COMPARISON OF SIMULATION METHODOLOGIES FOR SUCTION AND DISCHARGE SYSTEMS

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

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

Fabrício C. Possamai
Empresa Brasileira de Compressores S.A.
Caixa Postal 91
89219-901, Joinville, SC

Abstract. *Suction and discharge systems have an important role in the efficiency of reciprocating machines. Several authors have dedicated attention to this subject, proposing different simulation methodologies to solve the pulsating flow that prevails in such systems. In this work, two numerical approaches are applied to predict the unsteady gas flow through the suction muffler of a reciprocating refrigeration compressor: a linear acoustic method and a finite volume model. The acoustic model considers mufflers as Helmholtz resonators and requires empirical loss coefficients to bring about pressure oscillation damping. In the finite volume methodology, the governing equations are integrated over control volumes placed along the solution domain, resulting in a system of algebraic equations that is then solved. Both models are implemented into a computaional code written to simulate the dynamic behavior of refrigeration compressors. Numerical results for pressure pulsation in the suction chamber, compressor capacity and compressor consuption are compared with experimental data to assess each model capability. Although the present paper is concerned with suction systems of refrigeration compressors, the analysis can be extended to other applications, such as IC engines.*

Keywords: *Suction systems, discharge systems, valve systems*

1. Introduction

The efficiency of reciprocating machines, such as IC engines and compressors, are affected to a great extent by the suction and discharge systems. For instance, in the case of IC engines, combustion is limited by the amount of air available and, for this reason, an optimum suction system should supply the cylinder with as much air as possible. Moreover, the suitable mixing between fuel and air necessary for combustion is also affected by the flow field developed by the port/valve assembly. On the other hand, such systems play also a crucial role on the performance of compressors, affecting directly the volumetric and energy losses, as well as the compressor noise level.

Most of the development verified in suction and discharge systems has been provided by highly skillful and experienced designers with help of some knowledge acquired along the years from experimental results. Early measurements were restricted to pressure distribution but provision of local velocities started being available in the 1970's through new techniques such as hot-wire anemometry (HWA) and laser-doppler velocimetry (LDV). Additionally, an understanding of the overall characteristic of the flow pattern can be attained via particle image velocimetry (PIV), in which tiny particles are seeded in the working fluid and then photographed in motion. Due to the several parameters affecting the flow, a systematic experimental investigation of the phenomenon is difficult and expensive.

An alternative commonly adopted is the theoretical analysis of the flow by means of the numerical solution of the governing equations. In this respect, the one-dimensional analysis of the unsteady gas dynamics equations has been used to describe the flow in intake and exhaust systems, allowing a better understanding of the problem and a reduction in the time required to develop a new engine. With this purpose, several different methods of solution for the governing equations have been developed, such as acoustic models, method of characteristics, finite difference schemes and finite volume methodology.

This paper is concerned with the modeling of one-dimensional compressible flow in suction mufflers of refrigeration hermetic reciprocating compressors, represented schematically in Figure 1a. Reed valves are essential parts in such compressors. These valves automatically open and close depending on the pressure difference between the cylinder and the suction/discharge chamber, established by the piston motion. Once the valves are open, the pressure flow field is responsible for the resultant force acting on the reed. For this reason, it is crucial to predict the phenomena associated with the flow through the valve.

Figure 1b shows the indicator diagram for a typical compressor cycle. When the piston moves downwards, it reaches a position where low pressure gas is drawn in through the suction valve, which is opened by the pressure difference between the cylinder and the suction chamber. The gas keeps flowing in during the suction stroke as the piston moves towards the bottom dead center (BDC), filling the cylinder volume with gas at suction pressure, p_s. After reaching the BDC, the piston starts to move in the opposite direction, the suction valve is closed, the gas is trapped, and its pressure rises as the cylinder volume decreases. Eventually, the pressure reaches the pressure in the discharge chamber, p_d, and the discharge valve is forced to open. After the opening of the discharge valve, the piston keeps moving towards the top dead center (TDC), represented by point A. It should be noted that suction and discharge processes do not take place at constant pressure. This phenomenon is associated with the valve dynamics and the restriction imposed by the valve orifices and suction and discharge systems. At point A there remains an amount of refrigerant in the cylinder at discharge pressure, since a gap between the piston and cylinder head is necessary to accommodate the valves. Therefore, as the piston moves downwards, the refrigerant in the clearance is re-expanded and its pressure decreases, as represented by process AB. When the piston reaches point B, the cylinder pressure is lower than the suction chamber pressure, p_s, causing the suction valve to open and the gas to flow into the cylinder up to approximately point C.

