SCREW COMPRESSORS
Incorrect contact of screw machine rotors

Jaromír Švígler
Faculty of Applied Sciences, Department of Mechanics,
University of West Bohemia, Czech Republic

ABSTRACT

One of the most important part of screw machines, i.e. screw compressors or expanders, is a work space with its boundary surface which is produced by the surfaces of teeth and an inner cylinder surface of a machine box. This contribution deals with the analysis of the correct and incorrect contact of the teeth of machine rotors created with screw surfaces. Both conjugate surfaces touch each other at the curve by the correct contact. The curve contact changes into contact of isolated points by incorrect contact. The creation of a technically undesirable gap between both conjugated surfaces is a consequence of this change. Incorrect contact of surfaces is caused by a large parallel displacement of the axis of one of the surfaces dealt with in this contribution. Geometrical visualisation of the gap is performed in instantaneous time.

1 INTRODUCTION

The work space with its boundary surfaces has a geometrically complicated, non-stationary shape in the course of working process. Mostly processes of screw machines which are analysed from the point of view of statics, kinematics, dynamics, fluid dynamics, heat arise, heat transfer i.e. heat convention and heat conduction, heat removal and other phenomena connected with these basic processes, take place in the work space and on its boundaries optimally in its inlet or outlet. The analysis of the mentioned phenomena which take place during the state change of the medium in work space has a great significance, since the phenomena exercise a decisive influence on the machine operation. The processes on the boundary surface where the compressed or expanded medium comes in contact with the outer environment or with the environment of neighbouring work space, are of crucial importance because they act on the above mentioned phenomena. From this point of view the study of causes producing a gap by incorrect contact is significant. The deformation of rotors and the machine box takes place under the influence of heat and acting forces. Temperature on the surface of the machine box was measured 76.4 °C at the place of discharge by the injected compressor with axis distance 85 mm and discharge pressure $7 \cdot 10^5$ Pa [1]. The surface temperature of teeth may be estimated at 110 °C approximately [3]. Entry parameters of the medium of the screw engine [2] would be 600 °C and $6 \cdot 10^5$ $Pa$. In this case the surface temperature of teeth both rotors is essentially greater.
In this contribution the gap by incorrect contact is created by parallel displacement of female rotor axis that simulates defect of the correct contact of screw surfaces. The inner deformation of screw surfaces is not involved into solution and likewise general position of rotor axes that arises by real deformation of the machine box is not accepted. This simplification was necessary for the first step of the solution the complicated problem. The kinematical and geometric solution of the contact of tooth surfaces was considered as 3D problem for instantaneous time. The used methodology of the solution and computation presented in this contribution is valid and available for the compressor as well as for the expanders. In [6] the rotor interference i.e. the clearance between the teeth was solved as two-dimensional problem.

2 CORRECT CONTACT OF SCREW SURFACES

Screw surfaces which create tooth faces of rotor of screw machines are represented with two two-dimensional manifolds. Surface \( \sigma_3 = \sigma_3(p_{3L}, \chi) \) is defined by parametric equation 

\[
\sigma_3, r_L = \sigma_3, r_L(p_{3L}, \chi)
\]

whose concrete form in frame \( R_3 \), see Figure 1, is followed

![Figure 1. Generating pair of mutually conjugate screw surfaces \( \sigma_2 \) and \( \sigma_3 \)](image)
\[
\sigma_R^2 r_L = T_{R_i R_j} \left( -p_{3L} \right) T_{R_j R_k} \left( \phi_{SL} \right) \begin{bmatrix}
    r_s \sin \varphi - r_i \sin \chi \\
    -r_s \cos \varphi + r_i \cos \chi \\
    0 \\
    1
\end{bmatrix},
\]

where \( T_{R_i R_j} \) denotes transformation matrix from frame \( R_i \) to \( R_j (R_i \to R_j) \). Conjugate surface \( \sigma_2 = \sigma_2 \left( p_{3L}, \chi, \varphi_2 \right) \) is created in the direct envelope way according to the Distelli theorem. Problem of the explicit determination the corresponding conjugate profile of two meshing cylindrical gears is solved in [5]. The explicit solution is performed by solving the ordinary differential equations. The solution which is presented in this contribution is founded on kinematical principles. Generating the surface \( \sigma_2 \) is performed in the basic frame and then the transformation \( t : R_3 \to R \) was made necessarily

\[
\sigma_R^2 r_L = T_{R_i R_j} \left( \pi \right) T_{R_{3o} R_i} \left( -a_w \right) T_{R_j R_{3o}} \left( \varphi_3 \right) \cdot \sigma_R^i r_L.
\]

Conjugate screw surface \( \sigma_2 \) is described by equations

\[
\sigma_R^2 r_L = T_{R_i R_j} \left( \varphi_{2ko} \right) T_{R_j R_k} \left( \varphi_2 \right) \cdot \sigma_R^k r_L,
\]

\[
\sigma_R^k n_L \cdot \mathbf{v}_{L32} = 0.
\]

The second equation creates the necessary condition for the contact of both surfaces i.e. the condition of perpendicularity of relative velocity between the both surfaces and vector of normal in contact point. In these equations \( \varphi_3 = i_{32} \varphi_2 \) where \( i_{32} \) is the requested transmission ration. Instantaneous position of rotors, in which the contact of both surfaces is determined, is defined with angle \( \varphi_{3Ko} = i_{32} \varphi_{2Ko} \). Unitary vector of normal to screw surface \( \sigma_3 \) in basic frame \( R \) is described by equation

\[
\sigma_R^2 n_L = S_{R_j R_i} R_{R_i} n_L \cdot \sigma_R^2 n_L = \frac{r_{i3} \times r_{i3} \times t_2}{r_{i3} \times r_{i3} \times t_2},
\]

where \( S_{R_j R_i} \) is the matrix expressing the rotational displacement of the system \( R_j \) with respect to system \( R \), \( \sigma_R^2 n_L \) is the normal vector to the surface \( \sigma_3 \) in contact point \( L \) expressed in space \( R \), and \( r_{i3} t_1, r_{i3} t_2 \) are tangential vectors of surface \( \sigma_3 \). Vector of relative velocity surfaces \( \sigma_3, \sigma_2 \) in contact point \( L \) is given by equation

\[
r v_{L32} = r v_{L31} - r v_{L21} = S_{R_j R_i} R_{R_i} v_{L31} - r \omega_{21} \times r L = v_{L21} \times r L,
\]

in basic frame \( R \). The correct contact of both surfaces takes place in a contact curve \( ch_2 \). This situation is graphically demonstrated in Figure 2 for instantaneous time. Screw surfaces in Figure 2 illustrate only part of complete screw surfaces of rotor teeth. For the solution, a SLF4 profile [4], which is shown in Figure 3, was used. As observed from this figure, the tooth profiles of both rotors consist of more conjugate curves as \( k_1 - k_2, k_3 - k_4, k_5 \) is trochoid curve, \( k_6 \equiv U, k_6 - k_6, k_9 - k_{10} \) and \( k_{11} - k_{12} \) where \( k_1, k_3, k_7, k_9, k_{11} \) are defined curves of female rotor and \( k_2, k_4, k_8, k_{10}, k_{12} \) are their envelopes.
Figure 2  Curve correct contact of conjugate screw surface $\sigma_2$ and $\sigma_3$

Figure 3.  Tooth profiles in the cross section through rotors of the screw machine
3 INCORRECT CONTACT OF SCREW SURFACES

The incorrect contact of screw surfaces is caused, as described in this paper, Figure 4, by a parallel displacement $\Delta r_{o_3} = [\Delta x_{o_3}, \Delta y_{o_3}, 0]^T$ of axis $o_3$ of surface $\sigma_3$ which is displaced in position $\sigma_3^d$. The curve contact by correct meshing surfaces changes into point contact by incorrect touch. Contact of surfaces $\sigma_2$ and $\sigma_3^d$ takes place, see Figure 4, under condition

$$ r_K^R = r_X^R = \sigma_2 r_L \wedge \sigma_3 n_L \times \sigma_3 n_L = 0 \wedge \phi_3^p = \min \{\phi_j^p\} $$

(6)

in cross section $\tau$ that is defined with $\phi_3^p = \min \{\phi_j^p\}, j \in (1, m)$, where $\phi_j^p$ is the angle of rotation of profile $p_j$ of surface $\sigma_3^d$ into the contact position with profile $p_2$ of surfaces $\sigma_2$. Profiles $p_2 \equiv \sigma_2 \cap \tau$ and $p_3 \equiv k_3 \equiv \sigma_3^d \cap \tau$ appear as intersections of surfaces $\sigma_2$ and $\sigma_3$ with plane $\tau$ which is orthogonal to axes $o_2, o_3$ optimally to $o_3^d$. Similarly, as by correct contact the instantaneous position of rotors, in which the contact of both surfaces is searched, is determined with angle $\phi_{3Ko} = i_{32} \phi_{2Ko}$. Incorrect contact of surfaces $\sigma_2$ and $\sigma_3^d$ is demonstrated in Figure 5 where $CH_{32}$ is the contact point and $m_2, m_3$ are curves of minimum.

Figure 4. Searching of incorrect contact of screw surfaces $\sigma_2$ and $\sigma_3^d$
distance of surfaces $\sigma_2$ and $\sigma_3^d$. In consequence of the parallel displacement, one of the axes of the surfaces $\sigma_2$, $\sigma_3$, the contact in the curve $ch_{32}$, at which the two surfaces touch by correct meshing, changes into contact at point $CH_{32}$. Separation of both surfaces takes place outside the contact point. The situation is well observed in cross sections in Figure 6 where the detail of the approaching of both profiles $p_2$ and $p_3^d$ is shown on the right side.

4 INCORRECT CONTACT OF TEETH

Incorrect contact of teeth of male and female rotors was made for following basic geometrical parameters of the screw compressor: axis distance $a_w = 85\,mm$, gear ratio $i_{32} = 1,2$, helix angle on the rolling cylinder of both rotors $\gamma = 45^\circ$, real displacement of axis $o_3$, $\Delta x_{o_3} = 0,005\,mm$ and $\Delta y_{o_3} = 0,15\,mm$. For graphical visualization the displacement $\Delta x_{o_3} = 0,1\,mm$ and $\Delta y_{o_3} = 1\,mm$ was used. Length of tooth part of both rotors, which is $l = 193,8\,mm$, is divided into four periodical segments so that each of them is determined by the same position of tooth profile at its beginning and at its end. Tooth profil position is changed in consequence of its screw moving along rotor’s axis. In every segment only one point of contact exists. Subsequently we obtain four contact points along the length of axes. Among neighbouring contact points a tooth space between both surfaces $\sigma_2$ and $\sigma_3^d$ exists. Curves of minimal distance $m_2$, $m_3$, see Figure 5, between surfaces was determined. The situation can be observed in Figure 7, where the upper figure demonstrates the position of rotors and the lower figure shows the distance function i.e. the distance between both surfaces in the cross section. Comparison of rotor positions by correct and incorrect contact of the teeth with the corresponding distance function is shown in Figure 8.
Figure 6. Cross sections of surfaces $\sigma_2$ and $\sigma_3^d$ with the incorrect contact
Figure 7. Position of rotors and distance function
Figure 8. Comparison of profile position by correct and incorrect meshing with distance function
5 CONCLUSION

The paper deals with the preliminary analysis of the incorrect contact of two screw surfaces which create tooth faces of rotors of screw machines. Incorrect contact of teeth is caused by a large parallel displacement of the axis of female rotor as described in this contribution. Curves of minimal distance between both surfaces were determined. Problem was solved as a three-dimensional one. The curve at which the two surfaces contact each other by correct meshing teeth changes into isolated points. Contact points were determined by incorrect meshing both surfaces. In the solved case, the number of contact points was four. The tooth space exists between both surfaces $\sigma_2$ and $\sigma_3^\Delta$ among neighbouring contact points. The phenomenon is important namely at injected screw machines where the force transmission takes place. From the point of view of force transmission ensues that the strain of the covering layer of the tooth surface of rotors which take incorrect contact is greater than it is usually assumed. Solution of the incorrect contact of tooth surfaces by the general displacement of the axes of both surfaces is further logical step in investigating this problem.

Acknowledgment

It should be acknowledged that this work was supported by the project MSM 4977751303 of the Ministry of Education of the Czech Republic to which I express my thanks.

REFERENCES


Improving screw compressor performance

Nikola Stošić, Ian K. Smith, Ahmed Kovačević
Centre for Positive Displacement Compressor Technology, City University, London, U.K.
Jung-uk Kim, Jinwoo Park
Airplus Co Ltd, Korea

ABSTRACT

Screw compressor efficiency is improved by reducing internal leakage and this can be effected by minimising the clearance between the rotors and the casing. The effect on performance of three factors which influence the working clearance was analysed. These are: the interlobe clearance, adjusted to the rotor temperature change, rotor contact on the lobe flat side and displacement of the discharge bearing centres. Due account of the results of this study was taken in the design of a large screw compressor, which was then built and tested. The efficiencies obtained were the highest ever reported for screw compressors in the open literature. This confirmed the validity of this approach.

1 INTRODUCTION

Since the performance of screw compressors is highly affected by leakage, any reduction of the clearances within them must improve their efficiency.

Modern rotor manufacturing methods, such as grinding with simultaneous measurement, control and correction of the profile, enable the profile tolerance to be maintained within ±5 µm. This enables the clearances between the rotors to be kept below 15 µm. With such small clearances, rotor contact is very likely and hence the profile and its clearance distribution must be generated in such a manner that damage or seizure will be avoided should this occur.

Current practice to avoid rotor contact and seizure is to make clearances larger than required by manufacturing limitations. However, the clearances can be reduced by making their distribution non-uniform around the profile so that should hard rotor contact occur, it will not be in rotor areas where sliding motion between the rotors is dominant.

The design of screw compressors is an interactive process in which the performance estimated in the design process is compared with that obtained from prototype testing. The prototype is then repeatedly modified until the desired values are obtained. However, as more accurate methods of simulating the performance become available, the need for modifying the prototype can be minimised. To this end, the authors have developed a suite of subroutines for the estimation of screw compressor performance. These include facilities for the generation of new
rotor profiles, the estimation of the thermodynamic processes within the compressor and hence, the performance changes that are likely to result from modifying the profiles. The accuracy of these estimates has been enhanced by extensive comparison with test results on many machines, followed by modification of the model to obtain better agreement not only with the bulk parameters, such as flow delivery and power consumption, but also with the cyclic variation of important instantaneous values, such as the pressure distribution within the compressor working chamber. More details of this are given by Hanjalic and Stosic 1997. Essentially, the computational procedure is as follows:

(i) A pre-processor generates the lobe profiles from which the complete screw rotor pair is formed and the working chamber volume function is generated.

(ii) The performance is estimated by the numerical solution of a set of differential equations, which account for the conservation of mass, momentum and energy and which include estimates of the thermodynamic properties of the working fluid and associated flow processes. The results include not only estimates of bulk parameters, such as the delivery flow rate and power input, but also instantaneous values of how the working fluid pressure and temperature vary within the compressor cycle.

(iii) A post-processing program is then used to present the results in graphical and tabular form.

This procedure was used to estimate what effect the three main factors that govern the compressor working clearance have on its performance. The results thus obtained were then applied to the design of a 4/5-220 mm rotor diameter oil injected air compressor.

2 ROTOR MODIFICATION AND ITS EFFECT ON PERFORMANCE

In order to maximise screw compressor delivery rates and efficiencies, interlobe clearances must be made as small as possible without the likelihood of hard rotor contact between the rotors, in regions where the sliding velocity is high. Three main effects must be allowed for in the design in order to ensure this.

2.1 Thermal Expansion of the Rotors and Housing

Although the temperature range over which screw compressors operate is not large, the effects of thermal expansion are highly significant if the small clearances required between the rotors and between the rotors and the housing are to be maintained under working conditions. Thus, the rotor clearances obtained under manufacturing conditions must be estimated while taking account of thermal distortion that will occur when the compressor reaches its operating temperature and pressure and the calculation must allow for the unequal expansion of the rotors in different coordinate directions. An example of this is given in Fig. 1, where the left diagram shows the estimated clearance distribution when the rotors are cold, while, the centre and right diagrams show the clearances after the rotors reach their working temperatures. Additional information about the screw compressor clearance management and other means of improving efficiency may be found in Stosic, 2004.

Despite the cold clearances being adequate, under operating conditions, rotor contact may occur on the round flank, as shown in Fig 1b, or on the flat flank, as shown in Fig 1c.
2.2 Rotor contact on the lobe flat side

Oil flooded compressors have direct contact between their rotors. In well designed rotors, the clearance distribution will be set so that this is first made along their, so called, contact bands, which are positioned close to the rotor pitch circles. Since the relative motion between the contacting lobes in this region is almost pure rolling, the danger of their seizing, as a result of sliding contact, is thereby minimised. As already shown in Fig 1, the contact band may be either on the rotor round flank b), or on the rotor flat flank c).
The traditional approach is to maintain a high, so called, positive gate rotor torque, which ensures round flank contact. What is not widely appreciated is that there are significant advantages to be gained by maintaining a negative gate rotor torque to ensure that contact, when it occurs, will be on the flat face. The reason for this can be understood by examination of the sealing line lengths, shown as item 5 in Fig 2. Here it can be seen clearly that the flat flank sealing line is much longer than that of the round flank. Thus, minimising the clearance on the flat flank will reduce the interlobe leakage more than minimising the round flank clearance. Also, negative gate torque is achieved by making the gate rotor lobes thicker and the main rotor lobes correspondingly thinner. The displacement is thereby increased. Thus both these effects lead to higher compressor flows and efficiencies.

2.3 Displacement of the bearing centres

Since there must be some clearance in the bearings, the pressure loads will tend to push the rotors apart and displace their centres from their design position with respect to the compressor housing. Thus, if the bearing centres are set to be the same as those of the rotors, the rotors will be eccentric and as a result the clearance between the rotors and housing will be smaller at the low pressure side of the rotors and larger at the high pressure side. Since leakage is caused by the pressure difference, this displacement creates the least favourable rotor position for efficient compressor operation.

Also the resulting rotor displacements from their design positions may result in contact between the rotor tips and the housing unless allowance is made for them during the design. The situation can be remedied by making the bearing centre distance smaller than that of the rotor housing and aligned to maintain a uniform clearance between the rotors and housing. To minimise the rotor interlobe clearance, the bearing centre distance must be even further reduced.

2.4 Optimising the rotor profile

Although rotor profiles are designed to minimise the blow-hole area, this is frequently associated with increase in the sealing line length. The optimum profile shape is therefore that which minimises the sum of both the blow-hole and sealing line leakage areas.

2.5 Performance simulations of optimised rotors

The three possible improvements described in Sections 2.1-2.3 were applied systematically to performance simulations of an oil flooded air compressor of 223mm male rotor diameter when running at a shaft speed of 2100rpm with a suction pressure of 1 bar and a discharge pressure of 9 bar. The results of these are presented in Table 1.

<table>
<thead>
<tr>
<th>Q[m³/min]</th>
<th>ηv</th>
<th>P[kW]</th>
<th>Psp[kW/m³/min]</th>
<th>η</th>
<th>t_out[°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>15.9</td>
<td>0.901</td>
<td>103.7</td>
<td>6.506</td>
<td>0.835</td>
<td>86</td>
</tr>
<tr>
<td>16.1</td>
<td>0.911</td>
<td>102.9</td>
<td>6.383</td>
<td>0.850</td>
<td>85</td>
</tr>
<tr>
<td>16.2</td>
<td>0.916</td>
<td>102.6</td>
<td>6.327</td>
<td>0.858</td>
<td>84</td>
</tr>
<tr>
<td>16.0</td>
<td>0.906</td>
<td>103.7</td>
<td>6.444</td>
<td>0.843</td>
<td>85</td>
</tr>
</tbody>
</table>
The first row gives the results without any modification. The second row shows the performance obtained by introducing ‘hot clearances’. The third row shows the effect of displacing the discharge bearings 50 µm in the direction opposite to that of the line of action of the pressure radial forces and the fourth row represent the effects of inducing rotor contact on the flat flank side.

3 MODIFICATIONS IN SCREW COMPRESSOR DESIGN

Following the above analysis, a 220 mm diameter rotor oil flooded 4/5 air compressor was designed with a L/D ratio of 1.55, including all the proposed improvements described in Section 2. The rotors are shown in Fig 3 and its estimated performance in Figs 4 and 5.

![Fig. 3 4/5 220 mm rotors](image)

The rotors were made of steel and were manufactured by grinding, thus ensuring excellent precision and contact. The rotor L/D ratio was kept optimally low to minimise deflection, maintain favourable dynamic characteristics and achieve the required compressor compactness. The housing was designed for manufacture from normalized nodular iron for easy and precise machining, while ensuring both tightness and structural integrity.

Cylindrical roller bearings and two-point contact ball bearings for the axial loads were employed to ensure long operating life. A high quality mechanical shaft seal was provided for safe and reliable operation.

After manufacture of all the components, the compressor was assembled and tested.
Fig. 4 Estimated flow as function of compressor power

Fig. 5 Estimated specific power as function of shaft speed
4 TESTING OF THE PROTOTYPE COMPRESSOR

A photograph of the test rig with the compressor installed in it is presented in Fig 6 and a basic piping and instrumentation diagram of it is shown in Fig 7. The compressor was driven by a 100 kW electrical motor, with its speed controlled by a frequency converter. A six band belt drive with a speed reduction ratio of 1:2.52 was used to transmit the power from the motor to the compressor drive shaft, which was mounted on two pedestal bearings to resist the side load imposed on it. After discharge from the compressor, oil was removed from the compressed air in a separator. The separated oil was then recirculated into the compressor, via a water cooled heat exchanger, and reinjected through a nozzle, as a result of the pressure difference between the discharge pressure and the pressure in the compressor working chamber at the oil admission point. The separated discharge air was then passed through an orifice plate, in order to determine its flow rate and then vented to a flue below the test cell.

Fig. 6 Layout of the test rig

Instrumentation scheme of the test rig is shown in Fig 7. The measurements from which performance estimates of the compressor were derived are compressor speed and torque, compressor inlet pressure and temperature $p_1$ and $t_1$, compressor discharge pressure and temperature $p_2$ and $t_2$, the orifice plate inlet pressure and temperature $p_3$ and $t_3$ and the pressure drop across the orifice plate $\Delta p$. The orifice plate section of the discharge pipe was 100 mm diameter and the orifice plate was designed according to BS 1042. Additionally, the atmospheric pressure $p_0$ and temperature $t_0$ were measured.
Apart from the laboratory atmospheric pressure, which was input manually, all measurements were obtained from electrically generated signals derived directly from the speed, torque, pressure and temperature measurements. These were logged in an Instrunet data logger, programmed to display average values at 5 second intervals. In addition, instantaneous values were displayed visually by a variety of digital electronic meters and centrally at the computer screen. The latter were very helpful in setting up the required test conditions. The recorded data was then input to a computer as an unformatted data file for further processing. The test schedule employed followed the guidelines laid down by PNEUROP and CAGI in document PN2CPTC1.

The test data were obtained at discharge pressures of 7, 9 and 11 bars. The estimated delivery, power and specific power are compared with the experimentally derived values, as shown in Figs 8 and 9. The results show good agreement between the predicted and test values.

The specific power obtained, as a result of the optimisation procedure described, is 5-8 % lower than that of any other machine of this type, as reported to date in published literature.
Fig. 8 Comparison of the estimated and measured flow and power

Fig. 9 Comparison of the estimated and measured specific power
5 CONCLUSIONS

Three suggested improvements to the clearance management of screw compressor rotors were applied to the design of a compressor, which was then built and tested. Test results, obtained from it, confirmed the advantages that were claimed for them and the compressor achieved the best performance results yet reported for machines of this type. The implementation of such design refinements requires no additional time or cost. They are therefore recommended for inclusion as a regular part of screw compressor design practice.

REFERENCES

Calculation of Bearing Forces and Drive Torque of Rotary Displacement Machines

Knut Kauder and Jörg Temming
FG Fluidenergiemaschinen, University of Dortmund

ABSTRACT

The thermodynamic simulation of rotary displacement machines by means of a chamber model is an accepted method for the development and analysis of these machines. The software system KaSim, developed at the University of Dortmund, allows the simulation of constant and variable speed operation of the machines by calculating the thermodynamic behaviour of the working fluid.

This paper presents a new enhancement of KaSim, which enables the user to calculate bearing forces and driving torques of rotary displacement machines. Information about bearing forces makes it easier to select the correct bearings and optimise the design, including durability calculations. Information about the driving torque is necessary to analyse the transient operation behaviour, for example of screw-type superchargers in automotive applications. Also the influence of different geometric machine parameters on the thermodynamic behaviour and the acceleration performance of rotary displacement machines can be analysed.

First this paper describes the method for calculating forces and torques with KaSim on a simple model of a rotating disc. Furthermore a model based on a dry-running screw-type supercharger without timing gear is generated and analysed with this method. A comparison of the simulation results with measurement data concludes the paper.

NOMENCLATURE

\[ A, \bar{A} \quad \text{Area} \quad [m^2] \]
\[ F, \bar{F} \quad \text{Force} \quad [N] \]
\[ l \quad \text{Length} \quad [m] \]
\[ M_T, \bar{M} \quad \text{Drive torque, torque} \quad [Nm] \]
\[ n \quad \text{Rotation speed} \quad [\text{min}^{-1}] \]
\[ p \quad \text{Pressure} \quad [\text{Pa}] \]
\[ P_i \quad \text{Indicated power} \quad [\text{W}] \]
\[ r \quad \text{Radius} \quad [m] \]
\[ \bar{s} \quad \text{Position vector} \quad [m] \]
\[ \text{II} \quad \text{Pressure ratio} \quad [-] \]
\[ \omega \quad \text{Angular velocity} \quad [\text{s}^{-1}] \]

Subscripts

\[ \text{HR} \quad \text{Male rotor} \]
\[ \text{NR} \quad \text{Female rotor} \]
\[ n \quad \text{Normal direction} \]
\[ i \quad \text{Index} \]
\[ x, y, z \quad \text{Directions} \]

1 INTRODUCTION

The simulation of rotary displacement machines is an accepted means for the analysis, evaluation and development of these machines. The degree of sophistication in modelling and simulation can be varied. A simple model describes an isentropic, adiabatic compression with only one equation, while very complex models use a combination of
FEM (Finite Element Method) and CFD (Computational Fluid Dynamics) to simulate machine behaviour, [4].

A common method for the simulation of rotary displacement machines is by means of a chamber model. The University of Dortmund has developed the program KaSim, which allows the simulation of constant and variable speed operation of any rotary displacement machine on the base of chamber models, [2]. The use of rotary displacement machines in automotive applications has been experiencing a renaissance. Consequently, the simulation of transient machine behaviour is at present becoming more and more important. This paper introduces a method for calculating the driving torques and the bearing forces of rotary displacement machines with the program KaSim. First the method is described using a simple rotating disc chamber model. The influence of simulation steps on the degree of accuracy is analysed and discussed. Then a model of a dry-running supercharger, also developed at the University of Dortmund [3], is generated and simulated by this method. Finally the results are discussed and compared with measurement data for the analysed screw-type supercharger.

2 SIMULATION SYSTEM

The heart of the program system KaSim for simulating operating behaviour is the calculation of the thermodynamic processes inside the machine. The model of a screw-type compressor consists of separate chambers connected by clearances or interface areas. A more detailed description of the modelling and the simulation system structure is given in [2]. The influence of different parameters on the simulation, for example of different clearance heights, is discussed in [1].

The aim of the new enhanced version of KaSim is the calculation of the indicated power during compression in a different way. The indicated power also can be calculated from forces, torques and velocities. To obtain data for forces and torques it is necessary to know the pressure in the chamber, the surface area on which the pressure acts, and the centre of gravity of this surface related to a fixed coordinate plane. With the knowledge of these parameters, forces and torques acting on each component as a result of pressure can be calculated. The indicated power, for example in a piston compressor, is defined by the pressure on the cylinder and its velocity. In a rotary displacement machine the indicated power derives from the drive torque and rotation speed. In a correct model the resulting indicated power calculated from the volume curve or from the mechanical properties has to be equal.

With the bearing position in a fixed coordinate plane, it is also possible to calculate the bearing forces. For this purpose a linear system of the equations of motion has to be set up and solved. In applications with constant rotation speed the system supplies data for the bearing forces and drive torque. In non constant speed applications with given drive torque the system supplies the actual rotor accelerations as well as the bearing loads. The acceleration behaviour of rotary machines can be analysed in this way. A more detailed description of the modelling process and the calculation algorithms is given below and in Cap. 3.

For first experiences with the new enhanced KaSim the simple rotating disc model is not adequate\(^1\). It was only intended to be used for analysing the simulation process and

\(^1\)because the model of the rotating disc is no machine model. It is only a model to analyse the simulation process and accuracy.
was built because it was easier to analyse than a complex model of a Roots blower or screw-type supercharger, Fig. 1. The model consists of three chambers, an inlet and an outlet valve. Chamber C is filled through the inlet, chamber A compresses the gas and expels it through the outlet valve. Chamber B runs through the cycle without changing its volume. One result of the simulation is the indicated power during the compression process calculated from the thermodynamic behaviour:

$$ P_i = \int p \, dV $$

(1)

But the indicated power also can be calculated from the mechanical properties of the rotary displacement machine:

$$ P_i = \omega \, M_T $$

(2)

Rotation and model speed are synchronised. The pressure is a thermodynamic parameter of each chamber. The indicated power of this simple model is then given by:

$$ P_i = \sum_{i=1}^{3} \omega \frac{r}{2} F_i = \sum_{i=1}^{3} \omega \frac{r}{2} p_i r l $$

(3)

Fig. 2 shows the results of the rotating disc simulation using a different number of simulation steps. The calculated indicated powers tend towards the same marginal value with increasing time steps per period, and the relative tolerance decreases to zero. Above 2500 simulation steps per period the tolerance between the indicated powers is lower than 1%. It is also significant that the indicated power calculated through the drive torque is lower than the marginal value, while the indicated power calculated by volume curve is higher and tends more quickly towards the marginal value. The reason can be found in the calculation process for each time step, Fig. 3 a to c.

Fig. 3 a shows a cutout from the flow chart of the calculation process. The first step in calculating the indicated power by volume curve is an isentropic, adiabatic compression in each chamber. After this we dealt with the clearance flows, Fig 3 a). These clearance flows
lead to a lower chamber pressure, particularly in chambers connected to the discharge area, Fig. 3 b. Consequently the calculated indicated power is higher than the marginal value. In the drive torque calculation the pressure at the beginning of the time step is decisive. If the chamber volume decreases or increases, the pressure changes. Especially during the compression phase a deviation can be observed, Fig. 3 c. Therefore calculation errors are implicit in both methods due to the use of time steps. But if there are enough time steps the error is small and acceptable.

The model of the rotating disc shows that the enhancement of KaSim is an alternative approach for calculating the indicated power. The calculation of the drive torque leads

![Diagram](image-url)

Figure 3: Comparison of accuracy a) cutout from the flow chart b) calculated by volume curve c) calculated by drive torque
Figure 4: Description of a rotor surface segment

to the same indicated power as the calculation via the volume curve. In the next step the new method can be implemented in the more complex models of screw-type machines. The aim is not only to calculate the drive torque but also the bearing forces.

3 GENERATION OF SUPERCHARGER MODEL

Because of the complex geometry of a screw-type supercharger the calculation of the chamber volumes and the rotor surfaces is difficult. A special program was developed to solve this problem, a chamber model generator for rotary displacement machines. In this program the rotors (male and female) are divided into 2000 cross segments along the z-axis, and each segment is analysed. The process for generating the chamber model of a screw-type machine with chambers and connections is explicitly described in [1]. The generation of the screw-type supercharger model is analogous.

The chamber model generator also calculates the area of the rotor surfaces for each chamber. The profile cross-section of male and female rotors is defined by a series of points. In each segment a part of the rotor surface area is thus determined by four points, Fig. 4. The force vector $\vec{F}$ resulting from the pressure $p$ in the chamber on the surface vector $\vec{A}$ is defined by:

$$\vec{F}_i = -p \vec{A}_ni$$

(4)

This force results in a torque $\vec{M}$ on a point with a distance $\vec{s}$ from the rotors origin:

$$\vec{M}_i = \vec{s}_i \times \vec{F}_i = - p \vec{s}_i \times \vec{A}_ni$$

(5)
The geometrical centre of each rotor is fixed in the centre of the cross-section on the discharge site.

Because the points P1 to P4 are not in the same plane, the quadrangle is divided into two triangles with the Points P1, P2, P3 and P2, P3, P4, Fig. 4. To calculate the force and torque using *Kasim* the chamber model generator determines pressure depending on force and torque area for each chamber:

\[
\frac{\vec{F}}{-p} = \sum\frac{\vec{F}_i}{-p} = \sum (\bar{A}_{n1} + \bar{A}_{n2}) \quad (6)
\]

\[
\frac{\vec{M}}{-p} = \sum\frac{\vec{M}_i}{-p} = \sum (\bar{s}_1 \times \bar{A}_{n1} + \bar{s}_2 \times \bar{A}_{n2}) \quad (7)
\]

In [2] the simulation system *Kasim* was presented and the aims and basics of the system were explained. At this early stage of development of the simulation system the implementation of mechanical capacities and mechanical connections was already planned. The object-oriented paradigms and the intensive use of derived classes in *Kasim* now allow an easy implementation of the new algorithms.

The new capacity class, the *mechanical capacity*, stores the mechanical properties of a rigid body. Rigid bodies in the screw-type supercharger model are the male and female rotors. Typical mechanical properties of the rotors are their mass moments and their bearing positions (related to the geometrical centre). A new connection class, the *surface connection*, takes into account forces and torques resulting from the pressure on the rotors in the chambers. Therefore at each time step the pressure of all chambers is multiplied with the corresponding force and torque areas as presented in Eq. 6 and Eq. 7. Pressure forces and torques on the rotors are related to the geometrical centre of the rotors.

After pressure forces and torques are applied to the rotors, a linear equation system can be built to calculate the bearing forces, the drive torque and the acceleration of the rotors. The system consists of Eulers equations of motion for each rotor, the *principle of linear momentum* and *angular momentum*. The bearings lock the rotors in position, Fig. 5. All forces in each direction, and the moments around the x- and y-axis add up to zero. In the supercharger application the rotors can rotate freely around their z-axis, so the motion of the rotors can be summarized as follows:

\[
\sum \vec{F}_{\text{Rotor}} = \vec{0} \quad (8)
\]

\[
\sum M_{x,y,\text{Rotor}} = 0 \quad (9)
\]

\[
\sum M_z,\text{Rotor} = \Theta_{\text{Rotor}} \dot{\omega}_{\text{Rotor}}^2 \quad (10)
\]

But the male and the female rotor are kinematically connected through the constraint of the meshing profiles, which results in an additional equation:

\[
\dot{\omega}_{HR} = i \dot{\omega}_{NR} \quad (11)
\]

The newly-developed supercharger transfers the torque of the female rotor directly within the rotor intermesh area, so a timing gear is not necessary, ref. Fig. 5. A special profile

\footnote{For the torque this is a given area multiplied by the distance from the geometrical centre. In the following, both force and torque are described as an area.}

28
Figure 5: Forces and torques on male and female rotor
(contact force not shown in this figure, located between male and female rotor in the rotor intermesh area)

has been developed to make this direct female rotor drive possible. To avoid damaging the rotors a wear-resistant coating is applied to the rotor surfaces. The contact force is also considered in the linear equation system. During the rotation of the rotors the contact point varies with the rotation angle. In the simulation model the contact point is fixed between the rotors near the rolling circle. Further geometric machine parameters of the newly-developed dry-running screw-type supercharger are given in Table 1

<table>
<thead>
<tr>
<th></th>
<th>male rotor</th>
<th>female rotor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of lobes</td>
<td>3</td>
<td>5</td>
</tr>
<tr>
<td>Wrap angle</td>
<td>200°</td>
<td>120°</td>
</tr>
<tr>
<td>Crown diameter</td>
<td>72 mm</td>
<td>67.5 mm</td>
</tr>
<tr>
<td>Rotor length</td>
<td>101 mm</td>
<td></td>
</tr>
<tr>
<td>Displacement</td>
<td></td>
<td>283 cm³</td>
</tr>
</tbody>
</table>

4 SIMULATION RESULTS

With the help of a specially-developed algorithm this linear equation system now can be solved in order to obtain the bearing forces, the drive torque and the rotor accelerations.
Figure 6: Calculated bearing forces (axial and radial) and drive torque
\( n_{\text{male}} = 15000 \text{ rpm, } \Pi = 2 \)

\[
\begin{align*}
    \text{a} &= F_{A_{z,HR}}, \quad \text{b} = F_{A_{z,NR}}, \\
    \text{c} &= F_{B_{xy,HR}}, \quad \text{d} = F_{B_{xy,NR}}, \\
    \text{e} &= F_{A_{xy,HR}}, \quad \text{f} = F_{A_{xy,NR}}, \\
    \text{g} &= \text{drive torque } M_T
\end{align*}
\]

Fig. 6 shows the calculation results for bearing forces and drive torque of the supercharger model without timing gears at \( n_{\text{male}} = 15000 \text{ RPM (at constant speed) and a pressure ratio of } \Pi = 2 \).

Inside the supercharger model the rotor phase corresponds to a rotation angle of 120 degrees (because there are 3 male rotor teeth). After a rotation of 120 degrees the working cycle repeats again. Significant results are a lower axial force on the female rotor than on the male rotor and the higher load on the bearings at the discharge side compared to the suction side. Both results correspond to experiences and measurements on different screw-type compressors [5]. The drive torque (curve g in Fig. 6) and bearing forces rise until a rotor phase of 0.3. Related to this point, the highest pressure in the chambers occurs shortly after the rotor teeth cross the guiding edges on the discharge side. The pressure rises until connection to the discharge port leads to pressure equalisation between chamber and discharge port.

Going into more detail, we have found that approximately 10% of the total drive torque should be applied to the female rotor. The direction of the torque is constant during the phase. This is necessary to prevent the rotors from rattling and to achieve a clearly-defined contact area on the front flank (male rotor) and rear flank (female rotor) of the rotors [3].

Fig. 7 and Fig. 8 show a comparison of the calculated and measured drive torque and effective power of the screw-type supercharger. In the simulation, additional torque is
Figure 7: Comparison of measured (a) and calculated (b) drive torque in [Nm]

taken into account due to friction in bearings and seals. The tendency in drive torque
conforms in both curves but the gradient of the measured data is higher.
In the central area of the characteristics diagram the calculated and measured effective
power conforms, but at lower and higher pressure ratios and rotation speeds the accuracy
is lower due to the curves of the drive torque. To put it briefly, the curves for drive
torque and effective power suggest that the deviation is affected by pressure and speed of
rotation.
Influence coefficients affecting accuracy are the adiabatic modeling of the supercharger,
the model for friction at the seals and bearings and at the contact point between male
and female rotors. These influences will be taken into account in the future in order to
achieve better concordance between measurement and simulation.

5 CONCLUSION

The enhancement of KaSim with pressure-dependent force and torque areas makes pos-
sible a calculation of bearing forces and drive torque within the chamber model. The
availability of data on bearing forces makes durability calculations possible. In variable
speed applications, drive torque and rotor acceleration provide indicators of the accele-
ration behaviour of rotary machines.
In comparison with measured data the simulation achieves a moderate degree of accuracy.
Models for friction loss in the seals, the bearings and between the male and female rotors
still have to be developed and integrated into the software. More complex challenges
will be the further integration of heat transfer and rotor deformation models in order to
increase the accuracy of the simulation system.
Figure 8: Comparison of measured (a) and calculated (b) effective power in [kW]

REFERENCES


Identification of constraints in the optimal generation of screw compressor rotors by the pressure angle method

Nikola Stošić
Centre for Positive Displacement Compressor Technology, City University, London, UK

ABSTRACT

The profile gradient method has been recently introduced as a means of generating screw compressor rotor profiles. As a single parameter method, this procedure is convenient for the optimisation of screw compressor rotors and evaluating their quality. In this paper, the procedure is modified to adopt the pressure angle instead the profile gradient and applied to a screw compressor rotor rack. Optimisation constraints are analysed and an example of optimal profile generation is presented.

NOMENCLATURE

<table>
<thead>
<tr>
<th>C</th>
<th>Rotor centre distance</th>
<th>Subscripts</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>Parameter in Box optimisation method</td>
<td>1 Main rotor</td>
</tr>
<tr>
<td>k</td>
<td>Constant in Box optimisation method</td>
<td>2 Gate rotor</td>
</tr>
<tr>
<td>n</td>
<td>Constant in Box optimisation method</td>
<td>i Index</td>
</tr>
<tr>
<td>R</td>
<td>Random number</td>
<td>l Index</td>
</tr>
<tr>
<td>r</td>
<td>Radius</td>
<td>r Rack, index</td>
</tr>
<tr>
<td>x</td>
<td>Coordinate, Box variable</td>
<td>w Pitch circle</td>
</tr>
<tr>
<td>y</td>
<td>Coordinate</td>
<td></td>
</tr>
<tr>
<td>α</td>
<td>Line slope angle, Box reflection constant</td>
<td></td>
</tr>
<tr>
<td>φ</td>
<td>Profile parameter, pressure angle</td>
<td></td>
</tr>
<tr>
<td>θ</td>
<td>Meshing Angle</td>
<td></td>
</tr>
</tbody>
</table>

1 INTRODUCTION

New procedures for screw compressor rotor profile generation are continually being published. One of the more recent is to use the profile gradient, which was introduced to the screw compressor rotor profiling by Rinder and Grafinger, 2002, both to generate a profile and to
evaluate it. As a single parameter procedure, this method is convenient for the optimisation of screw compressor rotors, if used together with the constraints required to match the rotor geometry to its functional requirements. The pressure angle, which is the angle of the profile normal and which is the arc tangent of the profile gradient $dx/dy$, is used in this paper for the same purpose. It is applied here to the rack form, from which the rotor profiles are generated, rather than to the actual rotor profiles. By this means, pressure angle profiling can be combined with previously published optimisation procedures, to design screw compressors with the lowest manufacturing and running costs.

In industrial practice, rotor profiles are defined by Cartesian coordinates in the plane normal to their axes of rotation and by the gradient of these coordinates in that plane. These can be expressed in term of $x$ and $y$ and their derivatives, either $dy/dx$ or $dx/dy$. Alternatively they may be defined by use of polar coordinates, as functions of the angular position, $\varphi$, which in majority of cases corresponds to the pressure angle. The gradient values are obtained from the profile curves, expressed in analytical form, or numerically, from the rotor coordinates. Specification of both the coordinate values and their gradients results in redundant information. Therefore, it may be possible to reduce the information required by defining a profile gradient as a generic function from which all the information required can be derived. Thus starting with the profile gradient and one of the coordinates, any other coordinate can be estimated by:

$$x = x_0 + \int_{y_0}^{y} \frac{dx}{dy} dy$$

(1)

If $dx/dy$ is given in analytical form and its integral is known, $x$ may be found by direct integration. The integration can also be performed numerically if the integral is not known, or if the gradient is specified in a discrete form.

When generating screw compressor rotors, the pressure angle method has the advantage of easily maintaining profile continuity not only of the profile curves, but also of their gradients. Thus, if a succession of curves is used, the final coordinates of one curve serve as the initial condition for the next curve and therefore the profile continuity is retained. Moreover, the continuity of the profile gradient will be maintained even if the pressure angle value is retained only at the profile connections.

Since the profile curvature is a derivative of the profile gradient, it can be used to advance the coordinate step for a smooth profile, as well as a constraint required by the tool. Moreover, since the contact line between the rotors is defined by the profile gradient, its shortest distance from the rotor cusp is a good measure of the blow-hole area. Finally the profile gradient directly defines the rotor meshing condition $\theta$.

2 ANALYSIS OF THE PRESSURE ANGLE PROCEDURE IN PROFILING SCREW COMPRESSOR ROTORS

The majority of rotor profiles currently used by industry are generated from curves based on the actual rotor of interest. The majority of these curves are circular. Trochoids are then generated from them to obtain their counterpart profile on the meshing rotor. Circles are very suitable functions for the pressure angle method, because the pressure angle is the same as their profile parameter $\varphi$ and the profile gradient is a simple arc tangent function of the angle parameter, which is usually the same as pressure angle. Thus, $x=r \cos \varphi$ and $y=r \sin \varphi$, while
\[ \frac{dx}{dy} = -\tan \phi \]. The situation is very similar with straight lines, which have constant derivatives. Other generating curves, like ellipses, parabolae and hyperbolae have more elaborate, but still simple analytical forms, usually expressed as functions of the profile coordinates, \( x \) or \( y \). Moreover, if properly distributed, even the discrete point values of the profile gradient can be sufficient for generating the profile.

Alternative approach is to generate the rotor profile on a rack, which is a rotor of infinite radius. This is then equally applicable to both the main and gate rotors. Currently only two rack generated screw rotor profiles are in industrial use. These are that of Rinder, 1984, as adopted by the Trane Company in the USA for their refrigeration compressors, and the author’s ‘N’ profile. The latter is used by a growing number of manufacturers throughout the world in air, refrigeration and process gas compressors and in expanders.

In terms of the gradient profile generation, rack generated curves have a substantial advantage over those which are rotor generated. Namely, the rack rolling coordinate \( y = r_1 \psi \), unlike that of any rotor generated coordinate, continuously increases with \( x \) for all known screw rotor profiles. This means that the profile gradient \( \frac{dx}{dy} \) is a bounded function, which never goes to infinity and, therefore, the pressure angle is also bounded. Hence it cannot generate discontinuities.

![Figure 1 Rotor coordinate systems](image)

Since \( \frac{dx}{dy} \) is a bounded function for a rack generated profile, it may be integrated numerically by a simple procedure. However, it has been found out that second order, more accurate methods are advantageous. Therefore, the trapezoidal rule has been applied in this work. A Runge Kutta fourth order method was also used for the integration of \( y \), but the results so obtained were not different to those obtained from a second order solution.
Once generated, either by a fixed geometry approach, or through an optimisation procedure, the rack profile is used for the rotor profile generation. The envelope method was used for that purpose. Its application for the generation of screw compressor rotors was presented by Stosic, 1998.

A pair of screw compressor rotors, together with their rack, is presented in Fig. 1, with the male rotor on the left and the female on the right and the rack in bold between them. The centre line distance between them is \( C = r_{1w} + r_{2w} \), where \( r_{1w} \) and \( r_{2w} \) are the rotor pitch circle radii. Let \( x_r, y_r \) and \( dx_r/dy_r \) be the rack coordinates and their gradient respectively. The meshing condition \( \theta \) is a rotation angle of the main rotor at which the rotors contact. It results from the envelope condition:

\[
\frac{dy_r}{dx_r} (r_{1w} \theta - y_r) - (r_{1w} - x_r) = 0
\]

As may be seen, this equation is explicit in angle \( \theta \), which can be derived directly from it. Once solved, \( \theta \) serves to generate the rotors profile and also to calculate the rotor sealing lines. The rotor profile coordinates \( x_{01} \) and \( y_{01} \) of the main rotor and \( x_{02} \) and \( y_{02} \) of the gate rotor are:

\[
x_{01} = x_r \cos \theta - (y_r - r_{1w}) \sin \theta
\]
\[
y_{01} = x_r \sin \theta + (y_r - r_{1w}) \cos \theta
\]

\[
x_{02} = C - x_r \cos \theta - (y_r - r_{2w}) \sin \theta
\]
\[
y_{02} = x_r \sin \theta + (y_r - r_{2w}) \cos \theta
\]

**Table 1 ‘N’ profile rack gradients and pressure angles**

<table>
<thead>
<tr>
<th>Profile part</th>
<th>Curve</th>
<th>Gradient</th>
<th>Limits</th>
</tr>
</thead>
<tbody>
<tr>
<td>D-C</td>
<td>Circle</td>
<td>( dx_r/dy_r = -\tan \phi )</td>
<td>( 0 &lt; \phi &lt; \alpha_1 )</td>
</tr>
<tr>
<td>C-B</td>
<td>Straight line</td>
<td>( dx_r/dy_r = \text{const} = \tan (\pi - \alpha_1) )</td>
<td>( \phi = \alpha_1 )</td>
</tr>
<tr>
<td>B-A</td>
<td>Parabola</td>
<td>( dx_r/dy_r = 0.5y^{0.5} )</td>
<td>( \tan \alpha_1 &lt; dx/dy &lt; 0 )</td>
</tr>
<tr>
<td>A-H</td>
<td>Trochoid on the rack from the circle on the main rotor</td>
<td>( dx_{01}/dy_{01} = -\tan \phi ) on the main rotor</td>
<td>( 0 &lt; \phi &lt; \alpha_1 )</td>
</tr>
<tr>
<td>H-G</td>
<td>Trochoid on the rack from the circle on the gate rotor</td>
<td>( dx_{02}/dy_{02} = -\tan \phi ) on the gate rotor</td>
<td>( 0 &lt; \phi &lt; \alpha_1 )</td>
</tr>
<tr>
<td>G-F</td>
<td>Straight line</td>
<td>( dx_r/dy_r = \text{const} = \tan (\pi - \alpha_2) )</td>
<td>( \phi = \alpha_2 )</td>
</tr>
<tr>
<td>F-E</td>
<td>Circle</td>
<td>( dx_r/dy_r = -\tan \phi )</td>
<td>( 0 &lt; \phi &lt; \alpha_2 )</td>
</tr>
<tr>
<td>E-D</td>
<td>Straight line</td>
<td>( dx_r/dy_r = \text{const} = 0 )</td>
<td>( \phi = \pi )</td>
</tr>
</tbody>
</table>
Any rotor profile may be used to demonstrate pressure angle profile generation. However, taking into account that rack gradient generation is superior to rotor gradient generation, an “N” profile rack generated rotor has been used as an example in this paper without any loss in generality. Stosic and Hanjalic, 1997, who included a full specification of the curves from which the rack is constructed, described this profile and included analyses of several aspects of its generation. The ‘N’ rotor profile is presented in Fig. 2 while its curves and their gradients are defined in Table 1.
4 OPTIMISATION OF THE ROTOR PROFILE

The criteria for screw profile optimisation are valid irrespective of the machine type and duty. Thus, an efficient screw machine must admit the highest possible fluid flow rates for a given machine rotor size and speed. This requires that the fluid flow cross-sectional area through the rotors must be as large as possible. In addition, the maximum delivery per unit size or weight of the machine must be accompanied by minimum power utilization. This implies that the efficiency of the energy interchange between the fluid and the machine is a maximum. Accordingly unavoidable losses such as fluid leakage and energy losses must be kept to a minimum. However, increased leakage may be acceptable if it is associated with greater bulk fluid flow rates. Overall, the required compressor delivery rate must be obtained by simultaneous optimisation of the rotor size and speed to minimise the compressor weight while maximising its efficiency. It follows that a multivariable minimisation procedure is needed for screw compressor design with the optimum function criterion comprising a weighted balance between compressor size and efficiency or specific power.

The algorithm of the thermodynamic and flow processes used in optimization calculations is based on a mathematical model comprising a set of differential equations of conservation of mass and energy and algebraic equations of state and instantaneous compressor volume which fully describe the physics of all the processes within the screw compressor.

The energy equation is in form of internal energy rather than enthalpy as the derived variable. This was found to be computationally more convenient, especially when evaluating the properties of real fluids because their temperature and pressure calculation is not explicit. However, since the internal energy can be expressed as a function of the temperature and specific volume only, pressure can be calculated subsequently directly. All the remaining thermodynamic and fluid properties within the machine cycle are derived from the internal energy and the volume and the computation is carried out through several cycles until the solution converges.

Leakage in a screw machine forms a substantial part of the total flow rate and plays an important role because it affects the delivered mass flow rate and compressor work and hence both the compressor volumetric and adiabatic efficiencies.

