Purdue University
Purdue e-Pubs

International Compressor Engineering Conference
School of Mechanical Engineering

2008

CFD Applications for Development of Reciprocating Compressor

Akira Nakano
Matsushita Electric Industrial Co.

Kenji Kinjo
Matsushita Electric Industrial Co.

CFD Applications for Development of Reciprocating Compressor

Akira NAKANO[1]*, Kenji KINJO[1]*

[1] Refrigeration, Air Conditioning & Heating Research Laboratory,
Matsushita Home Appliances Company, Matsushita Electric Industrial Co., Ltd.
2-3-1-1 Nojihigashi, Kusatsu, Shiga 525-8520, Japan
Phone: +81-77-567-9801 Fax: +81-77-561-3201 E-mail: nakano.akira001@national.jp

ABSTRACT

This paper describes the analysis of pressure changes in the suction muffler of a reciprocating compressor using CFD, aiming at increasing cooling capacity and efficiency.

The numerical calculations take place in 2 stages. In the first stage, we calculate pressure changes in the cylinder, considering the vibration of the suction lead valve, using transfer matrix methods (4-pole relations) [1]. In the second stage, we carry out a numerical analysis of three-dimensional unsteady compressible viscous flow, taking pressure changes in the cylinder obtained in the first stage as the boundary condition. Then, the pressure changes in the suction muffler are analyzed and the mass flow rate of the refrigerant is maximized. For second stage computation FLUENT is used. As a result, along with obtaining findings for the reduction of the heating loss and pressure loss of the suction muffler, we could improve the suction muffler to increase the mass flow rate by 11%.

1. INTRODUCTION

In recent years, efforts aimed at energy saving have been accelerating throughout the world from the perspective of conservation of the global environment. Including the countries of Asia, where economic growth is marked, and the countries of Europe, which are working positively on energy saving, regulations on energy saving are becoming increasingly stricter every year. This trend has accelerated the energy saving competition among manufacturers of domestic refrigerators and in the future it will be necessary to pursue the positive development of higher efficiency for refrigerator cooling systems and reciprocating compressors.

Looked at globally, the reciprocating compressor is the main type of compressor used in domestic refrigerators, and it does not appear that this trend will change in the foreseeable future. Until now, competition for higher efficiency of reciprocating compressors has been conducted actively and various developments have been advanced from a number of perspectives. These efforts include the reduction of friction loss between reciprocating piston and cylinder, crankshaft and bearing, the reduction of compression loss caused by pressure changes in the suction line and discharge line, the reduction of motor loss based on the development of material such as magnetic steel sheets and the development of electronic control technology in areas such as capacity control.

On the other hand, due to the striking evolution in the processing speed and memory capacity of computer hardware and the development of the computational algorithms, CFD (Computational Fluid Dynamics), which previously could only be used in research and development for aircraft and automobiles, has reached the point where it can now also be used for the mass-production development of household appliances.

In particular, in cases in which CFD is used for purposes of energy saving in domestic refrigerators, CFD has been used for various viewpoint and CFD has become an essential development tool in the improvement of thermal insulation performance, the improvement of cooling performance within the refrigerator compartments, and the increase of efficiency in the cooling system.

In the development of compressors, there are issues in the complexity of part shapes and the complexity of the physical phenomenon of handling the compressible viscous flow that goes with oil and the arrival at practical calculation can be considered to be comparatively recent. However, positive efforts have also been made taking approaches that raise important themes in the area of scientific computation, including multi-scale analysis of the shell interior overall[2] and FSI (Fluid Structure Interaction) [3] analysis of the flow through the suction lead valve and motion of the suction lead valve, so there is some remarkable research in the development of CFD in the field of compressor development. By applying CFD to analysis of the behavior of the refrigerant and oil within the compressor in the development of reciprocating compressors for domestic refrigerators, we are currently refining test production specifications on the desktop and continuing to succeed in the reduction of test production expenses and the abbreviation of development periods. There are many examples of the application of CFD[4, 5] to the improvement of suction muffler specifications, in particular, and there is also a trend towards the very efficient use of numerical analysis in combination with DOE (Design of Experiment). However, there are thought to be few examples of research that has assumed flow phenomena in the suction muffler as the unsteady flow and discussed those phenomena in detail.