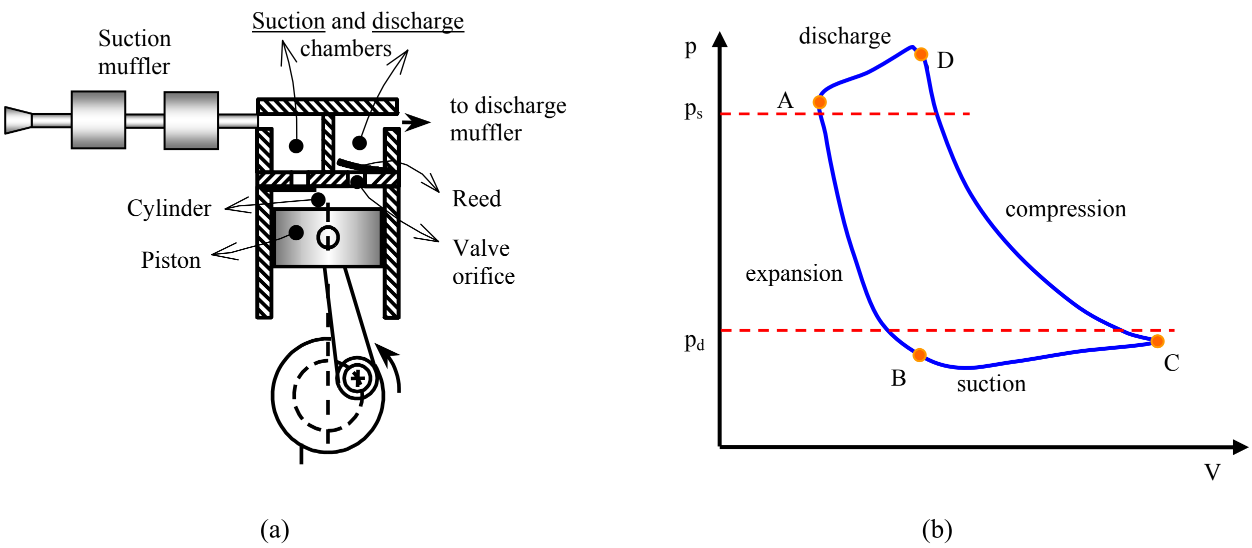


Figure 1. Schematic of a reciprocating compressor and its indicator diagram.

The fact that hermetic compressors adopt automatic valves means that their dynamics are affected by pressure pulsations in the muffler and vice versa, requiring both problems to be solved simultaneously. If pulsations are small compared to the mean pressure, its behavior can be approximated by the acoustic theory (Elson and Soedel, 1974). In the case of large pressure amplitudes, the non-linear partial differential equations governing the unsteady one-dimensional compressible flow have to be solved (McLaren et al, 1975). More recently, Liu and Soedel (1994) presented a numerical model to solve a set of non-linear differential equations describing the unsteady one-dimensional compressible flow in an intake system, considering wall friction and heat transfer. Pérez-Segarra et al. (1994) developed an unsteady one-dimensional model for the whole compressor using a finite volume technique and solved the fluid dynamics and heat transfer in a very detailed manner. Ignatiev et al. (1996) presented a model for the suction system and applied it to analyze the effect of geometric parameters on the compressor performance. In the work of Bassi et al. (2000) an unsteady one dimensional model is proposed for the flow in the muffler and solved through a discontinuous Galerkin method. Their methodology, and that proposed by Pérez-Segarra et al. (1994), have shown promising results.

In this work, the pulsating flow in mufflers has been predicted according to two different methodologies: i) a Resonant Helmholtz model and ii) a one dimensional fluid dynamics model. Since the main interest is the suction system, the simulation of other processes in the compressor (valve dynamics, cylinder compression, discharge system) has been carried out with a global simulation program, which will be detailed shortly. For a comparative analysis of both methodologies, experimental data for pressure in the suction muffler, compressor capacity and compressor consumption were obtained on a calorimeter under controlled conditions. The performance of each model was assessed by comparing their predictions with experimental data.

