International Compressor Engineering Conference

School of Mechanical Engineering

2004

The Performance Optimization of Rolling Piston Compressors Based on CFD Simulation

Wei Geng
Shanghai Hitachi Electrical Appliance Co.

Chun Hui Liu
Shanghai Hitachi Electrical Appliance Co.

Yan Zhen Wang
Shanghai Hitachi Electrical Appliance Co.

THE PERFORMANCE OPTIMIZATION OF ROLLING PISTON COMPRESSORS BASED ON CFD SIMULATION

Wei Geng, Chunhui Liu, Yanzhen Wang

R&D Center, Shanghai Hitachi Electrical Appliance Co., Ltd
Shanghai, 201206, China
Phone: 086-021-50554560 ext. 5210
Fax: 086-021-58991084
E-mail: gengw@shec.com.cn
liuchh@shec.com.cn

ABSTRACT

In this paper, the simulation of rolling piston compressor is established using Star-CD, a general *computational fluid dynamics (*CFD) software. Based on the result, the P-V diagram of compressor's working process and all kinds of factors influencing the compressor's overall efficiency are analyzed deeply, then several optimum projects are put forward in order to improve the compressor's overall efficiency. Finally, all the former projects are proved by theoretical CFD simulation respectively.

1. INTRODUCTION

During the past few decades, tremendous improvements of system performance have been achieved in the field of refrigeration and air conditioning. How to reduce the energy consume is still the focus of the researchers in this field. The compressor companies are facing the topic of developing higher efficiency compressors.

The main function of rolling piston type compressors is to carry out the suction, compression and discharge of refrigerant gas. The refrigerant in the compressor is three-dimensional compressible fluid and turbulent flow. Because of the high speed of compressor, the structure parameters of the suction and discharge channels have a great effect on compressor efficiency. Without the properly designed suction line, throttling will occurred and the actual refrigerant enclosed into the cylinder will be decreased. Volumetric efficiency will be degraded. Serious suction throttling will lead to impetuously pressure pulsation in suction pipe. During compression, high-pressure refrigerant gas in the cylinder will go through the notch, discharge port, valve stop, valve recess and muffler step by step. The parameters of discharge line will significantly influence the over-compression and re-expansion loss of compressor. Only enlarging discharge line maybe decrease over-compression loss, while the re-expansion loss of high-pressure refrigerant in the clearance volume will be increased synchronously. Thus volumetric and indicated efficiency will be degraded. The target of the optimal design is to obtain the optimum overall efficiency by balancing all the efficiencies.

This paper shows how to simulate the transient refrigerant gas flow in a rolling piston type compressor using STAR-CD, a general-purpose computational fluid dynamics (CFD) code. Results show the detailed gas flow in the compressor, such as velocity distribution, pressure distribution, flow resistance, and so on. The P-V diagram and efficiency of compressor also can be calculated by the result. Then several optimum projects are put forward in order to improve the compressor's overall efficiency and all the projects are analyzed by theoretical CFD simulation respectively.

2.MODEL AND BOUNDARY

The refrigerant in the rolling piston type compressor is three-dimensional compressible fluid and turbulent flow. In order to show the fluid flow in the inner compressor and comprehensively analyze the compressor efficiency, this paper use the transient moving-mesh model to simulate the refrigerant working process (Liu C., 2003).

In order to simulate the suction, compression and discharge process of compressor, the computational domain includes the whole fluid in the compressor pump. The model begins from the intake of cylinder suction line and ends of the discharge outlet in the upper muffler (or upper bearing).

According to the compressor working process, muffler, suction hole in the cylinder and discharge port are immovable parts, which are immovable mesh. Along with the crankshaft rotating, the piston rolls along the inner wall of cylinder. The volume of suction chamber and compression chamber varies, which are moving mesh. The clearance between cylinder inner wall and piston outer wall is so small and only allows a little refrigerant flowing through. Also there is oil sealing when compressor operates. So we assume there is no flow between the above two chambers. The pressure change in suction chamber and compression chamber is mainly caused by the volume variety according to the crankshaft rotation. Though there is relative rotation between piston and crankshaft, the shear motion caused by self-rotation of piston is omitted because which influences the gas flow little. The motion of leaf valve is controlled by the pressure difference between compression chamber and muffler. This paper simplified the valve motion to pressure-activated motion. The valve opens when the pressure in the cylinder compression chamber is larger than that in the muffler. The higher the pressure difference, the larger the valve lifts. The valve begins to regress until the pressure in the cylinder compression chamber is less than that in the muffler.