In this paper, after the overview of our CFD applications are introduced, computational results of physical quantity changes on time in the suction muffler are discussed and the characteristic improvement of the suction muffler is tried.

2. CONSTRUCTION OF A RECIPROCATING COMPRESSOR

The cross-section of a conventional reciprocating compressor is shown in Fig. 1, and the basic construction of the suction and discharge paths in Fig. 2. The piston and cylinder mechanism is located in the upper part of the hermetic casing, while the motor is placed in the lower part. As the piston of this compressor moves toward bottom dead center (suction stroke), the pressure in the cylinder reduces. Then the suction valve opens and refrigerant in the hermetic casing is guided into the cylinder through the suction muffler, valve plate and suction lead valve. When the piston moves toward top dead center (compression stroke), the refrigerant in the cylinder is compressed. Then the discharge valve opens and the refrigerant in the cylinder flows out to the refrigeration system through the valve plate, discharge valve and cylinder head.

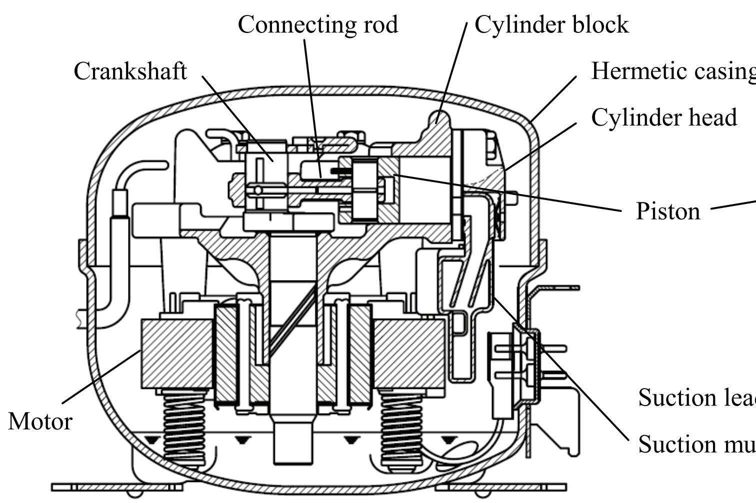


Figure 1: Schematic diagram of a reciprocating compressor

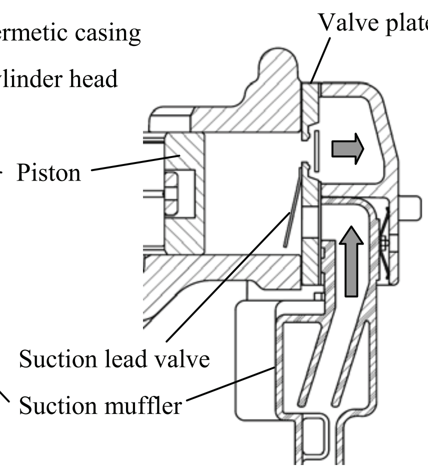


Figure 2: Close-up of suction and discharge paths

3. OVERVIEW OF CFD APPRICATIONS

Figure 3 shows a three-dimensional design model of a typical reciprocating compressor. At present, because developers often use the three-dimensional design in actual development, when CFD is used, they often simplify the design model overall or parts of it to create the CFD model. Here, we introduce a number of examples where we have applied CFD.

Figure 4 shows an example of analysis of the vicinity of the suction port, carried out to curtail pressure loss in the suction system. In these calculations, we modeled a suction lead valve that has begun to open and investigated the reduction of pressure loss in the path from the suction muffler outlet pipe to the cylinder.

