Numerical Investigation of Suction Muffler in Household Refrigerator Compressor

Umut Can Coskun and *Hasan Gunes*
Department of Mechanical Engineering
Istanbul Technical University
Istanbul, Turkey
ucoskun@itu.edu.tr, guneshasa@itu.edu.tr

Kemal Sarioglu
R&D Department
Arcelik A.S.
Istanbul, Turkey
kemal.sarioglu@arcelik.com

Abstract— *In this study a numerical investigation of a suction muffler in a hermetic reciprocating compressor of a domestic refrigerator is performed using a finite volume based flow solver (Fluent). In order to reveal the behavior of the flow realistically, unsteady experimental pressure data has been used in the outlet boundary condition for the simulations. Detailed investigations are carried out to reveal the instantaneous flow behavior in different muffler sections such as channel and chambers based on the mass flow rate variation with respect to crankshaft angle. It is shown that there is a phase shifting between the mass flow rates through the chamber inlets. Future studies are underway to improve and optimize the muffler design in an effort to maximize the flow rate while keeping the noise levels at a minimum.*

Keywords—reciprocating compressor; suction muffler; Computational Fluid Dynamics (CFD)

I. INTRODUCTION

In hermetic reciprocating compressors, pressure fluctuations at the suction and inside of muffler cause noise as well as a negative effect on the performance of the refrigeration. Therefore, the noise level should be kept at a minimum for both the performance improvement and our comfort [1].

Suction muffler is an important compressor component used for controlling the noise level. Many researchers have devoted their attention on suction mufflers with the aim of high performance and low noise levels.

Choi et al. (2000) optimized the suction muffler by analysing the flow and sound fields for an optimum design of a suction muffler [2]. In the same study, it is shown that according to the spectral characteristics of the noise, either one-chamber suction muffler or two-chamber suction muffler can be used for noise reduction.

Recently, Sarioglu et al. (2012) investigated the refrigerant flow inside the suction muffler experimentally and numerically. They performed time-dependent flow analysis and found that the muffler inner channel design is critical to reduce the overall pressure loss in the muffler. They also pointed out that after the expansion process, there are back flows during the exhaust period at muffler inlet [3].

Nakano et al. (2008) applied CFD simulations on several components of a reciprocating hermetic compressor including the lubrication system and the suction muffler. In their analysis, they solved three-dimensional unsteady compressible viscous flow with a commercial flow solver, taking pressure variations in the cylinder obtained from previous calculations where they have implemented these data as boundary conditions. As a result of their computations, the mass flow rate of the suction muffler is increased due to the heat and pressure losses [4].

Pereira et al. (2008) investigated the performance of reciprocating compressors via CFD and validated their data with experiments. They investigated the effect of modification in the design of suction and discharge parts and analyzed energy losses in mufflers and valves aiming to increase the compressor efficiency [5].

In this study, the flow dynamics in a two-chamber suction muffler is simulated in order to improve performance by increasing the mass flow rate through the muffler. The effect of the volumetric ratio of chambers is investigated numerically and experimentally. Solutions have been validated with the available experimental data. A good agreement has been achieved with experimental results, i.e., numerical simulations predict average mass flow rate of 4% higher than the measured value.

