2000

A Study on the Flow in a Discharge System of the Reciprocating Compressor Using Computational Simulation and PIV

M. K. Chung
LG Electronics Inc.

S. J. Baek
LG Electronics Inc.

I. S. Lee
LG Electronics Inc.

H. S. Rew
LG Electronics Inc.

Y. J. Bae
LG Electronics Inc.

See next page for additional authors

Authors
M. K. Chung, S. J. Baek, I. S. Lee, H. S. Rew, Y. J. Bae, Y. H. Moon, and T. W. Yun

A STUDY ON THE FLOW IN A DISCHARGE SYSTEM OF THE RECIPROCATING COMPRESSOR USING COMPUTATIONAL SIMULATION AND PIV

Moon Kee Chung*, Seung Jo Baek*, In Seop Lee*, Ho Seon Rew*,
Young Ju Bae**, Young Ho Moon**, Tae Whuan Yun**

*Digital Appliance Research Lab., LG Electronics Inc., 327-23, Gasan-Dong, Keumchun-Gu, Seoul, 153-023, Korea
**Digital Appliance Division, LG Electronics Inc., 391-2, Gaeumjeong-Dong, Changwon, Gyeong Nam, 641-711, Korea

ABSTRACT

This study presents some preliminary results on the flow characteristics in the discharge plenum of the reciprocating compressor using visualization technique and computational simulation. During the compression cycle, the flow from the cylinder through the discharge port induces a significant amount of pressure pulsation mainly due to the interaction of the discharge valve system and the flow. In order to improve the compressor efficiency, this pressure pulsation as well as the back flow through the valve must to be reduced. Additionally, in order to develop an optimum shape of the head cover in the sense of reducing the pressure loss, it is necessary to understand the characteristics of the compressed gas flow inside the discharge plenum. To investigate the flow precisely, PIV(particle image velocimetry) and computational simulation were used to visualize the overall flow field around the discharge system.

INTRODUCTION

The research topics on the household refrigerator can be divided into two main categories; the reduction of energy consumption because it is the largest household consumer of electricity and a reduction of noise level to create a more comfortable indoor environment[1]. To meet two main requirements of consumers, it is necessary to develop a more energy-efficient and low noise reciprocating compressor, which can be accomplished by characterizing the flow loss and flow induced noise through the complete flow survey inside the discharge system of the compressor as well as by optimizing of the mechanical parts and developing high-efficient driving motor.

Fig. 1 shows the schematics of the discharge system in the reciprocating compressor, which is composed of the discharge port, discharge valve, valve spring, retainer and head cover. A low pressure refrigerant gas from the evaporator is compressed to a high pressure that is balanced to the spring force of the discharge valve and valve spring, and it is discharged into the head plenum where the unsteady pulsating flow is relaxed to the specific pressure value(e.g. P_d=13.98 kgf/cm^2). The flow characteristics in the reciprocating compressor during the compression/discharge cycle shows complex flow pattern such as unsteady impinging jet through the discharge valve and valve spring, intense vortex generation due to the interaction between the high speed/pressure flow and the discharge system. Most of works concerning the discharge system were based on the assumption of the quasi-steady state, one-dimensional flow, and experimental works regarding full flow survey inside the head plenum have not been performed[2][3][4]. However, the detailed analysis of the flow characteristics inside the head plenum is essential to develop

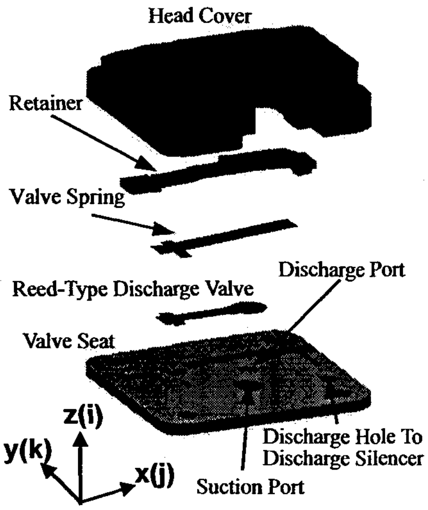

Fig. 1. Schematic view of the discharge valve assembly system and coordinate system

