International Compressor Engineering Conference | School of Mechanical Engineering

2016

Numerical Simulation for the Internal Flow Analysis of the Linear Compressor with Improved Muffler

Hyunjoo Oh
LG electronics, Korea, Republic of (South Korea), hyunjoo.oh@lge.com

Sungchul Kong
LG electronics, Korea, Republic of (South Korea), sungchul.gong@lge.com

Wonsik Oh
LG electronics, Korea, Republic of (South Korea), wonsik.oh@lge.com

Kyeong-bae Park
LG electronics, Korea, Republic of (South Korea), kb.park@lge.com

Numerical Simulation for the Internal Flow Analysis of the Linear Compressor with Improved Muffler

Hyunjoo OH[1]*, Sungchul KONG[1], Wonsik OH[1], Kyeongbae PARK[1]

[1]LG Electronics, L & A Research Center
Seoul, Republic of South Korea
hyunjoo.oh@lge.com
wonsik.oh@lge.com
sungchul.gong@lge.com
kb.park@lge.com

ABSTRACT

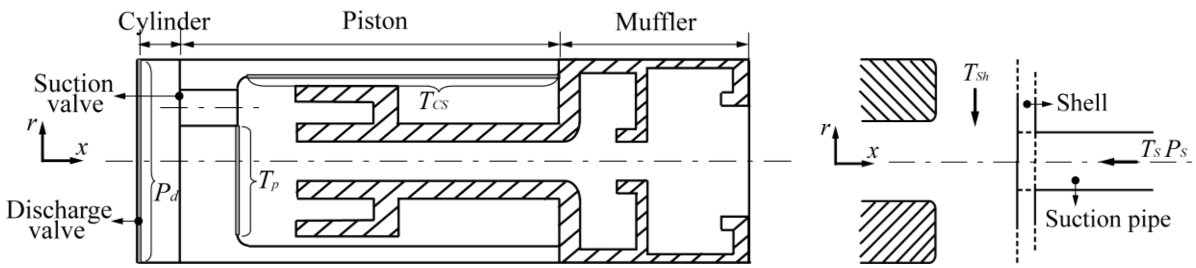
The numerical simulation is frequently used to estimate experimental results in fluid flow dynamics. In this paper, we focus on increasing the efficiency of linear compressor by applying modified muffler. Furthermore, we analyze a fluid flow in a suction line of linear compressor by using a computational fluid dynamics. We assume an unsteady flow, compressible, and two dimension axis-symmetric condition. In numerical simulation models, all processes of linear compressor, i.e., compression, discharging, expansion and suction, are continuously performed. Numerical simulation results are compared with experimental data to validate an accuracy of simulation. Furthermore, we show that numerical simulation results are well converged according to time. The results present that the mass compressor efficiency increases when the pressure field in the muffler exit is highly maintained during the suction process.

1. INTRODUCTION

In recent world, many countries have been strengthening the regulation of energy savings. Global home appliance markets interested on energy consumption have been also growing. Especially, because household refrigerator consumes about 5% (RECS, 2009) of using home energy, numerous manufacturers related household refrigerator have made a great effort to increase the energy efficiency. As example, the compressor accounting for up to 90% (LG homepage, 2016) of household refrigerator efficiency plays an important role to save energy in home appliance.

The efficiency of compressor can be improved by optimizing muffler design and by decreasing superheating loss and pressure loss. The muffler design especially affects efficiency of compressor by changing the flow field in suction line of compressor. Therefore, the suction muffler in compressor has been actively studied to improve energy efficiency of compressor. Furthermore, the computation fluid dynamics (CFD) has been used to expect results of compressor performance with respect to suction muffler, because the CFD commercial tool has been developed and we can obtain more exact results in short time. Hence, many works for muffler design have been studied by using CFD. Kemal *et al.* (2012) studied the refrigerant flow inside the suction muffler of compressor by using CFD. The results of numerical analysis shown the mass flow rate calculated by CFD is good agreement with the experiment results. C. J. *et al.* (2002) presented muffler affection in the compressor performance. This numerical model was applied one dimensional, compressible and thermal flow. Kwangha *et al.* (2006) developed the research of compressor by computational simulation. The compressor shell's modal simulation was conducted by CAE, and model with muffler and suction valve in suction line was analyzed by CFD with FSI. Umut and Hasan (2013) investigated change of the mass flow rate and pressure with respect to muffler shape. They solved simulation model of three dimension, unsteady, and validated experimental data by comparing CFD data.