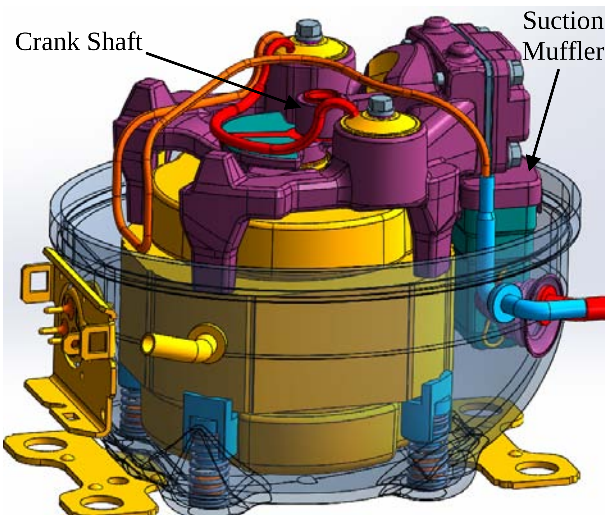


Fig.1: General appearance of a household reciprocating hermetic compressor

II. Computational Domain and Boundary Conditions

A. Geometry and Boundary Conditions

In a hermetic reciprocating compressor, the motion of the piston from top to bottom dead center causes the deformation of the valve reed due to the pressure differences between cylinder and the muffler. This deformation opens the path between muffler and the low pressure cylinder. To model the full system (muffler, piston-cylinder and valves), a very sophisticated numerical procedure including fluid structure interaction and dynamic mesh have to be performed. However, we are interested in the flow inside the muffler thus, the flow domain is restricted by the muffler. This simplification is only viable if one can provide the "correct" pressure variation in the muffler exit as a boundary condition.

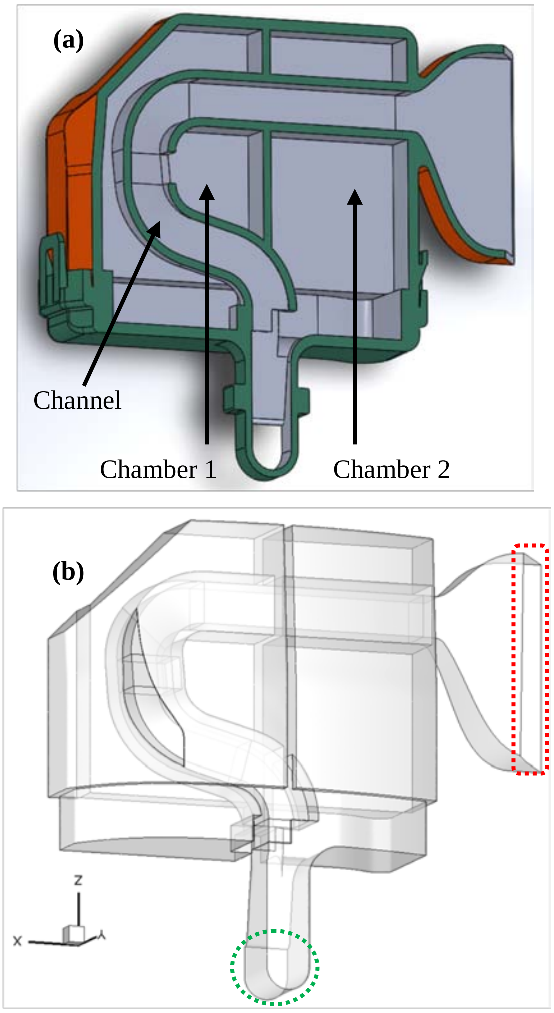


Fig.1. (a) Cross-section of the muffler solid body, (b) flow domain

Figure 1a shows a cross-section of the muffler body and Fig.1b shows the flow domain inside the muffler body. Figure 2 indicates a representative cross-sectional view of three-dimensional computational mesh consisting of nearly 4,200,000 cells. The grid consists of a combination of tetrahedral and structured hexahedral cells. Since the muffler geometry is complicated, it is not practical and feasible to generate structured grid entirely. On the other hand, the channel inside the muffler has an applicable geometry for structured hexahedral mesh as shown in Fig.2. Other sections including the two chambers of the muffler have tetrahedral cells.

