2018

Refrigerant Liquid Slugging In The suction System Of Reciprocating Compressor

Tadeu Tonheiro Rodrigues
Embraco - Research and Development, Brazil, tadeu.t.rodrigues@embraco.com

Sergio Koerich Lohn
Embraco, R&D, sergio.k.lohn@embraco.com

Refrigerant Liquid Slugging In The Suction System of Reciprocating Compressor

Tadeu Tonheiro RODRIGUES

Embraco, R&D
Joinville, Santa Catarina, Brazil
tadeu.t.rodrigues@embraco.com

ABSTRACT

Liquid slugging in reciprocating compressors is highly concerning due to the exceptionally harmful effects of liquid hammering. The high pressure generated by the liquid compression leads to high stresses on the structural components, resulting many times in compressor failure. The suction muffler is the mean path to the compression chamber for the slug flow, therefore its design influences the amount of liquid that reaches the compression chamber. This paper describes the numerical study of the two-phase liquid-gas flow in four typical simplified muffler designs employing VOF method, aiming to analyze the influence of the muffler design parameters on the liquid phase mass flow rate reaching the compression chamber. The main conclusions of the present work are the muffler design can influence significantly the flow towards the compression chamber and the numerical procedure can provide insightful information about the slug flow in the reciprocating compressors suction system.

1. INTRODUCTION

Compressors are designed to work with refrigerants in vapor state however, there are situations where return of liquid refrigerant or oil can occur resulting in wet compression, this phenomenon is referenced in the literature as liquid slugging (Danfoss, 2009a). The process of liquid compression can ~~exceeds~~ significantly typical cylinder pressures and the structural load the compressor was designed to support, resulting in critical failure of components like valves, connecting rods and crankshafts. Two typical situations of liquid slugging are: during the compressor steady state running operation, also known as refrigerant flood back (Danfoss, 2009b) or defrost (Simpson and Lis, 1988); and during compressor starting transient, known as flooded start (Danfoss, 2009c). Research about the subject of liquid slugging is very limited, therefore in order to contextualize the state of the art a careful bibliography review was performed.

The first relevant works on the field were published in two complementary papers by Sing *et al*. (1986), which conducted preliminary experiments to observe the cylinder pressure during liquid suction and develop simplified models for the wet compression in the cylinder, aiming to study the main influencing parameters. The authors confirmed the decreasing inlet quality of the suctioned mixture increases the maximum pressure peak during compression, also the abrupt increase of pressure is greatly correlated with the discharge process of the incompressible flow through the ports.

Simpson and Lis (1988) evaluated several instrumentation procedures aiming to accurately identify the cylinder chamber pressure. It was found that typical pressure transducers provide inaccurate pressure data, values far above the compressor operation limits being observed and the sample could not last the tests duration. The most promising methods were the application of strain gauges in the suction valves, pressure strain gauges in the cylinder and strain gauges in the main bearing of the crankshaft, this last one provides indirectly the cylinder pressures through the monitoring of the load transmitted though the piston and connecting rod to the main shaft bearing. The authors advocate the use of strain gauge on the main bearing once the instrumentation process is simpler and lasts more than on the valve system and cylinder instrumentation procedures. They employed the method in several tests, varying the refrigerant charge to observe the correlation with the cylinder pressure, obtaining an empirical correlation for the maximum cylinder pressure based on the compressor bore, stroke and nominal motor horse power.

Liu and Soedel (1994) developed a numerical model to simulate the liquid compression in a rolling piston compressor to study the effects of wet compression. The model was an extension of traditional compressor simplified models for performance evaluation with the inclusion of the state equations for the two-phase mixture of vapor and liquid. Numerical experiments where performed to evaluate the effect of the inlet quality in the suction system on the compression process. Similarly, to previous works, the quality reduction in the suctioned mixture implies increase of the pressure peaks, but additionally the model predicts the evolution of the quality during the compression process. High mixture qualities can develop in superheated vapor during the compression process, but low qualities can result in saturated liquid, which is a situation that must be prevented. Based on the interpretation of the equations of the model, the authors found that one of the main factors in the pressure peaks is the volume compression gradient, the higher the volume compression gradient, the higher the pressure peaks for liquid slugging. This information reveals reciprocating compressors are most probable to facing liquid compression, followed by rolling piston and scrolls compressor, once they have compression and discharge processes of 180, 360 and 540 degrees of rotation, respectively.

Shiva Prasad (2002) developed a simplified numerical model to predict the maximum loads in large reciprocating compressors used for pumping natural gas from reservoirs, where the liquid slugging occurs due the presence of water. Simulations were carried out to evaluate the correlation of the pressure increase with the increase of water volume suctioned. The empirical relation obtained by Simpson and Lis (1988) was used for reference purpose and, despite based on a different compressor and application, the author found relative good agreement.

Laughman *et al*. (2006) developed fault detection methods in compressor during field operation due to liquid slugging. The oscillations in the compressor shaft power due to the liquid slugging reflect in the electric motor, therefore monitoring the patterns of the input electrical parameters leads to the possibility to detect liquid slugging. The liquid flood back and flood start conditions were tested, monitoring the instantaneous motor power and varying the quantity of liquid through controlled injection. Similar to the previous works, it was found significant influence in the input power due to the liquid injection, with an increase of the necessary input power with the increase of the amount of liquid injected. In a complementary paper, Laughman *et al*. (2008) aimed to validate the previous work monitoring, together with the electrical parameters, the mechanical quantities of cylinder pressure, discharge pressure and the instantaneous rotation speed for liquid flood back condition and varying the liquid injection period.

