2000

Numerical Simulation of a Reciprocating Compressor For Household Refrigerators

F. Bassi
Universita degli Studi de Ancona

L. Pelagalli
Universita degli Studi de Ancona

S. Rebay
Universita degli Studi di Brescia

A. Betto
Electrolux Compressors Co.

M. Orefice
Electrolux Compressors Co.

See next page for additional authors

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

https://docs.lib.purdue.edu/icec/1372

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
Authors
F. Bassi, L. Pelagalli, S. Rebay, A. Betto, M. Orefice, and A. Pinto

This article is available at Purdue e-Pubs: https://docs.lib.purdue.edu/icec/1372
NUMERICAL SIMULATION OF A RECIPROCATING COMPRESSOR FOR HOUSEHOLD REFRIGERATORS

F. Bassi, L. Pelagalli
Dip. di Energetica, Università degli Studi di Ancona, via Brecce Bianche, 60100 Ancona, Italy

S. Rebay
Dip. di Ing. Meccanica, Università degli Studi di Brescia, via Branze 38, 25123 Brescia, Italy

A. Betto, M. Orefice, A. Pinto
Electrolux Compressors, via Consorziale 13, 33170 Pordenone, Italy

ABSTRACT

This paper presents a method for the numerical simulation of reciprocating compressors for household refrigeration. The compressor is regarded as a set of volumes and one-dimensional, variable-area ducts connected together. The model describes all relevant features of the compressor, including suction and discharge mufflers and automatic valves. The equations governing the unsteady flow of a real gas in the ducts are numerically solved with a high order accurate discontinuous Galerkin method. The model accounts for the interaction between the gas flow and the automatic valves motion by solving the governing equations in a strictly coupled fashion. The method has been validated by comparing the numerical results obtained in the simulation of a compressor for household refrigeration with experimental data. The results show the potentialities offered by the proposed approach.

INTRODUCTION

In this paper we present a new numerical procedure for the simulation of the hermetic reciprocating compressor used in the household appliances. The method can simulate accurately the unsteady flow of a real gas through the compressor (including the flow in the cylinder leakage) and can handle the interaction between the fluid flow and the automatic valves motion. Moreover, the effects of friction and heat exchange between gas and solid walls can be taken into account by means of appropriate correlations. The space discretization of the flow equations is based on a recently developed Discontinuous Galerkin (DG) method [1] that has proved to be very accurate and numerically robust.

For the application presented in this paper we have used a one-dimensional (in space) version of the code. However, work has been planned to employ multidimensional spatial discretizations to compute the fluid flow in those components which greatly influence the fluid dynamic behavior of the whole compressor.

In the following we present and discuss the governing equations of the flow in the compressor components; then we briefly outline the DG solution procedure; finally we present the numerical results of an unsteady simulation, that takes into account fluid flow, heat exchanges, fluid structure interaction with the automatic valves, cylinder leakage, and compare the results with available experimental measurements.

GOVERNING EQUATIONS

This section briefly describes 1) the equations governing the gas flow through ducts and volumes, 2) the equations governing the heat flux among the various components of the compressor and 3) the equations governing the suction and discharge valve motion.

We first begin with the fluid dynamic model of the compressor, which is constructed using two basic building blocks called "ducts" and "volumes". Ducts are used to model the regions of the compressor where the flow displays a prevailing velocity direction. Volumes are instead used to model more complex regions where the one-dimensional flow hypothesis does not apply. The various chambers inside the suction and discharge mufflers and the compressor cylinder are for example modeled as volumes, while the ducts connecting the various chambers are modeled as ducts.

The equations governing the gas flow through ducts with cross sectional area S can be written as
with suitable initial and boundary conditions, where the gas density \( \rho \), the velocity \( u \), the pressure \( p \), the total energy \( e \), and the area \( S \) are considered to be functions of the time \( t \) and the spatial coordinate \( x \) along the duct axis.