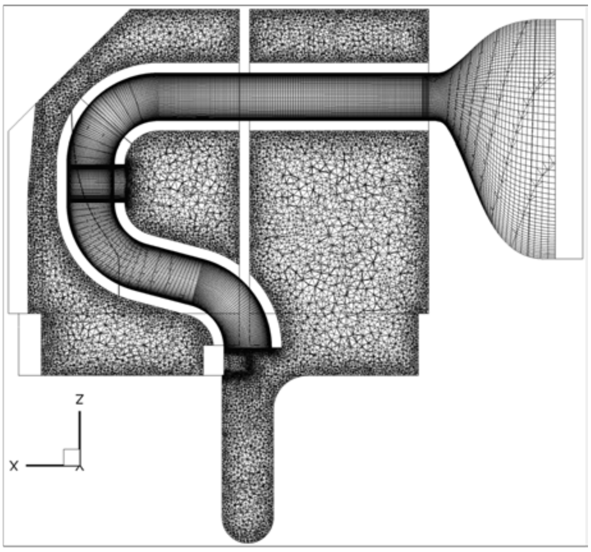


Fig.2. Cross-section of the computational domain of muffler

The fluid used in our calculations is isobutene which is commonly used as refrigerant in household refrigerators. Isobutene is assumed to be compressible ideal gas with constant physical properties. Operating pressure inside the compressor is 0.624 bar. The red dotted rectangle in Fig.1b shows the pressure inlet and green dotted circle in the same figure shows pressure outlet boundaries. Experimentally obtained pressure data at each crank angle is used as user specified transient pressure outlet boundary condition as shown in Fig.3. Constant pressure boundary conditions are specified at inlet. All other boundaries are stationary walls with no-slip condition. Due to the compressibility effects, energy equation has to be solved and all of the wall boundaries are prescribed with constant temperature.

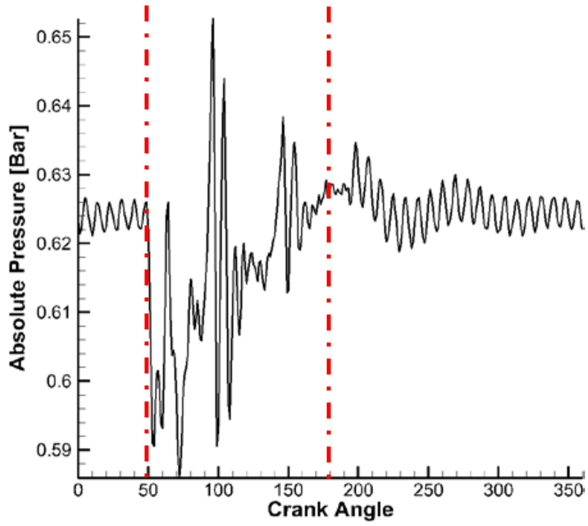


Fig.3. Experimentally obtained pressure data for outlet boundary condition

As we consider a fully periodic flow, the outlet boundary has been prescribed with two kinds of boundary conditions depending on the crank angle. Experimentally it is shown in Fig.3 that the suction reed opens approximately at the crank angle 50° and closes at 180°. The rapid closing and opening movement of the valve reed is called "flutter" which causes the pressure fluctuations between these angles (indicated between red dotted lines in Fig.3). In the numerical simulations 50° crank angle is determined as the beginning of the period and following 130° has been chosen as pressure outlet boundary profile. During the remaining 260° crank angle, the suction reed is considered closed for which we define the outlet boundary as wall with no-slip condition (i.e., zero flow-rate condition).

B. Governing Equations and Numerical Procedure

Our numerical results have been obtained using a finite volume based compressible transient flow solver. The governing equations to be solved are continuity, momentum and energy equations given in (1), (2) and (3) respectively:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{V} \right) = 0 \quad (1)$$

$$\rho \frac{\partial \vec{V}}{\partial t} + \rho \left(\vec{V} \cdot \nabla \right) \vec{V} = -\nabla p + \mu \nabla^2 \vec{V} + \frac{1}{3} \mu \nabla \left(\nabla \cdot \vec{V} \right) \quad (2)$$

$$\rho \frac{Dh}{Dt} = \frac{Dp}{Dt} + \nabla \left(k \nabla T \right) \quad (3)$$

In addition to conservation equations, equation of state has to be taken into account since the fluid is modeled as compressible ideal gas. Since the flow is turbulent, k-ε turbulence model is applied with non-equilibrium wall function for near wall treatment.

In the numerical solutions SIMPLE Algorithm is used for pressure-velocity coupling. The density, momentum and energy equations are discretized with second-order upwind scheme. The time step is $\Delta t = 5.68 \times 10^{-5}$ seconds considering the duration of the motion of crankshaft for 1 degree.

III. Results

Mesh quality plays a significant role in the accuracy and the stability of the numerical simulations thus, extensive grid checks have been performed to show grid independence. Grid refinement has been made in flow regions where large grandients occur such as walls.