Figure 5 shows an example where we analyzed refueling speed at the time the compressor is started to calculate how long oil retained in the lower part of the shell takes to reach the upper end of the crankshaft. In this example, having compared the calculated results and the experimental results, we confirmed that we forecast the speed of refueling after the compressor is started very accurately, with an experimental value of 0.86 seconds and a calculated value of about 0.8 seconds for the time required for the oil to reach the upper end of the crankshaft after the compressor is started. This analysis helped us to investigate the refueling specifications for the crankshaft.

Figure 6 shows an example of analysis of oil behavior between the piston and the cylinder. In this example, we were able to discover that the behaviour of oil retained in a groove was different as a consequence of the cross-sectional shape of the groove.

We have introduced above a number of examples showing our application of CFD, but it can still be thought that there is room for improvement in all of these examples in terms of both accuracy and calculation efficiency. On the other hand, we have also furthered the expansion of the scope of application positively.

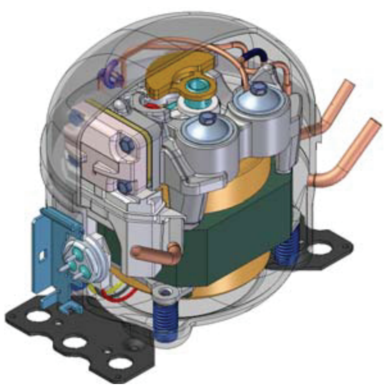
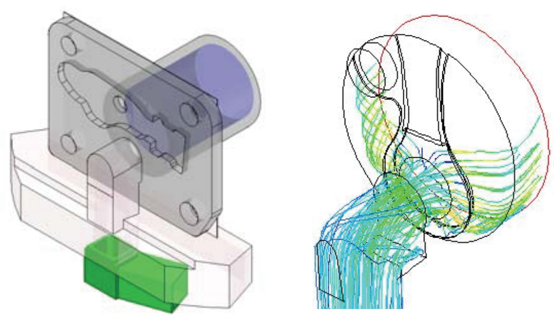
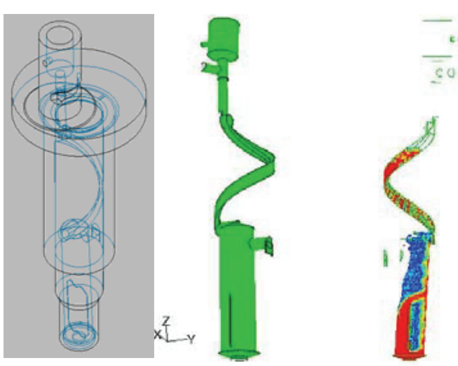


Figure 3: 3D-CAD model for mechanical design



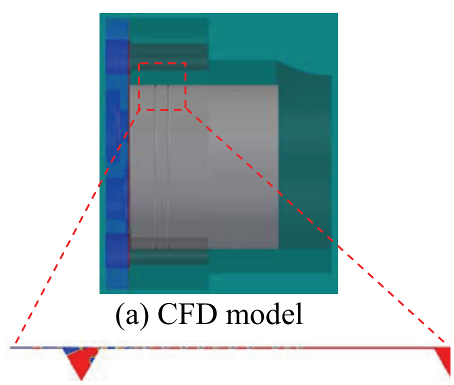
(a) CFD model (b) Stream lines

Figure 4: Examination of pressure loss of suction line



(a) CFD model (b) Volume % of oil (0.2s)

Figure 5: Examination of refueling hole specification



(a) CFD model

(b) Volume % of oil

Figure 6: Examination of leakage loss of cylinder

4. CFD APPLICATION FOR NUMERICAL ANALYSIS OF SUCTION MUFFLER

4.1 Outline of Analysis

We can calculate the characteristics of a suction system including a suction muffler using transfer matrix methods (4 pole relations)[1] of Yoshimura, et al, to forecast the variation over time of pressure in the cylinder, considerate of the eigenvalue of the suction lead. In this investigation, by carrying out a two-step calculation combining the analytical tools described above and CFD, we will consider the detailed shape of the suction muffler to describe the details of our efforts to reduce heat receiving loss. Figure 6 shows the calculation procedure.

