PAPER ID: 0396
DOI: 10.18462/iir.compr.2021.0396

LIQUID SLUGGING IN RECIPROCATING COMPRESSOR

Jan Tuhovcak[(a)], Eric Murakami[(a)], Jan Bossanyi[(a)]

[(a)] Secop Austria GmbH
Gleisdorf, 8200, Austria, jan.tuhovcak@secop.com

ABSTRACT

The refrigerant typically enters the compressor in super-heated vapor state, although in some specific operating conditions the refrigerant might enter in liquid state. This usually happens during a defrost cycle or cold compressor start up. In some cases, the liquid refrigerant might be injected deliberately into the compression chamber in order to increase the cooling capacity. However, in most of the cases the liquid state of the refrigerant entering the cylinder increases the risk of a failure due to significant pressure elevation. In order to avoid liquid slugging in reciprocating compressor, the suction flow path must be redesigned. This paper presents a study on two-phase flow of refrigerant applying numerical simulation. The effect of suction muffler design and boundary conditions on slugging are discussed.

Keywords: two-phase flow, reciprocating compressor, suction muffler, liquid separation

1. INTRODUCTION

Positive displacement compressors are designed to work with the gas or with the refrigerant in vapor state, nevertheless there are situations when instead of vapor there is a significant amount of liquid coming inside the compression chamber in uncontrolled manner. This process is called liquid slugging and might appear due to the expansion valve malfunction, high refrigerant charge or when the evaporator load is very low (e.g. during nights). High amount of liquid is coming into the compression chamber also when the defrosting cycle of the refrigeration unit is turned on and the expansion valve is opened to let the hot gas through evaporator. When the liquid ingestion occurs in the time of continuous running of the compressor, it is reffered as "steady-state liquid slugging". (Laughman, Foy, Wichakool, Armstrong, & Leeb, 2008) At the start-up of the compressor there is another phenomenon happening, called flooded start. Liquid refrigerant might flow into the chamber when the compressor is turned off and refrigerant condensates into the oil or in the suction line. However, in some cases the liquid refrigerant can be beneficial to the performance of the compressor, when it is injected in controlled manner. This way the temperature of the discharge gas is reduced, which also brings the benefit of lower temperature load on the whole compressor. Such studies were performed recently on scroll compressor (Kim, Jeon, Jang, & Kim, 2018) and screw compressor (Wen et al., 2020). On the other hand, reciprocating compressors are much more sensitive to liquid inside the compression chamber due to compression gradient. (Laughman et al., 2008)
Due the incompressibility of the liquid refrigerant inside the compression chamber, liquid slugging is one of the most common reasons for compressor failure. Typical fatal faults are deformed (cracked) valves, damaged pistons, rods or crankshafts. Furthermore, the liquid flow will wash out the oil from lubricated surfaces, shortening the lifetime of the compressor. High amount of liquid in a compressor shell has negative effect when the compressor is heating up. The refrigerant will start to boil out from the oil, which again decreases the ability of lubrication and increases the amount of oil dragged out of the shell to the system. First investigations in this topic were carried out by (Singh, Nieter, & Prater, 1986), who measured peak pressure due to the liquid inside the cylinder chamber and performed simulation using different level of vapor-liquid mixture. Results clearly shows considerable pressure increase not only during the compression phase but also during discharge process. (Simpson & Lis, 1988) analysed different measurement methods to identify pressure peaks accurately, concluding that most common piezo-electric transducers are not suitable for measuring cylinder pressure during liquid ingestion. The recommendation is to use strain gauges. Further studies on liquid slugging detection in reciprocating compressors were performed by (Armstrong, Laughman, Leeb, & Norford, 2006) and (Laughman et al., 2008; Laughman,

Norford, & Leeb, 2006). In this case the method to detect the liquid injection was changed to motor characteristics. By measuring the electrical current flowing into the aggregate or compressor itself it was possible to identify the liquid slugging during both steady-state operation and start-up phase. The electrical parameters were compared to data from pressure sensors, showing good agreement in identifying slugging. Using electrical parameters to identify the slugging is very helpful for maintenance as all the data might be easily collected during the run time of the compressor, however it does not help to estimate the loads on crank-train system, nor valve failure risk. Mathematical models are useful in describing the effect of liquid slugging more precisely, allowing to predict failures in the compressors itself. First simulation model was already introduced by (Singh et al., 1986), who adjusted the model from (Elson & Soedel, 1974). Later on (Liu & Soedel, 1994) presented more comprehensive paper on the thermodynamic model used for slugging simulations. Following publication in the field of liquid slugging was published much later. (Richard & Lawley, 2011) presented a model for liquid – flooded compression in scroll compressors showing a possible efficiency increase by using liquid injection. Following analysis of (Ding, Jiang, Ne, & Ste, 2016) in CFD model showed the interaction of an oil and refrigerant in vapor state inside the scroll compressor. The results showed the decrease of the gas temperature by using different amount of oil, but also the increase of the pressure during compression. There are more papers focused on performance optimization of rotary compressors using liquid injection (Kim et al., 2018; Wen et al., 2020), however only few papers are focused on similar topic in reciprocating compressors. (Rodrigues, 2018) has studied liquid slugging inside the muffler of a reciprocating compressor. The aim was to evaluate the design influence on the amount of liquid coming through the muffler. The author concluded that with current setting there was no correlation between the amount of liquid injected and leaving the domain, however the geometry changes could retain the liquid in the muffler. The aim of this paper is to analyse different ways how to avoid liquid coming into the cylinder of a reciprocating compressor during the suction phase and also compare two different simulation methods for two-phase flow.

