Numerical-experimental investigation of liquid slugging in the suction muffler of a hermetic reciprocating compressor

M Bianchi\textsuperscript{1}, C J Deschamps\textsuperscript{1}, T T Rodrigues\textsuperscript{2} and E E Paladino\textsuperscript{3}

\textsuperscript{1}POLO Research Labs, UFSC, Florianopolis, Brazil
\textsuperscript{2}R&D, Nidec-GA, Joinville, Brazil
\textsuperscript{3}SINMEC Computational Fluid Dynamics Lab, UFSC, Florianopolis, Brazil

deschamps@polo.ufsc.br

Abstract. Refrigerant in the liquid phase and lubricating oil that circulate in refrigeration systems may reach the compression chamber of compressors. This phenomenon of liquid slugging is detrimental to reciprocating compressors because it can damage valves and other components. A three-dimensional Eulerian multiphase model was developed using the software ANSYS CFX to predict the gas-liquid flow in a simplified suction muffler geometry. In addition, an experimental bench was designed to allow the injection of specified volumes of lubricant oil into the suction muffler and measurement of the volume that reaches the muffler outlet. The liquid slugging was observed through glass windows positioned in the test section and recorded with a high-speed camera. Measurements showed that the phenomenon of liquid slugging is random and that the volume of liquid reaching the muffler outlet increases with the oil injection pressure. The numerical model was able to predict the oil volume that reaches the muffler outlet in agreement with measurements in the case of high injection pressures, but it overpredicted the volume when the injection pressure was decreased. The visualization showed changes in the liquid phase morphology that are difficult to predict since closure models for a two-fluid model are typically developed for dispersed morphology.

1. Introduction

Under normal operating conditions, superheated vapor refrigerant is admitted into the cylinder of reciprocating compressors. However, liquid refrigerant and lubricating oil circulate in the refrigeration system to which the compressor is connected. Under some operating conditions, liquid can enter the suction system and reach the compression chamber. According to Singh et al. [1], in these situations the pressure inside the compression chamber can reach a value 10 times higher than the maximum pressure under normal operating conditions, due to the incompressibility of the liquid. This event, called liquid slugging, can damage valves and other components, reducing the reliability of the compressor.

In the suction of reciprocating compressors, an acoustic muffler is installed in order to attenuate the noise generated by the pressure pulsation in the flow, induced by the reciprocal movement of the piston. Due to the complexity of the flow inside the muffler, the effect of this acoustic muffler on the amount of liquid that reaches the compression chamber during a liquid slugging event is not well understood. The main objective of this study is to investigate the gas-liquid flow in a simplified geometry of the suction muffler of a small reciprocating compressor, aiming to understand the effect of the muffler geometry on the liquid volume that reaches the compression chamber after a liquid slugging event.
The majority of studies reported in the literature investigate the return of liquid refrigerant to the compressor, and its effects when it reaches the compression chamber [1-5]. Regarding lubricating oil, there are a variety of studies that analyze the effects of oil circulation in the compressor and the system, as shown by Youbi-Idrissi et al. [6]. However, no analysis of oil circulation related to the occurrence of liquid slugging, and subsequent damage to the compressor, could be found in the literature. An experimental investigation of gas-liquid flow in suction mufflers of reciprocating compressors also appears to be lacking. Regarding numerical modeling, several studies have been aimed at analyzing the flow features, heat transfer and noise attenuation in mufflers through CFD [7-10]. However, these studies consider the flow of pure gas refrigerant.

Rodrigues [11] adopted a two-dimensional formulation to numerically solve the gas-liquid flow in simplified geometries of suction mufflers, using the commercial software ANSYS CFX. The author sought to assess the influence of geometric parameters on the amount of liquid that reaches the muffler outlet from different liquid pulses of known volume. Rodrigues [11] observed that the geometry affects considerably the amount of liquid that reaches the muffler outlet.

