Purdue University
Purdue e-Pubs

International Compressor Engineering Conference

School of Mechanical Engineering

1994

Numerical Study of the Thermal and Fluid-Dynamic Behaviour of Reciprocating Compressors

C. D. Perez-Segarra
Universitat Politecnica de Catalunya

F. Escanes
Universitat Politecnica de Catalunya

A. Oliva
Universitat Politecnica de Catalunya

NUMERICAL STUDY OF THE THERMAL AND FLUID-DYNAMIC BEHAVIOUR OF RECIPROCATING COMPRESSORS

C. D. Pérez-Segarra, F. Escanes, A. Oliva

Laboratori de Termotecnia i Energetica
Departament de Maquines i Motors Termics. Universitat Politecnica de Catalunya
Colom 9, 08222 Terrassa, Barcelona (Spain)
FAX: +34-3-739.91.01; Tel.: +34-3-739.81.92

ABSTRACT

A numerical simulation of the thermal and fluid-dynamic behaviour of reciprocating compressors has been developed. The governing equations of the flow (continuity, momentum and energy) written in transient and one-dimensional form are integrated in the whole domain. The simulation covers a wide range of compressor types, open or hermetically sealed, which can include different elements in their suction and discharge ducts (mufflers, connecting tubes, orifices, etc), allowing the possibility of backflow. This formulation requires additional information about convective heat transfer coefficient, friction factor, and pressure drop through singularities (orifices, valves, sudden enlargements and contractions). The equations are discretized by means of an implicit control-volume formulation using an upwind scheme for the convective terms. The coupling between the governing equations is made by means of a pressure-based method taking into account the compressibility of the flow. In order to illustrate the possibilities of the software developed, some numerical results are presented for a hermetic compressor.

INTRODUCTION

Thermal and fluid-dynamic behaviour of reciprocating compressor units are characterized by complex heat transfer and fluid flow phenomena: three-dimensional turbulent compressible flow, fast transient processes, complex geometries, moving surfaces, etc. The challenge of new refrigerants and the need for high efficiencies are strong incentives to develop of general and accurate prediction methodologies.

Several models have been presented in the literature. The most sophisticated is based on the numerical integration of the Navier-Stokes equations in multi-dimensional and transient form. This methodology allows the determination of the local velocity, pressure, density and temperature distributions in the whole domain without empirical information (except for the empirical constants and functions in turbulence models). Although the computational time required is usually quite large and turbulence models introduce uncertainties, these methodologies give valuable information that can be used in less general models [1].

An important simplification can be made assuming one-dimensional flow, that is, mean values of the different variables are considered instead of the local ones. This approach requires considerably less computational time than the multidimensional simulation, however it needs empirical information in order to evaluate pressure drop across tubes and singularities (valves, orifices,..) and heat transfer between the gas and the solid walls (cylinder, tubes,...). Most of the models proposed in the literature for predicting compressor performance are of this kind [2][3].

The objective of this work is to simulate the thermal and fluid-dynamic behaviour of reciprocating compressor, especially those hermetically sealed (see Figure 1). The model is one-dimensional and the analysis has been restricted to the gas flow through the cylinder and the suction and discharge ducts (the model assumes known values for the solid wall temperatures and the compressor speed). The governing equations are written in a general form and they are discretized and solved in the whole domain using a pressure-based algorithm. From a given set of parameters (geometry, wall temperatures, ..) and boundary conditions (inlet pressure and temperature and outlet pressure) the numerical simulation evaluates, at each instant or crank angle, the distributions of temperature, pressure, velocity, density, heat flux, etc, together with the instantaneous mechanical power transmitted to the gas in the cylinder. From these results, the integral mean values of the impulsion temperature, mass flow rate, volumetric efficiency, mechanical power, etc, are evaluated.

In the next sections the mathematical model and the main numerical aspects considered are described. Some numerical results are also presented in order to illustrate the possibilities of the simulation.