2. SIMULATION MODEL

The simulation of two-phase flow was performed on simplified geometry of the suction muffler in order to keep the computational requirements at reasonable level. Internal structure of the suction muffler was preserved, but the 3D model was simplified to 2D model only. Nevertheless, the geometry is detailed enough to keep the main flow paths of the liquid coming to the muffler. Decomposition of the model is shown in Figure 1.

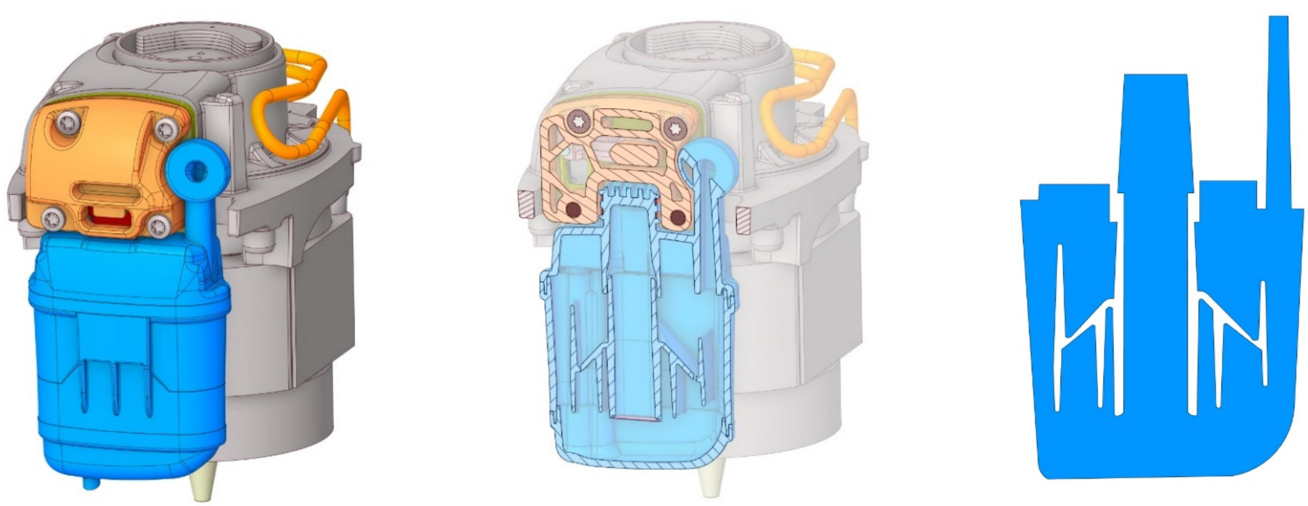


Figure 1: Compressor model

Except the simplified 2D geometry in Figure 1 there were three additional variants used in the simulation: model without the internal structure and two models with an additional feature at the inlet to the muffler in order to separate the liquid and gas phase on the way into the muffler. All additional variants are shown in Figure 2.

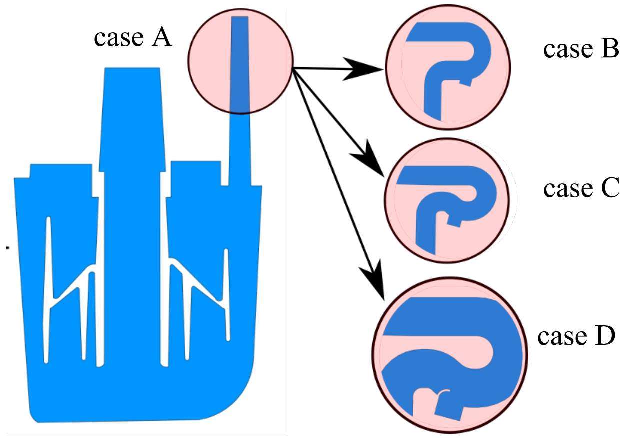


Figure 2: Design variants of inlet tube

2.1. Simulation settings

The boundary conditions are the key input for every simulation. In this case the formation of the liquid is considered to happen in the evaporator. Therefore, the liquid is coming through the connection pipeline into the muffler. This situation represents for example defrosting of the evaporator. This investigation was mainly focused on the processes inside the suction muffler. This area is highlighted by the red line in

Figure 3. This is important for the clarification of the boundary conditions in CFD simulations.