Ding and Jiang (2016) performed Computational Fluid Dynamics (CFD) simulations of the oil flooded process in scroll compressors. The authors simulated the unsteady three-dimensional process using the Volume of Fluid (VOF) method for modelling the interactions of the refrigerant in vapor state and the liquid oil. The main purpose was to predict the reduction in the cylinder chamber temperature through (CFD) simulation by varying the amount of injected oil, nevertheless the author also provided the cylinder pressure diagrams where can be verified the cylinder pressure increase during the discharge process due to the liquid compression.

One can observe from the bibliographic review limited development on the liquid slugging subject, most of the works focusing on the cylinder phenomena by using simplified models or instrumented tests. The amount of liquid is controlled by injection, disregarding the study on the suction process of the two-phase mixture which can influence the amount of liquid that reaches the cylinder chamber. The main purpose of this work is to evaluate the suction process of a two-phase refrigerant mixture in the suction muffler of a typical reciprocating compressor, for this the suction muffler was simplified for a two-dimensional model and the unsteady two-phase flow simulated through the VOF method implemented in ANSYS CFX® software. For additional insight about the phenomena, four typical muffler designs were used and the inlet mass flow liquid injection varied.

2. NUMERICAL PROCEDURE

2.1 Geometry Simplification

The physical domain simplification is illustrated in Figure 1, the full three dimensional model of a compressor is limited to the suction region, the main section of the suction muffler is extracted to build the two dimensional domain, which includes the internal cavity and suction tube attached to the shell. Details and curvatures, as the inlet nozzle, were eliminated to generate a simple model that preservers the main features which are: internal volume, tubes diameter and length, and the relative position among the suction system components.

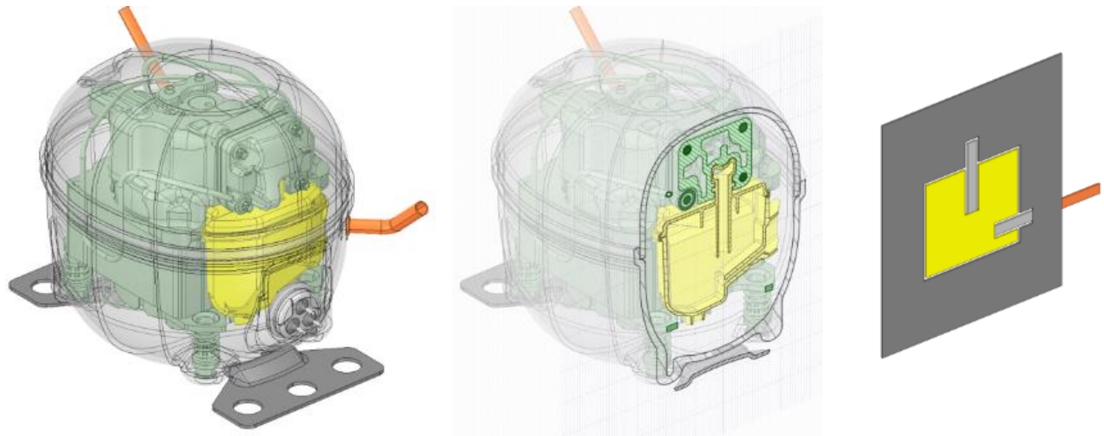


Figure 1: Simplification procedure, from three to two-dimensional domain.

2.2 Numerical Model.

The unsteady, compressible, turbulent and two-phase fluid flow through the simplified suction system was solved using the commercial code ANSYS CFX® release 18, based on the element-based finite volume method to discretize the partial differential equations of the mass conservation, momentum, energy and turbulence.

Advection terms were discretized using a High Resolution Scheme and the temporal discretization obtained by a second order backward Euler scheme. The Shear Stress Transport model was set to bridge the turbulent effects to the mean flow, which has been proven as a suitable model for compressor simulation (Rodrigues and Da Silva, 2014). The resultant system of equations is solved through Incomplete LU (ILU) decomposition, algebraic multigrid method and coupled strategy. The pressure evaluation correlation is obtained from the Redlich-Kwong library, available in the numerical tool.

The Volume of Fluid (VOF) method is used to treat the two-phase flow and belongs to the class of Eulerian methods, which basically employs an additional transport equation for the volume fraction that distinguishes the phases in the mesh domain, the interface is obtained using algorithms for tracking and reconstruction. The VOF method is suitable for non-disperse flows, e.g. free surface flows, which are considered similar to the present problem.

The two-phase flow was considered with buoyancy and gravity effects, the surface tension necessary for the liquid cohesion was obtained from the NIST data base for the fluid simulated.

The computational domain was discretized in quadrilaterals observing orthogonality quality parameter; the mesh refinement was set to higher resolution in the muffler, the suction tube and the internal cavity region between both, where is the mean flow. Two levels of mesh refinement were used in this study, resulting in mesh sizes of 43000 and 80000 nodes, an illustration scheme of the mesh used for discretization is presented in Figure 2. One can note a homogeneous and high quality mesh, the domain simplification was intended to generate a good mesh to obtain convergence and also good resolution in the phases interface, a well know requirement for two-phase flow simulation with VOF method.

At this point important considerations are presented, related to the numerical model limitations. The flow is assumed isothermal, so the evaporating and cavitation process that occurs during the liquid flow are disregarded. The evaporating process in the internal cavity modify the suction pressure and consequently the compressor mass flow, this phenomena is not considered and the pressure and mass flow boundary conditions are maintained constant. Despite the hypothesis and methodology limitations in this preliminary study, it is expected the simulations will provide insightful aspects of the flow and a conservative approach of the phenomena, once the obtained liquid mass flow rate through the muffler will be higher than in real process.

