Assessment of simplifying hypotheses adopted for valve leakage modeling

L G M De Luca, E Silva and C J Deschamps

Polo Research Labs for Emerging Technologies in Cooling and Thermophysics
Federal University of Santa Catarina, 88040-900, Florianopolis, SC, Brazil

deschamps@polo.ufsc.br

Abstract. The reed-type valves employed in refrigeration compressors must provide adequate sealing when closed to avoid leakage of gas between the compression chamber and the suction and discharge chambers. Recent studies show that valve leakage can considerably affect the performance of the small reciprocating compressors used for domestic refrigeration. The present paper reports an investigation on the adequacy of simplifying the hypothesis adopted in the simulation models of valve leakage. The results indicate that the transient effects related to both the valve deflection and fluid flow are negligible. Also, the ideal gas formulation was found suitable in some operating conditions found in domestic refrigeration. On the other hand, leakage was found to be overpredicted by almost 20% when the reed valve geometry was simplified to a circular plate in order to reduce the computational processing cost.

1. Introduction

The valve system is one of the main components of a reciprocating compressor, being responsible for controlling the suction and discharge processes of the refrigerant in the compression cycle. The compressors employed for domestic refrigeration adopt reed-type valves, which open and close depending on the pressure difference between the cylinder and the suction/discharge chamber that is established by the piston motion. When closed, these valves should provide complete sealing to avoid gas leakage, but geometric imperfections give rise to gaps in the region of contact between the reed and the seat. The pressure difference between the compression chamber and the suction and discharge chambers is the driving force of leakage and can also change the gap geometry due to the bending of the reed into the orifice.

Some studies available in the literature have shown that valve leakage can considerably reduce the compressor overall performance [1-4]. Silva and Deschamps [5] predicted a reduction of 2.7% and 4.4% in the volumetric and isentropic efficiencies, respectively, for a small compressor analyzed when a 1 μm gap was considered between both valve reeds and their seats. The authors developed a numerical model to estimate valve leakage in reciprocating compressors employed for domestic refrigeration, considering quasi-steady, adiabatic, compressible laminar flow of ideal gas. The reed valve geometry was simplified to the geometry of a circular plate. Rarefaction effects were also considered and the results indicated that under some conditions during the compression cycle the flow of gas in the gap occurs in the slip flow regime. The authors also observed that the valve deflection tends to increase leakage. Santos et al. [6] measured leakage in valves of low-capacity compressors before and after being subjected to wear.
The results indicate that the edge gap at the border of the valve orifice varies between 0.13 and 0.94 μm before wear and between 0.11 and 0.47 μm after wear.

The present study aims to investigate the influence of the following aspects on valve leakage: (i) inertia force associated with the fluid flow and reed valve bending; (ii) gas equation of state; and (iii) reed valve geometry.

2. Problem description

2.1. Fluid flow

The suction and discharge reed valves analyzed in the present study possess circular seats. The simulation model of valve leakage requires the time dependent pressure and temperature in the compression chamber as boundary conditions. These boundary conditions are determined by solving the compression cycle following isentropic expansion and compression processes, and assuming the suction and discharge processes at constant pressure. Therefore, there is no need to solve the valve dynamics.

The pressures in the suction and discharge chambers, $p_{sc}$ and $p_{dc}$, are also necessary and assumed equal to the evaporating and condensing pressures, $p_{evap}$ and $p_{cond}$, respectively. The temperature of the gas in the suction chamber $T_{sc}$ is assumed equal to the temperature at the suction line, while the temperature at the discharge chamber $T_{dc}$ corresponds to the temperature at the end of the isentropic compression process. The analysis is performed for a compressor operating with R600a under the following conditions: $T_{evap} = -23.3 \, ^\circ C$, $p_{evap} = 0.629 \, \text{bar}$, $T_{cond} = 54.4 \, ^\circ C$, $p_{cond} = 7.620 \, \text{bar}$, $T_{sc} = 54.1 \, ^\circ C$ and $T_{dc} = 129.8 \, ^\circ C$.

The flow solution domain corresponds to the region between the internal and external border of the seat, as indicated in the cross-section shown in Fig. 1. The radiuses of the internal and external border of the discharge valve seat considered herein had 2.25 and 3.05 mm, and the corresponding radiuses of the suction valve seat were 3.40 and 3.95 mm.

