THEORETICAL ANALYSIS OF TRANSIENT EFFECTS ON THE FLOW THROUGH REED TYPE VALVES

Rodrigo Link, Rodrigo_Link@embraco.com.br
Research & Development Group
EMBRACO
89219-901, Joinville, SC, Brazil

César José Deschamps, deschamps@polo.ufsc.br
POLO Research Laboratories for Emerging Technologies in Cooling and Thermophysics
Federal University of Santa Catarina
88040-900, Florianópolis, SC, Brazil

Simulation models adopted in the preliminary design of small reciprocating compressors usually employ the concepts of effective force area, A_F, and effective flow area, Av, to characterize the performance of reed type valves. The evaluation of such parameters considers a steady state hypothesis for the flow through the valve port, not taking into account flow inertia that takes place especially in the opening and closing stages of the valve. This paper presents a numerical study of inertial effects in the flow through the discharge valve of a small reciprocating compressor. The results show that the standard procedure for estimates of effective areas may result totally inappropriate in some flow situations.

***Keywords**: reed type valves, flow inertia, reciprocating compressors.*

1. INTRODUCTION

Several compressor simulation methodologies are available in the literature to simulate compressors. Despite the capabilities of multidimensional formulations, numerical methods based on integral formulation are usually adopted to simulate reciprocating compressors (Ussyk, 1984; Pérez-Segarra *et al.*, 1994; Longo and Gasparella, 2003). Such models are very convenient for preliminary analyses of compressor design, since their associated computational cost is much less expensive than that of multidimensional models.

In the integral formulation, each compressor component is mathematically described, with an account for piston displacement as a function of crankshaft angle, thermodynamic processes inside the cylinder, valve dynamics and gas pulsation in mufflers. Several parameters are calculated for a compressor cycle, such as instantaneous pressure and temperature, mass flow rate through valves, energy and mass losses, refrigerating capacity, etc. Thermodynamic properties for the refrigerant may be evaluated through a program link to different libraries.

Reed valves are essential parts in hermetic compressors. These valves are called automatic because they 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, and in order to obtain an optimum valve system, it is crucial to understand the phenomena associated with the flow through the valve as well as to its dynamics. The dynamics of such valves can be expressed in a simplified way, by using a one-degree-of-freedom model as follows:

$$m_{eq}\ddot{s}_v + c\dot{s}_v + ks_v = F_v - F_o \tag{1}$$

where m_{eq}, c and k are, respectively, the equivalent mass, damping coefficient and stiffness of the reed valve, which have to be specified. On the other hand, F_v is the flow induced force on the reed and F_o can accommodate any other force, such as reed pre-tension or oil stiction. Finally, s_v, $\dot{s}_v$ and $\ddot{s}_v$ are the instantaneous lift, velocity and acceleration, respectively.

The force F_v is commonly obtained with reference to the concept of effective force area, A_F ($=F_v/\Delta p_v$), which is determined from the pressure difference across the valve, Δp_v ($= p_u - p_d$). The effective force area can be understood as a parameter related to how efficiently the pressure difference Δp_v acts to open the valve. On the other hand, the mass flow rate through the valve is evaluated from data for the effective flow area, A_V. For a certain pressure drop, A_V expresses the actual mass flow rate, $\dot{m}_v$, with reference to an isentropic flow condition:

$$A_V = \frac{\dot{m}_v}{p_u\sqrt{2\gamma/[(\gamma-1)RT_u]}\sqrt{(p_d/p_u)^{2/\gamma}-(p_d/p_u)^{(\gamma+1)/\gamma}}} \tag{2}$$

where p_u and T_u denote pressure and temperature at the flow upstream, p_d is the pressure at the flow downstream, γ is the specific heat ratio and R is the refrigerant gas constant.

Generally, flow inertia is not considered in the evaluation of effective areas and this may bring considerable errors into the evaluation of F_v and $\dot{m}_v$, especially for high speed compressors. For instance, Habing and Peters (2006) adopted a transient flow hypothesis to express the pressure drop through a compressor valve, Δp_v:

$$\Delta p_v = \frac{1}{2}\rho\left(\frac{\Phi_v}{A_V}\right)^2 + \rho L\frac{d}{dt}\left(\frac{\Phi_v}{A_V}\right) \tag{3}$$

in which L is the valve port length and Φ and ρ are the volumetric flow rate and the density in the valve port, respectively. It becomes clear that, although the first term on the right side of equation (3) is always positive, the second term may result either positive or negative values, depending on whether the flow is accelerating or decelerating. Therefore, in flow situations with strong transients Δp_v may become positive or negative for a certain flow rate, Φ, regardless of the flow direction.

