2002

Numerical Simulation Of Pulsating Flow In Suction Mufflers

C. J. Deschamps
Federal University of Santa Catarina

F. C. Possamai
EMBRACO

E. L. L. Pereira
Federal University of Santa Catarina

NUMERICAL SIMULATION OF PULSATING FLOW IN SUCTION MUFFLERS

C.J. Deschamps[a], F.C. Possamai[b] and E.L.L. Pereira[a]

[a] Federal University of Santa Catarina, Florianópolis, SC, 88040-900, Brazil
[b] EMBRACO, Caixa Postal 91, Joinville, SC, 89219-901, Brazil

ABSTRACT

Fluid flow through acoustic mufflers is a complex phenomenon that has been investigated for many years due to its importance in compressor efficiency. A one dimensional computational fluid dynamics model based on the finite volume methodology is developed to solve this flow, considering compressibility and thermal effects. Head losses of a suction muffler are evaluated with the model taking into account the pulsating flow condition in a dynamic simulation of the whole compressor. The results are compared with those given by an analytical acoustic model and with experimental data so as to assess the model capability. The acoustic model uses an equation to estimate the friction losses but requires empirical loss coefficients that vary according to the compressor working conditions. On the other hand, the computational fluid dynamics model estimates directly the friction losses from its transport equations and therefore requires no such adjustments. Another benefit provided by the fluid dynamics model is the evaluation of gas temperature throughout the muffler, which is considered to be constant in the acoustic model.

NOMENCLATURE

A_s	flow cross section area	T	temperature
A_ℓ	volume lateral area	$\overline{V}$	mean velocity at the flow cross section area
h	enthalpy	$\forall$	volume
$\dot{m}$	mass flow rate	$\dot{Q}$	heat transfer
p	pressure	ρ	density
m	mass	τ_w	wall shear stress

INTRODUCTION

The intake system of a compressor plays an important role on the performance of reed type valves of small hermetic refrigeration compressors, specially suction valves designed without an opening limiter. The valve affects directly volumetric and energy losses and may have some influence on the compressor noise level as well. Therefore, mathematical models aimed to simulate a compressor thermodynamic performance should have a proper description of gas pulsations in the suction system. Finally, because the dynamics of reed type valves is affected by pressure fluctuations in the muffler and conversely, both problems must be solved simultaneously.

A number of methods have been proposed to analyze gas pulsations in intake and discharge systems. If pulsations are small compared to the mean pressure, its behavior can be approximated by the acoustic theory (see for instance 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. Examples of more complex analyses considering three dimensional muffler geometries are those presented by Choi et al. (2000) and Fagotti and Possamai (2000). Such approaches offer the most complete account of the flow through the muffler but because their high computational cost the interaction with other components of the compressor (such as reed and cylinder) is usually neglected.

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, flow in 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 muffler and for suction valve displacement were obtained for a compressor operating on a calorimeter under controlled conditions. The performance of each model was assessed by comparing their results with experimental data. The main goal is to verify whether the fluid dynamics model is a reliable tool to predict the compressor performance under different operating conditions.

MATHEMATICAL MODEL

A compressor simulation code originally developed by Ussyk (1984) is adopted to evaluate all the main parameters concerning the compressor operation. The code accounts for the piston displacement as a function of time, thermodynamic process inside the cylinder, fluid flow through the valves, valve dynamics, piston-cylinder clearance leakage, gas pulsation inside mufflers, motor momentum-power-efficiency relationship, bearing simulation, thermal simulation and refrigerant thermodynamic and thermophysical properties. Several parameters are calculated along the compressor cycle, such as instantaneous pressure throughout the compressor, mass flow rate, valve dynamics, energy and mass losses, refrigerating capacity, energy consumption, etc.

The differential equations associated to the compressor simulation code are solved via a fourth order Runge-Kutta method. Thermodynamic properties can be evaluated using four different equations: perfect gas, Martin-Hou, Carnahan-Starling-DeSantis and modified Bennedict-Webb-Rubin. The last two options are made available through a program linking to REFPROP database (Gallanger et al., 1993). Displacements of valves are modeled by modal superposition or by a single degree of freedom mass-spring model, whereas mass flow rate through them are obtained with reference to one-dimensional isentropic flow. In both cases effective flow and force areas have to be measured previously. Valve stiction is included according to the analytical model proposed by Khalifa and Liu (1998).

Gas pulsation is acoustically modeled considering mufflers as Helmholtz resonators. 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 politropic 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 obtained in Fagotti et al. (1994).