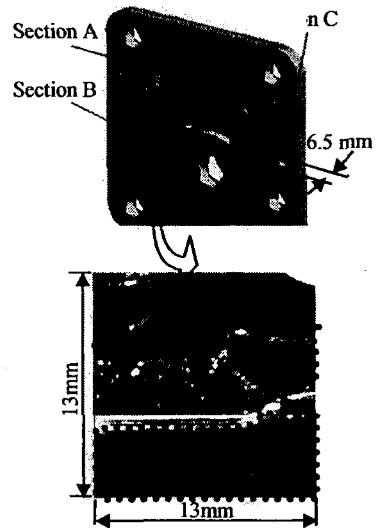

Fig. 2. Typical view of the discharge valve system and the measurement section

Fig 3. Grid system of discharge valve system

high-EER(energy efficiency ratio)/low-noise compressor because high-speed flow through the discharge port/valve induces a significant amount of flow loss and noise as well as the vibration of the reed-type valve. In order to get a discharge flow information during the running of the compressor, the pulsating flow inside the head plenum was investigated using the PIV and computational simulation. Gross flow patterns ahead of the discharge system(Fig. 1)(assembled parts of the discharge port, discharge valve, valve spring and retainer) were visualized according to the cycle-by-cycle variation and the detailed information about the flow interaction between the discharged gas flow and the valve system was obtained.

EXPERIMENTAL APPARATUS AND COMPUTATIONAL SIMULATION

For the visualization study, a reciprocating compressor except the shell was installed to the test jig which can drive the crank shaft of the compressor with a constant rotational speed of 3000rpm. The transparent visualization model of the compressor head cover was constructed using acrylic resin plate and aluminum base in order to prevent a leakage from the contact area between the head cover and the valve seat. Fig. 2 shows the assembled discharge system and the measurement section. Section A corresponds to an area having the highest velocity of the compressed gas flow from the discharge port/valve, section B to an area in the resonance volume taking part in reducing pulsation, and section C to an area where the compressed gas is discharged to the discharge silencer/loop pipe. All measurement sections were positioned off 6.5mm from the valve seat plane. To get an phase-averaged mean velocity field in an each section, the Z-pulse from the rotary encoder which was attached to the crank shaft was fed into the delay pulse generator. User-selectable delayed pulse from the delay pulse generator triggered the PIV system to synchronize with the crank angle. At each crank angle(θ=90°,135°,180°,225°,270° with respect to the bottom dead center), total 50 flow images were acquired and processed to give out an ensemble averaged velocity field using the cross-correlation PIV techniques. Since a maximum instantaneous velocity of the discharge gas flow was about 30m/s at the section A and the size of the measurement section was about $13\times13mm^2$, the separation time between the first flow image and the second one Δt had to be kept so small enough to minimize the error vector. In this study, particle image pairs with the time interval of Δt=2.6~5μs were captured on full-frame CCD camera which have spatial resolution of 1008×1018 pixels.

For accurate PIV measurements the seed particles should be distributed homogeneously over the measurement area, not alter the properties of the fluid or the flow, and accurately follow the fluid motions[5]. Glycerine and ester type lubricating oil were atomized by atomizer and supplied to the suction muffler of the compressor.

For the computational simulation, the geometry of the discharge system of the reciprocating compressor was rebuilt using 3D-CAD modeling and the grid was generated by body fitted grid method to apply the real geometry to numerical calculation(Fig. 3). Although the discharge system consists of head cover, retainer, valve spring, discharge valve and discharge port, the valve spring was excluded in the numerical calculation since the role of valve spring is to reinforce the stiffness of discharge valve. During the discharge period, the gap between retainer and valve spring, or between discharge valve and valve spring is very small. It was assumed that the flow disturbance by valve spring would not be large. The flow field was calculated by the commercial software, FLUENT using the k-ε turbulence model.