Kerpicci and Oguz (2006) conducted a comparative analysis between a transient fluid-structure simulation model and a stationary one-dimensional model. The authors observed a considerable difference between the results for mass flow rate of both models and suggested corrections to the one-dimensional model based on flow inertia. Haas et al. (2007) numerically analyzed the mass flow rate through a simplified IC engine valve geometry. Predictions obtained from transient simulations were compared to results returned by an integral formulation based on a standard definition of effective flow area that makes no reference to inertial effects. The authors found that errors in the opening and closing stages of the valve motion almost cancel each other, diminishing the overall impact of fluid flow inertia on estimates for mass flow rate obtained with the effective flow area.

This paper presents a theoretical study of inertial effects in a simplified geometry of compressor discharge valve. With this purpose, a two-dimensional numerical model was developed to predict the fluid-structure interaction, allowing the analysis of both the valve dynamics and the flow through the discharge port.

2. MATHEMATICAL MODEL

The computational model was developed with a commercial CFD code (ANSYS, 2006) based on the finite volume method. Due to the presence of moving surfaces such as the piston inside the cylinder and the discharge valve, 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 computational cost and solution accuracy. A second-order upwind scheme was adopted to interpolate the flow quantities, except for the turbulent quantities which were interpolated using the 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 via a segregated algorithm. The ideal gas hypothesis was used to predict the density of the working fluid (R600a), whereas polynomial regression curves based in the REFPROP 7.0 data base (Lemmon *et al.*, 2002) were adopted to evaluate all other proprieties, such as thermal conductivity, specific heat and viscosity.

The solution domain included the discharge valve, the discharge chamber and the compression chamber formed by the cylinder and the piston. During the discharge process, the valve moves simultaneously with the piston motion. The solution starts with piston in the bottom dead center (BDC) position. At a certain point of the compression process the discharge valve opens, with the simulation continuing until the valve is closed again. Figure 1 shows a representation of the axisymmetric solution domain at three piston positions: a) bottom dead center (BDC); b) discharge process and c) valve closed (end of simulation). It should be mentioned that the geometry dimensions in the figure are merely illustrative and do not represent the actual geometry of the compressor.

The piston motion is described by the following relationship:

$$L_p = \sqrt{L_{cr}^2 - e\sin\theta - L_{ps}} - e\cos\theta \tag{4}$$

where L_p is the piston position, L_{cr} is the connecting rod length, L_{ps} is the misalignment between cylinder and crankcase, e is the eccentricity and θ is the crankshaft angle.

The evaporating pressure and the temperature in the suction chamber were employed for initial estimates of thermodynamic properties inside the compression chamber. As far as boundary conditions are concerned, the temperature and velocity at the compression chamber walls were set to 80°C and zero, respectively. However, for the reed and piston surfaces the velocities were obtained from equations (1) and (4), respectively.

The differential equation for the valve dynamics, equation (1), was solved using an explicit Euler method and by considering the force F_v to be constant during each time step. An oil stiction force of 0.5N was assumed to act on the reed valve up to a lift of 10μm. Finally, the condensing pressure was imposed as the boundary condition at the exit boundary of the discharge chamber.

The numerical solution was verified for truncation errors associated with spatial and temporal discretizations. For the spatial discretization three levels of grid refinement were tested: i) 2745 volumes (coarse grid); ii) 4849 volumes (median grid) e iii) 10390 volumes (fine grid). Figure 2 presents predictions for mass flow rate $\dot{m}_v$ and force F_v obtained with the aforementioned meshes, showing that truncation errors are sufficiently small. In the same way, the influence of temporal discretization was performed by considering three time steps, specified in terms of crank angle increments: i) 0.100°; ii) 0.075° e iii) 0.050°. The results showed that a time step corresponding to 0.100° is enough to describe flow transients.