The above nonlinear system of partial differential equations can be solved for the conservation variables \( \rho \), \( \rho u \) and \( \rho e \), provided that the pressure is expressed as a function of density and internal energy by means of an equation of state. In the perfect gas case, this relation is the well-known equation \( p = (\gamma - 1)\rho e \), where \( \gamma \) is the ratio between the specific heats, whilst for a real gas the function \( p = p(\rho e) \) is interpolated using a table of thermodynamic data.

The terms \( h_f(x,t) \) and \( h_q(x,t) \) account for the effects of friction and heat flux along the duct, and are defined as

\[
\begin{align*}
h_f &= f(Re) \frac{1}{2} \rho u^2 \frac{S \vert u \vert}{d} \quad (4) \\
h_q &= g(Re, Pr) \frac{\lambda}{d} (T_w - T) S_q \quad (5)
\end{align*}
\]

where \( Re \) is the Reynolds number defined with the equivalent duct diameter \( d \) as reference length, \( Pr \) is the Prandtl number, \( \lambda \) is the thermal conductivity, \( T_w - T \) is the difference between wall and fluid temperature, \( S_q \) is the duct heat flux surface per unit length. The functions \( f(Re) \) and \( g(Re, Pr) \) are reported in the literature, see for example [2], and have been chosen taking into account the character of the flowfield (laminar or turbulent) and the cross sectional area shape (circular or annular).

The duct boundary conditions, relevant for the applications considered in this paper, are essentially of three types: inflow, outflow, closed end. Both inflow and outflow boundaries can be treated as reflecting or non-reflecting boundaries. As explained in the following, also the interfaces between ducts and volumes are treated as inflow or outflow boundaries.

The flow conditions for volumes are described using the average values of density, total energy and kinetic energy. The integral form of the conservation laws for the density and total energy in a generic volume \( v \) can be written as

\[
\begin{align*}
\frac{d}{dt} \int_V \rho dv + \oint_{\partial V} \rho (u - b) \cdot nds &= 0 \quad (6) \\
\frac{d}{dt} \int_V \rho e dv + \oint_{\partial V} \rho e (u - b) \cdot nds + \oint_{\partial V} p b \cdot nds - \oint_{\partial V} q ds &= 0 \quad (7)
\end{align*}
\]

where \( \partial V \) denotes the boundary of \( v \), \( u \) is the gas velocity vector, \( b \) is the boundary velocity vector, and \( n \) is the normal unit vector pointing outward from \( v \). As an example, for stationary walls \( u = b = 0 \) and for the piston \( u = b \neq 0 \). In the case of the cylinder, the heat flux \( q \) is given by the relation

\[
q = g(Re) \frac{2}{d_c} (T_c - T) 
\]

where \( d_c \) is the cylinder diameter, \( T_c - T \) is the difference between the cylinder and the gas temperature and \( g(Re) = 2.1 Re^{0.7} \), as reported in [3, 4]. The Reynolds number \( Re \) is defined with reference to the cylinder diameter and the piston average velocity.

To complete the knowledge of flow condition inside the volume we could use the integral form of momentum conservation equation. This approach however is not advisable because, in our experience, the crude approximation of the volume by means of a single computation cell introduces excessive losses that essentially depend on the discretization error. The momentum equation is thus substituted by a simple estimate of the volume average kinetic energy, as reported in [5]. If \( m_i \) denotes the \( i \)-th mass flow entering the volume \( v_i \) and \( T_{kj} \) the associated kinetic energy, the estimate of the kinetic energy \( T_j \) of the volume \( v_j \) is given by
where $\alpha_j$ is a dissipation factor to be specified as part of the model. The kinetic energy $T_j$, together with the density $\rho_j$ and total energy $e_j$, given by Eq. (6) and Eq. (7), allow the computation of an average volume value for any thermodynamic property as a function of $\rho_j$ and $e_j = e_j - T_j$.

It is important to notice that the computational procedure for the volumes explicitly avoids to determine the velocity direction. Lacking some information about the flow conditions at the interfaces between ducts and volumes, we can not use directly the interface flux evaluation as for the interfaces between elements in ducts. Instead, we consider the interfaces between ducts and volumes as inflow or outflow boundaries and we follow the procedure outlined in the next section to evaluate the boundary flux.