Injection of oil or other liquids for lubrication, cooling or sealing purposes, modifies the thermodynamic process in a screw compressor substantially. Special effects, such as gas or its condensate mixing and dissolving in or flashing out of the injected fluid must be accounted for separately if they are expected to affect the process. In addition to lubrication, the major purpose for injecting oil into a compressor is to seal the gaps and cool the gas.

The solution of the set of differential equations is performed numerically by means of the Runge-Kutta 4th order method, with appropriate initial and boundary conditions. As the initial conditions were arbitrary selected, the convergence of the solution is achieved after the difference between two consecutive compressor cycles becomes sufficiently small. Once solved, internal energy and mass in the compressor working chamber serve to calculate the fluid pressure and temperature.

The solution of the mathematical model of the physical process in the compressor provides a basis for a more exact computation of all desired integral characteristics with a satisfactory degree of accuracy. The most important of these properties are the compressor mass flow rate, the indicated work and power and specific power, volumetric and adiabatic efficiencies. A full
and detailed description of the presented model of the compressor thermodynamics is given in Hanjalic and Stosic, 1997.

When attempting to optimise a compressor design, the criterion for the desired result, such as the minimum power consumption or operational cost must first be resolved. However, the power consumption is coupled to other requirements, which should also be included, such as low compressor price, or investment cost. The problem becomes obvious if the requirement for low power consumption conflicts with the requirement for low compressor price, which is overcome by weighting the various elements of the target function.

The Box complex method was used here to find the minimum of the target function, which, in the case described in this paper, was the specific power. The constrained simplex method emerged from the simplex method, which was introduced by G. Box, 1957 and developed 1969 by M. Box. It is also suitable for the constrained cases because that only a few starting trials were needed, and the simplex immediately moves away from unsuitable trial conditions.

This is a multivariable constrained optimisation process. The task is to maximise a target function \( f(x_1, x_2, \ldots, x_n) \), subjected simultaneously to the effects of explicit and implicit constraints and limits, \( g_i \leq x_i \leq h_i, i = 1, n \) and \( g_i \leq y_i \leq h_i, i = n+1, m \) respectively, where the implicit variables \( y_{n+1}, \ldots, y_m \) are dependent functions of \( x_i \). The constraints \( g_i \) and \( h_i \) are either constants or functions of the variables \( x_i \).

Since the nonlinear problem is to be solved, it is necessary to use \( k \) points in a simplex, where \( k=2n \). These starting points are randomly generated so that both the implicit and explicit conditions in are satisfied. Let the points \( x^b \) and \( x^g \) be defined by

\[
\begin{align*}
f(x^b) &= \max f(x^1), f(x^2), \ldots, f(x^k) \\
f(x^g) &= \min f(x^1), f(x^2), \ldots, f(x^k) 
\end{align*}
\]  

(5)

calculate the centroid \( \bar{x} \) of those points other than \( x^i \) by

\[
\bar{x} = \frac{1}{k-1} \sum_{j=1}^{k} x_j, \quad x^i \neq x^j
\]  

(6)

The main idea of the algorithm is to replace the worst point \( x^i \) by a new and better point. The new point \( x' \) is calculated as a reflection of the worst point through the centroid. This is done as

\[
x' = \bar{x} + \alpha(\bar{x} - x^i)
\]  

(7)

where the reflection coefficient \( \alpha \) is chosen according to Box as \( \alpha=1.3 \).

The point \( x' \) is examined with regard to explicit and implicit constraints and, if it is feasible, \( x^i \) is replaced with \( x' \) unless \( f(x^i) \leq f(x') \). In that case, it is moved halfway towards the centroid of the remaining points. This is repeated until it stops repeating as the lowest value. However, this cannot handle the situation where there is a local minimum located at the centroid. The method used here is to gradually move the point towards the maximum value if it continues to be the lowest value. This will, however, mean that two points can come very close
to each other compared to other points, with a risk of collapsing the complex. Therefore, a random value is also added to the new point. In this way, the algorithm will take some extra effort to search for a point with a better value, but in the neighbourhood of the point of the maximum value. It is consequently guaranteed that a point better than the worst of the remaining points will be found. Expressed as an equation

\[
x^{r(new)} = 0.5\left[ x^{r(old)} + c\bar{x} + (1 - c)x^h \right] + (\bar{x} - x^h)(1 - c)(2R - 1)
\]

(8)

where

\[
c = \left( \frac{n_r}{n_r + k_r - 1} \right)^{n_r + k_r - 1}
\]

(9)

and \(k_r\) is the number of times the point has repeated itself as lowest value and \(n_r\) is a constant. Here \(n_r=4\) has been used. \(R\) is a random number in the interval \([0,1]\).

If a point violates the implicit constraints, it is moved halfway towards the centroid. In order to handle the case of the centroid violating the implicit constraint, the point is gradually moved towards the maximum value. If the maximum value is located very close to the implicit constraint, this will take many iterations and the new point will be located very close to the maximum value and will not really represent any new information. Therefore a random value is also added in this case.

All effects are present and often they exert an opposing influence. The geometry of screw machines is dependent on a number of parameters whose best values to meet specified criteria can, in principle, be determined by a general multi-variable optimization procedure. In practice it is preferable to restrict the number of parameters to a few, which are known to be the most significant, and restrict the optimization to them only. More information on screw compressor optimisation is published by Stosic, Smith and Kovacevic, 2003.

5 Optimisation Constraints

The optimisation constraints, which are explicit, are presented in Table 1. Those, which are implicit, will be discussed further.

Since \(y\) is an independent variable for this type of generation, its constraints are considered explicit too. The \(y\) coordinate on the rack is a uniform and rising function for any rotor, therefore, this must be satisfied as a constraint. \(y\) must never decrease. Moreover, it must not be constant, because this will result in an unbounded or discontinuous gradient for \(dx/dy\).

Implicit constraints are imposed upon the optimised variable, which is \(x\), since \(y\) is used as an independent variable.

Specifying the outer and root diameters of the rotors sets limits to the permitted values of \(x\) since the values of these for both rotors are defined by the lowest and highest values of \(x\) on the rack. Thus, the points at the lowest value of \(x\) on the rack are at the rotor root of the main rotor and at the outer diameter of the gate rotor, while, the highest value of \(x\) corresponds to the main outer and gate root diameters. Moreover, the rack must start and end at the same value of \(x\).
Constraints are usually imposed upon the curvature radii of the profile to satisfy the requirements of the applied tools. In such a case, all small circles on the rack and on the rotor profiles are subjected to restrictions, which are defined by the tool manufacturers. Therefore the smallest radii at certain areas of the profile are given additional length constraints.

Additional constraints are requested by the manufacturing process. Namely, to mill or grind rotors with the minimum tool wear, the rotor pressure angle at the rotor pitch circles has to be as large as possible, therefore its minimum value should be defined in advance as an optimization constraint. This angular requirement is a demand upon the profile gradient, because the profile gradient \( \frac{dx}{dy} \) is a tangent of the pressure angle. This requirement is implicitly satisfied by rack generated rotors if a straight line defines the profile in the vicinity of the pitch circles, when the line slope angles \( \alpha_1 \) and \( \alpha_2 \) define the rack and rotor pressure angles.

6  EXAMPLE OF ROTORS GENERATED BY THE PRESSURE ANGLE METHOD

For the purpose of validation a fixed geometry generation of a 4/5-220 mm ‘N’ profile has been performed, once by using an ordinary and well proven procedure and the second time by use of the profile pressure angle generation method.

No difference was detected between the fixed profile coordinates generated by the standard method and by the pressure angle method up to the 8th significant digit, which was 0.01 \( \mu \text{m} \).

A rotor optimisation was conducted for the lowest specific power as a target function. As a result of the optimisation, very similar rotor geometry was obtained for both generation procedures, with a specific power difference between them of less than 0.5 %. This outcome
was expected, because the optimisation procedure used, searches for the local minimum in a stochastic manner and, therefore some discrepancy in the optimised result was not a surprise.

The rotors calculated by the fixed procedure and their optimised counterparts are presented in Fig 3 in dashed and bold lines respectively.

7 CONCLUSION

The pressure angle procedure was incorporated into a rotor profile generation routine and used to generate ‘N’ rotors. New optimisation constraints were determined to suit the pressure angle method and applied to optimisation of the screw compressor rotors. The results presented in this paper confirm that the pressure angle method may be considered consistent with the standard ‘N’ profile generation procedure and the constraints imposed upon the variables in the optimisation process were found to be adequate.

REFERENCES


Rinder L, 1984: Schraubenverdichterlaeuf er mit Evolventenflanken, Schraubenmaschinen 1984 Dortmund, VDI Berichte 521


The development of a hybrid 2 stage micro-turbo/water flooded screw compressor

Adrian Alford, Gerd Cromm
Corac Group plc, UK

ABSTRACT

The combination of a direct drive, high speed micro-turbo compressor upstream of and in series with a water flooded screw compressor can provide an effective and efficient compression system for the supply of compressed air between 6 and 15 barg. The ability of the micro-turbo compressor to provide a high volumetric flow at low delivery pressure combined with the high pressure capabilities of the water flooded screw compressor can result in an extremely compact machine. The variability in rotational speed available from the micro-turbo compressor drive allows a variable pressure and density flow to be deliverable to the screw second stage which, rotating at constant speed, will deliver a variable mass flow rate at fixed outlet pressure. The “aftercompression” ability of the water flooded screw compensates for the low compression ratio of the micro-turbo compressor at low flow rates and allows a constant pressure to be delivered at flow rates varying between 55 and 100% of maximum free air delivery (F.A.D.).

This paper describes the development and testing of such a machine.

1. INTRODUCTION

Both the direct drive, high speed micro-turbo compressor and the water-flooded screw compressor are machines which are expected to dominate the oil-free air compressor market in the future. Both offer the prospect of reduced lifetime costs due to a combination of reduced complexity and therefore purchase price, and reduced energy usage due to an increase in compression efficiency. Due to the constraints imposed by materials and manufacturing methods, the water-flooded twin screw compressor is currently limited to a maximum size approximately corresponding with a 90kW frame size compressor. The micro-turbo compressor, by contrast, has a minimum size due to constraints imposed by materials, shaft dynamics and flow physics, which manifests itself as a reduction in efficiency at low flow coefficients and small frame sizes. Though there is no consensus on the effective minimum size, it is true to say that the micro-turbo compressor will achieve its most competitive advantage at frame sizes of 250kW and above. This leaves an important size range which is imperfectly served by either machine. The hybrid, 2 stage micro-turbo/water-flooded screw
will very effectively serve the compressor size range between 90 and 300kW using existing compression elements from a sub 90kW water-flooded screw range and a 200kW plus micro-turbo compressor range.

The combination of a gear-driven centrifugal compressor followed by a two-stage dry screw compressor has previously been offered by a manufacturer (1).

The layout and flow details of the hybrid compressor considered in this study can be seen in figure 1.

![Figure 1 – Hybrid Compressor Layout and Flow](image)

The micro-turbo compressor is well suited to its role of first compression stage of the proposed machine due to its inherent strengths; firstly, its ability to compress high volumetric flow rates in a compact unit; secondly, a high adiabatic efficiency for compression ratios of 2 or 3:1; thirdly, the ability to supply a variable mass flow rate at varying pressure ratio by control of rotational velocity. The water-flooded screw is well suited to its role of final compressive stage due to its ability to accept close to a constant inlet volumetric flow at constant speed, and to efficiently compress this charge over a variable pressure ratio using either after compression or a variable volume ratio slide valve.

In the hybrid machine the screw is used as the final compression stage, where its inlet density can be up to 3 times as great as the atmospheric air it would ingest when used as a single stage compressor. As the second stage of a two stage compressor it will ingest approximately 3 times the mass flow rate of the single stage screw compressor on which it is based. Delivery mass flow can be varied by control of the speed of the centrifugal first compression stage. As speed is reduced from its maximum value, the pressure ratio of the centrifugal compressor will reduce, so reducing the density of the flow ingested by the screw final stage. As the screw is a positive displacement machine operating at a constant speed, it will therefore ingest a mass
flow rate which will vary with the density of the flow supplied to it. Though the pressure ratio of the screw must vary to achieve a constant outlet flow, this is well within its capabilities.

In order to understand the operation of the integrated system, it is necessary to understand the characteristics of both compressors. In the following sections, the salient features of the micro-turbo and the twin screw compressors are introduced, and the modifications necessary for their integration to form the two stage hybrid machine discussed. Finally, the characteristics of the completed hybrid unit are presented.

2. FLOW AND EFFICIENCY CHARACTERISTICS – MICRO-TURBO COMPRESSOR

Due to the small component sizes of the micro-turbo compressor, use of the centrifugal compressor is ubiquitous. The centrifugal stage will ingest, at constant inlet conditions and constant rotational velocity, a volumetric (and therefore mass) flow rate which varies with outlet pressure. There is a point of minimum flow rate caused by the onset of surge, and a maximum flow rate caused by diffuser choke. Both these points occur at varying inlet flow rate with varying rotational velocity of the impeller, both tending to occur at increasing inlet flow rate with increasing rotational velocity. The established method of showing the characteristics of a centrifugal compressor stage is for inlet flow rate and outlet pressure to be non-dimensionalised and values plotted for discrete values of corrected rotational speed. The flow characteristics of the current micro-turbo compressor centrifugal stage are shown in figure 2. The points of minimum inlet flow rate are defined by the surge line, while those of maximum flow rate are nominally defined by the choke line, though this is indiscrete. As compressor efficiency can drop to very low values towards choke, stages are rarely used in this operating region; the choke line is therefore not shown on figure 2. The isentropic efficiency characteristics of the stage are shown in figure 3. By reference to these plots, it can be seen that the centrifugal compressor stage can operate at close to its peak efficiency over a wide range of pressure ratios, delivering a variable mass flow rate by control of rotational velocity.

![Figure 2 – Flow Characteristics of Micro-Turbo Compressor Stage](image-url)
3. FLOW AND EFFICIENCY CHARACTERISTICS – SCREW COMPRESSOR

As the two-stage compressor considered here is to operate with a fixed speed water-flooded screw compressor as its second stage, only the characteristics of the device at a single speed will be considered. Screw compressors of all configurations suffer from leakage flows occurring in the reverse direction with respect to the overall flow rate. These manifest themselves in a reduction of both volumetric and isentropic efficiency with increasing leakage rate. Volumetric efficiency is adversely affected by reverse flow leakage as a proportion of the geometric volume trapped during the inlet part of the cycle is filled by the leakage flow, reducing the volume available to be filled by the inlet flow. As reverse flow leakage tends to increase as the pressure difference which drives it increases, the volumetric efficiency of a screw compressor tends to decrease with increasing pressure ratio.

Isentropic efficiency is also adversely affected by reverse flow leakage due to two major loss mechanisms. The first is a result of the energy wasted in compressing the leakage flow, which is dissipated as heat during the throttling process across the leakage path. The second is a result of the heating of the gas earlier in the compression process, which is being diluted with gas at a higher temperature, and will consequently require a greater energy input to be compressed over the ratio between its pressure at that position and the delivery pressure of the unit. This second effect will be ameliorated by the cooling effect of the injected water.

The screw compressor can be designed with a range of volume ratios, defined as the ratio of the trapped volumes before and after compression within the mesh of the rotors (2). Due to the increase in temperature during the compression process, the internal pressure ratio of the device will be greater than the volume ratio, provided the gas is not cooled below its inlet temperature during compression. For water flooded screw compressors, the temperature rise during compression will be limited by the high heat capacity of the water, and the pressure ratio will be closer to the volume ratio than with non-cooled screw compression processes such
as the dry screw compressor. For a series of water injected screw compressors with the ideal volume ratio and water injection rate for the required compression ratio, the isentropic efficiency reduces with increasing compression ratio for the range of pressure ratios applicable here. Though the compression process is extensively cooled and therefore approaches isothermal conditions, leakage flows increase the compression losses, and while increased compression ratio has the potential to increase the isentropic efficiency of a compression process with constant isothermal (and polytropic) efficiency, both the isentropic and therefore the isothermal efficiencies fall in the typical water-flooded screw compressor. It is therefore apparent that the water-flooded screw compressor, like the oil flooded screw compressor, can benefit from the inclusion of a second compression stage in the quest for increased compression efficiency.

The general characteristics of a family of water-flooded screw compressors with a series of volume ratios and fixed speed have been described. Though this situation can be approached in a single compressor by the use of a variable volume ratio slide valve, the water-flooded screw compressor used in this study was of a constant geometry. Though its internal compression ratio remains fixed, it is able to increase its overall compression ratio using external (or after) compression, a condition characterised by a pulsating flow at the outlet port of the screw compressor. When low external pressure ratios are utilised, the isentropic efficiency of the compression process will remain high. Though external compression ratios of above 1.5:1 will reduce the compressors isentropic efficiency, the losses will again be ameliorated by the water present.

Commercially available water injected screw compressors are single stage devices for the supply of compressed air of between 7 and 13 bar (3). The water injected screw used in this study was a 55 kW, 8 barg output pressure unit with an internal pressure ratio of around 8:1. For use as the second stage of a two stage machine it was therefore necessary for it to be modified to give a reduced pressure ratio. This was achieved by modification of the porting. From figure 4, the peak isentropic efficiency point shows that the water injected screw compressor has an internal pressure ratio of around 3.6:1. As can also be seen from figure 4, the water flooded screw can compress efficiently at pressure ratios from its internal pressure ratio up to an overall compression ratio of around 1.5 times this amount, using cooled external compression. Though external pressure ratios of above this value are possible, efficiency continues to fall. At overall compression ratios below its internal compression ratio, the flow will be throttled as it exits via the compressors outlet port, a high loss process resulting in a rapid reduction of isentropic efficiency with reduction in pressure ratio.

The screw compressor used in this investigation was undersized for this duty and as a consequence was run at a rotational speed of 4800 rpm. A larger screw compressor rotating at around 3000 to 3500 rpm would be expected to compress at a significantly improved isentropic efficiency. Water was injected into the screw compressor at a temperature of 35°C.
Though the screw compressor considered here is a constant displacement device running at a constant rotational rate, the requirement of a variable pressure ratio will result in a varying volumetric efficiency and therefore a slightly varying inlet volumetric flow at constant outlet pressure. As can be seen from figure 4, the volumetric efficiency of the modified water-flooded screw compressor varied between 89% at a pressure ratio of 3.5:1 to 87% at a pressure ratio of 6.1:1.

4. FLOW AND EFFICIENCY CHARACTERISTICS OF COMBINED MACHINE

Though the centrifugal compressor is the primary compression stage, it is constrained to provide only the flow that the fixed speed screw final stage can accept. Hence the demand line for the screw stage will define the overall machines flow characteristic. As intercooling is used between stages, the flow accepted by the screw stage is at a relatively constant temperature. To investigate the matching of the two stages it is therefore most appropriate to plot the characteristic graph of the centrifugal compressor using its outlet volumetric flow at the temperature after intercooling. Before the demand line of the screw compressor can be plotted on this graph, the effects of its particular use providing a varying pressure ratio need to be considered. As the mass flow rate of the machine will be varied by varying the interstage pressure, and the machine will be operated with a constant delivery pressure, the pressure ratio duty of each stage will be varying with mass flow rate. Thus when the centrifugal compressor is operating towards its highest pressure ratio, the screw compressor will be operating towards its lowest pressure ratio and its volumetric efficiency and, as a consequence volumetric flow rate, will be towards its maximum value. Hence the demand line will run approximately parallel to the surge line of the centrifugal stage, as shown in figure 5. The demand line can, with correct design, pass close to the peak efficiency points for the centrifugal compressor for the majority of the machines outlet flow range.
Due to the variance in pressure ratio with mass flow rate, the proportion of overall input power supplied to each stage will vary. The power input to the centrifugal first stage will increase approximately with the cube of its speed, as its pressure ratio will increase approximately with the square of its speed and its mass flow will increase with interstage density after cooling, which will be proportional to interstage pressure, assuming constant outlet pressure from the intercooler. Though the screw stage will be accepting a variable mass flow rate, it will be compressing it over a reducing pressure ratio as mass flow increases. Thus the power input to the screw stage remains approximately constant over much of the range of the coupled machine. It is only at the lowest flow rates that input power to the screw stage drops off, as can be seen from figure 6. Thus the screw stage motor will operate at close to its maximum efficiency over the machines entire range.
It can also be seen from figure 6 that the power drawn by the screw stage peaks at over 73kW. For this reason, the packages original 55kW motor was replaced with a 75kW unit. The drive belt system was not modified and was therefore over-stressed and was estimated to be dissipating a number of kilowatts of power as heat.

The efficiency of a compressor can be expressed by its specific power requirement in kW/m³/min as expressed in BS ISO 1217:1996 (4). In figure 7 this parameter is plotted against percentage of peak free air delivery for the combined machine. It can be seen that peak efficiency is realised at 95% of peak mass flow rate. This is due to the efficiencies of both the centrifugal stage and the screw compressor falling off slightly above this point. The pressure ratio of the screw reduces below its optimum value beyond this point, and throttling across the outlet port causes energy losses. Below 95% of peak mass flow rate, the pressure ratio of the screw will increase with decreasing mass flow rate. The overall efficiency of the combined machine will therefore drop slightly with reducing mass flow rate below 95% of its peak rate due to the reduction in efficiency of the screw stage as its pressure ratio moves away from its optimum value and external compression is increasingly used.

![Figure 7 – Efficiency Characteristics of Combined Machine (BS ISO 1217:1996)](image_url)

The peak efficiency point of the machine shows a specific power figure of 6.79 kW/m³/min. This figure includes all parasitic losses and is calculated from the 3 phase power entering the machine and outlet mass flow at 8 barg. It is expected that the use of a larger direct drive water injected screw with an improved profile and a lower rotational speed will result in an overall specific power figure of 6.4 kW/m³/min at 8 barg due to an increase in the peak isentropic efficiency of the screw stage from 77% to 82%.
5. CONCLUSIONS

Though the centrifugal and water injected screw compressors have very dissimilar characteristics, each was able to operate at high efficiency under all tested operating conditions of the hybrid machine. Free air delivery could be varied between 55 and 100% of its maximum value with the centrifugal stage operating at a consistent and safe distance from its surge line. Overall efficiency was very good, with improvements expected by the use of a larger, current generation water injected screw compressor. This would also enable a 132kW shaft power compressor to be built within the dimensions of the current unit (l-1400mm; w-750mm; h-2000mm) The hybrid, 2 stage micro-turbo/water-flooded screw will very effectively serve the compressor size range between 90 and 300kW, providing a compact, low cost of ownership compressor system above the predicted size range of water injected screws and below the optimum size range of centrifugal compressors.

REFERENCES

2) Mechanical Design and Efficiency of Screw Compressors – P. O’Neill – Paper presented to the Institute of Mechanical Engineers (Scottish Branch, Graduates Section) 23 Nov, 1966.
Development of micro screw compressor

K. Venu Madhav
ELGI Equipments, India

ABSTRACT

This work describes the design and development of a small and efficient screw compressor for industrial applications. Basically screw compressors are not well appreciated in industrial norms with reference to cost when the required flow rates are small. Also the size of the compressor has direct impact on the performance, smaller the compressor, lesser the efficiency hence limits the application to moderate and large flow rates. However, when noise, reliability and space requirements are the deciding factors for selecting a compressor, screw compressor fits well in to low flow applications. To meet the market requirements in this type of applications, a small screw compressor was designed developed and tested. Innovative design and manufacturing methods, advanced rotor profile etc., makes this compressor unique of its kind in terms of size cost and efficiency. This compressor was exhibited in the recent Hanover Compressor show and the response to this compressor was excellent. This compressor finds application in innumerable areas ranging from miniature electronic circuits; Power tools etc. to critical surgical applications.

1. INTRODUCTION

In general the positive displacement compressors are classified as reciprocating and rotary compressors. Reciprocating compressors are used for compressing air and other gases when desired pressures are high and flow rates are relatively low. For large flow rates and moderate pressure ratios screw compressors are the best choice. The size of the screw compressor has direct impact on the performance. For a constant tip speed, the displacement is proportional to the diameter square and the leakage and clearance volume, where applicable, are proportional to the diameter. As the size of the rotor comes down, at some point the leakages will dominate the displacement and hence the volumetric and adiabatic efficiencies become poor. These factors limit the screw compressor application when the flow rates required are relatively low. However, when the rotary motion is readily available as a supplement of the main process, or the noise, reliability and space are the deciding factors for selecting a compressor, the screw compressor fits well in to that application. To capture the market in this type of applications, a small screw compressor was designed developed and tested.
2. COMPRESSOR DEVELOPMENT

The specification for the small screw compressor is given for air delivery of 100–300 litres per minute at 7 bar discharge pressure with the target improvement in efficiency of 5 to 8% in comparison to a reciprocating compressor for same specification. The maximum working pressure is 12 bar. The reciprocating compressor is taken for comparison here, because there are no screw compressors available at that size.

The ’N’ profile with a 4/5 lobe combination was selected to meet the requirements.

The Design and Development of the compressors started from the estimation of the compressor size and its performance. Based on the extensive design calculations and comparisons on various possible profiles and lobe combinations, the 4/5 rotor configuration with ‘N’ profile was found to have the advantage of high displacement with a short sealing line together with a small blow-hole area, when optimised for medium pressure air compression. The combination of ‘N’ profile and 4/5 configuration resulted in a low torque to the gate rotor and the inbedded involute rotor contact caused low rotor surface stress, while the ‘negative’ torque on the gate rotor imposed the rotor contact along the straight flanks, which minimised the interlobe leakage path. Hence, the 4/5 rotor configuration with ‘N’ profiles was selected to meet the requirements. The rotor profiles were generated using the rack principle. The ultimate attention was paid to the interlobe clearance between the rotors.

The required flow range was thereby achieved by a compressor of 35 mm male rotor size and 28 mm female rotor size with 1.5 L/D ratio. These are shown in Fig. 1 together with a photograph of the micro rotary compressor as shown in Fig.3

![Fig 1. 4/5 ‘N’ rotors for the small screw air compressors](Image)

The main performance parameters are calculated and presented in Table 1.
Table 1 Predicted Performance of the Micro Rotary compressor for oil-flooded application

<table>
<thead>
<tr>
<th>Tip Speed (m/s)</th>
<th>Male Rotor Speed RPM</th>
<th>Discharge Pressure bar(g)</th>
<th>FAD m³/min</th>
<th>Vol Eff (%)</th>
<th>Power kW</th>
<th>Specific Power kw/m³/min</th>
<th>Dis. Temp Deg.C</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.8</td>
<td>4914</td>
<td>7.3</td>
<td>0.07</td>
<td>0.477</td>
<td>0.666</td>
<td>9.514</td>
<td>80</td>
</tr>
<tr>
<td>11.9</td>
<td>6005</td>
<td>7.3</td>
<td>0.09</td>
<td>0.498</td>
<td>0.833</td>
<td>9.256</td>
<td>80</td>
</tr>
<tr>
<td>14.1</td>
<td>7097</td>
<td>7.3</td>
<td>0.112</td>
<td>0.526</td>
<td>1.027</td>
<td>9.170</td>
<td>80</td>
</tr>
<tr>
<td>15.9</td>
<td>7971</td>
<td>7.3</td>
<td>0.164</td>
<td>0.687</td>
<td>1.403</td>
<td>8.555</td>
<td>80</td>
</tr>
<tr>
<td>18.0</td>
<td>9033</td>
<td>7.3</td>
<td>0.193</td>
<td>0.713</td>
<td>1.626</td>
<td>8.425</td>
<td>80</td>
</tr>
<tr>
<td>20.1</td>
<td>10096</td>
<td>7.3</td>
<td>0.223</td>
<td>0.737</td>
<td>1.865</td>
<td>8.363</td>
<td>80</td>
</tr>
<tr>
<td>22.2</td>
<td>11159</td>
<td>7.3</td>
<td>0.253</td>
<td>0.754</td>
<td>2.108</td>
<td>8.332</td>
<td>80</td>
</tr>
<tr>
<td>24.3</td>
<td>12222</td>
<td>7.3</td>
<td>0.283</td>
<td>0.771</td>
<td>2.41</td>
<td>8.516</td>
<td>80</td>
</tr>
</tbody>
</table>

2.1 Mechanical Design of the Compressor

On completion of the rotor profile generation and thermodynamic performance estimation, the component size and shapes, as well as the resulting force loads thus estimated. In addition, modern design concepts, such as late closing of the suction port and early exposure of the discharge port were included, together with improved bearing and seal specification, to maximise the compressor endurance and reliability.

Fig. 2 Cross sectional view of the Microrotory Compressor
The key factor for all screw compressor applications is the rotor design. This was fully followed in the design. Although advanced rotor profiles are a necessary condition for a screw compressor to be efficient, all other components were designed to enhance rotor superiority and therefore the full rotor advantage was achieved. Thus rotor to housing clearances especially at the high-pressure end, are selected properly to follow the minimal interlobe clearances. This in turn required either expensive bearings to be applied with smaller clearances or cheaper bearings with their clearances reduced to the acceptable value by preloading. The latter practice was chosen as the most convenient and economic solution.

A screw compressor, especially of the oil flooded type, which operates with higher pressure differences, is heavily loaded by axial and radial forces which are transferred to the housing by the bearings. Considering the low loads and space limitations, Deep groove ball bearings were selected to achieve a life of 100,000 hours. Also the distance between the rotor centre lines was in part determined by the bearing size and internal clearance. A cross sectional view of the compressor is shown in Fig 2.

Same oil is used for rotor flooding and for bearing lubrication but the supply to and evacuation from the bearings is separate to minimise the bearing friction losses. Oil is injected into the compressor chamber at the place where thermodynamic calculations show the air and oil inlet temperature to coincide. The position is defined on the rotor helicoid with the injection hole located so that the oil enters tangentially in line with the gate rotor tip in order to recover as much as possible of the oil kinetic energy.

Special attention was paid to minimise the flow losses in the suction and discharge ports. The suction port is positioned in the housing to let the air enter with the fewest possible bends and the air approach velocity is kept low by making the flow area as large as possible. The discharge port size was first determined by estimating the built-in-volume ratio required for optimum
thermodynamic performance. It was then increased in order to reduce the exit air velocity and hence obtain the minimum combination of internal and discharge flow losses.

The cast iron casing was optimally dimensioned to minimize its weight, after which the casting was hydraulically tested at a pressure of 22.5 bar(g).

2.2 Testing of the Compressor Prototype
Three compressor prototypes were made using existing manufacturing facilities with innovative methods and are validated against the specifications. The compressor performance is in line with the theoretical predictions and hence the industrial requirements. One of these compressors was packaged as shown in Fig. 4 and as that presented at the Hanover Compressor Exhibition. The reception of this compressor was very good and it generated many enquiries for the various applications. Considering the requirements, about 150 of these compressors are currently under manufacturing as a pilot lot and the compressor is dedicated for mass production.

Fig.4 : Micro Rotary Compressor Package: Length 500 mm, Width 275 mm, Height 300 mm
Table. 2 Comparison of the estimated and measured performance

<table>
<thead>
<tr>
<th>Tip Speed</th>
<th>Male Rotor Speed</th>
<th>Discharge Pressure</th>
<th>FAD</th>
<th>Power</th>
<th>Specific Power</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>m/s</td>
<td>rpm</td>
<td>bar(g)</td>
<td>Specified</td>
<td>Actual</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>m³/min</td>
<td>kW</td>
</tr>
<tr>
<td>9.8</td>
<td>4914</td>
<td>7.3</td>
<td>0.07</td>
<td>0.069</td>
<td>0.666</td>
</tr>
<tr>
<td>11.9</td>
<td>6005</td>
<td>7.3</td>
<td>0.09</td>
<td>0.089</td>
<td>0.833</td>
</tr>
<tr>
<td>14.1</td>
<td>7097</td>
<td>7.3</td>
<td>0.112</td>
<td>0.118</td>
<td>1.027</td>
</tr>
<tr>
<td>15.9</td>
<td>7971</td>
<td>7.3</td>
<td>0.164</td>
<td>0.169</td>
<td>1.403</td>
</tr>
<tr>
<td>18.0</td>
<td>9033</td>
<td>7.3</td>
<td>0.193</td>
<td>0.2</td>
<td>1.626</td>
</tr>
<tr>
<td>20.1</td>
<td>10096</td>
<td>7.3</td>
<td>0.223</td>
<td>0.215</td>
<td>1.865</td>
</tr>
<tr>
<td>22.2</td>
<td>11159</td>
<td>7.3</td>
<td>0.253</td>
<td>0.26</td>
<td>2.108</td>
</tr>
<tr>
<td>24.3</td>
<td>12222</td>
<td>7.3</td>
<td>0.283</td>
<td>0.294</td>
<td>2.41</td>
</tr>
<tr>
<td>26.4</td>
<td>13284</td>
<td>7.3</td>
<td>0.313</td>
<td>0.33</td>
<td>2.636</td>
</tr>
</tbody>
</table>

3. CONCLUSION

A result of a screw compressor development, presented in this paper, confirms that it is possible to produce a small efficient screw compressor which covers a range of small reciprocating compressors. The testing results confirm a superior performance of the micro compressor in that range and consequently a decision has been taken to start a serial production of the micro compressors.

REFERENCES


Konka K-H, 1988: Schraubenkompressoren (Screw Compressors) VDI-Verlag, Duesseldorf


Rinder L, 1979: Schraubenverdichter (Screw Compressors), Springer Verlag, New York

Sakun I.A, 1960: Vintovie kompresorii (Screw Compressors, Mashinstroyenie Leningrad


LINEAR/NOVEL COMPRESSORS
Problems and possibilities of the springless oscillating motor-compressor

S. Kudarauskas, R. Didziokas*, L. Simanyniene, A. Senulis
Department of Electrical Engineering, Department of Mechanical Engineering*, Klaipeda University, Lithuania

ABSTRACT

The usage of oscillating motor to drive piston compressor enables to avoid transformations of mechanical motion, but widespread design with the mechanical spring complicates the device. A mechanical spring can balance kinetic energy of moving bodies through potential energy, can balance continuous component of loading force of the compressor and stabilise oscillation centre position. Modelling and simulation of the device shows that it is possible to design springless structure, because the spring functions can be realised by oscillating motor itself. The oscillating synchronous pulsating current motor is suitable for the springless compressor.

NOTATIONS

\[\begin{align*}
a & \quad \text{acceleration (m/s}^2) \\
E & \quad \text{complex electromotive force (V)} \\
\bar{F} & \quad \text{complex driving force (N)} \\
f & \quad \text{force (N)} \\
f_c & \quad \text{resistant force of compressor (N)} \\
f_{fr} & \quad \text{resistant force of friction (N)} \\
f_i & \quad \text{force of inertia (N)} \\
f_m & \quad \text{driving force of motor (N)} \\
h & \quad \text{mechanical co-ordinate (m)} \\
i & \quad \text{current (A)} \\
I & \quad \text{complex current (A)} \\
K & \quad \text{constant of exciting (N\cdot A}^{-1}=\text{Wb}\cdot\text{m}^{-1}) \\
m & \quad \text{mass (kg)} \\
R & \quad \text{resistance (\Omega)} \\
R_c & \quad \text{mechanical resistance of compressor (N\cdot s\cdot m}^{-1}) \\
R_{fr} & \quad \text{mechanical resistance of friction (N\cdot s\cdot m}^{-1}) \\
U & \quad \text{complex voltage (V)} \\
u & \quad \text{voltage (V)} \\
V & \quad \text{complex velocity (m/s)} \\
v & \quad \text{velocity (m/s)} \\
X & \quad \text{reactance (\Omega)} \\
X_c & \quad \text{mechanical reactance of compressor (N\cdot s\cdot m}^{-1}) \\
X_{fr} & \quad \text{mechanical reactance of friction (N\cdot s\cdot m}^{-1}) \\
X_i & \quad \text{mechanical reactance of inertia (N\cdot s\cdot m}^{-1}) \\
X_m & \quad \text{mechanical reactance of spring (N\cdot s\cdot m}^{-1}) \\
X_s & \quad \text{mechanical reactance of spring (N\cdot s\cdot m}^{-1}) \\
x & \quad \text{electrical reactance (\Omega)}
\end{align*}\]

1 INTRODUCTION

The main benefit of the oscillating motor to drive positive displacement reciprocating compressor (piston or diaphragm) is related to the direct drive principle. In the direct drive devices properties of mechanical movement of the motor coincide with ones of the driving mechanisms, and any transformer of mechanical motion can be avoided. In this way, structure of the device becomes simpler, cheaper and mechanical losses are reduced.
Oscillating electrical machines are ones of periodical movement (that is, with specific temporal properties of mechanical movement), which is in co-ordination with the reciprocating movement of the compressor’s piston. Electrical machines could also differ by their spatial properties of the mechanical movement (a trajectory of the movable part). Thus, we can observe electrical machines of rotating, linear movement and machines with complex trajectory (1). It should be noted that spatial properties of movement are independent from the temporal properties. In most cases the oscillating linear motors are used to drive piston compressors, though oscillating rotating motor-compressors are applied too. That is why, the term linear compressor sometimes used for the considered devices is not correct. The term oscillating motor-compressor reflects more exactly the principal structure of the compressor driven by an oscillating electrical motor (2).

The compressor driven by an oscillating motor is free-piston device, that is, the piston stroke is not defined by mechanical means and it becomes variable. On the one hand, this circumstance gives supplement opportunity to control capacity of the compressor. But on the other hand, requirement to limit possible overstrokes of the piston, to control amplitude and centre of oscillation could arise. Moreover, in the direct drive of oscillating motor-compressor, we lose the possibility to utilise kinetic energy of rotating bodies of the conventional drive with continuous rotation. The stored kinetic energy is desirable to balance non-continuous nature of the loading force of piston device. Therefore a spring as potential energy storage element is desirable in the mechanical system of the considered device. This is one of reasons to use a spring (other reasons will be discussed further).

Unfortunately, the mechanical spring makes the device more expensive and reduces its reliability. This shortcoming markedly decreases benefits of the oscillating motor-compressor. Although springs are used in a great many of produced or proposed oscillating motor-compressors, the possibility to create springless device also exists. That is main reason to study possibilities and analyse problems of the springless oscillating motor-compressor.

2 STRUCTURES OF OSCILLATING MOTOR-COMPRESSOR

The considered device can differ by structure of the oscillating motor and structure of the compressor (in sense as a cylinder-piston set).

Almost without exception, oscillating synchronous motors are used to drive piston compressor (3). The strict relation between the frequencies of oscillation of mover and supply voltage (or voltages) determines the synchronism of the oscillating motor. Mostly these frequencies are the same or multiple. In fact, oscillating synchronous excited and so-called pulsating current motors are used in the oscillating motor-compressors.

The oscillating synchronous excited motors have two sources of magnetomotive force: the winding of AC and the field winding of DC (or permanent magnet). Operation of this motor is based by variable permeance of mutual inductance between the windings (1). An example of oscillating synchronous excited motor with two windings in the piston compressor drive is shown in Fig. 1a. The sources of magnetomotive force can have the different position with respect to stator or mover. In principle, the performance characteristics of motor are independent of the location of windings. Analogous motors with exiting permanent magnets are shown in Fig. 1b, 1c. Of course, there are various topologies of the oscillating synchronous excited motors, but presented principle of structure and operation is general.
The simplest oscillating synchronous pulsating current motor has one magnetomotive force, which is created by a winding with a pulsating current. The unidirectional pulsating current is formed by a semiconductor element (non-controlled or controlled). Operation of such motor is based on variable permeance of the magnetic path, which depends on position of the movable part. As a rule, a monotonous dependency is used, which causes continuous component of the driving electromagnetic force. Such motor driving a single-sided piston compressor is shown in Fig. 2a.

The doubled oscillating synchronous pulsating current motors are used too, as it is shown in Fig. 2b. The continuous component of pulsating current and its even harmonics of separate sections of the motor (the current $i_1$ and $i_2$ in Fig. 2b) could be balanced in the whole current $i$ of the motor. Besides, the continuous component of the driving force could be balanced too, which is very important in the springless drive design.
The main variants of the piston compressor are also presented in Fig. 2, that is, single-sided or double-sided compressors. The principal qualitative difference of these structures from standpoint of possibility of a springless drive is related to the existence or absence of continuous component of the resistant force (or loading force) of compressor. Thus, the continuous resistant force could be balanced in the double-sided compressor.

3 FUNCTIONS AND PROBLEMS OF THE SPRING

3.1 Functions

One of the spring functions in the oscillating motor-compressor, as it was mentioned in the introduction, is the compensation of kinetic energy of moving bodies during their reciprocating motion. From this point of view, often the considered devices are treated as resonance compressors (e.g. (4)), and this debatable term is associated with existence of a spring in the mechanical system (without taking into account the spring properties of the piston compressor itself). Moreover, often one affirms that the natural frequency of the pure mechanical system must be close to the frequency of oscillation (e.g. (5)).

Of course, the resonance phenomena are an intrinsic property of any system with dynamic mechanical elements as inertia and stiffness. However, first of all, the potential energy could be stored not only in the mechanical spring, but also during compression process of gas in the compressor. Consequently, the resonance phenomena depend not only on the compressor mechanical properties but on technological process of gas too. Sometimes the spring stiffness of the compressor is evaluated as some constant value of the equivalent spring stiffness (e.g. (4)). But in reality the equivalent compressor spring stiffness is non-linear, which depends on the suction and discharge pressures, and on the piston stroke (3). Moreover, the resonance phenomena of whole device depend as well on the parameters of elements of the electrical system of the drive. The following analysis of the oscillating synchronous excited motor can elucidate such a situation.

As it was mentioned above, operation principle of the oscillating synchronous excited motor is predetermined by dependence of the mutual permeance (between two sources of magnetomotive force) on the position (co-ordinate \( h \)) of a movable part. If this dependence are linear and permeances of self inductance of the separate magnetomotive forces is constant, we have a unique case, when oscillating motor can be described by linear equations (1). By the way, this case is very suitable to simplify control system of the motor, especially by realisation of the sensorless control (e.g. (5), (6)). In this case the considered device could be presented by corresponding variants of equivalent schemes, which are depicted in Fig. 3.

In Fig. 3a, the equivalent scheme with two-port electromechanical device is shown. Here the variables are presented by their complex amplitudes. Parameters of elements are evaluated by their resistances or reactances (under given frequency of oscillation). The mechanical system is presented by analogous electrical signs of elements, with respect to analogy current-velocity. The loading properties of compressor are presented by non-linear elements \( R_c \) and \( X_c \). The mechanical reactance \( X_m \) evaluates the inertia of all movable bodies.
Consider the aforesaid case when the oscillating synchronous excited motor is described by linear equations. In such a case the driving electromagnetic force is proportional to the alternating current and induced electromotive force is proportional to the velocity of a movable part (1):

\[ F = K \cdot I, \]
\[ E = K \cdot V, \]  

Taking into account these expressions, we have the electrical (Fig. 3b) and mechanical (Fig. 3c) equivalent schemes (here $F_{BL}$ is the complex driving force of the blocked motor). The grey background emphasises “imported” elements of other physical nature in the electrical and mechanical equivalent schemes. These schemes obviously show that resonance phenomena of the device depend on all reactive elements of different nature, but not only on the pure mechanical dynamic elements (spring and inertia). On the other hand, these schemes show that spring reactance $X_s$ is not obligatory balancing kinetic energy, because other existing elements (spring reactance of the compressor $X_c$ and electrical reactance $x$ of the motor) could accomplish this function. This circumstance shows the possibility of the springless drive design of the device through realisation of the stored energy balance by all conservative elements of the system.

The second function of the mechanical spring is the compensation of the continuous component of resistant force of the single-sided compressor. Such situation is shown in Fig. 2a. Though the continuous component of driving force of the asymmetric pulsating current motor in principle could balance corresponding component of the compressor force, the realisation of such balance is complicated. Therefore single-sided compressor practically requires a mechanical spring. Consequently, the springless device is possible when oscillating motor drives a double-sided compressor.

The third function of mechanical spring in oscillating motor-compressor is related to the problem of stabilisation of oscillation centre. Though continuous components of compressor force of double-sided compressor could be balanced, more exhaustive analysis of the piston compressor properties shows that this balance is not stable from the point of view of the position of oscillation centre (3). In some cases of the springless compressor, a drift of oscillation centre could take place, and the asymmetrical operation of the double-sided
compressor cylinders could be observed, as it will be shown in further sections. The mechanical spring can limit the undesirable drift of oscillation centre. This spring function could be replaced by corresponding properties of itself oscillating motor.

### 3.2 Problems
Functions of the mechanical spring presented in previous section are very important for good operation of the oscillating motor-compressor. However, there are two essential shortcomings of the device structure with a mechanical spring: the spring makes the device more expensive and spring reduces reliability of the device.

Historical development of the oscillating motor-compressor obviously confirms problems caused by a mechanical spring. For example, the spring problem caused failure of refrigerators of the *Chausson Inc.* that had been produced in the 1955-66. The abundant number of patents with different variants of the spring confirms efforts of this incorporation to find a suitable and reliable form of the spring (e.g. (7)).

The experience of Korean firm *LG Electronics* also confirms problems of the spring reliability. Though they started to study an oscillating motor-compressor with a spiral spring, which was proposed by *Sun power Inc.* (5), (8), and which is used by other authors (e.g. (9)), later a helical spring (10) or eight smaller helical springs (11) were tried to use.

Sometimes the gas spring instead of the mechanical spring in the single-sided compressor drive is proposed (12). Unfortunately, this attempt is not rational. In this case usage of double-sided compressor should be a more successful solution.

Solving the problem of the mechanical spring reliability, special material is proposed (e.g. the titanium-copper alloy for the spiral spring (9)). This decision makes the device more expensive, though authors unfoundedly declare its cost being low.

These circumstances induce searches of conditions of the springless oscillating motor-compressor. Modelling and simulation of the springless device disclose some conditions and requirements.

### 4 MODELLING AND SIMULATION OF THE SPRINGLESS OSCILLATING MOTOR-COMPRESSOR

Modelling and simulation of different variants of oscillating motor-compressor were carried out by *MathCAD* software. The theoretical indicator diagram evaluated properties of the compressor.

#### 4.1 Oscillating synchronous excited motor
The oscillating synchronous excited motor can be more easily controlled, but its main advantage is the possibility to achieve high efficiency, especially if the permanent magnets are used as a source of exciting magnetomotive force. When the considered oscillating motor can be correctly described by linear equations, the motor does not acquire properties, which could stabilise the centre of oscillation. The simulation results confirm such situation.
The curves of variables of the starting of double-sided compressor are shown in Fig. 4. As we can see, the steady oscillation is achieved after the few cycles at start-up mode. Though in the curve of acceleration $a$ higher harmonics are visible (they are caused by non-linear properties of the compressor), variation of co-ordinate $h$ is close to sine wave. This situation is related to inevitable inertia of the moving part, which damps higher harmonics of oscillation. Also we can observe a dangerous approach of the displacement to the limit value $H_{\text{max}}$ during the transient process (see grey circle in Fig. 4). So, this motor must be protected from overstrokes.

In Fig. 5 the cyclograms of forces in a steady state operation of the considered oscillating motor-compressor are shown. The cyclograms express relationships between variables of oscillating system: between corresponding forces $f$ and position of the movable part of motor and compressor piston $h$ in the considered case. If the oscillating system has linear characteristics, the cyclograms should be shaped as ellipses. In a case of sine-shaped oscillation of linear system, a phasor diagram could present the same information about mutual relationship of variables as cyclograms. In the general case, the cyclograms are more
informative in comparison with phasor diagrams, because they reflect non-linear properties of the system.

The device operation can be stable with the spring, therefore we can observe corresponding force of the spring \( f_s \) in Fig. 5. Also we can see complex cyclogram of the inertia force \( f_i \), which expresses corresponding balance of non-linear resistant force of the compressor \( f_c \).

![Fig. 6 The plots of the variables of asymmetric regime of the compressor driven by oscillating synchronous excited motor (a) and cyclogram of the compressor force (b)](image)

Operation of the springless variant of this oscillating synchronous excited motor could have undesirable drift of the oscillation centre, if the accidental continuous component of the force arises (for example, the force of gravity, unbalanced force of the compressor etc.). Sometimes the asymmetric oscillation could settle, when one of cylinders acts without a gas discharge. We can observe such situation in the Fig. 6. Of course, this is poor operation of the compressor. Consequently, the springless operation of the oscillating synchronous excited motor, which was presented in Fig. 3, is problematic. It signifies that it is necessary to stabilise oscillation centre of the oscillating synchronous excited motor.

### 4.2 Oscillating synchronous pulsating current motor

Modelling and simulation of the springless compressor driven by doubled oscillating synchronous pulsating current motor (see Fig. 2b) was carried out. The model of the motor evaluates opposite monotonous variation of the permeances (or inductances) of both sections of the motor, when the position of a movable part changes.

Very important property of the considered motor is that mean permeances of both motor sections are equal in symmetrical oscillation mode with constant position of oscillation centre. If the oscillation centre moves to one direction (e.g. to the right side), the mean permeance (and inductance) of one section (of the right section) increases and of the other section decreases. The increased inductance diminishes current of the section, consequently, the mean value of the attractive force of this section decreases too. The result is that the unbalanced mean force of both sections can arise that stabilises the oscillation centre position.

The typical results of modelling and simulation are shown in Fig. 7 as curves of variables during the starting of the device. The steady oscillation after a few cycles of operation at start-up mode without over-strokes during the transient regime is observed.
Fig. 7 The start-up plots of the double-sided compressor driven by oscillating synchronous pulsating current motor

Fig. 8 Cyclograms of the forces at steady operation

The cyclograms of forces of the steady oscillation are presented in Fig. 8. The modelling and simulation results show good possibilities of realisation of a springless design of the oscillating motor-compressor with the synchronous pulsating current motor.

6 CONCLUSIONS

The mechanical spring in the oscillating motor-compressor can balance kinetic energy of moving bodies through transformations it in potential energy, can balance continuous component of loading force of the compressor and stabilise oscillation centre position.
The mechanical spring complicates the device, makes it more expensive and decreases reliability of the oscillating motor-compressor.

The springless oscillating motion motor-compressor is possible in the case of double-sided compressor. The oscillating synchronous pulsating current motor is more suitable for the springless device in comparison with the oscillating synchronous excited motor.

It is possible an expedient to create springless oscillating motor-compressors.

REFERENCES

Heat and fluid flow in a free piston stirling refrigerator

J. W. F. Heidrich, A. T. Prata  
Department of Mechanical Engineering, Federal University of Santa Catarina, Brazil  
D. E. B. Lilie  
Empresa Brasileira de Compressores S. A. – EMBRACO, Brazil

ABSTRACT

The present work aims to develop a mathematical model for numerical simulations of free piston Stirling coolers. Stirling refrigeration cycles are found on some applications such as low operating temperature coolers, medical diagnostic equipments and sophisticated electronics (superconductivity components and devices). In this work, it is explored a Stirling cooler machine for domestic refrigerators, as an alternative for conventional hermetic compressors and vapor compression cycles. The model explores the working gas thermodynamic behavior and evaluates the performance of the Stirling cooler components. To simulate the working fluid behavior inside the chambers, use was made of a global thermodynamic model. To model the heat exchanger and the regenerator a discretized approach was adopted using the finite volume methodology. Dynamic equations were used to simulate the system composed by the piston, displacer, and springs. An electric circuit represented the linear electric motor and its components. A computational code was generated to simulate the refrigerator and to help in the design process of new coolers and machines. Validations of the results obtained by comparisons with experiments indicated consistence of the physical and mathematical model. As explored, the Stirling cooler reached values of performance comparable to conventional hermetic compressors.

1. INTRODUCTION

Stirling engines and refrigerators are known since the nineteenth century (1) and recent advances have pushed its applications to a diversity of areas including domestic refrigeration (2). In this regard the present work will be exploring a linear free piston Stirling refrigerator as an alternative for replacing conventional hermetic compressors commonly employed in vapor compression cycles.