The main objective of the present work is to replace the acoustic model for the suction muffler by a fluid dynamics model and analyze the effect that this may have on the prediction of the compressor performance. For the fluid dynamics model, the conservation equations that govern the flow through a typical control volume in the muffler (Figure 1) are those related to mass, momentum and energy:

$$\frac{\partial m}{\partial t} + \dot{m}_o - \dot{m}_i = 0 \tag{1}$$

$$\frac{\partial\, m\overline{V}}{\partial t} + \left[\dot{m}\overline{V}\right]_o - \left[\dot{m}\overline{V}\right]_i = (p_i - p_o)A_s - \tau_w A_\ell \tag{2}$$

$$\frac{\partial\, m\,(h + \overline{V}^2/2)}{\partial t} + \left[\dot{m}\,(h + \overline{V}^2/2)\right]_o - \left[\dot{m}\,(h + \overline{V}^2/2)\right]_i - \forall\frac{\partial p}{\partial t} = \dot{Q} \tag{3}$$

All variables appearing in the above equations are listed in the nomenclature section, whereas sub indices “o” and “i” denote quantities at outlet and inlet sections of each control volume, respectively. The effect of the wall on the flow was taken into account through estimates of friction force and heat transfer, implied by the no-slip and impermeable wall boundary condition. The shear stress τ_w acting on the volume lateral area A_ℓ is evaluated from standard correlations for laminar and turbulent flow regimes. Pressure drop at geometric singularities (sudden contraction or sudden enlargement) have been estimated with reference to area ratio (Potter and Wiggert, 1991).

The internal heat transfer coefficient was evaluated also from standard correlations and then combined with thermal resistances of conduction at the wall and external convection to give an overall heat transfer coefficient U. A state equation for the gas, $p = p(\rho, T)$, completes the system of equations required to solve the flow.

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 in small control volumes (Figure 1), using a staggered grid scheme, and 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.

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 iteration 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 chamber, mufflers 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.

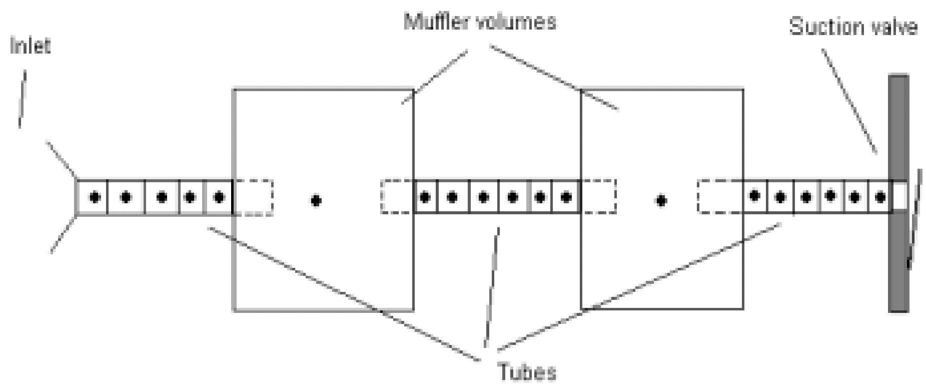


Figure 1: Suction muffler model and its discretization.

EXPERIMENTAL SETUP

Experimental data were used for an assessment of each model (acoustic and fluid dynamics) capability to predict the flow through the muffler. The quantities to be measured in the experimental setup are pressure in the muffler next to the suction valve, valve displacement, 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 ^{o}C and 42 ^{o}C, respectively. Rotation speed of the compressor is assumed constant, with a frequency of 60 Hz, and the refrigerating fluid is R134a.

RESULTS AND DISCUSSION

The performance of the compressor obtained from measurements was compared with 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 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. In the CFD model the energy equation is solved and hence the temperature at the entrance of the suction valve can be evaluated. The external heat transfer coefficient was estimated with reference to available data for similar compressors.

Figure 2 shows a comparison between experimental data and numerical results for suction valve displacement and pressure at the valve entrance corresponding to the compressor operating under ASHRAE LBP conditions. Overall, the level of agreement with experimental data is better for the CFD model when compared to both versions of the acoustic model. The same is true regarding the pressure pulsation originated by the muffler at the entrance of the suction valve. It can also be noticed that CFD results are approximately in phase with the experimental data when the suction valve is closed, a feature not captured by the acoustic models.