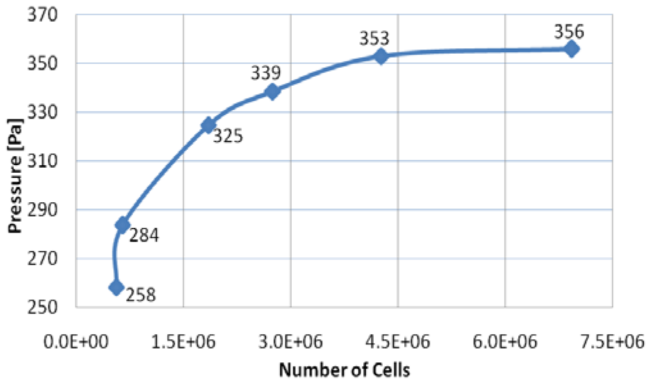


Fig.4. Grid independence showing pressure differences vs. number of cells

Figure 5 indicates mass flow rate at muffler inlet and outlet. As the outlet mass flow rate denoted by the red line shows, due to the pressure fluctuations between 50° and 180° crank angles, we observe fluctuations in the mass flow, too. Figure 6 shows the mass flow rates from chamber 1 and 2 to channel and exit region respectively. The reason why the initial opening of the suction reed corresponds to 0° crank angle instead of 50° as mentioned earlier relies on a manipulation to ease the implementation of the changes in the boundary conditions. In Fig.3 the periodic pressure variation till the beginning of the valve reed fluttering has been removed to the final part of the pressure data since it is a periodic flow.

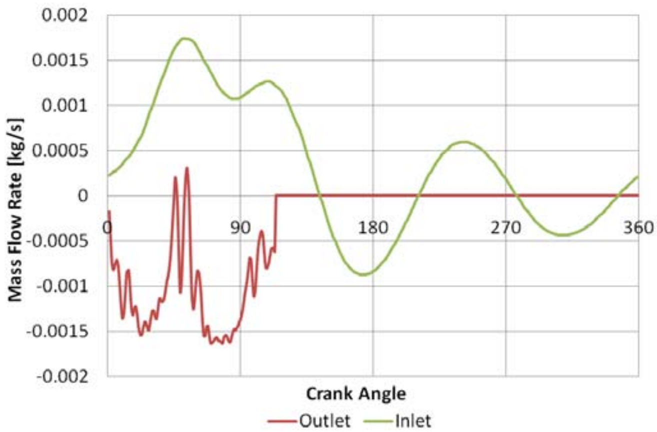


Fig.5. Mass flow rates of inlet and outlet sections

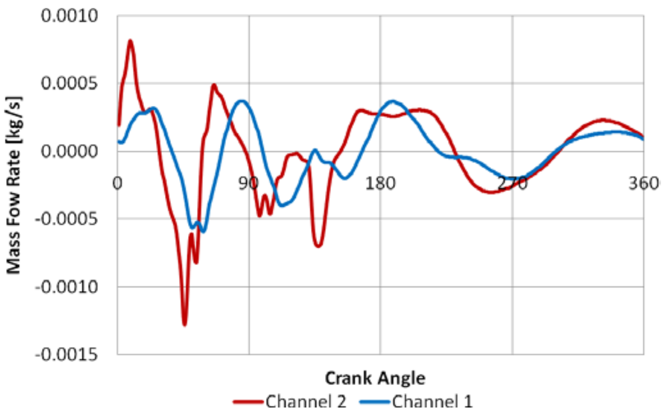


Fig.6. Mass flow rates of chamber 1 and 2 inlet section.

Figure 7 and 8 shows pressure distribution at outlet and chamber inlets respectively. In Fig.7 during the first 130^o crank angle, shows the experimental pressure profile as it is used as pressure outlet boundary condition while the remaining 260 steps represents the solution for surface pressure at the outlet boundary.