Mesh is assembled by diversified type. The immovable mesh is accomplished using Proam package. The moving mesh is deforming mesh and accomplished using Prostar package. The movement of rolling piston is controlled by .cgrd file. When the leaf valve open and close is controlled by condition event.

The initial inlet and outlet boundaries of this model are all set to pressure boundary. The attachment between discharge port and compression chamber is arbitrary sliding mesh. The refrigerant's thermophysical property is defined based on simple PR equation, whose density is the function of both temperature and pressure as listed in equation (1). Turbulence in the flow is modeled using the standard k-ε high Reynolds model. Heat exchange is not considered in this analysis.

$$\rho = f(T, P) \tag{1}$$

3. SIMULATION RESULTS

The compressor with lower discharge port, as Fig.1 shows, has been analyzed. The compressor's suction pressure is 0.625MPa. The suction temperature is 35°C. The discharge pressure is 2.146MPa. The rotating speed of compressor is 2840rpm. The refrigerant is R22.

3.1 P-V Diagram and Compressor Efficiency

The P-V diagram of compressor working chamber is shown in Fig.2. From it we can see, the over compression during the discharge process is high. The maximum pressure is close to 2.5MPa and energy loss is very big.

On the P-V indicator diagram, the indicator work of each compression cycle, ω, is identified as the enclosed area by the suction and compression chamber pressure curves. The actual mass flow rate of delivered refrigerant per cycle, m, is obtained from the simulation results. Then the cooling capacity of per cycle, q, can be calculated from equation (2). J represents the cooling capacity per weighing. We use the COP defined in equation (3) as the parameter to evaluate compressor performance.

$$q = m \times J \tag{2}$$

$$COP = q/\omega \tag{3}$$

Volumetric efficiency is defined as the ratio of the actually mass flow rate to the ideally delivered mass flow rate, which is proportional to the actual mass delivered by the compressor per cycle. In this paper, heat exchange between inner and outer calculated domain is ignored, so leakage and gas re-expansion are the main factors influence compressor volumetric efficiency. Compressor efficiency is listed in table 1.

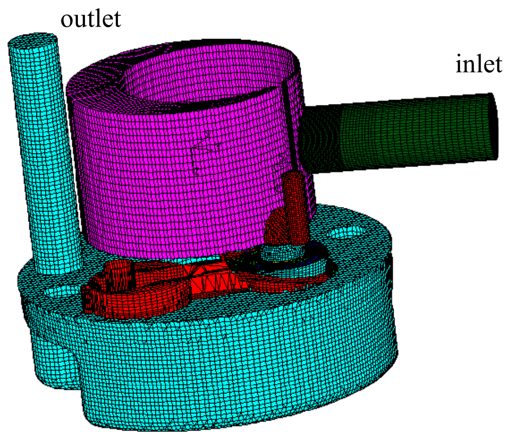

Figure 1: model

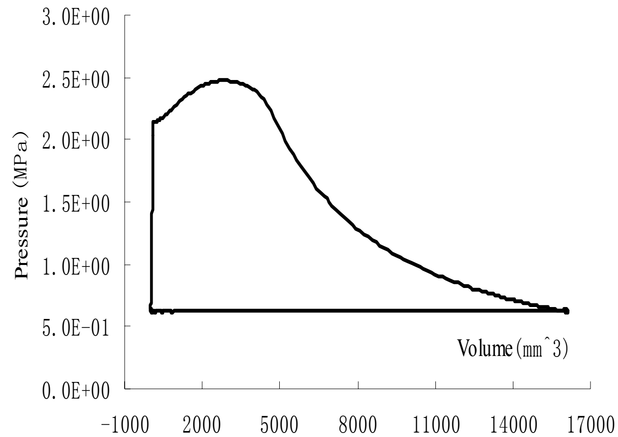

Figure 2:P-V diagram

Table1 Efficiency of compressor with lower discharge port and surge hole

cooling capacity of per cycle q（J/rev）	56.4
indicator work of per cycleω（J/rev）	13.9
COP =q/ω	4.057
Volumetric efficiency decrease by gas re-expasion（%）	9.9
Volumetric efficiency decrease by leakage（%）	6.2
Total volumetric efficiency（%）	83.9

