NUMERICAL MODELING OF A REFRIGERATION RECIPROCATING COMPRESSOR

Evandro L.L. Pereira, evandro@polo.ufsc.br
Programa de Pós-Graduação em Engenharia Mecânica
Universidade Federal de Santa Catarina
88040-900, Florianópolis, SC, Brazil

Cesar J. Deschamps, deschamps@polo.ufsc.br
Departamento de Engenharia Mecânica
Universidade Federal de Santa Catarina
88040-900, Florianópolis, SC, Brazil

Fernando A. Ribas Jr.
EMBRACO
89219-901, Joinville, SC, Brazil
Fernando_A_Ribas@embraco.com.br

Abstract. *A two-dimensional model has been developed to simulate reciprocating compressors for household refrigeration under actual operating conditions. The suction and discharge systems of this type of compressor adopt automatic valves that open and close according to the pressure difference between the cylinder and the suction/discharge chambers. Once the valves are open, the time dependent flow field through the valve dictates the pressure load on the valve. Since the valve displacement has a strong effect on the flow field, the valve dynamics has to be coupled and solved simultaneously with the flow. In the present work, a one-degree of freedom model is used to describe the valve dynamics, whereas a finite volume methodology is employed to predict the flow field. Due to the valve and piston motions, a deforming grid was chosen to allow the variation of the computation domain. Results for velocity flow field, valve displacement, pressure pulsation in mufflers are presented for an assessment of the model capability and an analysis of the main thermodynamic losses.*

Keywords*: refrigeration, reciprocating compressor, compressor valves, valve dynamics.*

1. INTRODUCTION

Reed type valves are essential components in a number of reciprocating compressors, particularly the hermetic ones employed in refrigeration. These valves open and close depending on the pressure difference between the cylinder and the suction and discharge chambers. These pressures, however, are function of piston reciprocating movement, valve dynamics and muffler's geometry. In designing the valve system in reciprocating compressors four main features related to the valve performance are sought: fast response, large mass flow rate, low pressure drop when opened, and good backflow blockage when closed. To satisfy the performance requirements necessary for a competitive compressor, a detailed understanding of the fluid flow through the valve as well as the dynamics of the valve is necessary.

The impressive growth in interest in computational fluid dynamics (CFD) for engineering applications is mainly associated with the fact that computers are becoming more powerful and less expensive, making CFD feasible for simulations of much larger problems. However, in the case of reciprocating compressors, most simulation methodologies are still based on integral formulations, usually evaluating the compression process through the use of polytropic exponents or the first law of thermodynamics. For instance, in the simulation procedure proposed by Ussyk (1984) the flow through valves is modeled through the concept of effective flow and force areas, with the valve dynamics being described according to a simple one-degree-of-freedom model. Some analyses require a more accurate approach and differential formulations are usually preferred instead of integral formulations. For instance, Fagotti and Possamai (2000) and Ottitsch (2000) have adopted CFD to design specific components of reciprocating compressors, such as valves and mufflers, analyzing their performance in isolation from the other components.

Several works have been carried out to study the flow through compressor valves, by using a simplified radial diffuser geometry. Some of these studies have considered the flow through the valve in a fixed opening position and others have included the effect of the valve dynamics. Simulations employing the radial diffuser geometry have also been used to provide the effective flow and force areas data for the integral formulations. Salinas-Casanova *et al.* (1999) present a numerical analysis for the turbulent flow in radial diffusers, considering the effect of the reed inclination on the flow. Results were obtained for different Reynolds numbers and valve displacements and indicated that the RNG k-ε model, proposed by Orzag *et al.* (1993), is appropriate to predict this kind of flow. Lopes and Prata (1997) developed a numerical methodology to analyze the incompressible laminar flow in radial diffusers, with the frontal disk dynamics being solved through a one-degree-of-freedom model. Numerical predictions for an imposed periodic flow condition showed a good agreement with experimental data. Later on, Matos *et al.* (2002) extended the methodology proposed by Lopes and Prata (1997), so that turbulent flow could also be considered.

A two-dimensional differential methodology to simulate the whole operating cycle of a small refrigeration reciprocating compressor was presented by Matos *et al.* (2006), including the compressible turbulent flow inside the cylinder and the time dependent flow field through the discharge valve. The suction valve was modeled by the effective flow and force areas concept, while the mufflers were not considered. Results for the cylinder pressure field demonstrated a significant restriction for the flow along the cylinder clearance when the piston approaches the top dead center and the discharge valve is open. A similar model was presented by Pereira *et al.* (2006) employing a commercial code, but including the effect of the discharge muffler through a simplified geometry of tubes and chambers. Because the main emphasis was on the discharge system, an ideal suction process was assumed in the analysis.