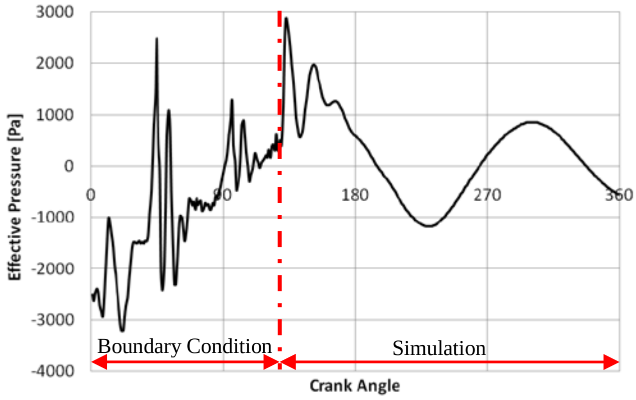


Fig.7. Pressure distribution of chamber 1 and 2 inlet sections

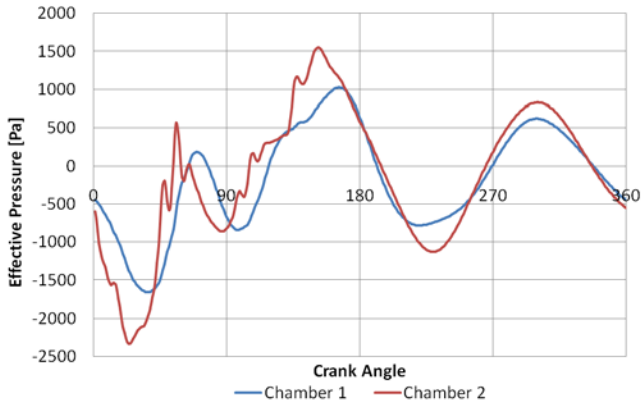
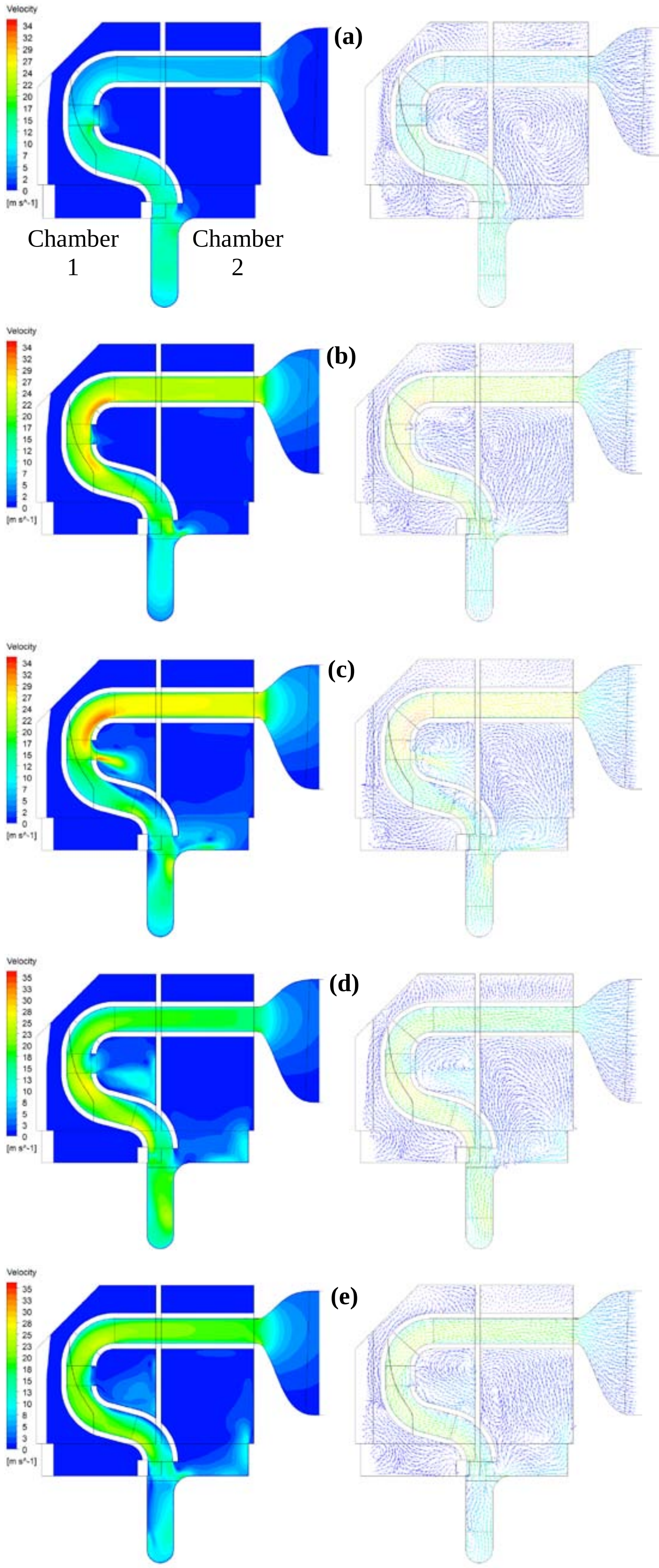


Fig.8. Pressure distribution of chamber 1 and 2 inlet sections

When the suction reed has opened at the beginning of a period, there exists a pressure difference between cylinder and muffler exit region which forces the fluid move from muffler to cylinder. The initial abrupt change in the mass flow rate at the outlet is supplied by the mass flow rate from the 2. chamber first. This effect spreads through the channel until it gets the outer cooler refrigerant.