The calculation domain is from the top of cylinder where the refrigerant flows into discharge plenum starts, to the pipe where the refrigerant flows out to the discharge silencer. The boundary conditions for inlet, outlet and plenum wall were obtained from experimental data operated in ASHRAE condition. The valve displacement was measured at the tip of the discharge valve and it was assumed linear in the numerical calculation.

RESULTS AND DISCUSSION

Fig. 4 shows the phase-averaged mean velocity fields and speed contour at the section A. Since this section corresponds to the plane just ahead of the retainer, a compressed high-speed flow from the discharge port vigorously interacts with the valve and retainer resulting in the severe flow loss at the

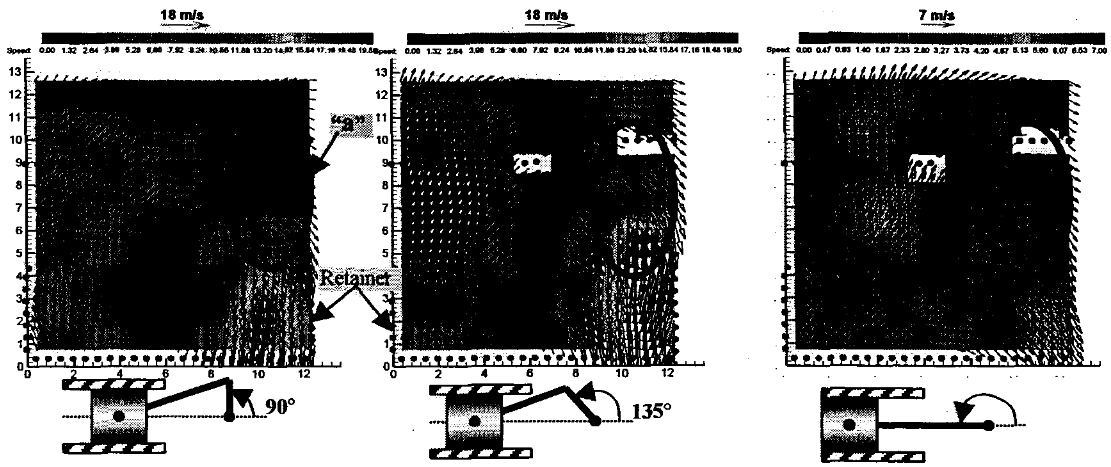

Fig. 4 Phase-averaged velocity vector field and speed contour at each crank angle(Section A)

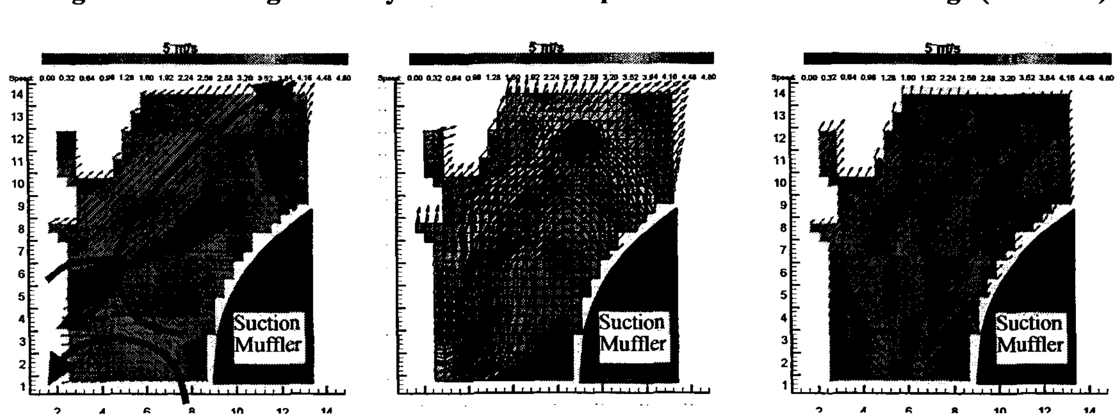

Fig. 5 Phase-averaged velocity vector field and speed contour at each crank angle(Section B)