In this paper, we analyze the internal flow of the linear compressor with improved muffler by using CFD. The simulation of compressor is carried out two dimensional axis-symmetry, unsteady, compressible and turbulence flow.



(a) Piston, cylinder and muffler (b) Suction part around shell

Figure 1: Schematic in suction system of the linear compressor and temperature condition on boundary

We considered four cases to evaluate the effect of suction muffler. Furthermore, we optimize suction muffler by analyzing change of flow field in suction muffler exit and suction muffler entrance of linear compressor.

2. NUMERICAL METHODOLOGY

In this section, we define initial condition, boundary condition and simulation method for computational model of linear compressor. Figure 1 shows schematic in suction system of the linear compressor and boundary conditions for temperature and pressure. As shown in Figure 1(a), we denote temperature of the inner piston's cylindrical surface and the inner piston's surface on suction port entrance as T_{cs} and T_p, respectively, which is about 333 Kelvin and 336.5 Kelvin. Moreover, we denote compression pressure applied on discharge port as P_d. We denote boundary conditions in suction pipe and around muffler entrance connected shell space, respectively, as T_s, P_s and T_{sh} in Figure 1(b). T_s, which is about 298 Kelvin and P_s respectively represent the temperature and the pressure of refrigerant flowing from evaporator. We involve shell space in simulation domain that connected muffler entrance. The boundary temperature on shell space is about 310.7 Kelvin, is denoted as the T_{sh}. The T_{cs}, T_p and T_{sh}, which are already mentioned, are obtained by experiment, and they are constant values in according with time.

The number of mesh of simulation model to finite volume method is about 65000. We generate rectangular meshes by using ANSIS ICEM program, and simulation domain is solved unsteady, compressible, viscous flow with ANSYS FLUENT V.15 commercial program. We set the simulation fluid as a real-gas-isobutan based on physical property in REFPROP. To implement motion of linear compressor, we use dynamic mesh including layering and re-meshing method. SIMPLE scheme is used by pressure-velocity coupling algorithm. We adopt second order upwind to interpolation between the nodes and nodes. The time step size to run calculation and time step to converge are set as $5xe^{-05}$ and 50, respectively.

In this CFD simulation, whole process of the linear compressor, i.e., compression, discharging, expansion, suction is continuously generated by using user defined function and scheme-language until the cycle converges. Figure 2 shows the pressure-volume diagram of whole processes. The compression process is defined that pressure of area-weighted average in cylinder increases from P_s to P_d as a polytropic process. In the discharging process, the pressure in cylinder is affected by refrigerant flowing out cylinder through discharse valve until the piston reaches to top dead center. After that, the pressure in cylinder decreases from P_d to P_s as a polytropic process. We call this process as the

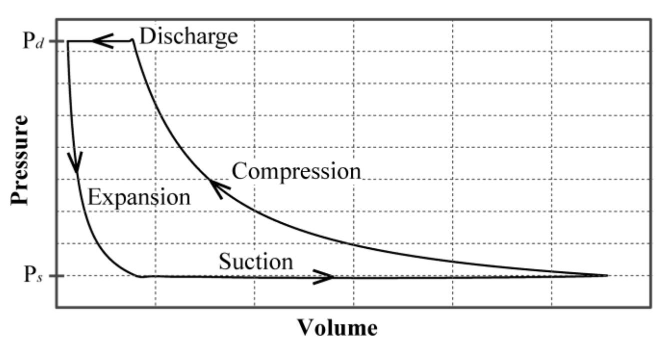


Figure 2: The pressure-volume diagram of whole processes by using the scheme language

Table 1: The four cases of modified suction muffler

Applied item	case1	case2	case3	case4
Partition wall	none	none	applied	applied
Cylindrical projection	none	applied	none	applied