Boundary conditions are provided in Figure 2. At the suction tube liquid velocity is prescribed, at the suction muffler outlet bulk mass flow rate correspondent to operating condition, at the cavity bottom the domain is set as outlet condition with suction pressure to eliminate the excess of rejected liquid. The last condition was necessary once the absence of three-dimensional domain the volume available at the bottom is insufficient; in preliminary simulation, the rejected liquid flooded the internal cavity up to the muffler inlet. To force two-dimensional effect it was applied symmetry condition at the front and back faces of the domain. The simulation is initialize with suction pressure of the operating condition, zero flow and gas refrigerant, i.e., liquid volume fraction of zero.

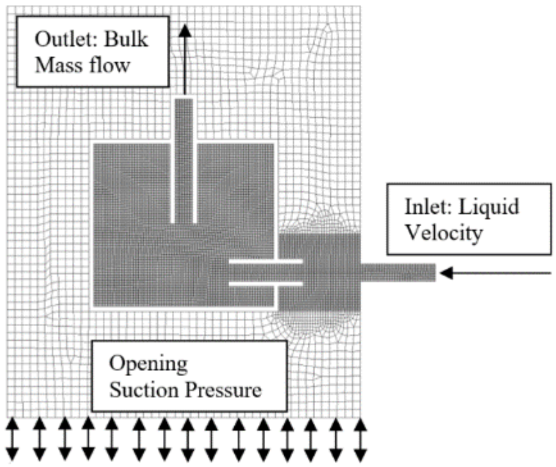


Figure 2: Mesh and boundary conditions used for simulations.

2.3 Geometries and slugging conditions evaluated.

Four typical muffler configurations were evaluated, based on actual mufflers designs. For comparison purposes, features like tube diameters, external geometry profile and compressor cavity were kept the same. Figure 3 illustrates the geometries investigated, two with one internal volume and two with internal division for two volumes.

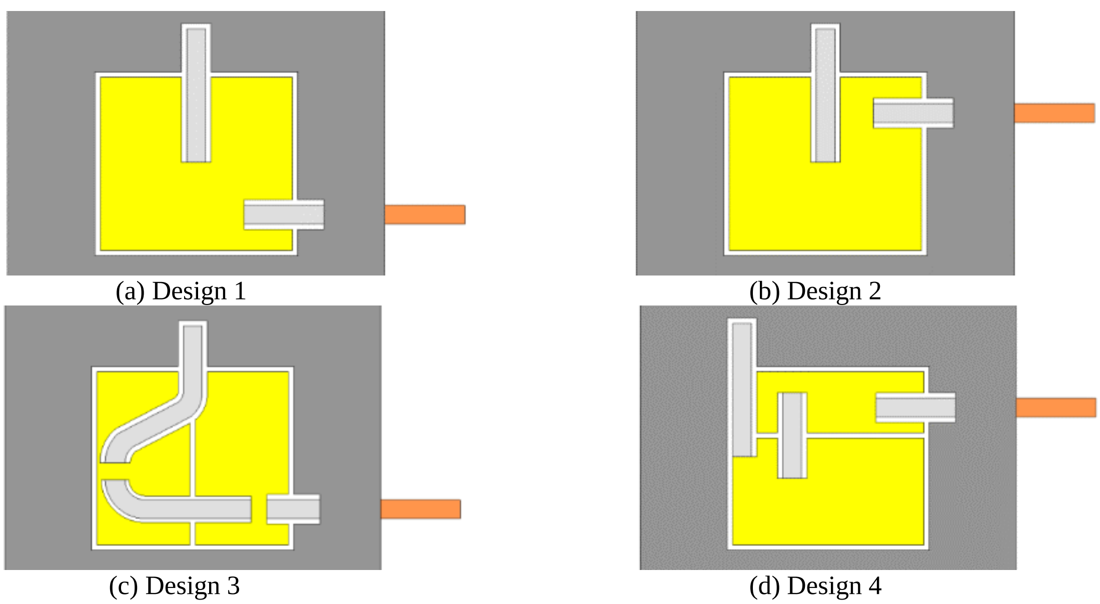


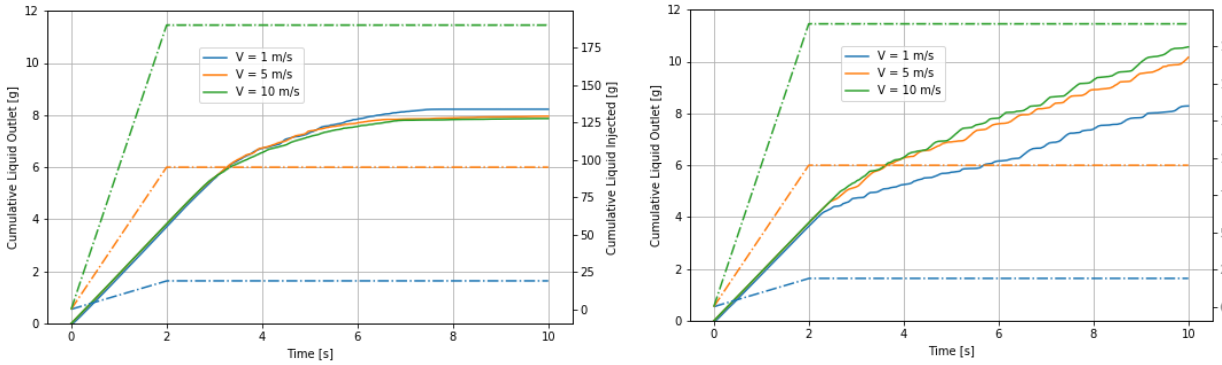
Figure 3: Muffler designs investigated.

