Purdue University
Purdue e-Pubs

International Compressor Engineering Conference | School of Mechanical Engineering

2010

Numerical Analysis of Transient Effects on Effective Flow and Force Areas of Compressor Valves

Rodrigo Link
Embraco

Cesar Jose Deschamps
UFSC

Follow this and additional works at: http://docs.lib.purdue.edu/icec

Link, Rodrigo and Deschamps, Cesar Jose, "Numerical Analysis of Transient Effects on Effective Flow and Force Areas of Compressor Valves" (2010). *International Compressor Engineering Conference.* Paper 1996.
http://docs.lib.purdue.edu/icec/1996

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at https://engineering.purdue.edu/Herrick/Events/orderlit.html

Numerical Analysis of Transient Effects on Effective Flow and Force Areas of Compressor Valves

Rodrigo LINK[1]*, Cesar J. DESCHAMPS[2]

[1]EMBRACO, Research & Development Group
Joinville, Santa Catarina, Brazil
Rodrigo_Link@embraco.com.br

[2]POLO Research Laboratories for Emerging Technologies in Cooling and Thermophysics
Federal University of Santa Catarina
Florianópolis, SC, Brazil
deschamps@polo.ufsc.br

* Corresponding Author

ABSTRACT

Current models adopted to evaluate effective flow and force areas of compressor valves consider a steady state condition for the flow through suction and discharge ports. Therefore, such models do not take into account flow inertial effects that occur especially in the opening and closing stages of valve displacement. This paper presents a study of inertial effects in the flow through discharge ports. To that extent, a numerical model based on the finite volume method is adopted, considering the interaction between the valve dynamics and the flow through the valve port. The results indicate that inertial effects can significantly affect the flow through the valve and make the standard concept of effective areas inappropriate.

1. INTRODUCTION

The correct understanding of the phenomena associated with the flow through reed type valves is essential for the design of high efficiency compressors. Thermodynamic losses due to suction and discharge valves can be assessed by using numerical and experimental procedures. In the experimental approach, measurements are usually provided for valve lift and pressure in the compression, discharge and suction chambers. Figure 1 illustrates a typical p-V diagram obtained experimentally, in which energy losses in the discharge and suction systems are identified by the areas above the pressure in discharge line and below the pressure in the suction line, respectively.

The most comprehensive method to numerically analyze the performance of compressor valves is offered by three-dimensional numerical models, notwithstanding its high computational cost and some modeling difficulties, such as the uncertainty in the evaluation of oil stiction force. Kim *et al.* (2006) presented an optimization procedure for compressor valves, by coupling the flow simulation with a genetic algorithm. 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.

Kim *et al.* (2008) numerically analyzed the impact of discharge valves against its stopper, commonly used in reciprocating compressors, and its influence on the valve reliability. Pereira *et al.* (2008) conducted an experimentally validated numerical analysis of flow through three-dimensional geometries of suction and discharge valves of a small reciprocating compressor, considering the coupling between the flow and the valve dynamics. The authors evaluated the contributions of both the reed valve and the port orifice on the thermodynamic loss and investigated design alternatives to improve the compressor efficiency.