From the literature review, it becomes clear that integral models are useful for analyses of the compressor global performance, despite their limitations to provide detailed information about the many physical phenomena that affect the compressor efficiency. On the other hand, most differential models are just concerned about either the suction or the discharge system, and only very recently some attempts have been made to simulate the complete compressor, including the piston motion.

The present paper proposes a two-dimensional numerical model for the analysis of reciprocating compressors typically employed for household refrigeration purpose, taking into account the piston motion, the valve dynamics, and the gas pulsation in suction and discharge mufflers. The interaction between reed valve motion and fluid flow is included in the model, using a simple one-degree-of-freedom model to solve the valve dynamics. Experimental data are used to validate the numerical predictions.

2. MATHEMATICAL MODEL

The compressible turbulent flow that prevails in the compressor was solved through the concept of Reynolds-averaged quantities, in which the value of a computed variable represents an ensemble average over many engine cycles at a specified spatial location. The turbulence transport contribution was modeled through the RNG k-ε model, which has been extensively used and validated for flow through simple geometries of compressor valves (Salinas-Casanova *et al.*, 1999). Non-Equilibrium wall functions approach was chosen to modeling the near-wall region. An equation of state for an ideal gas completes the system required to solve the compressible flow.

In this study the reed was considered to be parallel to the valve seat, with its dynamics being represented by a one-degree-of-freedom model:

$$m_{\mathrm{eq}} x + c\,\dot{x} + k\,\ddot{x} = F_{\mathrm{p}} + F_{\mathrm{o}} \tag{1}$$

where m_{eq}, c and k are the reed equivalent mass, damping coefficient and stiffness, respectively. On the other hand, F_p is the force induced by flow on the reed and F_o can accommodate any other force, such as reed pre-tension and also stiction that may occur due to the presence of a lubricating oil film between the reed and valve seat. Finally, x, $\dot{x}$ and $\ddot{x}$ are the instantaneous reed lift, velocity and acceleration, respectively. The equivalent mass m_{eq} is determined from data for reed stiffness, k, and natural frequency, f_n. In order to solve Eq. (1) for the valve displacement x, force F_p has to be evaluated from the pressure field created by the flow through the valve. For the discharge valve, a booster is set to act when the reed valve reaches a certain displacement and the valve lift is limited to a maximum value.

3. NUMERICAL SOLUTION PROCEDURE

The computational model was developed with a commercial CFD code (Fluent Inc., 2006) based on the finite volume methodology. Due to the presence of moving surfaces such as the piston inside the cylinder and suction and discharge valves, a moving grid strategy had to be applied to simulate the compression process. Grid and time refinement analyses were performed pursuing the best compromise between time simulation and solution accuracy. A second-order upwind scheme was adopted to interpolate the flow quantities, except for the turbulent quantities which were interpolated using a Power Law scheme for numerical stability reasons. A first-order implicit formulation was chosen for time-dependent variables. The coupling between the pressure and velocity fields was achieved with the SIMPLEC scheme. The system of algebraic equations was solved with a segregated algorithm.

The available moving mesh model in the commercial CFD code adopted in this work imposes restrictions on the maximum possible time step. The model does not allow that a moving surface shifts more than the smaller cell height adjacent to it along a certain time step. In this way, smaller grids require smaller time steps. Due to numerical stability aspects of the solution procedure and also because of constraints associated with the deforming mesh method, two different time steps had to be used according to whether the valves were open or closed. Thus, the time steps were set to values corresponding to 0.5 degrees of crankshaft angle when the valves were closed and 0.1 otherwise. In relation to the grid refinement it was adopted a structured mesh containing about 40,000 cells, with dimensions between 0.05 and 0.5 mm.

The solution domain includes the geometries of the suction and discharge valves and mufflers, and the compression chamber formed by the cylinder and the piston. The compressor shell is not included at this stage. An axisymmetric geometry was defined and the suction and discharge valves were positioned opposite to each other in relation to the compression chamber. Therefore, the suction and discharge muffler geometries were simplified to tubes and chambers, according to geometric dimensions. During the suction and discharge processes, the valves move simultaneously with the piston motion. Figure 1 shows a representation of the axisymmetric solution domain at two piston positions: (a) near the bottom dead center (BDC) and (b) near the top dead center (TDC). It should be mentioned that the geometry dimensions in the figure are merely illustrative and do not represent the actual geometry of the compressor. Furthermore, the arrows point out the flow direction.