3.2 Internal Flow

Figure 3 shows the velocity vector plot of discharge port at the crankshaft angle of 240 degree. When gas flow to the field marked with circle line, the flow section area expands suddenly. Big back flow exits in this area and also some sections have low velocity, which leads to clearance volume but little good effect on the pressure resistance decreasing. Since the target of expanding section is to increase the effective flow rate, this is a possible negative source to compressor performance. Expanding the section area gradually maybe have better effect.

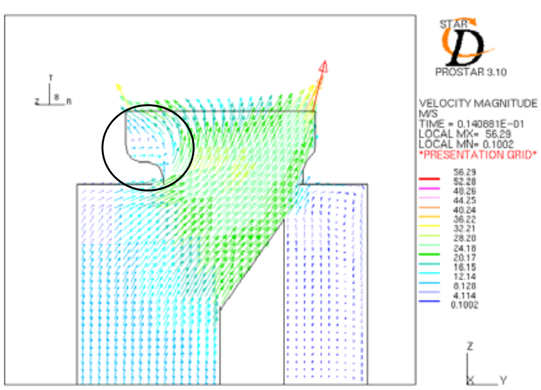

Figure 3: velocity vector plot of discharge port at the crankshaft angle of 240 degree

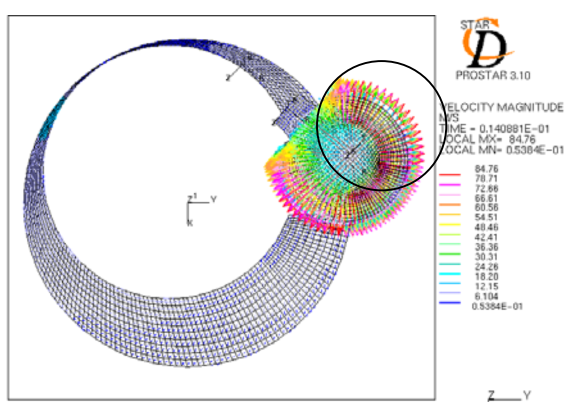

Figure 4: discharged gas flow of inner leaf valve at crankshaft angle of 240 degree.

Fig 4 shows the discharged gas flow of inner leaf valve at crankshaft angle of 240 degree. If the valve recess lies on the high velocity direction of gas flow marked with circle line, the gas flow will be improved more.

4. OPTIMAL PROJECTS

Based on the above analysis, several projects improving the gas discharge line have been put forward to increase the compressor efficiency and calculated to get the optimal project.

4.1 Comparison Between Compressors With Notch and Without Notch

The original intent of notch was to aid the discharge flow process by increasing the effective area where the circular port in the main bearing meets the cylinder compression chamber, consequently reduce the flow resistance and over compression loss and improve the compressor efficiency. While the re-expansion loss caused by high pressure gas in the notch at the end of discharge period can't be omitted. Fig 5 shows the PV diagram of single discharge compressor with notch, single discharge and double discharge compressor without notch. In the single discharge compressor without notch, the flow section area is reduced, which caused higher discharge resistance and larger over compression loss. The compression power is increased by 2.2%. Though the cooling capacity is increased by 1.7% and volumetric efficiency increased 1.6% because of the smaller clearance volume, the COP is descended. The effect of notch can't be omitted in single discharge compressor. While the COP of double discharge compressor without notch is increased by 5.4% with cooling capacity rising and compression power decreasing. The double discharge without notch structure is an effective way to improve the performance of small displacement compressors.