This paper reports the results of a three-dimensional simulation model for gas-liquid flow in a simplified geometry representing a suction muffler. Furthermore, an experimental bench was also designed to measure the volume of oil that reaches the muffler outlet after the injection of an oil pulse of known volume, and to allow the visualization of the flow inside the component. For simplicity, lubricating oil was adopted for the liquid phase. The refrigerant R134a was selected for the gas phase as it is associated with a high probability of the occurrence of liquid slugging [11].

2. Numerical model

2.1. Flow geometry

The actual geometry of the suction muffler (figure 1) is modeled as a three-dimensional rectangular muffler with square inlet and outlet tubes, as well as a domain extended around the muffler to represent the internal environment of the compressor housing and the housing walls, as shown in figure 2. The suction duct is connected to the housing, corresponding to a semi-direct configuration. A storage reservoir is placed at the muffler outlet to monitor the oil volume, which is later used for comparison with the experimentally obtained volumes.

![Figure 1. Actual suction muffler.](image1)

![Figure 2. Solution domain. a) three-dimensional view and b) side view.](image2)

The internal volume of the muffler and the dimensions of the tubes were approximated to the dimensions of an actual reference muffler. An important difference of this 3D geometry, in relation to the 2D geometry of Rodrigues [11], is the presence of a small orifice (purge) under the muffler to assist in the removal of the liquid that enters the component. A structured mesh with hexagonal elements was adopted to reduce truncation errors and improve the convergence of the solution procedure. Mesh
refinement analysis was also performed, with the mesh being refined until the liquid mass reaching the outlet showed an asymptotic value.

2.2. Governing equations
For the modeling of the gas-liquid flow in the suction muffler, mass and momentum conservation equations for each phase were numerically solved, in the framework of the two-fluid model, enabling the evaluation of velocity, pressure and volume fraction fields for liquid and gas phases inside the muffler. The adopted Eulerian heterogeneous model (two-fluid model) solves the governing equations for each phase, with interfacial transfer terms, which are responsible for modeling the interaction between phases. The liquid volume fraction is equal to 1.0 at the entrance of the solution domain during the oil injection. After that, the liquid distribution is variable within the domain and will result from the solution of the equation of mass conservation for the liquid phase. The set of governing equations was solved using the finite volume method available in the commercial software ANSYS CFX.

The turbulent flow was considered isothermal, compressible for the gas phase (R134a) and incompressible for the liquid phase (oil). The transient problem is characterized by an oil pulse prescribed at the suction line for a short period, after which the duct is closed. The R134a fluid properties are obtained using the Redlich-Kwong equation of state, while the properties of the liquid phase (oil) were prescribed (density $\rho = 926 \text{ kg/m}^3$, viscosity $\mu = 0.01389 \text{ Pa.s}$, interfacial tension $\gamma_{LG} = 12 \text{ mN/m}$). The liquid phase was set as dispersed and the gas phase as continuous. In fact, although the liquid phase enters as a continuous morphology in solution domain, it breaks-up within the suction muffler changing its morphology to dispersed droplets with different sizes. This represents a challenge from the modelling point of view, as model closures are usually developed for a given phase morphology.

The oil droplet diameter, which is an input parameter for the interfacial transfer closure model, was used as a fitting parameter of the model. Although this set-up was able to reproduce qualitatively the experimentally observed flow, further investigation on the numerical model is necessary to capture adequately the phase configurations and morphological changes within the muffler and obtain more accurate quantitative results.

Turbulence modelling in gas-liquid two-phase flow is another challenging aspect of the numerical simulation, especially when in the presence of changes in the phase morphology. Adequate phase-dependent turbulence closure requires complex models and are, in general, specific closure models. Since there is no detailed information about the turbulence interactions between phases, we have used the SST turbulence model [11] with both phases sharing the solved turbulence field.