Stirling refrigerators are reciprocating machines in which the working fluid is a pressurized gas such as helium. The working fluid is shuttled between hot and cold chambers by means of a displacer and the change in temperature experienced by the gas is provided by the oscillating movement of a piston that converts the pressure wave (mechanical energy) into a temperature wave (thermal energy). The pressure and temperature variations are of constant amplitude and
the period depends on the electrical frequency of a driven electric motor. A key feature of the Stirling refrigerator in particular and of the Stirling machine in general is the resonance frequency of the thermal pendulum.

Figure 1 presents a schematic view of the β-configuration linear free piston Stirling refrigerator employed here. As shown, the basic parts are the electric motor, springs, piston, displacer, regenerator, the hot and cold chambers, and the hot and cold heat exchangers.

The electric motor drives the piston that compresses and expands the gas inside the chambers, and the springs (one attached to the piston and the other to the displacer) allow the operation at the resonance frequency. Referring do figure 1, as the piston moves to the left, the gas volume is decreased and the gas temperature is increased. The piston displacement to the left also causes the displacer displacement to the right which tends to push the gas from the hot chamber to the hot heat exchanger. Thus, the displacer is responsible for moving the gas from the chambers to the heat exchangers. The piston is free to move with respect to the rod which is attached at its left to the displacer and at its right to one of the springs (the other spring is attached to the piston). Pressure in both chambers tend to be uniform throughout the entire cycle and the difference in area at the left and right sides of the displacer, caused by the rod cross sectional area, creates a resultant force in it which points to the right when the piston moves to the left and vice versa. The regenerator storage the thermal energy during the operation of the cycle and acts as a thermal barrier between the hot and cold ends of the machine. When the working gas is displaced from the hot to the cold chamber, due to the displacer movement, heat is transferred from the gas to the regenerator and thermal energy is temporarily stored in the regenerator matrix. When the working gas is displaced from the cold to the hot end of the machine, the stored thermal energy is released from the regenerator matrix to the gas. Piston and displacer lubrication is performed aerostactically, using the gas itself. In the present study a 100 W cooling capacity machine will be explored but the methodology can be easily extended to other machines.
2. MATHEMATICAL MODEL

Considerable work has been performed on the cycle analysis of the Stirling machine since the classical first order analysis due to Schmidt (3). Those analyses are divided into two main categories by Organ (3), engineering thermodynamics approach and flow-field approach. In either approach numerical calculations are employed to obtain the cycle performance for a given operating condition. Uriel and Berchowitz (4) modeled a free-piston Stirling machine as a generalized single degree of freedom system and derived general relations for piston-displacer amplitude ratio and phase angle in terms of the machine operating frequency. The system dynamics approach of Uriel and Berchowitz was further improved by Huang and Lu (5) that derived two transfer functions for the free displacer Stirling refrigerator, and by Huang and Chen (6) that derived a transfer function model for the system performance analysis of the integral-type Stirling refrigerator. In order to capture the irreversibilities of the Stirling cycle, Kaushik and Kumar (7) performed a finite time thermodynamics analysis of a general Stirling heat engine cycle. For completeness it is worth mentioning the contributions of Ataer (9) that employed a Lagrangean formulation in the numerical analysis of regenerators of free piston Stirling engines, so that neither time nor frequency appear in the governing equations.

The present work differs from the previous one for presenting a detailed analysis of variables such as pressure, velocity, temperature, and density, along the regenerator and as a function of time. Pressure drop and heat transfer within and along the regenerator allow for a more realistic condition and the influence of the regenerator thermal conductivity, heat capacity and density can be inferred from the model. Furthermore, the full dynamics of piston and displacer are taken into account, together with the performance of the linear electric motor.

2.1 Working fluid

Helium is the working fluid and its thermodynamic properties are assumed to be uniform inside the hot and cold chambers, varying only with time. Conservation equations for mass, momentum and energy are employed to describe the gas behavior during the operating cycle. For the temperatures and pressures encountered here, helium can be assumed an ideal gas with

\[ R = 2077 \, \text{J/kg/K}. \]

Ignoring kinetic and potential contributions, the energy equation for the gas inside the chambers can be written as

\[ \dot{Q} + \dot{W} + \dot{m}_h h_h = \dot{m}_o h_o + d(\text{me})/dt \]

(1)

where \( \dot{m} \) is the mass flow rate going in or out of the chamber, \( h \) is enthalpy and \( e \) is internal energy. The chambers are taken adiabatic, that is, \( \dot{Q} = 0 \) and the mechanical power is given by

\[ \dot{W} = -p \frac{dV}{dt} \]

Mass conservation for the gas inside the chambers are expressed by

\[ \frac{dm}{dt} = \dot{m}_o - \dot{m}_i \]

(2)

As will be explained later, the mass flow rate going in or out of the chambers is determined from the average velocity through the regenerator which is function of the pressure at its hot and cold extremes.
2.2. Heat exchangers
The heat exchanger employed in the present configuration is a metallic strip folded in several layers to form a corrugated structure that can be approximated by a stack of parallel plates channel, as seen in figure 2. The gas temperature distribution along the flow direction, x, was obtained from

\[ m c_p \frac{dT_g}{dx} = h_{ex} P [T_w(x) - T_g(x)] \]  \hspace{1cm} (3)

where \( T_g \) and \( T_w \) are, respectively, the gas and the wall temperature at a particular streamwise location; \( h_{ex} \) is the heat transfer coefficient and \( P \) is the overall perimeter. The channel wall temperature, \( T_w \), depends on the external temperature of the heat exchanger and on the conduction mechanism along its parts. For simplicity, in the present investigation the external temperature of both heat exchangers will be prescribed, and the channel wall temperature will be taken as an average between those temperatures and the gas temperature.

At a given time the local mass flow rate \( m \) along the heat exchanger is taken to be constant. From its value and the overall gas temperature difference between entrance and exit, the instantaneously heat transfer at each heat exchanger can be obtained.

2.3. Regenerator
The regenerator in the present work is a plastic strip wrapped in spiral form around the cylinder as shown in figure 2. The volume occupied by the gas in the regenerator is of the same order of magnitude of that in the chambers and the gas is subjected to density variations along the working cycle. Within the regenerator the flow is considered to be two dimensional and compressible between two parallel plates; curvature effects are negligible and can be disregarded since the average radius of curvature of the plates is of the order of \( 10^{-1} \) m and the distance between two plates is of the order of \( 10^{-4} \) m.

\[ \text{Figure 2 – Schematic view of (a) heat exchanger and (b) regenerator.} \]

Conservation of mass and momentum for the gas inside the regenerator can be written as, respectively,

\[ \frac{\partial p}{\partial t} + \frac{\partial (\rho u)}{\partial x} = 0 \hspace{1cm} \rho \frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} = - \frac{dp}{dx} + \frac{4}{3} \mu \frac{\partial^2 u}{\partial x^2} - 12 \mu \frac{u}{a^2} \]  \hspace{1cm} (4)

where \( u \) is the local velocity component along the flow direction integrated over the cross sectional area of the channel, and \( a \) is the channel height. The right most term on the second equation represents the viscous contribution that is obtained under the assumption of a parabolic velocity profile within the channel, which is a laminar assumption justified due to the
small Reynolds number encountered between the channel walls. It should be noted that $u$ varies with time and along the longitudinal direction. To obtain the overall mass flow rate through the regenerator at a particular location, the local mass flow rates at each channel are added up considering that density, average velocity and area are the same for all channels.

The density variation with temperature and pressure requires that the energy equation for the gas should also be considered. For a compressible one-dimensional flow, the temperature within the regenerator satisfies the following equation,

$$c_v a W \frac{\partial (\rho T_g)}{\partial t} + c_p \frac{\partial (m T_g)}{\partial x} = h_{\text{reg}} W [(T_{\text{reg}}(x) - T_g(x))]$$

where $W$ is the total width of the plastic strip employed in the regenerator, $m$ is the local mass flow rate, and $T_{\text{reg}}$ is the regenerator wall temperature at a given location obtained from the following conduction equation

$$\rho_{\text{reg}} c_{\text{reg}} \frac{\partial (T_{\text{reg}})}{\partial t} = k_{\text{reg}} \frac{\partial^2 (T_{\text{reg}})}{\partial x^2} + \frac{h_{\text{reg}}}{b} ([T_g(x) - T_{\text{reg}}(x)])$$

in which $b$ is the thickness of the plastic strip.

Equations (4) to (6), in addition to the ideal gas law, form a complete the set of five equations required to calculate $\rho$, $u$, $p$, $T_g$, and $T_{\text{reg}}$. Those equations are discretized and solved numerically following the finite volume methodology (9).

2.4. Linear electric motor

The linear electrical motor is modeled assuming a simplified electric circuit where the resistance, $R$, the inductance, $L$, and the actuator voltage, $V_m$, are in series and fed by an electric voltage, $V$,

$$V = R i + L \frac{di}{dt} + V_m$$

where $i$ is the electric current through the motor. The actuator voltage can be written as,

$$V_m = \alpha \frac{dx_p}{dt}$$

in which $x_p$ is the instantaneous piston location, and $\alpha$ is the actuator constant. The magnetic force, $F_m$, that drives the piston is obtained from, $F_m = i \alpha$.

To obtain full benefit from the mechanical energy generated by the motor, the magnetic force should be in phase with the piston displacement. Further information about the electric motor and the electric circuit can be found in (10).

2.5. Piston and displacer dynamics

Ignoring spring damping, the piston and displacer equations can be written as

$$F_p = k_p x_p + m_{p+a} \ddot{x}_p$$

25
\[ F_d = k_d x_d + m_d \ddot{x}_d \]  (10)

where \( F \) is the resultant force, \( k \) is the spring stiffness, \( m \) is mass, and \( x \) is the location; subscripts \( p \) and \( d \) apply for the piston and displacer, respectively; \( m_{p+a} \) is the mass for the piston and actuator and \( m_d \) is the displacer mass.

The resultant forces acting on the piston and actuator, respectively, are

\[ F_p = F_m - p_h (A_p - A_r) + p_o (A_p - A_r) \]  (11)

\[ F_d = -p_c A_d + p_h (A_d - A_r) + p_o A_r \]  (12)

where \( A_p, A_d \) and \( A_r \) are, respectively, the areas of piston, displacer, and rod, and \( p_c, p_h, \) and \( p_o \) are, respectively, the pressure in the cold and hot chamber, and the pressure at the back chamber. The back chamber is located behind the springs and remains at a constant pressure along the operating cycle, equal to the cycle average pressure. The working gas inside the back chamber has a small influence on the refrigerator performance.

Equations (9) and (10) allow the calculation of the piston and displacer locations as a function of time once the thermodynamic and fluid flow quantities have been determined. Due to the coupling among all the governing equations, the Euler method was employed and the solution proceeded explicitly at each time interval. A time step of \( 10^{-5} \) s was adopted for all simulations and a converged periodic solution was considered to be established when the heat transfer at the heat exchangers and the mechanical power delivered by the piston did not altered at the second decimal figure.

3. RESULTS AND DISCUSSIONS

As pointed before, the Stirling refrigerator under investigation has a cooling capacity of 100 W. The chosen machine and operation conditions were used to explore the thermodynamic properties of the gas during a working cycle, as well as the heat and fluid flow at the different parts. Except otherwise noted, all results to be presented refer to a single baseline condition. Other conditions were also investigated to compare results obtained using the present model and experiments, and also to allow comparisons with refrigeration systems that make use of conventional vapor compressor cycles.

Results for the instantaneous heat transfer at both hot and cold heat exchangers are presented in figure 3. In this figure, as well as in the other figures to be presented, two working cycles will be shown for the periodic regime. The rejected heat from the gas to the ambient is taken as positive whereas the heat received by the gas from the ambient is negative. Time is represented in the abscissa by the angle in degrees which values serve only to indicate the periodic nature of the heat transfer, and that both heat transfers are out of phase. Figure 1 also indicates the cold and hot stages. The cold stage begins at 95° and finishes at 281°, lasting 186°, and the hot stage begins at 237° and finishes at 411°, lasting 174°. Interesting to note is the overlapping between the two stages which is made possible due to the compressibility effect, that is, at a given time mass can simultaneous enter at both ends of the regenerator.
Figure 3 – Heat transfer at the hot and cold heat exchanger.

Figure 4 – Mass present at both hot and cold chambers and at the regenerator.

Figure 4 presents the instantaneous gas mass stored at both hot and cold chambers and at the regenerator. The total gas mass stored at those three components is approximately 200 mg, which is not the total mass stored in the refrigerator because some mass is also stored at the back chamber. From figure 4 it can be observed that the gas mass presented in one chamber does not reach the other chamber because of the great amount of gas presented in the regenerator. This is a positive influence on the performance of the Stirling refrigerator since a fluid particle does not need to travel the entire regenerator going from one chamber to another, which would otherwise cause a large pressure drop and a large temperature variation.
Figure 5 - Volumes occupied by the gas inside the hot and cold chambers.

Time variation of the volume occupied by the gas in the hot and cold chambers is presented in figure 5. The average volumes are 6.67 cm$^3$ and 13.74 cm$^3$, respectively, for the hot and cold chambers, and the gas volume in the regenerator is, approximately, 26 cm$^3$. The phase shift observed between the peaks of maximum volume in figure 5 is of fundamental importance for obtaining the refrigeration effect and its adjustment directly affects the refrigerator performance.

Figure 6 – Comparison between model and experiment for the Stirling refrigerator.

For the present configuration and heat exchanger temperatures, the optimum performance is achieved when the displacer oscillation is 75° advanced with respect to that of the piston and the ratio between both total displacements is 0.73. For temperatures of hot and cold heat exchangers equal to 35°C and -15°C, respectively, after adjustment for higher performance the values of phase shift and displacement ratio obtained from the simulation were 65° e 0.85, respectively, which agreed well with experimental results that yielded 60° and 0.87.
Further comparisons with experiments were performed for the COP (ratio between cooling capacity and electric power) considering different temperatures at the hot heat exchanger. Figure 6 presents the refrigerator COP as a function of the temperature at the hot heat exchanger having the temperature at the cold heat exchanger, $T_c$, as a curve parameter. It should be noted that the COP values of figure 6 make direct use of the temperatures of the heat exchangers without considering the refrigeration system. The simulations were performed for a cooling capacity of 100 W and the data indicated in the figure represent experimental results obtained from prototypes treated at EMBRAÇO. As can be observed, large deviations occurred between calculated and measured results. Those deviations reflect the absence of thermodynamic and mechanic losses in the theoretical model develop here. The introduction of those losses is a further step that is beyond the scope of the present work which main purpose was to discuss the Stirling performance and viability for domestic refrigeration.

Results obtained with the present model were also compared with data for a refrigerator that operates with hermetic compressor in a conventional vapor compression cycle. To this extent figure 7 was prepared for a cooling capacity of 100 W. Comparison on a same basis required that for the Stirling refrigerator arbitrary temperatures should be prescribed between the heat exchangers and the hot and cold environments. For the hot environment a temperature difference of 15 °C was employed, and for the cold environment the difference was 10 °C. Those temperature differences represent further inefficiencies associated to the refrigeration system. Again, it should be noted that the results for the Stirling refrigerator were obtained without considering the existing thermodynamic and mechanical losses. Had those losses being included, the performance of the Stirling refrigerator would be pushed downward, deviating even more from the conventional vapor compression cycle.

Despite the difference in performance between the conventional Rankine and the Stirling cycles, the results indicate that the Stirling refrigerators have potential for domestic applications and further developments are expected to bring its performance more and more competitive with other available technologies.
4. REFERENCES


Modeling and simulation of a pneumatic piston for reciprocating hermetic compressors

P. R. C. Couto, A. T. Prata
Department of Mechanical Engineering, Federal University of Santa Catarina, Brazil

D. E. B. Lilie
Empresa Brasileira de Compressores – EMBRACO, Brazil

ABSTRACT

This work deals with modeling and numerical investigation of a pneumatic lubrication system for the piston in hermetic compressors employed in domestic refrigerators. This piston lubrication adopts a new concept where small holes located on the cylinder walls allow the refrigerant to return from the compressor discharge providing a gas film within the radial clearance between piston and cylinder. The mathematical model includes the dynamic behavior of the piston displacement, and takes into account both axial and radial misalignments. A finite element methodology is employed to solve the Reynolds differential equation that governs the lubrication phenomenon. Different geometric parameters and operating conditions are investigated. Results for the piston orbits, lubricant mass flow rates, and pressure fields within the clearance between piston and cylinder skirt, are presented and discussed. With a gas leakage of 5% of the total discharge mass flow rate of the compressor the pneumatic system was able to sustain the load and lubricate the piston.

1. NOMENCLATURE

c - radial clearance between piston and cylinder, [m]
h - local radial gap between piston and cylinder, [m]
L - cylinder length, [m]
p - pressure, [Pa]
R - piston radius, [m]
\( R \) - universal gas constant, [J/(kg.K)]
U - piston axial velocity, [m/s]
\( V_{R+h} \) = 0 - cylinder radial velocity, [m/s]
\( V_R \) = -\( \partial h/\partial t \) - piston radial velocity, [m/s]
x - coordinate according to figure 3, [m]
y - axial coordinate in Reynolds equation; also coordinate according to figure 3, [m]
z - coordinate according to figure 3, [m]
\( \varepsilon_x, \varepsilon_y \) - instantaneous eccentricity ratio according to figure 3, [-]
\( \mu \) - lubricant viscosity, [Pa.s]
\( \theta \) - circumferential coordinate in Reynolds equation, [rad]
\( \rho \) - lubricant density, [kg/m^3]
2. INTRODUCTION

Hermetic compressors are widely employed in industry and domestic refrigeration systems. Common types of such compressors make use of the reciprocating movement of a piston caused by the rotation of an eccentric shaft coupled to the piston through a connecting rod. In addition to the axial movement, the connecting rod also transmits to the piston undesirable secondary and tertiary motions perpendicular to the piston primary displacement along its axis. The existence of those secondary and tertiary motions and the associated resultant forces and moments, require a careful lubricating systems which, in general, makes use of oil as the working fluid.

Recently, use has been made of a new family of compressors where the reciprocating motion is achieved via a varying electromagnetic field, eliminating the connecting rod mechanism. In those linear systems, a significant reduction is observed in the radial loads allowing for alternative lubrication systems where a gas, and not oil, is the lubricating fluid. The possibility of operating without oil reduces the compressor cost and avoids oil contamination in the refrigerating circuit. Figure 1 presents schematic views of the connecting rod and the linear reciprocating mechanisms.

![Figure 1 – Reciprocating mechanisms for the piston; (a) connecting rod (b) linear.](image)

A natural choice for the lubricating fluid in a linear reciprocating mechanism is the refrigerating gas. In this regard, the present investigation introduces a lubrication system, known hereafter as pneumatic piston, where a fraction of the discharge gas is returned to the cylinder through a network of small channels and is injected in the radial clearance between piston and cylinder forming a gas cushion that precludes the contact between piston and cylinder wall. In what follows a detailed description of the pneumatic piston will be given and a mathematical formulation for the problem will be presented.

3. PROBLEM DESCRIPTION AND FORMULATION

The geometry under investigation is shown in figure 2. On the left hand side of the figure cylinder and piston are presented as they are assembled, and on the right hand side a detailed view of the cylinder is depicted showing the leaking orifice, the restrictors and the feeding
holes. The leaking orifice is connected to the compressor discharge and through it refrigerant is fed to the lubricating system via a network of small channels and restrictors. Eight feeding holes supply the gaseous lubricant to the clearance between piston and cylinder. The restrictors play a major role in the performance of the pneumatic piston and have to be carefully designed to introduce adequate pressure drops and thus provide a self-compensating mechanism for the continuum inflow of gas into the fluid film.

3.1 Governing equations for the pressure field

To predict the piston dynamics in the reciprocating motion and to establish the piston orbit within the cylinder, the pressure field in the radial clearance is required. As a usual approximation for this type of problem (1), the fluid film is taken to be two dimensional and the curvature effects are neglected. Taken the y coordinate in the radial direction and the θ coordinate in the circumferential direction, mass and momentum conservation require that pressure satisfies the Reynolds equation according to,

\[
\frac{1}{R^2} \frac{\partial}{\partial \theta} \left( \rho h^3 \frac{\partial p}{\partial \theta} \right) + \frac{\partial}{\partial y} \left( \rho h^3 \frac{\partial p}{\partial y} \right) = 12\mu \frac{\partial}{\partial y} \left( \rho h \frac{U}{2} \right) + 12\mu \left( \rho V_{R+h} - \rho V_R \right)
\]

where \( \rho \) and \( \mu \) are the lubricant density and viscosity, respectively, \( h \) is the local radial clearance between piston and cylinder, \( U \) is the piston axial velocity, and \( V \) is the radial velocity of the solid surfaces that limit gap between piston and cylinder. At \( r = R \), \( V = -\partial h/\partial t \) which is the piston radial velocity; at \( r = R+h \), the radial velocity is zero except at the locations of the feeding holes where the velocity is that of the inflow refrigerant gas.

\[ \frac{1}{R^2} \frac{\partial}{\partial \theta} \left( \rho h^3 \frac{\partial p}{\partial \theta} \right) + \frac{\partial}{\partial y} \left( \rho h^3 \frac{\partial p}{\partial y} \right) = 6\mu \frac{\partial}{\partial y} (p h) + 12\mu \partial \left( \rho V_{R+h} \right) + 12\mu \left( \partial h/\partial t \right) \]

The fluid compressibility needs to be taken into account and here an ideal gas behavior is assumed. Equation 1 can then be written as

\[ \frac{1}{R^2} \frac{\partial}{\partial \theta} \left( \rho h^3 \frac{\partial p}{\partial \theta} \right) + \frac{\partial}{\partial y} \left( \rho h^3 \frac{\partial p}{\partial y} \right) = 6\mu \frac{\partial}{\partial y} (p h) + 12\mu \partial \left( \rho V_{R+h} \right) + 12\mu \left( \partial h/\partial t \right) \]
The pressure field is spatially periodic along the $\theta$ direction which requires that $p_{\theta=0} = p_{\theta=2\pi}$. Along the axial direction, the instantaneous pressure is prescribed at top and bottom of the piston, respectively,

$$p(t)_{y=0} = p(t)_{\text{cylinder}} , \quad p(t)_{y=L} = p(t)_{\text{suction}}$$  \hfill (3)

where $L$ is the cylinder length. The compressor under investigation here operates with a semi-open intake, and the pressure inside the compressor shell, which is also that at the bottom of the piston, is the suction pressure as established by the second condition in equation 3.

A further parameter required in equation 2 is the instantaneous local radial clearance between piston and cylinder, $h$, which is obtained from the instantaneous eccentricity ratios, $\varepsilon_x$, $\varepsilon_z$, of top and bottom of the piston, and from the location of the piston top with respect to the cylinder top, $y_{\text{pst}}$, according to figure 3. The value of $h$ is obtained from,

$$h(\theta,y) = c \left[ 1 - \left( 1 - \frac{y}{L} + \frac{y_{\text{pst}}}{L} \right) \left( \frac{\varepsilon_x}{L} \cos \theta + \varepsilon_z \sin \theta \right) - \left( \frac{y - y_{\text{pst}}}{L - y_{\text{pst}}} \right) \left( \frac{\varepsilon_x}{L} \cos \theta + \varepsilon_z \sin \theta \right) \right]$$  \hfill (4)

where $c$ is the radial clearance.

The refrigerant mass flow rate at each feeding hole is determined from mass and momentum conservation equations assuming compressible one dimensional flow (2). Even though the lubricant is fed via a network of channels and restrictors, due to the small dimensions of the restrictors compared to that of the channels, only the former need to be considered in determining the mass flow rate at the feeding holes. The instantaneous mass flow rate at each hole depends on the discharge pressure and the pressure at the hole which is obtained from the Reynolds equation. Because of the term containing the velocity of the inflow refrigerant gas in the Reynolds equation, $V_{R+h}$, the two problems are coupled and need to be solved simultaneously.

### 3.2 Piston Dynamics

The reciprocating motion of the piston is obtained by an electromagnetic device which, in addition to the axial displacement, imposes transversal and angular displacements on the piston that need to be balanced by the lubricant pressure to avoid the mechanical contact between the solid parts. This equilibrium is achieved requiring that the piston dynamic equations be satisfied along the reciprocating motion. In the present investigation, the dynamic equations are solved ignoring the piston mass, and at each time interval the piston radial velocity at top and bottom are such that the induced lubricant pressure field equilibrates the electromagnetic load. From the piston radial velocities at top and bottom, the piston precise location is determined and its orbit is obtained. This is a standard methodology that has been commonly employed in the literature (3).
4. NUMERICAL METHODOLOGY

The solution of the governing equations can only be achieved numerically. To this extent, the Reynolds equation was made dimensionless and integrated via the finite element technique, using triangular elements and quadratic interpolation functions. Use was made of this technique due to its feasibility in providing a good approximation for the solution domain in particular the feeding holes.

For each triangular element a system of equations is established and a global system of equations is then obtained which relates all nodal points and their corresponding pressure values. The global linear system of equations are solved using LU decomposition according to (4) and (5). Further information related to the numerical methodology can be found in (6).

5. RESULTS AND DISCUSSIONS

In what follows, results will be presented for a particular compressor and refrigeration system operating with condensation pressure and temperature of 761.3 kPa and 54.4 ºC, respectively, and evaporation pressure and temperature of 62.43 kPa and -23.3 ºC, respectively. The cooling capacity is 500 Btu/h and the refrigerant is R600a.

The electromagnetic load and the compressor P versus V diagram are known a priori and due to the reciprocating motion the piston respond with a periodically variation with time. Except otherwise noted, in what follows the compressor geometric features and system conditions will be kept fixed for all the results to be explored.

Results for the piston orbit at the top and bottom are presented in figure 4, (a) and (b), respectively. Those orbits are obtained after the periodic regime is established and the piston is stabilized. Shown in the abscissa are the $\varepsilon_x$ values and in the ordinate the $\varepsilon_z$ values, according to the reference system depicted in figure 3. It can be observed from figure 4 that the piston oscillation at the top is greater than that at the bottom. Furthermore, at the top the oscillation is more pronounced along the z direction, and a very minor oscillation is observer along the x direction. For the bottom, the oscillation is also greater along the z direction, and for the x direction almost no oscillation is observed. Another feature of the piston top and bottom oscillations is its occurrence along a particular plane, with little departures from this plane. This is due to the nature of the electromagnetic load applied to the piston. In general, it is observed that the piston is rather stable, with maximum oscillations around 0.2, which justify gas lubrication to be employed. Associated with the piston displacements observed in figure 4 are variations in the radial clearance distribution between piston and cylinder and, in turn, in the instantaneous pressure field. All that affects the inflow of refrigerant gas into the lubricant film.
Instantaneous results for the mass flow rate at each feeding hole are shown in figure 5. The holes located at the upper part of the piston, close to the piston top, are shown in solid lines; the holes located at the lower part of the piston, close to the piston bottom, are shown in dashed lines. It is interesting to note that, despite small variations in the mass flow rate for the upper and lower set of holes, the general behavior are the same for each set. Those variations in the mass flow rate at each set of either superior or inferior holes are due to the piston oscillation. At a given piston axial location, the closer the piston is to the cylinder surface, the smaller the local radial clearance which, in turn, result in a higher pressure at this particular location and, subsequently, in a smaller mass flow rate due to the reduction in pressure difference along the channels and restrictors.
Figure 5 – Refrigerant mass flow rate at the feeding holes as a function of time.

Another interesting aspect observed in figure 5 is the sharp drop in the mass flow rate at the superior holes after 0.07 s and altering a mild monotonic decay, yielding to minimum values followed by a similar sharp increase. The minimum values observed for the superior holes correspond in time to the minimum values of the mass flow rates for the inferior holes. This outcome occurs due to the piston axial location. As the piston moves up inside the cylinder and approaches its superior dead end, the pressure at the piston top approaches the discharge pressure. In turn, at the superior holes, the pressure also tends to that of the discharge reducing the pressure difference along the channels and restrictors that lead to those holes resulting in the observed drop of the mass flow rate. Due to the increase in pressure at the piston top, the entire gas film experiences a pressure increase and similarly, the inferior holes are also submitted to higher pressures that will decrease the pressure difference along their channel and restrictors, also reducing the mass flow rate therein.

The rationale presented in the previous paragraphs for the behavior of the mass flow rates in the feeding holes as a function of time is corroborated by the pressure variation with time shown in figure 6 for the feeding holes. Comparisons between figures 5 and 6 indicate that the time interval corresponding to the abrupt variation in the mass flow rate at the superior and inferior holes coincides with that where the pressures substantially increase at those holes.

Next, results for the influence of some geometric parameters on the performance of the pneumatic piston will be explored. To this extent, figure 7 presents the effect of the hydraulic diameter of the restrictors on the mass leakage as a percentage of the total compressor mass flow rate, and on the maximum piston radial displacement at top and bottom. Results for the mass leakage flow rates should be read on the ordinate at the left hand side, and results for the piston displacements should be read on the ordinate at the right hand side. As expected, as the restrictor hydraulic diameter is increased, more mass is drawn from the discharge line because of the decrease in pressure drop along the lubricating circuitry, which will improve the lubricating performance of the gas film resulting in a more stable piston orbit. On the other hand, as the restrictor hydraulic diameter is reduced, less inflow of mass will be available at the feeding holes and, consequently, poorer lubrication will lead to higher oscillations of top and bottom of the piston. For reference, the reference case is indicated in figure 7 and corresponds to a hydraulic diameter of 30.
Figure 6 – Pressure at the feeding holes as function of time.

Figure 7 – Mass leaking and maximum piston radial displacement for different hydraulic diameters of the restrictors.

Results showing the influence of both the feeding hole diameter and the radial clearance between piston and cylinder on mass leaking and piston radial displacement are presented in figure 8, (a) and (b), respectively. Increase in the feeding hole diameter causes increase in the leaking of refrigerant mass and decrease in the piston maximum radial displacement, as expected. On the other hand, increase in the radial clearance leads to decrease in the mass leaking and to increase in the maximum piston radial displacement, which is a less obvious result. The tendency to reduce mass leaking with decreasing in radial clearance as observed in figure 8 (b) for radial clearances less than 7 µm shows that the piston proximity to the feeding hole, the piston dynamics, and the gas pressure at the piston top affect the refrigerant mass.
inflow in a complex manner. As observed from the figures, the effect of the feeding hole diameter on the leaking of refrigerant mass and on the piston top and bottom displacements is less pronounced than that of the radial clearance. Furthermore, comparing figures 7 and 8, it is seen that the restrictor geometry plays a more important role on the operational conditions of the pneumatic piston than the other geometric parameters investigated, that is, the feeding hole diameter and the radial clearance.

Figure 8 – Influence of the feeding hole diameter (a), and of the radial clearance (b), on the mass leaking and on the piston maximum radial displacement.
6. CONCLUSIONS

A physical model and a numerical methodology has been successfully introduced and explored in the present work for a pneumatic piston in a linear reciprocating compressor. The pneumatic piston is lubricated with a fraction of the refrigerant compressed gas that is drawn from the discharge chamber back to the radial clearance between piston and cylinder. Both the lubricating problem for the gaseous fluid film and the dynamic problem associated to the piston displacement were considered. For the solution of the compressible Reynolds equation governing the lubrication problem a finite element methodology was employed. For the piston dynamics, the piston mass was ignored and a time marching procedure was adopted.

The results indicate that with a 4% leaking of the compressed gas back to the clearance between piston and cylinder a good lubrication is achieved with small radial oscillations of the reciprocating motion. It was also shown that in designing the gas circuitry that feeds the lubricating film, the presence of restrictors properly chosen play a crucial role in the performance of the lubricating system.

7. REFERENCES

Development of acoustic compressor using large amplitude waveform obtained in closed tube

Md. Anwar Hossain, Tamotsu Fujioka
Compressor Department, Anest Iwata Corporation, Japan
Masaaki Kawahashi
Department of Mechanical Engineering, Saitama University, Japan

ABSTRACT

This paper describes the development of an acoustic compressor by using a large amplitude pressure waveform obtained at the closed end of a resonant tube. Considering the stability of the resonant frequency at different driving accelerations, this study revealed that the half cosine shaped resonator is the best design for an acoustic compressor. The flow rate of a prototype acoustic compressor is estimated experimentally, which explored the possibilities of developing acoustic compressors for commercial use.

1. INTRODUCTION

The resonant oscillation of gas columns in closed tubes has been extensively investigated in recent years for its application in practical systems (1, 2). The majority of the published work on finite amplitude oscillations of interior medium in closed tubes deals with a rigid wall at one end and a vibrating piston at the opposite end of a straight cylindrical tube. An extensive theoretical and experimental investigation (3, 4) of the air column oscillation in a cylindrical tube showed that the waveform is accompanied by a shock which limits the amplitude of the waveform. The shocks are sawtooth-like waveforms contain very sharp gradients that lead to increased dissipation of energy in the system through viscous mechanisms. When the tube is forced at the fundamental natural frequency, a single shock travels back and forth between the ends of the tube. The steepened waveform of the shock is due to the presence of higher harmonics of the fundamental oscillations in the tube, which are generated by nonlinear effects. In cylindrical tubes, the natural resonant frequencies of the system are integer multiples of the fundamental. Hence, energy is readily transferred from the fundamental mode to its higher harmonics through the action of nonlinearities, resulting in the formation of shock waves. Recent investigations (5, 6) revealed that the geometry of the tube shape is the most important factor in determining the shape and amplitude of the waveform. Theoretical and experimental observations explored that it is possible to avoid the formation of shock wave and thereby acoustic saturation by changing the shape of the tube. This finding explored the possibility of generating large amplitude pressure waveform in a resonant closed tube which can be used to develop a practical system for an acoustic compressor (7). Furthermore, nonlinear phenomena,
namely acoustic streaming and thermoacoustic effect, are induced near the oscillatory boundary layer by the oscillation of the interior medium in a closed tube (8, 9). The acoustic streaming increases the velocity of the convection current and makes the unsteady convection into a steady convection (10). The thermoacoustic effect can be used to develop thermoacoustic engines and refrigerators (11).

In this context, the purpose of this report is to introduce an acoustic compressor, which is developed by using large amplitude pressure waveform obtained at the closed end of a properly designed resonator. The flow rate of a prototype acoustic compressor is estimated when the resonator is filled with HFC134a.

2. ACOUSTIC COMPRESSOR

Acoustic compressor is simple in design and easy to operate. The main parts of acoustic compressor are resonator, valves and driving system. The oscillations of the interior medium in the resonator are forced by using piston as a sound source at first mode resonant frequency so that the ends of the resonators are out of phase. The valves are placed at the closed end because the maximum pressure fluctuations are obtained at the end of the resonators. The operating principle of acoustic compressor is, when low pressure is induced in front of the valves, the suction valve is open and the discharge valve is closed which results low pressure gas is sucked into the tube. After half a period, when high pressure is induced in front of the valves, the suction valve is closed and the discharge valve is opened which results high pressure gas is discharged out of the tube. An overview of acoustic compressor is described below:

![Fig.1 Pressure waveform and frequency spectrum in cylindrical resonator](image)

2.1 Resonator

The resonator is the heart of the acoustic compressor. An acoustic compression cycle creates to compress the interior medium in the resonator. The amplitude of the pressure waveform strongly varies with the shape of the resonator. In a resonator with constant cross sectional area (cylindrical shape), the shock wave propagates with the wave motion that limits the amplitude of the pressure waveform. In fact, in cylindrical resonator, a series of higher harmonics are generated. As shown in Fig.1, All those harmonics are coincident with the modal frequency which results transfer of energy to the higher harmonics instead of increasing the amplitude of the waveform. In a resonator with cross sectional area change, it is possible to control the generation of higher harmonics that avoids the problem of shock formation. Therefore, as
shown in Fig. 2, it is possible to obtain shockless large amplitude pressure waveform in the area change resonator. This means that the resonator geometry is the most important factor in determining the amplitude of the pressure waveform. In this research, different resonator geometries are studied to explore the optimum resonator shape to design acoustic compressor. The details on resonator geometry will be explained in section 3.

![Fig.2 Pressure waveform and frequency spectrum in area change resonator](image)

2.2 Valves
The valves of acoustic compressor operate at very high frequencies. The valve response determines the performance of acoustic compressor. The response of the valves should be as same as the resonant frequency of the system. The resonant frequency depends on the length of the resonator and the sound speed. In this research, we used a mechanical reed valve which is made by a special type material named GIN6. The thickness of the valve is 0.108 mm. The cross sectional view of an acoustic compressor and the detail view of the valve geometry are shown in Fig. 3. A small hole of 2 mm diameter is provided in between inlet and outlet valves to measure the pressure fluctuation at the closed end of the resonator when the acoustic compressor is in operation. The pressure transducer is flash mounted in that small hole.

![Fig.3 Cross sectional view of an acoustic compressor](image)
2.3 Driving System

There are two ways to drive the interior medium of the resonator. Among them one is the piston drive system and the other one is the entire resonator drive system. We used piston drive system to oscillate interior medium in the resonator. The oscillations of piston are forced by using vibration generator through power amplifier. One of the advantages of piston drive system is that it can easily induce plane wave propagation in the resonator. The piston is made of aluminum with a diameter just slightly less than the inner diameter of the resonator at driving end to make the piston easy for reciprocating movement parallel to the axis of the resonator. The O-ring is used to seal the gap between the piston and the resonator.

3. OPTIMUMIZATION OF THE RESONATOR SHAPE

It is necessary to find out the optimum resonator shape to design practical systems like an acoustic compressor. For this purpose, finite amplitude standing wave in different resonator geometries was studied by theoretical analysis, numerical simulation and also by experiment. First, the effect of cross sectional area contraction ratios (ratio of cross sectional area at driving end to the closed end) and the effect of resonator lengths on wave motion are investigated by the numerical simulation. Based on numerical investigations, three resonators are designed and the performance of each resonator is investigated experimentally. The shape of the resonator is conical, exponential and half cosine shaped which cover all possible simple resonator geometries. Particularly, the shape of the waveform, compression ratio, and frequency responses are verified and the best resonator shape is explored for the development of an acoustic compressor.

3.1 Numerical Simulation

One-dimensional governing equations are used for the numerical simulation of wave motion in cross sectional area change tube. The finite difference MacCormack scheme is used by preserving the computational accuracy with second order in time and fourth order in space. The continuity, momentum and energy equation in conservative form is expressed as follows:

\[
\frac{\partial QA}{\partial t} + \frac{\partial JA}{\partial x} = H
\]  

(1)

Here \( Q, J \) and \( H \) is expressed in the following matrix form:

\[
Q = \begin{bmatrix} \rho \\ \rho u \\ E \end{bmatrix}, \quad J = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ (E + p)u \end{bmatrix}, \quad H = \begin{bmatrix} 0 \\ p \frac{\partial \phi}{\partial x} + M \\ Mu \end{bmatrix}
\]

(2)

\[
M = \frac{4}{3} \mu \frac{\partial^2 u}{\partial x^2} + F
\]

where \( \rho \) is the density, \( u \) is the oscillating velocity, \( E \) is the total energy composed of internal energy and kinetic energy, \( p \) is the pressure, \( A \) is the cross sectional area of the tube, \( \mu \) is the coefficient of viscosity, \( F \) is the friction, \( t \) is the time and \( x \) is the Cartesian coordinate.
Numerical simulation is carried out in an area change resonator with exponentially area contraction from the driving end towards the closed end. The geometry of the exponential area change resonator is expressed as follows:

$$A(x) = A_0 \exp(mx)$$

(3)

where, $A_0$ is the cross sectional area at closed end and the constant $m$ is written as follows:

$$m = \frac{1}{l} \ln \left( \frac{A_p}{A_0} \right)$$

where, $A_p$ is the cross sectional area at driving end, $l$ is the length of the resonator, and $A_p/A_0$ is the area contraction ratio.

The resonant frequency of gas column oscillations in closed area change exponential tube varies with the variation of resonator shapes. To design acoustic compressors, the consideration of resonant frequency of pressure oscillation is very important. This is because the valve response is related to the frequency of oscillation. The change of resonant frequency with area contraction ratio is shown in Fig.4 for piston acceleration amplitude of 200 m/s². It is assumed that the resonator is filled with HFC134a at normal atmospheric pressure and temperature. The resonant frequencies were normalized by the theoretical resonant frequency of the cylindrical shaped resonator. The theoretical resonant frequencies for different area contraction ratios are also shown in the figure. It is seen that the resonant frequency increases with the increase of area contraction ratio. The rate of increase of resonant frequency is linear at first, where the rate decreases at high area contraction ratios. It is noteworthy that the resonant frequency obtained by numerical simulation agrees well with the resonant frequency obtained by the linear acoustic theory.

The limit of cross sectional area contraction ratio to obtain a shockless pressure waveform is investigated numerically. The pressure fluctuation at the closed end of an exponential tube with area contraction ratio of 4, 9, 16, 25 and 36 and piston acceleration amplitude of 200 m/s² is shown in Fig.5. It is seen that the shock wave appears up to the area contraction ratio of 16 and then disappears for the ratio of 25 and more. It is observed that the amplitude of the waveform...
increases with the increase of area contraction ratio, however, the waveform has a narrower peak and a broader trough at higher area contraction ratios.

The evaluation of finite amplitude standing wave for some practical applications, especially for acoustic compressors, can be made by the compression ratio obtained at the closed end of the resonant tube, which is estimated by the ratio of the highest to the lowest absolute pressure of the waveform. Figure 6 shows the compression ratio obtained at the closed end corresponding to the area contraction ratio for piston acceleration amplitude of 200 m/s$^2$. It is seen that the compression ratio increases with the increase of area contraction ratio. The rate of increase of compression ratio is linear at first, which comes to the saturation state after a certain limit of area contraction ratio. The limit of area contraction ratio for acoustic saturation further varies with the variation of piston acceleration amplitudes.

The effect of resonator length on wave motion in closed axisymmetric tube needs to be realized to design any practical devices like acoustic compressors. Depending upon the applications, the resonators can be scaled in length to affect the frequency of operation and capacity. In this report, numerical simulation results are presented for different tube lengths. The area contraction ratio of the exponential area contraction tube is considered as 100. Figure 7 shows the change of compression ratio and resonant frequency with regards to resonator length when it is filled with HFC134a at normal atmospheric pressure and temperature. The piston acceleration amplitude is kept at 200 m/s$^2$. In each case the frequency is kept at the first mode resonant frequency. It is observed that the length enlargement decreases the resonant frequency while increasing the compression ratio. Higher frequency operation reduces not only the size of the resonator but also its efficiency. Therefore, to design any practical devices, optimization of the resonator length is essential according to the application requirement.

### 3.2 Experimental Observation

Based on numerical simulation results, three resonators are designed and the performance of each resonator is estimated experimentally. The shape of the resonators was conical, exponential and half cosine shaped, respectively. The length of each resonator is 500 mm. The diameter at closed end of the resonator is 10 mm and the diameter at the driving end (piston end) is 100 mm. This means the cross sectional area of each ducts are kept 100 so that the
waveform would not be accompanied with shock even at high acceleration amplitudes. Experimental results are presented considering that the resonator is filled with HFC134a at normal atmospheric pressure and temperature.

The pressure waveforms obtained at closed end of conical, exponential and half cosine shaped resonator at resonant state are shown in Fig 8. The waveforms are presented for piston acceleration amplitudes of 100, 300 and 500 m/s². In contrast to the cylindrical resonator, shock wave does not appear in spite of large amplitude of acoustic pressure. In conical resonator, it is seen that as the piston acceleration amplitude increases, the amplitude of the pressure waveform increases, however the waveform has a narrower peak and a broader trough. The shockless large amplitude pressure waveform is also observed in exponential resonator, however a few micro-shock appears at the peak pressures for piston acceleration amplitude of 500 m/s². Experimental results obtained in half cosine shaped resonator represents that the waveform is almost sinusoidal even at large piston acceleration amplitudes.

Fig.8 Pressure fluctuations at closed end of conical, exponential and half cosine resonator for three piston acceleration amplitudes

The frequency response curves in conical, exponential and half cosine resonator are shown in Fig 9. The peak-peak pressure amplitude at the closed end of the tube was monitored during the frequency sweeps. The curves are plotted for upward and downward frequency sweeps. In the conical resonator, it is seen that for high piston acceleration amplitudes, the resonance curve obtained at upward frequency sweep leans towards the upper frequency. Furthermore, for piston acceleration amplitude of 300 and 500 m/s², an abrupt discontinuity in the response curve occurs near the resonant frequency. The graph also shows that the resonant frequency
shifts towards the upper frequency region with the increase of piston acceleration amplitude. In the exponential resonator, though the abrupt fall of pressure does not appear however, the shift of resonance frequency with the increase of piston acceleration amplitude clearly observed. In the half cosine resonator, the shift of resonant frequency with the increase of piston acceleration amplitude and the sudden fall of pressures near the resonant frequency is not observed. Among three resonator geometries, the stability of resonant frequency at different driving accelerations is predicted only in half cosine shaped resonator. These results suggest that the half cosine resonator is the best candidate for the development of an acoustic compressor.

3. FLOW RATE OF ACOUSTIC COMPRESSOR

The large amplitude pressure waveform as well as high compressor ratio at closed end of area change resonators has been developed and a best resonator shape for stable resonant frequency has been explored. Therefore, it is possible to design a prototype acoustic compressor. The valves are placed at the small end of the closed area change resonators. Experiments are carried out by filling the entire system with HFC134a at atmospheric pressure and temperature.

The change of compression ratio with piston acceleration amplitudes is shown in figure 10.
The compression ratio increases with the increase of piston acceleration amplitudes. Among three area change resonators, the compression ratio is higher in half cosine shaped resonator compared to the others. Particularly, for piston acceleration amplitude of 500 m/s$^2$, the compression ratio is nearly 6 in the half cosine resonator. Usually, the compression ratio of 6 or more is required to design any practical system. This means our predicted value is sufficient to design an acoustic compressor for practical use.

![Fig.10 Compression ratio with piston acceleration in three resonators](image1)

![Fig.11 Flow rate of acoustic compressor for piston acceleration 500 m/s$^2$](image2)

Figure 11 shows the flow rate of acoustic compressor. Acoustic compressor is operated for piston acceleration amplitude of 500 m/s$^2$ at resonant state. The operating resonant frequency of acoustic compressor varied with the variation of resonator geometries. The operating resonant frequencies of acoustic compressor vary from 200-250 Hz. The flow rate is calculated from the discharge pressure for each resonator. The least square fit of fifth order polynomial is used for the curve fitting.

4. CONCLUSIONS

A simple prototype acoustic compressor is designed by using properly designed resonator. The half cosine shaped resonator is the best because it can develop a large amplitude shock free pressure waveform. The stability of resonant frequency is confirmed in the half cosine shaped resonator. The flow rate of acoustic compressor is estimated which reveals the possibilities of using acoustic compressor in practical fields.

REFERENCES


Experimental evaluation of an innovative rotary compressor with variable speed displacers

Pontifical Catholic University of Rio de Janeiro, Department of Mechanical Engineering, Brazil
(*) Corresponding author

ABSTRACT

The paper describes preliminary tests of a new rotary compressor with variable speed displacers. Two displacers rotate concentrically, in an annular space, at variable and phased angular velocities, thus creating two variable-volume compression spaces between them. The displacers are individually driven by two concentric shafts. An innovative driving mechanism imposes phased variable angular speeds to the shafts and, consequently, to the displacers. The driving mechanism also offers a convenient way of capacity control, from zero to 100%, at constant electric motor speed. A prototype was constructed and tested. First performance results are presented, showing the compressor behaviour under different operational conditions.

1. INTRODUCTION

Recent advances in the development programme of an innovative rotary compressor [1] are presented in this paper. Rotary compressors are, of course, positive displacement machines where the traditional piston-slider crank mechanism of the reciprocating compressor is substituted by some sort of arrangement where the displacer (the piston in reciprocating compressors) presents a predominantly rotating movement. For this reason, rotary compressors are less subjected to mechanical vibration and, owing to potentially higher velocities, are usually smaller than their reciprocating counterpart, for a given volumetric capacity. Different types of rotary compressors are now available, including the sliding vane, stationary-vane rolling piston, scroll and screw compressors, all of which have found commercial application in many different fields. Innovative rotary compressors have long been reported in the literature, showing different stages of development and success (see, for example, references [2-8]).

This paper presents a preliminary experimental evaluation of a new rotary positive displacement compressor. The Kopelrot Compressor Programme started with the construction of a demonstration scale model, to provide the first clues to the proposed compression device with its driving mechanism. A first proof-of-concept prototype was then constructed to evaluate, at relatively low speeds, the behaviour of the volume variation mechanism. Since no similar machine had yet been constructed, a simulation model became an essential tool to
establish basic geometric relations for the design and construction of a new prototype. A mathematical model simulating the performance of the new rotary compressor was then developed [9]. A traditional angle-by-angle simulation method for positive displacement compressors [10] was employed, based on mass and energy conservation equations, in differential form, applied to the control volumes (two compression spaces). A few important lessons were learned from the simulation effort. Given proper geometric relations, the proposed compressor showed to be able to function properly, with pressure-volume diagrams typical of traditional positive displacement compressors [9]. Furthermore, the driving mechanism proved to be well suited for capacity control. This capability was further explored in the preliminary tests of the second prototype, the results of which are reported in this paper.

2. COMPRESSOR DESCRIPTION

Figure 1 depicts the schematics of the proposed compressor. Two displacers rotate concentrically, in an annular space, at variable and phased angular velocities, thus creating two variable-volume compression spaces between them. Each displacer is attached to its own rotor. The movement of the rotor/displacer assemblies is dictated by fixed-length cranks pivoted to a common slot of a driving disk, Figure 2, driven, by its turn, by a constant speed electric motor. The pivoting point of each displacer is allowed to slide along the driving disk slot, as shown in Figure 2. Figure 3 depicts rotors A and B, the driving disk and the rotors assembly, providing a close view of the driving mechanism. Note that rotors and disk are eccentrically mounted (see Figure 2).

The resulting movement of the displacers and their respective sliding pivot points (guides) is depicted in Figures 4a and 4b, showing displacer and guide positions at each 45-degree of the driving axis. The locations of the suction and discharge ports are also shown in Figure 4a.

In spite of the guides (pivoting points) translational and rotational movements, the cranks are limited to rotate about the compressor axis. The angular velocity of the cranks, and their respective rotor/displacer assemblies, is determined by the position of their pivoting points. If a guide is in the centre of the disk, it will rotate around its own axis and will not rotate the crank. When it moves along the slot, it gains translational movement, which is converted to pure rotational movement of the cranks. The angular velocity increases when the guide moves...
farther away from the disk centre and decreases when otherwise. This innovative mechanism, patented by Kopelowicz [1], thus provides the required variable and phased angular velocity of the displacers. Inspection of Figure 4a shows that this is a positive displacement double-acting rotary compressor, with two full compression cycles per revolution of the driving axis. Suction and discharge ports, eventually with valves, must be strategically located at the circumference of the cylinder, in the valve plate (see Figure 5). Note that the displacers perform a circumferential trajectory, with centre in the cylinder axis, thus avoiding direct contact between the moving parts and the cylinder surface (as in rotary vane and rolling piston compressors). This would dispense, in theory, the use of sealing parts, moreover, as most of the leakage paths are relatively long.

Figure 2 – Overall and exploded view of the proposed compressor with its driving mechanism.

Figure 3 – Views of (counter-clockwise): (a) Driving disk; (b) Rotor A assembly (with displacer, shaft and crank); (c) Rotor B assembly; (d) Rotors A and B assembled.
Figure 4 - Principle of operation of the Kopelrot compressor: a) displacers positioning at every 45 degrees for one full shaft revolution; b) corresponding positions of levers’ pivoting points.