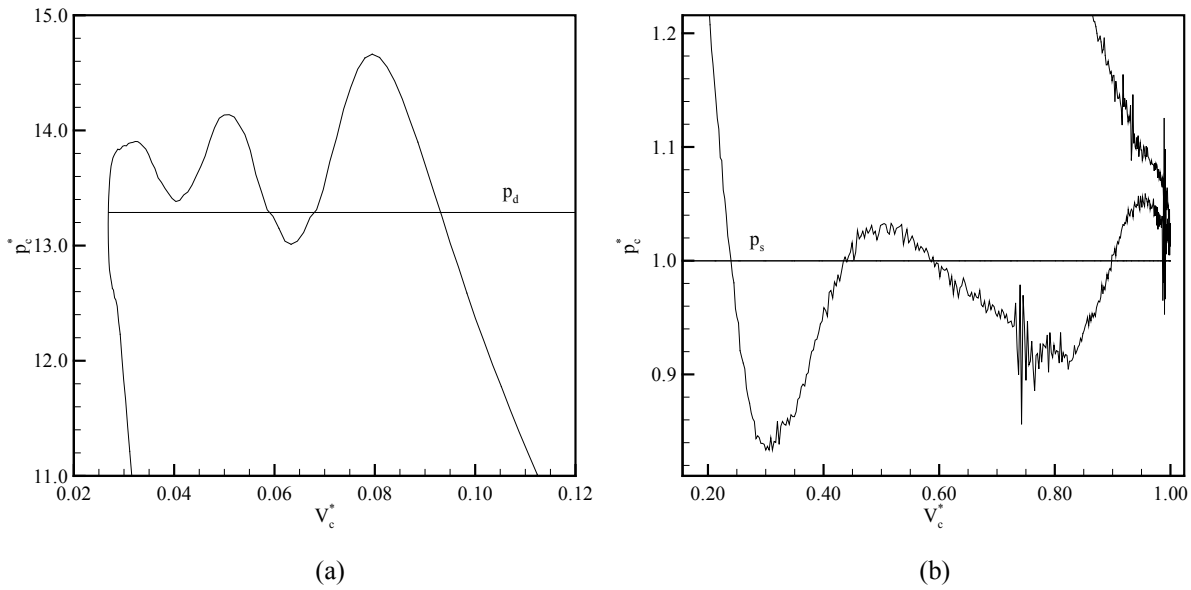


Figure 1: Experimental data for dimensionless p-V diagram: (a) discharge process; (b) suction process.

Despite the capabilities of multidimensional formulations, several numerical methods based on integral formulation have been proposed and 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 open and close depending on the pressure difference between the cylinder and the suction/discharge chamber established by the piston motion. 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 (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.

Haas and Deschamps (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 inertial effects on estimates for mass flow rate obtained with the effective flow area.

This paper presents a theoretical study of flow 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 (Fluent Inc., 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 move 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 2 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 3 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.

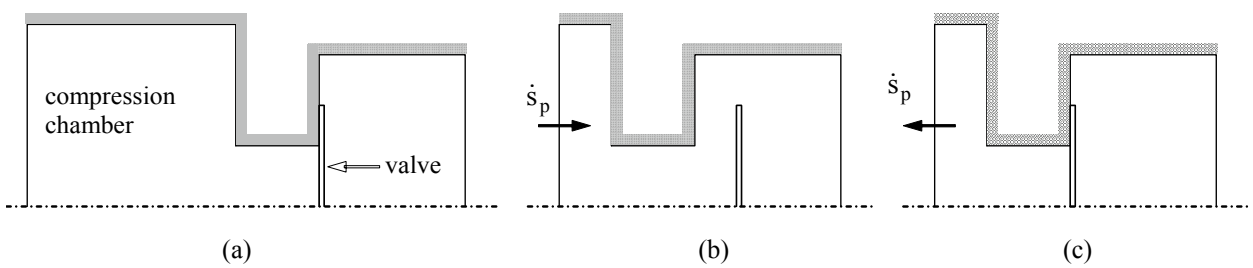


Figure 2: Solution domain at different time steps during the numerical simulation: (a) bottom dead center (BDC); (b) discharge process and (c) valve closed (end of simulation).