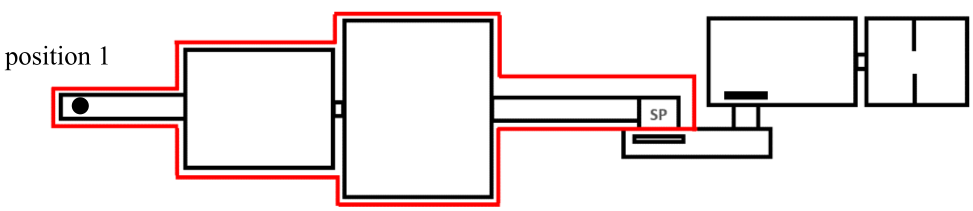


Figure 3: Compressor model scheme

During regular workload the conditions for a compressor are rather stable. At the entrance to the compressor (position 1) there is an evaporating pressure. On the other hand of highlighted area (SP – suction port) there is basically the cylinder pressure which changes with the respect to the compression chamber volume. The volumetric flow through the whole system can be assumed as constant for a given speed of the compressor, independent from evaporating and condensing pressure. During the steady-state liquid slugging, there is a significant change in the boundary conditions and the mass flow through the compressor is changed seriously as well. Another aspect to consider is the stability of the simulation. For all these reasons the boundary condition of the inlet (position 1) was set to the pressure, which is related to the evaporating pressure of the cooling circuit and at the outlet there was a normal velocity prescribed.
The simulation was performed using propane as a refrigerant, employing the Peng Robinson equation of state. The inlet pressure was set in accordance to the temperature of -25 *°C* and the velocity outlet was set to 2 *m/s*. The whole simulation lasts for 5 *s*, however the liquid is injected only in the first second of the simulation using the ratio of 80 *%* of the liquid and 20 *%* of the vapor. After this time the amount of liquid entering the domain is set to zero and only gas is entering through the inlet condition. Case A is analysed in order to have base value for the amount

of liquid coming to the suction muffler and exiting through the outlet condition. For cases B, C and D simple design change was created in order to separate the liquid and gas phase. In these three cases additional boundary condition was set up to represent the connection with the environment inside the shell of the compressor. Opening boundary condition was used in this case and the opening pressure was set to same value as at the inlet. Only gas phase could enter through this port during the simulation. Surface tension coefficient was obtained from Coolpack for given conditions. Turbulence model was set to Shear Stress Transport model and the flow was considered isothermal in this first simulation. Evaporation of the refrigerant will play a significant role in the suction muffler, but the simulation goal was to check the possibilities of mechanical separation of liquid and gas phase before entering the cylinder. For this reason, two different settings for two-phase flow modelling were tested. First simulation was conducted using VOF method with the flow field solved for each phase separately as well as the turbulence of each phase was dependant on the fluid phase. In the second approach the liquid refrigerant was defined as a dispersed fluid with a mean diameter of the droplets equal to 0.25 *mm*. The rest of the settings was kept the same as in the first approach and the simulations were performed on the same geometries.
For the initialization of the simulation there is zero velocity used in all directions and the pressure is set to the same level as at the inlet boundary condition. Inside the muffler there is no liquid refrigerant and the beginning of the simulation. The simulation domain is considered as quasi two-dimensional and for the front and back surface of the domain a symmetry condition was applied.

3. RESULTS AND DISCUSSION

Gradual flooding process of default geometry setup is shown in the Figure 5 and Figure 6. As it can be seen from the pictures, the suction muffler is quickly flooded by the incoming liquid and almost immediately the liquid is leaving through the outlet into the cylinder. Therefore, muffler design does not help to retain the liquid inside which would help to evaporate the liquid due to higher temperature of the walls of the muffler. Liquid injection is stopped in the simulation after 1s and the rest of the liquid is drawn towards the outlet. Figure 4 shows the mass flow of liquid phase through the outlet which drops shortly after the liquid injection is stopped. After approximately 1.3 *s*, the liquid flow through the outlet is almost negligible. Small amount of liquid is however trapped inside the muffler. Distribution of the liquid inside the suction muffler from 2 *s* up to the end of simulation (5 *s*) is basically not changing. As it was mentioned before, the simulation is set up to be isothermal, therefore the evaporating effect is not considered, which would help to eliminate the rest of the liquid in the muffler. Moreover, the vibration of the compressor would also help to disrupt the stable liquid in the muffler, helping the gas flow to drag the rest of it away.

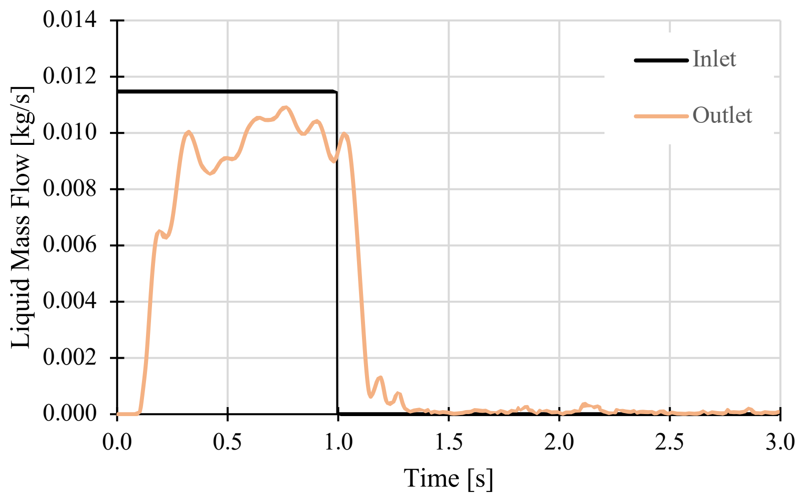


Figure 4: Liquid mass flow through outlet boundary of the suction muffler – Case A.