The liquid slugging was simulated in physical process of 10 s, were 2 s were taken to liquid injection and the remainder for observation of the muffler emptying process. The amount of liquid injected was evaluated by varying the injection velocity in 1, 5 and 10 m/s, which considerers a constant supply of liquid refrigerant. It was considered as refrigerant R134a, which is currently one of the refrigerants with highest probability to occur liquid slugging, due to limitations of charge for hydrocarbons. Nevertheless, the fluid refrigerant applied does not limit the conclusions of the present study.

3. RESULS

3.1 Effect of mesh refinement.

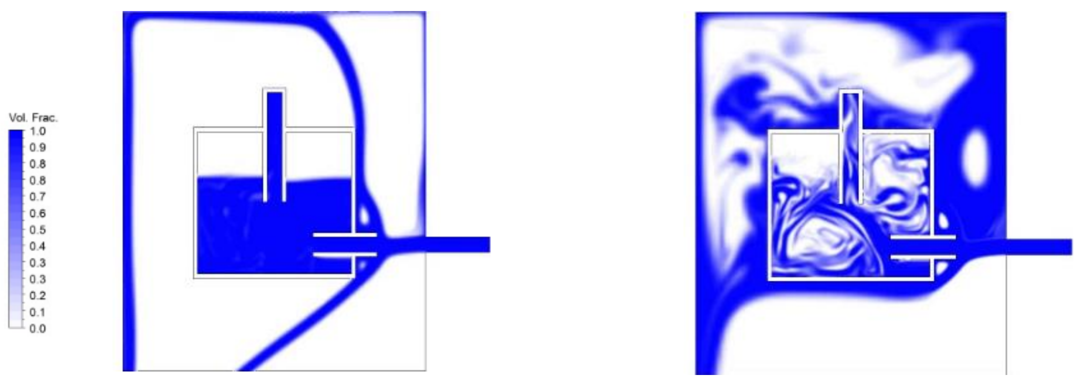
The mesh refinement evaluation was performed first to select a mesh basis for the full study. Comparison results for cumulative outlet liquid mass are presented in Figure 4, for the Design 1 only. The reason for the cumulative outlet mass to be chosen is due to the high oscillations observed in the mass flow rate, making difficult the interpretation.



(a) Mesh 1. (b) Mesh 2.

Figure 4: Cumulative outlet liquid mesh refinament results comparison, for Design1.

One can note Mesh 1 presents little difference among the three liquid injection velocities, for Mesh 2 differences are prominent mainly after the liquid injection ceases. The visualization of the liquid volume fraction in Figure 5 at the end of the injection process for inlet liquid velocity of 5 m/s illustrates the root cause. The VOF requires fine mesh refinement to adequately capture interfaces between phases, Mesh 1 was not able capture the turbulent interaction between the gas and liquid, which results in several gas bubbles inside the muffler during the filling process. Despite the results obtained for Design 1, simulations for the other designs were carried with both mesh refinements and the same conclusion was obtained, in the next section only results with Mesh 2 are fully present.



(a) Mesh 1. (b) Mesh 2.

Figure 5: Vizualition comparison of liquid fraction contours results at end of injection process, Design1.

3.2 Results for design effect on the liquid slugging.

The liquid slugging results in the suction muffler are presented in two graphical forms for time sequence: cumulative mass data for liquid injected (dashed lines) overlapped with cumulative outlet liquid mass (solid lines), and the muffler volume fraction with liquid. Results are presented in Figures 6 to 9, for muffler Designs 1 to 4, respectively.

In general, observing the cumulative outlet liquid, much little difference is found by varying the inlet liquid velocity or the muffler design during the injection process, this is basically due the bulk outlet mass flow rate boundary condition, which forces the liquid towards the outlet at the same rate imposed. The effect of the muffler design and the liquid inlet velocity can be, then, observed using the muffler volume fraction with liquid and the emptying process through the cumulative outlet liquid mass.

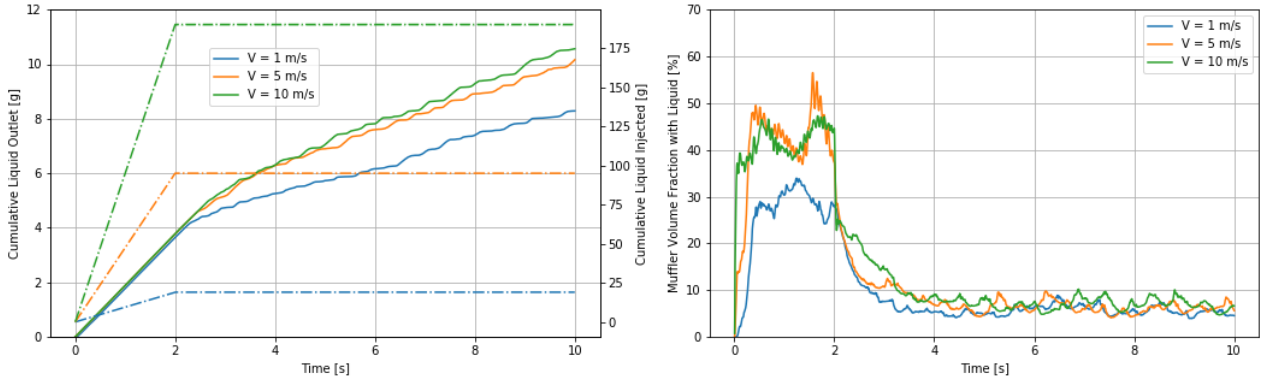


Figure 6: Liquid slugging results for muffler Design 1.