As shown in Fig. 1, the dimension of the gap varies with the radial position. At the flow outlet this parameter assumes the minimum value, referred to as the edge gap, $\delta_e$ [7]. The magnitude of the gap at the flow inlet, $w$, depends on the reed bending and varies during the compression cycle. When the reed valve is represented by a circular plate, the valve deformation and the flow are considered axisymmetric. However, this condition does not hold when the actual geometry is considered in the simulation, and a longitudinal symmetry condition is used instead.

The simulation model assumes the hypothesis of compressible laminar fluid flow. However, slip boundary conditions are applied as boundary conditions at the walls due to the reduced size of the gap. Moreover, the isentropic flow is assumed to predict the flow properties at the solution domain inlet, with the corresponding thermodynamic state being characterized by stagnation properties. Finally, the field forces and viscous dissipation were neglected.

![Figure 1. Closed valve and gap geometry between the reed and seat.](image-url)
The solution domain is illustrated in Fig. 2. The instantaneous pressure and temperature in the chamber from which the gas leaks are prescribed as boundary conditions at the inlet (a). At the outlet (b), the boundary condition is the pressure in the chamber where the gas flows to. Face (c) is a wall condition that represents the reed surface whose position is determined as a function of the reed valve bending. Face (d) represents the wall boundary condition associated with the seat surface. Finally, faces (e) and (f) represent the symmetry flow conditions.

The flow simulation is carried out only when the valve is closed. The initial condition for the solution domain of the suction valve is set as uniform and equal to the conditions in the suction chamber. In the case of the discharge valve, the initial condition is prescribed as that in the discharge chamber. The velocity field was assumed null for all cases.

During the flow simulation the thermodynamic and transport properties were obtained from REFPROP [8] when the ideal gas assumption was not used. Two parameters are necessary to define the slip boundary condition at the walls: the tangential momentum accommodation coefficient, determined experimentally by Silva et al. [9] for R600a; and the characteristic length associated to the Lennard-Jones potential, which was estimated from the gas viscosity assuming a hard sphere intermolecular interaction model [10].

2.2. Reed valve bending
The valve material is assumed to behave as a linear isotropic elastic solid with the following physical properties: $v = 0.3$, $\rho_s = 7860 \text{ kg/m}^3$ and $E = 210 \text{ GPa}$ ($v$ is the Poisson’s ratio, $\rho_s$ is the density and $E$ is the modulus of elasticity). The following assumptions were adopted in order to simplify the valve bending model: (i) contact points occur only between the reed and seat and at the reed clamer; (ii) the valve is assumed to be in contact only with the internal border line of the seat. Non-linearities originating from the contact between the reed and the seat are neglected; (iii) it is assumed that the reed surface covering the orifice is subjected to a uniform pressure load brought about by the pressure difference between the compression chamber and the suction or discharge chamber; (iv) the valve is at rest when closed (null velocity) and there is no initial deflection; (v) deformation generated by the impact of the valve against the seat is neglected.

Figure 3 illustrates the structural domain and the boundary conditions imposed. The reed is fixed at the clamer region and simply supported along the contact line with the internal border of the seat, with the pressure load acting only in the area covering the orifice. Alternatively, the valve was described as a circular plate with the diameter equal to the external diameter of the seat. In this case, the reed is simply supported along the contact line with the internal border of the seat.

![Figure 2. Flow domain and boundary conditions.](image1)

![Figure 3. Structural domain and boundary conditions.](image2)
3. Numerical solution

3.1. Fluid flow
The software ANSYS Fluent was used to solve the fluid flow through the valve gap. The mesh generated to discretize the fluid domain consisted of hexahedral volumes. For the case in which the reed was approximated by a circular plate, the fluid flow domain was reduced to an eighth of its original size due to the axisymmetric condition. On the other hand, for the case in which the original valve geometry was considered, only longitudinal symmetry was adopted. Since the domain changes according to the reed deflection during the compression cycle, moving mesh methods are required. The Smoothing method was adopted because of its reduced computational cost and adequacy for hexahedral meshes. In this sense, the mesh nodes at the wall follow the wall movement while those in the interior of the mesh move according to a convenient algorithm [11]. In the present study, the Linearly Elastic Solid Based Smoothing Method was employed, which transfers the node displacements to the interior of the mesh by assuming that the mesh deforms as a linearly elastic material.

