1998

Application of Computational Fluid Dynamics to Rotary Compressor Efficiency and Noise

J. O. Ban
LG Electronics Inc.

U. S. Lee
LG Electronics Inc.

B. H. Ahn
LG Electronics Inc.

Y. S. Kim
Pukyoung National University

APPLICATION OF COMPUTATIONAL FLUID DYNAMICS TO ROTARY COMPRESSOR EFFICIENCY AND NOISE

Ban Jong-Ok *, Lee Un-Seop** Ahn Byoung-Ha***
*Researcher ** Senior Researcher Rotary OBU
***Senior Researcher Living System Laboratory
LG Elec. Co., Ltd. Changwon, Kyoungnam, KOREA

Kim Young-Soo
Professor, Pukyoung National University, Pusan, KOREA

ABSTRACT

In this paper CFD(Computational Fluid Dynamics) analysis for the rotary compressor focused on the valve environment including muffler and cylinder has been investigated. From this analysis, it is possible to obtained flow pattern and environment from cylinder to muffler part in rotary compressor. Among the results of this analysis, it shows that the shape of retainer is very closer relationship between EER(Energy Efficiency Ratio) and noise.

1. INTRODUCTION

Recently, most electric living goods are required to have more efficient air-conditioning and more quiet environment in the room. These aspects make it possible to study of compressor about the phenomena of flow using various tools by means of various parameter. One part of these research movement, CFD is accepted as one of the tools for EER improvement and noise reduction based on the flow resistance analysis. So this paper focused on the designing geometry of valve environment of rolling piston type compressor of air-conditioning system.

Pumping parts of compressor is very complex and very sensitive in EER and noise. Therefore, it is difficult to decide the range of analysis. Generally valve system located between muffler and cylinder and it affected severely these two kinds of characteristics ,i.e. EER and noise. Due to the analysis of valve alone is meaningless job, CFD analysis must be contained the cylinder and muffler condition to get more exact solution. From this point of view, scope of analysis should be expanded to cylinder and muffler.

CFD is helpful tool in many point of view. Actually, it is important that what kind of procedure or what particular data obtained from CFD analysis, can help to increase the EER and to reduce the noise. The first helpful data is flow losses across the discharge port and valve. Information of this data can be used as a guide information to the geometry improvement to reduce the flow losses.

. In this study, design alternatives of valve parts such as retainer or notch angle, were estimated considerably, and output data of analysis such as pressure or kinetic turbulence energy term, also was used. Finally, How applied the data of analysis to the compressor also must discussed.

2. ANALYSIS

All the geometry in a rotary compressor, is three dimensional and the fluid flow pattern is turbulent and compressible. Available CFD package contain above requirements and there are several leading programs. In the presented paper, Fluent was adopted on the workstation and all analysis executed a steady state assumption having stationary boundary conditions. This commercial program have several restriction such as moving grid and boundary and based on the $k-e$ model. Dynamic valve motion or pressure characteristic analysis is very difficult with itself. But stationary boundary condition and steady state analysis can provide pressure drop information due to the flow losses. But, in actual system, rolling piston and valve are moving parts at every time without any stationary state. It was very difficult to decide the critical point in a rotating roller angle. From the empirical concept, the analysis points were decided to focus on discharge process. Among the discharge process, analysis point are fixed on the starting state of discharge process. This process would be expected rapid velocity and pressure changes in the pumping parts and most sensitive notch angle and retainer height . Pressure is applied to the roller wall surface as a mean pressure distribution and back pressure is also applied in the outlet port of muffler.

Starting point of discharge process provides many useful information about cylinder, roller and vane. Additionally it was helpful to the analysis of flow pattern of cylinder. The main concerning aspects are not leakage of vane or roller but the pressure generation and propagation phenomena in the cylinder volume.

$$k=\frac{1}{2}(\overline{U}_1^2+\overline{U}_2^2+\overline{U}_3^2) \quad (1)$$

This paper shows some results , one is pressure distribution and the other is kinetic turbulence energy(Eq.1). Pressure drop can be calculated from pressure distribution and kinetic energy distribution which present the velocity energy of a certain point in three dimensional analysis. Therefore, it represents the value of kinetic component. Generally fluid borne noise depends on the velocity as following relationship[1]:.

$$Aerodynmic\_Noise \propto V^{6-9}$$

$$Vibrational\_Induced\_Noise \propto V^{2.5}$$

. where v is velocity. Above relationship is a little bit different according to the each case of flow but it shows that the higher velocity zone have a lot of possibility which produce noise.

