2002

Numerical Simulation Of The Dynamics Of Reed Type Valves

F. F. S. Matos
Federal University of Santa Catarina

A. T. Prata
Federal University of Santa Catarina

C. J. Deschamps
Federal University of Santa Catarina

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

https://docs.lib.purdue.edu/icec/1563

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 SIMULATION OF THE DYNAMICS OF REED TYPE VALVES

F. F. S. Matos, A. T. Prata, C. J. Deschamps,
Federal University of Santa Catarina,
Department of Mechanical Engineering,
88040-900, Florianópolis, SC, Brazil.

ABSTRACT

The present work considers the development of a computational model to simulate the dynamics of reed type valves of reciprocating compressors. Because the valve dynamics and the time dependent flow field through the valve are coupled, they are solved simultaneously. A one-degree of freedom model is adopted for the valve motion and the finite volume methodology is employed to obtain the flow field through the valve, which is assumed incompressible, turbulent and axisymmetric. In view of the valve motion a numerical technique has been developed to take into account the variation with time of the computation domain. Through a coordinate transformation a moving coordinate system is obtained so as to expand and contract according to the valve displacement. Results for the turbulent flow (streamlines and turbulence intensity) and for the reed motion (resultant force, valve lift and pressure distribution on the reed surface) are obtained for a prescribed periodic mass flow rate at the valve entrance.

INTRODUCTION

Reed type valves are essential components in a number of reciprocating compressors, particularly the hermetic ones employed in refrigeration. These valves open and close depending on the pressure difference established by the piston reciprocating motion between the cylinder and the suction and discharge chambers. In designing the valve system in reciprocating compressors four main features related to the valve performance are sought: fast response, large mass flow rate, low pressure drop when opened, and good backflow blockage when closed. To satisfy the performance requirements necessary for a competitive compressor, a detailed understanding of the fluid flow through the valve as well as the dynamics of the valve is necessary.

The basic features related to the flow across the valve and the reed dynamics can be explained referring to Fig. 1, where a simplified discharge valve is represented by a circular disk. In this valve the unbalanced pressure distribution acting on the reed separates it from the valve seat generating a gap through which the flow is initiated. As illustrated in the figure, the fluid enters the valve through the orifice flowing axially and then, due to the presence of the reed, it is forced to flow radially for the rest of the valve passage. The pressure difference between the orifice entrance and the discharge chamber together with the valve lift govern the fluid flow throughout the valve. On the other hand, the flow dictates the pressure distribution on the reed and, consequently, the resultant force that will govern the valve dynamics and its displacement from the seat.

Valve performance is a intricate problem where fluid mechanics and solid dynamics play a definite role. For some recent investigations on valve modeling reference is made to Cyklis (1994), Machu (1994), Deschamps et al. (1996), Lopes and Prata (1997), Ezzat Khalifa and Liu (1998), Salinas-Casanova et al. (1999), Chung et al. (2000), Matos et al. (2000) and Ottitsch (2000). Most works in the literature related to reed valves either model the valve dynamics in detail but pay little attention to the description of the flow field, or focus on the fluid mechanics without considering the coupling between valve motion and pressure distribution on the reed. The work of Lopes and Prata (1997) considered for the first time a numerical methodology to explore the interaction between valve dynamics and fluid flow. In that work, a periodic flow rate was prescribed at the entrance of the valve orifice and a numerical solution was obtained for the laminar flow field. From the pressure distribution on the valve reed the resultant force was determined and a one-dimensional dynamic model was employed to solve for the reed acceleration, velocity and displacement. The work to be described herein prescribes a periodic flow rate through the valve in the same manner as Lopes and Prata (1997) but considers the more realistic condition of turbulent flow through the valve.
The dynamics of the reed can be modeled as

\[ F - F_o = m\ddot{s} + C\dot{s} + Ks \]  

(1)

In the present work the reed is assumed to move parallel to the valve seat, with \( s \) being the instantaneous gap between seat and reed, whereas \( \dot{s} \) and \( \ddot{s} \) are the velocity and the acceleration of the reed, respectively. The valve stiffness and damping coefficient, \( K \) and \( C \), respectively, as well as the valve mass, \( m \), are determined experimentally. \( F_o \) is a pre-load force that has not been considered in the present work.

To solve equation (1) for \( s \), the resultant force \( F \) acting on the reed is obtained from

\[ F = \int_0^{2\pi} \int_0^{D/2} p \, rdr \, d\theta \]  

(2)

where \( p \) is the instantaneous local flow-induced pressure distribution on the reed determined from the solution of the flow field.