A second-order upwind scheme was adopted to evaluate convective terms while the Least Squares Cell-Based scheme was used for diffusive terms. The SIMPLE algorithm was chosen for the pressure-velocity coupling and the convergence criteria were set to $10^{-4}$ for the mass and momentum equations and $10^{-6}$ for the energy equation. In addition to that, the variation of the mass flow rate at the inlet and outlet was verified after each 10 iterations of the solution procedure for all timesteps. The convergence was achieved when both parameters varied less than $10^{-4}$.

3.2. Reed valve bending
The solution of the structural problem was obtained using the finite element method available in the software ANSYS Mechanical. The mesh was generated assuming a constant number of elements along the width of the reed and the maximum aspect ratio was 1:2. The Solid-Shell element (SOLSH190) [11] was used due to its adequacy for thin structures. The Newmark implicit method was used for time integration and the linear system of equations was solved with the Full Solution Method. The software uses the Newton-Raphson method for an iterative solution procedure [11].

The timestep was adjusted automatically between three different levels by the software during the solution procedure: initial, minimum and maximum. The initial timestep $\Delta t_{in}$ was defined based on the largest natural frequency of interest $f_n$ (obtained from modal analysis) according to: $\Delta t_{in} = (1/20) * f_n$. The minimum timestep was used to prevent indefinite solutions to the equations, being prescribed as $\Delta t_{in}/100$. The maximum timestep was equal to the timestep used in the fluid flow simulation. In order to account for the internal damping of the material, the Rayleigh damping model was adopted with a damping coefficient of 0.1.

3.3. Fluid-structure interaction
The partitioned approach was adopted to account for the fluid-structure interaction. This approach provides solutions to the fluid and structure problems by distinct numerical methods using a sequential procedure, as well as the use of non-conformal meshes in which the nodal positions of both meshes do not need to coincide. Besides, a one-way approach was assumed, in which the structural solution is obtained first and used to define the position of the boundary conditions in the fluid flow domain. The transfer of data between ANSYS Mechanical and ANSYS Fluent was carried out with the ANSYS System Coupling.

The solution procedure consists in providing the values of pressure and temperature in the compression chamber at each timestep. Then, the ANSYS Mechanical solves the structural problem and the ANSYS System Coupling transfers the resulting reed deflection to ANSYS Fluent, allowing the fluid solution domain and mesh to be updated. Then, the iterative solution procedure follows until convergence.
4. Results

The errors associated with the temporal and spatial discretization were evaluated through the Richardson extrapolation method based on the grid convergence index (GCI). The final mesh in the solid domain consisted of an unstructured mesh with 5336 elements for the case when the complete reed geometry is considered and with 2600 elements when the reed is simplified as a circular plate. The fluid domain adopted a structured mesh with 16 x 90 x 180 volumes in the axial, radial and circumferential directions, respectively. The processing times of the quasi-steady flow simulations for the simplified and complete reed geometries were approximately 16 h and 21 h, respectively. The simulations were carried out in a computer with an Intel Core i7-7700K CPU 4.20 GHz processor.

In order to validate the structural model, the results of the reed deflection under a sinusoidal load with a frequency of 120 Hz were compared with the analytical solution presented by Weiner [12]. Figure 4a shows the displacement of the point of the discharge reed located in the center of the orifice, $w_x$, during the period $\tau$ in which the valve is closed. Figure 4b shows the discharge valve displacement along the radial direction, $w_r$, for three points in time. As can be noted, there is good agreement between the numerical prediction and analytical solution.

The fluid flow model was validated by comparing its predictions with the results found in the literature for the mass flow rate of the radial flow between parallel and inclined disks, which is similar to the geometry adopted in the present study. Figure 5 shows the comparison of the mass flow rate predicted by the fluid flow model with predictions and experimental data provided by Zuk [13] for the radial flow between two parallel disks under different pressures ratios. On the other hand, Table 1 shows the comparison between the results obtained in the present study and the predictions provided by Zuk and Smith [14] for inclined disks. The greatest difference was observed for the parallel disk condition, being approximately 10%. This agreement was considered satisfactory and the model considered adequate for predicting valve leakage.