2. Mathematical Model

A compressor simulation code originally developed by Ussyk (1984) was adopted to implement the acoustic and finite volume models and to evaluate the main parameters concerning the compressor operation. The code accounts for piston displacement as a function of crankshaft angle, the thermodynamic process inside the cylinder, mass flow rate through the valves, valve dynamics, gas pulsation inside the mufflers and refrigerant thermodynamic properties. Several parameters are calculated during the compressor cycle, such as instantaneous pressure throughout the compressor, mass flow rate, valve dynamics, energy and mass losses, refrigerating capacity, etc.

The transient equations associated with the compressor simulation code are solved via a fourth order Runge-Kutta method. Thermodynamic properties can be evaluated using the perfect gas hypothesis or through a program link to the REFPROP database (Gallanger et al., 1993).

Reed valves are usually made of stainless steel and their dynamics can be expressed in a simplified way, using a one degree of freedom model as follows:

$$m\ddot{\delta}_1 + C\dot{\delta}_1 + K\delta_1 = F - F_0 \tag{1}$$

where F_0 is a pre-load force acting on the reed and F is the force resulting from the pressure distribution on the reed surface. The valve stiffness and damping coefficients, K and C, respectively, as well as the valve mass, m, are determined experimentally. Valve stiction can be included according to the analytical model proposed by Khalifa and Liu (1998).

The resultant force F acting on valve reeds and the mass flow rate through the valves are obtained with reference to effective force area A_{ef} and effective flow area A_{ee}, respectively. From the pressure difference across the valve, Δp_v, A_{ef} is determined from $A_{ef} = F/\Delta p_v$. The effective force area can be understood as a parameter related to how efficiently the pressure difference Δp_v opens the valve. On the other hand, for the same pressure drop, A_{ee} expresses the ratio between the actual mass flow rate through the valve and that given by an isentropic flow condition.

Bearings are modeled through short bearing theory and, optionally, a fixed value can be set for the bearing power losses. Finally the thermodynamic process for the gas inside the cylinder can be evaluated either by a polytropic model or by the first law of thermodynamics.

The compressor internal temperatures have to be supplied as input for the simulation program. This is accomplished by an interface with a second simulation code, which evaluates the temperature in eight control volumes through energy balances and using some of the compressor simulation program outputs. The control volumes considered are: gas in the suction muffler, cylinder walls, gas in the discharge muffler, discharge gas, internal environment, compressor housing, electric motor and bearings. Steady state condition is assumed for all temperatures with the exception of the in-cylinder gas. The control volume balance equations are simultaneously and iteratively solved since they depend on all compressor energy fluxes. More details on the compressor simulation program can be obtained in Fagotti et al. (1994).

Gas pulsation can be acoustically modeled considering mufflers as Helmholtz resonators or by solving a fluid dynamics model. For the latter, the conservation equations that govern the flow through a typical control volume in the muffler (Figure 2) are those related to mass, momentum and energy:

$$\frac{\partial m}{\partial t} + \dot{m}_o - \dot{m}_i = 0 \tag{2}$$

$$\frac{\partial m\overline{V}}{\partial t} + \left[\dot{m}\overline{V}\right]_o - \left[\dot{m}\overline{V}\right]_i = (p_i - p_o)A_s - \tau_w A_\ell \tag{3}$$

$$\frac{\partial m(h+\overline{V}^2/2)}{\partial t} + \left[\dot{m}(h+\overline{V}^2/2)\right]_o - \left[\dot{m}(h+\overline{V}^2/2)\right]_i - \forall\frac{\partial p}{\partial t} = \dot{Q} \tag{4}$$

In the equations above, $\forall$, A_s and A_ℓ are the volume, the cross section area and the lateral surface area, respectively, of the control volumes employed to discretize the muffler. On the other hand, m and $\dot{m}$ stands for mass and mass flow rate. Sub indices "o" and "i" denote quantities at outlet and inlet sections of each control volume. A state equation for the gas, $p = p(\rho, T)$, completes the system of equations required to solve the flow.