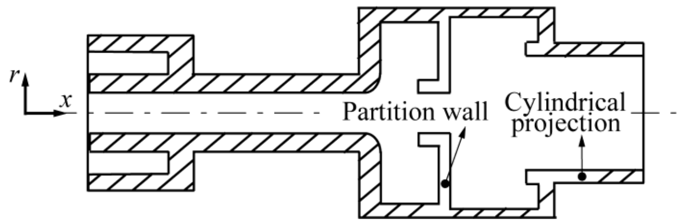


Figure 3: Schematic of the modified suction muffler

expansion process. Finally, the suction process is affected by refrigerant flowing in cylinder through suction valve. Last process is continued until pressure in cylinder reaches the P_s. Hence, valves of linear compressor only controlled by pressure difference between cylinder and muffler exit area and by simulation time. The suction valve and discharge valve are assumed as ideal valve. Therefore, valve motion is carried out to fully-open, fully-close. Because the simulation model is solved in two dimensional axis-symmetry method, it is assumed to be three dimensional structures based on the rotation axis. Therefore, suction valve is made by donut shape.

3. RESULTS AND DISCUSSION

3.1 Validation of the Computational Simulation

The internal flow of the linear compressor is affected by suction muffler structure existed to suction line from suction pipe to suction valve. Therefore, we analyze the internal flow of the linear compressor with improved muffler by using computational simulation.

We consider four cases of modified suction muffler in Table 1. Figure 3 shows schematic of the modified suction muffler. As shown in Figure 3, the partition wall is located between muffler entrance and exit, and the cylindrical projection is connected to the entrance of the muffler. The internal and an external diameter of partition wall is 0.008m, 0.025m, respectively. The height and diameter of cylindrical projection are 0.015m. In Table 1, case1 indicates base model without partition wall and cylindrical projection. case2 is applied cylindrical projection only.

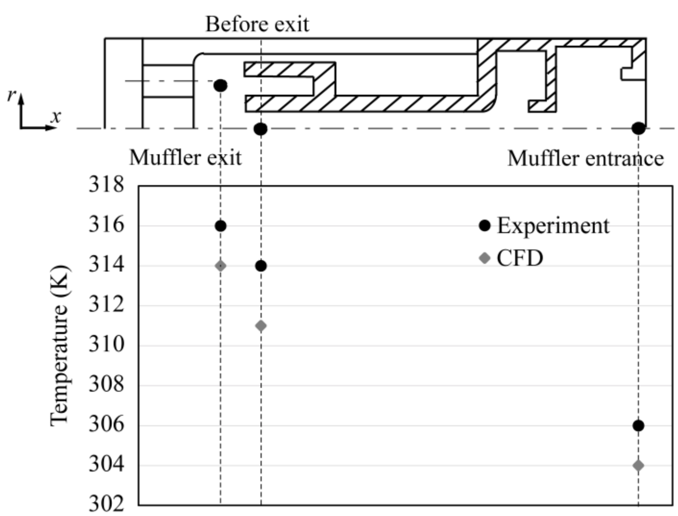


Figure 4: Temperature-monitoring point diagram for comparing experiment results with CFD data in case1

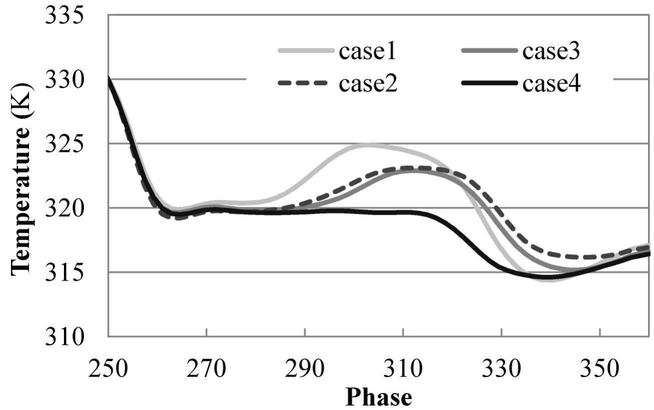


Figure 5: The temperature-phase diagram in muffler exit in case1, case2, case3 case4 during the suction process

On the other hand, case3 is applied only partition wall and the last one is applied all of them. We compare experimental data with CFD results to validate an accuracy of simulation. Figure 4 shows the temperature of each monitoring point in model with partition wall without cylindrical projection. The experiment and CFD data is monitored at the muffler exit, the muffler entrance and the distance of 0.01m from the muffler entrance. The results show a good correlation. The temperature difference of each monitoring point is less than 1%. Therefore, there results verify that CFD results are sufficiently reliable.