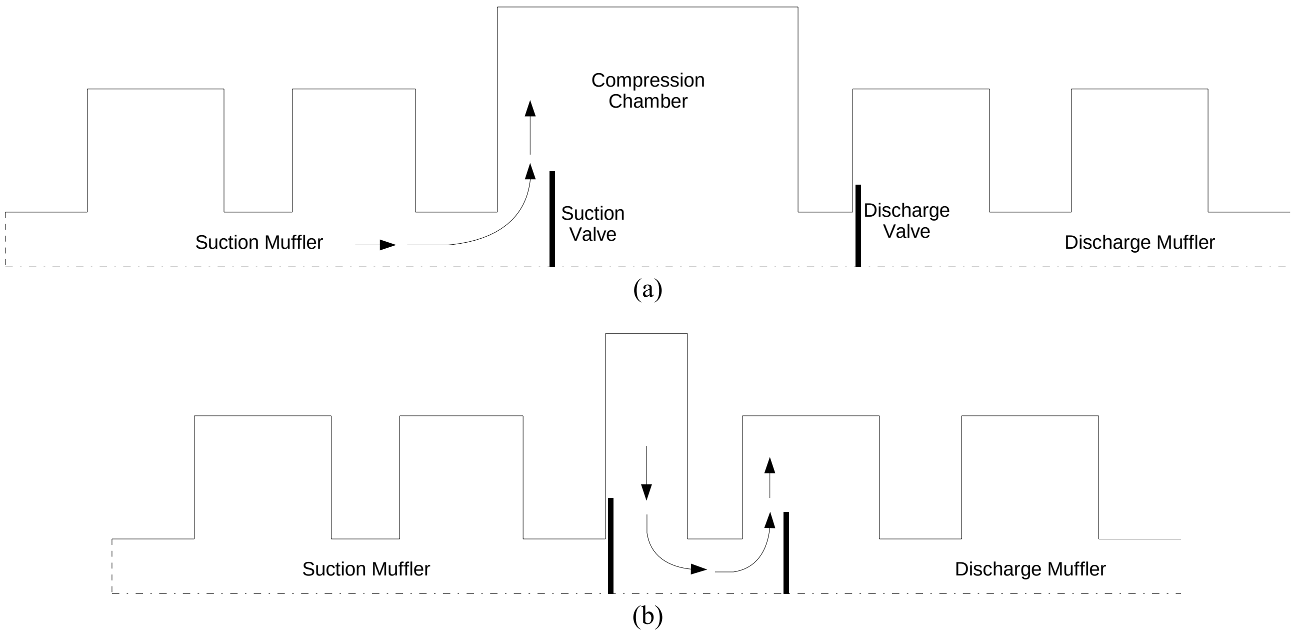


Figure 1. Schematic representation of the solution domain: (a) piston near BDC; (b) piston near TDC.

Evaporation and condensation pressures were imposed as boundary conditions for the suction and discharge mufflers, respectively. The temperature of the compression chamber walls was prescribed, but all remaining solid walls were assumed to be adiabatic. This major simplification was needed in this study because no data were available for the external heat transfer condition affecting the flow through mufflers. For this reason, the temperature at the inlet of the suction muffler was set to the value measured in the suction chamber, so as to guarantee the correct temperature for the gas entering the cylinder. At the solid walls all velocity components were set to zero, but for the reed and piston surfaces the velocities were obtained from Eq. (1) and the crankshaft mechanism, respectively. A turbulence intensity of 3% and turbulence length scale corresponding to the tube hydraulic diameter were adopted as the inlet boundary condition for the turbulent flow in the mufflers. The differential equation for the valve dynamics, Eq. (1), was solved using an explicit Euler method and by considering the force F_p to be constant during each time step. A stiction force, due to the presence of lubricating oil in the valve system, was assumed to act on the reed valves up to a valve lift of 10 μm. The values prescribed were 0.5 N for the suction valve and 1.5 N for the discharge valve.