Let's now turn to the model used to simulate the automatic valves. Two different schemes have been considered to describe the dynamical behavior of the suction and discharge valves. In the first model, the valve leaf is simply regarded as a (nonlinear) one-degree of freedom (DOF) damped mass-spring system. The model is nonlinear since both equivalent mass $m(x)$ and stiffness $k(x)$ are functions of the displacement $x$.

The equation governing the motion of a mass-spring system may be written as the first order system

\[
\begin{cases}
\dot{x} = p/m \\
p = f(x) - \omega(\omega x + 2\zeta p)
\end{cases}
\]

where $p$ is the momentum, $\omega$ the natural frequency ($\omega = \sqrt{k/m}$), $\zeta$ the damping ratio ($\zeta = c/2\sqrt{km}$) with $c$ the damping factor. The equivalent mass and stiffness are determined by numerical and/or experimental analyses. The force $f(x)$ is computed as the product of the total to static pressure difference across the valve, times an "effective force area" $A_e(x)$ experimentally obtained.

An "effective flow area" $A_{ef}(x)$, accounting for the geometric area of the valve opening as a function of $x$ and for the irreversibility of the flow through the valve port, has also been experimentally obtained. This area is used to define the cross sectional area $A(x(t))$ "seen" by the flow through the valve port.

The second valve model has been developed in an attempt to model both the valve leaf and the valve damper as two interacting mass-damper-spring systems governed by equations analogous to system (10). The two leaves interact through a contact element. The coupling between gas flow and valve motion is done as in the previous case. The numerical results here presented have been obtained with a one DOF suction valve and a two DOFs discharge valve models.

Both the equations governing the gas flow in ducts and volumes and those governing the valve motion are simultaneously integrated in time by means of an explicit Runge-Kutta time stepping method.

**DISCONTINUOS GALERKIN SPACE DISCRETIZATION**

The ducts flow equations can be written in compact vector form as

\[
\begin{align*}
\frac{\partial u}{\partial t} + \frac{\partial f(u)}{\partial x} &= s(u) \\
\end{align*}
\]

where the unknown $u$, the flux function $f$, and the source term $s$ are defined with reference to Eqs.(1-3). By multiplying by a "test function" $v$, integrating over the duct length $L$ and performing an integration by parts, we obtain the weak statement of Eq. (11)

\[
\int_L v \frac{\partial u}{\partial t} \, dx + [vf(u)]^b_a - \int_L f(u) \frac{\partial v}{\partial x} \, dx = \int_L vs(u) \, dx \quad \forall v
\]
where $a$ and $b$ denote the duct endpoints. A discrete analogue of Eq. (12) is obtained by subdividing $L$ into a collection of elements $\{E\}$ and by considering functions $u_h$ and $v_h$ which are piecewise $K$ degree polynomials inside each element (Galerkin method). Notice that $u_h$ and $v_h$ are in general discontinuous at element interfaces.

By regarding the integral over $L$ as the sum of integrals over the elements $\{E\}$ and by admitting the functions $u_h$ and $v_h$, we obtain the semidiscrete equations for a generic element $E$, which can be written as

$$\frac{d}{dt} \int_E v_h u_h \, dx + \left[ v_h f_h \right]_l^r - \int_E f_h \frac{\partial v_h}{\partial x} \, dx = \int_E v_h s_h \, dx \quad \forall v_h$$

(13)

where $f_h = f(u_h)$, $s_h = s(u_h)$, and where $l$ and $r$ denote the endpoints of element $E$. Eq.(13) must be satisfied for any element $E$ and for any function $v_h$. However, within each element, $u_h$ and $v_h$ are linear combinations of $k+1$ shape functions $\phi_i$, and Eq. (13) is therefore equivalent to the system of $k+1$ equations

$$\frac{d}{dt} \sum_{i=1}^{k+1} \phi_i u_{hi} + \left[ \phi_i f_h \right]_l^r - \sum_{i=1}^{k+1} \phi_i \frac{\partial \phi_i}{\partial x} \, dx = \sum_{i=1}^{k+1} \phi_i s_{hi} \, dx \quad 0 \leq i \leq k$$

(14)

which can be solved for the $k+1$ degrees of freedom $U_j(t)$ of the unknown solution $u_h$.