2.3. Boundary and initial conditions
The inlet liquid velocity at the suction duct during the pulse period was estimated by tracking the jet front via camera frame counting for different injection pressures (3.2 m/s for 0.7 bar, 3.9 m/s for 1.0 bar, 5.0 m/s for 1.5 bar and 5.5 m/s for 2.0 bar). A turbulence intensity of 5% was assumed as inlet boundary condition. The constant average pressure measured in the experimental apparatus was prescribed at the outlet of the storage reservoir. The boundaries representing the internal housing regions were represented by the suction pressure conditions (front, rear and bottom faces), allowing the gas to enter the muffler inlet and the oil that does not enter the muffler to be removed from the solution domain. The initial conditions assumed were that the solution domain was full of gas refrigerant (zero liquid fraction) under stagnant conditions and the pressure was equal to the saturation pressure for a temperature of $-23.3 \, ^\circ \text{C}$.

2.4. Spatial and temporal discretization
The grid convergence index (GCI) derived from the theory of generalized Richardson extrapolation [12] was adopted to verify convergence regarding spatial and temporal discretization, i.e., to identify the point from which further refinement in these discretizations did not produce a variation greater than 3% in the volume of oil collected in the storage reservoir. The adopted computational grid had 965,324 cells with very small sizes near solid walls, whereas the time step was set to 0.0025 s.
3. Experimental setup and procedure

The experimental test bench is composed of three main components: calorimeter, oil injection system and test section (figure 3). It should be noted that the superheated gas cycle calorimeter was just used to submit the muffler (which is inserted in the test section) to different operating conditions and to provide the mass flow of R-134a through the test section. The compressor discharge pressure was regulated via a hand-operated needle valve. An oil separator was installed in the discharge line followed by an accumulator to dampen pressure oscillations. Downstream of it, another hand-regulated needle valve was operated to expand the gas from the intermediate pressure down to the desired suction line pressure.

The oil injection system (figure 3) consists of an oil reservoir under controlled pressure, supply line and a solenoid valve that controls the injections in the test section. The supply line is connected to the test section and the distance between its outlet and the muffler inlet is similar to that found in the reference compressor, corresponding to a semi-direct suction system.

The test section, illustrated in figure 4, consists of a container with a sight glass where the transparent muffler made of acrylic is inserted. The test section is positioned in the calorimeter circuit before the suction duct of the compressor, so that the whole container is under the selected suction pressure conditions, simulating operation inside the hermetic compressor.

![Figure 3. Schematic representation of the test bench.](image1)

![Figure 4. Schematic representation of the test section.](image2)
A high-speed camera is placed in front of the sight glass, to allow observation of the flow through the muffler. The muffler outlet is connected to the suction chamber, storage reservoir and calorimeter line. The storage reservoir is also made of acrylic with dimensions that allow the level to be observed and measured by image processing. After establishing the suction and discharge pressures in the calorimeter and in the test section, the oil reservoir is loaded with the desired pulse volume. The oil reservoir is then pressurized with gas refrigerant, as the flow in the test section is promoted by the pressure gradient. The solenoid valve is opened with the high-speed camera being activated simultaneously, capturing the oil jet at the entrance and the flow inside the muffler. Finally, the compressor is turned off and an image of the storage reservoir is used to measure the volume of oil collected.

4. Results

4.1. Quantitative analysis

The experimental tests require the definition of three main parameters: injection pressure in the oil reservoir, oil pulse volume and mass flow rate through the system, the latter being determined mainly from the suction and discharge pressures in the compressor. The aim was to analyze the effect of the injection pressure, keeping the pulse volume and mass flow rate at fixed values of 16 mL and 10 kg/h, respectively. The injection pressures were 0.7, 1.0, 1.5 and 2.0 bar above the suction line pressure of 1.15 bar corresponding to the evaporating temperature of -23.3 °C, with a total of 9 injections being performed for each injection pressure in order to obtain an average value for the volume of oil collected at the muffler outlet. In the numerical model, varying the injection pressure resulted in different velocities at the suction duct entrance.

The measurements in figure 5 indicate that the volume of oil collected increases with the injection pressure, but the relationship is not linear. This suggests that the increase in the jet velocity gives rise to flow patterns that are more likely to direct the oil towards the muffler outlet. Hence, the higher the velocity, the higher the oil dispersion. There is a considerable increase in the volume of oil collected when the pressure is raised from 1.0 to 1.5 bar and a tendency for the volume to stabilize when the pressure is raised from 1.5 to 2.0 bar. As also shown in figure 5, no oil was detected at the muffler outlet for the injection pressure of 0.7 bar.