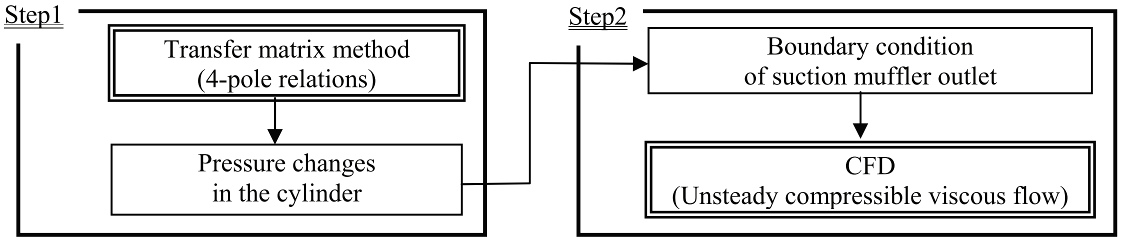


Figure 6: Procedure of analysis

The refrigerant gas in the suction muffler changes in accordance with the opening and closing of the suction lead valve in the inhalation and compression process caused by the moving back and forth of the piston in the cylinder. Table 1 shows the data for the reciprocating compressor used in this investigation and the average flow speed of the refrigerant gas at the suction muffler outlet estimated from this data. Because the flow speed of refrigerant gas is thought to be larger than the figures shown in Table 1, it is supposed that the refrigerant gas in the suction muffler is a pulsating flow of roughly subsonic velocity. Consequently, in this investigation, the pressure in the cylinder obtained using the 4 pole relations is assumed to be the pressure at the suction muffler outlet to calculate the unsteady compressible viscous flow of the refrigerant gas in the suction muffler using CFD.

Table 1: Main specifications of compressor and average speed of refrigerant flow

Volume of cylinder (cc, m^3)	Driving frequency (Hz)	Outlet Area of muffler (m^2)	Ave. flow speed at outlet (m/s)
10.0, 1.0×10^{-5}	50	5.0×10^{-5}	20.0

4.2 CFD conditions

The ASHRAE conditions are used as the compressor driving conditions. Table 2 and Figure 7 show the properties and boundary conditions used in the computation.

Table 2: Properties and boundary conditions

Properties	Refrigerant	R600a
	Density (kg/m^3)	Ideal-gas
	Cp (j/kgK)	1761.2
	Thermal Conductivity (W/mK)	0.0188
	Viscosity (kg/m・s)	7.938e-06
	Molecular weight (kg/kg・mol)	58.12
Boundary conditions (Figure 7)	B-1:pressure(kPa), Temperature(C)	63.6, 42.8
	B-2:coef. of HT(W/m^2K) , Temperature(C)	10.0, 60.0
	B-3:Temperature of valve plate(C)	100.0
	B-4:pressure (4-pole relations)	Time-depended