discharge period. There is large-scale vortex motion over the section A due to the jet-like flow impinging on the discharge valve and retainer. Because the deflected flow along the bottom plate of the valve meets the step on the valve seat and a part of the retainer(denoted region as "a" on the figure), it is directed into z-direction, and then, flows out to the section C. At θ=135°, mass flow through the port increases due to the full opening of the discharge valve, the maximum mean flow velocity attains 20m/s above the retainer, "a". At θ=180°, overall flow velocity decrease as the discharge valve come to close, but the instantaneous velocity is still comparable to the one at the previous crank angle. The turbulent intensity(not shown here) around "a" shows the maximum at θ=135° indicating strong flow-induced noise may occur around this location.

Contrary to the velocity field at section A, overall velocity at the section B shows relatively low value(Fig. 5). The unsteady pulsating components are effectively damped in this region, and a part of flow recirculates in the lower parts of the plenum around the suction muffler. At θ=135°, the pulsating flow pattern appears in this measurement section and then it is redirected into the discharge hole.

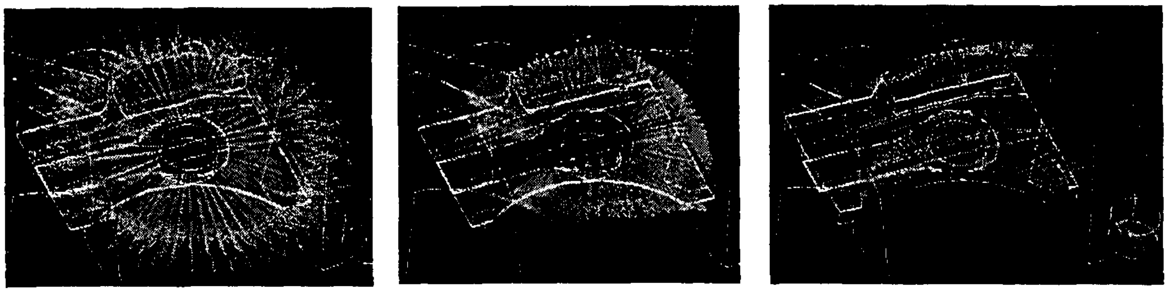

Fig. 6 Velocity vector field around the discharge valve system; (a) at the lower valve plane (b) at the upper valve plane (c) at the mid-plane between the valve and retainer

At the time the valve begins to open, the pressure in the cylinder reaches up to 20 kgf/cm^2, however, the pressure rapidly decreases and maintains to the discharge pressure, 16 kgf/cm^2. The pressure in the discharge plenum also varies, but the degree is much smaller, about 0.5 kgf/cm^2. The flow characteristics around the discharge valve by computational simulation is shown in Fig. 6. Since stagnating flow develops on the lower face of the discharge valve, velocity magnitude is small while pressure is high. However reaching to the tip of valve, static pressure turns into dynamic pressure and develops high velocity field(Fig. 6(a)). Strong velocity field appears in radial direction and the maximum velocity reaches up to 100 m/s. On the plane of upper face of the discharge valve(Fig. 6(b)), velocity magnitude varies a lot depending on the valve seat geometry. It is noticeable that the main flow separates into two parts, the one flows to left side where the discharge silencer is located and the other flows to right side, the contrary direction from the discharge silencer because of geometrical vicinity between the discharge valve and valve seat step. Fig. 6(c) shows 3 dimensional flow development by hitting the valve seat step. The flow turns the direction from horizontal to upward being divided to left and right side.