The effect of the wall on the flow is taken into account through estimates of friction force and heat transfer, implied by the no-slip and impermeable wall boundary condition. There is a question on whether flow friction factors can be applied to unsteady flow (Ribas Jr and Deschamps, 2003). When the transient flow is of relatively low frequency and amplitude, there is very little error involved in using a steady flow friction factor. However, if large and rapid disturbances occur, a significant error may be incurred. A similar argument can be extended to heat transfer correlations. In the present analysis the heat transfer, $\dot{Q}$, and the shear stress, τ_w, acting on the volume lateral surface area A_ℓ are evaluated from standard correlations developed for steady flow through pipes.

Pressure drop at geometric singularities, corresponding to sudden contraction or sudden enlargement at the muffler volumes, have been estimated with reference to the area ratio (Potter and Wiggert, 1991).

3. Numerical Methodology

The numerical solution of the governing equations for the flow in the muffler was performed using a finite volume methodology. For this practice the solution domain is divided into small control volumes (Figure 2) using a staggered grid arrangement. The governing differential equations are integrated over each control volume with the use of Gauss's theorem. The convection at the control volumes faces was approximated with the UPWIND interpolation scheme. A fully implicit time discretization scheme was applied to unsteady terms in the equations.

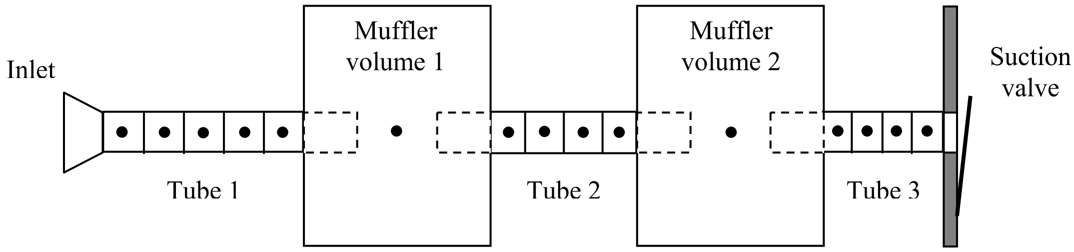


Figure 2. Suction muffler numerical model.

The system of algebraic equations were solved with the Tridiagonal Matrix Algorithm (TDMA), in a segregated approach. The coupling between pressure and velocity was handled through the SIMPLEC algorithm extended to flows of arbitrary Mach number (Van Doormal et al., 1987). Further details on the numerical methodology can be found in many references, such as Ferziger and Peric (1996).

The correct implementation of the code was verified by comparing its results with a number of benchmark solutions for different classes of compressible flow. Despite the non-linearity of the equations, no under relaxation factor was required in the iterative process. A very important aspect addressed was the validation of the numerical solution by means of sensitivity tests with respect to grid refinement and time step. The grid level used in all simulations had approximately 80 nodes equally distributed between the three tubes. Yet, only one node was placed in the volumes representing the compressor cylinder, muffler volumes and suction valve chamber.

The initial velocity field is set to zero everywhere inside the muffler, with pressure and temperature (therefore density) being taken as equal to the reference condition at the muffler inlet. The boundary condition at the suction valve is evaluated taking into account the pressure distribution in the muffler. The iterative procedure evaluates new flow properties for each time step until converge is reached, which is judged by examining whether the compressor operation conditions are cyclically repeated. A number of 6280 time steps was employed to resolve each of the 6 cycles required to establish such periodic condition.

4. Experimental setup

Experimental data were used to assess the capability of the acoustic and fluid dynamics models to predict the flow through the muffler. The quantities measured in the experimental setup are pressure in the muffler next to the suction valve, pressure in the cylinder and crankshaft position. The region at the entrance of the suction valve was chosen to place the pressure transducer because the highest gas pulsation is expected there. A piezoelectric transducer was selected for pressure measurements due to its high response frequency, small size and reliability regarding the hostile conditions inside the compressor. Small sensing windings are assembled in the valve plate seat to give the valve lift according to the crankshaft position. A second sensing winding is joined to the crankcase to collect the signal emitted by a magnet fixed to the crankshaft. The instantaneous crankshaft position is calculated taking into account the compressor mechanism characteristics.