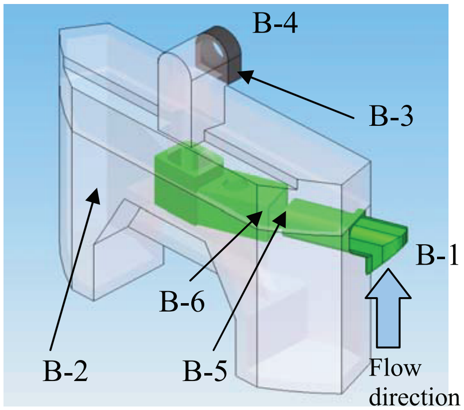


Figure 7: Suction muffler model

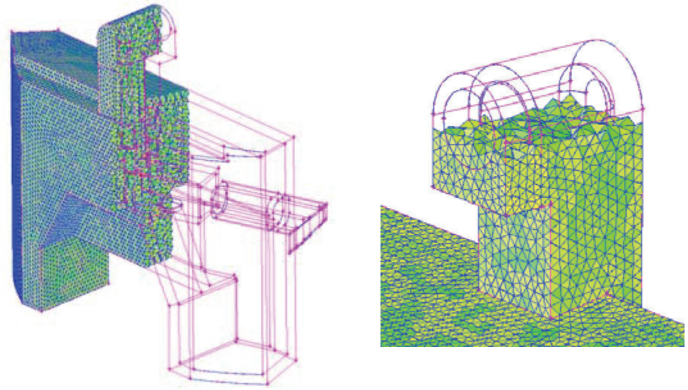


Figure 8: Space Division of flow field and solid body

We calculate density changes using a state eq uation supposing an ideal gas based on having R600a as the refrigerant and thermal properties are set for conditions as shown in Table 1.

Figure 8 shows an image of the space mesh division. The width of the finest mesh is 0.5mm, and the total number of mesh is about 500,000. In addition, because we also include in our calculations the thermal conduction of the suction muffler wall, the external wall of the suction muffler is part of the computational domain in addition to the fluid region in the muffler.

The boundary conditions for the inlet of suction muffler B-1 are set to evaporating pressure and temperature fixed at the experimental value. For the outer boundary of the suction muffler B-2, we set the thermal conduction rate supposing a shell-internal temperature of 60.0C. In addition, because we simulate the cylinder temperature, a valve plate is modeled and gave it a fixed temperature of 100.0C. Incidentally, we use experimental results for each of the settings. As for the boundary conditions of outlet B-4, we set the time-dependent variability of pressure in the cylinder obtained using the 4 pole relations and set velocity to zero after the pressure in the cylinder is risen to evaporating pressure.

Figure 9 shows the variation over time of the boundary condition of the suction muffler outlet B-4. From 0.0019 seconds, when the pressure in the cylinder starts to drop below the evaporating pressure, to 0.0119 seconds, when the pressure in the cylinder rises higher than the evaporating pressure again, we used the pressure obtained from the 4 pole relations and during the period when the pressure in the cylinder is higher than the evaporating pressure, we assumed that the suction lead valve is closed and set velocity to zero.

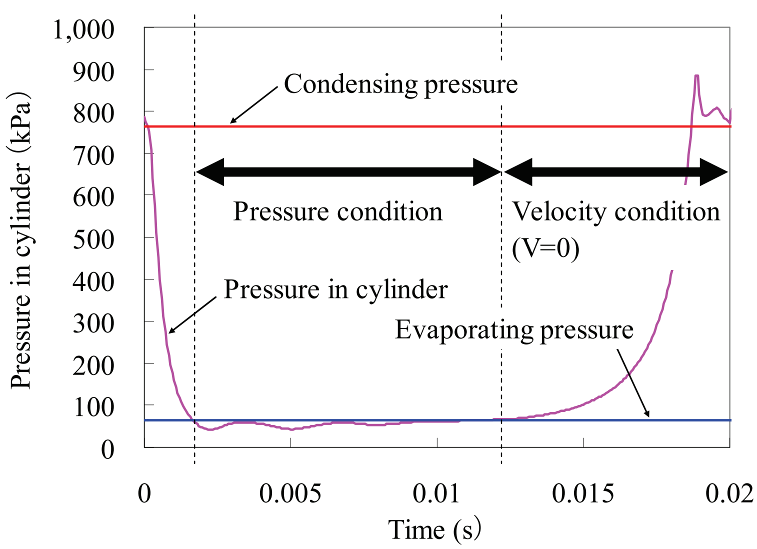


Figure 9: Boundary condition of suction muffler outlet B-4

4.3 Results of computation

Figure 10 shows the temperature transition of refrigerant at cross-sections B-1, B-5, B-6 and B-4 in the suction muffler in the 4th cycle of computation, when temperature transition had stabilized. The numbers of the cross-sections are shown Figure 7. In addition, Figure 10 shows average temperatures at suction muffler outlet boundary B-4.

The results of computation show inlet boundary B-1 at 42.8C and, in contrast, outlet boundary B-4 at 48.8C. In other words, the rise of temperature in the suction muffler is considered to be 6.0K. Having measured the gas temperature in the suction muffler using thermo couples under experimental conditions the same as in the calculations, the rise of temperature between the inlet and outlet was about 5.7K, so we were able to confirm that the computational results were good agreement with experimental values.