3. MODELING AND BOUNDARY CONDITIONS

Fig.1 show three dimensional grid model of pumping parts. This model consists of three parts. One is cylinder , another is notch and valve and the other is muffler part. The cavity of muffler do not allow any leakage. The discharge valve position was selected from experimental value and also selected the dimension of cylinder and roller position by the experiments.

Boundary conditions includes the geometric boundary and inlet and outlet boundary condition. In the geometric boundary condition, it should be careful to describe the valve port and retainer, because grid confusing has occurred by many intersection areas and lines.

Except cylinder and outlet port, all other parts are assumed as wall condition without having any leakage. The known boundary condition for the cylinder are roller velocity or roller wall pressure only. But roller velocity is very difficult to apply as an input condition because it must be considered the vector components which produced by crank shaft rotating. Pressure boundary condition is quite simple comparing of velocity input condition. In this paper the pressure which acting on the cylinder wall was assumed as a mean pressure. Outlet port of muffler needs another boundary condition. Also velocity or pressure boundary condition was expected as back pressure. This paper choose the mean back pressure across the outlet port as boundary condition considering the shell pressure.

4. SIMULATION RESULTS

There are several key parameters of to enhanced EER and to minimized noise for the rotary compressor. Among these parameters, select notch angle was appeared in Fig.3 and analysis was executed in the case of 35, 45, 55 degrees of notch angle as shown in Table.1. In this case, all the performance was enhanced in case C. The results appeared as a contour plot of pressure in Fig.3. These figure was 2-dimensional cross section which selected from 3-imensional analysis. This analysis show that the notch angle has very affectable on the defining of flow in discharge process in this analysis and there is lots of possibilities which depend on the flow volume. According to the reference[2] , it shows opposite result in this paper. The difference of this paper and the reference is the object of analysis. This paper was evaluated a standard 19,000 Btu/hr and reference was evaluated 12,000 Btu/hr grade.

Next, retainer height and angle variation was calculated as following table2. Pressure drop is biggest in case 1 and smallest in case 3.(Fig.2) But the difference kinetic turbulence energy was differ from each case of pressure drop. Case 3 was the smaller value than case 2 in almost area across the valve as following Table.3 Probably, it may happen that the existence of return flow in valve port area was conceived(Fig.4). Returning flow depends upon the shape of discharge recess space. According to this analysis, the more increase the notch angle, the more move toward to the center of pressure at circumference direction. Fig.5 shows the measuring point of kinetic turbulence energy term.

Table.1 parameter of notch angle

Item	Angle(degree)
Case A	35
Case B	45
Case C	55

Table.2 Parameter of retainer and height

item variable	Retainer Angle	Retainer end height
Case.1	10 degree	2.4 mm
Case.2	15 degree	3.0 mm
Case.3	20 degree	4.0 mm

Table.3 Calculation Results

Item	case	Case 1	Case 2	Case 3
Pressure Drop				
Kinetic Turbulence Energy	Pos.1	20.9	31.6	30.4
	pos.2	7.0	19.2	25.3
	pos.3	11.0	37.3	33.9
	pos.4	23.3	33.8	31.1
	pos.5	14.9	21.8	29.5

5.EXPERIMETAL RESULTS

To examine the effect of above analyses were estimated by a standard 19000 Btu/hr rotary compressors under ASRE/T condition. Table.4 shows the result of this experiment. Performance is proportional to the angle increase, but the noise is different. Case 2 is the lowest level of noise.

Table .4 Experimental Results

item		Case 1	Case 2	Case 3
Performance	Btu / hr	19318	19251	19130
	Consumer E(Watt)	1874	1856	1838
	EER	10.31	10.37	10.41
Noise	X -direction	65.6 dB(A)	64.6 dB(A)	66.5 dB(A)
	Y-direction	66.2 dB(A)	64.9 dB(A)	65.5 dB(A)

6. CONCLUSION

The CFD analysis is very helpful tool which can be tried various geometry changes to determine their difference using flow loss. Through the analysis , EER increase without noise increment.