The simulation of Case B and Case C finished with very similar results in terms of liquid distribution inside the muffler. In both cases the inlet tube was extended for the stabilization of the flow before entering to separation feature. The opening boundary condition was set up at the separation bend and to the same pressure level as it was prescribed at the inlet. This approach should represent the pressure inside the cavity of a compressor. The amount of liquid coming into the suction muffler through the inlet was 30 – 40 *%* lower compared to Case A, which is also visible in Figure 6. After ca. 1.3 *s* the liquid flow through the outlet dropped almost to zero in the same way as in Case A and after 2 *s* the liquid distribution stayed consistent till the end of the simulation.

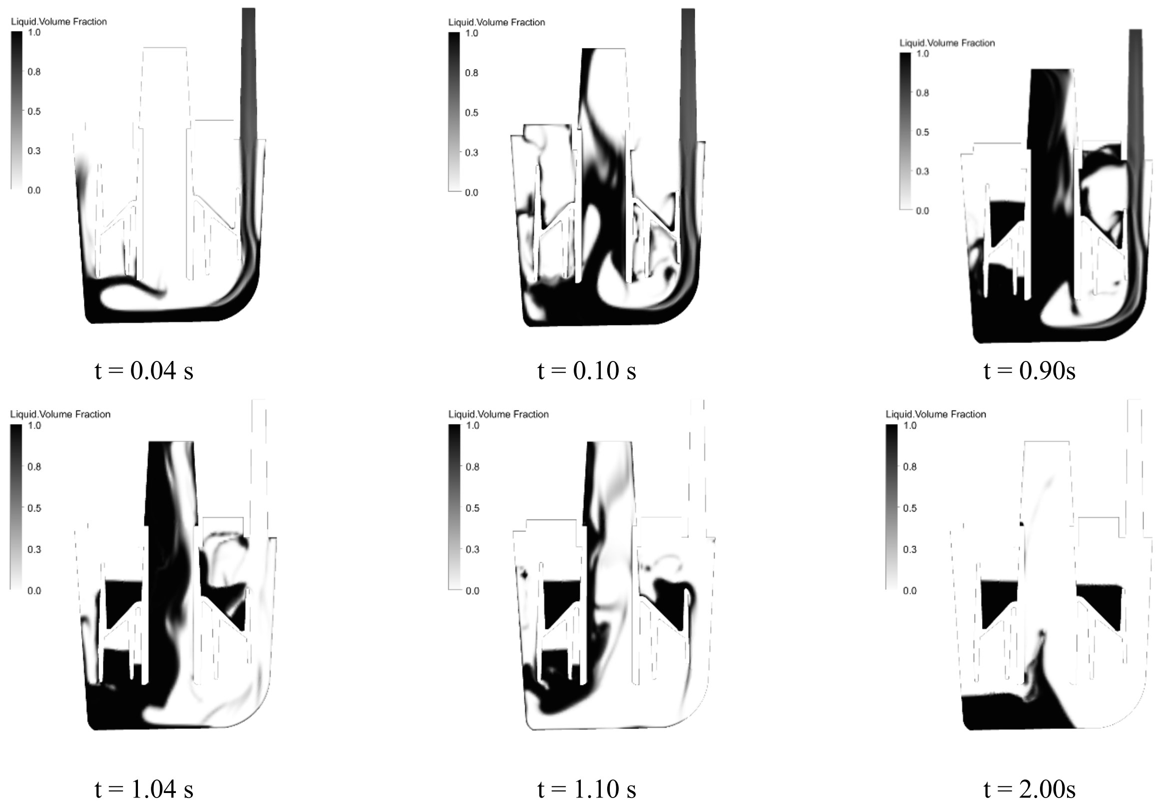
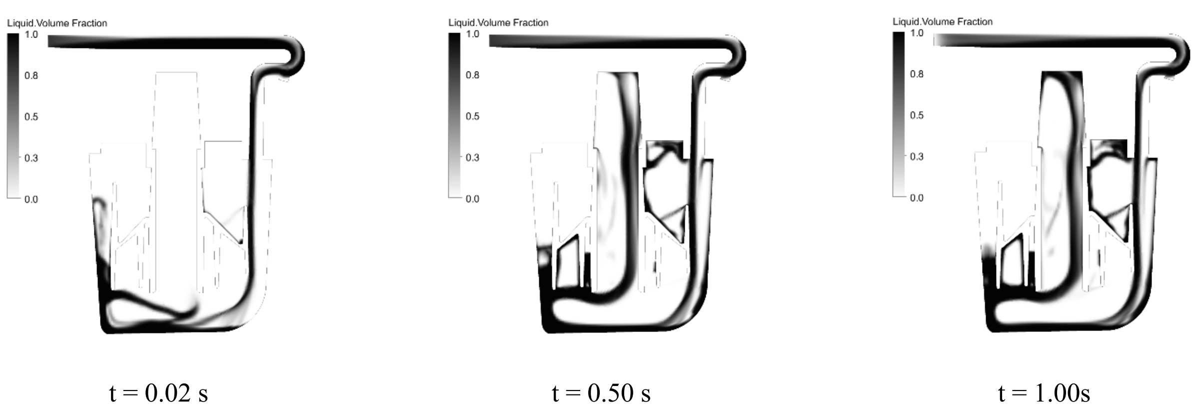
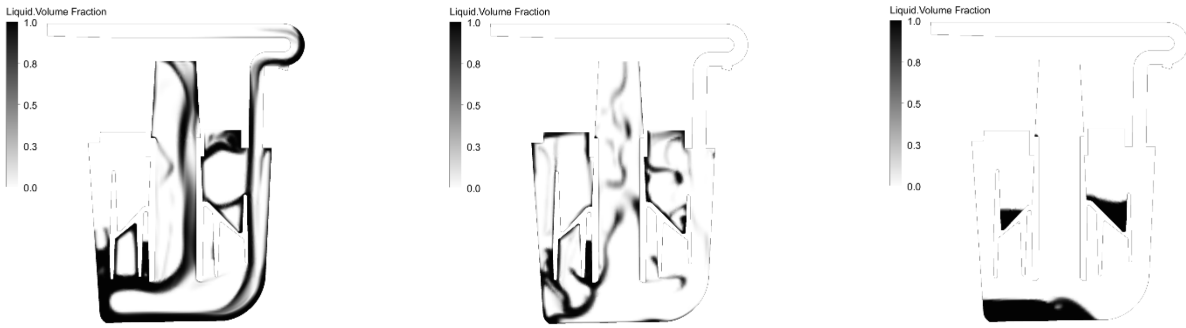