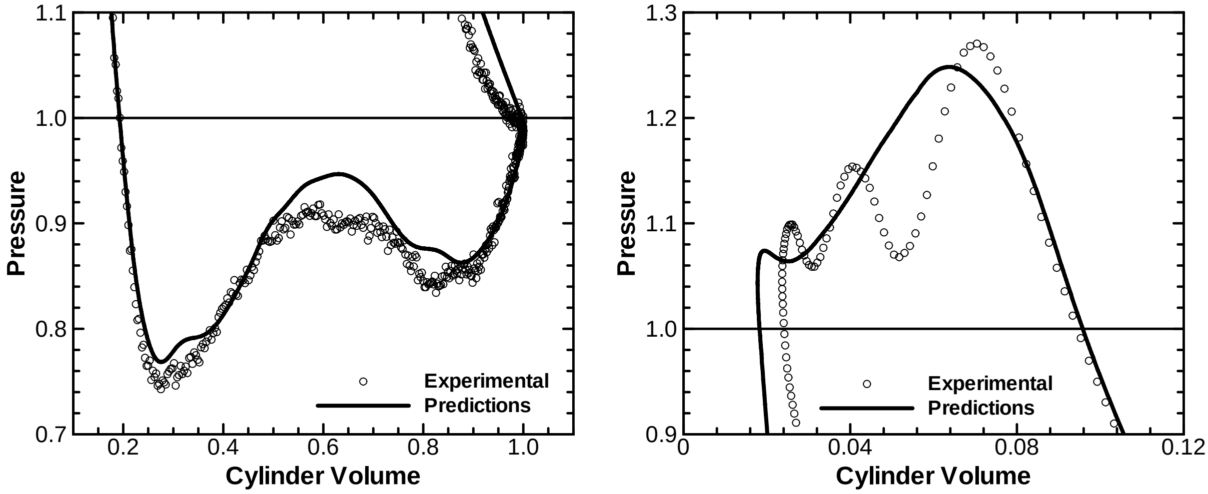
The solution starts with piston in BDC position, and with both valves closed. The evaporation pressure and suction temperature were employed as initial estimates for the suction muffler and for the compression chamber. Whereas, for the discharge muffler, the condensation pressure and discharge line temperature were used. The convergence of the procedure is assessed by examining whether the compressor operating conditions are cyclically repeated. By using such initial estimates, time steps and grid refinement, 3 cycles were required to establish a periodic condition, with a processing time of approximately 2 hours per cycle on a PC with a single Pentium IV 3 GHz processor.

4. RESULTS

The first step in the analysis was to compare the model predictions with experimental data obtained using a calorimeter facility. Besides crankshaft position, pressure measurements were acquired in the suction, discharge and compression chambers. A pressure piezoelectric transducer was selected due to its high response frequency, small size and reliability regarding the hostile conditions inside the compressor. A sensing winding was joined to the crankcase to collect the signal emitted by a magnet fixed to the crankshaft. The instantaneous crankshaft position was then calculated taking into account the compressor mechanism characteristics. Measurements were carried out for one 60 Hz compressor operating with R134a and with a nominal cooling capacity of 250 W.

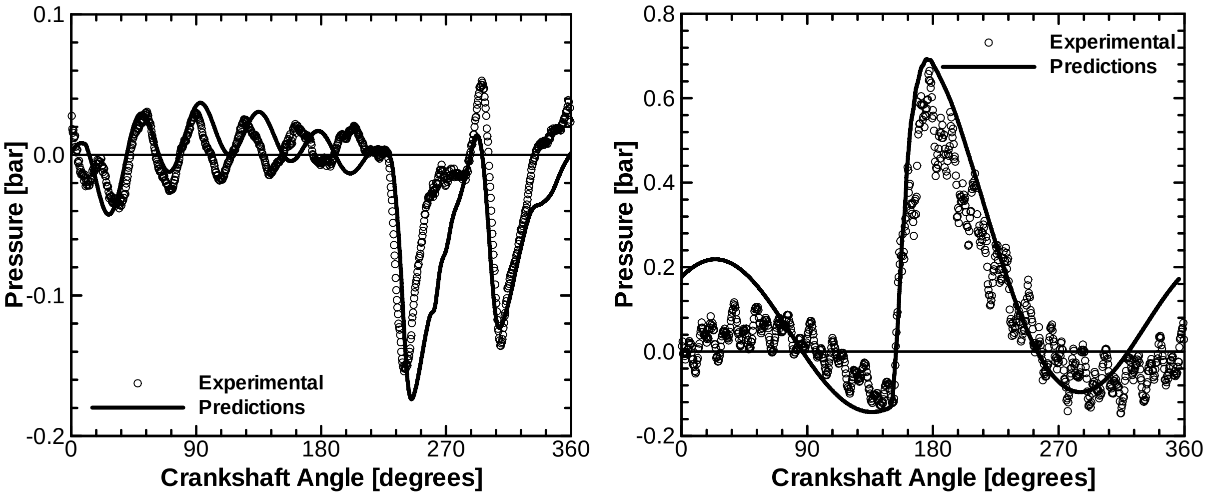
Figure 2 shows a comparison between numerical and experimental results of the compressor indicator diagram, corresponding to the suction and discharge processes, for a system condition represented by the condensation temperature T_c = 54.4 ºC. The pressure value was normalized by the pressure condition in the suction line or discharge line, depending on the process being examined. On the other hand, the instantaneous compression chamber volume was normalized by the compressor total volume displacement. The agreement between predictions and experimental data is very good in the case of the suction process. For the gas discharge, the prediction is also consistent, although some discrepancy is observed after the pressure reaches its maximum value. One possible reason for the pressure pulsation, observed experimentally but not predicted with the numerical model, can be associated with the manner the pressure transducer is installed in the cylinder. Accordingly, the experimental results represent the pressure at the cylinder wall in a position near the valve plate. The numerical results represent a mean value for the pressure on the piston surface. The first pressure peak that is observed in the discharge process is a consequence of the valve dynamics, since the cylinder pressure keeps rising until the valve lift is great enough to allow a larger mass flow rate. The other pressure peaks are associated to combined effects of valve dynamics, discharge chamber filling and increase in the flow restriction along the cylinder clearance, which occurs as the piston approaches the top dead center. During the suction process the effect of the transducer position is very small due to the large distance between the piston and the valve plate.