Figure 9 shows instantaneous flow fields at various crank angles selected out of one period after reaching periodic steady state condition. At the crank angle 0^o, the outlet boundary is still close and the flow rate to the inlet boundary with low magnitude is remnant of fluctuations from the previous cycle.

In Fig.9a the crank angle is at 20 degrees. The mass flow rate is still supplied mainly from chamber 1 and 2. Meanwhile the muffler starts to be fed with cool refrigerant through the inlet. In Fig.9b the crank angle is at 40 degrees. The mass flow feeding the muffler is entirely from the inlet section. During the cycle the pressure drop in chamber 2 is so high that it sucks refrigerant from the channel. As it can be seen in Fig.6 there is a phase shifting in the mass flow through the slots of chamber 1 and 2. In the following 20 degrees (crank angle at 60^o) suction to chamber 2 ends while chamber 1 is sucking fluid from the channel. As it can be seen from the instantaneous snapshots at following crank angles in Fig.9d-f this process repeats itself until the suction reed closes.



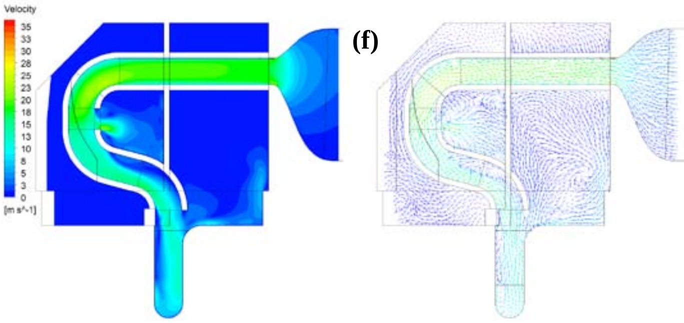


Fig.9. Crank angle (a) 20°, (b) 40°, (c) 60°, (d) 80°, (e) 100°, (f) 120°

As this study shows the size and volume ratios of the chambers influence the flow behavior thus, it is thought that they have an impact on the number and magnitude of the suction reed flutters. The volume ratio is determined to be an important optimization parameter for further studies.

IV. Conclution

This is a numerical study to investigate flow inside a muffler which is commonly used in household. The computations have been performed using a finite volume based compressible transient flow solver (Fluent). In this study we investigated the flow behavior inside a muffler based on mass flow rate variations with respect to crank angle both at the muffler inlet and outlet. This analysis gives a comprehensive insight to the interaction of muffler's chambers with flow through the channel. Furthermore this study aims to be an initial step to a more complicated optimization process.

Acknowledgment

We gratefully acknowledge the financial support of Arcelik A.S. through the project "Flow Analysis in Mufflers Used in Household Refrigerator Compressor".

References

[1] J. H. Lee, K. H. An, and I.S. Lee, "Design of the suction muffler of a reciprocating compressor," Int. Compres. Eng. Conf.,Purdue Unv., 2002.

[2] J. K. Choi, M. J. Joo, S. K. Oh and S.W. Park, "Smart suction muffler design for a reciprocating compressor," Int. Compres. Eng. Conf.,Purdue Unv., 2000.

[3] K. Sarioglu, A. R. Ozdemir, E. Oguz and A. Kaya, "An experimental and numerical analysis of refrigerant flow ınside the suction muffler of hermetic reciprocating compressor," Int. Compres. Eng. Conf.,Purdue Unv., 2012.

[4] A. Nakano and K. Kinjo, "CFD applications for development of reciprocating compressor," Int. Compres. Eng. Conf.,Purdue Unv., 2008.

[5] E. L. L. Pereira, C. J. Deschamps, and F.A. Ribas Jr, "Performance analysis of reciprocating compressors through computational fluid dynamics," Proc. IMechE, vol. 222, Part E: J. Process Mechanical Engineering