Figure 5: Liquid fraction in suction muffler in different time – Case A.

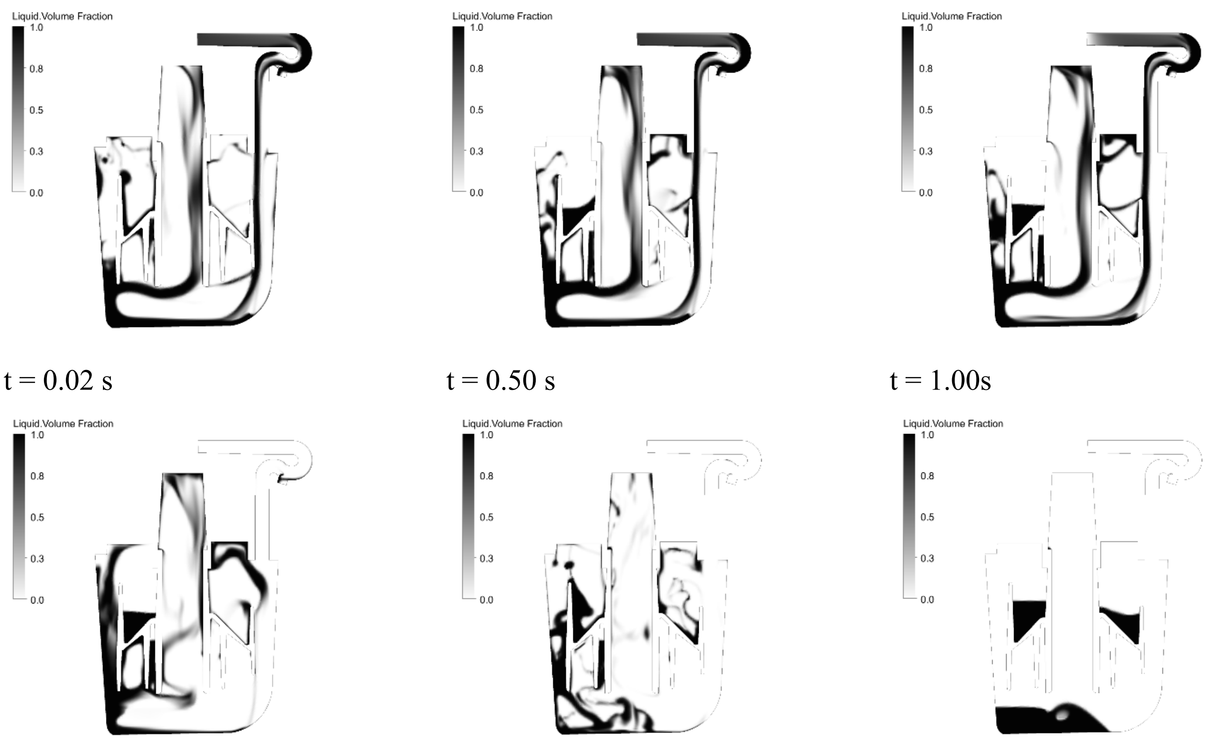




t = 1.02 s t = 1.10 s t = 5.00s

Figure 6: Liquid fraction in suction muffler in different time – Case C.