A compressor was assembled with the transducers described above and had its performance measured under two conditions: i) ASHRAE LBP; ii) Evaporating and condensing temperatures (T_E and T_C) equal to –27 ^oC and 42 ^oC, respectively. Rotation speed of the compressor is assumed constant, with a frequency of 60 Hz, and the refrigerating fluid is R134a.

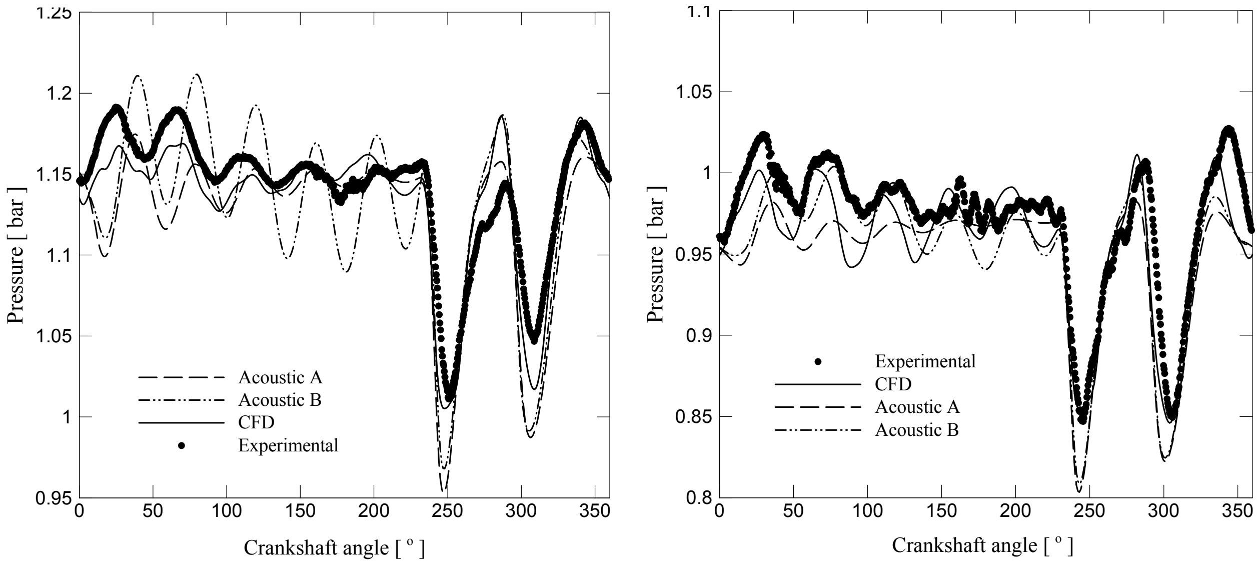
5. Results

Experimental data for the compressor performance were compared with numerical results provided by three models for the muffler: i) One dimensional computational fluid dynamics model (CFD); ii) Acoustic model with a variable loss factor (Acoustic A), depending on whether the valve is open or closed; iii) Acoustic model with a fixed loss factor (Acoustic B). The suction temperature was kept constant along the muffler for both acoustic models. Yet, in the case of the CFD model the temperature at the entrance of the suction valve can be evaluated since the energy equation is solved.

Figure 3 shows a comparison between experimental data and numerical results for pressure at the suction chamber as a function of the crankshaft angle, for two system conditions: a) ASHRAE LBP and b) T_E = -27 °C; T_C = 42 °C. As can be noticed, when compared to both acoustic model versions, the CFD model returns the best overall agreement with the experimental data. Pressure pulsation in the suction chamber predicted by the CFD model are approximately in phase with experimental data when the suction valve is closed, a feature not captured by the acoustic models. This is a particularly important aspect in the valve system efficiency, since the force on the reed and the flow through the valve orifice are dependent on the pressure difference between the suction chamber and the cylinder. For this reason, the pressure pulsation in the suction chamber has to be tuned to the crankshaft angle, by a proper design of the muffler, so as to maximize the compressor volumetric efficiency. The main shortcoming of the acoustic models is the necessity to adjust the loss factor for each different compressor model or system condition. Naturally, this is not an adequate basis to develop a compressor simulation model.