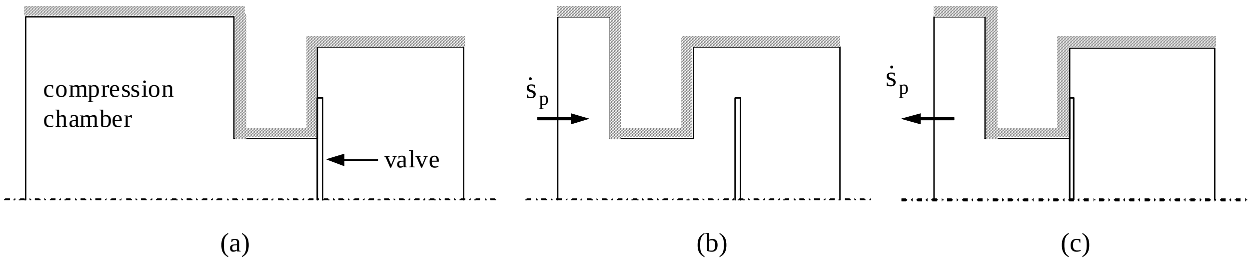


Figure 1: Solution domain at different time steps during the numerical simulation: (a) bottom dead center (BDC); (b) discharge process and (c) valve closed (end of simulation).

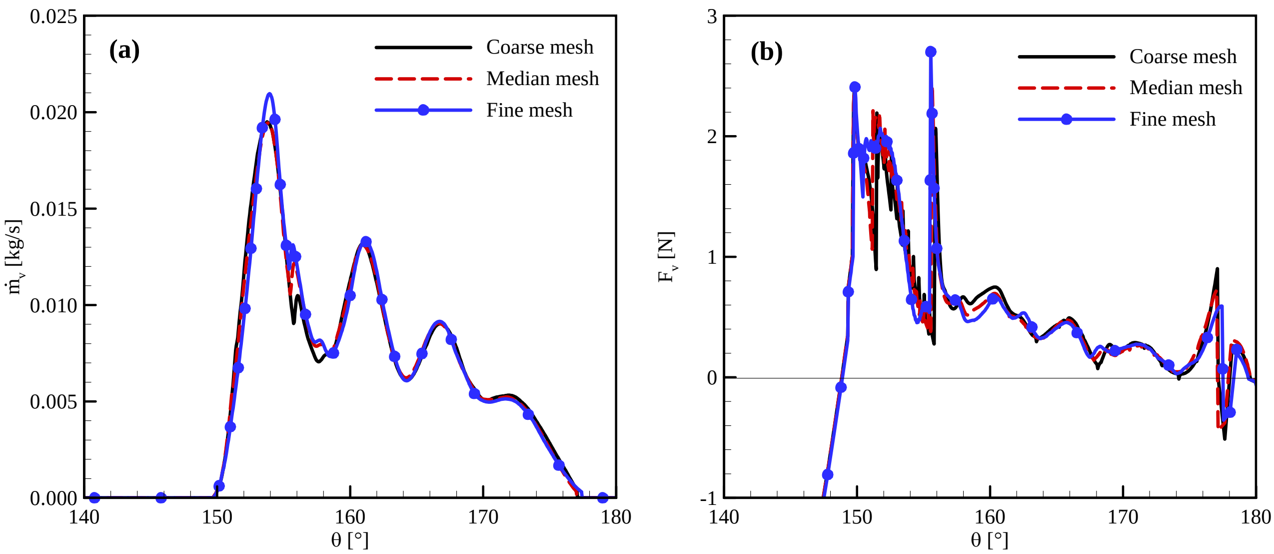


Figure 2: Numerical predictions for different grid refinements: (a) mass flow rate; (b) force.

3. RESULTS

The present numerical analysis is focused on predictions for mass flow rate, $\dot{m}_v$, flow induced force, F_v, and valve lift, s_v, as well as valve pressure drop, Δp_v. Flow inertia was investigated for two geometries of discharge valve, represented by two diameter ratios (D/d= 1.50 and 1.95), where D is the reed valve diameter and d is the valve port diameter. Three compressor speeds (f = 25, 50 and 75Hz) were tested, with the compressor displacement being adjusted to keep the same mass flow rate in all simulations.

When predictions for mass flow rate through the valve are written as a function of the pressure drop Δp_v, one can notice four distinct flow regions in Fig. 3. In regions II and III both the mass flow rate and the pressure drop have the same signs, in line with the standard definition of effective flow area. Accordingly, region II characterizes the normal flow direction in which the fluid exits the compression chamber and region III a backflow condition, which deteriorates the compressor performance.