Reference

(1) Chisachi Kato et al , Ph.D thesis, Tokyo University, 1995,

(2) Chisachi Kato Lecture note of Tokyo university

(Based on *: Chisachi Kato et al. "Numerical simulation of aerodynamic sound radiated from low mach number turbulent wakes" ASME 1995 Feb. Vol.219 pp53-58

(3) James R. Lenz et al, "Application of computational fluid dynamics to compressor efficiency improvement" 1994 Int. Comp. Eng. At Perdue University, 1994

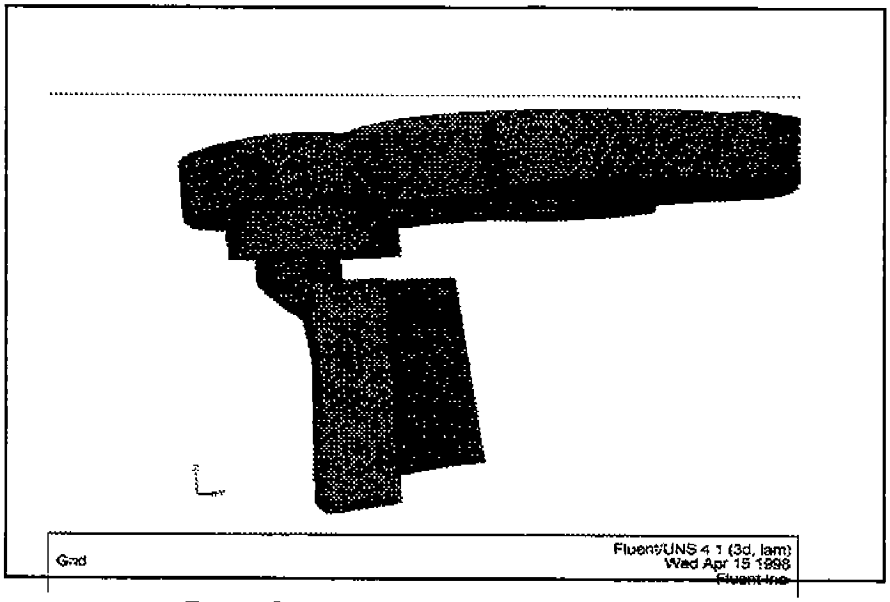

Fig.1 Grid of 3-dimensional Modeling

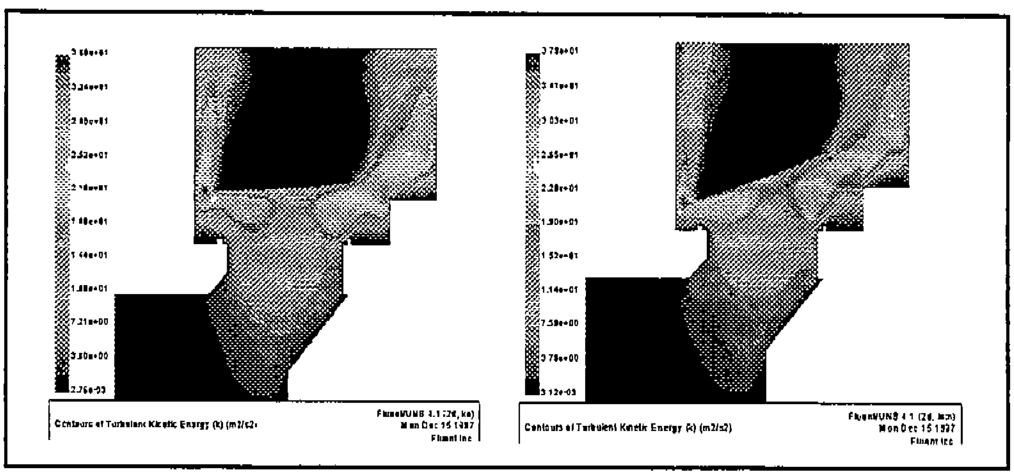

Fig.2 Cross section Drawing of rotary comp
(Left :10 degree / Right : 20 degree)