The axisymmetric geometry shown in Fig. 1 was employed to determine the pressure distribution on the reed, with the flow being assumed to be incompressible, turbulent and isothermal. The time/ensemble averaged Navier-Stokes equations were closed using the concept of 'turbulent' or 'eddy' viscosity \( \mu_t \). Under the effect of no body force the equations of motion are:

\[ \frac{1}{r} \frac{\partial}{\partial r} (r \rho v) + \frac{\partial}{\partial x} (\rho u) = 0 \]  

(3)

\[ \frac{\partial}{\partial t} (\rho u) + \frac{\partial}{\partial x} (\rho uu) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho vu) = -\frac{\partial}{\partial x} \left( p + \frac{2}{3} \rho k \right) + \frac{\partial}{\partial x} \left( \mu_t \frac{\partial u}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( \mu_t \frac{\partial u}{\partial r} \right) + \frac{\partial}{\partial x} \left( \mu_t \frac{\partial u}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( \mu_t \frac{\partial v}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( \mu_t \frac{\partial v}{\partial r} \right) \]  

(4)
\[
\frac{\partial}{\partial t}(\rho v) + \frac{\partial}{\partial x}(\rho uv) + \frac{1}{r} \frac{\partial}{\partial r}(\rho vv) = -\frac{\partial}{\partial r} \left( p + \frac{2}{3} \rho k \right) + \frac{\partial}{\partial x} \left( \mu_{\text{eff}} \frac{\partial v}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( \rho u_{\text{eff}} \frac{\partial v}{\partial r} \right) + \frac{\partial}{\partial x} \left( \mu_t \frac{\partial u}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( \rho u_t \frac{\partial v}{\partial r} \right) - \frac{\mu_{\text{eff}}}{r} \frac{v}{r} - \mu_t \frac{v}{r^2}
\]

(5)

where \( \mu_{\text{eff}} = \mu + \mu_t \) is the effective viscosity, which takes into account the molecular viscosity, \( \mu \), and the turbulence viscosity, \( \mu_t \). The latter is a measure of the turbulence transport mechanism in comparison with molecular diffusion and is evaluated in terms of the turbulence kinetic energy \( k \) and its rate of dissipation \( \varepsilon \). Following the RNG k-\( \varepsilon \) model of Orzag et al. (1993), the effective viscosity is evaluated as

\[
\mu_{\text{eff}} = \mu \left[ 1 + \left( C_u \frac{\rho k^{2}}{\varepsilon \mu} \right)^{1/2} \right]^2
\]

(6)

The kinetic energy \( k \) and the dissipation \( \varepsilon \) are obtained from their respective modeled transport equations. Such equations will not be shown here due to space limitations but can be found in Salinas-Casanova et al. (1999).

Equations (1) to (6), together with those related to the transport of \( k \) and \( \varepsilon \), completely describe the problem and seven unknowns are to be determined: \( s, F, u, v, p, k \) and \( \varepsilon \). The boundary conditions associated to each of these equations are discussed next.

At the solid walls all velocity components are zero with the exception of the reed surface. There, the radial component, \( v \), is zero but the axial velocity corresponds to the reed velocity \( \dot{s} \), which is obtained from equation (1). The symmetry conditions at the valve axis implies that \( v = \partial u / \partial r = 0 \). As for the outlet, the computational domain was extended beyond the valve exit (as indicated by the dashed line in Fig. 1) and there a pressure boundary condition was set to represent the discharge chamber. More details on this procedure can be found in Versteeg and Malalasekera (1995). For the velocity component parallel to the outlet boundaries the prescribed boundary condition was that of zero gradient.

Concerning turbulence quantities, no information is available for the turbulence kinetic energy at the inlet boundary. However, numerical tests indicated that when the level of the turbulence intensity \( I = (2k/3)^{1/2} / U_{in} \) is increased from 3% to 6% no significant change occurs in the predicted flow. Therefore, a value of 3% was used in the calculation of all results shown in this work. The dissipation rate at the entrance was estimated based on the assumption of equilibrium boundary layer, that is, \( \varepsilon = \lambda^{3/4} k^{3/2} / \ell_m \), where \( \ell_m = 0.07d/2 \) and \( \lambda = 0.09 \). At the solid boundaries, the turbulence kinetic energy \( k \) was set to zero and for the dissipation \( \varepsilon \), rather than prescribing a condition at the walls, its value was calculated in the control volume adjacent to the wall following a non-equilibrium wall-function.

**SOLUTION METHODOLOGY**

Following a practice described in details by Matos et al. (1999), a moving coordinate system was employed to solve the governing equations in the physical domain, which expands and contracts as the reed moves up and down, respectively. The main feature of this coordinate system is that it transforms the physical domain into a computational domain that remains unchanged regardless the reed motion.