MATHEMATICAL FORMULATION

Assuming one-dimensional flow and negligible potential energy and conduction heat flux in flow direction, the governing equations of the flow (continuity, momentum and energy) for a finite control volume (CV), together with the state equation, can be written in the following form:

$$\dot{m}_o - \dot{m}_i + \frac{\partial m}{\partial t} = 0 \tag{1}$$

$$\dot{m}_o v_o - \dot{m}_i v_i + \frac{\partial m\bar{v}}{\partial t} = (p_o - p_i)\,S - \bar{\tau}_w A \tag{2}$$

$$\dot{m}_o e_o - \dot{m}_i e_i + \frac{\partial m\bar{e}}{\partial t} - V\frac{\partial \bar{p}}{\partial t} - \bar{p}\frac{\partial V}{\partial t} = \dot{Q} - \dot{W}_s \tag{3}$$

$$p = z\rho RT \tag{4}$$

where: the subscripts *i* and *o* indicate the inlet and outlet flow sections of the CV respectively; t is the time; $\dot{m}$ and m are the mass flow rate and the mass of the gas in the CV; v, ρ, p, T and e are the velocity, the density, the pressure, the temperature and the specific energy (enthalpy plus kinetic energy); $\bar{v}$, $\bar{p}$ and $\bar{e}$ are integral velocity, pressure and specific energy averages over the CV; $\bar{\tau}_w$ represents the mean shear-stresses between the gas and the solid surfaces; z is the compressibility factor; R is the gas constant; $\dot{Q}$ is the net heat power transferred into the CV; $\dot{W}_s$, is the power transferred from the gas; S, A and V are the cross-sectional area, heat transfer area and volume of the CV respectively.

Momentum equation, eq. (2), has been written assuming uniform cross-sectional area (that is, $S_i=S_o=S$); shear stresses are evaluated from the expression $\tau_w=(f/4)\rho|v|v/2$, where f is the friction factor. When a singularity is presented in the CV (valves, orifices, sudden enlargements, sudden contractions), see Fig.2, across the singularity the momentum equation is replaced by the following specific equation:

$$p_1-p_2-\Delta p_o=\frac{v_o}{2S_o}\left[\dot{m}\left(\frac{1}{\sigma_2^2}-\frac{1}{\sigma_1^2}\right)+\max(\dot{m},0)\left(\frac{1}{\sigma_2}-\frac{1}{C_c}\right)^2+\min(\dot{m},0)\left(\frac{1}{\sigma_1}-\frac{1}{C_c}\right)^2\right] \tag{5}$$

where: subscripts 1 and 2 indicate the downstream and upstream sections (or vice versa); v_o and S_o are the velocity and the cross-sectional area of the singularity respectively; σ is a cross-sectional area ratio ($\sigma_1=A_1/A_o$, $\sigma_2=A_2/A_o$); C_c is the contraction coefficient; Δp_o is the minimum differential pressure to open the valves (this value is zero everywhere except for the valves). In the analysis presented here valve dynamics have been considered in a simplified way using the equation written above when p_1-$p_2 > \Delta p_o$ and otherwise v_o=0. Equation 5 has been obtained assuming: i) steady, one-dimensional incompressible flow; ii) isentropic flow behaviour from the inlet section to the vena contracta; iii) the pressure on the back face of the orifice is equal to the plenum pressure.

In the *energy equation*, eq. (3), the inlet heat flux through the CV solid surfaces is obtained from the expression $\dot{q}=\alpha(T_w-T)$, where α is the heat transfer coefficient, T and T_w are the gas and the solid wall temperatures respectively. The power rate transferred from the gas to the ambient, $\dot{W}_s$, is zero everywhere except in the cylinder where $\dot{W}_s=\bar{p}dV/dt$.

Empirical correlations. In the momentum equation, the friction factor has been calculated using a correlation presented by Churchill [cit. in 4] for forced convection inside tubes; when singularities are presented the contraction coefficient has been evaluated using the classical values of Weisbach [cit. in 5]. In the energy equation, an overall heat transfer coefficient proposed by Liu and Zhou [6] has been employed inside the cylinder; in the tubes and mufflers standard correlations for forced convection inside tubes have been used; finally, the convection in the pressure vessel has been evaluated using a correlation derived by Raithby and Hollands [7] for natural convection between concentric spheres.

NUMERICAL PROCEDURE

The domain where the gas is flowing is divided into strategically distributed control volumes (CV). For each CV a grid node is assigned at its center, as is shown in Figure 3. Some elements, such as the cylinder, constitute a single CV and they cannot be divided into smaller ones. Other elements, such as tubes between mufflers, can be divided into an arbitrary number of CV. For each grid node the different scalar variables (temperature, pressure and density) are calculated. Velocities are calculated at the faces of these control volumes (that is, between grid nodes) using a staggered CV.