In addition, in contrast to the temperature at inlet boundary B-1 of 42.8K, we understand that the temperature at outlet boundary B-4 in the vicinity of 0.0023 seconds when the suction lead valve was initially at its most open, was lower than the temperature at inlet boundary B-1. This is thought to be because the refrigerant gas had expanded in the suction muffler.

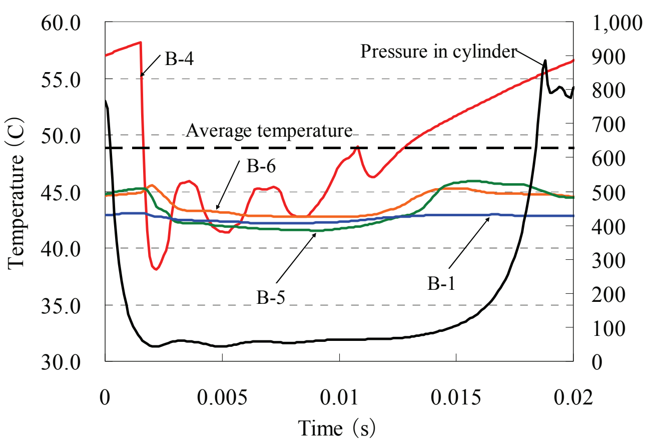


Figure 10: Temperature changes on time

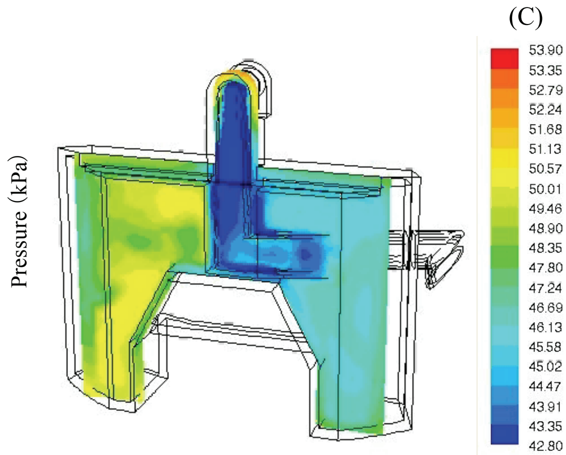


Figure 11: Temperature distribution at first peak of valve lift (0.0023s)

Furthermore, we were able to confirm that during the period when the suction lead was closed, the temperature of the refrigerant gas at the muffler outlet rises higher than the average temperature. This is thought to be because the temperature due to the compression of the refrigerant in the cylinder undergoes heat transfer by conduction from the cylinder block to the refrigerant gas via the valve plate. Because of this, by discovering specifications that suppress heat transfer from the cylinder block and the valve plate, we were able to discover the potential to suppress the rise of temperature of the refrigerant gas in the suction muffler.

Figure 11 shows the temperature distribution in the suction muffler at 0.0023 seconds. We can confirm form Figure 11 that the temperature at outlet boundary B-4 of the suction muffler is lower than at inlet boundary B-1. In other words, while the refrigerant gas is inhaled into the cylinder, the temperature of refrigerant gas actually inhaled is lower than the average temperature over time of refrigerant gas at B-4. It is considered that the temperature of the refrigerant gas while the suction lead valve is closed is a factor that rises the average temperature.

Figure 12 shows the density changes on time similar to Figure 10. We can confirm from Figure 12 that the density of 1.32kg/m^3 prior to the lead valve opening falls 27% to 0.97kg/m^3 when the suction lead valve is open and the refrigerant gas is inhaled into the cylinder. Pressure changes are the same as density changes.