The finite volume method was employed to integrate the partial differential equations governing the flow using a staggered arrangement for the control volumes for the axial and radial velocities. Interpolation of unknown quantities at the control volume faces were obtained using the QUICK scheme (Hayase et al., 1992) and a fully implicit discretization scheme was adopted for the unsteady terms. The equations were solved by means of a segregated approach and the coupling between pressure and velocity fields was handled through the SIMPLEC algorithm (Versteeg and Malalasekera, 1995). For the domain discretization, 37 x 32 (axial x radial) grid points were used in
the feeding orifice region and 33 x 80 grid points (axial x radial) were employed in the region between the reed and the valve seat. A more refined grid was not feasible due to the already extremely high computational cost required for the solution. However, the rather coarse grid employed in the computations was successful in capturing the main features of the flow field as will be discussed latter during the validation and presentation of the numerical results. For the time step, an interval of \(\frac{2\pi}{360\omega}\) was employed, with \(\omega = 2\pi f\) [rad/s]. The differential equation governing the valve motion, equation (1), was solved analytically considering the resultant force on the reed to be constant during each time step.

**RESULTS AND DISCUSSION**

The first step in the numerical analysis was to validate the computational code written for a moving coordinate system. The experimental data presented by Salinas-Casanova et al. (1999) for dimensionless pressure \(P^* = \frac{p}{(\rho u^2)/2}\) distribution on the reed, with a valve lift \(s/d = 0.05\) and a mass flow rate equivalent to \(Re = 25000\), is used to this extent. The Reynolds number \(Re\) is defined with reference to the mean velocity \(U_{in}\) at the entrance of the feeding orifice (i.e. \(Re = \frac{\rho U_{in} d}{\mu}\)).

Numerical results were generated for two flow conditions. In the first one the valve lift is kept constant and equal to \(s/d = 0.05\), whereas in the second the lift is set to vary harmonically as \(s/d(t) = 0.05 (1 + 0.2 \sin \omega t)\), where \(\omega = 2\pi f\). Because the main goal was to validate the methodology associated with the moving coordinate system, a low frequency (\(f = 0.1\)Hz) was adopted. Figure 2 shows a comparison between experimental data and numerical results corresponding to both constant and varying valve lifts. It is clear from the figure the good agreement between the results and this has yielded the required confidence on the present methodology.

The next step was to solve the reed dynamics coupled with the flow field through the valve. For all results to be presented here, the reed diameter \(D\), the orifice diameter \(d\) and the orifice length \(e\) are equal to 9.0 mm, 6.0 mm and 1.6 mm, respectively. The working fluid was taken to be air with \(\mu = 1.8 \times 10^{-5}\) Pa.s and \(\rho = 1.2\) kg/m\(^3\). At the outlet a pressure boundary condition (1.0132 \(\times 10^5\) Pa) was prescribed using the procedure already explained. The valve parameters used in equation (1) are \(K = 200\) N/m, \(C = 0.5\) N.s/m, \(m = 3.2\) g and \(F_0 = 0\). In line with actual compressor discharge systems, a lift limiter was included in the computational and positioned at \(s/d = 0.166\) so as to limit the opening of the valve.

![Figure 2: Comparison between numerical results and experimental data for pressure distribution on the reed surface, Re=25000, s/d=0.05, D/d=1.66.](image-url)
The flow rate at the feeding orifice is prescribed according to a periodic variation for the axial velocity component:

\[ u(t) = \frac{\mu \Re}{\rho d} \left[ 1 + 0.9 \sin(\omega t) \right] \]

where \( \omega = 2 \pi f \), \( f = 60 \) Hz and \( \Re = 25,000 \).

Results for valve lift and resultant force on the reed are presented in Fig. 3 as a function of time. For reference, in that figure is also shown the instantaneous Reynolds number prescribed at the valve feeding orifice as given by equation (7). The figure shows that large mass flow rates bring about large values of resultant forces and valve lifts, as would be expected. Another aspect that can also be seen is the limiter effect on the valve dynamics and the exact moment when the reed reaches its maximum lift. The present model can give an estimate for the velocity with which the reed hits the limiter.