When evaluating the boundary terms of Eq. (14) at an internal interface, the flux function $f_h$ is not uniquely defined due to the discontinuous function approximation. It is therefore necessary to introduce an interface flux $h[\,\cdot\,\cdot\,]_{(+)}$ depending on the states $r$ and $l$ of the elements $(-)$ and $(+)$ sharing the interface. Among the several numerical flux formulations available in the literature we have used the flux vector splitting formulation reported in [6].

The numerical flux function at the duct endpoints $a$ and $b$ is used to prescribe the boundary conditions. This is accomplished by defining an "external" state $u_{bc}$ and by computing the flux at $a$ and $b$ as $h[u_{bc}, u(a)]$ and $h[u(b), u_{bc}]$, respectively.

For closed ducts, $u_{bc}$ is equal to $u$ with reversed velocity. In this way the resulting $h$ contains only a pressure contribution. At inflow/outflow boundaries, $u_{bc}$ is computed by combining the prescribed boundary data and the Riemann invariant associated to outgoing characteristics.

Linear high-order numerical methods require some form of non-linear limiting in order to ensure solution boundedness if discontinuities develop in the flow field. To this aim, the numerical scheme is augmented with a “shock capturing” term which is analogous to similar terms considered in stabilized FE methods, such as the streamline diffusion (SD) or the streamline upwind Petrov Galerkin (SUPG) methods, and which depends on the residual of the solution. More details about the DG numerical solution of the Euler and of the Navier-Stokes equations can be found for example in [1, 7, 8].

**CODE APPLICATION AND RESULTS**

The simulation code requires as input data the whole geometry of the compressor, the refrigerant inlet temperature and pressure, the vapor discharge pressure and the wall temperatures. The code computes the unsteady distribution of the thermal and fluid dynamic parameters, as well as valves lifts, work and refrigerant mass flow rates, during the entire compression cycle.

According to the one-dimensional numerical approach employed, the real compressor circuit can be reduced as summarized in figure 1 in which ducts and volumes are denoted by symbols $T$ and $V$, respectively.

The working conditions are determined by the saturation pressures at the evaporating and condensing temperatures and by the over-heating of the evaporated gas. We here report the results compared to the experimental measurements at the following conditions:

Refrigerant: R134a  
Evaporating Temperature = -23.3 °C ; Suction pressure = 1.15 bar  
Condensing Temperature = +55 °C ; Discharge pressure = 14.91 bar  
Inlet fluid temperature = 32 °C  
Motor speed = 2942 rpm.
A dedicated experimental set up has been constructed and the compressor has been equipped with pressure transducers, thermocouples and an endoscopic instrumentation for the determination of the valve motion.

Figure 2 shows the pressure behavior in the suction muffler (V2 in fig. 1) versus the crank angle. A good agreement between predictions and measurements is found both in the closed and in the opened suction valve phase. Notice in particular a high frequency pulsation overlapped to a lower one.

Fig. 3 and 4 report the pressure and the temperature inside the cylinder (V3).

During the compression the refrigerant pressure and temperature increase until the opening of the self-acting discharge valve. After the valve opening, the internal pressure varies following a pulsating behavior due to the combined effects of the piston motion and of the outlet flow rate. The latter is in turn the result of the interaction of the fluid flow with the automatic valve leaf. The temperature, instead, starts to decrease mainly due to the heat exchange with the cylinder walls. The comparison between simulated and experimental results in this delicate phase of the cycle is rather satisfactory. Work is nevertheless in progress to improve the accuracy of the simulation in this phase of the cycle.