Differential pressure pulsations in the suction and discharge chambers are depicted in Fig. 3 for T_c = 54.4 ºC. The agreement between numerical and experimental data is again satisfactory, despite some phase and amplitude differences during the time the valves are closed. The pressure pulsation in the suction chamber directly affects the valve dynamics and, therefore, its correct prediction is of primary importance for the compressor simulation.



(a) (b)

Figure 2. Indicator diagram for T_c = 54.4 ºC: (a) suction process; (b) discharge process.



(a) (b)

Figure 3. Pressure pulsation: (a) suction chamber; (b) discharge chamber.

A further part of the validation procedure was to verify the capability of the model to predict variations in the compressor performance due to modifications in its design. In this respect, Tab. 1 shows results for power consumption when the diameter of the discharge valve orifice was increased by 30%. The experimental data indicate a smaller power consumption for the larger orifice diameter and no significant impact on the suction system. It should be noted that, although overestimating the compressor power consumption, the numerical model is able to predict the percentage of power reduction in close agreement with the measurements. Table 2 shows the compressor power consumption variation caused by a reduction in the condensation temperature from 54.4ºC to 40.5ºC. Again in this situation the numerical model was seen to correctly predict the decrease in the compression power indicated by the experimental data.

Based on the comparisons between predictions and experimental data, it can be said that despite some discrepancies verified in the indicator diagram, there is clear evidence that the numerical model can successfully predict variations in the compressor efficiency originated by modifications in its design and operating conditions.

Table 1. Power consumption [W] according to orifice diameter (T_c = 40.5ºC).

	Compression		Suction		Discharge	
	Experimental	Numerical	Experimental	Numerical	Experimental	Numerical
Small diameter	103.6	113.2	5.6	5.0	7.7	9.8
Large diameter	100.2	108.4	5.4	5.0	5.2	5.5
Variation %	-3.3	-4.2	-3.6	0.0	-32.5	-43.9

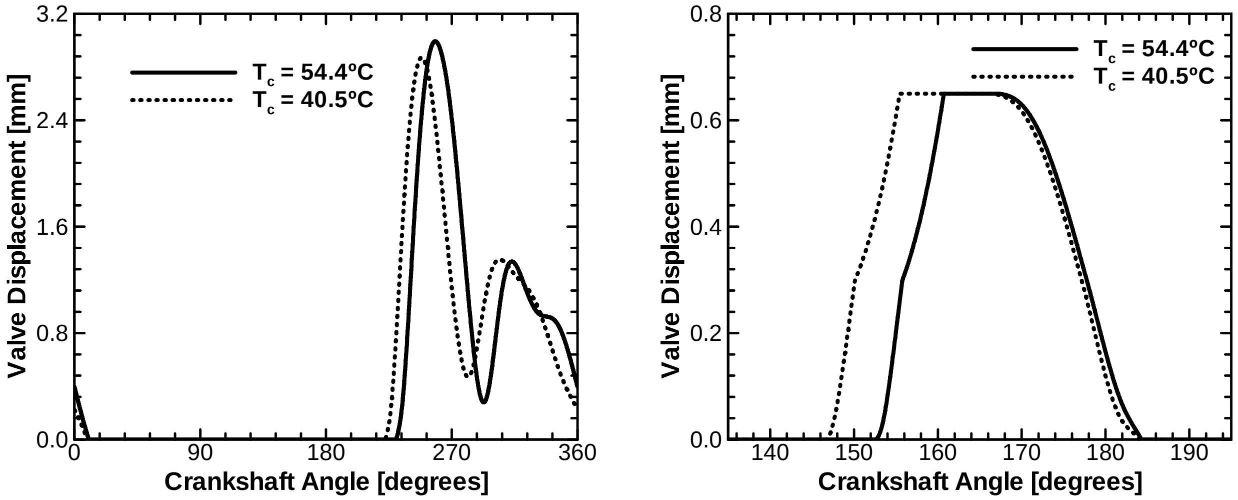
Table 2. Power consumption [W] for different condensation temperatures, considering the smaller discharge orifice.