The SIMPLEC algorithm of Van Doormaal and Raithby, extended to compressible flow, has been employed. In what follows, only the main features will be indicated; for more details see Patankar [8] and Van Doormaal and Raithby [9]. The governing equations (1, 2 and/or 5 and 3) are discretized by means of an implicit control-volume formulation; convective terms are numerically approximated using an upwind numerical scheme. At each grid node, temperature and density are obtained from the energy and the state equations respectively; velocities are calculated at the faces of the CV from the momentum equation (together with the equation for the singularity when it is presented). Pressures are computed at the grid nodes using a pressure correction equation; this equation is obtained from the continuity equation using a corrected velocity $u'=d\Delta p'$ (where the coefficient d is obtained from the momentum equation, and $\Delta p'$ is the difference of the corrected pressures of the adjacent grid nodes) and a corrected density $\rho'=Kp'$ (where the coefficient K can be obtained from the state equation). The set of discretized equations obtained from the momentum, energy and pressure-correction equations are solved by means of a TDMA (Tri-Diagonal Matrix Algorithm). All thermophysical properties have been evaluated at their local conditions.

For each time step and from guessed values of the pressure and temperature (usually taken from the previous instant), velocities and densities are obtained from the discretized momentum equation and from the state equation respectively. Then the pressure correction equation is solved and after that pressures, velocities and densities are updated. Finally, temperatures are obtained from the energy equation. This sequence is repeated unless convergence is reached. The process runs step by step in the time direction giving the transitory evolution of the different variables. After several cycles, global convergence is reached when instantaneous velocities, pressures and temperatures are cyclically repeated. Better convergence rates have been obtained when: i) the momentum equation is under-relaxed (a relaxation factor of 0.6 is normally used); ii) a modified velocity $\bar{v}$, defined as $\bar{v}=vS/\bar{S}$ (where: v is the velocity, S the cross-sectional area, and $\bar{S}$ a constant arbitrary reference area), is introduced in order to reduce the step changes produced in the velocity due to the cross-sectional variations.

NUMERICAL RESULTS

In order to show the possibilities of the software developed, a compressor of the type shown in Figure 1 has been analysed. In this compressor the low-pressure dry gas from the evaporator enters to the space situated between the shell and the motor-compressor unit, and after that goes across the suction ducts to the cylinder, where it raises its pressure. Then, the gas goes through the impulsion ducts to the condenser. The gas on its way across the suction and discharge ducts goes through different elements such as tubes, mufflers, valves, etc. The numerical results here presented corresponds to:

- *Fluid:* R-12
- *Geometry:* i) cylinder: bore=20 mm, stroke=20 mm, clearance volume=0.125 cm^3 (2 %), orifice

valve diameters=3.5 mm; ii) volume between the compressor shell and the motor-compressor unit= 78.5 cm^3; iii) contact surface gas/pressure vessel=640 cm^2, gas/oil=300 cm^2, gas/motor-compressor unit=400 cm^2; iv) two mufflers in suction line: diameter=30 mm, length=30 mm; v) two mufflers in discharge line: diameter=20 mm, length=20 mm; vi) connections between mufflers: diameter=2 mm, length=20 mm; vii) tube from the motor-compressor unit to the pressure vessel in the discharge line: inlet diameter=2 mm, outlet diameter=4 mm, length=100 mm.

- *Minimum differential pressure Δp_o to open the suction and discharge valves:* 0.005 bar and 0.050 bar respectively.
- *Boundary conditions (operating conditions):* i) inlet gas pressure and temperature: 1bar and 10 C; ii) outlet pressure: 8 bar; iii) temperature of the pressure vessel and oil: 40 C, iv) temperature of the motor-compressor unit: 60 C; v) compressor speed: 50 Hz.
- *Initial conditions (t=0):* i) in the whole domain the gas is at rest; ii) pressure and temperature in the suction ducts: 1 bar and 10 C; iii) pressure and temperature in the discharge ducts: 8 bar and 100 C.
- *Numerical parameters:* i) number of grid nodes: 19 (as is indicated in Fig. 2); ii) number of time steps per compression cycle: 180 (although other time steps have also been tested).