The amplitude of the angular speed variation of the displacers is dictated by the eccentricity of the device, i.e., the distance between compressor and electric motor axes. If the compressor axis is aligned with the driving plate axis, the distance from the crank guides to the disk centre will be equal, thus imposing equal velocities to their displacers and producing no compression. The displacers will rotate at constant speed, separated by a constant angle of 90 degrees. When an eccentricity is imposed, that distance will vary, resulting in a 180-degree phased movement and chamber’s volume variation. The eccentricity can be mechanically adjusted, allowing unique soft start-ups, with no compression and progressive increase of speed up to the required operating conditions. Soft start-ups contribute to extend equipment life and reduce maintenance costs. Also, capacity control systems could be implemented to modulate the compressor volumetric capacity, reducing energy consumption when lower capacity is required and maintaining continuous operation.

Figure 5 shows a sectioned view of the compressor fully assembled. Basic components are: rotors A and B, encompassing respective shafts, displacers A and B, firmly attached to their corresponding rotors, shell, top and lower covers, and the valve “plate”, in fact a 10mm-thick hollow disk where suction and discharge gas passages are drilled radially and valves are eventually installed. The discharge valve has to be positioned in the inner surface of the valve plate, so as to avoid an excessive clearance volume. To be noted the concentric movement of both displacers, leading to an operation with lower vibration levels. Internal and external gas leakage can be diminished with proper manufacturing tolerances or with the use of seals.
3. PREDICTED CAPACITY CONTROL CAPABILITY

Predicted results of volume variations for cylinder geometry of 100 mm, 88 mm and 38 mm of width, outer and inner diameters, respectively, revealed a relation between displaced and clearance (minimum) volumes with shaft eccentricity, as depicted in Figure 6. Clearly, as performance simulation results confirmed [9], capacity control could be enforced, since both the displaced and clearance volumes are altered by the shaft eccentricity. For each geometry there is a maximum eccentricity, which corresponds to zero clearance volume (with displacers, theoretically, touching each other at one point of their trajectory). In practice, the compressor must operate below this value. The swept volume is, of course, the difference between the maximum and minimum volumes.

![Figure 5 - Sectioned view of the compressor (fully assembled; driving mechanism not shown).](image)

![Figure 6 – Variation of maximum, minimum and displaced volumes with shaft eccentricity.](image)
4. PROTOTYPE AND EXPERIMENTAL APPARATUS

Figure 7 shows two views of the prototype compressor in the test bench. Inner and outer diameters of the compression cylinder were 38 mm and 88 mm, respectively. Length of the displacers, or cylinder width, was 100 mm. The displacers were made of steel 1010, and the cylinder, of nodular iron. Due to large manufacturing tolerances and undesirable material expansion (in spite of forced liquid cooling on the cylinder external surface), mechanical seals (made of phosphorous bronze) were installed in this prototype. Figure 8 shows the rotor and displacers with respective seals (longitudinally-\(a\), radially-\(b\) and circumferentially-\(c\) displaced).

The compressor operated with a conventional discharge valve and a suction port (no valve), both strategically located alongside the valve plate circumference. As indicated by previous simulation results, the angular positions of the suction and discharge passages are key factors to the compressor performance. Modelling predictions also indicated that, should better manufacturing tolerances and cylinder cooling be applied, adequate performance could be achieved without the mechanical seals, as seen on figure 8. The driving mechanism (crank pins, sliders and driving disc slot) was enclosed in a case containing oil, so as to provide “splash lubrication”. Lubrication of the rotors and displacers was provided, on a provisional basis, by oil injection in the suction.

The compressor was driven by a variable speed electric motor, so as to have it tested over a range of shaft speeds. A wattmeter was employed, although a shaft torque meter is planned to be used in the future, to measure the compressor-only work, independently from the electric motor efficiency. A turbine flow meter was installed upstream the compressor, with two large volume tanks between them, in series, in order to minimise the amplitude and influence of air pulsation in the flow meter. Compressed air was pumped to a high-pressure 100L air reservoir.
A pressure relief needle valve, installed in the reservoir, controlled the required air mass flow rate and, consequently, the discharge pressure. The maximum discharge pressure/no-flow condition (volumetric efficiency equal to zero) was achieved, for each combination of eccentricity and shaft speed, by operating the compressor against the air reservoir with the control valve fully closed. Temperature and pressure sensors were located at key points of the apparatus, to monitor the operation of the system.

5. RESULTS

Tests were carried out for a shaft speed of 1130 rpm. Figure 9 shows the normalized volumetric flow rate as a function of pressure ratio and shaft eccentricity. Three eccentricities to inner cylinder diameter ratios were evaluated, confirming the possibility of capacity modulation via adjustment of the shaft eccentricity. For example, for a pressure ratio of 3, the volumetric flow rate could be reduced to approximately 30% of its original value with mere 4 mm of adjustment in compressor shaft positioning. Experimental observation showed that the effort to apply a new eccentricity value (i.e., to displace the compressor body along its base slots – see detail in figure 7) is small and compatible with existing (i.e., commercially available) linear displacement servo-mechanisms. In the prototype, eccentricity adjustments were made manually. In Figure 9, the full lines correspond to linear best fits for each group of eccentricity data points.

Cylinder temperature remained at around 60°C for most of the tests. The discharge temperature, ranging from 160°C to 230°C, varied with operational conditions, notably, as expected, with the pressure ratio. It was also most affected by the lubrication conditions of this prototype, i.e., oil injection in suction port.

6. CONCLUDING REMARKS

A new type of rotary compressor was developed and tested. The Kopelrot compressor, for its inherent low vibration and simplicity, has shown great potential for several applications. The development programme will continue with further work on the present prototype and the use
of the simulation tool. A new concept for a four-45°-displacer double-stage intercooled CO₂ Kopelrot compressor is presently under way.

ACKNOWLEDGEMENTS

Thanks are due to FAPERJ, FINEP and CNPq, Brazilian research agencies, which have financially supported this work. The authors are also indebted to Mr. Lucas B. Vancini for his invaluable services with the manufacturing of the prototype.

NOMENCLATURE

\( V_{\text{max}} \) maximum chamber volume (mL)
\( V_{\text{min}} \) minimum chamber (clearance) volume (mL)

REFERENCES


Development of small size air compressor for mobile fuel cells

Kiyoshi Sawai, Atsushi Sakuda, Tatsuya Nakamoto, Noboru Iida, Tsutomu Tsujimoto, Hiroyuki Fukuhara, Hideki Murakami
Air Conditioning Research Laboratory, Matsushita Electric Industry, Co., Ltd., Japan
Tadashi Nagata
Air Conditioning Devices Division, Matsushita Electric Industry, Co., Ltd., Japan
Noriaki Ishii
Dept. of Mechanical Engineering, Osaka Electro - Communication University, Japan

ABSTRACT

In recent years, as the mobile equipment advances, higher output and higher energy density are required of the battery. As one of the solutions, development of mobile fuel cells (FC) with methanol used for fuel has been progressed and it will be soon put to practical use. The authors have developed a small size air compressor, which is mounted to mobile FC and forcibly supplies air for reactions with methanol. While being oil-less, this air compressor achieves an unparalleled small size and considerably high discharge pressure in the world, by adopting a rotary vane mechanism with self-lubricating vanes. In addition, by carrying out dynamic analysis to investigate the vane sliding form, a compression mechanism with low noise and small input is realized.

NOMENCLATURE

Symbol

\( \text{Oc} \) Center of the cylinder

\( \text{Or} \) Center of the rotor

\( \theta \) Rotating angle of the rotor

\( \omega \) Angular velocity of the rotor

\( B \) Vane thickness, Slit width

\( B_1 \) Length along the vane thickness in suction pressure at vane head

\( L \) Vane length \((L = L_1 + L_2)\)

\( L_1 \) Vane length inside the slit

\( L_2 \) Vane length exposed to the slit outside

\( H \) Cylinder height, Rotor height, Vane height

\( m \) Mass of the vane

\( r \) Distance from \( \text{Or} \) to gravity center \( \text{Gv} \) of the vane

\( \theta_1 \) Angle made by the vane and the line that connects \( \text{Oc} \) to vane head

\( \theta_2 \) Angle made by the vane and the line that connects \( \text{Or} \) to \( \text{Gv} \)

\( F_{pn} \) Force acting on the vane head by discharge pressure (normal direction to the slit)

\( F_{pb} \) Force acting on the vane by rear pressure (slit direction)

\( F_c \) Centrifugal force of the vane

\( F_{s1} \) Reactive force acting on vane side by the slit (at the head of the vane)
1. INTRODUCTION

In recent years, downsizing and increased functions of mobile information equipment have been amazingly advanced, and for example, notebook PCs not only with personal computer functions but also with AV functions and radio communication functions, have been put on market. As a result of this, power consumption of mobile information equipment has increased, and battery that drives this equipment must satisfy requirements for higher output and higher energy density. Presently, for this kind of battery, the Li-ion battery is used, but its energy density is almost reaching to its limit. One of the solutions to this problem is mobile fuel cells that use methanol for fuel, and development is positively underway in the world in an effort to put it into practical use in the near future.

In the mobile fuel cells, methanol fuel and oxygen (air) are supplied to the power generation section (stack), where methanol fuel is allowed to react chemically with oxygen to generate electric power. In order to increase the output power of the mobile fuel cells and improve its controllability, it is essential to timely provide necessary and sufficient air amount to the stack.

The air compressor plays a role to supply air to the stack, and is the most important device to offer technical advantages on the mobile fuel cells. For the air compressor, which is mounted to the mobile fuel cells, characteristics of (1) oil-less operation, (2) small-size, (3) high pressure, (4) low noise and (5) small input power are required. Then, the authors worked to develop small-size and high-performance air compressor.

In the first stage of development, efforts were made to choose the compression mechanism. Five kinds of compression mechanism were taken up, and performance, noise, size, and reliability are compared, as a result, a rotary vane mechanism was chosen. In this mechanism, the vane transports air while the vane comes in contact with a cylinder wall, then high volumetric efficiency, small size and high discharge pressure may be obtained at a small air flow rate. And also valve-less mechanism may be achievable, which is advantageous from the viewpoint of noise.

In the second stage of development, prototype compressor was made, and it was confirmed that the prototype exhibits the initial performance as targeted.

In the third stage, to aim at the stabilization of noise and input power, the authors worked for dynamic analysis of the compression mechanism. By this dynamic analysis, the magnitude of the forces acting on the vane and its meaning were clarified. Then, based on analytical results, an original vane sliding form was adopted, which stabilizes noise and input power over a long operation. Through these processes, the authors were able to develop an air compressor for mobile fuel cells, which provides features without parallel in the world; that is, small size, considerably high pressure, low noise, and small input.
This paper reports the structure of the developed air compressor, the process and results of the dynamic analysis, unique vane configuration and the compressor performance.

2. BASIC STRUCTURE AND FEATURES OF THE AIR COMPRESSOR

2.1 Basic structure of the air compressor

Fig.1 shows the basic structure of the developed air compressor. This air compressor adopts a rotary vane system for the compression mechanism, which is driven by a DC motor. The compression mechanism comprises a cylinder, a rotor, vanes, a front plate, and a rear plate. The rotor is arranged in the cylinder in the condition eccentric from the central axis, and to the rotor, two slits are provided, and to these slits, two vanes are fitted in the condition free to slide. The front plate and the rear plate are arranged as if they sandwich these cylinder, rotor, and vanes, and compression chambers are formed.

When the motor-rotor rotates, the rotor rotates via the shaft. And then the vane rotates in contact with the cylinder wall by the centrifugal force and discharge pressure to the rear of the vane. By expanding and contracting of the compression chambers, the pumping action is generated. The air is inhaled through a suction port in the front plate and after the air is pressurized inside the compression chambers, it is discharged through the discharge port in the front plate.

Fig.1 Basic structure of the air compressor

2.2 Specifications and features of the air compressor

Tab.1 shows final specifications of the developed air compressor and Fig.2 shows the performance characteristics (P-Q-W characteristics).

The first feature of this compressor is a small diameter (ϕ 30mm), this is achieved by arranging vanes eccentric from the rotor center as shown in Fig.1. The second feature is high discharge pressure (ΔP = 2 ~6 kPa) and small input power at small flow rate (Q = 1.5~4 L/min). The third feature is a low noise (40dB) achieved by optimizing built-in volume ratio (= suction volume divided by compression end volume) and eliminating a discharge valve. In addition, in order to stably maintain low noise, a unique vane sliding form is adopted. The fourth feature is the vane of this air compressor, which is formed with carbon composite material with self-lubricity. By an original surface treatment on the cylinder wall on which the vane slides, long-time (10,000 hours) oil-less operation has been enabled.
Table 1 Final specifications of the air compressor

<table>
<thead>
<tr>
<th>Compression mechanism</th>
<th>Rotary vane type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Discharge pressure $\Delta P$</td>
<td>2~6 kPa</td>
</tr>
<tr>
<td>Air flow rate $Q$</td>
<td>1.5~4 L/min</td>
</tr>
<tr>
<td>Power source (DC)</td>
<td>10~15 V</td>
</tr>
<tr>
<td>Noise at 50cm</td>
<td>40 dB(A)</td>
</tr>
<tr>
<td>Size</td>
<td>$\phi 30 \times L40$ mm</td>
</tr>
<tr>
<td>Weight</td>
<td>70 g</td>
</tr>
</tbody>
</table>

Fig.2 Performance diagram of the air compressor (P-Q-W characteristics)

3. DYNAMIC ANALYSIS OF COMPRESSION MECHANISM

3.1 Objectives of the dynamic analysis
The vane moves on cylinder while the head comes in contact with the cylinder wall. In the case of eccentric arrangement of the vane, there are two types of sliding form as shown in Fig.3. In the event that the angle made by the vane and the cylinder wall is less than 90°, the sliding form is called the “Trailing type” and in the event that it is larger than 90°, the sliding form is called the “Scooping type.”

It is assumed that the compressor characteristics such as noise, compression power, reliability, depend on the vane sliding form. Therefore, in order to make clear the effects of the vane sliding form on the compressor characteristics, efforts were made to clarify the forces acting on the vane, that is, dynamic analysis of the vane was carried out.
3.2 Process of dynamic analysis

In this paper, taking the “Trailing type” vane sliding form for an example, process of the dynamic analysis will be described. The authors worked to develop a technique, by which we can easily analyze the forces acting on the vane in contact with the cylinder wall or rotor slit. Using the analytical model of the compression mechanism in Fig.4, balancing expressions of forces acting on the vane and a balancing expression of moments are derived and the following equations are obtained, respectively.

The region in which the vane comes out from the slit (θ: 0 – 180°)

Balancing expression of forces on the vane in the slit direction.
\[-\mu_1(F_s1+F_s2)+F_f \cdot \sin \theta_1 - F_w \cdot \cos \theta_1 + F_pb + F_c \cdot \cos \theta_2 = 0\]  (1)

Balancing expression of forces on the vane in the normal direction to the slit.
\[F_s1 - F_s2 - F_f \cdot \cos \theta_1 - F_w \cdot \sin \theta_1 + F_pn + F_c \cdot \sin \theta_2 = 0\]  (2)

Balancing expression of the moment around point P on the vane.
\[F_s2 \cdot L_1 + \mu_1 \cdot R_2 \cdot B - F_f \cdot \cos \theta_1 \cdot L_2 - F_f \cdot \sin \theta_1 \cdot B_1 - F_w \cdot \sin \theta_1 \cdot L_2 + F_w \cdot \cos \theta_1 \cdot B_1 - F_pn \cdot L_2/2 - F_pb \cdot B_2 - F_c \cdot \sin \theta_2 \cdot (L/2 - L_2) - F_c \cdot \cos \theta_2 \cdot B/2 = 0\]  (3)

The region in which the vane enters into the slit (θ: 180 – 360°)

Balancing expression of forces on the vane in the slit direction.
\[\mu_1(F_s1+F_s2)+F_f \cdot \sin \theta_1 - F_w \cdot \cos \theta_1 + F_pb + F_c \cdot \cos \theta_2 = 0\]  (4)

Balancing expression of forces on the vane in the normal direction to the slit.
\[F_s1 - F_s2 - F_f \cdot \cos \theta_1 - F_w \cdot \sin \theta_1 - F_pn + F_c \cdot \sin \theta_2 = 0\]  (5)

Balancing expression of the moment around point P on the vane.
\[F_s2 \cdot L_1 - \mu_1 \cdot R_2 \cdot B - F_f \cdot \cos \theta_1 \cdot L_2 - F_f \cdot \sin \theta_1 \cdot B_1 - F_w \cdot \sin \theta_1 \cdot L_2 + F_w \cdot \cos \theta_1 \cdot B_1 - F_pn \cdot L_2/2 - F_pb \cdot B_2 - F_c \cdot \sin \theta_2 \cdot (L/2 - L_2) - F_c \cdot \cos \theta_2 \cdot B/2 = 0\]  (6)

Where, \(F_c\) which denotes centrifugal force of the vane, \(F_pn\) which denotes force acting on the vane head by discharge pressure, and \(F_pb\) which denotes force acting on the vane by rear pressure, are estimated by the following equations, respectively.
\[
\begin{align*}
\dot{=} \ m \cdot r \cdot \omega^2 \\
F_{pn} & \dot{=} L_2 \cdot H \cdot \Delta P \\
F_{bp} & \dot{=} B_1 \cdot H \cdot \Delta P
\end{align*}
\] (7) (8) (9)

By simultaneously solving equations from (1) to (3), or from (4) to (6) by using equations from (7) to (9), it is possible to find out the forces \( F_{s1}, F_{s2}, \) and \( F_w \) with respect to each rotating angle \( \theta \) of the rotor. These calculations can be easily performed by the use of Excel on the personal computer. In addition, from \( F_{s1} \) and \( F_{s2} \), which are reactive forces acting on vane side by the slit, the compression power can be calculated.

![Analytical model of the compression mechanism](image)

(Trailing type, region in which the vane enters into the slit)

<table>
<thead>
<tr>
<th>Comp. Spec.</th>
<th>Number of vane</th>
<th>2</th>
<th>_____</th>
</tr>
</thead>
<tbody>
<tr>
<td>Suction volume</td>
<td>1.5</td>
<td>cc/rev</td>
<td></td>
</tr>
<tr>
<td>Vane mass</td>
<td>0.4</td>
<td>g / piece</td>
<td></td>
</tr>
<tr>
<td>Anal. Cond.</td>
<td>Discharge pressure ( \Delta P )</td>
<td>4</td>
<td>kPa</td>
</tr>
<tr>
<td>Rotational speed</td>
<td>( N )</td>
<td>2,300</td>
<td>rpm</td>
</tr>
<tr>
<td>Frictional coeff. ( \mu_1 )</td>
<td>0.2 ~ 0.45</td>
<td>_____</td>
<td></td>
</tr>
<tr>
<td>Frictional coeff. ( \mu_2 )</td>
<td>0.2 ~ 0.45</td>
<td>_____</td>
<td></td>
</tr>
</tbody>
</table>

3.3 Results of dynamic analysis
Tab.2 shows main specifications of the compressor and analytical conditions in dynamic analysis. The forces acting on the vane was analyzed on both “Trailing type” vane and “Scooping type” vane. Fig.5 shows the analytical results of the reactive force \( F_w \) in case of “Scooping type” vane, and Fig.6 shows that in case of “Trailing type” vane.
In general, in the case of “Scooping type” vane, the friction acting on the vane head works to prevent the vane from coming out from the slit. This analytical results indicate that when the frictional coefficient $\mu_1$ between the vane side and the slit increases, the reactive force $F_w$ by the cylinder wall becomes extremely small at rotating angles 120° to 180°. This suggests that the friction acting on the vane side degrades a smooth slide of the vane in the slit and the vane may not come in contact with the cylinder wall. That is, in the case of the “Scooping type” vane, jumping phenomena occur in actual operation, and as a result, the noise would increase or the air would leak around the vane head, then the performance would decrease.

On the other hand, in the case of “Trailing type” vane, in general, the friction acting on the vane head works to help the vane come out from the slit. This analytical results evidenced that because in the case of “Trailing type” vane, there is no region in which the reactive force $F_w$ approaches to zero even when the frictional coefficient $\mu_1$ or $\mu_2$ varies, the jumping phenomena don’t occur and the stable low noise would be maintained.
However, it has been found that when the frictional coefficient $\mu_1$ or $\mu_2$ increases, the reactive $F_w$ increases in the region where the vane enters into the slit (rotating angle: 180-300°). When the reactive force $F_w$ increases, there is a fear of increasing the compression power, to which care must be taken.

4. EXPERIMENTAL INVESTIGATION OF PERFORMANCE

In order to verify the analytical results, two prototypes of the air compressor were made, which has different kind of vane sliding form. Long-time (400 or 800 hours) operations were carried out, and changes of compressor performance were investigated. Fig.7 shows the experimental results of the noise change characteristics, and Fig.8 shows the experimental results of the input power change characteristics.

**Noise change characteristics**
The compressor with “Scooping type” vane gradually increased noise at 100 hours after the operation was started, and increased noise by about 20dB (A) after 400 hours. On the other hand, in the case of “Trailing type” vane, the compressor stably maintained low noise over 800 hours. These results indicated that the increase of noise in the “Scooping type” would be caused by the jumping phenomena of the vane.

**Input change characteristics**
In the case of the compressor with the “Scooping type” vane, small input was stably maintained, whereas in the case of the “Trailing type,” the increasing phenomenon of the input appeared after 100 hours. It is assumed that this phenomenon is due to the increase of frictional coefficient $\mu_1$ as in the case of the increase of noise on the “Scooping type.” The increase ratio of the input in the case of “Trailing type” is as large as 50%. But, since the absolute value of the input after the long operation is as small as 1.5W, it is assumed that this wouldn’t cause any problem in actual use.

These experimental results of noise characteristics and input characteristics were compared with the dynamic analysis results discussed before, and conclusively, the “Trailing type” was chosen as the appropriate vane sliding form.

![Fig.7 Noise change characteristics of the air compressor](image_url)
By going through these development processes, the authors were able to develop an air compressor for mobile fuel cells with features such as small size, low noise, and small input power, which have never been seen before.

5. CONCLUSION

We made efforts to develop a small size rotary vane type air compressor with high performance for mobile fuel cells, and the following conclusions were obtained.

(1) An analytical method was introduced to calculate by Excel the forces acting on the vane. It has been confirmed that the analytical results are effective for investigation of noise around the vane.

(2) In the rotary vane type air compressor, which operates under the oil-less atmosphere, by adopting “Trailing type” vane, the vane jumping phenomena can be prevented and long time noise stabilization is achieved.

By the way, for this air compressor, various kinds of reliability tests are presently underway. We hope that this air compressor will put into practical use in the near future, and offer the advantages on the mobile fuel cells.

REFERENCES

(1) I HONDA , K KANAZAWA, H OOBA, T KONDO, and Y NAKASHIMA, A study on a Sliding Vane-Type Compressor, TRANS. JSME, Vol. 57, No.534 (1991), pp168-172
(2) M FUKUTA, M TANAKA, T SHIMIZU, and T YANAGISAWA, Analysis of Film on Vane Slides of Vane Compressors, TRANS. JSME, Vol. 57, No.538 (1991), p81
Design of large scroll compressors

C. Ancel, P. Ginies, D. Gross
Danfoss Commercial Compressors, France

ABSTRACT

This paper investigates the possibility of building large scroll compressors for air conditioning and refrigeration. Orbiting and co-rotating technologies were evaluated and compared. Compressors key points like material, arrangement, clearances, dimensions, weights, bearing loads, lubrication were considered for two fluids: R134a (low density) and R407C (medium density). Rules for sizing the involutes and the bearings were determined according to the pattern of forces in each case. The critical points of the design were underlined for both technologies.

1. INTRODUCTION

The current scroll compressors for air conditioning applications available on the market have a cooling capacity ranging from 1.5 tons (~5kW) to 30 tons (~32kW). There are various possibilities to increase the maximum cooling capacity but the usual way is to increase displacement. One interesting question was to find out what the maximum cooling capacity would be, using one pair of scrolls at a typical speed of 3500 rpm.

This study was with a motor mounted on the low pressure side, in order to have the motor cooled by the refrigerant. This choice was determined by the use of a low motor coil temperature, because a high pressure side motor configuration does not allow the maintaining of a safe temperature for copper over the whole operating range.

Two scroll technologies were evaluated based on the kinematics of the scrolls pairs: an orbiting scroll motion and a co-rotating scroll motion.

Various criteria are used to decide if a scroll design is acceptable or not:

- Resistance criteria, such as the maximum stress at the bottom of the involute wall,
- Deformation criteria, such as the deflection of the involute under loads,
- “Good Proportions” criteria, such as the maximum thickness of the wall, or the Aspect Ratio (ASR) of the involutes - i.e. the ratio of the involute height over the pitch. The maximum scroll height is determined by the optimum thickness of the wall in relation to the orbiting scroll stability. The ASR helps to find the right proportions to limit the thrust bearing energy losses,
- The journal bearings size.

Two fluids were investigated: R134a (low density) and R407C (medium density).

2. THE ORBITING SCROLL TECHNOLOGY

2.1 Description
An orbiting scroll compressor (fig 1 and 2) consists of a fixed and an orbiting scroll arranged within a hermetic shell. The orbit motion is performed by the eccentric motor shaft and an Oldham coupling. The orbiting scroll is in orbit around the main compressor axis, but the axes of this orbiting scroll keep the same directions in relation to a fixed coordinates system. Pairs of compression pockets are formed between the scrolls and the pocket volume is decreasing in a direction towards the center of the scrolls resulting from the orbiting movements.

The orbiting scroll is driven using a controlled orbit design, which means that the motor shaft directly drives the orbiting scroll without any sliding intermediate parts such as swing links or slide blocks. A precise machining of the components enables the use of this driving principle of controlled orbit. If the tightness needs being improved because of the machining precision (or clearances and tolerances), a sliding part can allow a radial movement of the orbiting scroll to try to correct the relative position of the scrolls, using either the inertia force or a spring.

Thanks to its controlled orbit and precise scroll profile, the orbiting scroll rolls and slides on an oil film with no friction or wear, securing absolute radial tightness. In the axial plane the scroll tips (fig 3) are dynamically kept in contact with the opposite scroll base via opposed floating seals. This technology reduces the contact surface area, but the contact pressure as well, substantially reducing friction losses and increasing efficiency. The soft touch of the tip seals reduces the noise level and is able to compensate both mechanical deflexion and thermal expansion.

Orbiting scroll basic design

Controlled orbit design

![Orbiting Scroll Basic Design](Fig1)

![Controlled Orbit Design](Fig2)
The forces acting on this type of orbiting scroll are the following:

FORCES ACTING ON THE ORBITING SCROLL (fig 4)

- \( F_{ag} \) = the axial gas force resulting from the pressure difference between the internal scroll pockets and the suction pressure. This force is applied midway of the axes of the mobile and fixed scrolls. This vertical force is not reported on sketch.

- \( F_{tg} \) = the tangential force resulting from the internal pressure differences between pockets. This force is perpendicular to the pockets sealing points lines and is applied at the middle point between the main axis of the mobile and of the fixed scroll.

- \( F_{tg \ TR} \) = the transmitted tangential force at the pin center

- \( F_{rg} \) = the radial force resulting from the internal pressure differences between pockets. This force is parallel to the pockets sealing point lines and is applied at the center of the orbiting scroll.

- \( OSINF \) = the Orbiting Scroll Inertia Force resulting from the orbit motion. This force is opposed to the radial force and helps to maintain the tightness between the sealing points. It is applied at the center of gravity of the orbiting scroll. \( OSINF \) contributes to the sealing of the pockets by compensating for the assembly clearances and tolerances. This force, which is always greater than \( F_{rg} \), pushes the orbiting scroll following the contact lines in the \( OSINF \) centrifugal direction. This enables the controlling of the components clearances and tolerances, in order to achieve an ideal positioning pattern, which sets the possible gap between the involutes at its minimum value.

The orbiting scroll orbits following the M direction around the B axis. \( F_{tg} \) is transmitted at the pin center location, where the local resistant force is \( F_{tg \ TR} \). \( F_{tg \ TR} \) is equal to \( F_{tg} \), hence the resistant torque on the shaft is \( F_{tg} \times Ro \). Since there is a difference between the actual torque on the orbiting scroll (\( F_{tg} \times Ro / 2 \)) and the transmitted torque (\( F_{tg} \times Ro \)), this torque difference is supported by the Oldham coupling.
Frg and OSINF do not generate any torque on the shaft, because their direction passes through the orbit axis.

The forces applied to the orbiting scroll bearing in the eccentric direction are OSINF and Frg. Ftg is perpendicular to these forces. The resulting force has a fixed direction in the crankshaft rotating coordinates system.

2.2 Mechanical design
The orbiting scroll stability was studied using a mechanical modelization, which included the computation of all forces and torques. The necessary reaction force to stabilize the system was computed. The stability limit was defined as the minimum required diameter for the thrust bearing in order to keep the origin of the reaction force within this diameter. This led to a practical limitation of the ASR parameter, hence of the involute height.

The bearing design requires larger bearings for large compressors. When the orbiting scroll bearing diameter increases, the inner diameter of the thrust bearing increases too, and this in turn increases the unsupported surface of the orbiting scroll. In order to keep an acceptable parts deflection, the orbiting scroll base plate height (thickness) has to be greater, which significantly increases the orbiting scroll mass and this increases again the inertia force...which makes bearings even larger.

2.3 Results
Orbiting scroll design
The investigated materials include a light metal alloy, in order to decrease the inertia force, which is a limiting factor for orbiting scrolls.

The orbiting scroll mass (fig 5) dramatically increases with the cooling capacity but the low density R134a leads to high weights which can be compensated using a light metal alloy to build the orbiting scroll : the weight can get even lower than the weight of a cast iron orbiting scroll using medium density R407C.

With the plotted weight per kW (fig 6) over the cooling capacity graph, it appears that the weight per kW is greater for a big scroll compressor than for a small one. This is due to the circular shape of the orbiting scroll, which makes the weight increase quicker than the cooling capacity.
The ratio of the inertia force OSINF over the drive force (fig 7) $F_{tg}$ increases with the cooling capacity and the ratio of 1.5 (considered as critical) is quickly reached, which can limit possibilities for manufacturing large scroll compressors, since inertia represents too big a part of the load compared to the drive force $F_{tg}$.

The force at orbiting scroll bearing (per kW) (fig 8) increases with the cooling capacity, leading to big size bearings with the associated friction losses, which are damageable to performance. Nevertheless, for one curve plotted for medium cooling capacities (R407C + cast iron), an optimum value exists.

2.4 Analysis
First step: selecting scroll geometries
With the ASR criterion and maximum involute wall thickness, the scroll geometries were chosen.

Second step: compatibility with journal bearings
The journal bearings are calculated for two main conditions, the first one is the high load condition and the second one is the flooded condition (lubrication with a low viscosity mixture of oil and liquid refrigerant).

With current medium size scroll compressors, the key criterion is the bearings calculation at high load, with high forces and an average oil viscosity. The load is mainly the gas forces and depends mainly on the operating conditions.

With large scroll compressors, the key criterion is the flooded condition, with already high forces and a low oil viscosity. The load is mainly the inertia load and does not depend a lot on the operating conditions.

This has an impact for large scrolls:
- firstly, the inertia load is applied instantaneously when the compressor starts, this will require a fast oil supply system for the bearings.
- secondly, for large orbiting scroll compressors, the bearing design criterion is the flooded condition: the load is already high and the oil viscosity is low.

The bearing design for low oil viscosity is possible, but for the large orbiting scroll compressors design, it becomes difficult. The high inertia load at low viscosity requires a large bearing in order to create a sufficient oil film thickness, but in normal conditions the oil
viscosity is greater, and this increases bearing losses. A good compromise is to use the smallest oil film thickness in the low viscosity case, and a small bearing clearance, but due to the high inertia forces, the crankshaft deflected profile has to match the deflected profile of the journal bearing, which is required in order to avoid bearing edge loading.

**Oldham coupling design**
Large scroll compressors require “high” orbiting radiiuses and larger orbiting scroll diameters. The Oldham coupling diameter will be larger as well. To keep a deflection matching the requirements, the Oldham ring section must also be large which increases the Oldham coupling mass. The coupling stroke associated with the orbit motion increases, thus the acceleration increases too - and this increases the Oldham contact surfaces load.

**Conclusion**
The first part of the study shows that for large compressors the inertia force is great but there are possibilities to deal with this situation. Moreover, it was found that a scroll compressor less sensitive to inertia would be interesting to evaluate.

The other two types of technologies are the co-rotating scrolls and the co-orbiting scrolls. It was decided to evaluate the co-rotating design first because the two scroll wraps are rotating around their axis and there are no heavy parts in orbit.

3. **THE CO-ROTATING SCROLL TECHNOLOGY**

3.1 **Description**
A co-rotating scroll compressor consists in a master and slave scrolls (fig 10 and 11) arranged within a hermetic shell. A motor gives a rotating movement to a master and to a slave scroll. The axes of the master and of the slave scroll (fig 9) are separated by a distance equal to the orbit radius used for the usual orbiting arrangement. A mechanical device is necessary to drive the slave scroll that turns around a free axis. The absolute movements of scrolls in relation to the compressor frame are true rotations. The relative movement between the scrolls is still an orbit motion in a rotating coordinates system attached to one of the scrolls. The pocket volume is decreasing in a direction towards the center of the scrolls resulting from the rotating movement. The controlled orbit design is once again a good solution, especially because the inertia force is not available to move away the master scroll using a sliding part. A sliding part design would have to resort to the force of a spring to improve the tightness.

In the same way, the axial tightness is achieved with tip seals.

The scrolls and the thrust bearing cannot be arranged like in the orbiting scroll case, since the peripheral speed of the involutes supporting plates increases with the radius. As a consequence, the points at the periphery would have a high speed, while speed is constant at any point of the
supporting plate for the orbiting scroll design. This high speed combined to the standard Fag force could lead to an excessive friction on the thrust bearing surface, and cause damages. This is why the following design is proposed in order to set the usual relative speed (i.e. the same speed as for an orbiting design) between the co-rotating master scroll and the thrust bearing of the slave scroll. The following assembly is without apparent axial force Fag.

If the inertia force disappears, then the Frg force tends to separate the involutes, hence the possible gap is set at its maximum value according to the parts clearances and tolerances. As a consequence, the co-rotating design needs special care to secure the tightness at sealing points.

The scroll stability is computed with rules, which are different from the orbiting case, but based on the same principle with the search of the application point for the stabilizing reaction force. The stability is not a limiting factor for co-rotating scrolls. The effect of the centrifugal force and an excessive involute wall thickness are more a problem.

Another special feature of co-rotating scrolls is the need for a rotating seal in order to discharge the compressed gas.

Since there is no eccentric mass in rotation, the counterweights are suppressed, which is an advantage.

The inertia force is suppressed for co-rotating scrolls. The forces acting on such an orbiting scroll set are the followings:
FORCES ACTING ON THE MASTER SCROLL  (fig 12)

- $F_{ag}$ = the axial gas force resulting from the pressure difference between the internal scroll pockets and the suction pressure. This force is applied midway of the axes of the master and slave scrolls. This vertical force is not reported on sketch.

- $F_{tg}$ = the tangential force resulting from the internal pressure difference between pockets. This force is perpendicular to the pockets sealing points lines and is applied at the middle point between the main axes of the master scroll and of the slave scroll.

- $F_{rg}$ = the radial force resulting from the internal pressure difference between pockets. This force is parallel to the pockets sealing points lines and is applied at the center of the master scroll. This force tends to separate the two scrolls and can create leakages.

The master scroll rotates following the M direction around the A axis, while the slave scroll rotates following the S direction around the B axis. $F_{tg}$ M directly generates a resistant torque $F_{tg}$ M * Ro / 2 and $F_{tg}$ S directly generates a resistant torque $F_{tg}$ S * Ro / 2.

If a single motor is used, $F_{tg}$ S * Ro / 2 is reported on the master scroll by a transmission device, hence the total resistant torque on shaft will be $F_{tg}$ * Ro.

$F_{rg}$ M and $F_{rg}$ S do not generate any torque on shafts, since their direction passes through the orbit axes.

The sealing points lines keeps the same direction, since the axes positions are fixed. As a consequence, the direction of forces is also constant.

3.2 Mechanical design
The scroll bearing load comparison with the orbiting technology shows that in the co-rotating design the inertia is ‘0’ and the bearing forces are mainly the combination of the radial gas force and the tangential gas forces. As the radial gas force tends to separate the two scrolls, and since there is no inertia force, the bearing radial clearance and parts geometry allow master and slave scroll axes motions on the bearings plane and consequently increase the leakage potential. Conversely, with the orbiting scroll, the inertia force tends to pull the orbiting scroll in order to get the largest possible orbiting radius (this tends to reduce the scrolls radial clearances).

The force directions are fixed in the compressor frame coordinates.

To start the compressor, the momentum of all turning parts is greater than that of an orbiting compressor, and the motor start cycle must be adapted to prevent damaging the motor with current surges.
3.3 Design Analysis

Involutes design

A finite elements analysis has been carried out to evaluate the impact of the rotational speed on the parts.

The figures 13 below show in dark the critical stress zones with the action of the centrifugal force on the peripheral areas.

A number of modifications were simulated to prevent the deformation of the involutes outer edge caused by the centrifugal force:

<table>
<thead>
<tr>
<th>Version</th>
<th>Modification</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Relative radial end displacement of the wrap</td>
<td>0.618</td>
<td>0.474</td>
<td>0.600</td>
<td>0.705</td>
<td>0.370</td>
</tr>
</tbody>
</table>

The best results were achieved with the combination of the thicker base (version 2) and the tangential rib (version 5).

Bearings

The force on the orbiting scroll bearing is lower for a co-rotating scroll than for an orbiting scroll, since there is no inertia force in the co-rotating case. Thus the bearings of a large co-
A rotating compressor will be computed mainly with the high load condition, as for the current medium size orbiting scroll compressors.

The crankshaft deflection may be also the key factor for bearing sizing instead of the oil film thickness.

Another key factor may be the forces level when the compressor starts. For the orbiting scroll, the kinetic energy is given by the scroll mass and the orbit radius. For the co-rotating scrolls, the kinetic energy is greater because the two scrolls rotate around their own axis. During the starting cycle, the motor starting torque produces forces shared between the scroll wrap and the Oldham coupling, which values depend on the momentum ratio between the master momentum and the slave momentum.

Moreover, the evaluated design shows that the oil delivery time to the bearings is critical, particularly for the slave upper bearing. A local oil reserve could be a solution to reduce the oil delivery delays.

The present oil delivery system used for the orbiting design would be reviewed taking care that for the co-rotating, loads directions are fixed.

**Oldham coupling**
The Oldham coupling motion is reduced and this gives less unbalance and less inertia forces on the Oldham coupling keys. For a more compact design, the synchronization between the two scrolls could be achieved with small cranks but the lubrication design would have to take this situation into account.

**Slave scroll design**
The slave scroll could be driven using either an Oldham coupling or small cranks. The lubrication of the slave scroll bearing is a difficulty, and canals inside the small cranks may be a solution.

### 4. CONCLUSION

For the design of large scroll compressors, additional criteria must be taken into account:

- **Orbiting scroll**
  - the orbiting scroll inertia force is large and changes the selection of the scroll geometry
  - the Oldham coupling inertia could be an issue.
  - the counterweights size is not always compatible with a simple thrust bearing design
  - the bearings dimensions depend on the low viscosity condition, which is the most critical

- **Co-rotating scroll**
  - the radial clearance is not naturally set at its minimum for co-rotating compressors
  - lubrication design is complicated
  - the design has to take into consideration the wrap deflection produced by the centrifugal force
This study highlights key points between Orbiting and Co-rotating scroll designs, with a motor mounted on the low pressure side. The maximum cooling capacity directly depends on the material the scrolls are made of. Because the size is the limiting factor for scrolls, it is clear that there is a big interest in using light materials with a high resistance, provided that the cost and the machining possibilities are economically viable.
NOVEL DUTIES
Heated scroll expander and its application for distributed power source

Y. M. Kim, D. K. Shin, J. H. Lee
Korea Institute of Machinery & Materials, Korea

ABSTRACT

Rankine cycle engines with a scroll expander, currently being developed for domestic cogeneration systems, have relatively low electrical conversion efficiency, because the scroll expander is being operated in the relatively low temperature range of about 200–300°C and has not been developed for expansion of high temperature gas. The main reason for that is thermal deformations of scroll elements due to the temperature drop of the expanding gas in the scroll expander, which causes different clearances in radial and axial directions. This paper proposes a heated scroll expander for the purpose of reheating the expanding gas, which can reduce a difference in thermal expansion between the innermost and outermost zones of the scroll wraps and simultaneously improve the cycle efficiency. The heated scroll expander can be used in the gas power cycle (Ericsson cycle) as well as in the vapor power cycle (Rankine cycle), in which the reheat process by heating the scroll expander and regeneration process by a recuperator contribute to improve the thermal efficiency and specific power of its system. From a preliminary experimental study, it might be said that the scroll expander can be fairly suitable for effective reheating due to the high area-to-volume ratio of scroll geometry and good gas-to-scroll heat transfer.

1. INTRODUCTION

Recently, micro-CHP (combined heat and power) replacing domestic boiler is expected to create a great potential market in the near future. Especially, as a prime mover for micro-CHP, the external combustion engines including the Stirling engine and the Rankine cycle engine are very promising, due to their low noise, vibration and emission levels. Some companies are trying to commercialize the Rankine cycle engine with a scroll expander for micro-CHP system [1][2]. However, their systems have the relatively low electrical conversion efficiencies, because the scroll expander is being used in the relatively low operating temperature range around 200–300°C [3]. The main problem in operating the scroll expander for expansion of high temperature gas is a difference in thermal expansion of scroll elements due to the temperature drop of the expanding gas [4][5].
Before the introduction of this study, the other point to be noticed is that some works on the scroll compressor have found that effective cooling of the gas during the compression in a scroll compressor could be achieved due to the high area-to-volume ratio of scroll geometry and good gas-to-scroll heat transfer. Raymond W. Moore, Jr [6] pointed out that when the scrolls were cooled by circulating city water through flow passages in the fixed scroll only, the actual discharge temperature of helium with a pressure ratio of 10 and an inlet temperature of 24°C was only 110°C, while the adiabatic discharge temperature of helium would be 463°C. Recently, Air Squared’s scroll air compressor [7] eliminates the need for an aftercooler by allowing a temperature rise of just 52°C with a pressure ratio of 9, while the adiabatic temperature rise of air would be 256°C.

As a beginning of this study, it has been supposed that, inversely in the scroll compressor, the effective heating of the gas during the expansion in a scroll expander could be achieved. Then, in this paper, a heated scroll expander is proposed for the purpose of reheating the expanding gas to reduce a difference in thermal expansion between the innermost and outermost zones of the scroll wraps and simultaneously improve the thermal efficiency of the cycle. It can be used in a gas power cycle as well as the vapor power cycle. The characteristics for both cycles are presented in this paper. In addition, a preliminary experimental study was carried out to investigate the effect of heating the experimental scroll expander on the temperature drop of expanding gas, with air as a working fluid.

2. HEATED SCROLL EXPANDER

![Fig. 1 Temperature drop in the scroll expander](image1)

![Fig. 2 Heated scroll expander](image2)

For the scroll expander to produce power efficiently, as shown in Fig. 1, all of the radial width of gas pocket (1, 2, ..., 6) between the flanks of the scroll wraps in line contact must be equivalent to two times the radius of orbital motion by rotation of drive shaft, and the axial
clearances between the bottom or top plate and the scrolls must be minimized. However, if the scroll expander is operated for the expansion of high temperature gas, the continuous expansion of the gas causes a wide range of temperature distribution over the whole scroll wrap. This leads to a difference in thermal expansion between the innermost and outermost zones of the scroll wraps and causes different clearances in radial and axial directions. In this paper, a heated scroll expander is proposed for the purpose of reducing the difference in thermal expansion. In the process of expansion, if heat is added to the working fluid in the expansion space from an external source, the temperature drop of the expanding gas can be reduced. This also contributes to improvements in the thermal efficiency and specific power of its systems. To reduce the difference in thermal expansion, it is desirable to heat the scroll expander as uniformly as possible so as to maintain the same temperature over the whole scroll wrap. However, it is not easy to heat the scroll expander uniformly by direct heating by flames. So a flame-heated heat pipe coupled to the external fins of a double-sided scroll expander is preferably proposed as the best way, as shown in Fig.2.

3. APPLICATIONS OF THE HEATED SCROLL EXPANDER

3.1 Reheat and Regenerative Rankine cycle

As mentioned earlier, the cycle efficiency of the present Rankine cycle engine with a scroll expander for micro-CHP system is much lower than other prime movers because the scroll expander is operated in the relatively low temperature range. However, the proposed heated scroll expander is designed to run in high temperature range by accommodating the difference in thermal expansion of scroll elements. If it is applied in the Rankine cycle engine as shown in Fig.3, in which the expanded hot steam is used to heat feed water from the water pump in a counter-flow heat exchanger (recuperator), it can be possible to improve the performance of the engine with a increase in the operating temperature of scroll expander. Both reheat process and
regeneration process contribute to improve the thermal efficiency and specific power of the engine. Although this advantage is not big in the low operating temperature of 200~300°C, it could be larger with the higher operating temperature.

3.2 Scroll-type Ericsson cycle engine/refrigerator
The Ericsson cycle is very much like the Stirling cycle in that both of two cycles consist of isothermal compression, isothermal expansion and regeneration processes, except that the two constant-volume processes are replaced by two constant-pressure processes. The efficiency of the Ericsson cycle is the same as that of the Carnot cycle, as in the Stirling cycle. However, Stirling and Ericsson cycles are difficult to achieve in practice because neither the isothermal compression nor isothermal expansion is practical in the cylinder or with the turbomachinery due to the insufficient area for heat transfer and the short time for the process. In the Brayton cycle with turbomachinery, as the number of compression and expansion stages is increased, the Brayton cycle with inter-cooling, reheating, and regeneration processes will approach the Ericsson cycle with the Carnot efficiency. However, this inevitably leads to the added complexity of the turbomachinery and the need for many additional components. In this paper, it is proposed that the scroll-type compressor and expander is fairly suitable for approaching the isothermal compression and expansion processes due to the high area-to-volume ratio of scroll geometry compared to other machines. The new-type Ericsson cycle engine consists of one pair of scroll compressor and scroll expander, as shown in Fig.4. In the process of compression, the heat generated in the compression space between the orbiting scroll and the fixed scroll of the scroll compressor is carried to the external dump (cooling fluid) through the inner scroll wrap and external cooling fins. Then the compressed working fluid enters the inlet port of the scroll expander. In the process of expansion, heat is added to the working fluid in the expansion space from an external heat source (heating fluid) through external heating fins and the inner scroll wrap. Expanded hot and compressed cold working fluid streams enter the counter-flow heat exchanger from opposite ends, and heat transfer takes place between them. But even in the scroll-type Ericsson cycle engine as shown in Fig. 4, since the processes of compression and expansion are not ideally isothermal, it may have remote heater and cooler. The more closer to isothermal process the processes of compression and expansion approach, the smaller the heater and the cooler become. Figure.5 shows a comparison of P-V diagrams for the Brayton and Ericsson cycle engines with air (specific heat ratio, k=1.4) as a working fluid, between given limits of pressure and temperature. In general, the gas compression or expansion process can be modeled as polytropic \((Pv^n = constant)\) process, where the value of \(n\) varies between \(k\) and 1: an isentropic (\(n=k\)) process (involves no heat transfer), an polytropic (\(1<n<k\)) process (involves some heat transfer), and isothermal (\(n=1\)) process (involves plenty of heat transfer). The shaded areas \(A_1\) (decrease in compression work, \(W_{comp}\)) and \(A_2\) (increase in expansion work, \(W_{exp}\)) represent the additional work made available by substituting isothermal processes for adiabatic processes. The reversed Ericsson cycle, like the reversed Brayton cycle, can be used for a refrigerator (or heat pump) if drive power is input and then heat can be absorbed into the expansion space at a temperature lower than that of the compression space, as shown in Fig.6. Figure.7 shows a comparison of P-V diagrams for the reversed Ericsson and Brayton cycles with air as a working fluid, between given limits of pressure and temperature. The shaded areas \(A_1\) (decrease in compression work, \(W_{comp}\)) and \(A_2\) (increase in expansion work, \(W_{exp}\)) represent the reduced input of work and the greater cooling capacity by substituting isothermal processes for adiabatic processes.
Compression Side Expansion Side

Fig. 4 Scroll-type Ericsson cycle engine

Fig. 5 Brayton and Ericsson cycle (P-V plot)
3.3 Comparison of the Rankine cycle engine and the Ericsson cycle engine

For the micro-CHP system with the heated scroll expander, as mentioned earlier, two kinds of systems are possible, the Rankine cycle engine and the Ericsson cycle engine. The comparison of the two systems is summarized in Table 1. In the Rankine cycle, the back work ratio, defined as the ratio of the compression work to the expansion work, is a few percent, because the working fluid is compressed in liquid state. However in the Ericsson cycle, the back work...
ratio is very high in the range from 30% at a high operating temperature of 900°C to 80% at a low operating temperature of 200°C, and the situation is even worse when the efficiencies of the compressor and the expander are low. Practically, at a low operating temperature range of 200 to 300°C, the Rankine cycle is preferable to the Ericsson cycle which may produce no output at all due to the very high back work ratio. On the other hand, at a high operating temperature range of 700°C, the theoretical maximum thermal efficiency of the Ericsson cycle can reach up to 60%, approaching the Carnot cycle, and much higher than 40% of the Rankine cycle. It is supposed that the Rankine cycle is easier to commercialize in the short term than the Ericsson cycle since it does not necessarily need a high temperature resistant scroll expander but has its simple system. However, in the long term, the Ericsson cycle is preferable to the Rankine cycle if the more efficient and higher temperature resistant scroll expander can be developed. It is because the practical thermal efficiency of the Ericsson cycle is very sensitive to the efficiencies of the compressor and the expander in spite of the potentiality of high thermal efficiency.

**Table 1. Comparison between Rankine cycle engine and Ericsson cycle engine**

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Rankine</th>
<th>Ericsson</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Compression</strong></td>
<td>Liquid state</td>
<td>Gaseous state</td>
</tr>
<tr>
<td><strong>Expansion</strong></td>
<td>Gaseous state</td>
<td>Gaseous state</td>
</tr>
<tr>
<td>$W_{\text{comp}}/W_{\text{exp}} (\text{rbw})$</td>
<td>1~2%</td>
<td>30%(at 900°C)~80%(at 200°C)</td>
</tr>
<tr>
<td><strong>Low Temp. range</strong></td>
<td>Ideal</td>
<td>High $W_{\text{comp}}/W_{\text{exp}}$ may produce no output work</td>
</tr>
<tr>
<td>(200~300°C)</td>
<td>Approach Carnot cycle</td>
<td></td>
</tr>
<tr>
<td><strong>High Temp. range</strong></td>
<td>Max. thermal efficiency ~ 40% (different from Carnot cycle)</td>
<td>Max. thermal efficiency ~ 60% (approach Carnot cycle)</td>
</tr>
<tr>
<td>(700°C)</td>
<td>Simple configuration and system matching</td>
<td>Important system matching with compressor and expander</td>
</tr>
<tr>
<td><strong>System matching</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Weak point</strong></td>
<td>Limited max. thermal efficiency</td>
<td>Real thermal eff. is very sensitive to eff. of compressor and expander</td>
</tr>
<tr>
<td><strong>R &amp; D strategy</strong></td>
<td>Easy to commercialize in short term</td>
<td>Ideal with high efficient &amp; high temperature scroll expander</td>
</tr>
</tbody>
</table>

### 4. PRELIMINARY EXPERIMENT OF THE SCROLL EXPANDER

A experimental investigation on the potential isothermality of the scroll compressor was carried out with a experimental scroll compressor modified from a commercial scroll air compressor, as shown in Fig.8. Its designed pressure ratio is 8, and it has many cooling fins on both the fixed and orbiting scrolls together with a cooling fan. Furthermore, it was modified to be suitable for water-cooling. The outlet air temperatures for different cooling methods were measured, maintaining the inlet air temperature at 20°C and the discharge air pressure at 800kPa, the designed pressure ratio of the scroll compressor. And it was tested for four speeds of rotation varying between 1000 and 2500 rpm. Figure 9 shows the results of the test of the scroll compressor. For the case 1 with the original air-cooled orbiting and fixed scrolls, the outlet temperature goes up to 150°C from the inlet temperature of 20°C at 2500 rpm. For the case 2 with the air-cooled orbiting scroll and the water-cooled fixed scroll, there is a distinct
improved cooling effect at low speeds. However, as the revolution speed is increased, the improved cooling effect is decreased. For the case 3 with both the water-cooled orbiting and fixed scrolls, the cooling effect is so much more improved that the outlet temperature is about 90°C at 2500 rpm. Theoretically, the adiabatic discharge temperature for air would be 256°C, with a pressure ratio of 8 and a inlet temperature of 20°C.