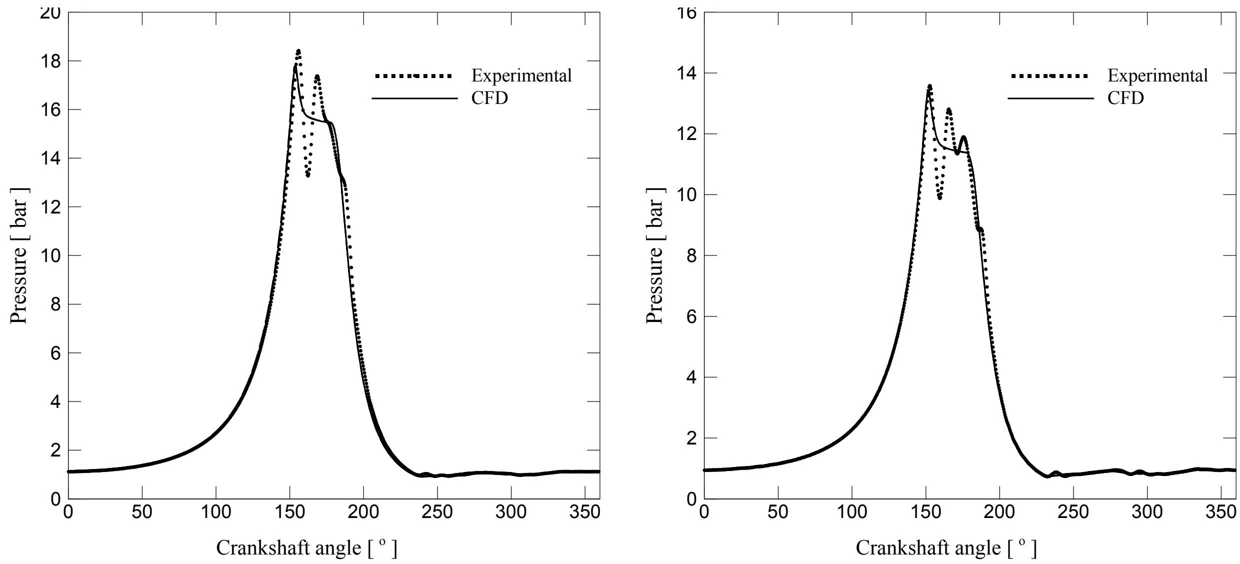
Experimental data and CFD predictions for the in-cylinder pressure as a function of crankshaft angle are shown in Figure 4. The most significant difference between both results occurs at the top dead center, during the opening of the discharge valve. This weakness of the physical modeling can be circumvented by the adoption of a differential formulation to solve the in-cylinder flow around the piston top dead center.

Tables 1 and 2 show measurements and predictions for the compressor performance. From the ASHRAE LBP condition presented in Table 1, one can see that the compressor simulation combined with CFD model offers the best estimate for the compressor efficiency. As for the condition represented by T_E = -27 °C and T_C = 42 °C, the three models predict similar results, although the CFD model returns a value of consumption that is in closer agreement with the experimental data.



(a) ASHRAE LBP (b) T_E = -27 °C; T_C = 42 °C

Figure 3. Pressure at the suction valve entrance.



a) ASHRAE LBP

b) T_E = -27 °C; T_C = 42 °C

Figure 4. Pressure in the compressor cylinder.

Table 1. Results for compressor performance: ASHRAE LBP condition.

	Capacity (W)	Consumption (W)
Experimental data	227.8	136.1
CFD model	230.7	137.1
Acoustic model A – variable loss factor	235.7	140.1
Acoustic model B – fixed loss factor	234.8	139.3

Table 2. Results for compressor performance: TE = -27 °C; TC = 42 °C.

	Capacity (W)	Consumption (W)
Experimental data	208.2	117.2
CFD model	203.5	117.7
Acoustic model A – variable loss factor	205.3	118.9
Acoustic model B – fixed loss factor	205.3	118.7

6. Conclusions

The present work considered an analysis between different approaches for modeling the pulsating flow in suction systems, with reference to refrigeration compressors. One of these methodologies is represented by an analytical acoustic model, which requires empirical coefficients to estimate friction losses. The other model is a one dimensional computational fluid dynamics model based on the finite volume methodology, which considers compressibility and thermal effects. Both models are applied to predict head losses of a suction muffler in a dynamic simulation of the whole compressor. The main weakness of the acoustic model is the necessity to adjust loss factors according to the compressor operating condition, or depending on whether the valve is closed or not. This limitation is not present in the CFD model since friction losses may be directly evaluated according to the flow condition. Furthermore, the CFD model includes the energy equation and can predict the temperature distribution in the muffler, an important parameter in the compressor efficiency. Based on comparisons between experimental data and numerical results, it is clear the benefit brought by the CFD model to the prediction of the gas dynamics in suction and discharge systems.