After being validated, the model was used to verify three hypotheses commonly adopted in predictions of leakage: (i) quasi-steady condition for the valve deflection and fluid flow; (ii) ideal gas; (iii) reed valve as disk plate. The quasi-steady assumption for the valve deflection was tested by observing the valve deflection at the external border of the seat, $w_r$, during the period in which the valves are closed. Figure 6 shows the solutions obtained with the model adopting the transient and quasi-steady formulations. The horizontal axis shows the compressor shaft angle, $\theta$, during the period the valve is closed, normalized by the shaft angle immediately before the valve opening, $\theta_{\text{max}}$. The greatest deviations observed between the results of both formulations were 1% for the discharge valve and 0.4% for the suction valve. These results show that the inertial force resulting from the valve acceleration due to the transient loading being negligible and do not having a significant influence on valve deflection.

![Figure 4. Deformation of the discharge reed valve: (a) point located in the center of the orifice; (b) along the radial direction at different points in time.](image-url)
Figure 5. Comparison of mass flow rate for the parallel disks configuration.

Table 1. Comparison of mass flow rate for the inclined disks configuration.

| Heights [mm] | \( \delta_\alpha \) | \( \dot{m} \) [kg/h] | Zuk and Smith [14] | Present study | \( \Delta \dot{m} \) [%] |
|--------------|----------------|----------------|------------------|---------------|----------------|
| 8.255        | 6.985          | 2830.3        | 2784.6           | 1.62          |
| 10.795       | 9.525          | 6096.2        | 5805.0           | 4.78          |
| 13.335       | 12.065         | 10314.7       | 9568.4           | 7.23          |

Figure 6. Predicted valve deformation considering transient and quasi-steady approaches for an operating frequency \( f = 120 \) Hz: (a) suction valve; (b) discharge valve.

The inertia effect on the flow was verified through the difference between the mass flow rate at the inlet and outlet of the solution domain, \( \Delta \dot{m} = \dot{m}_{in} - \dot{m}_{out} \), during the compression cycle. Figure 7 shows the results of \( \Delta \dot{m} \) for the discharge valve normalized by the average mass flow rate during the compression cycle, \( \dot{m}_{in} \), for three operating frequencies (30, 60 and 120 Hz). It is possible to note that the greatest \( \Delta \dot{m} \) occur when the pressure difference \( \Delta p \) shows steep variation, which is more pronounced at higher operating frequencies. It is worth mentioning that \( \Delta \dot{m} \) is positive when \( \Delta p \) increases and
negative when $\Delta p$ decreases. In fact, in addition to leakage, the gap also acts as a reservoir, receiving, accumulating and delivering an amount of mass to the compression chamber periodically.

The overall effect of inertia on the flow was also analyzed in terms of the mean mass flow rate at the inlet and outlet of gap for two intervals of time during the compression process and three operating frequencies (Fig. 8). These time intervals are referenced herein by their corresponding shaft angles. The positive values indicate that the mass flow rate enters the gap and vice-versa for negative values (hatched bars). Considering $\theta/\theta_{\text{max}} = 65\%$, Fig. 8 shows that the mean mass flow rate at the inlet increases with the frequency, while the mean mass flow rate leaving the domain is practically insensitive. Close to the valve opening ($\theta/\theta_{\text{max}} = 95\%$) these variations are no longer observed even at the inlet. In fact, the mean mass flow rate predicted with the quasi-steady formulation ($Q_{\text{S}}$) is quite close to those obtained considering the transient flow conditions. Therefore, although present in the flow, we conclude that inertia effects are not important for the prediction of valve leakage.

Simulations were also conducted to evaluate the adequacy of the ideal gas assumption. Figure 9 shows the mass flow rate associated with the leakage of gas in the suction and discharge valves, considering different values of the edge gap. The results show that the maximum difference between the results obtained with the ideal gas and real gas formulations is approximately 5% for the discharge valve when $\delta_0 = 0.8 \mu m$. Given the small difference between the results of both formulations, and the higher computational cost (~ 20%) of simulations with the real gas formulation, the ideal gas hypothesis is a suitable approach for leakage predictions.

Finally, the influence of the reed valve geometry on the predictions was considered. The results of the mean deflection, $w_m$, (calculated along the perimeter of the inlet flow region) as a function of the pressure difference are shown in Fig. 10, considering the actual geometry (NS) and the simplified geometry of the circular plate (S). As can be seen, the mean deflection is always smaller for the actual geometry regardless of the pressure difference, resulting in a smaller leakage area and, consequently, less leakage than when adopting the simplified geometry case. The net leakage through the suction and discharge valves is 0.025 kg/h when solving for the actual geometry and 0.031 kg/h if the simplified geometry is used instead, a difference of 19%. However, the computational processing cost of the simulations with the simplified geometry is approximately 35% lower than those of the actual geometry.