Yet, mass flow rate and pressure drop have opposite signs in regions I and IV, due to dominant inertial effects. As Fig. 3 shows, the standard definition of effective flow, given by equation (2), is not applicable to regions I and IV, since it neglects inertial effects and considers the mass flow rate to be uniquely related to the pressure drop, Δp_v. In fact, for steady flow condition the pressure drop represents the valve head loss, which is fully correlated with the mass flow rate.

However, this is not the case for transient flow and the pressure drop can be positive or negative, regardless the flow direction, as indicated by equation (3). Hence, given the negative pressure drop in region I, equation (2) would indicate a backflow condition when the flow was in fact exiting the compression chamber. In order to circumvent this problem, a new definition of effective flow area should include flow inertia.

Figure 4 depicts results for the flow induced force on the reed valve, F_v, according to the pressure drop Δp_v. For an assessment of inertial effects on F_v, two flow conditions identified by point A (s_v = 0.016mm, Δp_v = 155.4kPa) and point B (s_v = 0.160mm, Δp_v = 39.69kPa) in Fig. 4b were simulated for the hypothesis of steady state flow. As Table 1 shows, the results for F_v predicted with the steady state hypothesis are considerably different from those related to the actual transient flow condition. This is an outcome of the very distinctive pressure distributions that prevail on the valve surface for both flow conditions, as illustrated in Fig. 5.

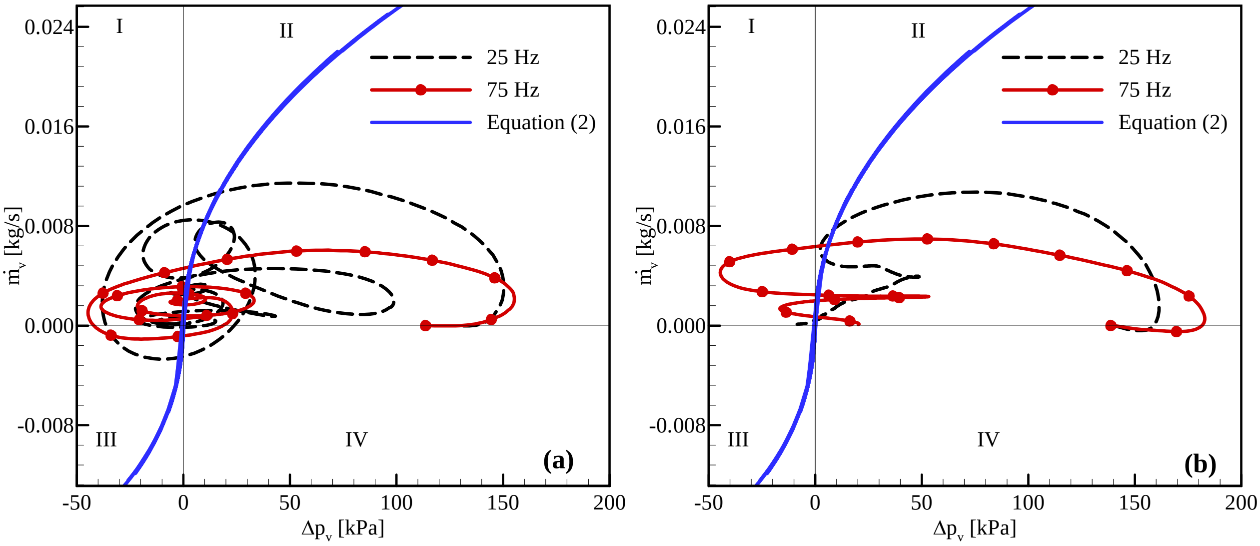


Figure 3: Predictions for mass flow rate through the valve: (a) D/d = 1.50; (b) D/d = 1.95.