	Compression		Suction		Discharge	
	Experimental	Numerical	Experimental	Numerical	Experimental	Numerical
T_c = 54.4ºC	115.6	122.8	5.5	4.8	6.0	7.1
T_c = 40.5ºC	103.6	113.2	5.6	5.0	7.7	9.8
Variation %	-10.4	-7.8	2.0	4.2	28.1	38.0

Figure 4 presents the displacements predicted for the suction and discharge valves, considering the two operating conditions aforementioned. The main difference originated for the motion of both valves is that of the opening point. The change in the curve inclination observed during the opening of the discharge valve (Fig 4b) is a consequence of the contact between the reed and the booster, which increases the valve stiffness. In order to improve the valve reliability, the opening of the discharge valve is limited by a stopper. The positions for the booster and stopper have to be properly chosen to prevent backflow through the valve, without increasing the compressor power consumption. Results for the suction valve displacement are important and can be used, for instance, to identify any contact between the reed and the piston. The numerical model can also be used to estimate the impact velocity of the reed valve against its seat, so that impact stress can be evaluated.

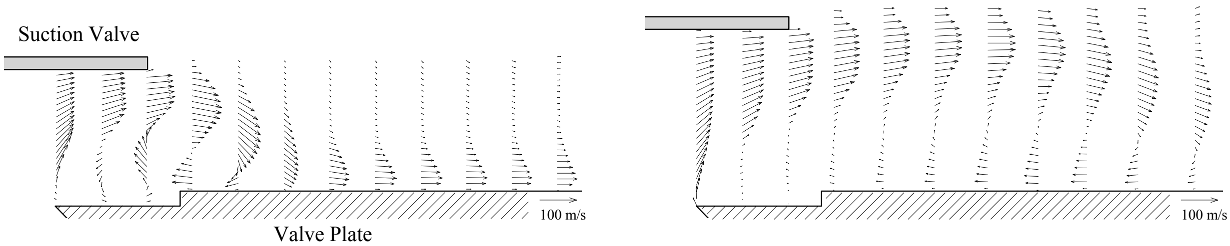
Figure 5 displays velocity vectors in the suction and discharge valves at intermediate and maximum valve lifts. The large separating flow regions verified in all situations are undesirable because they act to reduce the flow passage area and, hence, the compressor volumetric efficiency. Immediately after the valve opens, there is a high pressure difference between the compression chamber and the suction (or discharge) chamber that may produce even a sonic flow condition in the valve. In general, velocity levels are higher in the discharge valve because the pressure difference is greater there.

Figure 6 shows the local pressure difference in relation to the discharge pressure along the cylinder clearance, as the piston approaches the top center (TC) crank position. As can be observed, the pressure drop along the clearance is very high and is associated with viscous friction and flow acceleration. As the gas flows along the clearance towards the discharge orifice, it is accelerated due to a reduction in the flow passage area. The pressure decrease resulting from this flow acceleration causes the gas density to decrease, increasing even more the flow velocity. At position (a) the piston is near to the TC position and Fig. 7 shows that the valve is already open. As the valve continues to open, at position (b) the piston becomes closer to the TC position. The pressure increase in the clearance region verified between positions (a) and (b) is caused by an increase in the flow restriction given by the proximity of the piston in relation to the cylinder head and also due to the small valve lift. As the valve continues to open, the cylinder pressure level eventually decreases.

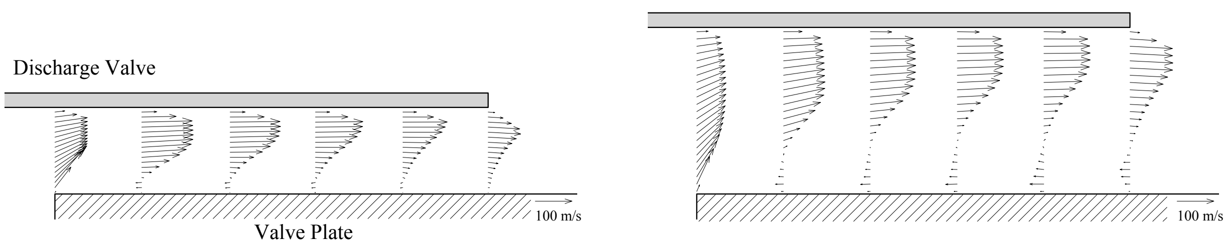


(a) (b)

Figure 4. Valve displacements: (a) suction valve; (b) discharge valve.