A rather unexpected flow feature in the detail of Fig. 3 is the force decrease observed just before the reed reaches the limiter and the local minimum then originated. This can be explained with reference to dimensionless streamlines \( \Psi^* (= \Psi/\bar{m}) \) in Fig. 4 and pressure distributions on the reed in Fig. 5, for \( 4 \pi/180, 20 \pi/180, 25 \pi/180, 39 \pi/180 \) and \( 45 \pi/180 \). Such time positions are identified in Fig. 3 by letters \( a, b, c, d \) and \( e \), respectively. As can be seen in Fig. 5, the presence of the reed creates a plateau on the central part of the pressure distribution (r/d < 0.5), whereas the sharp pressure drop at the radial position r/d = 0.5 is associated with both the change in the flow direction and the reduction of the flow passage area. The latter is particularly strong in the range of disk displacement and Reynolds number considered here since a separated flow region is always present on the valve seat. The reduction in the passage area originated by this separation results in a pressure drop due to the increase of local velocity. After reaching a condition of minimum, the pressure level on the reed is increased as the flow progresses towards the valve exit. In all lift situations the pressure at the exit of the valve is seen to be higher than the pressure in the discharge chamber.

The dimension \( h \) indicated in Fig. 4d is a measure of the minimum flow passage area related to the recirculating region. For \( 4 \pi/180 \) (Fig. 4a) this minimum occurs at r/d = 0.635, where \( h = 0.30 \) mm, and originates the minimum in the pressure distribution shown in Fig. 5. As the reed moves from locations \( a \) to \( b \) in the cycle, the recirculating flow region becomes larger, but the height \( h \) is also increased (h = 0.38 mm) and the radial position for the minimum passage area is now r/d = 0.678. This originates a smaller pressure drop on the reed surface and an increase in the resultant force.

As can be seen in Fig. 4c for \( 25 \pi/180 \), as the valve opens the height \( h \) is increased even further (h = 0.41 mm) despite the presence of a larger recirculating flow region. This acts to lessen the negative pressure region on the reed surface but because the pressure level in the stagnation region (r/d < 0.5) becomes smaller (Fig. 5), the overall result is a reduction in the resultant force (Fig. 3). A similar reduction in the force takes place for \( 39 \pi/180 \), point \( d \) in Fig. 3, at the exact moment when the reed hits the limiter and \( h = 0.56 \) mm.

Finally, for \( 45 \pi/180 \) (point \( e \) in Fig. 3), the reed is totally open for some time and a flow configuration identical to that for \( 39 \pi/180 \) is verified, with \( h = 0.55 \) mm. Despite of that, the mass flow rate (represented by the Reynolds number) is still increasing and therefore the pressure level in the stagnation region becomes larger (Fig. 5) and the same occurring with the resultant force.

Results for turbulence intensity \( I (= [2k/3]^{1/2} U_{in}) \) are presented in Fig. 6 for those time positions \( \alpha t \) already discussed. With reference to Figs. 4 and 6, one can see that a common feature for all valve lifts is the high level of turbulence generated by the recirculating flow regions, as should be expected. On the other hand, the decrease in the turbulence intensity verified with the increase of the valve lift is a result of smaller strain rates caused by larger flow passage areas.
Figure 3: Valve lift and resultant force on the reed in response to a periodic mass flow rate.

- Reynolds, • Force, s/d.

(a) $\omega t = \frac{4\pi}{180}; s/d = 0.076$

(b) $\omega t = \frac{20\pi}{180}; s/d = 0.103$

(c) $\omega t = \frac{25\pi}{180}; s/d = 0.117$

(d) $\omega t = \frac{39\pi}{180}; s/d = 0.166$

(e) $\omega t = \frac{45\pi}{180}; s/d = 0.166$

Figure 4: Dimensionless streamlines $\Psi^* (= \Psi/\dot{m})$. 
Figure 5: Pressure distribution on the reed as the maximum valve lift is reached.

- \( \omega t = 4 \pi / 180; s/d = 0.076 \)
- \( \omega t = 20 \pi / 180; s/d = 0.103 \)
- \( \omega t = 25 \pi / 180; s/d = 0.117 \)
- \( \omega t = 39 \pi / 180; s/d = 0.166 \)
- \( \omega t = 45 \pi / 180; s/d = 0.166 \)

Figure 6. Turbulence intensity \( I = \left( \frac{2k}{3} \right)^{1/2} / U_{in} \).
The present work considered the modeling of the dynamics of automatic valves commonly found in reciprocating compressors. The complex interaction between reed dynamics and gas flow through the valve was incorporated into the physical and the mathematical models. Because the main focus here is the development of a numerical methodology to solve the reed dynamics, at this stage a simple periodic flow rate condition was set at the valve entrance. The present methodology is capable to capture some features found in the dynamics of actual valves, such as the impact against the valve limiter for conditions of large mass flow rates. An important step in the modeling is the inclusion of compressibility effects on the flow. Moreover, the mass flow rate through the valve should be evaluated as a function of the pressure drop between the cylinder and the discharge chamber. Such improvements in the model will be considered in future works.

ACKNOWLEDGEMENTS

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

REFERENCES