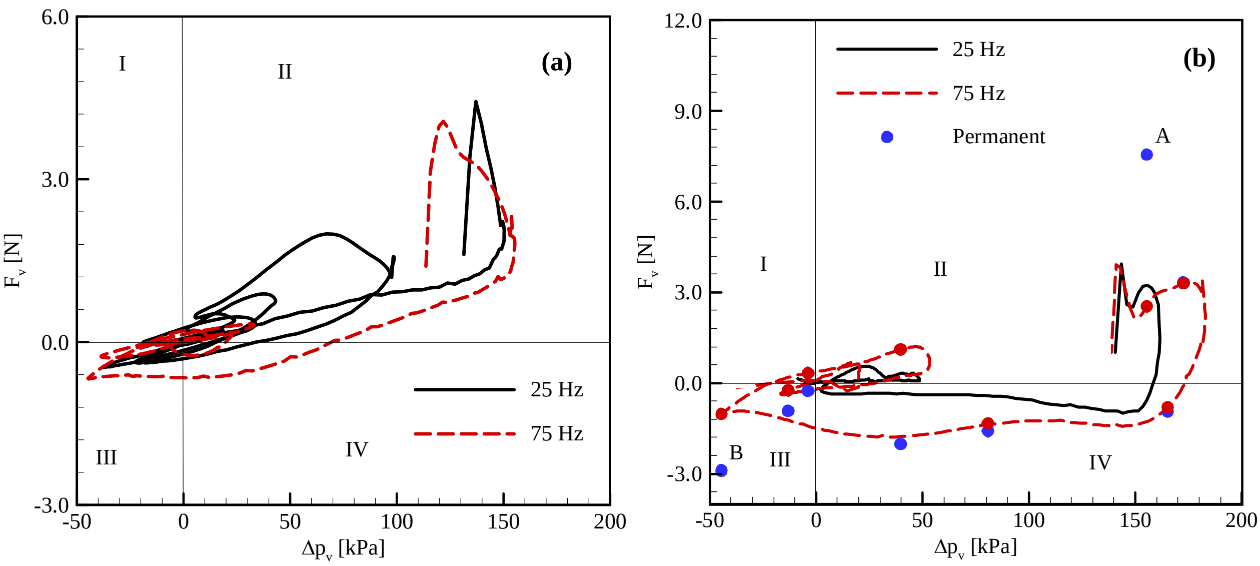


Figure 4: Predictions for flow induced force on the valve: (a) D/d = 1.50; (b) D/d = 1.95.

Table 1: Predictions for flow induced force, F_v, considering steady state and transient flow conditions.

Flow condition		Transient flow	Steady state flow
A	s_v = 0.016mm, Δp_v = 155.4kPa	2.6N	7.6N
B	s_v = 0.160mm, Δp_v = -44.0kPa	-1.1N	-2.9N

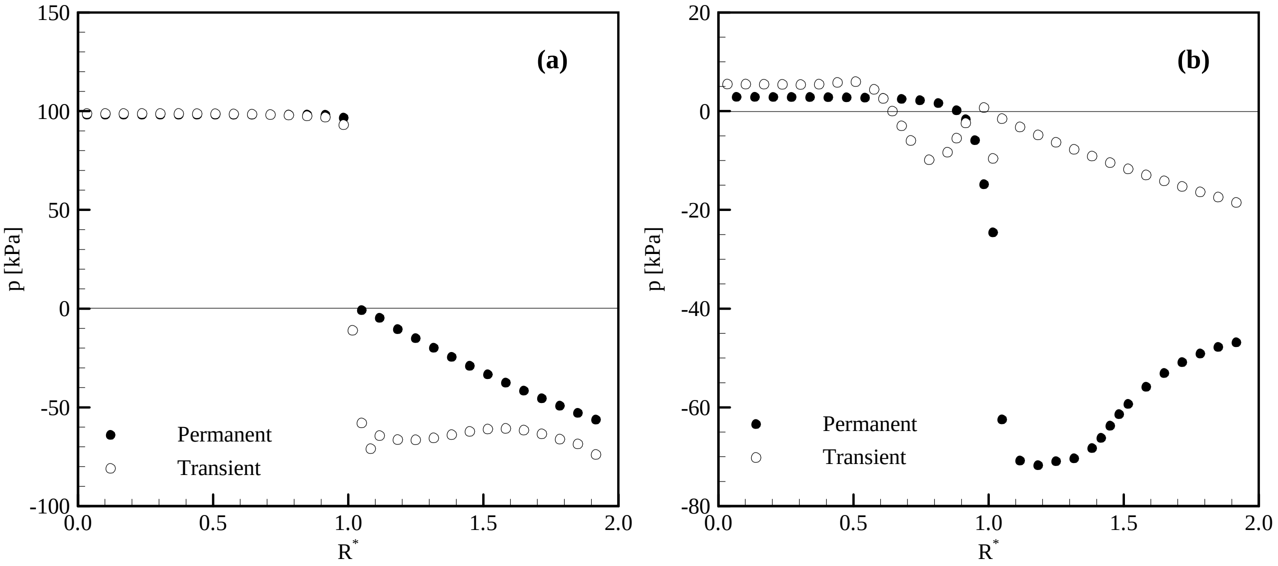
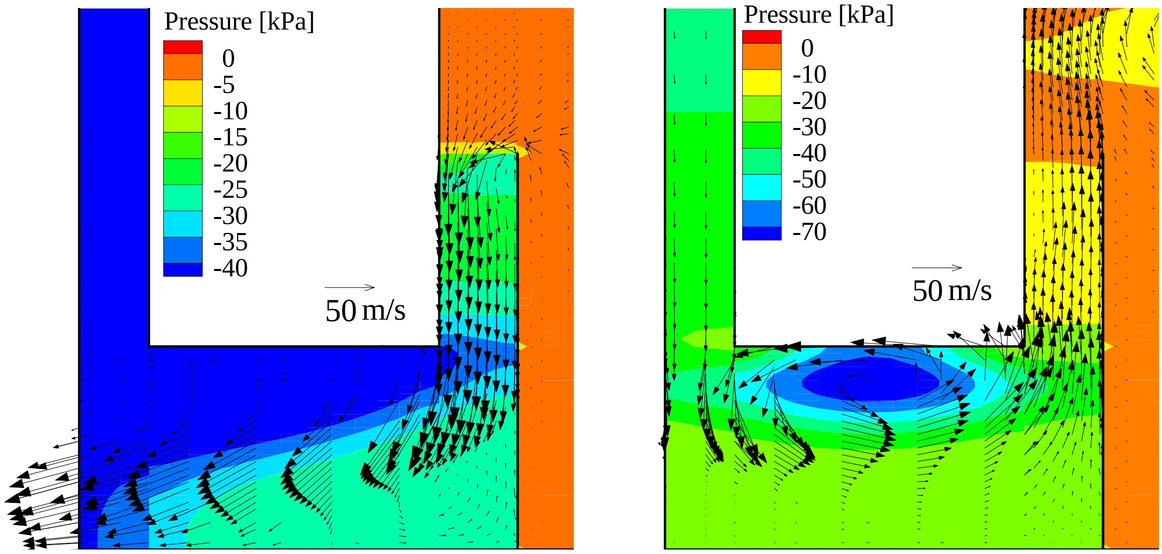