Table 2 Efficiency of compressor with and without notch (no surge hole)

	Single discharge	Single discharge	Double discharge
Notch	with	without	without
Cooling capacity, q（%）	57.7	58.7	59.2
Compression power, ω（J/rev）	13.9	14.2	13.5
COP=q/ω	4.15	4.13	4.38
Volume efficiency（%）	87.9	89.3	90.5

4.2 Performance Influenced By Notch Axes Intersect Cylinder Axes Or Not

Shown as the velocity vector plot, fig 4, high velocity gas flows through the circular port outlet mainly along the radial direction of cylinder center. While the valve recess in the main bearing is along the tangential direction of cylinder radius. So the flow resistance is big in this field. The compression gas is discharged along the tangential direction of cylinder center since the piston rotates along the cylinder inner radius. The gas is imposed to change the flow direction along the radial line since the notch axes intersects the cylinder axes. There must be larger pressure loss there. A new notch is designed to reduce the pressure loss, whose axes does not intersect the cylinder axes but along the tangential direction of cylinder radius. The analysis result of double discharge compressor with new notch is shown in Fig 6. From the velocity vector plot we can see that the flow direction of high velocity gas has been changed to consistent with the valve recess processing direction. The gas flow resistance is reduced and compression power during discharge process is decreased too. The COP is increased by about 0.8% listed in table 3.

Table 3 Efficiency of compressor influenced by notch axes intersect cylinder axes or not (no surge hole)

Notch axes intersect cylinder axes?	Yes	No
Cooling capacity, q（J/rev）	58.0	58.4
Compression power, ω（J/rev）	13.5	13.4
COP=q/ω	4.30	4.34
Volume efficiency（%）	87.4	88.5

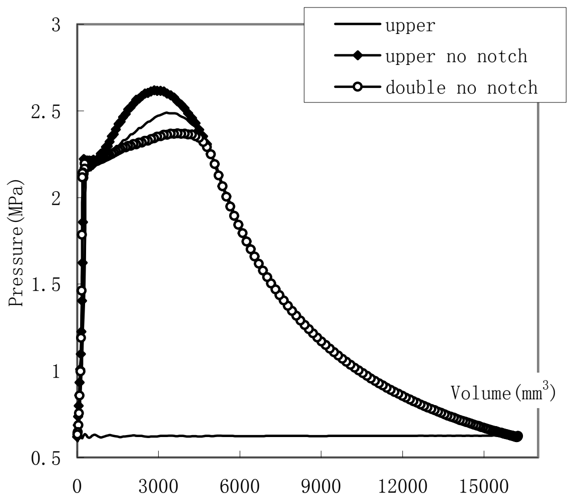

Fig 6 P-V diagram of compressors
with and without notch

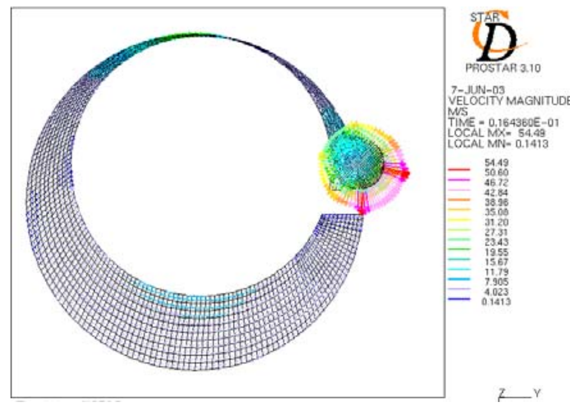

Fig 7 Velocity vector plot of notch axes
not intersect cylinder axes compressor

5. CONCOLUSIONS

This paper used the computational fluid dynamics analysis to simulate the working process of rolling piston type compressors. Compressor internal flow is shown and several optimal projects are analyzed. The results show double discharge compressor without notch and notch axes along tangential direction of cylinder radius are the effective way to improve compressor efficiency.

NOMENCLATURE

ρ	density	kg/mm^3
T	temperature	^oC
P	pressure	MPa
ω	indicator work of each compression cycle	J/rev
m	mass flow rate of delivered refrigerant per cycle	kg/rev
J	cooling capacity per weighing	J/kg

REFERENCES

Liu, C., Geng, W., 2003, A CFD based simulation of refrigerant gas flow in the rolling piston compressors, *Proc. ICCR*: p.1-8.

James, R.L., 1994, Application of computational fluid dynamics to compressor efficiency improvement, *ICECP*: p.441-446.

Ban Jong-Ok, 1998, Application of computational fluid dynamics to rotary compressor efficiency and noise, *ICECP*, : p.105-110.

Liu, C., Geng, W., 2003, CFD and compressor design, *Proceedings of SHEC* : p.97-101.