The effect of liquid separation was negligible. The distribution of liquid in Figure 6 at time 0.02 *s*, 0.5 *s* and 1 *s* shows that all the liquid is only passing over the separation exit and due to lower pressure it actually drags small amount of gas inside the muffler. This behaviour was observed on both cases, B and C. Additional feature in the separation bent was created to improve the liquid separation and drag bigger amount of liquid of the suction muffler. The results in Figure 7 show that this configuration performs better in reducing the amount of liquid inside the suction muffler, see Table 1.



t = 1.02 s t = 1.10 s t = 5.00s

Figure 7: Liquid fraction in suction muffler in different time – Case D.

The amount of liquid flowing into the suction muffler in case D was very similar to case A, however nearly 20 % of the incoming liquid was separated in case D, see Table 1. The rest of the liquid leaves the muffler in the same manner as in previous cases. The liquid fraction at the outlet drops to zero after 1.3 *s* in all cases, see Figure 8.

Table 1: Ratio of the liquid leaving from the suction muffler and being injected on the inlet.

	Case A	Case B	Case C	Case D
Liquid leaving through the outlet	100%	96%	96%	76%
Liquid leaving in separation point	0%	0.6%	0.2%	21%

The lowest volume fraction of liquid in Figure 8 is obtained for case C, but it is necessary to take into an account also the fact that the liquid mass flow was much lower in this case, see **Figure 9**.
Together with the VOF method for two phase flow there was also tested the approach using dispersed fluid as liquid refrigerant inside the suction muffler. In this case, it was necessary to set up the mean diameter of refrigerant droplets, which was prescribed to 0.25 *mm*. The rest of the parameters were kept the same as in the analysis using VOF method.

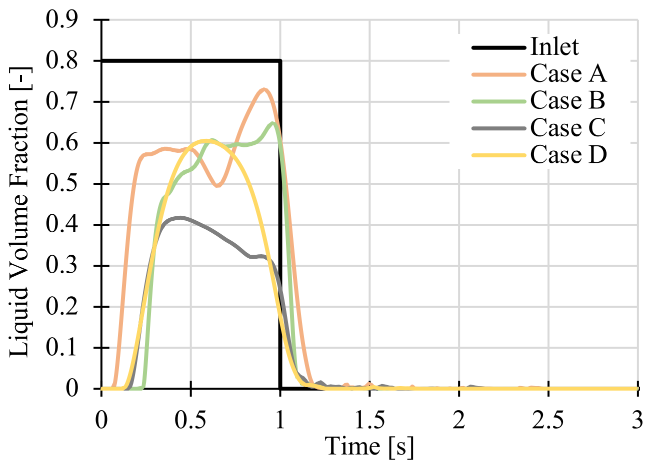


Figure 8: Liquid volume fraction at the outlet.

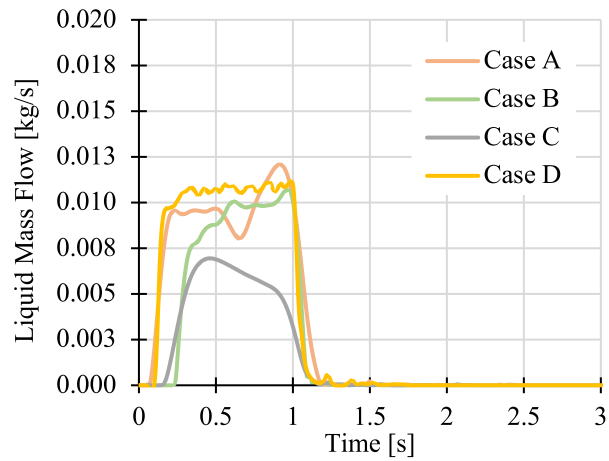


Figure 9: Liquid mass flow at the outlet.