As the discharge period undergoes, the velocity of the refrigerant in the plenum rapidly decreases and the average is just about 10 m/s due to the expansion of flow path. Fig. 7 and Fig. 8 show the velocity field and pressure field on the middle plane of the discharge plenum during the discharge period respectively. If the discharge period is to say, T, during the beginning of the period((a),(b) in Fig. 7,8) the flow separates into two directions. One flows to the left side where discharge silencer is located so that flows out through it even though some collision loss occurs at the upper left corner. However the other part, flowing to the right side along the wall of the suction port, becomes accumulated and develops high pressure region. As the discharge valve starts to close, the refrigerant which was accumulated in the right region begins to flow out through the discharge silencer((c),(d),(e),(f) in Fig 7,8).

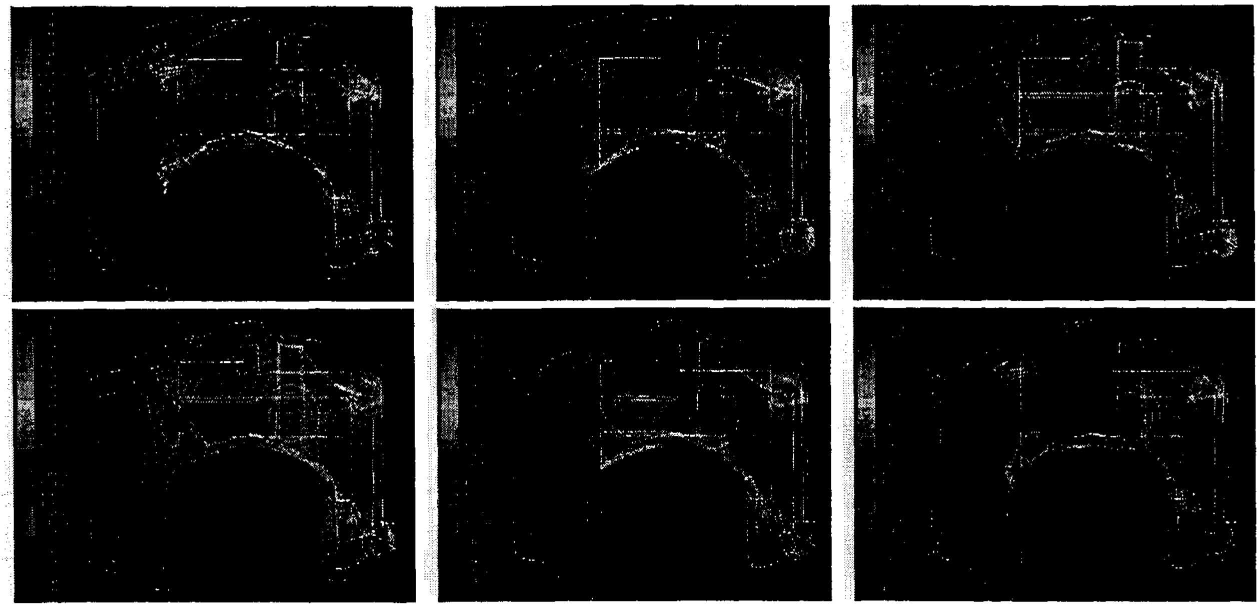

Fig. 7 Velocity field on the middle plane during the discharge period
(a) 1/6T, (b) 2/6T, (c) 3/6T, (d) 4/6T, (e) 5/6T, (f) 6/6T