These data have been arbitrarily estimated with the intention of showing some numerical aspects and characteristics of the software developed. Different number of time steps per cycle (90, 180, 270, 360, and 450) have been tested. For the initial conditions selected, converge solutions (in the sense that instantaneous pressures, temperatures, etc, are repeated every cycle) were typically obtained in about 300 cycles. Minor differences were observed in the transitory process (from the initial conditions to the converge solution) when the number of time steps per cycle was higher than 180. The effect of the time step on some compressor working parameters is shown in Table 1, where n is the number of time steps per cycle, $\dot{W}$ is the compressor input power, $\dot{m}$ the mass flux rate, η_v the volumetric efficiency, and w the input power per unit mass of refrigerant. When 180 time steps per cycle are employed, a typical computational time consumption of about 15 s/cycle has been obtained in a work-station of 75 MIPS.

Table 1. Influence of the time step on some compressor working parameters

n	90	180	270	360	450
$\dot{W}$ (W)	48.6	46.6	45.9	45.6	45.4
$\dot{m}$ (Kg/h)	2.36	2.36	2.36	2.36	2.36
η_v (%)	39.0	38.9	38.9	38.9	38.9
w (kJ/Kg)	74.3	71.2	70.2	69.6	69.4

The instantaneous pressure and temperature distributions along the compressor domain are presented in Figure 4 for four different crank angles ϕ (or instants), corresponding to the different process of the compression cycle: compression (ϕ=120), impulsion (ϕ=170), expansion (ϕ=210) and suction (ϕ=330). As can be seen, the instantaneous pressures and temperatures in the suction and discharge lines are almost uniform along the cycle except in the vicinity of the cylinder; pressure drop effects are specially important in the suction line and a superheating is produced when the gas goes across the compressor shell. The instantaneous pressure and temperature evolution per cycle in the cylinder are shown in Figure 5. The discharge process begins at $\phi \approx 160$ (where the pressure distribution reaches a maximum) and it ends at $\phi \approx 180$ (where an abrupt change in the slope of the pressure distribution is produced). The suction process begins at $\phi \approx 230$ (see temperature distribution); while the valve is open the pressure remains nearly constant; the suction process ends at $\phi \approx 20$

Finally, the pressure-volume diagram is shown in Figure 6. The numerical results obtained with the software developed are compared to the ones obtained from the same software, without considering heat transfer and pressure drop across both suction and discharge ducts, and from a standard idealized thermodynamical model. The idealized thermodynamical model considers neither the heat transfer and the pressure drop in suction and discharge ducts nor in the cylinder; this model assumes isobaric suction and discharge process and polytropic compression and expansion process (a polytropic coefficient of 1.125 has been employed).

CONCLUSIONS

A numerical simulation of the thermal and fluid-dynamic behaviour of reciprocating compressors has been presented. The modelization is based on the solution of the one-dimensional governing equations of the flow in the whole domain. Empirical correlations for the gas-solid surface interaction (friction and heat transfer) are needed to provide for the closure of the conservation equations. The numerical solution has been made using a control-volume formulation and a segregated pressure-based algorithm. The formulation is flexible and general, and a great variety of geometries and boundary conditions can be solved. In order to show the possibilities of the software developed some illustrative results have been presented. A comparison with less general models has been made in order to show the influence of the heat transfer and pressure drop. In the near future more effort will be made to implement valve dynamics and a multidimensional heat transfer calculation of the solid elements (compressor shell, cylinder walls and motor). Furthermore a more specific empirical correlations in order to increase the accuracy of the simulation will also be introduced.

ACKNOWLEDGMENT

This study has been supported by the company Unidad Hermética S.A. (group ELECTROLUX COMPRESSORS), by the Comisión Interministerial de Ciencia y Tecnología (ref. no. PTR-90060) and by the Dirección General de Investigación Científica y Técnica (ref. no. PB90-0606). The authors thank J.M.Serra, J.Pons and A.Castillo of Unidad Hermética S.A. for the technical support given.

REFERENCES

[1] G.W.Recktenwald, J.W.Ramsey and S.V.Patankar, Predictions of Heat Transfer in Compressor Cylinder, 1986 Int. Compressor Eng. Conf., ed. by J.F.Hamilton and R.Cohen, Purdue University, pp.159-174, 1986.

[2] W.H.Hsieh, T.T.Wu, C.L.Yeh, K.K.Kuo, Numerical Simulation of Gas-Dynamic and Heat-Transfer Processes in Two-Stage, Very High-Pressure Gas Compressors, 1992 Int. Compressor Eng. Conf., ed. by J.F.Hamilton, Purdue University, pp.289-298,1992.