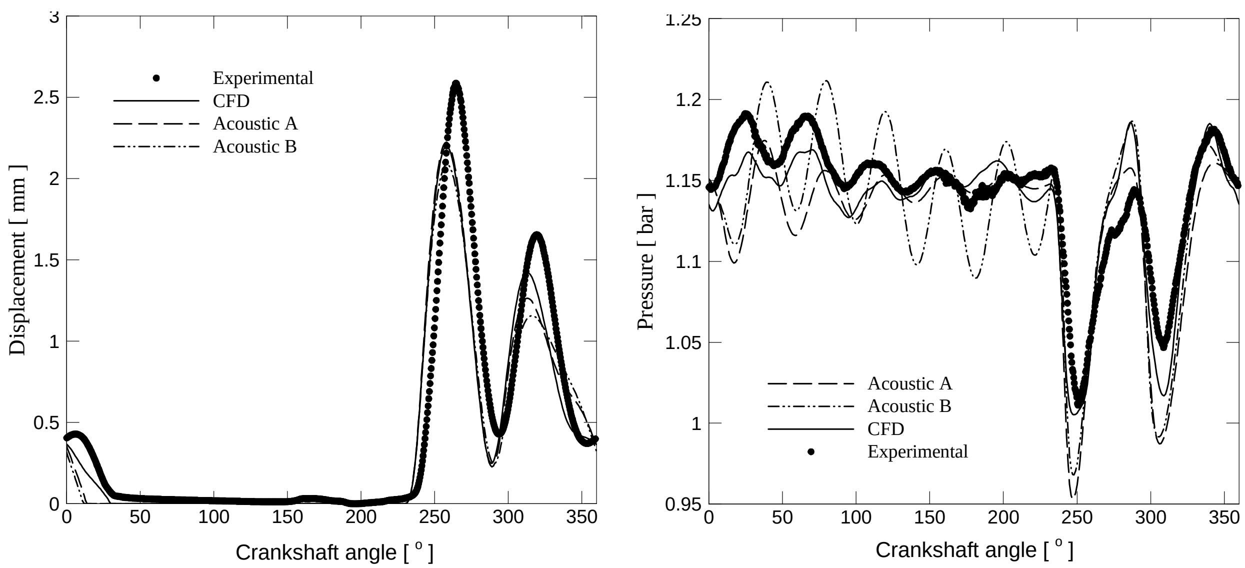


Figure 2: Suction valve displacement and pressure in the muffler for ASHRAE LBP conditions.

Figure 3 presents experimental data and numerical results for pressure pulsation at the inlet of the suction valve, considering the compressor working with T_E = -27 °C and T_C = 42 °C. As for the ASHRAE LBP conditions, the CFD model predicts more accurately the pressure behavior measured in the experimental unit.

During the suction process occurs an important interaction between the gas flow and the valve dynamics. As seen from the previous results, all models adopted in this work seem to predict this phenomenon fairly well, with the CFD model being slightly superior to both acoustic model versions. It is worthwhile to recall that friction losses are evaluated in the CFD model according to standard correlations developed for stationary flow. An analysis of such losses for strong transients as found in the suction muffler could provide useful information to refine further the CFD model.

Results for pressure in the cylinder as a function of crankshaft angle are shown in Figure 4. The most significant difference between numerical and experimental results occurs around the top dead center position, during the opening of the discharge valve. On the other hand, it is difficult to detect from the figure any visible variation caused by the employment of the models in the suction process.

Tables 1 and 2 show measurements for the compressor performance, and the corresponding estimate given by each model, for both operating conditions. For the ASHRAE LBP condition in Table 1 the comparisons indicate that the models slightly overestimate the compressor efficiency (maximum of 0.7%). As for the condition represented by T_E = -27 °C and T_C = 42 °C, all models predict also very similar results, with the EER being underestimated by approximately 2.8%.