![Fig. 8 Scroll compressor/expander modified from a commercial scroll air compressor](image)

In the same way, a experimental investigation on the potential isothermality of the scroll expander was carried out with an experimental scroll expander in backward mode of the scroll compressor. The outlet air temperatures for different cooling methods were measured at each revolution speed, while maintaining the inlet air temperature at 200°C and the pressure at 800kPa, with a direct discharge to ambient air. Figure 10 shows the results of the test of the scroll expander. For the case 1 with no-heating of scroll element, the outlet temperature goes down to 50°C from the inlet temperature of 200°C at 2000 rpm, with the temperature drop of 150°C. Theoretically, for that case, the adiabatic temperature drop of air would be 212°C. For the case 2 when the fixed scroll is heated by thermal oil at a temperature of 200°C, the temperature drop of the air is decreased to 70°C. Furthermore, for the case 3 when both the fixed and orbiting scrolls are heated, the temperature drop of the air is reduced to 40°C. It is supposed that if the scroll expander can be well designed for effective heat transfer, the more effective heating can be achievable.

The noticeable point in the result of scroll expander is that the temperature drop is almost irrelevant to the revolution speed, that’s different from the case of the scroll compressor. It can be explained in the following way. In the compressor mode, the contacting area for cooling is decreasing with compression into the center and the temperature increase. However, in the expander mode, as the gas is expanded to the outer space with the temperature drop, the contacting area for heating is increasing and sufficient to be irrelevant to the revolution speed. Therefore, it can be supposed that with the scroll machines, expansion process can approach the more closer to the isothermal process than compression process.
Fig. 9 Effect of cooling the scroll compressor

Fig. 10 Effect of heating the scroll expander
5. CONCLUSIONS

In this paper, a heated scroll expander and its applications are proposed, and a preliminary experimental study was carried out to investigate the effect of heating a scroll expander. The following conclusions can be drawn from the present study.

1) A heated scroll expander was proposed to reheat the expanding gas for the purpose of reducing a difference in thermal expansion between the innermost and outermost zones of the scroll wraps and simultaneously improving the thermal efficiency of the cycle.

2) The proposed scroll expander can be used both in a gas power cycle (Ericsson cycle) and in a vapor power cycle (Rankine cycle), furthermore in a gas refrigeration cycle (reversed Ericsson cycle). Both the reheat process by heating the scroll expander and the regeneration process by a recuperator contribute to improve the cycle efficiency of its systems.

3) From a preliminary experimental study, it might be said that the scroll expander can be fairly suitable for effective reheating due to the high area-to-volume ratio of scroll geometry and good gas-to-scroll heat transfer.

6. REFERENCES

Operating performance of scroll expander working with water-mixed air

Tadashi Yanagisawa, Mitsuhiro Fukuta, Yasuhiro Ogi, Emi Yamada
Department of Mechanical Engineering, Shizuoka University, Japan

ABSTRACT

Performance of a scroll-type air expander working with water mixed air was investigated theoretically and experimentally. The theoretical performance was analyzed with a mathematical model taking account of the passage loss and the leakage loss. The experimental expander was realized by supplying high-pressure air with water to an oil-free scroll air compressor and letting it rotate in the reverse direction. The water mixing into the supply air had two effects on the expander performance: decreasing the supply air flow rate and increasing the output shaft torque. The performance of the expander is not sensitive to the water/air volume ratio in the supply air, though the ratio must be kept high to avoid the frost formation at the expander exit.

1. INTRODUCTION

Scroll type fluid machinery can be used for both compressors and expanders based on its working principle. For the compression purpose, it is put into practical use as air compressors and refrigerant compressors. Also oil-free scroll air compressors and vacuum pumps are now on the market (1). On the other hand, for the expansion purpose, several works have been reported on an oil-lubricated air expander (2) and an oil-lubricated refrigerant expander for the Rankine cycle (3, 4). Though practical scroll expanders have not appeared on the market, it will be realized soon as the expander for power recovery in CO₂ refrigeration cycles. Oil-free scroll expanders are another prospective application of scroll machinery in the near future, and one report (5) was presented by us at the 2001 IMechE conference on compressors and their systems.

In this study, performance of a scroll air expander working with water mixed air is investigated theoretically and experimentally. The test machine is realized by conversion from an oil-free scroll air compressor on the market. Operating performance of the scroll air expander under the no-water mixed condition reported by us (5) was promising. To pursue the possibility to improve the efficiency of the scroll air expander, water mixed air is supplied to the expander and its performance is analyzed.
2. THEORETICAL ANALYSIS

Figure 1 shows a schematic view of a pair of scroll wraps, so-called fixed and orbiting scrolls, formed by the involute spiral. When the scroll wrap combination is run as an expander, high-pressure air is supplied to an intake port at the center of the fixed scroll and it expands in the scroll pockets, which forces the orbiting scroll to move around with a constant radius in the fixed scroll.

The expansion performance of the scroll expander is analyzed theoretically with the aid of a mathematical model modified from our scroll compressor model (6). The model calculates the pressure in scroll pockets with the assumption of the adiabatic or polytropic process in consideration of the change in each pocket volume, the flow through the intake port, the flow through the intake and exhaust openings at the starting and ending points of the scroll wraps, and the leakage flow through the clearances between the two scroll wraps. The flow through each passage is treated as a compressible flow through a throttling area with an appropriate flow coefficient. The tangential leakage flow through the wrap-radial clearance is treated as a compressible flow with fluid friction at a narrow channel. The radial clearance with a converging-diverging shape is approximated as a frictional channel with a constant cross-sectional area and an appropriate length, and the flow rate through the frictional channel is calculated as a Fanno flow in practice.

When water is mixed with the supply air, the leakage flow in the expander is treated as the gas-liquid two-phase flow. At the frictional channel, the Lockhart-Martinelli method for two-phase flow is applied for convenience.

Based on the pressure $P$ in the scroll pocket with volume $V$, the gas expansion work of the expander is calculated by integration of the $P-V$ work from the intake to exhaust processes. The flow rate of the expander is calculated as the substantial flow through the intake port. For the performance calculations, the flow coefficients at the intake port, the intake opening and the exhaust opening are assumed as 0.7 practically.
3. EXPERIMENT

As oil-free scroll expanders are not available on the market, a commercial oil-free scroll air compressor was converted into an oil-free air expander. The rated power and the nominal flow rate of the original scroll compressor are 1.5 kW and 160 L/min at 1920 rpm, and its operating performance has been reported by us (7). The experimental expander, with no modification of the compressor except the removal of a cooling fan, has a structure shown in Figure 2. The fixed scroll is mounted on the casing, and the orbiting scroll is supported by three auxiliary crank mechanisms that prevent the self-rotational motion of the orbiting scroll and meet the thrust load on the orbiting scroll. Tip seals are equipped on the tip of scroll wraps to control the leakage through the tip clearance of the wraps, while the crank mechanism with a constant radius keeps some clearance between flanks of the wraps to avoid mechanical contact. Main dimensions of the expander are; height of wrap: 23.5 mm, thickness of wrap: 4.5 mm, pitch of involute wrap: 20.5 mm, involute angles at starting and ending points of wrap: 0.31 and 7.25 $\pi$ rad; equivalent to the ideal intake and exhaust stroke volumes of 31.5 and 100.1 cm$^3$/rev, respectively. The built-in volume ratio of the expander is 3.18 and the built-in adiabatic pressure ratio is 5.05 for air as a working fluid.

In experiments, high-pressure air with or without water is supplied to the experimental expander, and is exhausted to the atmospheric pressure. The shaft of the expander is connected to a dynamo as a load, and the shaft torque is measured with a strain gauge type torque meter set between them. The flow rate of the supply air is measured with a rotameter. Pressure of the supply air and temperatures of the supply air and the exhaust air are measured with a Bourdon-tube pressure gauge and T-type thermocouples, respectively. At the same time, three piezoelectric pressure transducers mounted on the fixed scroll detect the instantaneous pressure variation in the scroll pockets. An electromagnetic sensor and an eddy current sensor detect the rotational speed and angle position of the shaft, respectively.

In order to investigate the influence of water mixing into the supply air on the expander performance, water in a pressurized reservoir is simply mixed to the supply air at a T-joint piping 0.3 m before entering the expander, and the water flow rate is measured with a gear type flow meter. In experiments, the expander is operated in the room atmosphere under various supply pressure, rotational speed and water/air ratio conditions. The volumetric and total adiabatic expansion efficiencies are obtained from the air flow rate and the output shaft torque.

4. RESULTS AND DISCUSSION

4.1 Performance under various supply pressure conditions

Figure 3 shows the volume flow rate $Q_s$ at the supply pressure condition and the volumetric efficiency $\eta_v$ of the experimental expander with and without the water mixing against the supply air pressure $P_s$ at the constant rotational speed $N = 1800$ rpm. The water mixing ratio $\alpha_v$ as a parameter denotes the volumetric ratio of mixed water to air at the supply pressure condition. $\eta_v$ is the ratio of the ideal supply air flow rate, which is decided by the intake stroke volume and the rotational speed of the expander, to the actual supply air flow rate $Q_s$. 

145
In Fig. 3, $Q_s$ shows a slightly increasing tendency, which leads to the slight decrease of $\eta_v$ with the increase of the supply pressure $P_s$ under the no-water mixed condition ($\alpha_v = 0$). Under the water mixed condition ($\alpha_v = 0.4\%$), $Q_s$ reduces almost evenly not depending on $P_s$ because of the sealing effect of the water on the internal leakage through wrap clearances. The corresponding $\eta_v$ increases by about 10\%, from about 70\% of no-water condition to about 80\% of water mixed condition, though its increase is slightly smaller at the higher $P_s$ condition that imposes the larger pressure difference across the clearance. In Fig. 3, theoretically calculated lines for $Q_s$ and $\eta_v$ are illustrated and they show the same tendency as the measured data. Details of the calculation method is described in the later section.

Figure 4 shows the measured shaft torque $T_s$, the indicated gas expansion torque $T_{ind}$ and the total adiabatic expansion efficiency $\eta_t$ corresponding to Fig. 3. $T_{ind}$ is obtained from the indicated $P$-$V$ work based on the measured pressure curve in the scroll pocket, and $\eta_t$ is derived as the ratio of the actual output power based on the shaft torque $T_s$ to the ideal output power that is expected when the supply air with actual flow rate expands adiabatically from the supply air pressure to the atmospheric exhaust pressure.

In Fig. 4, $T_s$ increases almost linearly in parallel with the ideal torque, which is decided by the scroll wrap geometry and the applied pressure condition, with the increase of $P_s$ under the no-water mixed condition ($\alpha_v = 0$). Under the water mixed condition ($\alpha_v = 0.4\%$), $T_s$ rises with the larger increase at the higher $P_s$. The rise is expected by the reduction of mechanical friction loss between the wraps due to the water as a kind of lubricant and by the increase of $T_{ind}$ as shown in Fig. 4. Based on the double effect of the decreased air flow rate and the increased shaft torque, the total expander efficiency $\eta_t$ increases by about 20\% by the water mixing not depending on $P_s$. $\eta_t$ shows a convex shape against $P_s$, and it reaches its maximum value of about 70\% at $P_s = 0.4$ MPa [gauge] in this case. In addition, the difference between $T_s$ and $T_{ind}$ means the mechanical loss in the expander, and it is relatively large because the tested scroll expander is originally designed as a compressor with nominal power of 1.5 - 2.2 kW, or nominal torque of 7 - 8 Nm. If the scroll machine is designed properly as an expander, the mechanical loss is reduced and its total efficiency will be improved.
$T_{\text{ind}}$ plotted in Fig. 4 is obtained from the measured pressure-angle ($P-\theta$) diagram shown in Fig. 5. The pressure during the expansion process under the water mixed condition ($\alpha_v = 0.4\%$) is much higher than that under the no-water mixed condition ($\alpha_v = 0$), which leads to the increase of the indicated work or the indicated torque $T_{\text{ind}}$. In Fig. 5, theoretically calculated pressure curves show the similar tendency to the experimentally measured curves. Details to obtain the theoretical pressure curve are discussed later.

### 4.2 Performance under various rotational speed conditions

Figure 6 shows the flow rate $Q_s$ at the supply pressure condition and the volumetric efficiency $\eta_v$ of the experimental expander with and without the water mixing against the rotational speed $N$ at the constant supply air pressure $P_s = 0.39 \text{ MPa [gauge]}$. $Q_s$ increases almost linearly with the increase of $N$, and it becomes small under the water mixed condition ($\alpha_v = 0.4\%$) as compared with that under the no-water mixed condition ($\alpha_v = 0$). The corresponding $\eta_v$ is also elevated by about 10% under the water mixed condition, and reaches 90% at $N = 2400 \text{ rpm}$.

Figure 7 shows the measured shaft torque $T_s$, the indicated gas expansion torque $T_{\text{ind}}$ and the total adiabatic expansion efficiency $\eta_t$ corresponding to Fig. 6. $T_s$ decreases almost linearly with the increase of $N$, though the ideal torque is constant not depending on $N$. It rises under the water mixed condition ($\alpha_v = 0.4\%$) as compared with that under the no-water mixed condition ($\alpha_v = 0$) because of the increasing indicated torque $T_{\text{ind}}$ and the decreasing friction loss by water as lubricant. But the increasing degree of $T_s$ becomes smaller at the higher $N$ because too much amount of mixed water at the higher $N$ with constant water volume ratio $\alpha_v$ becomes some resistance to the motion of the orbiting scroll beyond the lubricating and sealing effects. As a result of the decreased flow rate $Q_s$ and the increased shaft torque $T_s$ by the water mixing, the total expander efficiency $\eta_t$ is elevated greatly and its increasing degree is larger at the lower $N$ in this case.
4.3 Performance under various water/air ratios

Figure 8 shows performance of the experimental expander against the water/air volume ratio $\alpha_v$ under the constant rotational speed $N = 1800$ rpm and the constant supply air pressure $P_s = 0.39$ MPa [gauge]. In experiments, the water mixing ratio $\alpha_v$ was limited to values above 0.04% because the operation of the expander under the less water mixed condition was difficult due to the frost formation at the expander exit portion. With the increase of $\alpha_v$ in the range over $\alpha_v = 0.4\%$, the supply air flow rate $Q_s$ decreases slightly because of the reducing leakage loss and the increasing pressure drop at the inlet port, which leads to the slightly increasing volumetric efficiency $\eta_v$. On the other hand, the shaft torque $T_s$ and the indicated torque $T_{ind}$ are almost constant. These variations in the flow rate and the torque bring the slightly increasing total expander efficiency $\eta_t$ in the experimental water mixing range. When the water mixing ratio $\alpha_v$ is changed at the higher rotational speed, both $Q_s$ and $T_s$ decrease with the increase of the water ratio, which leads to the almost constant $\eta_v$. These results reveal that it is favorable to limit the water mixing ratio to the minimum value avoiding the frosting of the mixed water.

4.4 Investigation on theoretical analysis

As mentioned before, the water mixing into the supply air of the expander resulted in the decrease of the supply air flow rate and the increase of the shaft torque. The effect was also seen on the pressure-angle diagram of Fig. 5, in which the pressure curve under the water mixed condition ($\alpha_v = 0.4\%$) descended less steeply during the expansion process than one under the no-water mixed condition.

To predict theoretically such performance of the expander working with water mixed air, the mathematical model described in Section 2 is used. In the theoretical calculation of the model, as investigated in our previous study (5) on the dry air scroll expander, the expansion process is treated as a polytropic change with a polytropic exponent which satisfies the polytropic relationship between the supply and exhaust pressures and temperatures for each experimental operating case, and the radial leakage through the wrap tip clearance is nullified by taking account of the existence of the tip seal. However, the treatment of the tangential leakage through the wrap radial clearance is not easy under the water mixed condition in the supply air.
Figure 9 shows the influence of the way to treat the tangential leakage through the radial clearance on the pressure curve and the volumetric efficiency $\eta_v$ under the experimental condition of water/air mixing ratio $\alpha_v = 0.4\%$. When the water/air ratio $\alpha_l$ of the leakage flow through the radial clearance is kept constant being equal to the supply ratio (= 0.4%), the pressure curve descends a little faster than the experimental pressure curve, and its volumetric efficiency (74.9%) is not high enough as the experimental value (79.6%). On the other hand, when the leakage water/air ratio $\alpha_l$ is changed to increase with the rotational angle in a parabolic way from the supply ratio (0.4%) at the expansion starting angle to an appropriate ratio (4%) at the expansion ending angle, the pressure curve descends in the similar way to the experimental curve, and its volumetric efficiency (80.0%) is almost equal to the experimental value (79.6%). This indicates that the sealing effect of the mixed water on the leakage through the wrap radial clearance is intensified more at the larger involute angle or the outer location of the wrap.

In Fig. 5, the theoretical pressure curves calculated in the same manner were illustrated, and they were in good agreement with the experimental curves. Based on the theoretical analysis, the flow rate $Q_s$ (cal), the volumetric efficiency $\eta_v$ (cal) and the indicated torque $T_{ind}$ (cal) were calculated for each operating condition, and they were illustrated in Figs. 3, 4, 6, 7 and 8. As a whole, they were in good agreement with the experimental data, which reveals the effectiveness of the theoretical analysis.

5. CONCLUSIONS

Operating performance of the scroll air expander, which was converted from an oil-free scroll air compressor on the market, working with water mixed air was investigated theoretically and experimentally, and the results are summarized as follows.

1. The water mixing into the supply air brings the decrease of the supply air flow rate and the increase of the output shaft torque. Under the water mixed condition with water/air volume...
ratio 0.4%, the volumetric efficiency and the total expander efficiency were improved by about 10% and 20% respectively as compared with those under the no-water mixed condition.

2. The water mixing into the supply air reduces the tangential leakage through the wrap radial clearance and its sealing effect is intensified at the latter part of the expansion process, which leads to the slower descent of the pressure in the scroll pocket.

3. The performance of the expander working with the water mixed air is not sensitive to the water/air volume ratio in the supply air, though the ratio must be kept high to avoid the frost formation at the expander exit.

In this study, energy or enthalpy of the water in the high-pressure supply air is not counted in calculation of the expander efficiency because the water mixed high-pressure air can be produced easily with no extra energy, e.g. by just adding water into the compressor suction air. If the water is mixed by a pump before the expander, the pumping power should be included from the viewpoint of the total energy analysis.

REFERENCES

The effects of liquid injection on performance of a rotary compressor

K.T. Ooi
School of Mechanical and Production Engineering,
Nanyang Technological University, Singapore

ABSTRACT
In this paper, the effects of liquid injection on the performance of a rolling piston compressor using R22 will be discussed. The mathematical model describing the basic working principle of the compressor incorporating the liquid injection has been formulated and used to study the various effects of liquid injection on the performance of the compressor, with particular focus on the compression power and discharge temperature.

NOMENCLATURE

\[ A \quad \text{Area (m}^2\text{)} \]
\[ C_d \quad \text{non-isentropic and flow loss coefficient} \]
\[ \text{COP} \quad \text{coefficient of performance} \]
\[ d \quad \text{diameter (m)} \]
\[ h \quad \text{enthalpy (J/kg)} \]
\[ L \quad \text{normalised loss} \]
\[ m \quad \text{mass (kg)} \]
\[ Q \quad \text{heat transfer (J)} \]
\[ Q_{\text{ref}} \quad \text{normalised refrigerant capacity} \]
\[ P \quad \text{pressure (Pa)} \]
\[ r_m \quad \text{injected liquid over refrigerant mass ratio} \]
\[ T \quad \text{temperature (K)} \]
\[ V \quad \text{volume (m}^3\text{)} \text{ or velocity (m/s)} \]
\[ W \quad \text{normalised power} \]
\[ v_{\text{inj}}, v_s \quad \text{specific volume (m}^3\text{/kg)} \]
\[ \omega \quad \text{angular speed (º/s)} \]
\[ \theta \quad \text{angle (º)} \]
\[ \rho \quad \text{density (kg/m}^3\text{)} \]
\[ \eta \quad \text{normalised efficiency} \]

Subscripts:
\[ i \quad \text{in} \]
\[ o \quad \text{out} \]
\[ inj \quad \text{injection} \]
\[ c \quad \text{chamber} \]
\[ v_r \quad \text{at constant specific volume} \]
\[ T_c \quad \text{at constant chamber temperature} \]
\[ vol \quad \text{volumetric} \]
\[ mec \quad \text{mechanical} \]
\[ suc \quad \text{suction} \]
\[ dis \quad \text{discharge} \]
\[ mot \quad \text{motor} \]
\[ ind \quad \text{indicated} \]

1 INTRODUCTION

Literature [1-6] shows that fluid injection such oil, refrigerant and others fluids into the working space of various types of positive displacement compressor has been experimentally and theoretically studied. However, very little information on the liquid injection in the rolling
piston compressor can be found in the open literature. This paper deals with a theoretical study of liquid refrigerant injection in the working space of a rolling piston compressor. In general, liquid injection can be introduced at various stages of the compressor cycle. It can be introduced at the compressor intake line to reduce the temperature of the suction gas and thus reducing the overall temperature of the compressor system unit by lowering the discharge temperature of the refrigerant. This approach is simple and easy to incorporate which requires minimum system alteration. However, it may reduce [1] the capacity of the compressor and result in a lower refrigeration capacity, as it indirectly exaggerating the heating of the suction gas by lowering the suction gas temperature. Another method is to introduce the liquid injection at the early stage of the compression process. This method reduces the temperature without the possibility of reducing the capacity of the compressor. However, the latter method is more difficult to incorporate as it must be designed, fabricated and assembled together with the other compressor components during the production stage, it may also result in a higher power input. In this paper, the latter approach is looked into.

2 REFRIGERANT LIQUID INJECTION

Figure 1(a) shows the schematic of the refrigeration system with liquid injection. In this case the feeding liquid is channelled from the outlet of the condenser ‘I’ directly into the working chamber of the compressor ‘Z’ through an insulated capillary tube. The flow through the capillary may result in the flashing of the liquid refrigerant to occur as the pressure in the tube gets lower as it flows. Flashing causes the flow of the refrigerant to accelerate and result in a significant additional pressure reduction. This situation is undesirable as it causes the formation of the refrigerant vapour and it may further choke the flow at the tube exit [7]. Therefore the selection of the capillary tube must be such that the single phase fluid is present in the capillary, and that the downstream pressure at the capillary exit should not be less than the saturated pressure of the refrigerant at the point of injection. Fig 1 (b) shows the Pressure-enthalpy diagram for R22. Line a-b represents the duration of the liquid injection and line c-d is the compression process after the injection.

![Fig. 1 Schematic of a vapour compression cycle with liquid injection at compressor.](image-url)
3 DETERMINATION OF THE INJECTOR LOCATION

It is assumed that the injector is positioned at one of the cylinder head face. Figure 2(a) shows that the design of the location of the injector position which can be determined by the position of the roller, with reference to the pressure variation in the compression chamber shown in fig. 2(b). Fig. 2(a) shows that the two roller positions at ‘A’ and ‘B’ have been marked for the location of the injector at ‘C’. Location ‘A’ is the position where the liquid injection process commences and the location ‘B’ is where it ends.

In fig. 2(a), the circles with broken line represents the location of the roller at 90° and 190° measured from the initial roller position at ‘Y’. The injector upstream pressure, \( P_{\text{inj}} \), is determined by the pressure at location ‘A’ in fig. 2(b). Following this, the length and diameter of the capillary that links the feed liquid to the injector can be determined.

In fig. 2, the injection begins when the roller-cylinder contact reaches ‘A’ (90°) and it ends when it reaches ‘B’ (190°), giving the injection span of 100°. The injection may also end when the pressure in the chamber is higher than \( P_{\text{inj}} \). In this analysis, it is assumed that soon after the fluid is injected into the working chamber of the compressor, it mixes homogeneously and attains the equilibrium with the gas in the chamber after the mixing [1]. This assumption is reasonable, since the injected liquid is saturated, and the mass of liquid injected is assumed to disperse in atomise form. The later is dependent on the injector size and the pressure different across the injector. The amount of the liquid injected will affect the compression power due to an increase in the pressure as a result of additional compression of the injected fluid.

Figure 2 Determination of the injector position and injector upstream pressure

In fig. 2(a), the circles with broken line represents the location of the roller at 90° and 190° measured from the initial roller position at ‘Y’. The injector upstream pressure, \( P_{\text{inj}} \), is determined by the pressure at location ‘A’ in fig. 2(b). Following this, the length and diameter of the capillary that links the feed liquid to the injector can be determined.

In fig. 2, the injection begins when the roller-cylinder contact reaches ‘A’ (90°) and it ends when it reaches ‘B’ (190°), giving the injection span of 100°. The injection may also end when the pressure in the chamber is higher than \( P_{\text{inj}} \). In this analysis, it is assumed that soon after the fluid is injected into the working chamber of the compressor, it mixes homogeneously and attains the equilibrium with the gas in the chamber after the mixing [1]. This assumption is reasonable, since the injected liquid is saturated, and the mass of liquid injected is assumed to disperse in atomise form. The later is dependent on the injector size and the pressure different across the injector. The amount of the liquid injected will affect the compression power due to an increase in the pressure as a result of additional compression of the injected fluid.
4 MATHEMATICAL MODEL

In the previous study [8], the mathematical model describing the basic working principle of a rolling piston compressor taking into account of the various leakages and frictional losses has been verified by comparing its prediction with measured results. In this analysis liquid injection is taking into consideration. Thus, the change in the working fluid temperature in the working chamber with angular position of the roller is given by,

\[
\frac{dT}{d\theta} = \frac{1}{\omega m_c} \left\{ \frac{dQ}{d\theta} + \sum \frac{dm}{d\theta} (h_i - h_j) + \frac{dm_{\text{inj}}}{d\theta} (h_{\text{inj}} - h_i) - \sum \frac{dm_{\omega}}{d\theta} (h_j - h_i) + \frac{dv}{d\theta} \left[ V \left( \frac{\partial P}{\partial v_i} \right) - m_c \left( \frac{\partial h_c}{\partial v_i} \right) \right] \right\}
\]

and, the variation of the pressure is given by:

\[
\frac{dP_c}{d\theta} = \frac{1}{\omega} \left( \frac{\partial P_c}{\partial T_c} \right) \frac{dT_c}{d\theta} + \left( \frac{\partial P_c}{\partial v_s} \right) \frac{dv_s}{d\theta}
\]

If the injection is in the liquid phase, then the mass flow rate of the liquid injected can be obtained from:

\[
\frac{dm_{\text{inj}}}{d\theta} = \rho_{\text{inj}} A_{\text{inj}} V_{\text{inj}}
\]

where

\[
V_{\text{inj}} = \sqrt{\frac{2(P_{\text{inj}} - P_c)}{\rho_{\text{inj}}}}
\]

However, for gas injection, the flow of a superheated refrigerant gas is assumed as an isentropic flow with the actual non-isentropic condition and other flow losses taking care of by using the coefficient of discharge \(C_d\). Therefore, the mass flow rate is computed by:

\[
\frac{dm_{\text{inj}}}{d\theta} = \frac{C_d A_{\text{inj}}}{\omega v_{\text{inj}}} \sqrt{2(h_{\text{inj}} - h_s)}
\]

where \(h_s\) is the downstream enthalpy by assuming an isentropic flow and \(\omega\) is the rotational speed of the compressor. The speed of the gas flow must be restricted to be always less than or equal to the speed of sound.

The change of the total mass in the compression chamber can be expressed as:

\[
\frac{dm_c}{d\theta} = \left( \sum \frac{dm}{d\theta} + \frac{dm_{\text{inj}}}{d\theta} - \sum \frac{dm_{\omega}}{d\theta} \right)
\]

In the above model, real gas equations are used to determine the refrigerant properties.
5 BASIC SIMULATION RESULTS

In this section, the simulation results are presented. The compressor operates at 3000 rev/min with R22 as the working fluid. The evaporating and condensing conditions are 280.15 K, 635 kPa and 328.15K and 2170 kPa, respectively. The motor efficiency is assumed constant for all cases at 80% and the compressor has a capacity of 5.5 kW. Three cases of liquid and gas injection have been studied and these are:

**CASE 1**: Saturated liquid injection with varying injector diameters.

The saturated refrigerant liquid injection begins when the roller is at 450º and ends at 550º (which corresponds to injector position at roller-cylinder contact of 90º and 150 º). The upstream of the injector is at the saturated pressure of 17.3 bar and temperature of 318 K. An injector diameter of 0.5 mm for one case and 1.0 mm for the other are used.

Figure 3(a) shows the variation of the mass of the refrigerant in the compression chamber. In this case since the suction process was first simulated, therefore the compression process starts 360 º thereafter. The liquid injection begins when the injector hole is uncovered at 450º (=360º +90º), it can be seen that the refrigerant mass increases after this angular position. The injection stops when the pressure in the compression chamber is greater than the $P_{inj}$ or when the injector is covered by the roller, which ever comes first, this can be observed by the constant mass region in figure 3(a) just before the discharge process begins. The results in Fig. 3(d) suggest that the injection mass increases in a square manner with the injector diameter; i.e. the mass injected at the injector diameter of 1.0 mm is the square of that when the injector diameter is 0.5 mm, as the flow area is square of the diameter of the injector. The results also show that the pressure and indicated power (Fig. 3(c)) is almost unaffected by the liquid injected but the temperature (Fig. 3(b)) is significantly affected. It seems to suggest that cooling effects provided by the liquid injected even-out the additional compression that is needed with the presence of the gas vapour due to the liquid injected. The result in figure 3(b) demonstrates that for the injection conditions given, the temperature of the refrigerant in the compression chamber is greatly reduced when the injector size is 1.0 mm in diameter. The variations in the injection velocity, injected mass and the discharge valve displacement are also shown. Figure 3(f) shows that discharge temperature drops significantly as the diameter of the injector increases. The variation of the discharge temperature with mass ratio, defined as the mass of the injected liquid over the mass of the refrigerant flows through the evaporator into the compressor.

Table 1(a) shows that there is no significant change in the other compressor performance parameters. Though, the discharge losses affect noticeably but the overall effects on COP is not significant. This is because the amount of liquid injection is relatively small, less than 0.6%.

**CASE 2**: Saturated liquid injection with a varying injection position.

Saturated refrigerant liquid is injected into the compression chamber of the compressor with injector diameter of 0.75 mm. The injection upstream conditions remain the same, but with three different injection positions at 360º, 400º, 450º.

Figure 4(a) shows that as the injection begins earlier, the amount of the mass injected is more, as would be expected. Fig. 4(b) shows that the earlier the injection begins, the lower the discharge temperature in the compression chamber would be, as shown in fig. 4(f). Fig. 4(c) shows that the injection does not significantly affect the pressure in the compression chamber.
as the mass of the liquid injection is small. The variations in the injection velocity, injected mass and the discharge valve displacement are also shown. Table 1(b) shows that the power input of the compressor is not significantly affected, only a very marginal performance variation is observed. In practice, it would expect that the overall compressor performance will improve.

**CASE 3: Superheated gas injection with varying injector size.**
For the purpose of comparison, in this case, a superheated refrigerant gas is injected into the compression chamber of the compressor at the injector position at 450º. The injected gas is superheated at the pressure of 15.3 bar and at temperature of 318.15K. Injectors with diameter of 0.5, 1.0, 1.5, 3.0 and 5.0 mm are attempted.

The results of the study are shown in Figure 5. Since this is the case of gas injection, the cooling effect is expected to be less, as it is only based on the sensible heat of the gas which is many times lower than the latent heat as compared to the case of the liquid injection. Moreover as the gas does not provide effective cooling, its presence in the working chamber affects the compression work significantly. Therefore it is expected that in gas injection, the performance of the compressor deteriorate significantly with increase mass injection. Fig. 5 shows that the temperature of the gas in the compression chamber reduces while the pressure increases. For the same discharge temperature reduction, a lot more gas than liquid has to be injected. Even with about more than 60% of the mass (relative to the total mass in the chamber without injection) injected, the discharge temperature does not seem to reduce that significantly.

Table 3 shows that gas injection affects the performance of the compressor significantly. The COP reduces drastically as the amount of the gas injected increases.

6 CONCLUSIONS

The results show that the temperature of the refrigerant in the compression chamber is significantly affected by the liquid injection and less affected by the gas injection. This can be seen in figure 6 where the discharge temperatures with mass ratio of the three cases are shown. However, the pressure in the compression chamber is significantly affected by the gas injection and less significantly affected by the liquid injection.

The result also shows that the liquid injection of up to 27% of the chamber mass does not seem to affect the chamber pressure that noticeably. However, for gas injection of 1%, a noticeable effect on pressure variation is observed.

Liquid injection tends to reduce motor power input and gas injection increases it. Apart from lowering the discharge temperature, liquid injection does not seem to negatively affect the power input of the compressor noticeably, but gas injection reduces the performance of the compressor significantly by increasing the compression power.

The results also shows that earlier liquid injection reduces discharge temperature more significantly than if it is injection later, as more liquid can be injected.
7 REFERENCES


(3) Stosic, M., Kovacevic, A., Hanjalić, K., Milutinovio, L., Mathematical modelling of the oil influence upon the working cycle of screw compressors, International Compressor Engineering Conference at Purdue, 1988, 354-361

(4) B. Sangfors; Svenska Rotor Maskiner AB, Sweden; Computer Simulation of Effects From Injection of Different Liquids in Screw Compressors, 1998, 595-600


(8) Ooi, K.T., Design optimization of a rolling piston compressor for refrigerators, Applied Thermal Engineering, v 25, n 5-6, April, 2005, 813-829
Figure 3: Saturated liquid injection at 450° with varying injector size.
Figure 4: Saturated liquid injection with varying injection position.
( Inj. Position: 360°, 400°, 450°, $D_{inj}$=0.75 mm, $P_{inj}$=17.292 bar and $T_{inj}$=45°C)
Figure 5: Superheated gas injection at 450° with varying injector size.
(\(D_{\text{inj}}=0.5, 1.0, 1.5, 3.0, 5.0\) mm, \(P_{\text{inj}}=15.292\) bar and \(T_{\text{inj}}=45^\circ\)C)
Fig. 6 Variation of discharge temperature with mass ratio.

Table 1: Normalised simulation results for cases 1, 2, and 3.

(a) Case 1

<table>
<thead>
<tr>
<th>$d_{inj}$ (mm)</th>
<th>$r_m$</th>
<th>$\eta_{vol}$</th>
<th>$\eta_{mech}$</th>
<th>$W_{ind}$</th>
<th>$L_{dis}$</th>
<th>$L_{suc}$</th>
<th>$W_{mot}$</th>
<th>COP</th>
<th>$Q_{ref}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>0.072</td>
<td>1</td>
<td>1.000</td>
<td>1.000</td>
<td>1.054</td>
<td>1</td>
<td>1.000</td>
<td>0.9997</td>
<td>1</td>
</tr>
<tr>
<td>1.0</td>
<td>0.268</td>
<td>1</td>
<td>0.999</td>
<td>0.994</td>
<td>1.209</td>
<td>1</td>
<td>0.995</td>
<td>1.006</td>
<td>1</td>
</tr>
</tbody>
</table>

(b) Case 2

<table>
<thead>
<tr>
<th>$\theta_{inj}$ (º)</th>
<th>$r_m$</th>
<th>$\eta_{vol}$</th>
<th>$\eta_{mech}$</th>
<th>$W_{ind}$</th>
<th>$L_{dis}$</th>
<th>$L_{suc}$</th>
<th>$W_{mot}$</th>
<th>COP</th>
<th>$Q_{ref}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>350</td>
<td>0.150</td>
<td>0.999</td>
<td>1.000</td>
<td>0.999</td>
<td>1.124</td>
<td>1</td>
<td>0.998</td>
<td>1.009</td>
<td>1</td>
</tr>
<tr>
<td>450</td>
<td>0.120</td>
<td>1.000</td>
<td>1.000</td>
<td>1.001</td>
<td>1.089</td>
<td>1</td>
<td>1.001</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>550</td>
<td>0.066</td>
<td>1.000</td>
<td>1.000</td>
<td>1.001</td>
<td>1.054</td>
<td>1</td>
<td>1.001</td>
<td>1.000</td>
<td>1.000</td>
</tr>
</tbody>
</table>

$\theta_{inj}$ angle of cylinder-roller contact at which injection begins

(c) Case 3

<table>
<thead>
<tr>
<th>$d_{inj}$ (mm)</th>
<th>$r_m$</th>
<th>$\eta_{vol}$</th>
<th>$\eta_{mech}$</th>
<th>$W_{ind}$</th>
<th>$L_{dis}$</th>
<th>$L_{suc}$</th>
<th>$W_{mot}$</th>
<th>COP</th>
<th>$Q_{ref}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>0.013</td>
<td>1.000</td>
<td>1.000</td>
<td>1.008</td>
<td>1.021</td>
<td>1</td>
<td>1.007</td>
<td>0.993</td>
<td>1.000</td>
</tr>
<tr>
<td>1.0</td>
<td>0.052</td>
<td>1.000</td>
<td>1.002</td>
<td>1.028</td>
<td>1.073</td>
<td>1</td>
<td>1.026</td>
<td>0.975</td>
<td>1.000</td>
</tr>
<tr>
<td>1.5</td>
<td>0.110</td>
<td>1.000</td>
<td>1.004</td>
<td>1.059</td>
<td>1.155</td>
<td>1</td>
<td>1.054</td>
<td>0.949</td>
<td>1.000</td>
</tr>
<tr>
<td>3.0</td>
<td>0.338</td>
<td>1.000</td>
<td>1.011</td>
<td>1.174</td>
<td>1.492</td>
<td>1</td>
<td>1.161</td>
<td>0.861</td>
<td>1.000</td>
</tr>
<tr>
<td>5.0</td>
<td>0.621</td>
<td>1.000</td>
<td>1.016</td>
<td>1.301</td>
<td>1.927</td>
<td>1</td>
<td>1.280</td>
<td>0.781</td>
<td>1.000</td>
</tr>
</tbody>
</table>
RECIPROCATING COMPRESSORS
Performance evaluation of reciprocating compressor using an engineering toolbox

P. Grolier
Tecumseh Europe, Engineering Department, France

ABSTRACT

The performance evaluation of reciprocating compressor performance is estimated by the use of an engineering toolbox. A modular approach is described. New improvements of the model are listed. Areas for development are investigated. Factors affecting compressor performance are discussed. Possible solutions are identified: on the one hand, clearance volume, suction gas superheat reduction and valve timing improvement to increase refrigerating capacity; on the other hand, diminution of compression losses, friction losses and pressure drop, to decrease the power input. A current and an improved compressor are simulated and results presented through figures and tables. Other uses of the toolbox are discussed.

1 INTRODUCTION

Compressor manufacturers must develop new products to provide suitable answers to customer needs, within the shortest lead time. Efficiency, sound, reliability and pricing are the usual design criteria. Compressor designers look for novel solutions in the quest to match suitable features on existing or future products. A key issue is the establishment of objectives among design criteria. A strategy is necessary to investigate areas for development. When product objectives, design process and development strategy are defined then an engineering toolbox can be of assistance.

Readers will find helpful information in relation with design process and development strategy in the two mentioned articles. Reference (1) presents main stages of typical engineering process including idea generation, concept analysis, product design and feasibility analysis. Such methods can help to build a step by step process in compressor design. Reference (2) describes areas for development through a list of improvement points related to technical subjects. A table presents factors affecting compressor performance versus causes and components. Such an approach can be an answer to the expected strategy for compressor development.
A computer-aided design tool can guide engineering activities, if it is able to bring automation and a better understanding of main phenomena taking place in a product. It implies having a structure which supports the engineer through his design process, and organizes development strategy. It doesn’t mean that solutions will appear like miracles from the computer, because a computer software generally requires basic knowledge to be used. Such a computer-aided engineering tool needs to demystify the product, which can sometimes remain a black box system with inputs and outputs. In order to insure its use by engineers in their daily jobs, a computer software must have a user-friendly interface. It must also be able to bring quick answers to the engineer’s issues and concerns, with sufficient accuracy, to help decision making.

The aim of the engineering toolbox is to provide a software to help compressor designers. A computer program was developed to solve the mathematical equations and simulate compressor performances (refrigerating capacity, power input and reliability). The performance assessment is now possible on all existing products of Tecumseh Europe’s wide range of reciprocating compressors. Simulations toolbox expedite the initial stages of the design process, but also provide helpful information during the manufacturing stage. For example, deviations to specifications can be analysed to make decisions. Due to current environmental concerns, significant gains on compressor performance are more and more demanded to get energy savings on the final application. The engineering toolbox is now mainly used in the quest for efficiency improvements on existing compressors.

The purpose of this paper is to present the use of an engineering simulation toolbox for reciprocating compressor performance evaluation. It includes a description of the steps followed to improve a current compressor. The investigation focuses on solutions to increase cooling capacity and to decrease the power consumption. Solutions are simulated and compared to experimental results.

2 DESCRIPTION OF THE MODEL

2.1 Outline
The outline of the toolbox has already been presented in reference (3) with sufficient details to get a general overview of the model. It includes the main modules used to describe the phenomena and to help the designer estimate the compressor performance. Geometrical data are necessary to get the boundary conditions and the main dimensions of the compressor required for the description of the mechanical kit and the valve features. Motor curves are included to yield the electrical efficiency. The refrigerant property formulations are included to run the thermodynamic cycle. The dynamic viscosity of oil is supplied by the database to get the hydrodynamic performance of bearings.

2.2 Modular approach
The advantage of a modular approach in compressor simulation is discussed in reference (4). The actual toolbox includes seven modules which provide input data and store output results from simulation.

2.2.1 MecaDim
Main dimensions of the pump can be visualised to run the mechanical analysis.
2.2.2 DeadVol
The geometric clearance volume can be estimated with the required features: cylinder, top of piston, gasket shape and selection, valve plate ports and suction valve leaf.

2.2.3 DisPort
Thermodynamic parameters can be calculated as a function of working conditions and inside compressor temperature assessment: compression ratio, polytropic exponent, density, speed, Mach and Reynolds numbers of suction and discharge gas.

2.2.4 Motor
The suitable speed and efficiency of motor versus torque of compressor can be obtained from the curves in the database.

2.2.5 Bearing
The module determines the oil film thickness and friction losses of main, outboard and throw crankshaft bearings; wrist pin-connecting rod, piston-cylinder and thrust face bearings.

2.2.6 DefValv
A dynamic valve analysis is used to predict the motion of the suction and discharge self-acting valves. It considers the valve as a mass-spring-damper system with one-degree of freedom. Empirical coefficients are included in the model, and can be modified by users. The pressure state in the two volumes of the cylinder head on suction and discharge sides can be monitored to take into account suction under pressure and discharge over pressure, assuming an arbitrary pressure fluctuation as a percentage of line pressures. The heat transfer coefficient can be multiplied by a factor within a certain range. The power losses due to the pressure drop through the valve can be estimated for a full valve motion cycle. The mass flow rate to use for the cooling capacity of the compressor is calculated as the mean value between the mass flow at the suction side and the discharge side, taking into account the mass flow leakage due to the piston blow by. The cooling capacity is a product of the mass flow rate and the enthalpy increase during evaporation. The module enables the plotting of the pressure in the cylinder versus the crank angle, the pressure-volume diagram, the suction and discharge valve motions versus the crank angle, the valve-timing diagram, the temperature, density, mass of gas, and the heat flow inside the cylinder bore versus time.

2.2.7 PowerCal
The power input is calculated as a result of all the estimated powers determined in previous modules: useful power, mechanical friction losses, valve power losses due to pressure drop, and motor losses. The coefficient of performance can be determined based on the knowledge of the compression efficiency, the mechanical efficiency and the motor efficiency.

Since the last presentation, new improvements have been implemented in the toolbox.

2.3 Model improvements
Among the investigated improvements to refine accuracy, it was found that the heat transfer model was of major importance. The previous model wasn’t suitable for the analysis of the energy balance during compression cycle: a new thermal model has been used. The two empirical coefficients related to force gas and mass flow have been modified too, with a more simple approach than previous ones. The main changes applied are described below.
2.3.1 Heat transfer model
The heat flow $\phi$ through the cylinder wall can be calculated as soon as the heat exchange coefficient $h_{\text{heat}}$, the cylinder area $A$ and the temperature difference $\Delta T$ are determined.

$$\phi = h_{\text{heat}} \ A \ \Delta T$$ (1)

The heat exchange coefficient is calculated using the Nusselt number $Nu$, with the help of Reynolds $Re$, as set out in the following formula according to reference (5).

$$Nu = 0.7 \ Re^{0.7}$$ (2)

A multiplying factor of the heat exchange coefficient can be used, within a range comprised between 1 to 3, as advice in reference (6).

2.3.2 Valve system model
The equation of the valve motion is based on the fundamental equation of the dynamics, with $M$ describing the valve mass, $x$ designing the lift, $\frac{d^2 x}{dt^2}$ the acceleration of the valve, total force acting on the valve is composed of gas force $F_g$, spring force $F_s$, and damping friction force $F_f$.

$$F_g - F_s - F_f = M \frac{d^2 x}{dt^2}$$ (3)

The gas force relates to the pressure difference, between upstream and downstream, across the flow path. The gas force depends on the valve area $A_v$ exposed to pressure, the pressure difference $(P_u - P_d)$ and an empirical coefficient $C_g$, the gas force coefficient.

$$F_g = C_g A_v (P_u - P_d)$$ (4)

The empirical gas force coefficient $C_g$ is now constant and equal to 0.6.

2.3.3 Flow model
The mass flow through the valve $\dot{m}_{\text{valve}}$ can be evaluated by the following formula.

$$\dot{m}_{\text{valve}} = C_d \ A_v \ \sqrt{2 \ \rho \ (P_u - P_d)}$$ (5)

The empirical flow coefficient $C_d$ is now constant and comprised between 0.4 and 0.6.
3 DEVELOPMENT STRATEGY

3.1 Background
The quest for energy saving on current compressors implies having a good knowledge of current performance to identify potential efficiency improvement. A method must be used to succeed in such a job: reference (1) and (2) provide useful information. Computer simulation can help to predict potential energy saving, which can be difficult to measure on calorimeter equipment due to the usual standard deviation on compressor from production. The efficiency improvement solutions very often concern the addition of solution of few percent very difficult to measure separately. The computer software helps us to understand what kind of improvement can be expected; however measurement is the only definitive answer.

3.2 Refrigerating capacity
The mass flow rate is a product of the gas density $\rho$, the volumetric efficiency $\eta_v$, and the swept volume of the compressor per revolution $V_s$. The refrigerating capacity $Q$ is calculated as a product of the mass flow rate and the enthalpy increase $\Delta H$ during evaporation.

$$Q = \rho \eta_v V_s \Delta H$$

(6)

The volumetric efficiency is defined as the ratio of the effective volume of gas entered during the suction process to the swept volume during a revolution.

The determination of suction gas density is based on the ideal gas relationship, where $M$ is the molar mass, $R$ the ideal gas constant, $P$ and $T$ absolute pressure and temperature respectively.

$$\rho = \frac{M P}{R T}$$

(7)

These well known equations clearly show that an increase of refrigerating capacity, for given displacement compressor, refrigerant gas and working conditions, depends on the volumetric efficiency and the gas density. The goal is to find solutions to increase both of them, by preventing the mass flow losses due to re-expansion loss, suction gas super heating, blowby leakage loss, and valve dynamics losses due to backflow.

3.3 Power input
The power input $W$ can be easily estimated with the simple formula below, with $\gamma$ the polytropic exponent, $P$ the absolute pressure, $V$ the volume, $\tau$ the compression ratio. The following efficiencies are also requested: mechanical efficiency $\eta_m$, compression efficiency $\eta_c$, and electrical efficiency $\eta_e$.

$$W = \frac{\gamma}{\gamma - 1} \frac{P V}{\tau^{\gamma - 1}} n_c \eta_v \eta_c \eta_m \eta_e$$

(8)

The previous equation teaches us that the mechanical, compression and electrical efficiency need to be as high as possible to minimize the power input.
3.4 Coefficient of performance

The coefficient of performance \( \text{COP} \) is defined as the ratio between the refrigerating capacity \( Q \) and the power input \( W \). With the help of formulas (6) and (8), \( \text{COP} \) can be expressed as:

\[
\text{COP} = \frac{Q}{W} = \frac{\rho V_s \Delta H \eta_c \eta_m \eta_e}{\gamma - 1} P V \left( \frac{\gamma - 1}{\tau} - 1 \right)
\]

The mechanical efficiency \( \eta_m \) can be estimated after all the mechanical losses are determined, and compared to the useful power. The compression efficiency \( \eta_c \) is defined as a ratio between the indicated and theoretical power. The electrical efficiency \( \eta_e \) reflects the power losses versus the useful motor power.

4 PERFORMANCE EVALUATION

4.1 Description

The performances of a current compressor AEZ and its improved version TTA have been evaluated with the toolbox software and compared with the values measured at the calorimeter. The 4425Y model is a compressor for commercial application on R134a for high back pressure. Most of its mechanical features are presented in MecaDim module below:
The new TTA compressor includes many new features to bring efficiency improvements. A new discharge reed valve system replaces the actual horse shoe shape. Suitable mass, stiffness of valve and stop height of retainer have been determined to increase the mass flow during intake process by preventing the back flow losses and the valve flutter. The dead volume has been reduced thanks to a thinner valve plate, optimised port size, and a new valve plate gasket shape. All these new features help to decrease the valve losses and increase the refrigerating capacity, specially at low evaporating temperature, where the pressure ratio is high. The oil viscosity has been decreased to minimise the power losses at bearings.

4.2 Thermodynamic audit
The compressor is equipped with a set of pressure transducers and thermocouples in both sides of the cylinder head, respectively at suction and at discharge, as close as possible to the cylinder bore. Other thermocouples are placed at different positions to measure other important temperatures. A thermodynamic audit is ran at calorimeter to get pressure and temperature values.

4.2.1 Temperature
A thermal map is plotted with the main temperature inside compressor, at nominal working conditions: compressor components temperature (housing, oil, motor), gas temperatures throughout the circuit, in the cylinder head and in the discharge tube.

![Figure 1 – Thermal map](image)

The thermal model included in the toolbox software shows the closeness of the calculated and measured temperatures, with a suitable multiplying factor of heat transfer coefficient within the advice range. Suction and discharge gas temperatures are essential data for accurate simulation results.

4.2.2 Pressure
The pressure value in the cylinder head can be measured as under pressure on the low side and over pressure on the high side. The values are monitored as a percentage of the pressure line. The percentage range measured is included within +/- 5% of the pressure line.
4.3 Performance simulation and measurement
The expected performances of AEZ4425Y and TTA4425Y have been calculated with the toolbox software and compared with the values measured at the calorimeter, in accordance with the European standard EN 13771-1. The table below shows results on refrigerating capacity, power input and coefficient of performance, at three different working conditions.