[3] P.J.Singh, A Digital Reciprocating Compressor Simulation Program Including Suction and Discharge Piping, 1984 Int. Compressor Eng. Conf., ed. by R.Cohen, Purdue University, pp.129-138,1984.

[4] S.Lin, C.C.K.Kwok, R.Y.Li, Z.H.Chen and Z.Y.Chen, Local Frictional Pressure Drop During Vaporization of R-12 Through Capillary Tubes, Int. J. Multiphase Flow, Vol.17, No.1, pp.95-102, 1991.

[5] P.R.Bullen, D.J.Cheeseman, L.A.Hussain and A.E.Ruffellt, The Determination of Pipe Contraction Pressure Loss Coefficients for Incompressible Turbulent Flow, The Int. J. of Heat and Fluid Flow, Vol.8, No.2, pp.111-118, 1987.

[6] R.Liu and Z.Zhou, Heat Transfer Between Gas and Cylinder Wall of Refrigerating Reciprocating Compressor, 1984 Int. Compressor Eng. Conf, ed. by R.Cohen, Purdue University, pp.110-115, 1984.

[7] G.D.Raithby and G.T.Hollands, A General Method of Obtaining Approximate Solutions to Laminar and Turbulent Free Convection Problems, Ad. Heat Transfer, ed. by T.F.Irvine and J.P.Hartnett, Academic Press, New York, pp. 266-315, 1975.

[8] S.V.Patankar, Numerical Heat Transfer and Fluid Flow, McGraw-Hill, New York, 1980.

[9] J.P.Van Doormal, G.D.Raithby and B.H.McDonald, The Segregated Approach to Predicting Viscous Compressible Fluid Flows, Int. Gas Turbine Conf. Exibit, Dusseldorf, Germany, 1986, ASME paper 86-GT-196.

FIGURES

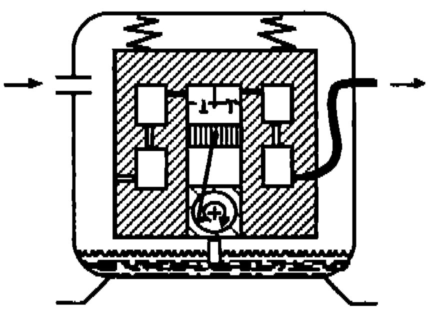

Figure 1. Hermetic compressor schematic

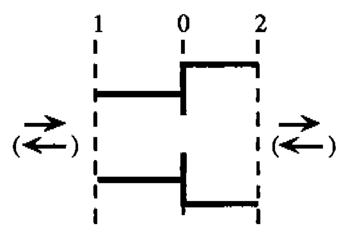

Figure 2. Flow through a generic singularity

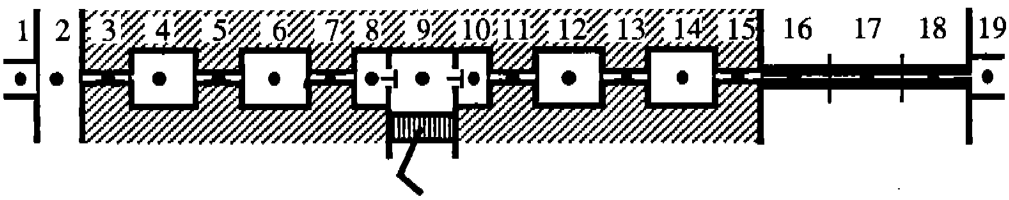

Figure 3. Example of control-volume and grid-node distribution for the hermetic compressor represented in Fig.1