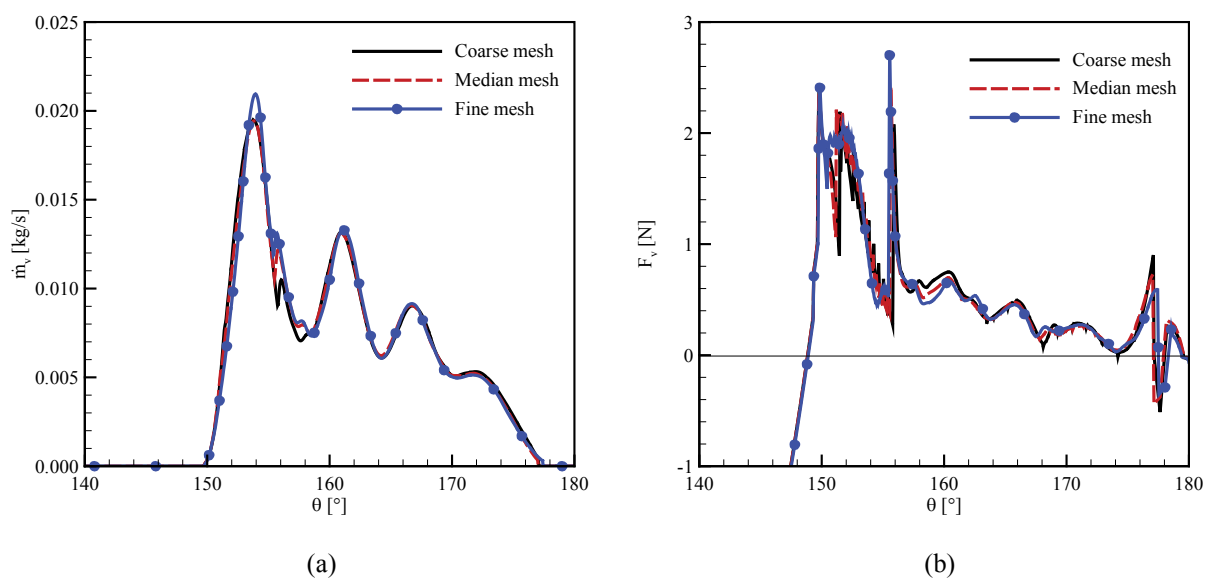


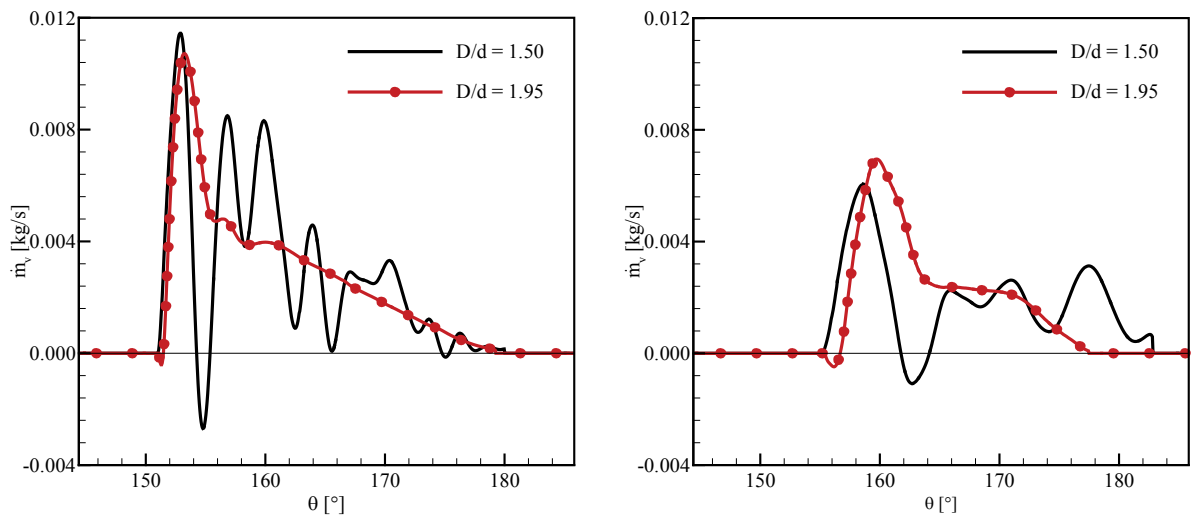
Figure 3: Numerical predictions for different grid refinements: (a) mass flow rate; (b) force.