Table 1 – Performance

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>AEZ</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>T amb. (°C)</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>T inlet (°C)</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>T liq. (°C)</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>T cond (°C)</td>
<td>55</td>
<td>55</td>
<td>55</td>
<td>55</td>
<td>55</td>
<td>55</td>
</tr>
<tr>
<td>T evap (°C)</td>
<td>-15</td>
<td>5</td>
<td>15</td>
<td>15</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Capacity (W)</td>
<td>219.8</td>
<td>230.4</td>
<td>655.2</td>
<td>660.6</td>
<td>994.8</td>
<td>975.9</td>
</tr>
<tr>
<td>Power (W)</td>
<td>186.7</td>
<td>184</td>
<td>298.3</td>
<td>305.2</td>
<td>348</td>
<td>359.8</td>
</tr>
<tr>
<td>COP (W/W)</td>
<td>1.18</td>
<td>1.25</td>
<td>2.20</td>
<td>2.16</td>
<td>2.86</td>
<td>2.71</td>
</tr>
<tr>
<td><strong>TTA</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>T amb. (°C)</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>T inlet (°C)</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>T liq. (°C)</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>T cond (°C)</td>
<td>55</td>
<td>55</td>
<td>55</td>
<td>55</td>
<td>55</td>
<td>55</td>
</tr>
<tr>
<td>T evap (°C)</td>
<td>-15</td>
<td>5</td>
<td>15</td>
<td>15</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Capacity (W)</td>
<td>265.9</td>
<td>259.7</td>
<td>689.1</td>
<td>674</td>
<td>1028.5</td>
<td>989.9</td>
</tr>
<tr>
<td>Power (W)</td>
<td>213.3</td>
<td>194.9</td>
<td>314.4</td>
<td>301.1</td>
<td>360.7</td>
<td>352</td>
</tr>
<tr>
<td>COP (W/W)</td>
<td>1.25</td>
<td>1.33</td>
<td>2.19</td>
<td>2.24</td>
<td>2.85</td>
<td>2.81</td>
</tr>
</tbody>
</table>

The results presented above show a good agreement between experimental results and calculation. Deviations for the refrigerating capacity and for the compressor efficiency are acceptable for the preliminary calculation. The accuracy of the program is quite sufficient to help in the analysis of design changes. Trends for each compressor are right.

4.4 Interpretation
As seen above, the toolbox software can be a very useful tool for the engineer in predicting the compressor performance. It also leads to further analysis and a better understanding.

4.4.1 Share of compressor consumption
A detailed knowledge of the compressor consumption is useful to provide a view of the power input through each source of losses, thanks to prediction by simulation.

Figure 2 – Sources of compressor consumption
Such percentages should help the engineer in his development strategy to make suitable choices among possible solutions, keeping in mind useful, electrical, mechanical and valves power accounting.

4.4.2 Efficiencies
The coefficient of performance provides a global overview of the compressor efficiency. The description of each efficiency details below the improvements by each source, with the percentage difference between the two designs TTA and AEZ:

Table 2 – Efficiencies summary at Tevap=-15°C

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Efficiencies</th>
<th>AEZ</th>
<th>TTA</th>
<th>% TTA/AEZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Capacity</td>
<td>volumetric</td>
<td>62.7</td>
<td>70.9</td>
<td>13.1</td>
</tr>
<tr>
<td></td>
<td>dead volume</td>
<td>79.6</td>
<td>86</td>
<td>8.0</td>
</tr>
<tr>
<td></td>
<td>valve</td>
<td>78.8</td>
<td>82.4</td>
<td>4.7</td>
</tr>
<tr>
<td>Power</td>
<td>input</td>
<td>51.1</td>
<td>54.5</td>
<td>6.6</td>
</tr>
<tr>
<td></td>
<td>compression</td>
<td>92</td>
<td>93</td>
<td>1.1</td>
</tr>
<tr>
<td></td>
<td>mechanical</td>
<td>84.3</td>
<td>87.4</td>
<td>3.7</td>
</tr>
<tr>
<td></td>
<td>electrical</td>
<td>65.9</td>
<td>67</td>
<td>1.7</td>
</tr>
</tbody>
</table>

The volumetric efficiency can be analysed as the product of two efficiencies: the one related to the dead volume alone and the other one related to the valve design.

4.5 Discussion
The engineering toolbox software helps the compressor designer to estimate performance. It also includes sound and reliability aspects with convenient parameters to make decision, because every new design needs also to take into account noise and endurance issues.

For valves, the speed of gas and the Mach number are determined to prevent any jet noise trouble. The impact speed and the extrusion stress of the valve on the valve plate’s seat are estimated and compared to acceptable safety factors. For the bending stress determination of valves, it is necessary to use a finite element analysis computer software such as the one mentioned in reference (7). For bearings, the oil film thickness and the pressure distribution are already included in the engineering toolbox, with the help of usual rule of thumb. The goal is to verify that no wear can take place at certain bearings, such as the wrist pin bearing.

Others technical solutions can be simulated in the engineering toolbox, too. The effect of a plastic suction muffler can be estimated by putting the suitable pressure drop and temperature decrease as input parameters. The method for designing a suction muffler has been very well described in reference (8). The toolbox includes a module, named Pdrop still under development, which will provide information on pressure drop at suction and discharge lines. Electrical efficiency improvements can be simulated with the direct link to motor database.

Further development of the engineering toolbox software will consist in providing additional information on thermodynamic parameters for a better use of European standard EN 12900 to get a view of the global energy balance with the calculation of the isentropic efficiency. Thanks to this approach, a better understanding will be available for a knowledge growth of compressors designers in their daily jobs.
5 CONCLUSIONS

The performance evaluation of reciprocating refrigeration hermetic compressors has been presented using an engineering toolbox. The software toolbox includes the main modules necessary to see an overview of the phenomena taking place in compressors. The toolbox is able to simulate the refrigerating capacity and the power input, to determine the coefficient of performance. New improvements have been included to refine accuracy. The comparison between analytical and experimental results on a current and an improved compressor has shown a very good degree of agreement. The toolbox confirms improvement’s trends measured at the calorimeter. A development strategy has been discussed. Areas for development must guide engineering activities. The toolbox software can be a very useful tool for better understanding of compressor design. It provides consistent results for advanced investigations. Thanks to its users-friendly interface, compressors designers can use the engineering toolbox in their daily jobs: it can be considered as a tool for compressor lifecycle management.

ACKNOWLEDGMENTS

The author acknowledges Julien Chaou and Arnaud Louel in Master Degree in Heat Engineering of Pau University, who brought improvements in the valve module of the toolbox software, using Visual Basic version 6.0, as programming language.

REFERENCES

1. ABSTRACT

Lubrication of piston-connecting rod assembly in reciprocating machines is grounded in practical experience. What is presented here partially reflects an engineer’s life experience. Employing inertia simplified the lubrication and made the design work more effective.

![Fig. 1. Oil in reciprocating mechanism.](image)

The metal parts of the assembly are subject to accelerations of the crank mechanism, while oil is sloshing under inertia force in opposite direction. Shaking a bottle of water and one of oil illustrates the role of viscosity. With the level of acceleration in high-speed machines, the role of viscosity is not considered in this paper. During the compression stroke, oil inside the rod is accelerated away from the critical load area, making the wrist pin junction vulnerable.

2. INTRODUCTION

The reciprocating engine and the compressor have been serving us for about a hundred years now, and will continue this performance for a while longer.
Understanding the oil motion within the components of a reciprocating machine can improve this service. An effective lubricating system is essential to longevity of the reciprocating mechanism. Metal parts moving and rubbing against each other generate friction and heat. The lubricant reduces the friction and carries away the heat. An efficient design can accomplish both duties at a reasonable cost, and extend the life of the machine.

Abrasive debris, if allowed entry into the narrow gap of the hydrodynamic bearing will in time erode both the journal and the bearing to the point when the load can no longer be supported. Hard metal particles of size larger than thickness of the oil layer get embedded in the softer bearing material to gouge the shaft journal. In the combustion engine, a pressurized oil filter is used to capture the grit along with the contaminants produced during operation.

Filtering is not very practical in a hermetic compressor: once assembled, the compressor is left to fend for itself. When used, the filter becomes clogged. If no by-pass is provided, oil flow ceases and the bearings burn out. Inertia force can spin metal grit out before oil enters the bearings. Within the radial acceleration field the heaviest particles are driven to the outermost radius wherefrom they can readily be expelled. Some very fine dust may enter the bearings and lap the surfaces. This run-in process actually improves the bearing function.

At the entrance to the oil pump the ambient pressure is somewhat reduced. With added heat, vapor flashes from the refrigerant-oil mixture. Being the lightest of the components, the released gas stays closer to the center of rotation and can be vented outside the pumping chamber. Since the stream of oil within the crankshaft gains heat along its path, venting may be needed at several locations. An efficient design can fully accommodate these contradictory aspects. The designs falling short in this respect result in excessive warranty costs, as the bearings wear prematurely. See References (1), (2). These patents, dated 1978, are expired and are useful to expand the scope of this paper.

A majority of reciprocating machines produced worldwide has some improvement potential. When implemented at the design stage, inertia based solutions are inexpensive. Inertia force is free and it works 100% of the time. The magnitude of radial acceleration in a high-speed compressor is best seen in terms of Earth acceleration $g = 9.81 [\text{m/sec}^2]\)  

Angular velocity $\omega$:

$$\omega = \frac{\pi \cdot n}{30} [\text{rad/sec}]$$  \hspace{1cm} (1)

Maximum radial acceleration ($a/g$):

$$\frac{a}{g} = \frac{\omega^2 r}{g} \left[\text{g's}\right]$$  \hspace{1cm} (2)

Taking 60 Hz operation, $n = 3500$ RPM sample and $r = 12.5$ mm crank throw

We get for 60 Hz:

$$\frac{a}{g} = \left(\frac{\pi \cdot n}{30}\right)^2 \cdot \frac{.0125}{9.81} = 17 \left[\text{g's}\right].$$  \hspace{1cm} (3)

At this level acceleration becomes a formidable tool begging to be used. Scrutiny of any oiling system along these guidelines will reveal areas where improvements can be made.
Compressor bearings are fed through axial passages drilled in the crankshaft to convey the oil along the shaft. Radial passages additionally serve as pumping devices.

The wrist pin load area is prone to failure with marginal oil supply. This is particularly evident in compressors working under a high-pressure differential. The force on the piston deck reaches the point where the wrist pin has no axial movement left. The oil is squeezed out, while the rocking motion of the rod continues to heat up the parts under much increased friction coefficient. With no oil to carry away the heat of friction, the temperature rapidly builds up to the point of melting the rod metal. Bearing failure may follow in short order.

The oil passage drilled through the stem of the connecting rod communicates with a single hole drilled in the crankshaft throw (eccentric) only once per revolution at BDC. All oil needed at the piston walls and the wrist pin-connecting rod is delivered through this passage. To extend this interception time and deliver a full supplement of oil to the wrist pin area, the oil supply outlet in the crank can be scalloped or amply chamfered.

To have oil available for the wrist pin/connecting rod interface during gas compression, oil storage should best be located above this point. Hollow wrist pin core is a logical storage place. During compression stroke inertia will drive oil downward. See Fig. 2 and 4.

3. SOLUTIONS

Drilling the connecting rod from the top, as shown in Fig. 2 enhances the oil flow. During the down stroke the spent oil is jetted against the piston cavity to cool it down. The piston deck is heated by gas compression. Refrigerant vapor is already highly superheated on its way to cylinder, as it absorbs heat from the compressor body and cools the motor, picking up to ~50°C. At a high-pressure ratio, temperature in the cylinder runs above 150°C. This heat flux is dissipated into the metal of surrounding piston skirt acting as a heat sink.

Typically the piston cavity is made rectangular with a flat deck on top. The asymmetric geometry of this area causes uneven, heat gradient-oriented thermal expansion leading to piston crown edge scoffing along the cavity length: (Fig. 3, right side). Adding a larger fillet
radius improves the heat transfer to the thin portion of the piston skirt and limits the excessive expansion, scoffing and friction: (Fig. 3, left side).

![Figure 3. Piston with reduced edge scoffing.](image)

A large diameter wrist pin reduces the unit load of the compression cycle.

Oil storage inside the wrist pin should be as ample as feasible. This is limited by the wall thickness of the wrist pin. For single piston machines, a portion of the oil mass should be included in balancing calculations. The exact amount depends on the supply and the outflow.

Design of the wrist pin should be based on deflection rather than strength. A thin wall wrist pin under load will deflect within the limits of steel elongation. Die cast aluminum rod elastic elongation hovers around the 4.5% mark, while SAE 1016-1018 steel shows ~25% limit. Under slugging pressure such wrist pin will crack the rod. One way to get around this problem is to use a cold-formed wrist pin with a web, as shown below.

![Figure 4. Thin wall wrist pin stiffened by central web addition.](image)

The bottom cross-hole is for fastening the wrist pin to the piston. “Spirol” pin is customarily used for this purpose: since it is subject to fatigue breaks, it requires attention during design.

The central hole in the web helps the flow of oil, and can be cold formed in the same step the baffle is made. The right-hand hole at the baffle communicates with oil passage in the rod. The open ends of the pin assure a supply of oil to lubricate the piston walls during the up and down stroke, utilizing the inertia forces generated by reciprocal acceleration.
Arrows in Fig. 4 show inertia forced flow of oil during the up and down stroke. In many designs the wrist pin partially emerges from the cylinder bore implying oil could escape. During the down stroke inertia keeps the oil away from the potential leak area.

Commonly the wrist pins of small compressors are made with the solid bottom portion to prevent the oil escape. In such cases the top of the pin spills the oil out into a circumferential groove above the pin. As the oil in the groove lubricates the piston wall and flows downward under gravity, the blow-by gas foams it, generating a very effective sound attenuation cushion compared to a piston with plain walls. In one study conducted on a single ~28 mm diameter piston running at 3500 RPM showed up to 8.0 LWA dB sound reduction after adding the oil groove. Additional ~ 2.2 dB was found after increasing the oil supply to the wrist pin.

Apparently the free-floating piston was oscillating at the central frequency of ~3000 Hz and responded to the viscous damping provided by the oil foam cushion in the groove. Approximate frequencies of most effective damping were related by a factor of ~3, as below:

<table>
<thead>
<tr>
<th>Frequency</th>
<th>Reduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>110 Hz</td>
<td>3 dB</td>
</tr>
<tr>
<td>330 Hz</td>
<td>9 dB</td>
</tr>
<tr>
<td>1000 Hz</td>
<td>7 dB</td>
</tr>
<tr>
<td>3000 Hz</td>
<td>12 dB</td>
</tr>
<tr>
<td>LWA</td>
<td>8 dB</td>
</tr>
</tbody>
</table>

### Table 1. Sound reduction for added oil groove to a piston.

3.1. Thermal expansion and contraction
During operation, the temperature of the assembly increases to approximately 90° C (190° F). The coefficient of thermal expansion for aluminum is about twice that of steel, so the running fit will increase from the assembly fit of 7 to 10 microns, (.0003” to .0004”) by about 14 microns, (.0006”) for a typical 19 mm diameter pin. This increase in the running clearance must be accounted for in any bearing calculations, normally done with exclusion of thermal expansion. Since the compressor must operate within a wide thermal range, all options must be evaluated. The same is true for the large end of the aluminum connecting rod, where the full running temperature can increase the clearance ~3-fold, depending on crank diameter.

The opposite end of the thermal expansion is also a problem: a compressor with aluminum connecting rods, exposed to freezing weather in the street soft drink vending machines can seize on the start up and trip the motor protection under excessive LRA. (Locked Rotor Amps). This is fairly common mode of failure, hard to recognize in the tear down analysis. The dimensional check performed at that time reveals nothing wrong since post-mortem analysis is performed at room temperature.

PTCR cartridge heaters are sometimes used to alleviate this situation with modest improvement in cold start mortality rate. The typical 30 W crankcase heater offers about 10° C raise in the oil temperature for medium sized hermetic compressor, easily overwhelmed by the influx of liquid refrigerant migrating to the coldest spot in the system. Since the heater is an added cost, it is usually left out, the cost of compressor replacement being born by the manufacturer unaware of this pitfall.
4. PUMPING OIL

With the lower end of rotating crankshaft submerged in the oil sump, radial acceleration generates force to lift the lubricant to where the oil is needed.

Increase in oil pressure due to rotation is best shown as \( \Delta h \), the column height the radial acceleration can deliver. Note the absence of density in the equations below: all particles regardless of their specific weight are subject to the same acceleration and experience different forces in proportion to their specific gravity. That is what drives the centrifugal separation process. Judicious use of this process is the main theme of this paper.

The oil pickup tube shown at right in Fig 5 is described: other arrangements are analogous.

\[
\Delta h = \frac{\Delta v^2}{2g} = \frac{\omega^2}{2g} (r_s^2 - r_0^2)
\]  

(4)

Where \( h \) - the head measured in units of length; \( r_0 \) and \( r_s \) are inlet and tube ID radii.

Adding baffle acting as an impeller, sets the value of initial radius - \( r_s \approx 0 \)

\[
\Delta h = \frac{\omega^2}{2g} r_s^2
\]  

(5)

The baffle increases the pumping head in \( R_h \) ratio:

\[
R_h = \frac{r_s^2}{(r_s^2 - r_0^2)}
\]  

(6)

Sample calculation is given below. These are theoretical values; the actual lifts will be somewhat lower.

For the bottom tube with \( r_1 = 4.0 \text{ [mm]} \) and \( r_0 = 2.2 \text{ [mm]} \), \( R_h = \frac{16}{16 - 4.84} = 1.43 \)

And \( \Delta h = \frac{\omega^2}{2g} r_s^2 = \frac{366.5^2}{2 \times 9810} 4^2 = 109.5 \text{ mm using baffle, or 76.6 mm without baffle.} \)

Second stage at \( r_2 = 6.3 \text{ mm} \) will add \( \Delta h = \frac{\Delta v^2}{2g} = \frac{\omega^2}{2g} (r_s^2 - r_1^2) = 162.2 \text{ mm} \)

This is the total head available; static pressure needed to raise the oil to the point of use has to be deducted to obtain dynamic pressure available to deliver a quantity of oil.

50 Hz operation reduces the head to 69% of figures quoted above. Thus we have:

Without baffle: \( \Sigma h = 76.6 + 162.2 = 238.8 \text{ mm at 60 Hz and 165.8 mm at 50 Hz operation.} \)

With baffle: \( \Sigma h = 109.5 + 162.2 = 271.7 \text{ mm, 60 Hz and 188.7 mm for 50Hz.} \)
When excessive static raise leaves the upper bearings starved for oil too long, an intermediate oil pool can help. See Ref. (4)

4.1. Separating the streams of metal particles, oil and vapor
The arrows show the streams entering and leaving the pump area. In particular, note the presence of recesses \( R \) where the heaviest particles, i.e. metal debris accumulate. This part of the stream must be prevented from entering the bearings. In the sketch on the left, a minor step addition eliminated most of the scoring. Likewise, the step at the entrance to the upward feed passage provides additional metal separation point. Note the oil flow takes the shape of a thin crescent shaped layer on the outer periphery of any conduit, leaving the inner portion of the passage for gas flow, an important feature to remember when it comes to venting.

The provision of a drain hole beneath the oil pump allows for continuous forced bleeding of the metal-oil mixture back into the oil sump. What metal dust is still left in the pump area is deposited on the outer wall of the upward conduit. When the compressor stops running the oil supply system drains back into the sump flushing all the contaminants out. Since during operation the system is being flushed continuously, it is immaterial what wear particles enter the pumping area: nearly all are going to be expelled, but the finest trapped by oil viscosity. Change in the direction of revolution does not affect the operation.

Note the central vent hole has a restriction at the upper end, leading to the radial passage out. This is necessary to prevent priming of oil. When priming occurs at full passage diameter, this wide-open channel siphons most of the oil out. With the restriction shown, the venting action will have some intermittent spitting of oil.

The arrangement on the right of Fig. 5 has similar features. The annular pocket created by pressing the downward pick up tube into the bottom of the crankshaft leaves sizeable gap above the tube, provides for the metal dust trap. The step at the entrance to the upward conduit prevents metal from entering the bearings above that level. Adding a magnet at the bottom of sump can be useful, but in a well-designed system this cost can be avoided.
5. CONCLUSIONS

This paper is a brief summary of common pitfalls every designer of reciprocating machines and hermetic compressors in particular will face in his/her career. Guidelines are scarce but through word of mouth of more experienced colleagues. It seems every engineer entering this field has to learn from scratch from his/her mistakes. This paper addresses lubrication in glossary fashion to point the milestones. Other areas of compressor design are also in need of a similar practical handbook. Any concerted effort to produce such commonly available source of information is likely to be welcome.

6. REFERENCES

The patent references are interactive. Since this paper will be published only in print, OCR scan or retyping in WORD is necessary to activate the links. When on Internet, the next paragraph should show underlined in blue to indicate the link is active. Clicking the underlined titles will bring up the patent pages. Allow time for the patent pages to load.


(1) 4,111,612  Hermetic compressor lubrication system. Paczuski Andrew. Sept. 5, 1978

(2) 4,131,396  Hermetic compressor lubrication system with two-stage oil pump
Privon, Paczuski. December 26, 1978

(3) 6,484,847  Lubricant pump with magnetic and centrifugal traps.
Paczuski Andrew. November 26, 2002

(4) 6,527,085  Lubricating system for compressor. Paczuski Andrew. March 4, 2003
A hybrid simulation methodology for reciprocating compressors

João B. Rovaris, César J. Deschamps
Department of Mechanical Engineering, Federal University of Santa Catarina, Brazil
Francisco F.S. Matos, Fabrício C. Possamai
EMBRACO S.A., Brazil

ABSTRACT

A hybrid numerical approach is proposed to simulate reciprocating compressors. The methodology adopts a differential formulation to predict the compressible turbulent flow through the discharge valve, using the finite volume approach. For the remainder of the compression cycle an integral formulation is applied, with effective flow and force areas being used to evaluate the resultant force and the flow rate for the suction valve. A one degree of freedom model is adopted to predict the valve dynamics. Results show that the methodology is able to capture important flow features of the discharge valve, at a reasonable computational cost.

1. INTRODUCTION

Reed valves are essential parts in hermetic compressors and should have a number of features, such as fast response, high mass flow rate, low pressure drop when opened, and good backflow blockage when closed. These valves are called automatic because they open and close depending on the pressure difference between the cylinder and the suction/discharge chambers, established by the piston motion. Once the valves are open, the pressure flow field has a strong influence on the resultant force acting on the reed. For this reason, and in order to obtain an optimum valve system, it is crucial to recognize and predict the phenomena associated with the flow through the valve and its dynamics.

In (1) a methodology was developed to analyze the non-linear behavior of discharge valve systems, considering a transient compressible flow. The valve dynamics was coupled to the flow field, which was solved from integral conservation laws written for a number of control volumes for the flow through the valve. The study demonstrated the importance of solving the valve dynamics coupled with the flow field through the valve. Several studies available in the literature related to automatic valves either model the valve dynamics in detail but pay little attention to the description of the flow field (2), or focus on the fluid mechanics without considering the coupling between valve motion and pressure distribution on the reed (3).
More recently, new efforts have been directed to model the valve dynamics coupled to the flow. In (4), for the first time, a numerical methodology was developed to explore the interaction between valve dynamics and fluid flow. The model prescribed a pulsating flow condition at the entrance of the valve orifice and from the continuity and momentum equations the flow field was obtained. From the pressure on the valve reed the resultant force was determined and a one-dimensional dynamic model was employed to solve the reed acceleration, velocity and displacement. Later, the model proposed in (4) was extended so as to take into account the turbulent regime that prevails in valves (5).

The main motivation for the present work was to offer a hybrid methodology to simulate the compressor, by combining the differential formulation developed in (5) for the discharge valve with an integral formulation for the remainder of the cycle. The simulation of the complete compressor cycle with a differential formulation would be very time consuming. On the other hand, the solution of the flow through the discharge valve is crucial to correctly predict its performance and, as a consequence, the compressor volumetric efficiency.

2. MATHEMATICAL MODEL

2.1. Integral Formulation
The integral formulation employed here is a simplified version of the code developed in (6). The code accounts for piston displacement as a function of crankshaft angle, the thermodynamic process inside the cylinder, mass flow rate through valves and valve dynamics. Several parameters are calculated during the compressor cycle, such as instantaneous pressure throughout the compressor, mass flow rate and valve dynamics. The transient equations associated with the compressor simulation code are solved via a fourth order Runge-Kutta method. Thermodynamic properties are evaluated using a perfect gas hypothesis.

Valve displacements are modeled by a one degree of freedom mass-spring model. The natural frequency and damping coefficient have to be specified for the valves. Valve stiction is included according to the analytical model proposed in (2). The resultant force acting on the reed and the mass flow rate through the valve are obtained with reference to data on the effective force area $A_{ef}$ and effective flow area $A_{ee}$. If specified, gas pulsation in mufflers can also be modeled. Finally, the thermodynamic process for the gas inside the cylinder can be evaluated either through a polytropic model or by the first law of thermodynamics.

2.2. Differential Formulation
For the Reynolds averaged flow equations (RANS), the value of a computed variable represents an ensemble average over many engine cycles at a specified spatial location. In general, two-equation closures, such as the k-ε model, have been used to model turbulent transport. The shortcomings of RANS models have been widely documented and a discussion of issues related to reciprocating IC engines is given in (7).

Another approach to accounting for the turbulent transport is large-eddy simulation (LES), where flow structures with sizes on the order of the mesh spacing are directly resolved, while the influence of smaller length scales is modeled using a sub-grid scale (SGS) model. Therefore, in LES the governing equations are spatially filtered rather than ensemble averaged. Because statistics of small-scale turbulence are expected to be more universal than those of large scale turbulence, LES offers the promise of wider generality and reduced modeling uncertainty. LES is particularly appealing for application to reciprocating engines, given its
inherent transient operation. In addition to this, the grid refinement commonly used for RANS simulations in IC engines is comparable to that required in LES for resolving a significant range of flow structure scales. Given the difficulties associated with RANS models, LES might actually be simpler, less computationally expensive and more accurate.

The contribution of unresolved (sub-grid scale) motions to the resolved scales can be estimated with the concept of an eddy viscosity, \( \mu_t \), similarly to that used for the Reynolds stress tensor in standard RANS turbulence models. One such a model was proposed in (8) and expresses the eddy viscosity as:

\[
\mu_t = \rho \left( C_s \ell \right)^2 \sqrt{2 S_{ij} S_{ij}}
\]  

where \( S_{ij} \) is the rate of strain tensor and \( \ell \) is a filter width, usually taken as proportional to the local grid spacing, \( \Delta \). According to (9), for non-uniform grids \( \ell \) can be evaluated as the grid spacing geometric average, i.e. \( \ell = (\Delta x \Delta y \Delta z)^{1/3} \). The coefficient \( C_s \) is referred to as the Smagorinsky constant and here is assumed to be 0.17. In this study density fluctuations have been considered negligible when compared to velocity fluctuations. Hence, the density variation is only accounted for through its spatial and temporal variations.

LES certainly implies a transient three-dimensional simulation but, in order to reduce computational cost, at this stage of the research it was decided to adopt an axis-symmetric flow condition, as represented in Figure 1. In this sense, the modeling approach being adopted could be regarded to some extent more like an algebraic eddy-viscosity turbulence model.

The differential governing equations can be written as follows:

- **Mass conservation**
  \[
  \frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} + \frac{1}{r} \frac{\partial \rho v}{\partial r} = 0
  \]  

- **Momentum conservation in the axial (x) direction**
  \[
  \frac{\partial \rho u}{\partial t} + \frac{\partial \rho u u}{\partial x} + \frac{1}{r} \frac{\partial \rho u v}{\partial r} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( \mu_{\text{eff}} \frac{\partial u}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( r \mu_{\text{eff}} \frac{\partial u}{\partial r} \right) + \frac{\partial}{\partial x} \left( \mu_{\text{eff}} \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial r} \left( \mu_{\text{eff}} \frac{\partial \bar{v}}{\partial r} \right)
  \]  

- **Momentum conservation in the radial (r) direction**
  \[
  \frac{\partial \rho v}{\partial t} + \frac{\partial \rho u v}{\partial x} + \frac{1}{r} \frac{\partial \rho v v}{\partial r} = -\frac{\partial p}{\partial r} + \frac{\partial}{\partial x} \left( \mu_{\text{eff}} \frac{\partial v}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( r \mu_{\text{eff}} \frac{\partial v}{\partial r} \right) - \frac{\mu_{\text{eff}} v}{r^2} + \frac{\partial}{\partial x} \left( \mu_{\text{eff}} \frac{\partial u}{\partial r} \right) + \frac{\partial}{\partial r} \left( \mu_{\text{eff}} \frac{\partial \bar{v}}{\partial r} \right)
  \]
• Energy conservation

\[
\begin{aligned}
\frac{\partial \rho T}{\partial t} + \frac{\partial \rho u T}{\partial x} + \frac{1}{r} \frac{\partial \rho v T}{\partial r} &= -p \bar{v} \bar{V} + \frac{\partial}{\partial x} \left[ \left( \frac{k}{c_v} + \frac{c_p \mu_t}{\text{Pr}_t c_v} \right) \frac{\partial T}{\partial x} \right] + \frac{1}{r} \frac{\partial}{\partial r} \left[ \left( \frac{k}{c_v} + \frac{c_p \mu_t}{\text{Pr}_t c_v} \right) \frac{\partial T}{\partial r} \right]
\end{aligned}
\]  

(5)

In the above equations, \( \mu_{\text{eff}} \) is the effective viscosity, comprised by the molecular viscosity, \( \mu \), and the eddy viscosity, \( \mu_t \), i.e. \( \mu_{\text{eff}} = \mu + \mu_t \). The fluid thermal conductivity is denoted by \( k \) and \( \text{Pr}_t \) is the turbulent Prandtl number, for which a value of 0.9 was prescribed. For simplicity, the perfect gas hypothesis was adopted for the state equation required to complete the equation system.

2.3. Valve Dynamics

Reeds are usually made of stainless steel and their dynamics can be expressed in a simplified way, using a one degree of freedom model as follows:

\[
m \ddot{\delta}_1 + C \dot{\delta}_1 + K \dot{\delta}_1 = F - F_0
\]

(6)

where \( F_0 \) is a pre-load force acting on the reed and \( F \) is the force resulting from the pressure distribution on the reed surface. The valve stiffness and damping coefficients, \( K \) and \( C \), respectively, as well as the valve mass, \( m \), are determined experimentally. As illustrated in Figure 1, in this study the reed is considered to be parallel to the valve seat.

In order to solve Eq. (6) for the valve lift \( \delta_1 \), force \( F \) has to be evaluated from the pressure field created by the flow through the valve as follows:

\[
F = \int_0^{D/2} \int_0^{2\pi} \rho \, r \, dr
\]

(7)

Since the flow through the suction valve is not solved, force \( F \) has to be obtained via the effective force area, \( A_{\text{ef}} \).

3. SOLUTION PROCEDURE

A key issue in solving the governing equations is the methodology chosen to handle the physical domain that expands and contracts, since both the piston and the valves are in motion. Here, following the practice adopted in (4), a moving coordinate system was employed. The main feature of this coordinate system is that it transforms the physical domain into a computational domain that remains unchanged regardless any surface motion.

A finite volume methodology was employed to integrate the partial differential equations governing the flow with a fully implicit approximation for transient terms. Staggered control volumes were adopted for the axial and radial velocities. Interpolation of unknown quantities at the control volume faces were obtained using the QUICK interpolation scheme (10), which is considered to be a second-order procedure. A segregate approach was employed to solve the equations and coupling between pressure and velocity was handled through the SIMPLEC algorithm.
The solution domain was discretized with a computational grid with 110x90 (x, r) volumes, as illustrated in Figure 1. Grid refinement was specifically adopted in regions where high gradients are expected to occur in the flow field. Of great help was some evidence of the discretization needed for the flow analysis in the discharge valve and made available in (3). Further details on this issue and on the computational domain dimensions can be found in (11).

![Figure 1: Solution domain and computational grid.](image)

At the solid walls all velocity components were taken to be zero, except at the surfaces of the reed and piston where \( u \) represents their corresponding velocity values. At the valve axis (\( r = 0 \)) the axis-symmetry conditions were imposed as \( \frac{\partial u}{\partial r} = 0 \). At the valve outlet, a boundary condition of pressure was prescribed. To obtain the velocity component normal to the valve outlet from the specified pressure value, a control volume half that of the control volumes adjacent to the boundary was used to integrate the corresponding momentum equation component. By doing so, the velocity could be expressed in terms of the pressure gradient and the neighboring velocity components. Because the velocity obtained from this equation satisfied the momentum conservation but not the mass conservation, such an estimate had to be corrected via the SIMPLEC algorithm, in the same way as in the solution domain. For the velocity component parallel to the outlet boundary, the prescribed condition was that of normal gradient equal to zero.

Concerning the energy equation, a value of 80 °C was prescribed for the wall temperature. For the discharge a local parabolic flow condition was used when the gas is leaving the domain. If the flow is entering the domain, the temperature at the boundary is considered to be that in the discharge chamber.

The coupling between integral and differential formulations needs to be addressed. The compressor simulation starts at the bottom dead center (BDC) with the integral formulation, as
represented in the illustrative diagram shown in Figure 2. The cylinder pressure is the parameter chosen as the reference for change from one formulation to the other. Thus, during the compression stroke the integral formulation is changed to the differential formulation when the pressure reaches a value set at 12 bar. At this point, values for pressure, \( p \), temperature, \( T \), and density, \( \rho \), are directly transferred and used as initial fields for the differential formulation. Other quantities given are the instantaneous piston position and cylinder volume \( V \), both related to the crankshaft angle \( \omega t \), where \( \omega = 2\pi f \) and \( f = 60 \) Hz. No information is available on the velocity field and its initial value is set to zero. For the eddy viscosity, \( \mu_t \), an estimate is obtained from the numerical solution given in (11). After the gas has been pushed out of the cylinder, the discharge valve has been closed and the compressor is in the expansion stroke, the procedure is changed back to the integral formulation when the pressure reaches 8.5 bar. Because flow quantities are available for each control volume of the domain, an average value is required for pressure, temperature and density.

The iterative procedure evaluates properties for each time step until convergence is reached, which is assessed by examining whether the compressor operation conditions are cyclically repeated. A time step corresponding to \( 10^{-3} \) rad was employed for the differential formulation and \( 10^{-2} \) rad for the integral approach. It should be mentioned that a shorter time step was required when the discharge valve was open. By using such time steps, 4 cycles were required to establish a periodic condition. By using such time steps, a number of 4 cycles were required to establish a periodic condition, taking approximately 39 hours on a computer with a single Pentium IV 2 GHz processor.

The differential equation for the valve dynamics, Eq. (6), was solved by considering the force \( F \) to be constant during each time step. As already indicated, in the case of the discharge valve, \( F \) is obtained from the pressure distribution on the reed surface associated with the flow. Yet, for the suction valve, \( F \) must be calculated from effective force area data.

![Figure 2: Schematic of hybrid numerical simulation.](image)
4. RESULTS

The results given below were prepared for four crankshaft positions during the opening of the discharge valve: \( \theta_t = 2.67 \) (a); 2.80 (b); 2.99 (c) and 3.13 (d) rad. Figure 3 shows how pressure at the cylinder wall and valve lift vary according to the crankshaft angle, during the discharge process. With reference to Figure 3, each of the aforementioned points can be associated with the following events during the compressor stroke: (a) discharge valve is opening and the cylinder pressure is near its highest value; (b) discharge valve is still opening and the pressure has dropped; (c) valve is closing and pressure has increased; (d) valve is returning to the valve seat and pressure is decreasing.

The pressure increase to a level above that of the discharge chamber is linked to distinct aspects: i) stiction force between reed and valve seat; ii) valve inertia; iii) increase in stiffness and natural frequency due to the presence of a booster; iv) flow through the valve. The first two effects are present when the valve is closed. In this simulation, the opening of the valve occurred when the cylinder pressure reached 15 bar. The valve lift was limited to 0.9 mm and a booster was set to act when the valve displacement was 0.3 mm.

The performance of the valve system can be analyzed by examining results for in-cylinder pressure and valve displacement. For instance, it is possible to identify when the valve opens and the associated overpressure condition created by the stiction force due to the oil film between the reed and valve seat. According to Figure 3, after the valve is open the pressure keeps rising up to point (a) due to the flow restriction brought about by a small valve lift. In this regard, low levels of velocity can be observed from results for velocity vectors shown in Figure 4a.

![Figure 3: Pressure at the cylinder wall and discharge valve lift.](image-url)
The discharge valve keeps opening and, when the lift is large enough, the cylinder pressure starts to drop down to point (b). From this crankshaft position the pressure is seen to increase again, reaching a new local maximum condition, point (c). This occurs for two reasons: i) increase in flow viscous dissipation, due to the small clearance between the piston and the cylinder head; ii) closing motion of the valve, restricting the flow passage area. The viscous dissipation effect in the cylinder clearance can be observed from the results for isobars, shown in Figure 5. As can be seen in Figure 5c, there is a pressure drop of approximately 1 bar between the cylinder wall and the valve orifice.

For the compressor model here considered, the top dead center (TDC) is reached at a crankshaft angle of 3.10 rad. Since point (d) represents an angle of 3.13 rad, in this position the piston has already started the expansion stroke, originating an abrupt pressure drop in the cylinder that acts to close the discharge valve. Depending on the valve dynamics, some backflow can occur before the valve is totally closed, deteriorating the volumetric efficiency of the compressor. This important aspect is captured by the simulation under study, as shown by the velocity vectors in Figure 4d.

Figure 4 shows the presence of recirculating regions on the valve seat, and even inside the feeding orifice. These flow features act to reduce the flow passage area and, as a consequence, the valve efficiency.

![Figure 4: Velocity vectors during the discharge process.](image-url)
5. CONCLUSIONS

This paper presented a methodology to simulate reciprocating compressors, with particular attention given to the discharge valve dynamics. In order to reduce the computational time, integral and differential formulations were combined. The compressible turbulent flow through the discharge valve was predicted via large-eddy simulation (LES), with the Smagorinsky model for the sub-grid scale motions. Results are seen to display several important flow features found in compressors, such as pressure over shooting in the cylinder, recirculating flow regions and back flow through the discharge valve. Future developments will be concentrated on extending the simulation to a three-dimensional geometry, implementing alternative sub-grid scale models and numerical techniques. In addition, the differential formulation will also be applied to the flow through the suction valve.

ACKNOWLEDGEMENTS

This work forms part of a joint technical-scientific program of the Federal University of Santa Catarina and EMBRACO. Support from the Brazilian Research Council, CNPq, is also acknowledged.

REFERENCES

(3) C.J. Deschamps, A.T. Prata and R.T.S. Ferreira, "Turbulent Flow through Reed Type Valves of Reciprocating Compressors", Proc. ASME International Congress and


ROTARY VANE COMPRESSORS
A comprehensive model of a sliding vane rotary compressor system

Roberto Cipollone\textsuperscript{b}, Giulio Contaldi\textsuperscript{a}, Antonio Sciarretta\textsuperscript{b}, Raffaele Tufano\textsuperscript{a}

\textsuperscript{a} Ing. Enea Mattei S.p.A., Italy
\textsuperscript{b} University of L'Aquila, DIMEG, Italy

ABSTRACT

This work presents a complete system-level model that simulates the geometry and the thermo-fluid dynamics of sliding vane rotary compressors. The detailed geometrical model includes realistic geometries for suction and discharge ports, non-radial vanes, and non-circular stators. The fluid dynamic model is a lumped-parameter model but it is able to represent wave propagation. The overall model is validated with measurements taken from a six-vane machine with varying rotational speed and discharge pressure.

1. INTRODUCTION

Using compressed air in the industrial and service sector is a common practice, since production, handling, and use are relatively safe and easy. Compressed air accounts for as much as 10\% of industrial consumption of electricity, or over 80 TWh per year in the European Union (1). Among various technical measures that could improve overall performance of a compressed air system, particular importance has been recognized to the optimal choice of the type of compressor as a function of specific end use and applications, the improvement of compressor technology, the use of sophisticated control systems, the recovery of waste heat for integration with other functions, the overall system design.

Like other volumetric compressors, sliding vane rotary compressors (SVRC) have an inherent advantage over centrifugal compressors, consisting of having a rather constant air delivery rate as a function of discharge pressure. Moreover, when compared to, e.g., reciprocating compressors, SVRCs are generally less noisy and better balanced machines, being capable of operating at higher rotational speeds, and suffering less from vibrations. A comparison over the specific energy - input power per volume throughput - is also usually favourable to SVRCs, typical values, 5-6 kW/(m\(^3\)min). A major drawback is that these machines suffer from internal air leakage losses that are overcome by reducing constructive clearances, at the expense of substantial lubricating oil circulation and the need of complex oil separation components. However, such a circumstance opens new interesting possibilities in terms of energy efficiency.
As a matter of fact, the substantial heat losses of SVRCs can be partially recovered, e.g., to feed Peltier modules and thus deliver electricity in addition to compressed air with an integrated, very compact system. Another possible configuration may recover waste heat by means of an absorption heat pump, and provide an efficient air conditioning device without additional moving parts. In such scenarios, the compressor itself becomes a single component of a strongly interconnected energy system that is oriented to improved energy saving, especially in all those applications that require flexibility of use and compactness like, e.g., automotive applications.

Considered as isolated machines, SVRCs have been long studied in terms of geometrical characteristics, to improve performance by optimizing some design parameters (number of vanes, vane inclination, shape of the stator, rotor radius, etc.), and in terms of the forces exchanged at the various contact points between the machine parts (2,3).

However, the evaluation of the machine performance additionally requires a thermodynamic model to calculate how the pressure varies in the cells as a function of time. This calculation is strongly coupled with the fluid dynamics modelling of suction, discharge, and oil circuits, particularly when the compressor is part of a multi-component energy system. Being such a complex task, circuit modelling has been often neglected in literature works (2,3).

The aim of the present work is to develop and validate a complete system model approach that simulates machine geometry and thermodynamics, as well as fluid dynamics and thermal status of various circuits and related components. In the paper, first a detailed geometric model is presented, including seldom considered aspects like internal clearances, frontal suction ports, non-radial vanes. Then, the system model is described for a general configuration, with some details on the approaches used to simulate fluid dynamics and oil saturation. The overall model is validated with measurements taken from (4) for a six-vane machine with varying rotational speed and discharge pressure.

2. GEOMETRIC MODEL

The aim of this sub-model is the evaluation of the volume of each cell as a function of the rotor motion.

A four-vane SVRC is sketched in Fig. 1. The figure shows the circular stator (S: stator axis), the circular rotor (O: rotor axis), the suction port L'L, and the discharge port M'M. Each of the four cells is defined by the stator surface, the rotor surface, the leading and trailing vanes. Fig. 2 shows the relevant geometric quantities for the vane characterization.

If C and B are the intersections of the leading vane axis with the rotor surface and the stator surface, respectively, and C', B' the corresponding points for the trailing vane, the cell area \( A_C \) is given by

\[
A_C = CC'B'B = OCC'B'OB'B-OCB - OC'B'
\]

(1)
\[
OCC' = SCC' + OSC - OSC'
\]

(2)

\[
SCC' = \frac{R^2}{2} \cdot (\beta_c - \beta_{c'})
\]

(5)

\[
OBB' = \frac{\pi \cdot r^2}{n}
\]

(3)

\[
OSC = \frac{1}{2} \cdot e \cdot R \cdot \sin \beta_c
\]

(6a)

\[
OSC' = \frac{1}{2} \cdot e \cdot R \cdot \sin \beta_{c'}
\]

(6b)

\[
OCB = \frac{1}{2} r_v \cdot (c - b)
\]

(4a)

\[
OC'C' = \frac{1}{2} r_v \cdot (c' - b)
\]

(4b)

**Table 1 – Geometric quantities**

The terms in equation (1) are reported in table 1, having defined the quantities \(b\), \(c\) and \(c'\) as

\[
b = AB = \sqrt{r^2 - r_v^2}
\]

(7a)

\[
c = AC = -e \cdot \sin \alpha + \sqrt{R^2 - (r_v - e \cdot \cos \alpha)^2}
\]

(7b)

\[
c' = AC' = -e \cdot \sin \alpha' + \sqrt{R^2 - (r_v - e \cdot \cos \alpha')^2}
\]

(7c)

being the rotor radius \(r\), the stator radius \(R\), the eccentricity \(e\), and the vane circle radius \(r_v\).

The \(\beta_c\) and \(\beta_{c'}\) are the SC and SC’ vector angles that are calculated as

\[
\sin \beta_c = \frac{r_v \cdot \sin \alpha + c \cdot \cos \alpha}{R}
\]

\[
\cos \beta_c = \frac{r_v \cdot \cos \alpha - c \cdot \sin \alpha - e}{R}
\]

(8a)

\[
\sin \beta_{c'} = \frac{r_v \cdot \sin \alpha' + c' \cdot \cos \alpha'}{R}
\]

\[
\cos \beta_{c'} = \frac{r_v \cdot \cos \alpha' - c' \cdot \sin \alpha' - e}{R}
\]

(8b)

while \(\alpha' = \alpha - 2\pi / n\).
Combining all the previous equations, the cell area is

\[ A_c = \frac{R^2}{2} \cdot (\beta_c - \beta_c') + \frac{1}{2} \cdot e \cdot R \cdot (\sin \beta_c - \sin \beta_c') - \frac{\pi \cdot r^2}{n} - \frac{1}{2} \cdot a \cdot (c - c') \]  

(9)

To the area \( A_c \), various terms should be subtracted or added to take into account the volume of the vane and the space between the vane and its slot. The area subtracted due to the occupancy of the vane between the point C and the point B is

\[ \Delta A_1 = -\frac{t_v}{2} \cdot (c - b) - \frac{t_v}{2} \cdot (c' - b') = -t_v \cdot \frac{c + c'}{2} + t_v \cdot b \]  

(10)

where \( t_v \) is the vane thickness. The area added due to the crevice space between the point B and the point Q (vane root) is

\[ \Delta A_2 = \frac{t_s}{2} \cdot [l_v - (c - b)] + \frac{t_s}{2} \cdot [l_v' - (c' - b')] = -(t_s - t_v) \cdot \frac{c + c'}{2} + (t_s - t_v) \cdot (b + l_v) \]  

(11)

where \( t_s \) is the slot thickness and \( l_v \) is the vane height. The area added due to the crevice space between the vane root (point Q) and the slot end is

\[ \Delta A_3 = \frac{t_s}{2} \cdot [l_v'' - (c - b)] + \frac{t_s}{2} \cdot [l_v''' - (c' - b'')] = -t_s \cdot \frac{c + c'}{2} + t_s \cdot (l_v - b - l_v) \]  

(12)

where \( l_s \) is the slot depth. The final expression for the cell volume is

\[ V = H \cdot (A_c + \Delta A_1 + \Delta A_2 + \Delta A_3) \]  

(13)

where \( H \) is the rotor length.

Two distinct ways of filling and emptying the SVRC cells exist. The passage areas may be located on the lateral surface of the stator (as slots, arc LL’ in Fig. 1), leading to a radial intake/exhaust flow. Otherwise, the passage areas may be located on the frontal surface of the stator (as ports, area between the stator and the segment LL’ in Fig. 1), leading to an axial intake/exhaust flow. In the first case the location of the slots is identified by two angles, \( \beta_L \) and \( \beta_L' \), which refer to the opening and closing of the passage area. When the leading vane contact point angle \( \beta_c \) lays between \( \beta_L \) and \( \beta_L' \), the passage area facing the control cell is calculated as

\[ A_v = \max\{\beta_L, \min\{\beta_c, \beta_L'\}\} - \max\{\beta_L, \min\{\beta_c, \beta_L'\}\} \cdot R \cdot H \cdot \xi_s \]  

(14)

where \( H \cdot \xi_s \) is the total axial length of the suction port. In case of frontal suction, still described by the stator angles \( \beta_L' \) and \( \beta_L' \), the frontal area is calculated with the integral

\[ A_v = \frac{1}{2} \cdot \int_a^q \left( \rho^2_c - \rho^2_d \right) \cdot d\alpha \]  

(15)

The point D is defined as the intersection of the vane with the inner surface of the frontal suction port. The calculation of the position of D requires a complex evaluation of various geometric parameters that is not described here for simplicity.
3. FLUID DYNAMICS MODEL

Generally, the fluid dynamics model includes several branches and capacities. The model uses the equations of the Quasi-Propagatory Model (5,6) to calculate flow properties (e.g., mass flow rates) in the branches and state properties (pressure, temperature, oil concentration) in the capacities.

3.1 Capacities

Each cell is described by a capacity (see Fig. 3) whose volume varies with time according to the law derived in the previous section. A cell capacity exchanges mass and energy with the other cells through the vane clearance and, if it is the case, with the suction and discharge circuits via the respective ports. Moreover, it exchanges energy with the external environment through the machine walls.

Suction and discharge ports are also described by capacities that exchange mass with all the cells that are simultaneously inducting and, respectively, discharging. The volume of these capacities is generally well defined for lateral ports. In case of frontal suction, the calculation of the port volume is more complex and strongly dependent on the single machine.

Additional capacities and branches may describe the discharge circuit, with its typical components (oil separators, aftercoolers, etc.). The boundary condition of the machine after all the accessories may describe the discharge pressure level in two ways. Either a discharge through a valve to an ambient pressure reservoir or a free discharge to a reservoir at the desired pressure level are both suitable as models. In the first case, the valve also represents the distribution line, including the final nozzle that discharge at ambient pressure. In the second case the machine is considered as discharging at the operating pressure level.

General equations for the capacities concern the variables pressure $p$, temperature $T$, oil concentration $c$, total mass $m$, oil mass $m_o$, internal energy $u$, molar mass $M$. Mass conservation law is written as

$$\frac{dm}{dt} = \sum \dot{m}$$ (16)
where the sum is extended to all the mass exchange paths (e.g., for a cell: the suction and discharge ports and the leakage with adjacent cells). The flow terms due to exchange between adjacent cells are calculated using a linear relationship between pressure difference between cells and mass flow rate across the leakage (i.e., \( \dot{m}_{\text{leak}} = k_{\text{leak}} \cdot \Delta p \)). The remaining mass flow rate terms are calculated with the branch equations. Similarly, mass conservation law for the oil fraction is

\[
\frac{d\dot{m}_o}{dt} = \sum \dot{m}_o - \dot{m}_{\text{inj}} - \dot{m}_{o,s}
\]

where the terms under the summation describe oil flow exchanged with air flow. The injected oil mass flow rate \( \dot{m}_{\text{inj}} \) is relevant only for the cell capacities. The third term in the right-hand side of Eq. 17 describes the oil mass that condenses when oil fraction is larger than the equilibrium fraction at the current mixture pressure and temperature

\[
\dot{m}_{o,s} = \dot{m}_o - f_{sa}(p,T) \cdot \frac{M_o \cdot m}{p}
\]

where the saturation pressure \( f_{sa}(\cdot) \) is a tabulated function of pressure and temperature. The mixture molar mass is calculated as

\[
M = \frac{1}{c} \cdot \frac{M_o \cdot m}{1-c}
\]

where \( c = \frac{m_o}{m} \) is the oil mass fraction. Energy conservation law implies the evaluation of enthalpy flows associated with mixture flows (\( \dot{m} h \)), mechanical work (\( -p \cdot \dot{V} \)) for cell capacities, heat flows (\( \dot{q} \)), latent heat of vaporization and condensation,

\[
\frac{du}{dt} = \sum \dot{m} h - \dot{q} - p \cdot \dot{V} + H_{LHV}(T) \cdot \left( \frac{d\dot{m}_o}{dt} - \frac{d\dot{m}_{\text{inj}}}{dt} \right)
\]

The heat flow rate is calculated as proportional to the temperature difference between cell air and walls via a global heat coefficient \( \chi_{hf} \). This represents a simplified but practical way to describe the heat convection between the compressor casing and the environment, as well as the heat removed by the air. Finally, mixture temperature and pressure are updated using ideal gas properties

### 3.2 Branches

In the present model, the branches are elements that connect two capacities, allowing for the evaluation of flow quantities such as mass and energy flow rates. General equations for the branches are within the framework of QPM. Branches are subdivided in "total-to-static" and "static-to-static" types (see Fig. 4). The former type (e.g., suction duct, single discharge branches) is bounded by an upstream capacity with total pressure \( p_u \) and a downstream capacity with static pressure \( p_d \). In the latter type, both \( p_u \) and \( p_d \) are static pressures. According to the value of \( p_u, p_d \) (and also \( T_u, T_d \)), a steady-state fluid velocity \( u_{\infty} \) is evaluated, possibly taking into account non-homentropic phenomena. Then the fluid velocity \( u \) is calculated with a first-order model
\[
\frac{du}{dt} = \frac{u_\infty - u}{\tau(p_u, p_d, \varphi, ...)}
\]  

or with a second-order dynamic model (which is further simplified to a quasistatic model),

\[
u = u_\infty
\]

according to the sign of a discriminating parameter \(\lambda\) that is a function of the boundary conditions, the partial restriction coefficient \(\varphi\), and non-hometropic factors (heat flows, cross-section variations, friction). The time constant \(\tau = -2 \cdot (l/a)/\lambda\) is a function of the same parameters and also of the branch length \(l\).