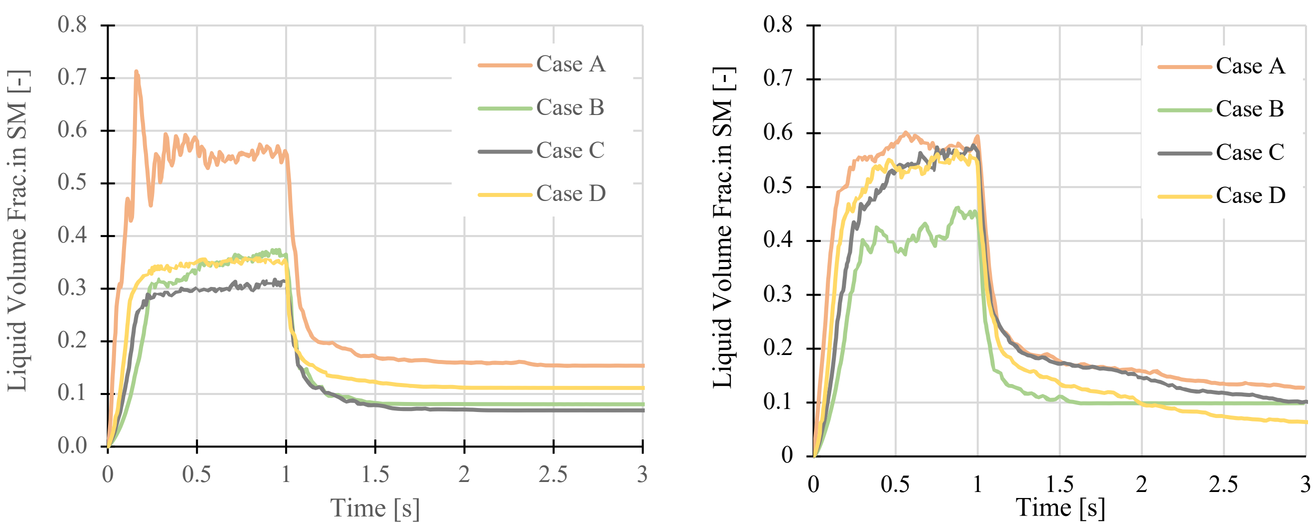


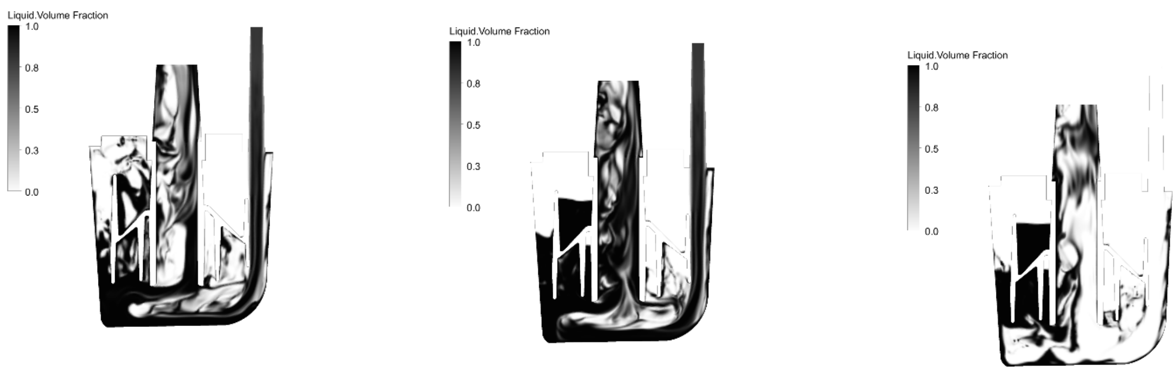
Figure 10: Liquid fraction in suction muffler for VOF method (left) and dispersed fluid method (right).

The results show that the liquid filling of the muffler is similar in both simulation approaches. Muffler is quickly filled with liquid (Figure 11 and Figure 12) and shortly the liquid injection is stopped, almost all of the liquid is dragged out of the muffler, see Figure 10. In the end of the simulation there is again a little amount of liquid trapped inside the muffler. The amount of liquid mass coming through the inlet is very similar in both simulation methods when configuration A is considered. The difference is only around 6 *%*. For configuration B, C and D the situation is quite opposite and there is more liquid mass coming into the muffler. Nevertheless, the effect of separation capability is the same for both simulation methods. As it is shown in Table 2, the most effective configuration is again configuration D, where 30 % of incoming liquid is separated from the inlet. Separation capabilities for B and C geometry are very small and results are like from VOF simulation.

Table 2: Ratio of the liquid leaving from the suction muffler and being injected on the inlet.

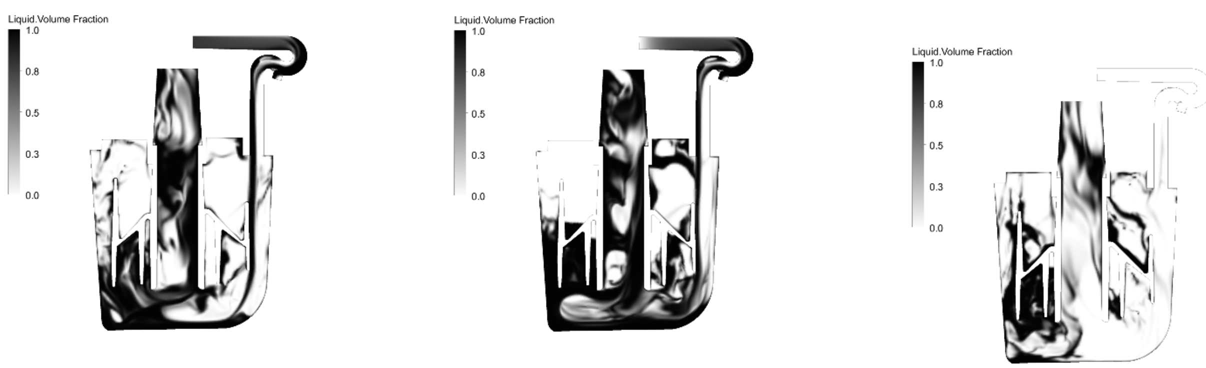
	Case A	Case B	Case C	Case D
Liquid leaving through the outlet	100%	93%	96%	68%
Liquid leaving in separation point	0%	1.83%	0.97%	31%