Figure 6 shows the experimental and numerical results for the volume of oil collected in the storage reservoir. The dashed lines correspond to the average value obtained in the experiments at each injection pressure, while the solid lines correspond to the numerical prediction of the oil reaching the storage reservoir as a function of time. Table 1 shows measurements of collected volumes compared with the maximum collected volumes obtained from the model over a period of 1 s. As can be seen, the model is able to capture the trend of the volume of oil collected increasing with the injection pressure. It is also observed that the model estimates for the volume of oil collected are in good agreement with the experimental measurements for the higher injection pressures (1.5 bar and 2.0 bar), but the model overestimates the volume for the lower pressures (1.0 bar and 0.7 bar). The explanation for this could be related to the magnitude of the jet velocity at the entrance of the solution domain. Cases with higher injection pressure have higher velocities, with inertial forces, better captured by the model, being dominant over interfacial forces, which rely on less accurate closure models, particularly for this case in which phase morphologies and their changes along the fluid domain are not properly resolved.

The numerical results showed that a small amount of oil is dragged out of the solution domain through the exit section and is thus not captured by the storage reservoir. In addition, the volume of oil collected in the storage reservoir varies over time, which is associated with the predicted gas flow in that region. The adoption of a dispersed configuration for the liquid phase may contribute to this phenomenon, with the oil level in the storage reservoir being disturbed due to oil being dragged out of the domain. This disturbance is more visible for the case with an injection pressure of 2.0 bar. This case results in a higher volume of oil being collected and, consequently, a greater height of the oil level in
the storage reservoir becomes exposed to the gas flow. It should be noted that it was not possible to quantify the amount of oil dragged out in the experimental bench system.

![Figure 5. Measured volume of oil collected as a function of injection pressure.](image)

![Figure 6. Experimental and numerical results for volumes of oil collected.](image)

### Table 1. Experimental and numerical results for the volumes of oil collected.

| Injection Pressure [bar] | Volume [mL] | Δ [mL] |
|--------------------------|-------------|--------|
|                          | Exp.        | Num.   |
| 0.7                      | 0.0000      | 0.0281 |
|                          | -0.0281     |        |
| 1.0                      | 0.0124      | 0.0732 |
|                          | -0.0608     |        |
| 1.5                      | 0.1061      | 0.0978 |
|                          | 0.0083      |        |
| 2.0                      | 0.1493      | 0.1436 |
|                          | 0.0057      |        |

The time of the experiment is 4 s from the start of the oil pulse to the compressor shutdown. If the total simulation time was also 4 s, it is possible that the volume of oil collected would still vary. On the other hand, the volume fraction iso-surfaces for the model and the high-speed camera videos indicate that after approximately 0.6 s there is no more liquid entering the muffler outlet tube. Hence, the oil directed to the outlet had already left, either being collected in the storage reservoir or having left the solution domain. Taking this into consideration and the fact that the computational processing time is high (an average 60 h/run of 1 s), the simulation was limited to a period of 1 s.

Some variability was found in the measurement of the volume of oil collected for each injection, as previously indicated by Singh et al. [1], which can be attributed to the chaotic characteristic of the injection process. figure 7 shows snapshots of the oil flow inside the muffler at the same time for three distinct injections with 2.0 bar. These images show that the jet breaks when it collides with the muffler walls, assuming very different flow patterns for each injection.

The results for the volume of oil collected in the nine experiments with injection at 2.0 bar are shown in figure 8, where the dashed line represents the average value. Table 2 shows the average value after three, six and nine injections, indicating that it reaches a well-defined value as the number of injections increases, despite the variability of the phenomenon.