4. RESULTS

This section presents a validation of the model with measurements taken from the literature work of Tramschek and Mkumbwa (4). The literature data are for a 5 kW compressor, with air delivery range of 10-20 l/s and an operating pressure range of 7.5-10.5 bar. The validated model is used to analyze the influence of various tuning parameters on the machine performance estimation.

4.1 Model Validation

In their original paper (4), Tramschek and Mkumbwa investigated five machines sharing the same stator but utilizing different rotors with vane slots of different inclination. Common geometrical data are listed in Table 2. The compressor stator is circular with lateral suction ports, which makes the implementation of the model particularly straightforward.

The fluid dynamic model structure in terms of branches and capacities was built using the level of detail summarized in Table 3. The fluid dynamic network is generalized as follows. There is a common branch at the suction side, connecting the external ambient to a suction capacity (typically, the volume between the intake valve and the suction port) via a constant restriction described by the area ratio \(\psi\). From hence, a number of \(n\) fictitious branches (\(n\) is the number of vanes) reaches in principle all the vanes. Of course, during the machine operation only one or few of them will be really active, depending on which vanes are actually in communication with the suction port. Similarly, at the discharge side there are \(n\) fictitious branches directed to a discharge capacity. The discharge circuit is described by two branches separated by a capacity (typically representing the oil separator). The end branch faces the discharge ambient at a pressure \(p = \Pi \cdot p_s\), being \(\Pi\) the pressure ratio. As already discussed in Section 3, this choice implies that there is no restriction coefficient at the end branch.

<table>
<thead>
<tr>
<th>Stator Radius</th>
<th>(R)</th>
<th>47.52 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor Radius</td>
<td>(r)</td>
<td>40.05 mm</td>
</tr>
<tr>
<td>Eccentricità</td>
<td>(e)</td>
<td>7.35 mm</td>
</tr>
<tr>
<td>Suction Port Opening Angle</td>
<td>(\beta_s)</td>
<td>244.75 deg</td>
</tr>
<tr>
<td>Suction Port Closing Angle</td>
<td>(\beta_c)</td>
<td>328.93 deg</td>
</tr>
<tr>
<td>Discharge Port Opening Angle</td>
<td>(\beta_M)</td>
<td>136.57 deg</td>
</tr>
<tr>
<td>Discharge Port Closing Angle</td>
<td>(\beta_m)</td>
<td>146.33 deg</td>
</tr>
<tr>
<td>Vane Height</td>
<td>(l_v)</td>
<td>22.54 mm</td>
</tr>
<tr>
<td>Rotor Length</td>
<td>(H)</td>
<td>170.00 mm</td>
</tr>
<tr>
<td>Discharge Port Axial Length Ratio</td>
<td>(\xi_d)</td>
<td>0.824</td>
</tr>
<tr>
<td>Suction Port Axial length Ratio</td>
<td>(\xi_s)</td>
<td>0.154</td>
</tr>
<tr>
<td>Rotor Slot Depth</td>
<td>(l_s)</td>
<td>22.83 mm</td>
</tr>
<tr>
<td>Vane Thickness</td>
<td>(t_v)</td>
<td>3.95 mm</td>
</tr>
<tr>
<td>Slot Thickness</td>
<td>(t_s)</td>
<td>4.10 mm</td>
</tr>
<tr>
<td>Number of Vanes</td>
<td>(n)</td>
<td>6</td>
</tr>
<tr>
<td>Vane Circle Radium</td>
<td>(a)</td>
<td>0 mm</td>
</tr>
</tbody>
</table>

Table 2: Compressor Geometrical Parameters
<table>
<thead>
<tr>
<th>Branch no.</th>
<th>Upstream Capacity</th>
<th>Downstream capacity</th>
<th>Type</th>
<th>$\phi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ambient at $p = p_a$</td>
<td>1</td>
<td>1</td>
<td>$\psi$</td>
</tr>
<tr>
<td>$2, \ldots, n+1$</td>
<td>1</td>
<td>$2, \ldots, n+1$</td>
<td>-1</td>
<td>1</td>
</tr>
<tr>
<td>$n+2, \ldots, 2n+1$</td>
<td>$n+2$</td>
<td>$n+3$</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>$2n+2$</td>
<td>$n+3$</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>$2n+3$</td>
<td>ambient at $p = \Pi \cdot p_a$</td>
<td>-1</td>
<td>1$^a$</td>
<td></td>
</tr>
</tbody>
</table>

$n$ is the number of vanes (see Table 2); type $\pm 1$ refers to "total-to-static" or "static-to-static" branches, respectively; $^a$: see text

**Table 3: Branches used in the model**

Some of the model parameters had to be tuned, since their values for the experimental campaign of (4) are not known.

The most important parameters that were calibrated with their respective value are listed in Table 4. This data set will be referred to as "base case". Notice that in the base case there is no oil mass injected. The cooling effect is lumped in the heat transfer process, which is set as infinitely effective, i.e., the thermodynamic transformations in the cells are practically isothermal.

Free air delivery varies with the compressor speed and the discharge pressure (or discharge/suction pressure ratio). Figure 5 shows the increase of free air delivery $Q_e$ as a function of compressor rotational speed $N$, as calculated for two values of the compression ratio, namely, $\Pi = 7$ and $\Pi = 11$.

The experimental results of (4) show instead a negligible dependency of $Q_e$ on $\Pi$. The reliability of the model proposed is confirmed by the fact that the experimental curve lays between the two curves calculated at the chosen values of $\Pi$.

The agreement obtained between measurements and calculations is also shown in Fig. 6, where $Q_e$ is plotted as a function of $\Pi$ for various values of $N$. This confirms that the measured free air delivery seems not to depend on $\Pi$ (constant dashed lines), while calculated data show a decrease of $Q_e$ as $\Pi$ increases. This is not due to air leakage, since this analysis used a zero value for $k_{\text{leak}}$ (see Table 3). The decrease of $Q_e$ is rather due to increased reverse flow from high-pressure zones to low-pressure zones of the machine.
Besides free air delivery, the other main machine parameter is the indicated power $P_i$, which is calculated as the average value during a stationary compressor cycle of the instantaneous power $p \cdot \dot{V}$ calculated for a cell. Figure 7 shows how the indicated power increases both with $\Pi$ and $N$ almost linearly. The correct evaluation of the shaft power would require a detailed model of the friction losses between moving parts of the machine, as well as of the power drained by the accessories. These aspects will be the subjects of further investigation.

4.2 Sensitivity Analysis
The model validation of the previous section required a fine tuning of some unknown model parameters, namely, the control flow restriction coefficient $\psi$, the coefficient $k_{\text{leak}}$ for the calculation of leakage flows, the global heat flow coefficient $\chi_{\text{hf}}$, the oil mass injected per revolution $m_{\text{inj}}$ (see Table 3). The aim of this section is to investigate how variations of such parameters influence the machine performance calculated. The figure of merit used to discuss the results is the relative sensitivity. In the range of the independent variable $x \in [x_{\text{min}}, x_{\text{max}}]$ the relative sensitivity of the variable $y$ is defined as:

$$s_y = 100 \cdot \frac{y_{\text{max}} - y_{\text{min}}}{y_{\text{max}}} \cdot \frac{x_{\text{max}} - x_{\text{min}}}{x_{\text{max}}}$$

(23)

In volumetric compressors, the air delivery is controlled by acting on a throttling device at the suction side of the machine. Figure 8 shows how the air delivery and the indicated power vary as a function of the restriction coefficient $\psi$ (0: fully closed, 1: wide open). Both quantities are monotonically increasing with $\psi$, with a nonlinear trend that is typical of throttled inflows from a large reservoir (in this case, the external ambient) to a pipe. However, while the air delivery tends obviously to zero as $\psi$ becomes smaller, the indicated power remains at a substantially higher value. This behaviour is better understood looking at the indicated cycle of a single cell on the p-V plane (Fig. 9).
Even with a very strong reduction of the inflow area ($\psi=0.01$), which causes the air delivery to be less than 15% of its maximum value, the indicated cycle has a residual area in the p-V plane that depends on the position of discharge and suction ports. This area is representative of work absorbed by the machine and then of indicated power. Free air delivery and indicated power vary almost linearly with the leakage coefficient $k_{\text{leak}}$ (Fig. 10). With respect to the reference case $k_{\text{leak}}=0$ (Table 4), the relative sensitivity of free air delivery is 17%, of indicated power 6%.

In contrast, the variation of machine performance with heat flow global coefficient is nonlinear (Fig. 11). Both $Q_e$ and $P_i$ decrease as $\chi_{hf}$ increases and the air compression approaches an isothermal process (base case), which is characterized by the lowest value of $P_i$ for a given $\Pi$. For the same reason, the pressure during the compression phase decreases as the polytropic coefficient decreases (ranging from the ratio of the specific heats in the adiabatic case to the value 1 in the isothermal case) and consequently so does the density and hence the mass inducted. The relative sensitivity of free air delivery is 7%, of indicated power 12%.

As the oil mass injected increases, the maximum temperature in the cell decreases. Consequently, air density and mass inducted per revolution increase. This trend is clearly shown in Fig. 12, together with a correspondent increase of indicated power.
5. CONCLUSIONS

A system-level model of a sliding vane rotary compressor and of its circuits was presented. The model comprises two parts, describing: (i) the variable geometry of the cells, and (ii) the fluid dynamics and thermodynamics of the air and oil flows in the compressor, as well as in the suction and discharge circuits.

A high flexibility, being capable of simulating various arrangements, including frontal or lateral suction ports, circular or non-circular stators, and radial or non-radial vanes characterize the former submodel. Clearances between slots and vanes are modelled in detail, as their influence on the geometric compression ratio was shown to be very important. The fluid dynamics description is based on a lumped-parameter representation of the flows between the various parts of the system, the Quasi-Propagatory Model having been adopted. Thermal aspects are included as well, particularly for what concerning the heat flows through the compressor walls, and the air saturation with the injected oil.

The model is generally based on physical relationships and first-principle analyses. Nevertheless, some parameter that is difficult to estimate were left to be tuned, in order to validate the model. Experimental data taken from the literature were chosen for this purpose. The comparisons between experimental and calculated data show a good agreement for different rotational speeds and compression ratios. As a general trend, the experiments used show a very small decrease of the free air delivery as the compression ratio increases. The model systematically overestimated such a decrease.

This aspect requires a more precise description of the boundary conditions to apply at the interfaces between the various parts of the system. In particular, the authors think that the air recirculation between discharge and intake ports predicted by the model under certain circumstances could not actually take place in the experimental set up, due to oil sealing effects. In order to further investigate such interesting modelling aspects, as well as to further validate the submodels of the circuit components, a proprietary experimental campaign is ongoing. The results of this investigation will be presented in a future publication.

REFERENCES

Concept of oscillating-roller rotary compressor

N. Dreiman, R. Bunch
Tecumseh Products Company, USA

ABSTRACT

The mechanical structure of a new rotary compressor and its operating principle is introduced in this paper. The crankshaft of the compressor is stationary. The motor-rotor, cylinder block and vane are formed as integral part, which rotates around axis of the stationary crankshaft in the magnetic field created by the motor-stator. The eccentrically mounted roller driven by the interaction with the vane rotates synchronously with the cylinder block. The vane is clamped without clearance between inner sides of heads to which free end of the vane is fixed. Analysis indicates that leakage and frictional losses have been eliminated at the following location: vane tip-roller O.D., vane ends-facing surfaces of the cylinder heads, and vane sides-slot sides of the rotor-cylinder.

1. INTRODUCTION

In the last few years’ carbon dioxide received increasing attention as possible replacement of fluorocarbon-based refrigerants used at present for vapor compression cycle technology. One concern with R –744 refrigerants is the effect of high operating pressure. Large difference between discharge and suction pressure can trigger in contemporary rotary compressors higher leaks and friction losses, increased load on the bearings, etc. Fatigue problems due to higher impact velocities (lift and closure of the valve, for example) when compressing such a relatively dense gas, as CO$_2$ (density is $\approx 5$ times higher than that of R-22), must also be considered. The conventional rotary compressors embodied a stationary cylinder block provided with peripheral movable vane engaged directly with the wall of an eccentrically mounted roller. The vane has to be holding against the roller wall through the action of spring, back gas pressure and is movable upon the cylinder block. Use of CO$_2$ as the refrigerant will increase frictional and leak losses and will affect performance and life span of the contemporary rotary compressor. A prototype compact hermetic rotary compressor has been developed for use with carbon dioxide to overcome problems specified above [1, 2].
2. OPERATING PRINCIPLE

The mechanical structure and operating principle of rotary oscillating-roller compressor is in some degree similar to that of contemporary rotary sliding vane or swing type compressor. The difference is that the crankshaft of new compressor does not rotate. The motor-rotor, cylinder block and vane are formed as integral part (rotor-cylinder), which rotates around axis of the stationary crankshaft in the magnetic field created by the motor-stator. The eccentrically mounted roller driven by the interaction with the vane rotates synchronously with the rotor-cylinder. Fig.1 shows schematic sectional views of the progressive positions for the integral cylinder block-vane and roller during operating cycle of the compressor.

3. MOTION ANALYSIS

3.1 Kinematics of the compressor

The rotor- cylinder and roller can be considered consequently as external cylinder (driver) and internal cylinder (follower) having rolling contact in point P with transmission of the motion from the driver to the follower through the rigid link C (coupler). The external and internal cylinders will rotate in one direction. According to the structure, the vane presented in Fig.2 as link C, separates suction and compression sides and extends radially inward toward rotor-cylinder center of rotation OC through the center of the bushing mounted in the roller. The line connected rotor-cylinder center OC and point PR (see Fig.2A) is considered as a driving link with OC PR = d, OC and roller center OR are individually taken as the two immovable points of the mechanism with OC OR = e (fixed link). The line OR PR = f considered as a follower. The equation of motion of the compressor elements such as integral rotor- cylinder – vane, roller and roller bushing can be derived by choosing the orthogonal coordinate system fixed to the stationary crankshaft with origin in rotor-cylinder-vane center OC, the x axis lie along common line, namely the line of centers directed to the point of contact P and the z axis coincided with the stationary crankshaft axis. For plane triangle ∇OCPROR (see Fig. 2A)

\[ e \sin \theta = f \sin \varphi ; \quad d = e \cos \theta + f \cos \varphi \]  

(1)
Due to the fact that the rotor-cylinder-vane are secured together and rotates as one solid body around $z$ axis, there is no reciprocating movement of the vane and implanted in the roller bushing flats slide along the suction and discharge sides of the vane with change of the turning angle $\theta$. The part of the vane $L(\theta)$ exposed to the suction and discharge pressure inside of the working chamber will be

$$L(\theta) = P_R P_C = R_C - d,$$

where radius of the cylinder $R_C = R_R + e$ and radius of the roller $R_R = f$. After substitution and transformation exposed length can be expressed as

$$L(\theta) = e ( 1 - \cos \theta ) + f \left\{ 1 - \left[ 1 - \left( \frac{\lambda \sin \theta}{2} \right)^2 \right]^{1/2} \right\},$$

where $\lambda = e / f$. After expanding the root into a binominal series with inclusion of the first two terms as the series rapidly converges, we obtain

$$L(\theta) = e ( 1 - \cos \theta + \frac{\lambda^2}{2} \sin^2 \theta - \frac{\lambda^3}{8} \sin^4 \theta + \ldots \ldots )$$

Thus the length of the exposed part of the vane very approximately (error 0.15%) is

$$L(\theta) = e ( 1 - \cos \theta + \frac{\lambda^2}{2} \sin^2 \theta )$$

An angle that define the roller bushing turning angle $\varphi_1$ (see Fig.2B) with change of the vane rotational angle is found to be

$$\varphi_1 = \arcsin \left[ e \sin \theta / ( R_R - R_B ) \right]$$

where $R_B$ is radius of the roller bushing. The values of the bushing turning angle $\varphi_1$ and exposed vane length $L(\theta)$ with change of the rotational angle $\theta$ in the limits 0-2$\pi$ (single revolution) are shown in Fig.3.
Fig. 3. Turning angle of vane bushing and exposed length of the vane with change of the rotational angle.

3.2 Piston-swept volume

The swept volume $V(\theta)$ of the working chamber at an arbitrary angle of the vane is

$$V_d(\theta) = A(\theta) \cdot h,$$  \hspace{1cm} (7)

where $h$ is the height of the cylinder and $A(\theta)$ is the area of compression chamber at an arbitrary angle $\theta$ of the vane position.

$$A(\theta) = \int_0^\theta (R^2_C - d^2) \, d\theta = \int_0^\theta [R^2_R (1 + \lambda)^2 - d^2] \, d\theta,$$  \hspace{1cm} (8)

where length of the driving link

$$d = R_C - L(\theta) = R_R (1 + \lambda \cos \theta - \lambda^2/2 \sin^2 \theta)$$  \hspace{1cm} (9)

or

$$d^2 = R^2_R (1 + 2\lambda \cos \theta + \lambda^2 \cos 2\theta)$$  \hspace{1cm} (10)

if we neglect the terms containing $\lambda^3$ and $\lambda^4$ due to the small value and putting $\cos 2\theta = \cos^2 \theta - \sin^2 \theta$.

Refer back to Eqv. 7 and Fig. 4 the compression chamber volume trapped within cylinder, vane and roller at an arbitrary angle $\theta$

$$V(\theta) = \lambda \cdot h \cdot R^2_R/2 \left[ (1 + \lambda) \theta - 2 \sin \theta - \lambda/2 \sin 2\theta \right]$$  \hspace{1cm} (11)

In this calculation we have neglected the volume occupied by the vane. To account for the vane thickness, the volume given by Eqv. (11) has to be reduced by the volume of the exposed part of the vane in the working chamber:

$$V_1(\theta) = V(\theta) - h \cdot t \cdot L(\theta) = V(\theta) - e \cdot h \cdot t \left( 1 - \cos \theta + \lambda^2/2 \sin^2 \theta \right),$$  \hspace{1cm} (12)

where $t$ is the thickness of the vane.
3.3 Velocity

Motion of external cylinder is transmitted to the internal cylinder through the driving link C. If driving link C rotates counterclockwise with constant angular velocity, the follower will revolve in the same direction at a varying (accelerating and decelerating) speed. The relative velocity between the cylinders is a function of the driver angle $\theta$, radii of the contact cylinders, centers distance. The linear velocities of the cylinder $v_C$ and roller $v_R$ are shown below:

$$v_C = \frac{dS_C}{dt} = \omega \frac{dS_C}{d\theta} = \omega R_C$$  \hspace{1cm} (13)

$$v_R = \frac{dS_R}{dt} = \omega \left[ \int_0^\theta d\theta \right] / \theta = \omega d$$ \hspace{1cm} (14)

where $S_C$ and $S_R$ are revolving arcs of the cylinder and roller.

The relative velocity of the roller to the cylinder

$$\Delta v = \omega (R_C - d)$$ \hspace{1cm} (15)

Taking derivative we find that

$$dv/dt = \omega \left[ (d/d\theta)(d\theta/dt) = \omega^2 (d^2/d\theta) = -\omega^2 \lambda R \sin \theta (1 + \lambda/2 \cos \theta) \right]$$ \hspace{1cm} (16)

The extreme values of the roller relative velocity are shown below:

$$\Delta v_{\text{max}} = 2\omega (R_C - R_R)$$ \hspace{1cm} (17)

$$\Delta v_{\text{min}} = 0$$ \hspace{1cm} (18)
Effect of the $R_C / R_R$ ratio on the values of the relative velocity between the roller and the cylinder at different vane angular positions is shown in Fig. 5. It is shown that rather smaller ratio give us lower relative velocity between rotating in one direction roller and cylinder.

![Fig. 5. Effect of the radii ratio on the relative velocity.](image)

### 4. STRUCTURE OF THE COMPRESSOR

A prototype compact rotary compressor has been developed for use with carbon dioxide as natural working fluid [2]. The rotary hermetic compressor shown in Fig. 6 comprises a motor, the cylinder block cavity and vane machined in the core of said motor-rotor so, that it formed integral rotor-cylinder -vane part (rotor-cylinder, see Fig. 7), the roller to rotate with the integral rotor-cylinder about an axis eccentric to the crankshaft axis of the rotor-cylinder. The vane projects inwardly from the circular wall in substantially radial line and opposite side edges of the vane engage the opposed inwardly facing surfaces of the heads, so that the vane is clamped without clearance between the inner sides of said heads which are immovable fixed to the rotor-cylinder in order to form an enclosed casing in which said vane and the roller are mounted. The impelling vane has been fixed at the free end by pin, which connects heads and said vane together. The vane can be considered as cantilever rectangular beam loaded transversely by the pressure differential forces across the vane within the cylinder. Reaction forces and vertical shear for the beam supported at both ends are twice less than that for single end support.

![Fig. 6. Longitudinal cross-sectional view of rotary oscillating-roller compressor](image)
The vane extends into transverse cylindrical pocket open through periphery of the roller in which are rotatably fitted bushing, preferably formed by segmental cylindrical blocks, the flat surfaces of which are machined to slidingly engage upon the opposed surfaces of the vane with only operating clearance.

![Diagram of the compressor](image)

**Fig.7. The cross-sectional view of the compressor in vicinity of the rotor-cylinder.**

### 5. LEAKAGE AND FRICTIONAL LOSSES

Rolling piston type compressors operate with clearances between moving parts, which induce internal leakage flows and associated leakage losses that affected the delivered flow of refrigerant, reducing the cooling capacity, volumetric efficiency and, consequently, increase the power consumed by the compressor. Analytical and experimental studies of rolling piston compressors indicate that dominant leakages occur at the radial clearance between the roller O.D. and cylinder I.D. and through the clearances between vane sides and facing surfaces of the cylinder [3]. The leakages through the roller radial, roller axial and vane axial clearances have great impact on the performance of the compressor [4]. Due to the fact that in the developed rotary compressor the rotor-cylinder, heads, vane are secured together and rotate as one solid body there is no movement of the vane in the slot of the rotor-cylinder and relative movement upon the inward facing surfaces of the heads as in contemporary rotary or swing type compressors. Analysis of leakage and frictional losses shows that the losses have been eliminated in oscillating-roller type compressor at the following locations:

- Vane tip-roller O.D. There is no radial gap between the vane and the roller.
- Vane edges - facing surfaces of the cylinder heads. The vane is stationary. There are no operating clearances between the vane edges and the cylinder heads facing surfaces.
- Vane sides- slot sides in the cylinder block. Stationary vane is integral part of the rotor-cylinder. There is no high – low side axial gaps and slot-slide boundary friction usual in contemporary rotary compressors.

Preliminary general evaluation indicates (see Table ) that mechanical relative losses of the developed compressor can be up to 20% lower than the losses of a conventional compressor and expected reduction of the leakage is around 3%.
Table. Relative Compressor Losses

<table>
<thead>
<tr>
<th>Point (line) of contact</th>
<th>Frictional Losses, %</th>
<th>Leakage flow fraction losses*</th>
<th>Frictional Losses, %</th>
<th>Leakage flow fraction losses*</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roller O.D- Vane tip</td>
<td>24.3</td>
<td>Eliminated</td>
<td>0.0025</td>
<td>Eliminated</td>
</tr>
<tr>
<td>Roller O.D-Cylinder I.D.</td>
<td>1.0</td>
<td>1.0</td>
<td>0.0309</td>
<td>0.0309</td>
</tr>
<tr>
<td>Roller I.D-Eccentric</td>
<td>18.7</td>
<td>18.7</td>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>Roller ends-Cylinder heads</td>
<td>2.7</td>
<td>2.7</td>
<td>0.0294</td>
<td>0.0294</td>
</tr>
<tr>
<td>Vane ends- Cylinder heads</td>
<td>N/A</td>
<td>Eliminated</td>
<td>0.0005</td>
<td>Eliminated</td>
</tr>
<tr>
<td>Vane sides-Slot sides</td>
<td>28.1</td>
<td>Eliminated</td>
<td>0.0310</td>
<td>Eliminated</td>
</tr>
<tr>
<td>Crankshaft- Bearings.</td>
<td>25.2</td>
<td>26.13</td>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>Vane sides-bushing block</td>
<td>-</td>
<td>25.67</td>
<td>---</td>
<td>0.0310</td>
</tr>
<tr>
<td>Bushing –pocket wall</td>
<td>-</td>
<td>4.82</td>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>Sub-Total</td>
<td>-</td>
<td>---</td>
<td>0.0943</td>
<td>0.0913</td>
</tr>
<tr>
<td>Total Loss, %</td>
<td>100</td>
<td>79.02</td>
<td>100</td>
<td>96.8</td>
</tr>
</tbody>
</table>

* Leakage flow fraction relative to delivered suction volume

Developed compact hermetic oscillating-roller rotary compressor designed for the capacity range 5300 W – 7032W (18000Btu/h – 24000Btu/h) has the following dimensions: height - 132mm(5.2in) and O.D equal 140mm. The compact oscillating roller rotary compressor does not required pressure on the back of the vane for proper operation, so it can be designed as low side or high side compressor.

6. CONCLUSIONS

The leakage and frictional losses in new oscillating rotary compressor are relatively lower than in contemporary rotary compressors due to elimination of the vane radial movement and synchronous rotation in one direction of the integral rotor-cylinder- vane and roller with small relative velocity.

A combined design in which the armature of the electric motor forms part of the integral cylinder block and vane reduce number of the parts and provide comparatively compact, small, well balanced, reliable and efficient rotary compressor.

REFERENCES

3. Costa, C., et al., ”Consideration About the Leakage Through the Minimal Clearance in a Rolling Piston Compressor”. 1990 ICECP, pp. 853-862
Research on tip profile of vane for rotary vane compressor

Cai Hong, Li Liansheng, Guo Bei, Shu Pengcheng
National Engineering Research Center for Fluid Machinery and Compressors, Xi'an Jiaotong University, P R China

ABSTRACT

The friction and abrasion of the vane tip has been one of the main factors for the design of rotary vane compressor. Besides the contact force between vane tip and cylinder internal wall, the moving range and regulation of the contact point have big effect on the friction and abrasion of vane tip. On the basis of geometric analysis the position and moving range of contact point is derived in this paper, and an algorithm for the perfect profile of vane tip is proposed. Given an instance, the relative calculation and analysis are done, and results show that improper tip profile can cause “cuspidal point slippage” in the movement of vane.

Keywords: vane; contact point; tip profile; rotary compressor

NOMENCLATURE

B_v vane thickness, m
r_t vane tip radius, m
F_d gas force on vane tip, N
P_q pressure of prior chamber, Pa
P_h pressure of posterior chamber, Pa
H axial length of vane, m
x_A horizontal coordinate of vane tip center, m
y_A vertical coordinate of vane tip center, m
K slope of line connecting vane tip center and intersecting point of vane centerline and vane tip centerline
l_1 length of line connecting vane tip center and intersecting point of vane centerline and vane tip centerline, m
l_2 length of line connecting rotor center and vane tip center, m
r_v radius of perfect profile of vane tip, m
l radial length of vane, m
r rotor radius, m
GREEK LETTERS

σ  half of central angle of vane tip arc, °
θ_i  inclined angle of vane slot, °
θ_t  inclined angle of vane tip, °
α  slipping angle of contact point, °
Φ  rotation angle in geometrical method, °
γ  rotation angle in numerical method, °
ρ  curve equation of inner cylinder wall in polar coordinate, m
ψ  included angle of radials vector and common normal at the contact point, °
ξ  included angle of radials vector and line connecting rotor center and vane tip center, °
r_c  the largest radius of vane tip in each rotation angle of vane, m
θ_c  inclined angle corresponding to r_c, °

1. INTRODUCTION

Rotary vane compressors have been widely used in the automobile conditioners because of their advantages such as compact structure, high efficiency, smoothly operation and easy machining. Figure 1 shows a schematic structure of this type of compressor.

There are some chambers, called basic element, separated by the rotor, vane, and cylinder internal wall. Because the vane tip must contact with the cylinder internal wall in the compression process at any working condition including varying rotation speed, the friction loss resulting from the contact force always takes great part of compressor power consumption (1). Edwards T C, Donald A T (2) and Peterson C R, Gahon W A (3) analyzed the mechanical friction in rotary vane machines and focused on deriving the friction coefficient between vane and cylinder wall. Kamiya H, Shimizu T (4) studied the frictional loss on vanes in through-vane compressor and suggested increasing the radius of vane tip is helpful to fluid lubrication according to experimental results. Guo Bei (5) studied the thermodynamic and kinetic performance of rotary vane compressor for automotive conditioner, described the movement regulation of the contact point, and considered having a perfect profile for vane tip that fits
better movement and the lubrication between vane and cylinder wall. However, the numerical method is complex and the perfect profile was not mentioned.

This paper presents a geometric analysis for the position and moving range of contact point, proposes an algorithm for the perfect profile of vane tip.

2. MOVEMENT REGULATION OF CONTACT POINT

The shape of vane tip, an unsymmetrical arc illustrated in Figure 2, is determined by three parameters: vane thickness \( B_v \), radius \( \theta_t \) and inclined angle \( r_t \). Then half of central angle of the arc \( \sigma \) can be calculated by

\[
\sigma = \arcsin \frac{B_v}{2r_t \cos \theta_t}.
\]  

\( \sigma \) is the sliding angle (get positive sign when contact point is on the right of center line of vane tip arc). If \(|\alpha| \leq \sigma\), during the operation of rotary vane compressor, the vane will roll and slide on the cylinder wall. If \(|\alpha| > \sigma\), the contact point between vane and cylinder will hold in the end point of vane tip, the phenomenon called “cuspidal point slippage” occurs, which can increase the friction and abrasion, scrape the oil film on the wall of cylinder, destroy the condition of fluid lubrication. Hence, it is essential for calculating moving regulation of the contact point to avoid the phenomenon of “cuspidal point slippage”.

Moving regulation of the contact point even effects the forces acting on vane tip (see. Figure 2), the gas force \( F_d \), due to pressure of gas in basic element, may be derived by

\[
F_d = (P_q + P_h)Hr_t \sin \sigma + (P_q - P_h)Hr_t \sin \alpha.
\]
The position of contact point between vane and cylinder and its vary regulation are important for the research in this paper, and two calculating methods are discussed as follow.

2.1 Numerical Method

According to the coordinate system used in Figure 3, a numerical method of calculating the position of contact point can be done by setting angle $\gamma$ as referenced rotation angle of vane \(^{(5)}\).

![Figure 3. Contact point of vane tip](image)

Coordinate of point A can be resolved by

\[
\begin{align*}
\begin{cases}
y_A - r \sin \gamma &= \tan(\gamma - \theta_t)[x_A - r \cos \gamma + \frac{r \sin \theta_t}{\sin(\gamma - \phi)}] \\
(x_A - \rho \cos \phi)^2 + (y_A - \rho \sin \phi)^2 &= r_i^2
\end{cases}
\end{align*}
\]

(3)

As two curved faces in related motion touch, there is a common normal line passing the contact point. So the contact point between vane and cylinder (point B) must be on connection line of curvature center of vane tip (point A) and the curvature center of cylinder wall, and the slope of line AB (letter K) can be derived by

\[
K = \frac{y_A - \rho \cos \phi}{x_A - \rho \sin \phi} = \frac{\frac{d\rho}{d\gamma} \sin \phi + \rho \cos \phi}{\frac{d\rho}{d\gamma} \cos \phi - \rho \sin \phi},
\]

(4)

where $x_A$ and $y_A$ refer to coordinates of point A.

The values of $\Phi$ and K can be obtained by resolving equations (3) and (4), then, the value of $\alpha$ is given:
\[
\alpha = \arctan \frac{K - \tan(\gamma - \theta_i)}{1 + K \tan(\gamma - \theta_i)} - \theta_i \tag{5}
\]

As equation (3) and (4) are nonlinear, the solution process will take a lot of time.

### 2.2 Geometrical Method

Referring back to Figure 3, choose \( \Phi \) as vane rotation angle, \( \alpha \) and \( \gamma \) can be described by

\[
\alpha = \arcsin \frac{r \sin \theta_i - l_1 \sin \theta}{l_2} + \psi + \xi - \theta_i , \tag{6}
\]

\[
\gamma = \phi + \xi + \arcsin \frac{r \sin \theta_i - l_1 \sin \theta}{l_2} , \tag{7}
\]

where \( \psi = \arctan \frac{d\rho}{\rho d\phi} \), \( l_1 = \sqrt{r_i^2 - \frac{B_v^2}{4 \cos^2 \theta_i}} \), \( l_2 = \sqrt{r_i^2 + \rho^2 - 2r_i \rho \cos \psi} \) \( \xi = \arcsin \frac{r_i \sin \psi}{l_2} \).

### 3. PERFECT PROFILE OF VANE TIP

There exist a profile of vane tip on which the moving range of contact points covers the whole arc of vane tip. This profile called “perfect one” can avoid “cuspidal point slippage” and make the arc radius get the maximum value.

As profile of vane tip is perfect, \( \sigma \) and \( \alpha \) meet

\[
\alpha_{\text{max}} = -\alpha_{\text{min}} = \sigma \tag{12}
\]

However parameters of perfect profile, \( r_v \) and \( \theta_v \), can be obtained by resolving equations (6), (7) and (12), the two uncertain parameters make iteration process time-consuming. Another algorithm is proposed as follow:

As shown in Figure 4, let the radius of vane tip arc increase step by step from \( B_v/2 \), contact point between vane and cylinder will move from inner point to end point of arc, at last arc radius gets the largest value \( r_c \), the corresponding inclined angle of arc is \( \theta_c \). Two parameters, \( r_c \) and \( \theta_c \), meet

\[
\sin \theta_c \sqrt{\rho^2 - \frac{B_v^2}{4 \cos^2 \theta_c}} = r \sin \theta_i , \tag{13}
\]

which can be transformed to
\[ \tan^4 \theta_c - \left[ \frac{4(\rho^2 - r^2 \sin \theta_v)}{B_v^2} - 1 \right] \tan^2 \theta_c + \frac{4r^2 \sin^2 \theta_v}{B_v^2} = 0. \]  \hspace{1cm} (14)

Solution of equation (14) is

\[ \theta_c = \arctan \sqrt{\frac{m - \sqrt{m^2 - 4n}}{2}}, \]  \hspace{1cm} (15)

where letter m and n refer to

\[ m = \frac{4(\rho^2 - r^2 \sin \theta_v)}{B_v^2} - 1, \]  \hspace{1cm} (16)

\[ n = \frac{4r^2 \sin^2 \theta_v}{B_v^2}. \]  \hspace{1cm} (17)

Therefore, we have

\[ r_c = \frac{B_v}{2 \cos \theta_c \sin(\psi + \xi)}, \]  \hspace{1cm} (18)

where \[ \xi = \arcsin \frac{B_v}{2 \rho \sin \theta_c}. \]  \hspace{1cm} (19)

Using numerical method we can get the rotation angle (\( \Phi \)) at which \( r_c \) gets the minimum one, then \( r_v \) and \( \theta_v \) can be calculated by equations (13) ~ (19).
Selecting coordinate system shown in Figure 4, coordinate of perfect profile center (point A) is obtained by

\[
\begin{align*}
\begin{cases}
  x_A &= \frac{B_v}{2} \sin \theta_v \sqrt{r_v^2 - \frac{B_v^2}{4 \cos^2 \theta_v}} \\
  y_A &= l - r_v
\end{cases}
\end{align*}
\]

(20)

\[\text{Figure 5. Coordinate of perfect profile center}\]

4. EXAMPLE OF CALCULATION

Here give an instance for a rotary compressor whose structure parameters are: \(r = 25\) mm, \(\rho = (28 + 9.5 \sin \Phi)\) mm, \(\theta_l = 15^\circ\) and \(B_v = 3.6\) mm. The parameters of perfect profile of vane tip:

\[r_t = 5.519\text{mm}, \ \theta_t = 13.053^\circ,\]

are obtained from equation (13)–(19), and the coordinate of perfect profile center:

\[(0.6255, 10.4811),\]

is calculated by equation (20).

For vane whose parameters of tip profile are \(r_t = 5.519\) mm and \(\theta_t = 13.053^\circ\), \(\sigma\) is calculated by equation (1) and \(\sigma = 19.561^\circ\). The relation of \(\alpha\) with \(\Phi\), determined by equation (5), is illustrated in Figure 5(a) and it shows that \(\alpha\) and \(\sigma\) meets equation (12).

If the parameters of tip profile change to \(r_t = 7.139\) mm and \(\theta_t = 14.395^\circ\), \(\sigma\) gets the value of \(15.089^\circ\) and the relation of \(\alpha\) and \(\Phi\) is shown in Figure 5(b). From figure 5(b), it can be seen that vane at one angular position, which is in the range of \((21^\circ, 56^\circ)\) and \((109^\circ, 161^\circ)\), there exists a phenomenon of “cuspidal point slippage”.

221
5. CONCLUSIONS

Because the numerical method for getting the movement regulation of contact point between vane and cylinder is time-consuming in calculating process, a geometrical method is proposed in this paper, and the regulation can be used to check the vane tip profile and analyze the force acting on the vane. At the same time, a perfect profile, which fits to build fluid lubrication, is more easily derived. Calculating results obtained from the example show that improper tip profile can cause an adverse phenomenon of “cuspidal point slippage” in the movement of vane.

REFERENCES


(2) Edwards T C, Donald A T. Analysis of Mechanical Friction in Rotary Vane Machines. ICECP, USA. 1972: 250~262.


TURBO COMPRESSORS
Low specific speed turbocompressors

Mechanical Engineering Department, Imperial College London, UK

ABSTRACT

The present contribution presents further results from a new class of turbocompressor developed at Imperial College based on recent turbomachine innovations, which is suitable for efficient operation at low specific speed.

Positive displacement machines are typically used in low specific speed applications (high pressure ratio, low shaft speed and low mass flow). The innovative turbocompressor is a fraction of the size, weight, and production cost of a positive displacement machine, delivering oil-free compressed gases reliably. The design may consist of multiple stages similar to a standard industrial turbocompressor, and more stages may be added to achieve the required pressure ratio. For a single stage, detailed computational fluid dynamic analysis has been validated against experimental results. A multi-stage demonstrator has been built to investigate stage interactions with preliminary results reported here.

The proposed innovation may be employed in applications as diverse as fuel gas compression for microturbines (50-750 kW), gas pressure boosting for medium sized gas turbines (1-5 MW), automotive super-charging, oil-free air compression, and air compression for fuel cells. Fuel cell prime mover units are gaining momentum as an alternative to IC engines for automotive use, with prototypes based on Proton Exchange Membrane (PEM) in demonstration trials. PEM uses hydrogen fuel and compressed air reaction to produce electrical power. Hydrogen is delivered to the fuel cell under pressure from a tank. Atmospheric air, after on board compression, is supplied to the fuel cell, but regulated to set fuel cell power. Air supply regulators are presently based on positive displacement systems and are large, inefficient, and noisy. A multi-stage compressor design based on the present low specific speed turbomachinery innovation was realised with test results showing characteristics superior to the currently used positive displacement machines.
1. INTRODUCTION

The worldwide explosion in mobility in the past two decades has seen an alarming increase in the transportation sector. The growing car production figures (to 40 million vehicles globally, in 1998) are exacerbated by growth in the number of miles per year driven by motorists [1]. Given the life span of an average car is 160,000 kilometres, during which time it consumes 13,700 litres of fuel (at 8.55 l/100km) and 230 litres of lubricating oil and creates 35 tonnes of carbon gases and particulates. The impact of transport pollutants emission on the environment has therefore been catastrophic.

The result of growing concerns on environmental issues is a demand for cleaner and more energy efficient vehicles, whilst maintaining vehicle performance, comfort, and safety. The most significant problem, identified as early as 1970, is CO₂ from private passenger vehicles. The present emission target is a 35% reduction in CO₂ by 2010 for new passenger vehicles (down to 120 g/km) [2]. These emissions requirements make the use of internal combustion engines difficult, although recent years have seen substantial improvements in engine technology. Recent advances in battery technology have resulted in several hybrid vehicles on the road. These use batteries in conjunction with an internal combustion engine in series and parallel arrangements. The target solution is the zero emissions vehicle.

The CO₂ problem augmented by battery limitations as well as advances in fuel cell technology development has lead to electrically driven Fuel Cell Vehicles (FCV). FCV uses a low temperature (80 degrees C) Proton Exchange membrane (PEM) fuel cell with hydrogen fuel as a replacement for the internal combustion engine [3]. The prime mover system includes five sub-systems; fuel cell stack, hydrogen fuel, air supply, water and heat management. The present contribution concentrates on the air supply sub-system by describing new compressor technology to service this requirement.

2. FUEL CELL LAYOUT AND AIR SUPPLY SUBSYSTEM

A typical PEM fuel cell system with fuel and air supply sub-systems is illustrated in Figure 1. It is noted that the water and heat management sub-systems are not relevant to the present contribution and hence not shown in Figure 1. In this configuration, the hydrogen fuel is provided from a pressurised tank, although some systems reform fuel on board from methanol or gasoline to a rich hydrogen gas stream. The latter clearly overcomes the hydrogen supply and storage problem but incurs penalties of size, weight, and cost.

The fuel cell also requires an air supply system. Unlike a naturally aspirated internal combustion engine, a fuel cell requires a regulated pressurised air supply sub-system. The supply pressure is fuel cell dependent, but a requirement for a pressure ratio of the order of 2 to 4 is typical. The air supply regulation is an important feature as it determines fuel cell power output. A wide range in flow rate is required (for an automotive fuel cell 10 to 100 g/sec) to cater for fluctuating demands on the fuel cell. Therefore the air subsystem also requires a quick dynamic response [4].

The regulated pressurised air required might be realised by an air compressor with speed control that would enable variable supply for a wide range of power output. To recover some of the energy from the pressurised exhaust air, an expander may be incorporated to assist the motor as shown in Figure 1.
The requirements for the air supply sub system are; compact size and weight, oil free delivery, low noise, low maintenance, low cost and high efficiency operation. The requirements of pressure ratio and mass flow rate suggest the use of a positive displacement compressor and to date this application has been serviced by positive displacement machines, usually of the screw type configuration [5]. Recently, very high speed (> 100,000 rpm) turbocharger type compressor systems integrated with a high-speed motor have also been developed [6]. Detailed comparisons of the two systems have been made [7] showing high-pressure fuel cell systems being smaller but less efficient than low-pressure versions. The selection of an appropriate compressor technology best suited for this application may be further investigated by considering specific speed values.

3. LOW SPECIFIC SPEED

The suitability of a particular compressor technology for a given application can be visualised by plotting specific speed versus adiabatic efficiency for a given Reynolds number (Figure 2). Non-dimensional specific speed, $N_s$ is given by:

$$N_s = \frac{\omega \frac{Q^2}{\Delta h}}{\Delta h}$$

Where $Q$ is the volumetric flow rate at inlet, $\omega$ is the rotational speed and $\Delta h$ is the specific isentropic enthalpy rise. A machine of a given rotational speed with a high pressure rise and a low volumetric flow has a low specific speed.
Low specific speed machines are required for high pressure, low volume flow rate, and low shaft speed of the driving motor. Positive displacement machines include screw, scroll, reciprocating and rotary vane compressors, although these have a low power density and high mechanical complexity when compared to turbomachines.

The present work describes recent advances in turbomachine design for moderate pressure ratios, to replace positive displacement compressors. The most common types of positive displacement compressors are shown in Figure 3.

Screw Compressors that have been employed in FCV applications work at the higher end of the specific speed range for positive displacement machines. They are more reliable and compact than other positive displacement machines and operate with reasonable efficiency. Oil ingesting designs require filtration. Oil-free designs require precision timing gears and very high manufacturing tolerances on a complex design. Screw compressors have been shown to have the highest power density (around 50 W/kg) of the positive displacement machines. Although this is still not compact for vehicular applications. A step change is required in terms of size and weight of positive displacement machines to make them suitable for this application. Other types of positive displacement compressors are described [9].
4. TURBOCOMPRESSORS

Low specific speed turbomachines of a conventional design are inefficient. This is because parasitic losses do not vary greatly with flow rate, therefore at low flow rate parasitic losses represent a greater proportion of the machine’s power.

Table 1, from [9], describes the main differences between a normal radial compressor, and one of low specific speed, both operating at the same diameter and rotational speed (‘Windage’ refers to the drag on the plane disc that forms the back-face of the compressor rotor):

<table>
<thead>
<tr>
<th>Design Aspect</th>
<th>Difference in Low Specific Speed Compressor</th>
<th>Consequence</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass Flow</td>
<td>Lower mass flow due to lower specific speed at the same pressure ratio and shaft speed.</td>
<td>Low specific speed rotor has a lower power.</td>
</tr>
<tr>
<td>Blade Height</td>
<td>Shorter blades, due to lower mass flow.</td>
<td>Frictional losses in rotor passages are higher.</td>
</tr>
<tr>
<td>Tip Clearance</td>
<td>Similar tip clearance, however, tip clearance is proportionally higher for low specific speed rotor. (Some low specific speed rotors are shrouded for this reason.)</td>
<td>Tip leakage proportionally higher, which effects efficiency to a greater extent.</td>
</tr>
<tr>
<td>Diameter</td>
<td>Same diameter, therefore equal power loss due to Windage</td>
<td>Due to reduced power of low specific speed rotor, windage has a greater effect on efficiency.</td>
</tr>
<tr>
<td>Seal Leakage</td>
<td>Inter-stage seal leakage is equal for both rotors.</td>
<td>As the flow is lower for low specific speed rotors, a given loss has a greater effect of efficiency.</td>
</tr>
</tbody>
</table>

Turbomachinery can be scaled down to reach low flow rates and maintain the optimum specific speed (as in turbochargers), but the tip speed must be maintained and this necessitates high shaft speeds. Automotive turbochargers are only feasible because they have access to the oil system of the engine. Recently, rolling element bearings have begun to replace hydrodynamic bearings. However there is a maximum linear speed for the bearings and this limits their size. This puts a constraint on the maximum shaft diameter, which affects the shaft natural frequencies. Grease packed bearings that do not require an oil system require even lower diameter shafts, which lower the natural frequencies and make it more likely that they will be excited. Other options for a bearing system include air bearings, magnetic bearings and hydrodynamic bearings. Each of these systems have a low stiffness and force the designer to tolerate a large tip clearance, seriously impacting efficiency [10].

In a fuel cell compression system there is an additional problem, there is not always enough power at the turbine to run the compressor. Unlike a conventional engine, the fuel cell requires the compressor to work at all times. Therefore an electrical motor that can transfer power to the compressor shaft is required. Use of high speed motor technology has clearly attracted attention particularly since recent advances in radial flux permanent magnet motor designs has led to compactness and reliability. However, the requirement for close radial clearances, advanced materials, and costly and complicated motor drives has been further complicated by cooling and structural stability problems. Shaft support is also difficult due to out of balance loads and high rotational speeds.

One solution based on turbocharger type turbomachinery [6] is being developed for a 50 kW PEM and is said to be about 10 litres (300 by 200 mm). The requirements as set out by US
Department of Energy (DOE) are a maximum pressure ratio of 3.2 at 76 g/sec with a 4 litre system volume. Wiartalla[4] suggests that the maximum overall efficiency is at a stack pressure of between 2.5 and 3 bar. The exact pressure required is likely to change as a result of future fuel cell developments. The developments reported by [6] are for a turbocompressor with a mixed flow radial machine (stated efficiency 70%) driven by a radial permanent magnet motor that is assisted by an expansion turbine based on the variable nozzle VNT™ (stated efficiency 80%) driven by fuel cell exhaust gases. The motor inverter is supplied by 300V dc. The integrated single shaft is supported by self-sustaining compliant foil air bearings at 108,000 rpm[7]. The performance achieved to date, that appears acceptable to DOE is 76 g/sec at a pressure ratio of 2.5 and system volume of 15 litres [6] (8.5 for the inverter and 6.5 for the compressor system). The main difficulties include:

- High speed low diameter turbomachinery with significant tip clearance losses.
- High speed motor design and construction is difficult. Cooling and end windings pose particular design problem.
- Motor drive inverter with fast switching with 300V dc is a challenge in technology and cost.
- Bearing design poses high current pulsation on the shaft rotor inducing additional heating problems for the bearings in addition to problems with mobile operation.
- System cost escalates with the high speed operation requiring costly inverter, bearing, motor, and cooling technologies.
- Smaller fuel cells would require air systems operating at higher speeds and hence more difficulty.

5. INNOVATIONS IN LSS TURBOMACHINE

To overcome these limitations, research into an efficient low specific speed turbocompressor has been ongoing at Imperial College London, partially funded by a European Union project [11]. A novel solution has been found, which can extend the specific speed range of radial compressors by radically changing the geometry, [12] and [13]. A Computational Fluid Dynamics (CFD) simulation using CFX-TASCflow [14] has been used to predict and optimise performance.

A number of single stage demonstrators have been designed, manufactured and commissioned. An air turbine drove a first demonstrator designed to operate at 60,000 rpm [9]. The pressure ratio and mass flow results indicated that the design concept was feasible. A second demonstrator with further design optimisations has also been evaluated experimentally on a specially designed test bed that operates at 20,000 rpm. Test results are detailed in Table 2.

Table 2: Test Results of Low Specific Speed prototypes

<table>
<thead>
<tr>
<th>Prototype</th>
<th>Number</th>
<th>Speed, rpm</th>
<th>PR</th>
<th>Mass flow, g/sec</th>
<th>Specific Speed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single stage</td>
<td>S1-03</td>
<td>60,000</td>
<td>1.43</td>
<td>10</td>
<td>0.19</td>
</tr>
<tr>
<td>Single stage</td>
<td>S1-05</td>
<td>20,000</td>
<td>1.52</td>
<td>75</td>
<td>0.19</td>
</tr>
<tr>
<td>Multi-stage (two modules)</td>
<td>M1-04</td>
<td>60,000</td>
<td>1.87</td>
<td>8</td>
<td>0.14</td>
</tr>
</tbody>
</table>

230
6. MULTI-STAGE ARRANGEMENT

It has been shown that leakage and windage losses can be critically important to a low specific speed machine. In order to assess the efficiency of a multi-stage machine the interactions of loss mechanisms must be considered.