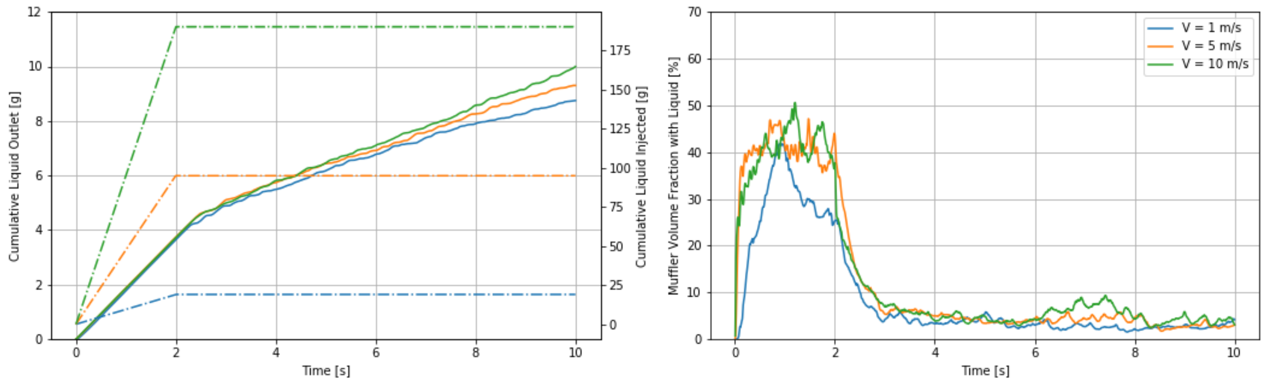


Figure 7: Liquid slugging results for muffler Design 2.

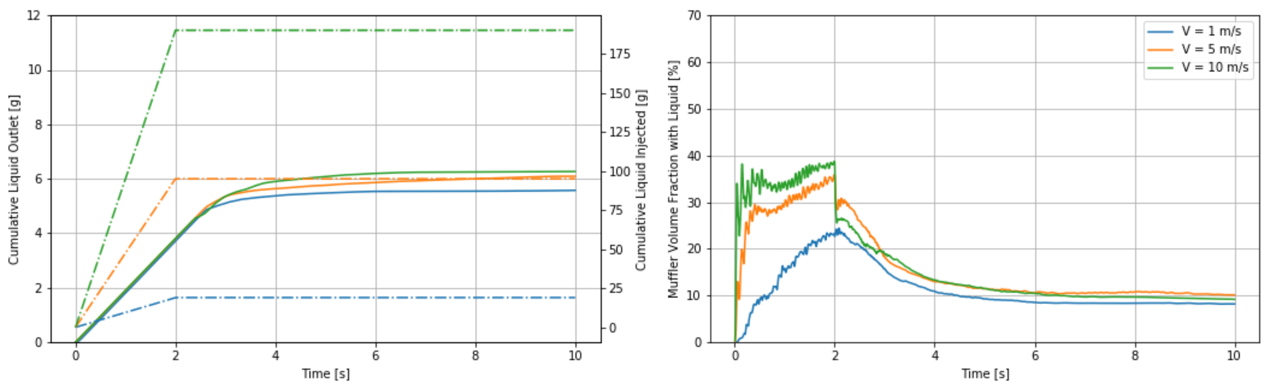


Figure 8: Liquid slugging results for muffler Design 3.

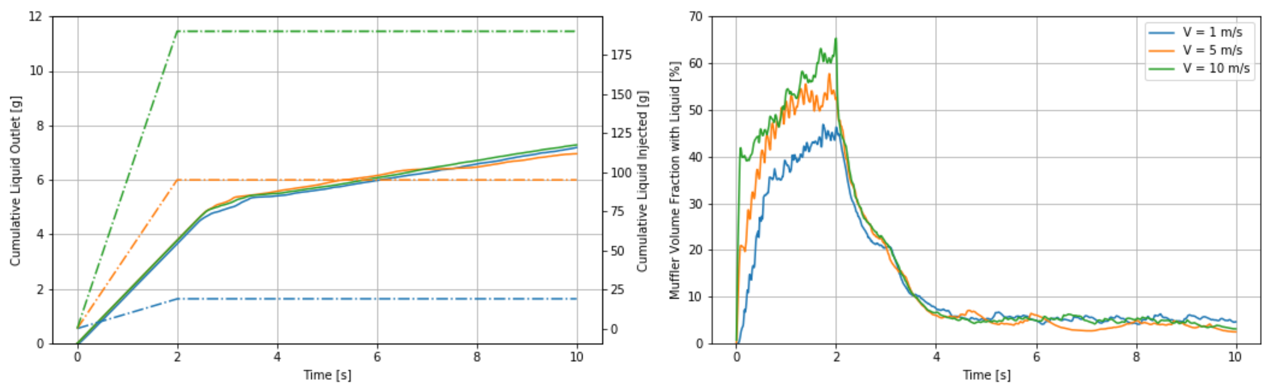


Figure 9: Liquid slugging results for muffler Design 4.

Direct results observation points Design 3 is the layout with the smallest capability to drive liquid to the cylinder, which contradicts the natural sense, the liquid contention capability interpreted by the muffler volume fraction with liquid also is the smallest. Design 4 is the second in terms of reduced outlet liquid mass leaving the muffler, but with the highest level of liquid containment. In both designs, little difference is found in the accumulated outlet liquid mass due the injected liquid velocity, it is observed difference in the liquid volume fraction between injection velocities of 1 m/s and 5 m/s, with higher liquid accumulated for higher injected velocity, however no significant addition is observed for injection velocity of 10 m/s.