4.2. Qualitative analysis

In order to visualize the oil flow from the numerical results, iso-surfaces were constructed with volume fractions ranging from 0.1 to 1.0, with increments of 0.1, totalizing ten iso-surfaces. These iso-surfaces were plotted simultaneously every 0.1 s and compared with snapshots from the camera video frame at
the same time, allowing a qualitative comparison of the flow configuration inside the muffler. The comparisons are presented herein for the first intervals (0.1 to 0.4 s) with an injection pressure of 2.0 bar. However, the qualitative observations are similar for the other injection pressures.

**Figure 7.** Flow pattern for $t = 0.2$ s for three distinct injections at 2.0 bar.

**Figure 8.** Volume of oil collected for different injections at 2.0 bar.

| Injection | Vcol [mL] | Mean 3-6-9 |
|-----------|-----------|------------|
| 1         | 0.2054    |            |
| 2         | 0.1880    | 0.1561     |
| 3         | 0.0750    |            |
| 4         | 0.0324    |            |
| 5         | 0.1574    | 0.1476     |
| 6         | 0.2276    |            |
| 7         | 0.1815    |            |
| 8         | 0.1395    | 0.1493     |
| 9         | 0.1371    |            |

**Table 2:** Volume of oil collected per injection at 2.0 bar.

Figure 9 was prepared for $t = 0.1$ s and shows the numerical model predicted higher vertical oil dispersion, which may be related to the adoption of a dispersed model configuration for the liquid phase. A slight downward slope is observed in the jet due to the effect of gravity. We found this jet inclination is even more prominent in cases with lower injection pressure, a consequence of the lower momentum of the jet at the entrance. The experiments showed that the flow presents a continuous-continuous pattern until approximately the end of the jet injection, with a well-defined interface between the oil and the gas. The numerical and experimental results of the oil film advancing on the bottom surface of the muffler are in good agreement and no oil is present in the outlet tube at this time.

As shown in figure 10 for 0.2 s, the numerical and experimental results for the oil level on the bottom surface of the muffler seem to be in good agreement, with oil present at the outlet tube walls. For a 16 mL pulse of oil, the injection time in the model is 0.1786 s. In the experimental test section, the oil is injected via a pressure load applied to the reservoir. Therefore, the end of the jet pulse is characterized by a mixture of gas and liquid, with the jet taking longer to cease when compared to predictions. This also gives rise to higher dispersion of the oil jet inside the muffler, which is well suited for predictions with the dispersed model adopted in the simulations. Part of the discrepancy between numerical and experimental results can be attributed to the measurement of the jet velocity, which is estimated by tracking the jet front via camera frame counting.
Figure 9. Qualitative comparison for t = 0.1 s.

Figure 10. Qualitative comparison for t = 0.2 s.

Figure 11 shows some similarity between the numerical and experimental results (t = 0.3 s) for the flow configuration at the left wall of the muffler. In addition, the oil level is seen to increase at the lower right corner of the muffler, both in the model and in the experiment. In fact, this increase was observed at 0.2 s (figure 10), but it was intensified at 0.3 s due to the phenomenon of “sloshing”, in which the oil slides from left to right. This sliding and the presence of oil in the outlet section become more evident in the subsequent figures as time progresses. At t = 0.4 s (figure 12), oil dispersed in the muffler central region is directed towards the outlet. It is also possible to observe the level of the oil settling at the bottom. The numerical result shows that the wave formed by the sloshing phenomenon begins to move towards the left side of the muffler.

As time continues to progress (figure 13), basically three phenomena are observed: (i) a reduction in the dispersion morphology in the muffler central region; (ii) stabilization of the oil level at the bottom of the muffler; (iii) a variation in the oil level in the storage reservoir, which is observed exclusively in the numerical model. The high-speed camera videos indicated that the oil reaches the outlet via two mechanisms: (i) dragging of dispersed oil particles that approach the entrance of the outlet tube; and (ii) suction of oil film that flows downwards on the external surface of the outlet tube, which is dragged as soon as it reaches the entrance of the outlet tube.
Figure 11. Qualitative comparison for $t = 0.3$ s.

Figure 12. Qualitative comparison for $t = 0.4$ s.