3.2 Comparative Analysis of Modified Muffler

In this paper, we analyze the internal flow in the muffler and modify the muffler to increase the compressor's efficiency. In Figure 5, we show the temperature-phase diagram at muffler exit during the suction process. The suction valve opens and refrigerant goes in the cylinder due to the low pressure field in cylinder, and this refrigerant temperature in the muffler exit of linear compressor represents the valve between 315 and 330 Kevin in all cases. In case 1 in which the partition wall and cylindrical projection are not adopted, the temperature increases about 325 Kelvin in about 300 degree of phase and decreases 314.5 Kelvin in about 340 degree of phase. In case2 and case3, as in case1, similar results are shown in which the temperature increases about 323 Kelvin, and decreases 316 and 314 Kelvin, respectively. The temperature in case3 is lower than that in case2 by difference between 0.5 and 1. However, the trends of temperature are greatly similar. In case4 in which the partition wall and cylindrical projection are adopted, the temperature is about 320 Kelvin in about 313 degree of phase and decreases about 314.5 Kelvin in about 338 degree of phase.

We investigate the temperature contour to explain the result aforementioned. In Figure 6, we show the temperature in case3 and case4 at the muffler entrance. We provide the temperature contour in case 1 and case4 in Figure 7. The cylindrical projection interrupts the diffusion of cool refrigerant going out in the shell space. Hence, in case4, the temperature can be lower than that in case3. Furthermore, in case1, 2, and 3, the temperature is higher than that in case4 due to the effect of the hotter refrigerant coming from the upper side of the muffler.

In Figure 8, we show the pressure-phase diagram in muffler exit of all cases in the whole process of compressor. In case2 and case3, the pressures are similar in the compression, discharge, and suction processes. Furthermore, the pressure of case1 is lower than that of case2, 3, and 4 in about 300 degree of phase.

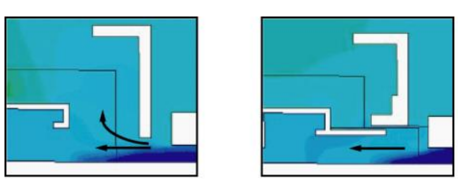


Figure 6: The temperature contour of case3(left) and case4(right)

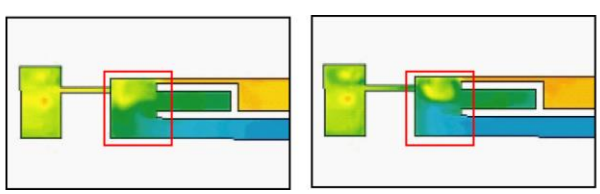


Figure 7: The temperature contour of case1(left) and case4(right) in the suction process

In Figure 9, we show the time average pressure at each position of suction line in the suction process. In case4, the narrow suction line made in the muffler entrance by cylindrical projection and partition wall, and that increases the pressure field in the suction line. Hence, in the case4, the pressure maintains highest in suction line, the pressures are similar in case2 and 3, and the pressure is lowest in the case1. And, this pressure field develops pressure difference of between inner cylinder and muffler exit. Note that the internal field of the cylinder has relatively low pressure in the suction process. The pressure differences between the inner and outer cylinder are about 146Pa, 157Pa, 176Pa, and 188Pa in case1, 2, 3, and 4, respectively. Due to the high pressure difference between the muffler exit and the cylinder, the cool refrigerant of suction line can be moved fast in the suction process, and it can interrupt hot refrigerator of muffler upper side with flowing in muffler exit. Hence, case4 represents the highest compressor efficiency as a result in Figure 10. We expect increasing the compressor efficiency when the pressure can be high at the muffler exit and the superheating loss is reduced in suction line.

4. CONCLUSIONS

We can estimate compressor efficiency in according with muffler design by using CFD tool. Through this study, we can obtain conclusions as in the following.

- We suggested improving suction muffler of the linear compressor by using CFD.
- The refrigerant of low temperature flows into the cylinder when the pressure in muffler exit is highly maintained during the suction process.
- The efficiency of compressor improved when refrigerant flows in the suction line without superheating loss.