(a)



(b)

Figure 5. Velocity vector field: (a) suction valve; (b) discharge valve.

The numerical model was also employed to analyze the influence of heat transfer at the cylinder wall on the compressor efficiency, based on two thermal conditions: i) adiabatic walls and ii) prescribed temperature of 94ºC at the cylinder wall. Figure 8 shows the gas temperature and pressure inside the cylinder along the compression cycle. The temperature is a volume averaged value, whereas the pressure is the mean value on the piston surface, normalized by the evaporating pressure. By comparing with the adiabatic condition, it can be seen that the heat transfer at the cylinder walls, especially during the suction process, makes the gas temperature to increase approximately 13ºC. Despite such difference in the temperature levels, the indicator diagrams are virtually the same in both situations

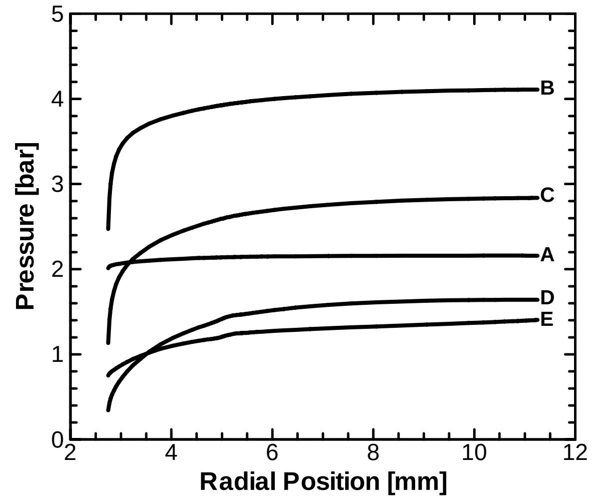


Figure 6. Pressure drop along the cylinder clearance.

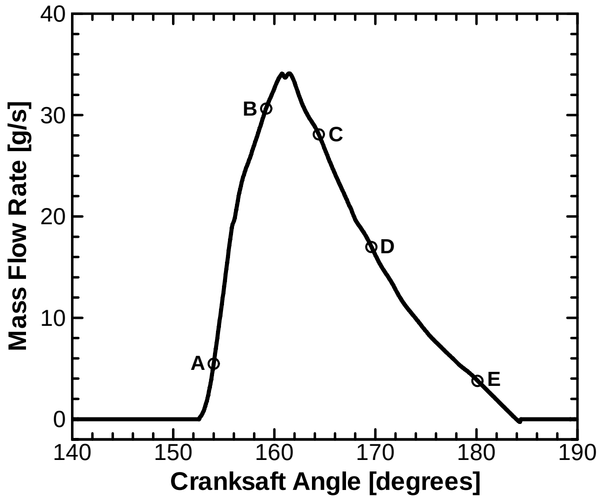


Figure 7. Mass flow rate through discharge valve.

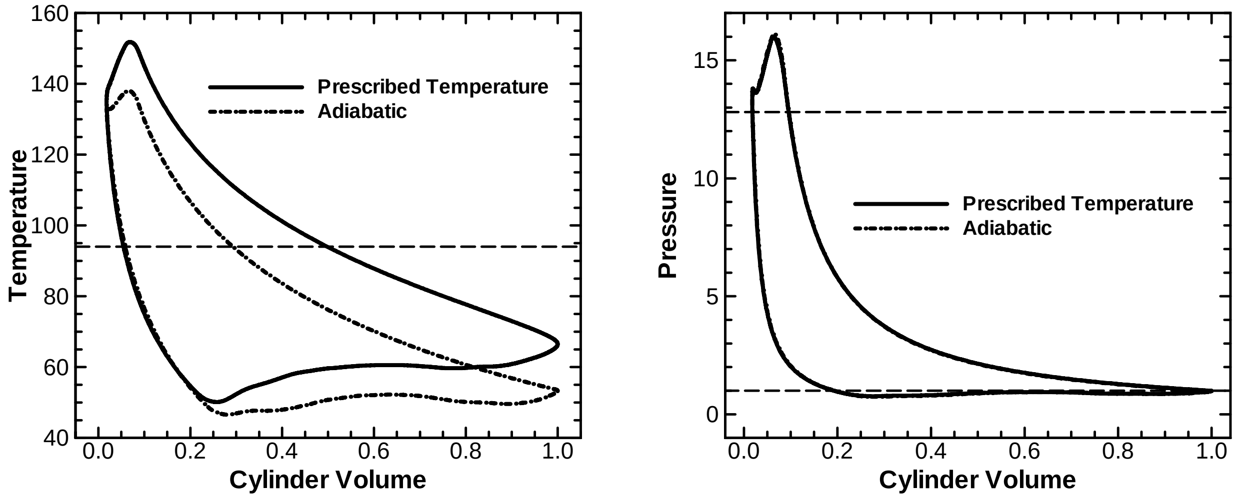


Figure 8. Results for in-cylinder gas properties according to different thermal boundary conditions.