Figure 5: Pressure distribution on the reed valve surface; D/d = 1.95. (a) s_v = 0.016mm, Δp_v = 155.4kPa; (b) s_v = 0.16mm, Δp_v =-44.0kPa.

In the condition A, the transient flow is being accelerated and this is the reason for the lower pressure levels in the diffuser region ($1.0 \leq R^* \leq 1.95$). For the condition B, there is a negative Δp_v and, therefore, a backflow is predicted with the steady state formulation. However, due to flow inertia, in the transient formulation the flow is found to be in the opposite direction (i.e., exiting the valve). It is worthwhile to note that for other flow conditions marked with blue dots in Fig. 4b, the inertial effect is almost negligible and the corresponding forces F_v become very close to the values observed for the transient flow condition (red line).

The velocity fields associated with steady and transient flow conditions are shown in Fig. 6 for flow condition B indicated in Table 1. As can be seen, there is a recirculating flow region in the valve port for the transient flow condition. In Fig. 6b one can also notice that mass exits the valve, when the pressure drop would indicate a flow in the opposite direction. When the flow is simulated for a steady state formulation with s_v = 0.16mm and Δp_v =-44.0kPa, a very different flow pattern results, as shown in Fig. 6a, and a backflow results in the valve.



(a) permanent.   (b) transient.

Figure 6: Velocity vector field for condition D/d = 1.95, s_v = 0.160mm and Δp_v = -44.0kPa.

The resulting negative force on the valve is not related to the flow direction but to the valve geometry. It is well established in the literature that large diameter ratios D/d give rise to a negative pressure distribution along the diffuser region formed between the reed and the valve seat. Depending on such negative pressure levels, a negative force occurs and the valve exhibits an unstable operating condition.

The mass flow rate through the valve as a function of valve displacement is also affected by the valve geometry and inertial effects. For *D/d* = 1.50, depicted in Fig. 7a, it is possible to verify that the valve opening is almost twice larger for the frequency 25Hz than that occur for 75Hz. Therefore, the mass flow rate ends up being very different in both frequencies. Figure 7a also shows the presence of negative mass flow rate when the valve reaches its maximum displacement. This phenomenon is associated with a decrease of pressure inside the cylinder and it is more evident at the operating condition corresponding to 25Hz.