![Figure 7](image_url)  
**Figure 7.** Effect of fluid inertia: (a) Difference between the mass flows rate at the inlet and outlet; (b) pressure difference acting on the discharge valve.
\textbf{Figure 8.} Mean mass flow rate at the inlet and outlet of the fluid domain for different operating frequencies ($f_1 = 30$ Hz, $f_2 = 60$ Hz and $f_3 = 120$ Hz): (a) suction valve; (b) discharge valve.

\textbf{Figure 9.} Leakage in a single compression cycle as a function of the equation of state and dimension of the edge gap: (a) discharge valve; (b) suction valve.

\textbf{Figure 10.} Mean reed deflection along the perimeter of the inlet region as a function of the pressure difference: (a) discharge valve; (b) suction valve.
Conclusions
A simulation model was developed for the numerical analysis of leakage in the reed-type valve of reciprocating compressors adopted in household refrigerators. The results of valve deflection and mass flow rate provided by the model were compared to the results available in the literature in order to validate the model. Then, the model was used to verify the adequacy of simplifying assumptions normally adopted in the modeling of valve leakage. The results showed that the transient effects related to the valve deflection and fluid flow can be neglected for operating frequencies smaller than 120 Hz. Also, it was found that the ideal gas formulation tends to underestimate leakage. However, given the small differences observed in relation to the real gas formulation and the reduction in the computational time, the ideal gas formulation can be considered suitable for predictions of valve leakage. Finally, the analysis showed that the actual valve geometry is important for accurate results despite being associated with computational processing cost approximately 35% higher.

Acknowledgements
The present study was developed as part of a joint technical-scientific program between the Federal University of Santa Catarina and EMBRACO. The authors are also grateful for the financial support provided by CAPES and EMBRAPII Unit POLO/UFSC.

References
[1] Machu E 1990 How leakages in valves can influence the volumetric and isentropic efficiencies of reciprocating compressors Int. Compressor Eng. Conf. at Purdue (West Lafayette, IN, USA)
[2] Fujiwara M and Kazama T 1998 Performance improvement of a reciprocating air microcompressor Int. Compressor Eng. Conf. at Purdue (West Lafayette, IN, USA)
[3] Elhaj M, Gu F, Ball A D, Albarbar A Al-Qattan and M Naid 2008 A Numerical simulation and experimental study of a two-stage reciprocating compressor for condition monitoring. Mech. Syst. and Sig. Proces. 22 374–389
[4] Breuker M and Braun J 1998 Common faults and their impacts for rooftop air conditioners. HVAC&R Research. 4 303–318
[5] Silva L R and Deschamps C J 2015 Modeling of gas leakage through compressor valves Int. J. Refrigeration 53 195–205
[6] Santos F L D, Silva E and Deschamps C J 2018 Influence of wear on leakage through reed-type valves of small reciprocating compressors Int. Compressor Eng. Conf. at Purdue (West Lafayette, IN, USA)
[7] Rezende G C, Silva E, Deschamps C J 2018 Edge gap as a geometric parameter to characterize the sealing performance of compressor valves Int. J. Refrigeration 90 257–263
[8] NIST 2007 Refprop – Reference fluid thermodynamic and transport properties (NIST)
[9] Silva E, Rojas-Cardenas M and Deschamps C J 2016 Experimental analysis of velocity slip at the wall for gas flows of nitrogen, R134a, and R600a through a metallic microtube Int. J. Refrigeration 66 121–132
[10] Hirschfelder J O, Curtiss C F and Bird R B 1954 Molecular theory of gases and liquids (John Wiley & Sons, New York) p 1249
[11] ANSYS Inc 2011 User’s guide v.15.0.0 (ANSYS)
[12] Weiner R S 1965 Forced axisymmetric motions of circular elastic plates J of Appl. Mech. 32 893
[13] Zuk J Fluid 1972 Mechanics of Noncontacting gas film seals Ph.D. Thesis Case Western Reserve Univers.
[14] Zuk J and Smith P J 1974 Computer program for quasi-one-dimensional compressible flow with area change and friction NASA