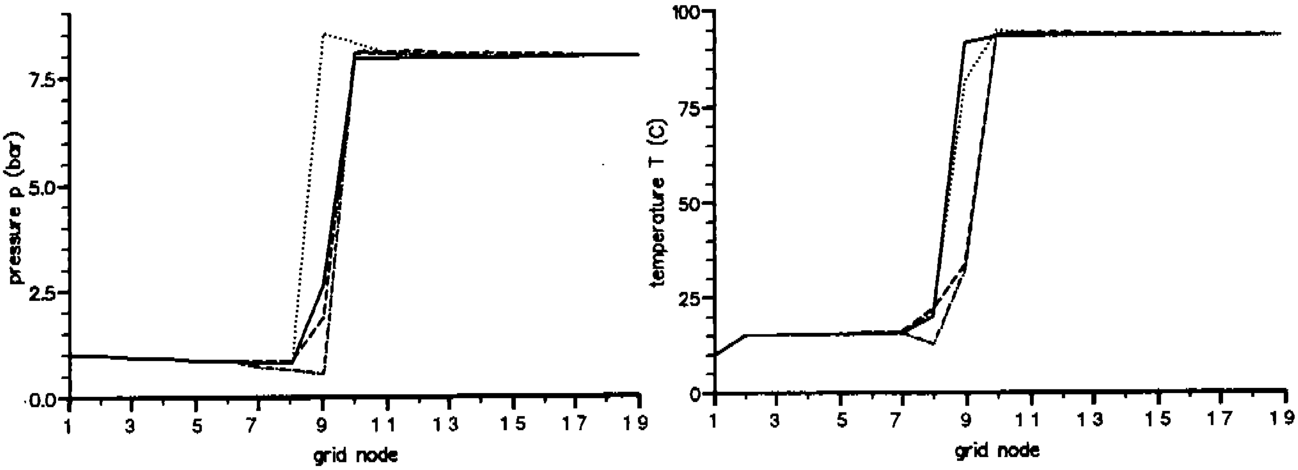

Figure 4. Pressure and temperature distribution along the compressor domain at different crank angles: —— : $\phi = 120$ (comp.); ...... : $\phi = 170$ (disch.); ---- : $\phi = 210$ (exp.); -·-·- : $\phi = 330$ (suct.).

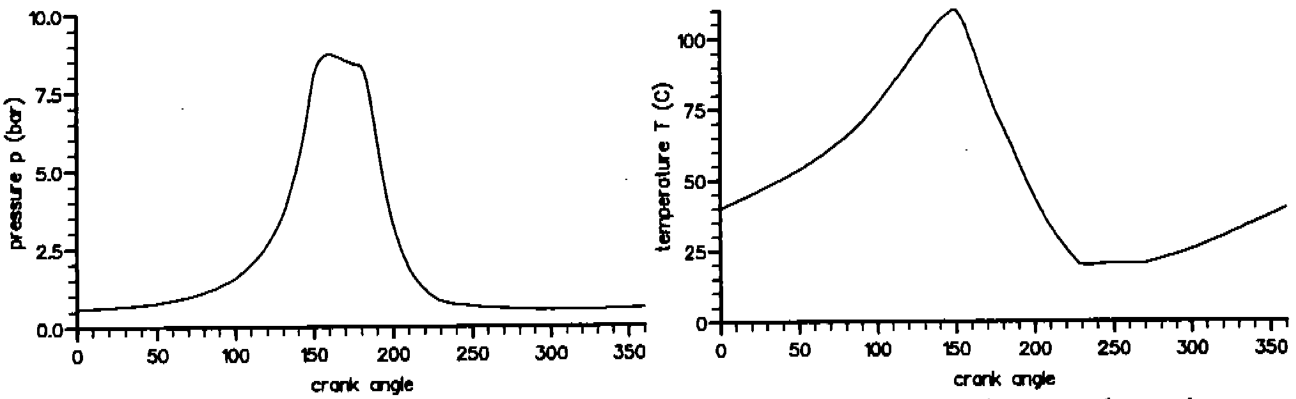

Figure 5. Instantaneous pressure and temperature distribution inside the cylinder during a complete cycle.

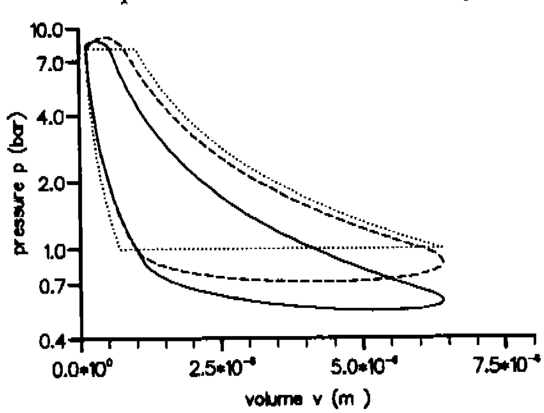

Figure 6. Pressure-volume diagram: —— : present model; ---- : idem but heat transfer and pressure drop are considered neither in the suction duct nor discharge duct; ...... : idealized thermodynamical model.