Turning the attention to the greater diameter ratio, D/d = 1.95, one can note a smaller influence of the operating frequency on the valve displacement (Fig. 7b). For this valve geometry, there is a greater viscous friction loss during the discharge process and stronger flow inertia, and the mass flow rate does not reach a reversal condition such as observed for the smaller diameter ratio (D/d = 1.50), although some backflow can be noticed at small valve lifts, s_v.

It is well known that the flow induced force on the reed is greatly affected by the valve geometry, with the force magnitude dropping sharply as the valve lift is increased. This can be confirmed in Fig. 8, even in the presence of inertial effects. In fact, the analysis of Fig. 8 shows that flow inertia considerably affects the magnitude of the resulting force, as the operating frequency is varied, but not the form in which F_v varies with the valve displacement.

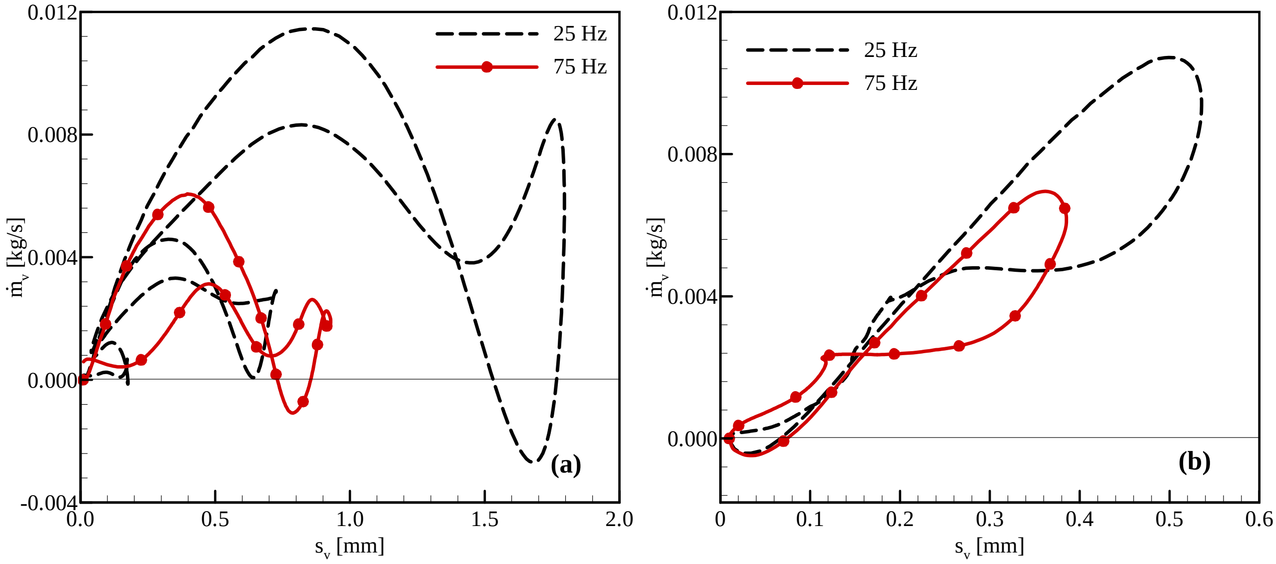


Figure 7: Predictions for mass flow rate through the valve: (a) D/d = 1.50; (b) D/d = 1.95.