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 inertial effects were investigated for two geometries of discharge

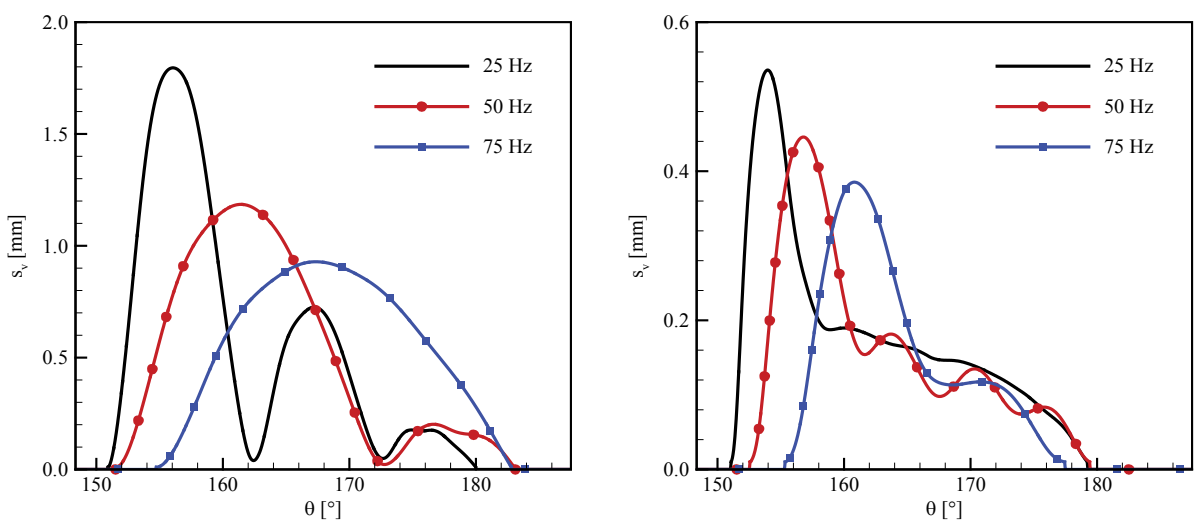
valve, represented by two diameter ratios (*D/d*= 1.50 and 1.95), where *D* is the reed valve diameter and *d* is the valve port diameter. Three compressor speeds (*f* = 25, 50 and 75Hz) were tested, with the compressor displacement being adjusted to keep the same mass flow rate in all simulations.

Figure 4 presents numerical results for mass flow rate through the valve as a function of the crankshaft angle for two compressor speeds (*f* = 25 and 75Hz) and two valve diameter ratios (*D/d*= 1.50 and 1.95). It can be seen that the higher diameter ratio (*D/d* = 1.95) gives rise to a more stable flow condition, characterized by fewer oscillations in the mass flow rate. Figure 5 demonstrates that valve displacement is strongly affected by both the operation condition and the valve geometry, as one would expect. Again, the greater valve diameter ratio returns a more stable opening motion for the discharge valve.



(a) (b)

Figure 4: Predictions for mass flow rate through the valve: (a) 25 Hz; (b) 75 Hz.