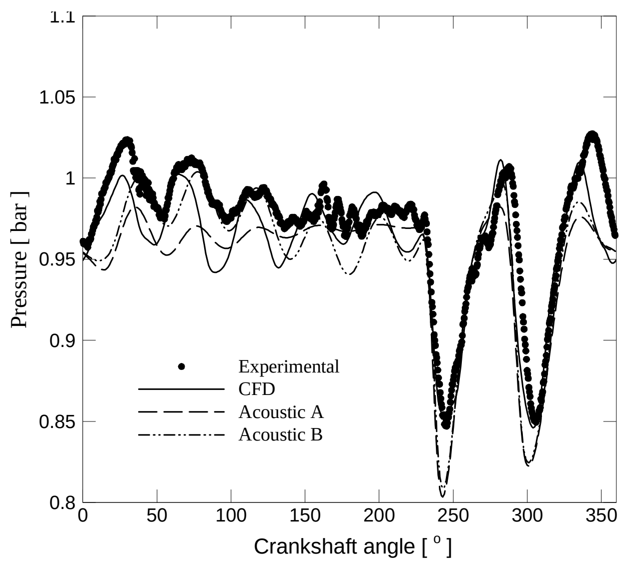
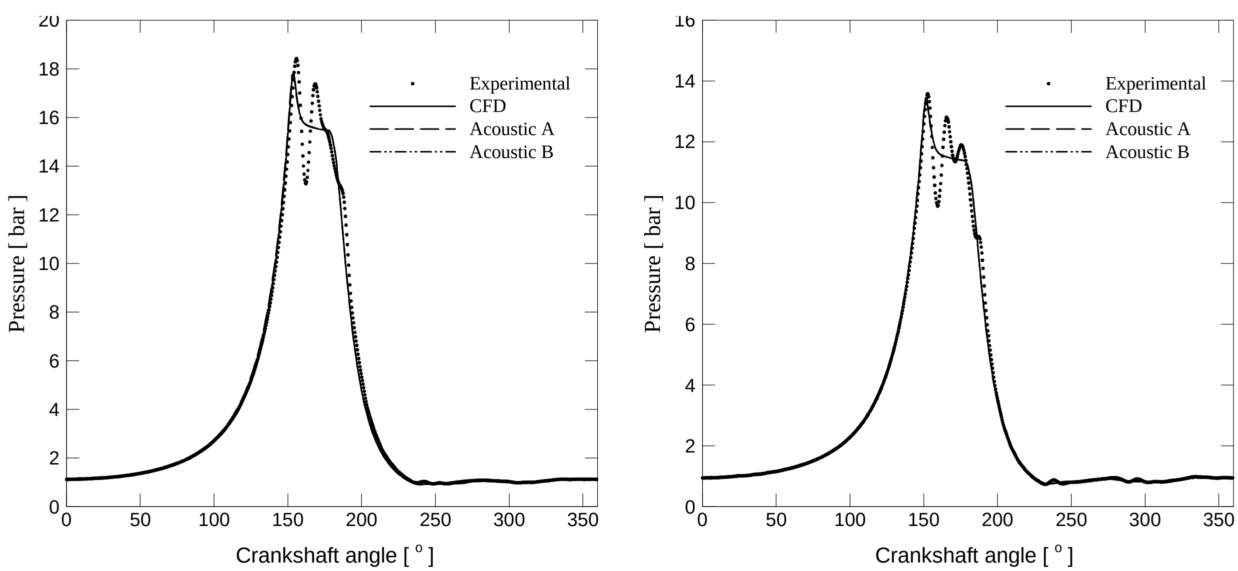


Figure 3: Pressure at the suction valve entrance for T_E = -27 °C; T_C = 42 °C.



a) ASHRAE LBP conditions

b) T_E = -27 °C; T_C = 42 °C

Figure 4: Pressure in the cylinder.

Table 1: Results for compressor performance: ASHRAE LBP condition

	Capacity (BTU/h)	Consumption (W)	EER (BTU/Wh)
Experimental data	778	136.1	5.72
CFD model	788	137.1	5.75
Acoustic model A – variable loss factor	805	140.1	5.75
Acoustic model B – fixed loss factor	802	139.3	5.76

Table 2: Results for compressor performance: T_E = -27 °C; T_C = 42 °C

	Capacity (BTU/h)	Consumption (W)	EER (BTU/Wh)
Experimental data	711	117.2	6.07
CFD model	695	117.7	5.90
Acoustic model A – variable loss factor	701	118.9	5.90
Acoustic model B – fixed loss factor	701	118.7	5.91

CONCLUSIONS

The present work considered a comparative analysis between two approaches for modeling the pulsating flow in mufflers and how they affect the prediction of compressor performance. The first methodology is an analytical acoustic model, which requires empirical coefficients to estimate friction losses. The other is a one dimensional computational fluid dynamics model based on the finite volume methodology that 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 need for different loss factors according to the compressor operating condition, or whether the suction valve is closed or open. Such difficulties are not present in the CFD model, which evaluates friction losses and has the additional benefit of providing the temperature distribution in the muffler. From comparisons between numerical and experimental results presented in this work, it has been concluded that the fluid dynamics model offers a good route to simulate the gas dynamics in the muffler.

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.

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.

Choi, J.K., Joo, J.M., Oh, S.K., Park, S.W., “Smart Suction Muffler Design for a Reciprocating Compressor”, Proc. Compressor Engineering Conference at Purdue, West Lafayette, Indiana, USA, p. 619-626, 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.

Fagotti, F., Possamai, F.C., "Using Computational Fluid Dynamics as a Compressor Design Tool", Proc. Compressor Engineering Conference at Purdue, West Lafayette, Indiana, USA, p. 137-144, 2000.

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.

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.