Figure 13. Qualitative comparison for $t = 1.0$ s.
5. Conclusions
A numerical-experimental investigation of gas-liquid flow in a simplified suction muffler was conducted, aimed at deepening our understanding of liquid slugging in reciprocating compressors of small refrigeration systems. Measurements were carried out in an innovative experimental apparatus and showed that the volume of liquid reaching the muffler outlet increases with the oil injection pressure at the compressor entrance. In addition, we found that the phenomenon of liquid slugging is random, requiring mean values to quantify the liquid volume that reaches the muffler outlet. A three-dimensional Eulerian multiphase model was developed to predict the gas-liquid flow. The model was able to predict satisfactorily the main flow features and the volume of oil that reaches the muffler outlet under high injection pressures. The visualization clearly showed that the liquid phase morphology changes from continuous to dispersed in some regions of the muffler, and to continuous again when the oil injection ceases. These morphological changes are difficult to predict since closure models for a two-fluid model are typically developed for dispersed morphology while interface-capture methods cannot deal with changes from continuous to dispersed morphology. This aspect of liquid slugging in reciprocating compressors merits extensive investigation in future studies.

Acknowledgements
The present study is part of a technical-scientific cooperation program between the Federal University of Santa Catarina (UFSC) and Nidec-GA (Embraco). Financial support provided by the National Institutes of Science and Technology (INCT) Program (CNPq Grant No. 443696/2014-4; FAPESC Grant No. 2018TR1576) and EMBRAPII Unit POLO/UFSC and CAPES is duly acknowledged.

References
[1] Singh R, Nieter J J and Prater G 1986 Prediction of Slugging-Induced Cylinder Overpressures Proc. Int. Compressor Eng. Conf. at Purdue (West Lafayette, USA)
[2] Simpson F and Lis G 1988 Liquid Slugging Measurements in Reciprocating Compressors Proc. Int. Compressor Eng. Conf. at Purdue (West Lafayette, USA)
[3] Shiva-Prasad B G 2002 Effect of Liquid on a Reciprocating Compressor. J. Energy Resources Technology, 124 187-190
[4] Liu Z and Soedel W 1994 An Investigation of Compressor Slugging Problems Proc. Int. Compressor Eng. Conf. at Purdue (West Lafayette, USA)
[5] Laughman C R, Armstrong P R, Norford L K and Leeb S B 2006 The Detection of Liquid Slugging Phenomena in Reciprocating Compressors via Power Measurements Proc. Int. Compressor Eng. Conf. at Purdue (West Lafayette, USA)
[6] Youbi-Idrissi M and Bonjour J 2008 The effect of oil in refrigeration: Current research issues and critical review of thermodynamic aspects. Int. J. Refrigeration, 31 165-179
[7] Nakano A and Kinjo K 2008 CFD Applications for Development of Reciprocating Compressor. Proc. Int. Compressor Eng. Conf. at Purdue (West Lafayette, USA)
[8] Morriesen A, Deschamps C J, Pereira E L L, Dutra T and Ribas Jr F A 2009 Numerical Prediction of Superheating in the Suction of a Hermetic Reciprocating Compressor Proc. 20th Int. Congress of Mech. Eng. (Gramado, Brazil)
[9] Sarioglu K, Ozdemir A R, Oguz E and Kaya A 2012 An Experimental and Numerical Analysis of Refrigerant Flow Inside the Suction Muffler of Hermetic Reciprocating Compressor Proc. Int. Compressor Eng. Conf. at Purdue (West Lafayette, USA)
[10] Coskun U C, Gunes H and Sarioglu K 2020 Numerical Investigation of Suction Muffler in Household Refrigerator Compressor. Int. J. Geology, 14 27-31.
[11] Rodrigues T T 2018 Refrigerant Liquid Slugging in the Suction System of Reciprocating Compressor. Proc. Int. Compressor Eng. Conf. at Purdue (West Lafayette, USA)
[12] Roache P J 1998 Verification and Validation in Computational Science and Engineering, (Albuquerque: Hermosa Publishers)