Figure 13 shows the stream lines in the suction muffler at the same instant (0.0023 seconds). We can confirm from Figure 13 that at the instant the lead valve opens the flow speed of the refrigerant gas at the bend in the outlet pipe and in the vicinity of the outlet boundary is in excess of 80m/sec. Considering that the acoustic velocity of R600a is about 220m/s, at the instant the refrigerant gas is inhaled into the cylinder, the flow in the suction muffler can be estimated to be at a subsonic velocity approaching more than mach 0.3 locally.

In addition, Figure 14 shows the mass flow rate changes on time of the refrigerant gas. From Figure 14, we can understand that suction muffler outlet boundary B-4 follows the pressure boundary condition profile sensitively, and the mass flow rate changes have sharp peaks. From these results, the refrigerant gas can be inhaled more efficiently by improving getting depressed of mass flow rate at outlet boundary B-4.

On the other hand, we can understand that along with the reduction in the peak value of mass flow to the extent of going back along the flow path from outlet boundary B-4, outlet pipe inlet B-6, inlet pipe outlet B-5, inlet boundary B-1, the flow slows to a crawl and a temporal lag is generated. This is thought to be due to the refrigerant gas retained in the suction muffler expanding. Then to improve getting depressed of mass flow rate and the expansion of refrigerant gas, the improvement of suction muffler was tried by the following viewpoints, decrease of pressure loss of outlet and inlet pile, spread of expansion volume.

Having tested the specification improvements based on the perspective described above, we were able to discover specifications that control the rise of temperature in the suction muffler at about 3K and improve the mass flow rate of refrigerant by 11%. Test production experiments that used CFD together are advanced in the future, aiming at increasing cooling capacity and efficiency.

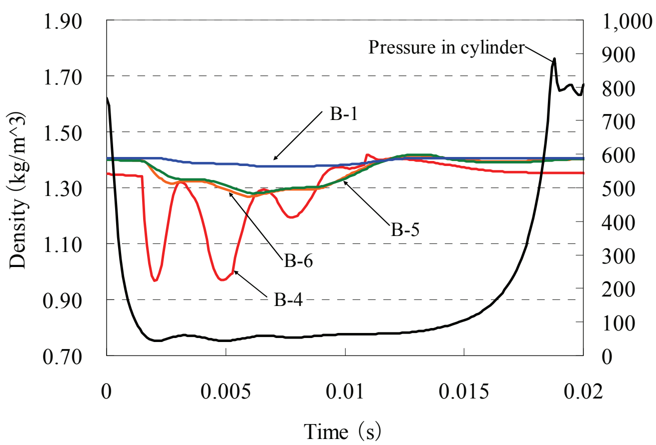


Figure 12: Density changes on time

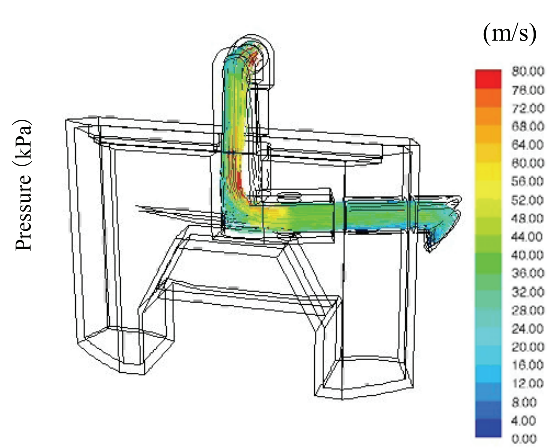


Figure 13: Stream lines at first peak of valve lift (0.0023sec)