Design 1 and 2 are similar in terms of outlet accumulated liquid mass for all injection liquid velocities, except for velocity 1m/s. Both designs delivers more liquid to the cylinder comparing to Designs 3 and 4, which also translates into smaller levels of muffler volume fraction with liquid. Similar to Designs 3 and 4, there is small difference between results for injection velocities of 5 m/s and 10 m/s.

In order to explain the quantitative data behavior, the visualization of the injection process was performed through contours of liquid volume fraction, provided for each design at times 2 s and 3 s. The contours are presented for Design 1 in Figures 10 and 11, for Design 2 in Figures 12 and 13, for Design 3 in Figures 14 and 15, and for Design 4 in Figures 16 and 17.

(a) V = 1 m/s. (b) V = 5m/s. (c) V = 10 m/s.

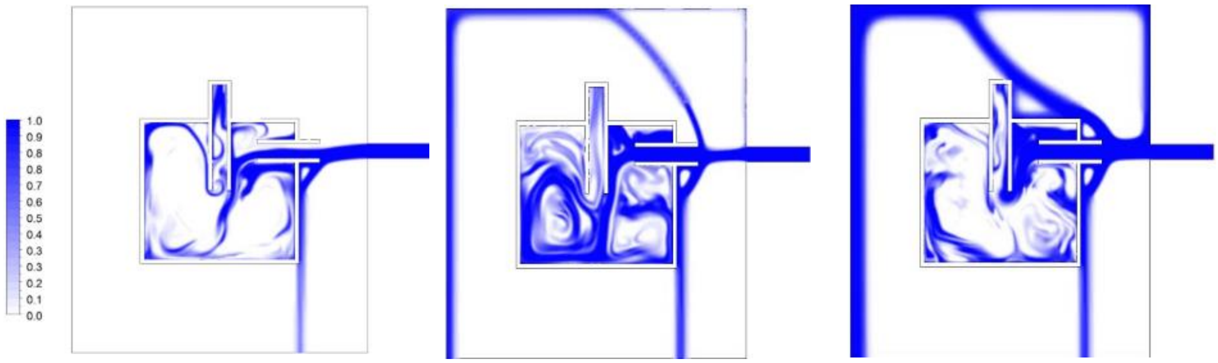
Figure 10: Liquid volume fraction visualization for Design 1, time 2 s.

(a) V = 1 m/s. (b) V = 5m/s. (c) V = 10 m/s.

Figure 11: Liquid volume fraction visualization for Design 1, time 3 s.

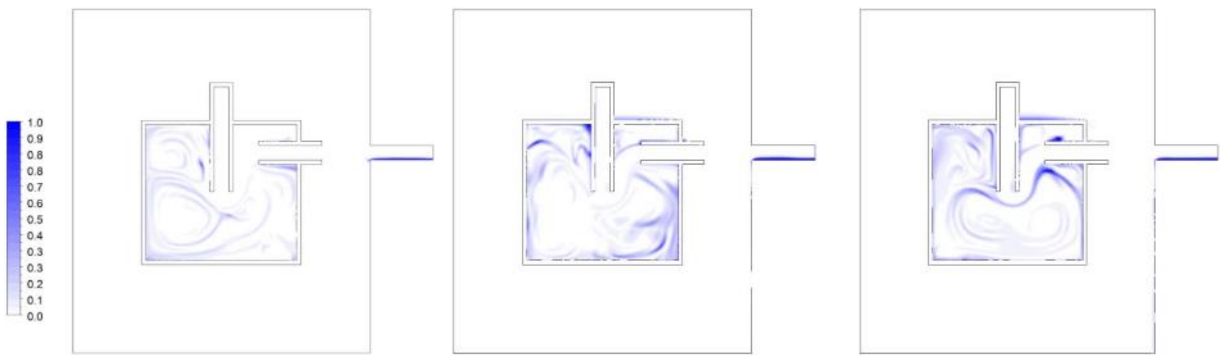
Liquid volume fraction contours for Design 1 evidences at a first moment, a high level of liquid rejected to the internal compressor cavity, associated to entrance effects at the muffler inlet tube and the resistance caused by the liquid already inside the suction muffler. The rejection is higher for higher injection liquid velocities, due to increased turbulence, justifying the absence of linear correlation with the monitored data.

The emptying process is quite similar for all injection velocities in Design1, the suction mass flow rate is able to reduce substantially fast the quantity of liquid inside the muffler. The liquid is carried by the gas flow, which drags and disperses the liquid inside the muffler; one can observe the geometry details can trap the liquid inside extending the emptying process.



(a) V = 1 m/s. (b) V = 5m/s. (c) V = 10 m/s.

Figure 12: Liquid volume fraction visualization for Design 2, time 2 s.

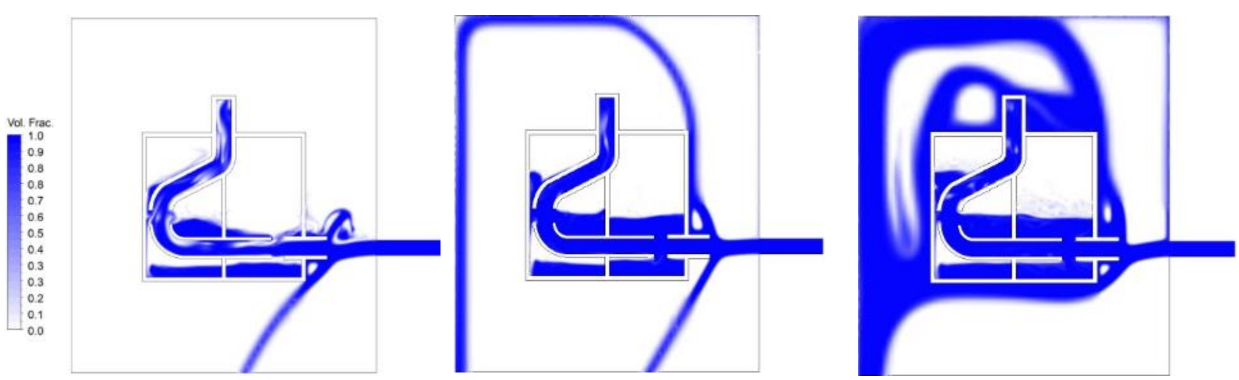


(a) V = 1 m/s. (b) V = 5m/s. (c) V = 10 m/s.