The liquid fraction at the outlet boundary has the same trend as in Figure 8. All the injected liquid is directly going to the outlet and further into the cylinder. Internal muffler geometry therefore does not reduce the liquid flow through the outlet or retain the liquid in the flow path, which would help to increase evaporation rate inside the muffler.



t = 0.2 s t = 0.80 s t = 1.10s

Figure 11: Liquid fraction in suction muffler in different time – Case A.



t = 0.2 s t = 1.00 s t = 1.10s

Figure 12: Liquid fraction in suction muffler in different time – Case D.

4. CONCLUSIONS

The simulation of two-phase flow inside the suction muffler was executed in the present study, using two different methods, VOF method and dispersed fluid method. The results from both of them are basically analogous, when the muffler separation capability is evaluated. Some geometry configurations with simple bending are not effective for separating the liquid inside the suction muffler. Moreover, the internal shape of the muffler does not help to retain the liquid inside it and all the fluid is dragged immediately towards the muffler outlet. This conclusion is however dependant on prescribed boundary conditions. In present study there was only one size of a mean droplet diameter used. Bigger droplets might be easier to separate in proposed configurations. Following studies should also take into the consideration the flow inside a reciprocating compressor, which is naturally pulsating. Current study showed that the flow can be very quickly stabilized which is not happening in a real situation. This would require detailed analysis of boundary conditions and experimental observation of the phenomenon.

ACKNOWLEDGEMENTS

The authors acknowledge Secop for the financial support.

REFERENCES

Armstrong, P. R., Laughman, C. R., Leeb, S. B., & Norford, L. K. (2006). Detection of rooftop cooling unit faults based on electrical measurements. *HVAC and R Research*, *12*(1), 151–175. https://doi.org/10.1080/10789669.2006.10391172

Ding, H., Jiang, Y., Ne, A., & Ste, A. (2016). CFD Simulation of An Oil Flooded Scroll Compressor Using VOF Approach * Corresponding Author 2 . CFD SOLVER AND GOVERNING EQUATIONS. *International Compressor Engineering, Refrigeration and Air Conditioning, and High Performance Buildings Conferences*, 1–8.

Elson, J. P., & Soedel, W. (1974). Simulation of the interaction of compressor valves with acoustic back pressures in long discharge lines. *Journal of Sound and Vibration*, *34*(2), 211–220. https://doi.org/10.1016/S0022-460X(74)80305-6

Kim, D., Jeon, Y., Jang, D. S., & Kim, Y. (2018). Performance comparison among two-phase, liquid, and vapor injection heat pumps with a scroll compressor using R410A. *Applied Thermal Engineering*, *137*(September 2017), 193–202. https://doi.org/10.1016/j.applthermaleng.2018.03.086

Laughman, C. R., Foy, R. R. L., Wichakool, W., Armstrong, P. R., & Leeb, S. B. (2008). *Electrical and Mechanical Methods for Detecting Liquid Slugging in Reciprocating Compressors*. (1986), 1–8.

Laughman, C. R., Norford, L. K., & Leeb, S. B. (2006). The Detection of Liquid Slugging Phenomena in Reciprocating Compressors via Power Measurements. *International Compressor Engineering Conference*, (1986), 1–8. Retrieved from https://docs.lib.purdue.edu/icec/1816/

Liu, Z., & Soedel, W. (1994). An investigation of compressor slugging problems. *International Compressor Engineering Conference*.

Richard, J. P., & Lawley, M. a. (2011). Theoretical and Experimental Analysis of Liquid Flooded Compression in Scroll Compressors. *Policy*, *9*(July 2010), 2003–2006. Retrieved from http://docs.lib.purdue.edu/dissertations/AAI3413897/

Rodrigues, T. T. (2018). *Refrigerant Liquid Slugging In The Suction System of Reciprocating Compressor*. (1994), 1–10.

Simpson, F., & Lis, G. (1988). Liquid Slugging Measurements in Reciprocating Compressors. *International Compressor Engineering Conference*, 479–487. Retrieved from https://docs.lib.purdue.edu/icec/663

Singh, R., Nieter, J. J., & Prater, G. (1986). *Prediction of Slugging-Induced Cylinder Overpressures*.

Wen, Q., Zhi, R., Wu, Y., Lei, B., Liu, S., & Shen, L. (2020). Performance optimization of a heat pump integrated with a single-screw refrigeration compressor with liquid refrigerant injection. *Energy*, *207*, 118197. https://doi.org/10.1016/j.energy.2020.118197