During the re-expansion of the dead volume after the discharge valve closing, the temperature and the pressure continue to decrease until the opening of the suction valve. In the suction phase, the temperature increases again because of the mixing with the sucked refrigerant while the pressure changes are small according to the volume variation law and the inlet flow rate. Notice that the interaction between fluid flow and valve dynamics plays a
fundamental role in the overall behavior of the system during this phase. The simulation seems however quite accurate thanks to an effective prediction of the suction valve motion.

To complete the analysis of the overall compression cycle it must be noted how the computed pressure is slightly underestimated during the compression phase while it is overestimated during the expansion phase, compared to the experimental results. This may be due to the computed heat transfer rate and/or to uncertainties of the measurements.

Fig. 5, 6 and 7 present the pressure behavior in the discharge line: in the plenum (V4) directly downstream the discharge valve and in the two subsequent mufflers (V5 and V6). The numerical results predict fairly well the pressure impulse generated by the discharge valve opening and the valve rising instant, while the simulated phase of the more intense successive fluctuations is not exactly captured. A deeper analysis concerning both the thermal phenomena and the friction losses seems opportune to improve the accuracy of the simulation in this phase.

Fig. 8 shows the comparison between the computed suction and discharge valves lifts and the experimental data obtained by means of an endoscopic measurement. As can be seen, the agreement between computed and experimental results is excellent, especially for the suction valve. The opening and closing times, as well as the global motion, are well captured. The disagreement of the discharge valve is partly caused by the difficulties in reading the exact lift in the endoscopic visualization. It is emphasized that the calculations regarding the valve motion have been performed with an increased dead volume because more space was needed into the cylinder to accommodate the endoscopes.

---

**Figure 5 - Pressure in the cylinder head**

**Figure 6 - Pressure in the 2nd discharge plenum**

**Figure 7 - Pressure in the 3rd discharge plenum**

**Figure 8 - Non-dimensional valve motion**
Table 1 summaries the computed and measured global performance of the compressor. It can be noted how the computed mass flow rate is slightly overestimated of about 1.8%. In this regard it is noteworthy that the compressor performance losses, due to fluid leakage between piston and cylinder, have been computed modeling the clearance as a duct that connects the cylinder with the shell volume where the unsteady one-dimensional equations govern the flow. In particular the friction term $h_f(x,t)$, that accounts for laminar losses with an annular shape of cross section, is here expressed as

$$h_f = 12\pi \mu \frac{d}{r}$$

where $\mu$ is the dynamic viscosity of the refrigerant and $r$ the piston to cylinder radial clearance.

This way of modeling has allowed the valuation of the flow rate leakage during the whole compression cycle, i.e. the discharge and suction phases.

<table>
<thead>
<tr>
<th></th>
<th>Computed</th>
<th>Measured</th>
<th>Error %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass Flow Rate</td>
<td>Kg/h</td>
<td>3.82</td>
<td>3.75</td>
</tr>
<tr>
<td>Cooling Capacity</td>
<td>Watt</td>
<td>197.2</td>
<td>193.5</td>
</tr>
<tr>
<td>Total Input Power</td>
<td>Watt</td>
<td>147.1</td>
<td>149.3</td>
</tr>
<tr>
<td>COP</td>
<td>W/W</td>
<td>1.34</td>
<td>1.29</td>
</tr>
</tbody>
</table>

Table 1. Global performances: comparison between experimental and simulated results

CONCLUSIONS

A simulation model for reciprocating compressors used in household refrigeration has been developed and implemented. It can be easily adapted to different configurations of the whole circuit, different working fluids and operative conditions and therefore it results in an useful tool for design and development purposes. At the present state, the experimental validation of the code has shown promising potentialities of the code both for the global performances of the system and for the local fluid dynamic phenomena, in particular in the interaction fluid structure of the automatic valves. Nevertheless further analysis is worthy to be pursued as far as the thermal exchange and friction losses are concerned.

In the next future the code will be implemented with a complete heat exchange model that values all the heat fluxes among the building blocks ducts and volumes and the heat sources internal to the circuit due to mechanical and electrical losses.

REFERENCES