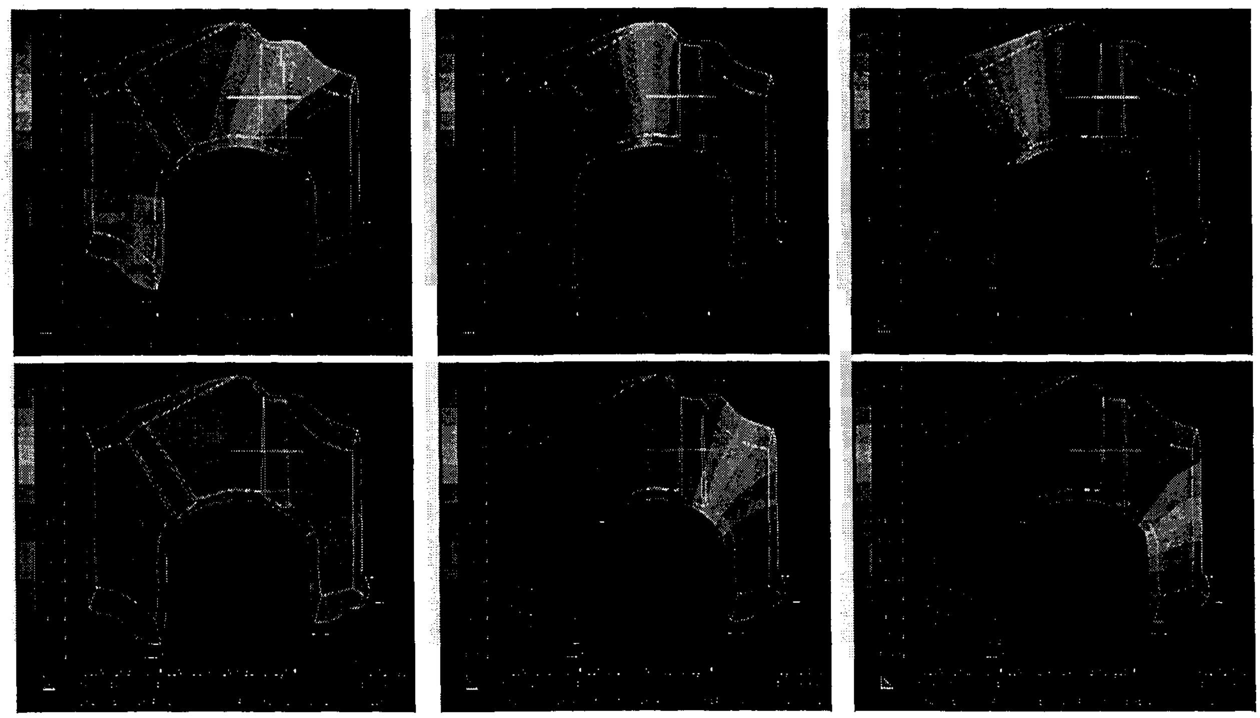

Fig. 8 Pressure field on the middle plane during the discharge period
(a) 1/6T, (b) 2/6T, (c) 3/6T, (d) 4/6T, (e) 5/6T, (f) 6/6T

CONCLUSION

In this study, the flow fields inside the head plenum of the reciprocating compressor have been investigated using the cross-correlation PIV technique and the computational simulation. The velocity field in the head plenum was analyzed by ensemble averaging according to the crank angle. Numerical simulation was conducted with 3D-CAD model and experimental data. Due to the significant velocity variation around the discharge system and the narrow flow passage between the valve and the valve seat, major flow loss occurred on the valve seat step. When crank angle is 135°, maximum velocity and turbulent intensity were measured on the upper region of retainer, which is believed to generate pressure pulsation in the plenum and induce the considerable noise.

REFERENCES

[1] Vineyard E.A., Sand J.R. and Bohman R.H.: Evaluation of Design Options for Improving the Energy Efficiency of an Environmental-ly Safe Domestic Refrigerator-Freezer, *ASHRAE Transactions.*, 101(1), p.1422, 1995.

[2] Leopold B.: Non Steady Flow in Valves, *Purdue Compressor Tech. Conf.*, p.664, 1990.

[3] Piotr Cyklis: CFD Simulation of the Flow Through Reciprocating Compressor Self-acting Valves, *Purdue Compressor Tech. Conf.*, p.427, 1994.

[4] C.J. Deschamps, R.T.S. Ferreira and A.T. Prata: Turbulent Flow Through Valves of Reciprocating Compressors, *Purdue Compressor Tech. Conf.*, p377, 1996.

[5] Westerweel J., Draad A.A., Hoeven J.G.Th. and Oord J. van.: Measurements of Fully-Developed Turbulent Pipe Flow with Digital Particle Image Velocimetry, *Exp. In Fluids*, Vol.20, p.165, 1996.