Figure 13: Liquid volume fraction visualization for Design 2, time 3 s.

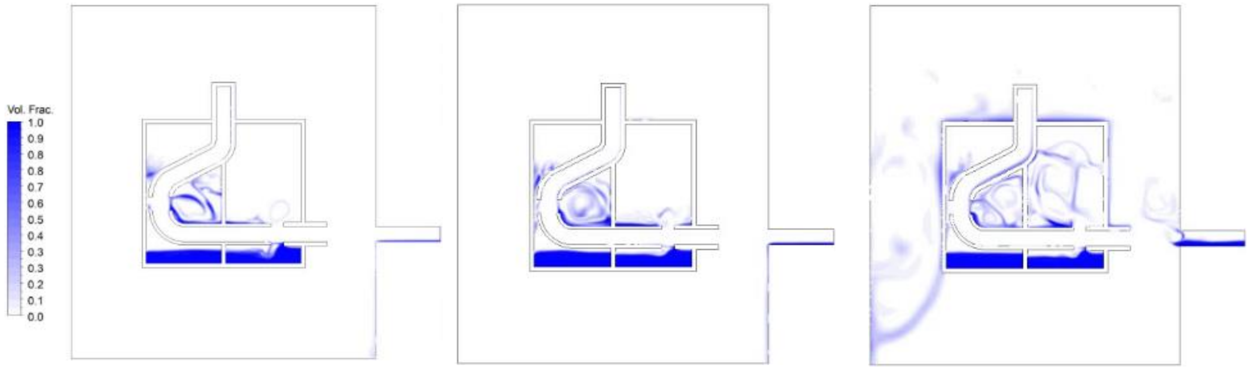
Design 2 presents very different flow patterns for the liquid compared to Design 1, mainly because of the obstacle imposed by the outlet muffler tube. Nevertheless, the outlet tube also acts as path, the liquid attaches to the wall and descends being captured by the suction flow imposed at the outlet.

The liquid is highly dispersed at 3s seconds of simulation, compared to Design 1, although in terms of muffler volume fraction with liquid both designs presents similar data. Considering the refrigerant volatility, the small quantity of liquid and high dispersion due the turbulent flow, the evaporation would have turn all the liquid into vapor at this point of the emptying process, for both Designs 1 and 2.



(a) V = 1 m/s. (b) V = 5m/s. (c) V = 10 m/s.

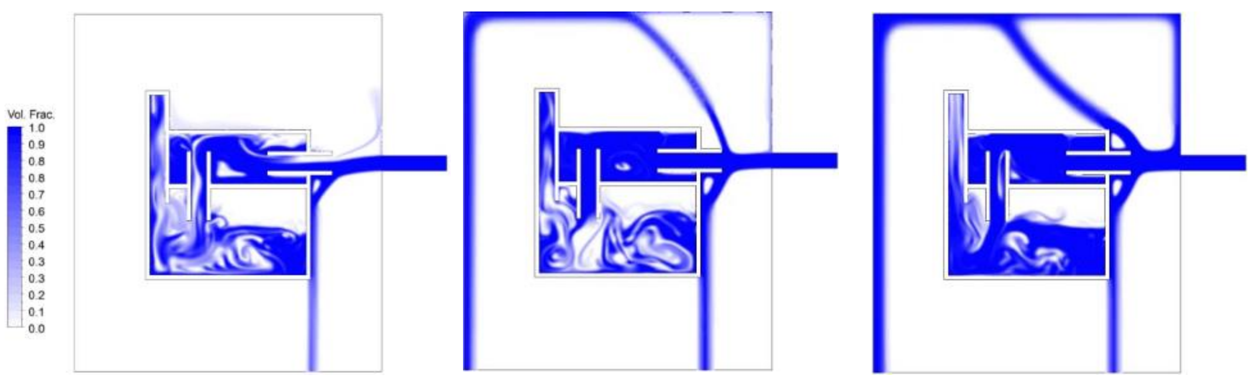
Figure 14: Liquid volume fraction visualization for Design 3, time 2 s.



(a) V = 1 m/s. (b) V = 5m/s. (c) V = 10 m/s.

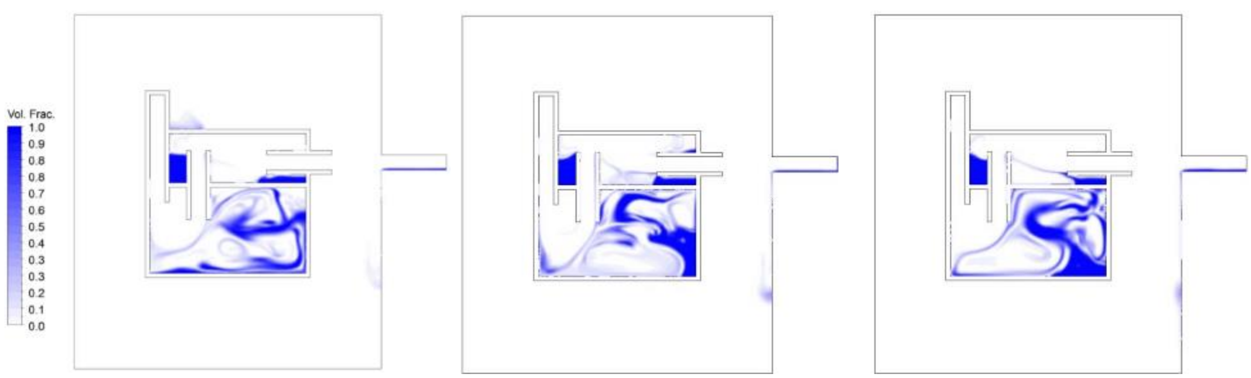
Figure 15: Liquid volume fraction visualization for Design 3, time 3 s.