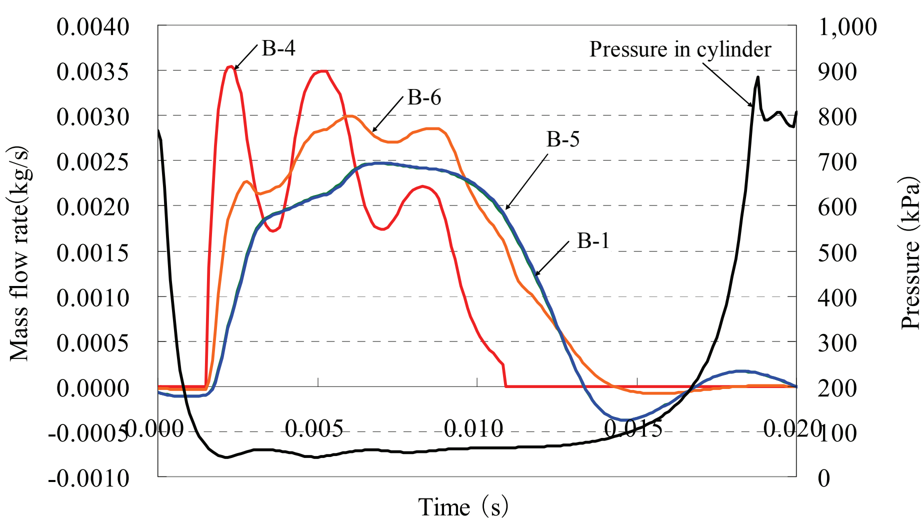


Figure 14: Mass flow rate changes on time

5. CONCLUSIONS

We estimated the pressure changes on time in the cylinder following consideration of the frequency of vibration of the suction lead valve using the 4 pole relations of Yoshimura, et al, and made that pressure in the cylinder the boundary condition to carry out three-dimensional CFD of the suction muffler. FLUENT is used for CFD.

The behavior of the refrigerant gas in the suction muffler showed itself to be essentially dependent on time based on the opening and closing of the suction lead valve during the inhalation and compression process. Consequently, we obtained the following findings as a result of testing numerical analysis of unsteady compressible viscous flow.

- The results of numerical analysis combining the 4 pole relations and CFD were very consistent with experimental data. The difference in the difference in temperature between the suction muffler inlet and the suction muffler outlet measured using thermo couples, and the average difference in temperature over time between the suction muffler inlet and the suction muffler outlet obtained using CFD was about 5%.
- The temperature of the refrigerant gas when the suction lead opened and it was actually inhaled into the cylinder was lower than the average temperature over time of the suction muffler outlet and there are cases when it is also lower than the temperature of the muffler inlet due to the expansion of the refrigerant gas in the muffler.
- The average temperature over time of the suction muffler outlet is thought to be pulled up by heat transfer from the cylinder block and the valve plate when the suction lead is closed.
- The flow rate of the refrigerant gas in the suction muffler is thought to achieve subsonic velocity in places.
- Comparing the mass flow rate changes on time of the suction muffler inlet cross-section and the suction muffler outlet cross-section shows that in contrast to the mass flow rate of the inlet cross-section changing comparatively gently, the suction muffler outlet cross-section responds sensitively to the vibration of the suction lead valve and the mass flow rate changes greatly. Consequently, increasing mass flow of the refrigerant is thought to be possible by raising the decline in the mass flow rate at the suction muffler outlet cross-section.

REFERENCES

1. Yoshimura T., Akashi H., et al., 2002, 'The Estimation of Compressor Performance Using A Theoretical Analysis of the Gas Flow through the Muffler Combined with Valve Motion', *International Compressor Engineering Conference at Purdue*
2. Y. V. BIRARI, et al., 2006, 'Use of CFD in design and Development of R404A Reciprocating Compressor', *International Compressor Engineering Conference at Purdue*
3. J. Kim, S. Wang., et al., 2006, 'Valve Dynamic Analysis of a Hermetic Reciprocating Compressor', *International Compressor Engineering Conference at Purdue*
4. B-H. Kim, et al., 2004, 'Design of the Suction Muffler of a Reciprocating Compressor Using DOE', *International Compressor Engineering Conference at Purdue*
5. V. K. Rao, et al., 2004, 'Analysis of Refrigerator Compressor Using CFD', *International Compressor Engineering Conference at Purdue*

ACKNOWLEDGEMENT

The authors would like to express their sincere appreciation to K. Sawai, F. Sakima, and K. Horii for their support in applying CFD to the development of reciprocating compressors.