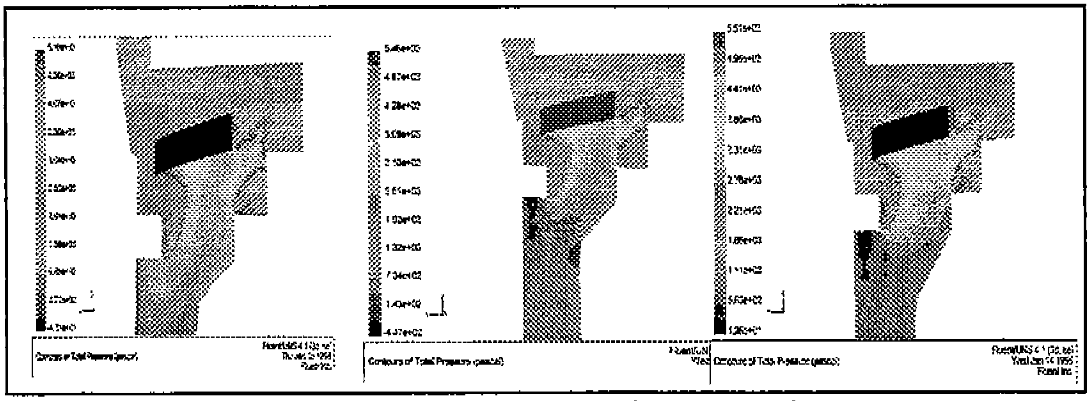

Fig.3 Pressure contour of notch angle
(Left : 35 degree / Middle : 45 degree / Right : 55 degree)

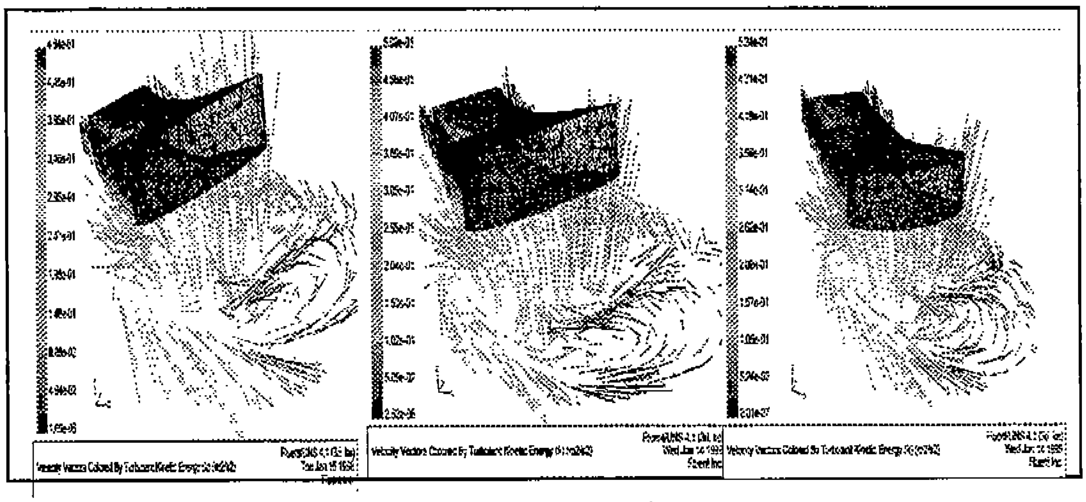

Fig.4 valve return flow
(Left : 35 degree / Middle : 45 degree / Right : 55 degree)

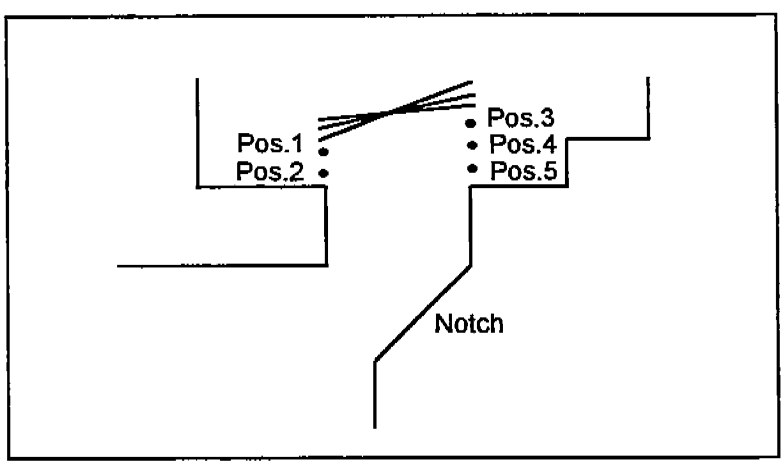

Fig.5 Kinetic Turbulence Energy measuring point