The internal tubes layout of Design 3 provides a direct path to cylinder, as observed by the significant and fast filling of the tubes with liquid. The liquid already inside the tubes causes resistance to the additional incoming liquid, resulting in a high level of rejected liquid. Design 3 can be interpreted as the most effective to conduct liquid to the cylinder during the injection process, aligned with natural sense, only limited by the compressor capability to suction the liquid, represented in the present study by the outlet boundary condition. However, Design 3 presents a resistance to accumulate liquid as well to emptying the muffler, due to the small connections with the tubes, one can expect the emptying process will happen probably through liquid evaporation in this design.



(a) V = 1 m/s. (b) V = 5m/s. (c) V = 10 m/s.

Figure 16: Liquid volume fraction visualization for Design 4, time 2 s.



(a) V = 1 m/s. (b) V = 5m/s. (c) V = 10 m/s.

Figure 17: Liquid volume fraction visualization for Design 4, time 3 s.

Design 4 analysis of the liquid fraction contours is the most direct. The internal tubes and barriers dispositions are restrictions to the liquid flow, the first volume retains most of the liquid, delivering slowly to the second volume and then to the outlet tube, as observed there is a significant amount of liquid present at 3s compared to previous cases.

4. CONCLUSIONS

In the present study a numerical investigation of the liquid slugging at the suction muffler of a reciprocating compressor was performed, employing the VOF method to simulate the two-phase flow of gas-liquid refrigerant. Design influence was evaluated in four typical simplified muffler designs, the influence of the amount of liquid injected was verified by varying the velocity of liquid injected. The main conclusions from this investigation are:

- There is no direct correlation between the outlet liquid mass and the amount of injected liquid. It was observed significant liquid rejection due to turbulence effects at the muffler inlet and resistance caused by the liquid already in the muffler.
- Muffler design can influence significantly the retention of liquid due to restriction in the flow path, reduce restrictions can orient directly the liquid towards the cylinder chamber.
- VOF method is suitable to evaluate design through a conservative manner, nevertheless a high level of liquid dispersion was observed, future works should include evaluation of models with transition for disperse flows and evaporation.

REFERENCES

DANFOSS. (2009). Why Compressors Fail: Part 1 - Refrigerant Flood Back, Field Service Note 006. Denmark.

DANFOSS. (2009). Why Compressors Fail: Part 2 - Flooded Starts, Field Service Note 007. Denmark.

DANFOSS. (2009). Why Compressors Fail: Part 3 - Liquid Slugging, Field Service Note 008. Denmark.

Ding, Hui & Jiang, Yu. (2016). CFD Simulation of An Oil Flooded Scroll Compressor Using VOF Approach. *Proceedings of the 23nd International Compressor Engineering Conference at Purdue* (Paper 1616). West Lafayette, USA: Purdue University.

Laughman, Christopher R.; Armstrong, Peter R.; Norford, Leslie K.; & Leeb, Steven B. (2006). The Detection of Liquid Slugging Phenomena in Reciprocating Compressors via Power Measurements. *Proceedings of 2006 International Compressor Engineering Conference* (Paper C084). West Lafayette, USA: Purdue University.

Laughman, Christopher R.; Foy, Roderick R. Lal; Wichakool, Warit; Armstrong, Peter R.; Leeb, Steven B.; Norford, Leslie K.; Rodriguez, John; Goebel, Kai; Patterson-Hine, Ann; & Lupton, E. C. (2008). Electrical and Mechanical Methods for Detecting Liquid Slugging in Reciprocating Compressors. *Proceedings of 2008 International Compressor Engineering Conference* (Paper 1437). West Lafayette, USA: Purdue University.

Liu, Z. & Soedel, W. (1994). An Investigation of Compressor Slugging Problems. *Proceedings of 1994 International Compressor Engineering Conference* (Paper 1017). West Lafayette, USA: Purdue University.

Singh, R., Nieter, J.J. & G. Prater, Jr. (1986). An Investigation of the Compressor Slugging Phenomenon. *ASHRAE Transactions, Vol. 92* (1), 250-258.

Singh, R.; Nieter, J. J.; & Prater, G. (1986). Prediction of Slugging-Induced Cylinder Overpressures. *Proceedings of 1986 International Compressor Engineering Conference* (Paper 544). West Lafayette, USA: Purdue University.

Simpson, Fran and Lis, Gordon. (1988). Liquid Slugging Measurements in Reciprocating Compressor. *Proceedings of 1986 International Compressor Engineering Conference* (Paper 663). West Lafayette, USA: Purdue University.

Rodrigues, T. T., Da Silva, D. L. (2014). Turbulence Modelling Evaluation for Reciprocating Compressor Simulation. *Proceedings of the 22nd International Compressor Engineering Conference at Purdue* (Paper 1121). West Lafayette, USA: Purdue University.

ACKNOWLEDGEMENT

The authors acknowledge the EMBRACO technical and financial support.