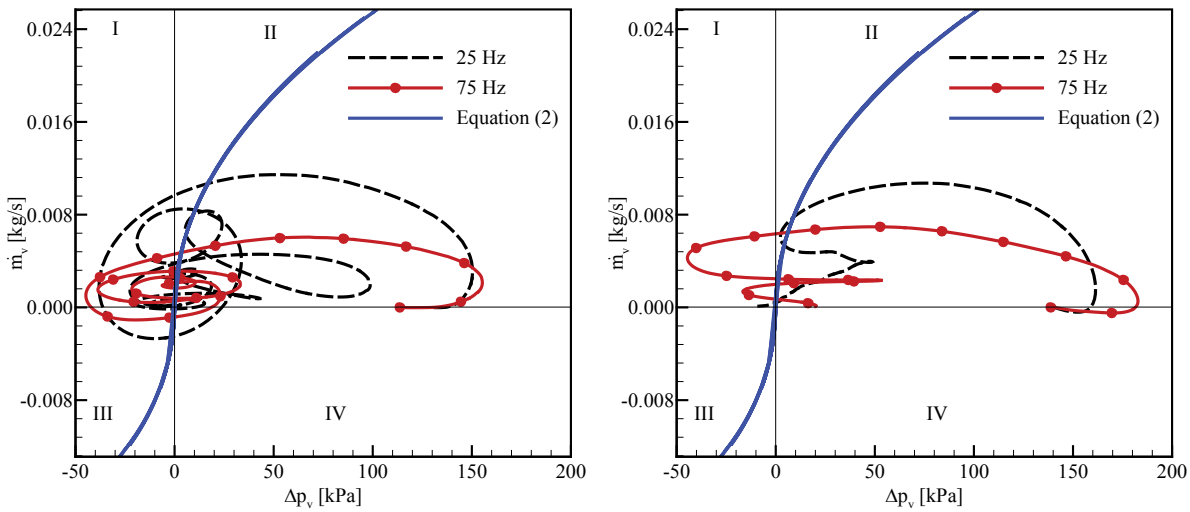


(a) (b)

Figure 5: Predictions for valve displacement: (a) D/d = 1.50; (b) D/d = 1.95.

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. 6. 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. 6 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 inertial effects.



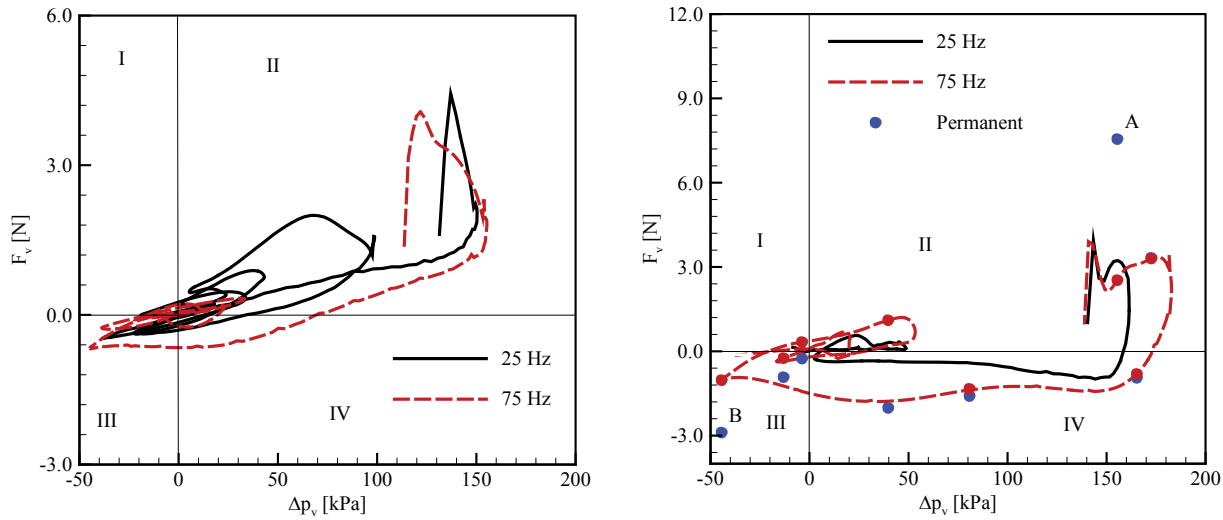
(a) (b)

Figure 6: Predictions for mass flow rate through the valve: (a) D/d = 1.50; (b) D/d = 1.95.

Figure 7 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. 7b 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. 8.