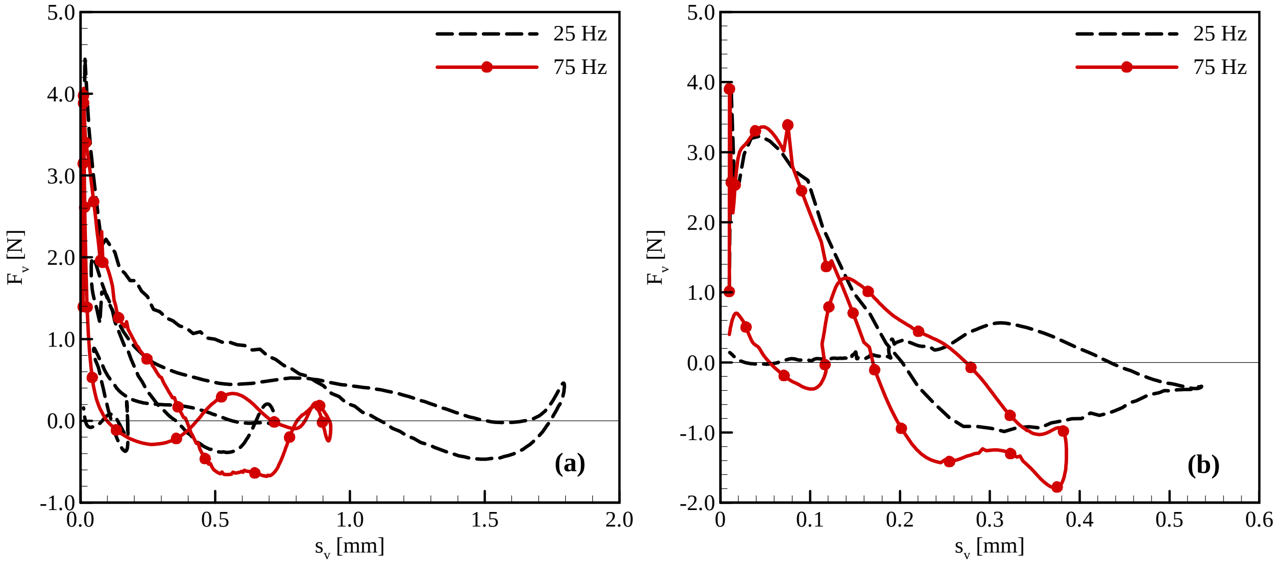


Figure 8: Predictions for flow induced force on the valve: (a) D/d = 1.50; (b) D/d = 1.95.

4. CONCLUSIONS

This paper presented a theoretical analysis of inertia effects in the flow through a simplified geometry of compressor discharge valve. A numerical model based on the finite volume method was developed, taking into account the complex interaction between the valve dynamics and the flow through the valve port. It has been observed that standard definitions of effective flow and force areas are not capable of describing the mass flow rate and the flow induced force in the initial and final stages of the valve opening. This study is an initial step towards the understanding of flow inertia in compressor valves, aimed at developing more general correlations for effective flow and force areas that can be used in transient flow conditions.

5. ACKNOWLEDGEMENTS

The material discussed herein forms part of a joint technical-scientific program between the Federal University of Santa Catarina and EMBRACO, partially funded by CNPq (Brazilian Research Council) through Grant No. 573581/2008-8 (National Institute of Science and Technology in Refrigeration and Thermophysics). Support from FINEP (Federal Agency of Research and Projects Financing) is also acknowledged.

6. REFERENCES

ANSYS Inc, 2006, Fluent 6.3 User’s Guide.

Haas, D.A., Bertoldi, D. and Deschamps, C.J., 2007, “Transient Effects in the Valve Flow of Engines” (in Portuguese), *Proc. SAE Conference*, São Paulo, Paper 2007-01-2960.

Habing, R., Peters, M.C.A.M., 2006, “An Experimental Method for Validating Compressor Valve Vibration Theory”, *J. Fluids and Structures*, vol. 22, p. 683–697.

Kerpicci, H. and Oguz, E., 2006, “Transient Modeling of Flows Through Suction Port and Valve Leaves of Hermetic Reciprocating Compressors”, *Proc. Int. Compressor Engineering Conference at Purdue*, West Lafayette, Paper C122.

Lemmon, E.L., McLinden, M.O. and Huber, M.L., 2002, “REFPROP 7.0, NIST”, USA.

Longo, G.A. and Gasparella, A., 2003, “Unsteady State Analysis of the Compressor Cycle of a Hermetic Reciprocating Compressor”, *Int. J. Refrigeration*, vol. 26, p. 681-689.

Pérez-Segarra CD, Escanes F and Oliva A, 1994, “Numerical Study of the Thermal and Fluid-Dynamic Behavior of Reciprocating Compressors”, *Proc. Int. Compressor Engrg. Conf. at Purdue*, pp. 145-150.

Soedel, W., 2007, “Sound and Vibration of Positive Displacement Compressors”, CRC Press, 342 p.

Ussyk, M.S., 1984, “Numerical Simulation of the Performance of Hermetic Reciprocating Compressors” (in Portuguese). *M.Sc. Thesis*, Department of Mechanical Engineering, Federal University of Santa Catarina, Brazil.

7. RESPONSIBILITY NOTICE

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