As one would expect, Tab. 3 shows that the cooling capacity is directly proportional to the volumetric efficiency. If heat is not transferred to the gas during the suction process, the cooling capacity can be increased about 5%. Since the power consumption was practically the same in both situations, the thermodynamical coefficient of performance, COPpV, was increased proportionally to the cooling capacity.

Table 3. Influence of thermal boundary conditions on the compressor performance.

Boundary Condition	$\dot{Q}_e$ [W]	$\dot{W}$ [W]	η_v	COPpV
Prescribed Wall Temp.	256.7	122.8	0.76	2,09
Adiabatic Wall	269.4	122.6	0.80	2,20
Variation %	4.9	-0.2	5.0	5.2

5. CONCLUSIONS

This paper presented a two-dimensional CFD model developed to simulate reciprocating compressors. The solution domain included suction and discharge mufflers, valves and the compression chamber. The numerical results were validated with reference to experimental data of pressure in the suction, discharge and compression chambers. The experimental data demonstrated that the numerical model is capable to predict variations in the compressor efficiency originated by changes in its design or operating conditions. Moreover, the model was shown to provide detailed information about the flow through valves, valve dynamics and pressure pulsation in the suction and discharge systems, which are required for a complete analysis of the compressor.

6. ACKNOWLEDGEMENTS

This study forms part of a joint technical-scientific program of the Federal University of Santa Catarina and EMBRACO. Support from FINEP (Federal Agency of Research and Projects Financing) and CAPES (Coordination for the Improvement of High Level Personnel) is also acknowledged.

7. REFERENCES

Fagotti, F., Possamai, F. C., 2000, "Using Computational Fluid Dynamics as a Compressor Design Tool", Proc. Int.Compressor Engrg. Conf. at Purdue, pp. 137-144.

Fluent Inc., 2006, "FLUENT User's Guide", v. 6.2.16, Lebanon/NH, USA.

Lopes, M. N., Prata, A. T., 1997, "Comportamento Dinâmico de Válvulas Tipo Palheta em Escoamentos Periódicos", Cob 1138, Anais Do Xiv Congresso Brasileiro De Engenharia Mecânica (Cd-Rom), Bauru, Sp.

Matos, F.F.S., Prata, A.T. and Deschamps, C.J., 2002, "Numerical Simulation of the Dynamics of Reed Type Valves", Proc. 2002 International Compressor Engineering Conference at Purdue (CD-ROM), West Lafayette, USA, 8 p.

Matos, F.F.S., Prata, A.T. and Deschamps, C.J., 2006, "A Two-Dimensional Simulation Model for Reciprocating Compressors with Automatic Valves", Proc. Int. Comp. Engineering Conference at Purdue (CD-ROM), Paper C053.

Orzag, S. A., Yakhot, V., Flannery, W. S., Boysan, F., Choudhury, D., Marusewski, J., Patel, B., 1993, "Renormalization Group Modeling and Turbulence Simulations", Near-Wall Turbulent Flows, Elsevier Science Publisher.

Ottitsch, F., 2000, "CFD: A Viable Engineering Tool for Compressor Valve Design or Just a Toy?", Proc. Int. Compressor Engrg. Conf. at Purdue, pp. 423-428.

Pereira, E. L. L., Deschamps, C. J., Lilie, D. E. B., Schreiner, J. E., 2006, "Análise de um Sistema Auxiliar para a Abertura de Válvulas", Proceedings of the 11th Brazilian Congress of Thermal Sciences and Engineering, Paper CIT06-0859, Curitiba, Brazil.

Salinas-Casanova, D.A., Deschamps, C.J. and Prata, A.T., 1999, "Turbulent Flow through a Valve with Inclined Reeds", Proc. International Conference on Compressors and Their Systems, Londres, p. 443-452.

Ussyk, M. S.,1984, "Simulação Numérica do Desempenho de Compressores Herméticos Alternativos", Dissertação de Mestrado, Departamento de Engenharia Mecânica, Universidade Federal de Santa Catarina, Florianópolis.

8. RESPONSIBILITY NOTICE