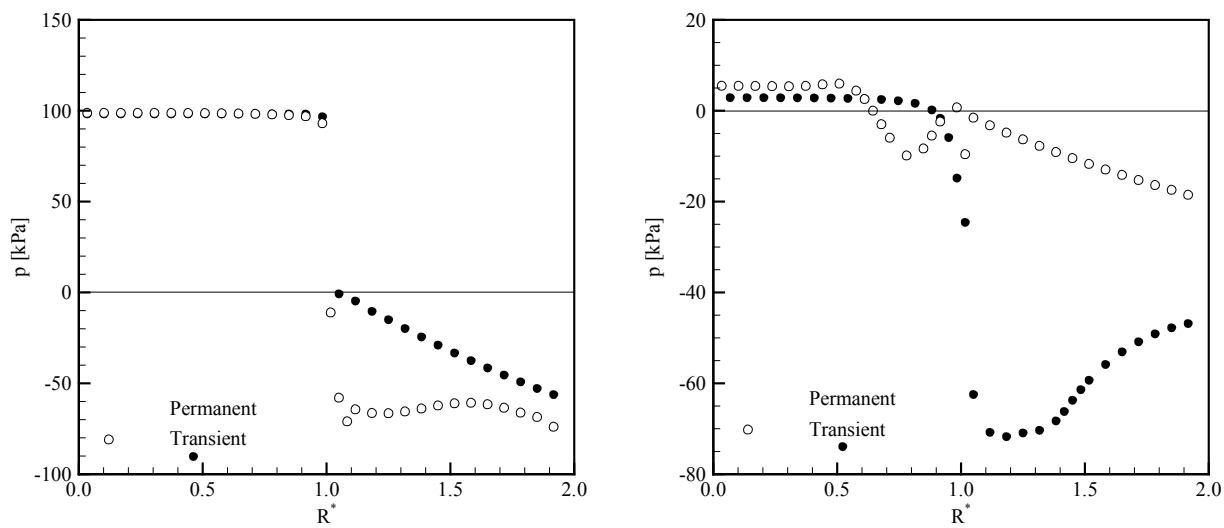
In the condition A, the transient flow is being accelerated and this is the reason for the lower pressure levels in the diffuser region ($1.0 \leq R^* \leq 1.95$). For the condition B, there is a negative Δp_v and, therefore, a backflow is predicted with the steady state formulation. However, due to 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. 7b, 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 inertial effect is stronger for the greater diameter ratio (D/d = 1.95), because longer diffuser regions are usually connected to negative force F_v even for the steady state flow condition. Since flow acceleration always decrease pressure, a negative F_v is more likely to happen in the case of D/d = 1.95. Naturally, such a phenomenon is also very dependent of valve displacement s_v.



(a) (b)

Figure 7: Predictions for flow induced force on the valve: (a) D/d = 1.50; (b) D/d = 1.95.



(a) (b)

Figure 8: 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.

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

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

4. CONCLUSIONS

This paper presented a theoretical analysis of flow inertial effects in a simplified geometry of compressor discharge valve. A numerical model based on the finite volume method was developed, taking into account the 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 mass flow rate and flow induced force in the opening and closing stages of the valve displacement. This study is an initial step towards the understanding of flow inertial effects in compressor valves, aimed at developing more general correlations of effective flow and force areas for transient flow conditions.

REFERENCES

Fluent 6.3 User's Guide, 2006, Ansys Inc.

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.

Kim, H., Ahn, J. and Kim, D., 2008, Fluid Structure Interaction and Impact Analyses of Reciprocating Compressor Discharge Valves, *Proc. Int. Compressor Engineering Conference at Purdue*, West Lafayette, Paper 1112.

Kim, J., Wang, S., Park, S., Ryu, K. and La, J., 2006, Valve Dynamic Analysis of a Reciprocating Compressor, *Proc. Int. Compressor Engineering Conference at Purdue*, West Lafayette, Paper C107.

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.

Pereira, E.L.L., Deschamps, C.J. and Ribas Jr, F.A., 2008, Performance Analysis of Reciprocating Compressors Through Computacional Fluid Dynamics. *J. Process Mechanical Engineering*, vol. 222, p. 183-192.

Pérez-Segarra CD, Escanes F and Oliva A, 1994, Numerical Study of the Thermal and Fluid-Dynamic Behaviour 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.

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.