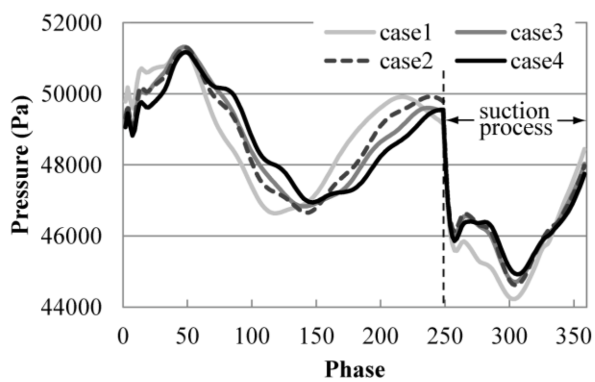


Figure 8: The pressure-phase diagram in muffler exit in case1, case2, case3 and case4 in the whole process

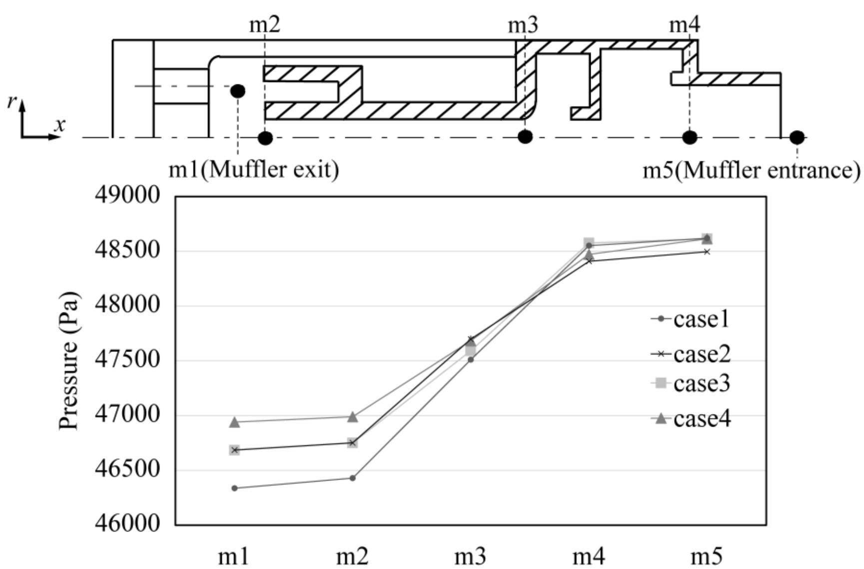


Figure 9: The pressure averaged time-monitoring point diagram in the suction process

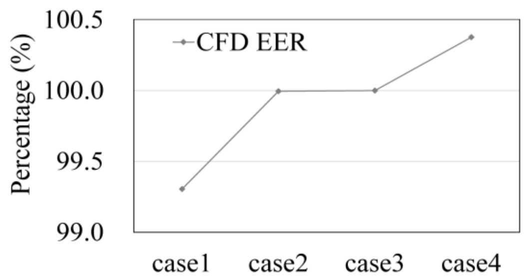


Figure 10: The percentage of the compressor EER for comparing experiment with CFD data in case3 and case4

NOMENCLATURE

T	temperature	(K)
P	pressure	(Pa)
r	radial direction	(m)
x	*x* direction	(m)

Subscript

cs	cylindrical surface
p	piston surface
s	suction
sh	shell
d	discharge

REFERENCES

U.S. Energy Information Administration, Residential Energy Consumption Survey (RECS) 2009, How Energy Is Used in Homes (2009).

www.lg.com, Discover LG Refrigerator Technology, Guaranteed Reliability.

Kemal, S., Ahmet, O., Emre, O., & Atilla, K. (2012), An Experimental and Numerical Analysis of Refrigerant Flow Inside the Suction Muffler of Hermetic Reciprocating Compressor, *International Compressor Engineering Conference at Purdue*.

Kwangha, S., Daenyoung, H., & Hyeongseok, K. (2006), CAE/CFD Application for Linear Compressor, *International Compressor Engineering Conference at Purdue*.

C, J. D., F, C. P., E, & L. L. P. (2002), Numerical Simulation of Pulsating Flow in Suction Mufflers, *International Compressor Engineering Conference at Purdue*.

Umut, C. C., Hasan, G., & Kemal, S. (2013), Numerical Investigation of Suction Muffler in Household Refrigerator Compressor, *International Conference on Mechanics, Fluids, Heat, Elasticity and Electromagnetic Fields.*

ACKNOWLEDGEMENT

This research was supported by L & A Research Center of LG Electronics.