUSIONS

Both visualization and simulation method are applied to research the oil flow inside compressor discharge plenum. From the comparison between visualization results and simulation results, we can conclude that:

- Visualization of the compressor plenum shows that the source of oil in the system is oil droplets injected by the reed valve opening and closing process.

- 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 droplets inside compressor plenum. The simulation results are compared to the video results at the compressor discharge under different compressor speed. The agreement between simulation and experiments validate the CFD model used for oil droplets trajectory prediction.
- Higher compressor speed will generate more oil droplets. The distribution of oil droplets size fulfills the Weibull distribution statically. Larger droplets tend to retain in the plenum while smaller droplets are more likely follow the refrigerant vapor flow.

REFERENCES

Toyama, T., Matsuura, H., & Yoshida, Y. (2006). Visual techniques to quantify behavior of oil droplets in a scroll compressor. *International Compressor Engineering Conference*. Paper 1788.

Wujek, S. S., & Hrnjak, P. S. (2011). Mist to annular flow development quantified by novel video analysis. ACRC Report TR285, University of Illinois at Urbana-Champaign, Urbana, IL.

Yokoyama, T., Shingu, K., Sekiya, S., Maeyama, H., & Nishimura, N. (2012). CFD Analysis inside a CO2 Rotary Compressor Shell to Improve Oil Separation and Reduce the Shell Size. *International Compressor Engineering Conference*. Paper 2074.

Xu, J., & Hrnjak, P. (2016). Refrigerant-Oil Flow at the Compressor Discharge (No. 2016-01-0247). SAE Technical Paper.

Zimmermann, A. J. P., & Hrnjak, P. S. (2014). Source Identification and In Situ Quantification of Oil-Refrigerant Mist Generation by Discharge Valve Opening Process. *International Compressor Engineering Conference*. Paper 2367.

Zimmermann, A. J. P., & Hrnjak, P. S. (2014). Visualization of the Opening Process of a Discharge Reed Valve in the Presence of Oil. *International Compressor Engineering Conference*. Paper 2369.

Zimmermann, A. J., & Hrnjak, P. S. (2015). Oil flow at discharge valve in a scroll compressor. Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering, 229(2), 104-113.

ACKNOWLEDGEMENT

The authors would like to acknowledge the support of the member companies of the Industrial Advisory Board for the Air Conditioning and Refrigeration Center at the University of Illinois through ACRC project 341.