7. Acknowledgements

This work is part of a technical-scientific program between Federal University of Santa Catarina and EMBRACO. Support from the Brazilian Research Council, CNPq, is also appreciated.

8. References

Bassi, F., Pelagalli, L., Rebay, S., Betto, A., Orefice, M., Pinto, A., "Numerical Simulation of a Reciprocating Compressor for Household Refrigerators", Proc. Compressor Engineering Conference at Purdue, West Lafayette, Indiana, USA, p. 97-104, 2000.

Elson, J.P. and Soedel, W., "Simulation of the Interaction of Compressor Valves with Acoustic Back Pressures in Long Discharge Lines", Journal of Sound and Vibration, v.34, n.2, pp. 211-220, 1974.

Fagotti, F., Todescat, M.L., Ferreira, R.T.S., Prata, A.T., "Heat Transfer Modeling in a Reciprocating Compressor", Proc. Compressor Engineering Conference at Purdue, West Lafayette, Indiana, USA, p. 605-610, 1994.

Ferziger, J.H. and Peric, M., "Computational Methods for Fluid Dynamics", Springer-Verlag, 1996.

Gallanger, J., McLinden, M., Morrison, G. and Huber, M., "NIST Thermodynamic Properties of Refrigerants and Refrigerants Mixtures Database (REFPROP)", Thermophysics Division, National Institute of Standards and Technology, US Department of Commerce, 1993.

Ignatiev, K.M., Chrustalev, B.B., Perevozchikov, M.M. and Zdalinsky, V.B., "Simulation and Parametric Analysis of the Suction Valve and Muffler of a Small Reciprocating Compressor", Proc. Int. Compressor Engineering Conference at Purdue, West Lafayette, USA, p. 205-209, 1996.

Khalifa, H.E. and Liu, X., "Analysis of Stiction Effect on the Dynamics of Compressor Suction Valve", Proc. Int. Compressor Engineering Conference at Purdue, West Lafayette, USA, pp. 87-92, 1998.

Liu, Z. and Soedel, W., "Using a Gas Dynamic Model to Predict the Supercharging Phenomenon in a Variable Speed Compressor", Proc. Int. Compressor Engineering Conference at Purdue, West Lafayette, USA, pp. 491-497, 1994.

Maclaren, J.F.T., Tramschek, A.B., Sanjines, A. and Pastrana, O.F. "A Comparison of Numerical Solutions of the Unsteady Flow Equations Applied to reciprocating Compressor Systems", Journal of Mechanical Engineering Science, v. 17, n. 5, pp. 271-279, 1975.

Pérez-Segarra, C.D., Escanes, F. and Oliva, A., "Numerical Study of the Thermal and Fluid-Dynamic Behaviour of Reciprocating Compressors", Proc. Int. Compressor Engineering Conference at Purdue, West Lafayette, USA, pp. 145-150, 1994.

Potter, M.C. and Wiggert, D.C., "Mechanics of Fluids", Prentice-Hall, 1991.

Ribas Jr, F.A., Deschamps, C.J., "Computational Simulation of Pulsating Turbulent Flow in Pipes", Proc. 17th Int. Congress of Mechanical Engineering, São Paulo, SP, Paper 0409 (CD-ROM), 7 p., 2003.

Ussyk, M.S., "Numerical Simulation of Hermetic Reciprocating Compressor Performance", (in Portuguese), M.Sc. Dissertation, Federal University of Santa Catarina, 1984.

Van Doormal, J.P., Raithby, G.D., McDonald, B.H., "The Segregated Approach to Predicting Viscous Compressible Flow", ASME J. Turbomachinery, v. 109, 268-277, 1987.

9. Responsibility notice

The authors are the only responsible for the printed material included in this paper.