Figure 4 shows a four-stage design of the first multistage demonstrator, which has been developed and driven by a low speed motor and step-up gearbox, operating at 60krpm. Abradable polymer seals allow for tight seal clearances. Shrouded impellors eliminate tip clearances. Compact ducting is permitted by low speed flow. This machine has been designed to assess the inter-stage characteristics of the low specific speed turbocompressor, as well as testing four different rotors of varying specific speeds. Air enters the rig through axial holes in the housing. Each stage consists of a rotor and a stator. After the second stage, the air is passed to an external cooler. The air is finally expelled through a tapping in the housing. The shaft is supported by angular contact bearings that can be either grease packed or oil-fed. The air blown seals of the bearings do not affect the measurements. The rig is driven by a square drive spool piece, which is connected to the step up gearbox and low speed motor. To measure the work input and calculate efficiency it is normal to measure the temperature increase of the gas. However this is not possible in this case as the rig is effective in cooling the gas due to the low flow rate and high wetted area. The test rig has been characterised by using two test
methods, a four-stage arrangement and a two-stage arrangement. The inter-cooler has only been utilised in the four-stage test method. Both test methods have been fully characterised with various inlet conditions. The data from the two-stage arrangement is relevant to the fuel cell system and produced a pressure ratio of 1.87 at 8g/s throughput (see Table 2).

7. DESIGN STUDY – FUEL CELL COMPRESSOR

Test results from the low specific speed turbomachines at two nominal speeds and flow rates have demonstrated their functionality (Table 2). Moreover a multi stage machine has also been tested to demonstrate system modularity and successful implementation of inter-stage issues such as sealing. Based on these results, scaling laws can be used to interpolate the performance of any required duty such as the one required for the fuel cell air system.

In the experimental work, it was found that the compressor obeyed the well-known turbo machinery law that the pressure ratio varied with the square of the running speed for a given throttling valve setting, and prior to shockwave losses. Data from CFD simulations has shown that a 20-30% increase in the operating speeds would be sustainable, achieving a pressure ratio of 1.68 for S1-05 (Table 2). Two-stage and three stage machines could therefore develop pressure ratios of up to 2.8 and 4.7 respectively, see Table 3. The design of the compressor rotor allows for a high-pressure ratio at a low tip speed and as a consequence, there is no stress problem with the operating speed of 26,000 rpm for S1-05.

For fuel cell applications size is a key concern. A very simple design study is shown here to give an impression of size. The rise in pressure (1-PR) for a stage varies with the square of the rotor tip speed which can be used with the shaft speed to scale the diameter from the baseline case. The axial length of a stage is relatively constant at 50mm including ducting and sealing. In addition to this, 1.1 litres has been allowed for a pair of bearings and seals.

<table>
<thead>
<tr>
<th>Table 3: Test Results of Fuel Cell Design Studies</th>
</tr>
</thead>
<tbody>
<tr>
<td>Speed, rpm</td>
</tr>
<tr>
<td>Baseline single stage</td>
</tr>
<tr>
<td>Design 1</td>
</tr>
<tr>
<td>Design 2</td>
</tr>
<tr>
<td>Design 3</td>
</tr>
</tbody>
</table>

This is similar to the size predicted by Gee [6], but larger than the guideline suggested by the DOE. This shows that conceptual designs based on the low specific speed innovation is most suitable for the fuel cell application in terms of the performance characteristics. It is noted that developments in [6] predict an integrated turbine and compressor with a pressure ratio of 2.5 and size of 6.5 litres (excludes inverter) whilst operating at 108,000 rpm.

The other turbomachinery issue is turbine design to match the low specific speed compressor in a complete fuel cell air system. Turbines are well known to be more amenable to high blade loading and less susceptible to separations due to the favourable pressure gradient. Using the present low specific speed compressor design innovation [12 and 13] for turbine design is
extremely attractive. A low specific speed turbine could be used to drive the low specific speed compressor, or even the twin-screw compressor [5] making either technology a more realistic competitor.

The low specific speed compressor has achieved exceptional characteristics in terms of mass flow and pressure ratio in laboratory tests. It is expected that performance and pressure ratio will increase significantly as a results of further developments at hand. A detailed design study focusing on size reduction is likely to significantly reduce the overall volume.

APPLICABILITY TO FUEL CELLS

Currently the only technology that is in production for the supply air system is the twin screw from Opcon [5], that does not have a turbine, and is expensive and heavy.

The compressor described by Gee [6] is running at a specific speed of 0.55 (calculated from a pressure ratio of 2.5, 108,000rpm, and 76g/s). Making an efficient turbocharger at this specific speed with an integrated motor will be an impressive achievement if it can be realised, and even more so if it is possible at the target system cost of $300-600 for 50 to 150 kW fuel cells by 2010 [6]. However, this specific speed is close to the limit of conventional turbo machinery so there will be a problem for smaller fuel cell vehicles, particularly small urban vehicles and scooters [15], [16].

The low specific speed compressor is also currently unproven as a fuel cell compressor. No work has so far been undertaken in packaging the compressor to achieve the stringent size requirements of the US dept of energy, unlike the project of Gee [6] where $3 million has been spent to take existing, well understood turbomachine components and integrate them in a package small enough for an automotive fuel cell. However, the low specific speed compressor has unique features which make it cheaper, easier to implement, and more suited to even lower flow rates.

8. CONCLUSION

The air supply system for automotive fuel cells is a complex and demanding problem. Positive displacement machines have failings in a number of areas, particularly size, reliability and oil ingestion. Turbocompressor technology also suffers due to complexity and cost issues. Initial studies of a new class of low specific speed turbocompressor have shown promising performance.

Laboratory testing of the two designs at 20,000 and 60,000rpm is reported showing pressure ratios and mass flow rates appropriate for FCV applications. This type of machine could extend the benefits of turbomachinery, namely compactness, simplicity, and reliability, to applications where positive displacement machines have traditionally been used.
REFERENCES

Early detection of a compressor impeller crack

Erik van Deursen  
Hoek Loos Linde, Netherlands

Gert Hoefakker  
Brüel & Kjær Vibro, Netherlands

Peter Surland, Mike Hastings  
Brüel & Kjær Vibro, Denmark

ABSTRACT

Rotor vibration and phase condition monitoring limits were exceeded for a 25 MW centrifugal compressor. Although the vibration levels were still 60% under the safety system alarm limits, the Rotating Equipment Engineer decided to shut down the machine for inspection. A 25 cm crack was found on the impeller. If the condition monitoring system had not been monitoring this machine, it is possible the safety system could not have detected the crack before failure. This could have resulted in destroying both the rotor and casing. Later investigation revealed that the shaft crack could have been detected even earlier if the automatic machine diagnosis program that runs on the condition monitoring system was used.

1 INTRODUCTION

The 4-stage, 25 MW centrifugal compressor is used in an air separation process. The 5-year old compressor is a critical machine to the process, and has no spares. The machine has been monitored since it was commissioned.

2 MONITORING STRATEGY

A combined condition monitoring and diagnosis system and safety monitoring system is used at the air-separation plant for monitoring three machines. The condition-monitoring portion of the system is used in a predictive maintenance strategy for detecting faults at an early stage of development. Both vibration and process parameters (many of these are imported from the distributed control system) are automatically monitored at periodic intervals. The monitoring system supplier also provides an “on-request” diagnostic service to the customer for analyzing faults.

The condition monitoring installation on the compressor includes X-Y displacement sensors on the inboard and outboard bearings, as shown in Fig. 1.
Measurements at each sensor point include:

- DC
- Bandpass (10-1000 Hz, RMS peak-peak)
- 1x, 2x vector measurements (RMS peak-peak magnitude and phase), ½x magnitude (RMS peak-peak)
- 6% Constant Percentage Bandwidth (10.3 to 1030 Hz, RMS peak-peak), 23% Constant Percentage Bandwidth (2 to 1000 Hz, RMS),

3 FAULT DETECTION

In January 2004 several condition-monitoring alarms were abruptly generated, primarily on the inboard X-Y displacement sensors on the compressor rotor:

- CPB6% exceeded its 54.38 µm alert limits at running frequency (61.30 Hz)
- 1x vector radial vibration level exceeded the 60 µm alert limits (25 µm is normal)
• 1x phase exceeded the 90° danger limits

The alert-alarm levels were relatively far from the safety monitoring system trip level of 130 µm overall vibration level (phase is not monitored for trip), but the Rotating Equipment Engineer was concerned anyway. This was partly because the 1x phase measurements have been 65° greater than normal for over a year (alert level is 85°), while the 1x vibration level has been gradually increasing from around 30 µm to over 50 µm for a period of around 3 years (see Fig. 2).

Figure 2 Vector radial vibrations on the inboard bearing: Alert alarm limits exceeded by 1x vector magnitude vibration at running speed. Danger alarm exceeded by 1x vector phase.

Figure 3 Spectrum radial vibrations on the inboard bearing: Alert alarm limits exceeded by 6% constant percentage bandwidth spectrum (CPB6%) at running speed.
The Engineer initially suspected the problem was mechanical unbalance caused by fouling, so a camera inspection was ordered.

4 MACHINE INSPECTION AND REPAIR

The first thing noticed during inspection was that the cooling water piping was seen cracked with parts of the pipe missing. When the compressor was opened, there was no evidence of fouling as previously thought, but a large crack was found on the base of one of the first stage impeller blades. The damaged impeller could be repaired but it was quicker to install a new one, which resulted in a 12-week shutdown. If the machine condition monitoring system had not been monitoring this machine, it is possible the protective monitoring portion of the system could not have tripped the machine in time to prevent a catastrophic failure. Such a failure could have destroyed both the rotor and casing, which could have resulted in 12 months of downtime.

Figure 4 First-stage impeller crack.

5 POST FAULT ANALYSIS AND DIAGNOSIS

Looking back at the data in Fig. 2, the changing 1X vibration and phase is an indication of a shaft/rotor crack. It is interesting to note that the vibration was higher at the inboard bearing position (stage 4) than the outboard (stage 1) position where the impeller crack appeared (see Figures 1 and 2). The compressor manufacturer said that this is normal for this type of machine. The effects of the mass balance between the first and fourth stage can influence the location where the first stage 1X vibration will show up.

The condition monitoring system had averted a catastrophic failure of the machine, but there was little lead-time to cost-effectively plan maintenance ahead of time. Moreover, the initial diagnosis was incorrect as it was based on the premise of unbalance due to fouling, which is quite different from a rotor crack. The compressor had to be shut down quickly which resulted in unplanned downtime. Could the fault have been detected automatically, earlier? Could it have been correctly and automatically diagnosed as an impeller crack?

\[1\] There is a lot of literature on rotor crack detection using vibration analysis. There are also a number of different vibration analysis techniques used. References [1-6] give an overview of some of these techniques and [7] gives a description of a crack detection technique using 1X magnitude and phase measurements for detecting cracks in gas turbine blades and rotors.
Figure 5 1x vibration magnitude increased gradually from 2001 to 2004, where there was an abrupt increase in January 2004. 1x vector phase increased abruptly first after the shutdown in June 2002, and then again in January 2004.

Looking at Fig. 5, there were obvious symptoms already back to September 2002 that indicated the beginning of a phase change. There had been a shutdown in June the same year but the compressor was never opened for inspection at that time. For this reason it is doubtful that this shutdown could have aggravated the problem. There were also indications even further back to September 2001 where the vibration levels began to gradually increase. But it is unlikely these symptoms would have ever been noticed since they were below the alarm levels. The phase increased in October 2002 but it quickly levelled off and remained stable for a year and a half.
after, so there was no real concern as of yet. The gradually increasing vibration level was still below alarm limits right up until the last moment, when both the magnitude and phase then suddenly increased and the machine had to be manually shut down.

The problem is that no one had noticed these early indications of a developing fault.

The monitoring system supplier offers an automatic machine diagnosis program that can be installed as an add-on to the existing monitoring system. No such diagnosis program was installed at the time of the impeller crack, so the customer asked the monitoring system supplier to retroactively scan the compressor database with the diagnosis program. The Rotating Equipment Engineer wanted to know if the developing impeller crack could have been detected earlier using this automatic diagnosis technique.

5.1 Automatic machine diagnosis system
The neural network-based automatic machine diagnosis program that runs on the condition monitoring system, classifies measurements recorded by the COMPASS System by looking at the pattern of the levels at different frequencies in the vibration spectrum. As soon as a machine shows “non standard” behaviour, this is identified by the pattern recognition mechanism and converted into a machine problem. This means all that is needed to automatically diagnose a fault is some initial data where the machine is assumed to be in good condition. It is not necessary to set up alarm limits to diagnose a fault.

The diagnosis software package evaluates the complete database on a regular basis (e.g. hourly) or it starts when it is triggered by an event. The results of the evaluation are:

- List of calculated certainty of diagnosis of potential faults
- Description of the diagnoses of the potential faults (rules, symptoms)
- Plots showing a trend of the certainty of the diagnosis over time

Figure 6 The Automatic diagnosis program lists all likely and unlikely diagnoses for a given database scan. The diagnosis with the highest certainty after 21 November 2002 is the “shaft change” (i.e. rotor crack). Other machine faults can be seen listed below this one.
Figure 7 Description explaining the rules and symptoms used in the diagnosis.

Figure 8 This plot shows the calculated certainty of the shaft crack diagnosis. Already by 21 November 2002 the certainty of diagnosis was up to 80%. Even though the certainty levels of the rotor crack diagnosis are relatively low prior to this date, the diagnostic trend has been increasing in terms of probability (certainty) the entire time.
6 CONCLUSION

Safety systems are not always effective in protecting a machine against faults such as blade or rotor cracks. It is difficult to detect the minute energy changes caused by a developing crack using overall vibration measurements. Once a crack has reached critical proportions, the rotor could conceivably fail before high vibrations are detected by the system.

There is a greater likelihood that rotor cracks can be detected at a relatively earlier stage of development by using any number of transient or steady-state vibration analysis techniques. 1X and/or 2X vector measurements are one possible technique. The changes, however, can be subtle and below the defined alarm limits, so it may be necessary to trend the values over time. This can be exhausting work for a diagnostician to look at all data even before they exceed alarm limits. An automatic trend alarm function for each measurement would be more useful, but this would have to be set up individually for each measurement.

An automatic expert system provides the optimal solution since it would look at several symptoms at the same time when making a diagnosis – not just one. As described in the case story in this paper, the diagnostic system can be used at any time to search the entire monitoring system database for symptoms. As the symptoms begin to develop over time to fulfill the criteria established in the diagnostic rules, the certainty of the diagnosis increases over time. The calculated diagnosis certainty can then be plotted and trended to give advance warning of a fault. The diagnoses created by the diagnosis program can also be automatically sent as a file to a maintenance management system or operator.

Using such a system can significantly reduce the workload of the system user while at the same time give an earlier warning of a developing fault. The program can also be used to fine-tune diagnosis rules and symptoms and to “verify” the diagnoses done by others.

REFERENCES

Gas dynamic design of powerful pipeline compressors not based on model tests

Y.B. Galerkin, K.A. Danilov
Compressor Dept., Technical University, Saint Petersburg, Russia

NOTATION

c_w – tangential component of the absolute velocity after an impeller
C_wfr – friction drag force coefficient
p_es – gas pressure at a compressor exit
p_inl – gas pressure at a compressor inlet
Re_u – Reynolds number related to mean surface velocity and blade’s length
u_2 – periphery velocity of an impeller
\bar{V} - volumetric flow rate
\Pi = p_es / p_inl - pressure ratio
\psi_r = c_w / u_2 - work coefficient

h_p – polytropic head
\psi_p = h_p / u_2^2 - polytropic head coefficient

\Phi = \frac{\bar{V}}{4 D_2^2 u_2} - flow rate coefficient

ABSTRACT

The urgent necessity to develop a new generation of pipeline compressors in Russia has forced the compressor manufacturers to apply gas dynamic designs based on the Universal Modelling method - (1), (2), (3), (4). Fourteen pipeline compressors with 2 - 6 stages, power 4.0 – 25 MW, pressure ratio 1.35 – 1.70, exit pressure 56 – 125 bar were designed, produced and tested in 1998 – 2004. The model tests were reduced to minimum in two cases, and were excluded completely in others. The predicted performances of compressors’ stages and their geometry create the bank of “virtual” model stages.
1. UNIVERSAL MODELLING

The engineering tool for gas dynamic optimisation of industrial centrifugal compressors (1), (2), (3), (4) is based on the physical scheme that follows from measurements of flow structure (rotating impellers are included), and its visualisation (5). In case of impellers main features of flow behaviour can be described as that:

- separation of flow can take place in a shape of a wake on suction sides of blades at design flow rates, and appears for sure at lower flow rates,
- no significant separation takes place at other surfaces at any flow regimes. Two effects prevent separation - the normal component of turbulence is activated at blades’ pressure sides by normal forces of inertia, boundary layers at hub and shroud surfaces remain thin due to positive effect of secondary flows.

The basic problem of gas dynamic curves modelling is calculation of head loss. The Universal modelling predicts friction losses at all surfaces, mixing losses due to wake at suction surfaces Incidence losses are calculated and added at off – design flow rates.

The general idea of the modelling is clarified by the sample of friction losses. A thin plate in a free flow creates a drag force due to friction on its surfaces. The well–known empirical approximation of the drag force coefficient for a smooth thin plate is:

\[ C_{wfr} = \frac{0.0307}{Re_w^{1/8}}. \]  

(1)

In case of impeller blades’ or hub and shroud surfaces the difference is that flow has gradient along surfaces and in a normal direction as well. The Universal modelling describes proper influences to friction losses in such a mode:

\[ C_{wfr} = \frac{X_1}{Re_w^{X_2}} (1 + X_4(1 - \bar{w})^{X_4}) \left(1 + X_5 (\bar{\omega} \partial \bar{w} / \partial \bar{n})^{X_5}\right). \]  

(2)

In the formula (2) there are:
- \(X\) - empirical coefficients that are defined by comparison of calculated and tested performances of numerous model stages,
- \(\bar{w} = w_2 / w_1\) - flow ratio along a surface,
- \(\bar{\omega} \partial \bar{w} / \partial \bar{n}\) - normalised velocity gradient normal to a surface that is responsible for turbulence and secondary flow activity.

It is evident that the structure of the formula (2) is of an arbitrary character. The function \(C_{wfr} = f \left(Re_w, \bar{w}, \bar{\omega} \partial \bar{w} / \partial \bar{n}\right)\) could be presented in many other ways. The chosen structure is based on the Authors’ intuition, but its validity is proven by practice also.

The set of formulae alike the formula (2) describes friction and secondary losses in all elements of a stage, influence of Mach number, incidence losses (formulae for incidence losses are shown in p.3), etc. Total number of empirical coefficients exceeds two dozens. Their values are established by the special procedure of numerous empirical data reduction. The model for performance curves’ calculation is the base for direct and reverse solution codes. The codes are applied as for the individual stages and for the multistage compressors as well.
2. OBJECTS OF DESIGN

The early versions of the Universal modelling codes were applied in design practice still in 1980’s. At these times the design solutions were validated by at least one of stages model tests of a multistage compressor. To mid – 1990’s more than 20 types of compressor flow paths were designed by the Compressor Department and constructed by compressor manufacturers. Many of these machines are still in production. More than 250 compressors of this generation operate in the pipeline industry now.

In the beginning of 1990’s “GAZPROM” starts the program of its compressors innovation. The “GAZPROM” compressor specialists formulated general and special demands to pipeline compressor performances:

- the maximum efficiency is necessary in a wide as possible flow rate range,
- the gas turbine drive with variable RPM do not solve completely the problem of a compressor – pipeline performance matching. The smaller is the surge coefficient - ratio of surge and design flow rates \( \frac{\Gamma_{\text{sur}}}{\Gamma_{\text{des}}} \) the better,
- a steep pressure curve is necessary,
- a power consumption curve must have the maximum at the design flow rate. It guarantees the gas turbine effective operation at high flow rates.

It appeared that Russian manufacturers had no ready prototypes or model stages to satisfy all these demands completely. Stages with vaned diffusers were applied in Russian compressors usually. It helps to obtain highest efficiency at a design flow rate and diminish necessary radial sizes of compressor bodies. The stages with vaneless diffusers and low work coefficients \( \psi_T \) appeared to be were necessary.

The TU SPb Compressor Department applied Universal modelling codes to design model stages and compressors of the new generation. The limited number of model stages with very short vaneless diffusers were developed – to substitute old flow paths with vaned diffusers in existing bodies. Fourteen types of the original compressors and so-called “Changeable flow path” – CFP** – are in regular operation now. Four more designs are still in work. The parameters of realised designs are:

- two-stage compressors and CFP for gas transportation along pipelines with power 16 and 25 MW, pressure ratio \( \Pi = 1.35 – 1.44 \), exit pressure 5.35 – 7.45 MP,
- two-stage CFP for compressors that boost gas from wells to pipe lines with power 16 MW, pressure ratio \( \Pi = 1.64 – 1.70 \), exit pressure up to 7.45 MP,
- multistage compressors and CFP to inject gas in underground storages with power from 4.0 to 10.0 MW, pressure ratio \( \Pi = 1.70 – 2.50 \) and exit pressure up to 12.5 MP.

3. ORDER OF DESIGN

The up-to-date order of the TU SPb gas dynamic design gives quantity answer to all possible demands. The first step is a compressor variants’ analysis, while RPM, number of stages and

---

* Proper texts in Russian can be found in the Transactions of the annual International symposiums “Compressor Users – Manufactures” in TU SPb.
** CFP – a new rotor and diaphragms to replace old ones in the same compressor body. An aim is to reach better performance curves, or obtain compressor parameters – flow rate and pressure rate – different from original ones.
impeller diameters can be analysed. Main parameters of stages’ dimensions are optimised after
for the chosen variant. Fast operating code automatically changes flow path shape and
calculates efficiency choosing the best variant. The stages’ and the compressor performance
maps (RPM and inlet parameters can be varied) are calculated for a chosen variant.

The details of an impeller configuration are taken into account by surface velocity diagrams’
analysis – inviscid Q-3D calculations by the singularity code 3DM-023 – (3), (4). This step of
design is qualitative mainly.

The main problem of this design mode is whether calculated performances are reliable, or not.
The Universal modelling method reliability depends mainly on the consistence of the used
physical model and on its proper description by the mathematical model.

Being field – type fast engineering tools, the proper codes do not take into account all details of
a shape of investigated or designed flow paths. Therefore it is not possible to expect complete
matching of measured performances and calculated by the Universal modelling ones. The
accuracy of a prediction that is enough for design practice must be accepted.

The practical application of the Universal modelling shows that the work input and compressor
efficiency are predicted with accuracy inside the gap of ±1,0 – 1,5% as a rule. Matching of
performances in all range of flow rates is not so satisfactory. This result seems to be quite
natural to the fast method with an approximate description of stages’ geometry.

Anyway, in all cases of design the manufacturers’ technical specifications were satisfied. But
having the data of compressors’ tests, the Authors achieved better matching of performance
curves by proper correction of the empirical coefficients X. In most cases the compressors’
efficiency and pressure ratio in a design point were so close, that there was not any necessity to
correct coefficients that control friction and separation losses.

The objects of corrections were in necessary cases the empirical coefficients that control
incidence losses. The way to calculate incidence losses is the same for impellers, vaned
diffusers, return channel blades and volutes. In case of an impeller the head of incidence
losses \( h_{\text{inci}} \) is:

\[
h_{\text{inci}} = X \left( \frac{c_{\text{inci}}^2}{2} \right) \left( 1 + X_{r1} M^2_{\text{omax}} \right), \tag{3}
\]

\[
\Delta c_{\text{i1}} = (c_{\text{inci}}^1 - c_{\text{i1}}) \tan \beta_{\text{inci}}, \tag{4}
\]

where:
- \( \Delta c_{\text{i1}} \) - a velocity tangential component appeared in result of flow sudden turn at incidence
  inlet,
- \( c_{\text{inci}}^1, c_{\text{i1}} \) - meridian velocity components at no incidence and an arbitrary flow rates,
- \( \beta_{\text{inci}} \) - flow angle that corresponds to no incidence inlet,
- \( M_{\text{omax}} \) - maximum Mach number at a blade profile.
Surface velocity diagram is estimated in a course of calculation to define \( M_{\text{rel max}} \) and other parameters necessary to calculate losses. Empirical coefficients \( X_1, X_{i+1}, X_{i+2} \) can be varied to match the measured efficiency performance curve with the calculated one.

4. SAMPLES OF APPLICATION

4.1. Gas storage compressor 10.0 MW
The compressor was provided with 5 stages and develops the pressure ratio \( \Pi = p_{\text{ex}} / p_{\text{in}} = 1.70 \) at \( p_{\text{ex}} = 12.50 \text{ MP} \). The cross section of the machine is presented at Fig. 1. After the design the model of the 1-st stage with axial inlet nozzle was developed and tested. Real radial inlet nozzle, four remained stages and exit collector were not tested but modelled by the Universal modelling code. The efficiency and the pressure ratio were predicted properly at the design flow rate but the tested performance appeared to be more effective at bigger flow rates. The proper empirical coefficients were modified. Comparison of the tested performances with calculated ones (corrected) is presented at Fig. 2.

It is worth to note that predicted and measured efficiency of this machine appeared to be 3% higher than the SOW (“Suggestion of Work”) limitation.

4.2. Two-stage pipeline compressors
The row of two-stage gas – transportation compressors and CFP 16,0 and 25,0 MW were developed, tested and are operating in industry now.

As the sample the compressor 16,0 MW that develops the pressure ratio \( \Pi = p_{\text{ex}} / p_{\text{in}} = 1.44 \) at \( p_{\text{ex}} = 7.45 \text{ MP} \) is shown at Fig. 3. The problem of this machine was too short vaneless diffusers as its body had limited diameter. As in the previous case the model of the 1-st stage with axial inlet nozzle was developed and tested. Real radial inlet nozzle, the second stage with exit collector were not tested but modelled by Universal modelling.

In other cases of two-stage compressors no model tests were used. Anyway, the predicted performances were validated by tests.

4.3. Booster CFP 16,0 MW
Two – stage CFP that develops the pressure ratio \( \Pi = p_{\text{ex}} / p_{\text{in}} = 1.70 \) at \( p_{\text{ex}} = 7.45 \text{ MP} \) had to replace the standard compressor flow path with the pressure ratio \( \Pi = p_{\text{ex}} / p_{\text{in}} = 1.44 \). Elevated work input was achieved by applying impellers with bigger work coefficients and with bigger diameters. Shortened vaneless diffusers and low specific speed make the design less effective. Anyway, application of CFP as the quick temporary solution gives big economy for the system in a whole.

Though the CFP stages had no tested analogues, the design was not accompanied by any model tests. In this case the predicted efficiency was a bit too optimistic, but as this possibility was taken into account, the design point parameters satisfied SOW. The elevated losses in the radial nozzle were found during the test. After the proper correction the comparison of performances shown at Fig. 4 is acceptable.
4.4. Gas storage compressor 4.0 MW
The compressor was provided with 6 stages and develops the pressure ratio $\Pi = p_{ex} / p_{int} = 1.70$ at $p_x = 7.45$ MP. The stages had no tested analogues and the design was not accompanied by any model tests. The favourable conditions to design high–effective machine took place this time. The optimal number of stages with low work coefficients and long enough vaneless diffusers were applied. The comparison of performances is presented at Fig. 5. No correction of empirical coefficients was necessary to obtain good correlation between measured and predicted performances.

5. VIRTUAL MODEL STAGES

The model stages in Russian practice are subdivided on three groups:
- intermediate stages: impeller + diffuser + return channel,
- suction stages: suction nozzle + impeller + diffuser + return channel,
- final stages: impeller + diffuser + volute (or collector).

In a course of fourteen types of compressors and CFP development there were designed 14 suction stages, the same number of final stages and 24 intermediate stages. As it was mentioned above, only two intermediate stages were tested as models.

The tested performances of all fourteen compressors and CFP were predicted in a course of the design. In necessary case the correlation of the empirical coefficients was executed to get better matching. The new values of empirical coefficients describe real performances satisfactory, it is possible to recon that performances of all stages of these tested compressors and CFP are described rather accurately.

The Authors recon, that the known geometry and reliably predicted performances open the possibility to use this information to design new compressors and CFP as if they would be normally tested model stages. As these stages were not tested in reality, we name them “virtual” model stages. The range of gas dynamic and geometry parameters of these stages can be seen in the table:

**Main parameters of the developed “virtual” model stages**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Design flow rate coefficient $\Phi_{des}$</th>
<th>Design work coefficient $\psi_{T,des}$</th>
<th>Hub ratio $D_{hub}/D_2$</th>
<th>Radial length of diffuser $D_4/D_2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>0.025 – 0.075</td>
<td>0.45 – 0.72</td>
<td>0.25 – 0.44</td>
<td>1.36 – 1.70</td>
</tr>
</tbody>
</table>

In accordance with the Compressor Department usual practice the name of a stage contains main information about a stage. For instance, the stage F-0370/510-350/1,65 is the final stage (I- for an intermediate, S – for a suction one) with $\Phi_{des} = 0.0370$, $\Phi_{des} = 0.510$, $D_{hub}/D_2 = 0.350$, $D_4/D_2 = 1.65$.

The performances of a group of virtual stages with parameters typical to pipeline compressors are presented at Fig. 7.
6. CONCLUSION

The design practise has shown that the Universal modelling is the adequate method for solving gas dynamic design problems. The new generation of pipeline compressors was created with minimal financial, labour and time spending. The compressors and CFP are superior in their gas dynamics to the existing analogues. As the optional result the set of effective “virtual” model stages was developed.

REFERENCES


FIGURES

Fig. 1 Cross-section of the gas storage compressor 10,0 MW.
Fig. 2. The gas storage compressor 10.0 MW. Comparison of the tested performances with the design
Points – test data, lines – calculation (efficiency is related to SOW demand)

Fig. 3 Cross-section of the pipeline compressor 16.0 MW.
Fig. 4. The booster CFP 16,0 MWt. Comparison of the tested performances with the design
Points – test data, lines – calculation (efficiency is related to SOW demand)

Fig. 5 The gas storage compressor 4,0 MW. Comparison of the tested performances with the design
Points – test data, lines – calculation (efficiency is related to SOW demand)
Fig. 7. Performances of a group of virtual stages with parameters typical to pipeline compressors.
Increasing in reliability of compressor machines by surface modification of highly-loaded parts

Efim I. Tesker, Sergey E. Tesker, Vladimir A. Guriev
Volgograd State Technical University, Russia

ABSTRACT
The data of applied research aimed at new laser surface modification technologies development are presented in this paper. Both polymer and metal surface coatings formed on steel parts under laser radiation have been tested and their influence on the friction conditions, reliability, bearing capacity and durability the drive item as well as relationships between laser treatment modes, coatings composition and various exploitation parameters have been studied. It is shown that the worked out technologies, to be used in production, provide compressor’s mechanical systems with increased reliability and operating properties.

INTRODUCTION
Intensification of nature gas and oil production and refinery processes requires continuous improvement of compressor equipment reliability. Exploitation experience testifies to limitation of compressor lifetime and working capability caused by low wear resistance of highly-loaded parts. Manufacturers develop many innovations in design and special materials used for the compressor parts their efficiency could hardly meet the multi-factor operating requirements. Having much more complicated construction design the compressor equipment often loose their maintainability for a service staff at a place (refinery, pipes, etc.).

The working out of new task-oriented techniques providing various highly-loaded compressor parts with demanded properties for operation under multi-factor actions (aggressive medium, hard friction, cycled loads) was the main goal of the studies are being presented. For the purpose in view the complex investigation of laser surface alloying and treatment of steels and polymers as well as many life-like tests have been conducted.

It is well known that insufficient damage and wear resistance of friction parts surface is the main cause of equipment’s failure. Traditionally using in production techniques of thermo- and chemical-thermohardening as well as many others are usually unable to provide items with demanded resistance especially under high dynamic loads and in aggressive medium. The process of surface wearing destruction caused by the number of mechanical and chemical phenomenon. The feature of contact interaction is the presence of considerable gradient fields of stress, temperature and deformation velocities. Without doubts that the surface layer’s role in formation of operating properties of every particular item is rising significantly thanks to its hardening but in this case the conditions for fragile or fatigue destruction could be sometimes
induced. Thus the surface modification should ensure necessary mechanical strength and that is why the investigation of physical and chemical processes in surface layers, revealing determinant factors for loading capacity of working surfaces as well as development on its basis the techniques and coatings for surface modification appears rather complicated but very actual scientific problem.

Based all mentioned above, the alloying powders chemical composition as well as CO2-laser alloying and modification techniques development was done using complex approach with concurrent consideration of following statements: items operating limits and changes in their surface layer's material properties at the final stage of lifetime; wear kinetic and failure mechanism; optimization of surface layer properties that influence on its bearing capacity, wear and failure resistance. The described approach allows to state necessary specific recommendations for a surface coating components choice and their ratio optimization as well as to make substantiated prediction of working capacity and lifetime range in the particular conditions.

Using laser advantages and features (such as highly concentrated and localized energy, huge heating and cooling rates and very short treatment times) we were forced to solve many experimental and research problems – alloys and polymer composition, treatment modes optimization, etc., that finally led us to elaboration of industry applicable technologies for laser hardening of steels, laser surface alloying on steels and irons, laser assisted formation of protective and solid-lubricating polymer films. Technologies features in detail and some examples of use will be given below.

RESULTS AND DISCUSSION

Nowadays, laser hardening of low-carbon corrosion-resistant steels which are utilized for cylinders, rods, crankshafts, plungers and crossheads production appears rather problematic. But we were capable of solving this task and had established that with specially developed and optimized laser treatment modes based on analysis of steel's chemical composition and phase conversions, that may well take place under highly concentrated energy beams, up to 1 mm (in thickness) surface layer with microhardness up to 7400 MPa can be formed. This layer has the homogeneous martensite structure with chrome-carbides distribution in its volume (see fig.1a). The indicated microhardeness value is significantly higher than it can be obtained after volume or induction heating. Besides, there is auspicious compressive stress field induced after the laser treatment which can also conduce to increasing in destruction and wear resistance. Proportional dependence between wearing intensity and microhardeness of local micro-volumes gives us grounds to suppose that wear resistance laser treated surface layer may grow up significantly. This supposition was approved by lifelike tests carried out on plungers of the opposite compressor "Créseau Luar" (France) where we obtained wear resistance rised up 2 times.

Other feature of lengthy surfaces laser treatment is the special laser track positioning, namely, with the track by track displacement value (S) less than beam’s diameter (d_b). Study of laser treated samples wear kinetics with surface waviness recording equipment let us to find out that thanks to definite distribution of laser tracks and the surface micro-hardness the specific surface micro-geometry can be formed (fig.1b). That micro-geometry provides surface with a kind of "oil-traps", which in their turn, allow to obtain resistant lubricating oil film which separates friction surfaces and is favorable for their wear resistance.
Fig.1. a - view of laser track traces in steel structure; b - surface micro-geometry after friction test.

The approval of laser hardening positive influence on the samples mechanical strength was obtained from impact tests data. The increasing in impact elasticity (up to 1.5 times) was achieved thanks to increased crack nucleation work and its propagation through the hardened layer as well as crack arrest on the hardened layer-matrix steel interface (fig.2 a, b).

Fig.2. Crack arrest on the hardened layer-matrix steel interface.

The worked out laser technologies for multipurpose surface-alloyed coatings creation are highly effective in application for renovation and production of new compressors and spare parts. The microstructures of thermal sprayed coating and laser alloyed surface coating based on Ni-Cr-V powder system is shown on fig.3. It can be seen that there is no any structural defects (pores, microcracks, incontinuity etc.) in the alloyed layer. The highest adhesion is defined by the matrix-coating metallurgical bonding. The data at fig.4 - 6 prove high operating properties of laser alloyed surface layers based on Ni-Cr-V and Ni-Cr-B-Si powder systems. At these figures the results of adhesion, strike and wear tests are presented.
Fig. 3. Microstructures of coatings (100×): a) - laser alloyed Ni-Cr-V based layer, where 1 - laser alloyed layer and 2 - matrix steel; b) - thermal spraying.

Fig. 4. Adhesive strength tests results:
1 – laser alloying (composition Ni-Cr-V);
2 – laser alloying (composition Ni-Cr-B-Si);
3 – flame spraying.

Fig. 5. Impact elasticity tests results:
1 – steel in its regular condition;
2 – laser alloying (Ni-Cr-V);
3 – laser alloying (Ni-Cr-B-Si);
4 – laser thermo-hardening.
Another one worked out and successfully applying technology is laser formation of polymer protective, solid-lubricating films on the steel friction surfaces. The actuality of such surface modification for compressor aggregates is stated by very intricate operating conditions of sliding bearings which working ability limit the whole equipment’s lifetime. Bearings operate under high dynamic loads induced by vibration processes. Vibrations, in their turn, are mostly produced by the failures of lubricating layer followed with “jamming” and “setting” of friction surfaces. The known lubricating medium phenomenon such as “dry whirl”, “oil whip” and “oil whirl” are accompanied with low-frequency vibrations, critical-frequency vibrations and dynamic disbalance with auto-vibrations. Besides, the listed damaging effects are always take place during self-accommodation of friction surfaces.

Presented below data of our research and test shows that using of worked out friction surfaces modification with solid-lubricating polymer films allow to avoid all of the damaging effects and rise up wear resistance and longevity of friction pairs.

The polymer composition was created using fluorine-containing elastomer solution in the special organic solvents mixture with following introduction of specific additives providing the polymer-steel surface chemical bonding. The airspraying of composition on the steel surface was chosen as coating method. CO₂-laser treatment was applied after 1 minute drying of obtained polymer layer. Laser treatment modes optimization was specially conducted.

The thickness of the polymer film was taken into consideration as a very important value for modified item operating properties and longevity of the film. Film thickness choice was made based on the statement that in order to obtain stable liquid friction using elasto-hydrodinamic lubricants the total thickness of lubricating films (oil, additives, solid lubricants) must be in limits 10⁻³ to 20⁻³ m. Specific thickness (λ) of lubricating film between friction surfaces can be found as total film thickness related to the sum of root-mean-square values of micro-geometrical heights (σ₁, σ₂) on these surfaces [1]. Direct microroughnesses contact is unable when λ ≥4.
Properties evaluation for the films has been done during tribotechnical tests on friction machine using oil lubricated sliding-friction between polymer coated disk and uncoated shoe. Test data presented at the fig.7. The load \( F \) was increased stepedly during the test. The loading "steps" showed as periods (I – VI) at the figures. The polymer coated disk samples tests were carried out in comparison with uncoated ones.

\[
\lambda = h/ \sqrt{\sigma_1^2 + \sigma_2^2}
\]

Fig.7. Friction test results: a) - uncoated steel sample; b) - steel sample with polymer film.
For uncoated samples the first initial testing period (I) was conducted under the load of 100 N/cm² and for the other periods (II – V) contact load was raised up from 200 to 800 N/cm² with the step of 200 N/cm². For the samples with polymer coatings the initial load equaled to 200 N/cm² and was stepped up to 1200 N/cm² with the same step.

As shown at the fig.7a, increasing in contact load up to 400 N/cm² for uncoated sample led to sharp rising of friction torque and its following stabilization. These processes are associated with sudden oil film destruction at the moment of increasing in contact load. Further increasing in contact load accompanied with growth of instability in the friction process and after 800 N/cm² it was unable to keep on the test because of oil destruction. The test results for the coated samples are shown on the fig.7b. The process is stable during all testing modes (200 N/cm² ≤ F ≤ 1200 N/cm²) and the breaking point was not reached even at 1200 N/cm² of contact load. Calculated average friction coefficient value was 0.006 what is equal to the coefficient value of liquid friction. It can be concluded that applying of worked out film coating allow to increase bearing capacity up to three times and result into 9 fold decrease of average friction coefficient. Other positive effect of polymer films application is the significant decreasing in local contact temperature.

Temperature range during the coated samples friction tests was 50 – 70°C. Absence of "temperature flashes" caused by surface "jamming" and unstable friction is also a favor for optimal lubricating. Besides, the effect of transferred polymer film on the counter body (shoe) was also revealed after the tests. The effectiveness of using worked out solid lubricating coating was proven by lifelike tests on the compressor plungers and crankshafts.

The other significant research stage was the friction tests conducted on the friction machine using vibro-measuring complex. During these tests we study vibration velocities acting on the friction item (disk-shoe). The results of the tests are presented at fig.8. The diagrams consist of eight periods. First period corresponds to the "zero-point" when neither friction machine nor vibro-measuring complex don't work. The second period characterizes the equipment's work without any load, i.e. disk spins with 600 Rpm, but the shoe is unloaded.

All vibrations and friction torque in this case produced by the friction machine drive and are equal to the testing machine characteristics (V₀, M₀). The third period describes that test conditions when the shoe is loaded with the tangential force equal to 600 N. The total friction torque has been increased at this period. The periods from 4 to 6 at the fig.8a shows that after number of cycles sharp increasing in friction torque has been registered and it was caused by the lubrication conditions change. At the seventh period the radical changes of lubrication were revealed. Oil destruction followed with its smoking became the evidence of dry friction with the highest vibration velocities and friction torque. The eighth period is equipment unloading. However, the measuring complex reading for the polymer coated sample at the periods from 4 to 6 is absolutely opposite to the ones obtained for uncoated sample (see fig.8b). There is stable friction torque with no oscillations and as a result no damaging vibrations were fixed. Conducted vibration tests show that application of worked out polymer coating allow to create favorable friction conditions, exclude friction torque fluctuations and thanks to this improve operating properties of compressor drive bearings.
Fig. 8.
Considering that the compressor machine parts are often operate in aggressive mediums which can significantly damage their surfaces, limit the lifetime of the equipment as well as became a cause of expensive repairs we have carried out corrosion test of the samples with polymer coatings. As a sample for this test the real-life compressor plungers were chosen made of corrosion-resistant steel containing up to 13 percent of chrome in its composition. The corrosion medium for this test was prepared using NACE instructions and consisted of the acid mixture (H₂SO₄ + HCl) with inclusion of metallic copper as a positive ions donor for the process promotion. The samples were exposed in the solution for 72 hours. It was found that those samples which were covered by polymer coating had no any trace of corrosion damage but the uncoated samples at the same time became worthless.

**CONCLUSION**

The conclusion is that the worked out technologies allow to:
- provide highly-loaded parts with the unique mechanical and tribotechnical properties;
- restore wearied out machine parts with no loses in their future longevity and durability;
- significantly decrease in friction coefficient, contact temperature, exclude vibrations;
- increase in parts wear resistance;
- protect surfaces against corrosion wearing.

The stated above investigation results serve as a basis for brand new resourcesaving technologies innovation at the oil and gas refinery plants. Industrial application of developed technologies confirms their high economic effectiveness and increased lifetime of compressor equipment.
Applying modern design/manufacturing concepts in the development of centrifugal air compressors for the global market

Bob Kolodziej, Scott Tackett
Cooper Compression, USA

ABSTRACT

Up until this time, the dry screw plant air compressor was the only choice to customers who were price and lead time sensitive in 200-450HP oil-free compressed air market. However, if the customer considered overall efficiency, maintainability and life cycle costs centrifugal compressors could be considered as the superior choice. By understanding the market drivers, it was apparent that if the oil-free centrifugal compressor was to be competitive in the global market a new design and manufacturing philosophy would be required. As a result, the modular design approach was adopted during the new product development process.

1. INTRODUCTION

For over 50 years the manufacturer, has been a successful producer of modern, high efficiency Turbo Compressors, which supplied an oil-free product to the air/gas market. Through most of the company’s manufacturing history, the designs were focused to a contract specific application, (designed and manufactured per customer’s specific requirements). The products were successfully designed and manufactured to support a variety of world wide industries, from air separation to snow making.

Despite the historical success of a quality engineered product, primarily serving the air separation market, it became necessary to diversify into broader markets. The air separation market is highly cyclical and for the business to maintain consistent volume in its factory, the business decided to pursue the standard plant air market in the mid 1980’s. Over the course of that time frame, using oil-free air in plant operations has become the industry standard. The market for oil-free air ranges from textile mills to automotive factories to bottling companies. Each facet has unique requirements in terms of discharge pressure and the amount of flow needed to run plant operations. An interesting trend develops in terms of customer requirements and how they impact the selection of compressor designs. Those customers who need lower flow machines are much more price sensitive than customers who need larger flow machines. Larger machine customers weigh compressor efficiency much more heavily in the decision making process. For example, a 1% efficiency gain for 1000 horsepower or greater compressors translates into a significant savings in energy costs. Smaller machine customers
are much more sensitive to “first cost” and lead times for these types of purchases. As a result, it was no surprise that the dry-screw compressor was outselling centrifugal designs by an estimate of 50-1 in the smaller range.

Consistent with the business’ past practices, on a yearly basis existing product offerings and new market requirements were reviewed. A determination was made as to which offerings should be modified, enhanced or when a new product design was necessary. It was determined at this point, if the business wanted to grow into the smaller plant air market, it needed to adopt a new design philosophy rather than to continue to design products that merely satisfy engineering specifications. The task was to deviate from the present practice of contract-specific manufacturing/assembly and move to a world sourced, standardize product; with the goal to compete with the high end dry-screw market. This process is referred to as gathering the “voice of the customer.” (VOC)

2. THE PLAN

To insure success the team turned to its Six Sigma philosophy to begin the process. For those unfamiliar with Six Sigma, probably the simplest explanation is that it is a process whereby data is collected and used statistically in order to make decisions. This data acquisition is a continual process where the entire new product development process begins at the customer and then moves through almost all of the business functions such as marketing, engineering, manufacturing, and sourcing before a final product is delivered to the customer.

2.1 Marketing - capturing the “voice of the customer”

In the case of new product development that data collection took on the form of surveying customers to obtain both quantitative and qualitative data on the smaller air compressor market in general. This consisted of sales and marketing personnel interviewing a broad range of indirect and direct customers. They asked about their likes and dislikes, and learned what they would do if they could build the ideal plant air compressor product. In essence, the team let the customers become a part of the design engineering team.

After collecting the data a Six Sigma tool known as Quality Function Deployment (QFD) was employed to sort through the results. In a QFD matrix the data is analyzed to determine its validity and importance. The end result is a prioritization of what matters most (in this case to the customer).

The results clearly showed that the centrifugal plant air compressor out performed the screw compressor in all areas except cost, lead time and customer familiarity with the product. It was clear many customers wanted the least expensive capital cost unit, regardless of the maintenance requirement or operating life cycle cost. It was reported that maintainability of the unit and operational costs were not a main factor in the initial purchase decision process. The purchasing group was measured mainly on the initial cost and not life cycle costs.

2.2 Establishing manufacturing targets

The data clearly showed if centrifugal compressors were to have an impact on the screw compressor market it would have to reduce the standard cost by approximately 20% and the lead time from 8 to 4 weeks, while maintaining existing performance and quality. At this point the businesses had established a baseline by performing a sensitivity analysis and identified the cost makeup of the existing product.
Even though the above task seemed simple, because each unit was built per order, identifying true material and labor cost was challenging. Because it was critical to get good base line information, the team tracked the build cycle of several similar basic units. It was found that the build time could vary significantly due in part to assembler’s skills and/or material availability. Next the team disassembled a finished unit, categorizing parts into functional systems, (i.e. oil system, inlet air system, etc), and rebuilt the same unit with tools and instructions available. It was determined that a reduction in labor hours and cycle time was possible. Now the team had a starting point and was able to clearly identify present material cost and labor hours for a specific contract, if all parts were available.

2.2.1 Assembly exercise observations
The assembly exercise not only quantified present performance rates but identified design and procedure enhancements which would further reduce cost. It became apparent that the business was not taking advantage of “quantity of scale” part order purchases. In many cases the procurement group was purchasing one part per contract. This practice increases cost exponentially. A VOC survey was also performed with the assembly team. From the results, it was also apparent the design was not “assembly friendly.” Not only were there many parts that could be combined or eliminated, reducing part count, there were many parts that were physically difficult to put together. It was also noticed during the assembly cycle, functions that were core competences did not consume excessive labor. Once assembled the unit looked and performed as designed, however it was labor intensive and unnecessarily expensive. The conclusion was obvious; the team had to design for easy manufacturing and assembly if the business was to achieve team lead time goal of four weeks with a 50% reduction in labor costs.

After analyzing the above data it was becoming clear what had to be done if the team was to achieve the cost and lead time goals. However, regardless of the new knowledge, incorporating a new design/assembly philosophy could not jeopardize the commitment to the new product launch schedule. Because additional experienced engineering design resources were not available, the engineering staff relied on the core competencies of its supply base to help in the new design and procurement process. What started out as a request to design a new cost reduced product evolved, into a new product development model.

3. A NEW WAY OF PERFORMING PRODUCT DEVELOPMENT
The new product development model went much further than simply applying standard engineering principles into the new product. To truly optimize the product it was critical to consider all operational functions to include manufacturability, procurement, assembly and most importantly customer delivery. As a result of the worldwide customer survey, it showed that the VOC varied by geographical location. Because the intent was to have a world wide offering, all the requirements had to be considered into the design. Just as critical was identifying the core competency of select vendors and partnering with those who supported the same goal. In the past all design activity was focused within the company’s engineering department. If the engineering staff had to support marketing’s entire present and future requests the department size would have to expand proportionally. With no additional internal engineering support available, the staff would have to leverage resources elsewhere. By relying on the core competencies of the supply base, university students and outsourced engineering, the staff was successful with keeping their core engineers managing the critical design applications of the compressor. The overall benefit of this approach was that the business was
able to satisfy its design requirements and introduce new technology that would not have been discovered if the design was done 100% internally.

Due to the overall global market, the assembly process had to be simplified to the point where the product could be assembled at various world wide locations, where the customer density was the highest. The benefit of this approach was to significantly reduce lead time while being able to incorporate locally sourced material. As a result, the engineering staff moved to a modular design concept in the development of the centrifugal plant air compressor. Because the design utilized modular components, vendors were now able to stock the module, ready for immediate shipment. Freezing the standard design and modularizing also proved to be a major benefit to controlling quality, which was a paramount requirement of the product. Rather than performing quality checks on numerous parts, sourcing quality engineers only had to check dimensions for the module. Finally, standard cost, which had been a mystery until the product was shipped is now readily available at time of order. The purchased cost of each module is tabulated along with the pre-determined labor hours to install it. The limited options are prepackaged, listing the material and labor cost to install it. Adding test time and summing will yield the standard cost for the product at time of quote.

There are many additional supporting benefits to the modular design. Each module performs a specific operational function; enhancement of a module for cost and/or performance reasons is concurrent while the product is being marketed. Upgrading an existing module will not affect overall functionality of the module or assembly of the package. In the past labor hours varied, dependent on how skilled the service technician was, when replacing a specific system or adding an enhancement option. Now each module and option kit is prepackaged with instructions, required tools and required labor hours to install. As an example, the water cooling manifold that originally had 130 parts is now down to a more manageable 17 parts as shown below.

4. THE MODULAR APPROACH IN FUTURE DESIGNS

There is no set format to utilize the modular design philosophy; however, there are areas in each phase where an appropriate risk mitigation plan is appropriate. Each design and/or project requires consistent dynamic interfacing activity among functions, modifying as required until project completion, to optimize success. Performing a sensitivity analysis during the concept phase of the project typically allows areas of high risk, cost and/or performance problem, to surface. This can be accomplished using Six Sigma tools such as a FMEA (Failure
Modes Effect Analysis). Consequently the problem design areas will be focused on for correction or eliminated where possible. Because each function affects the performance of a supporting function, (to various degrees), it’s essential that interfacing activity throughout the design phase be active. To support the dynamics of real time data transfer between functions, (internally, domestically and internationally), it is essential to select support services, (vendors, students, outside resources), or setup support services, with an electronic communication link. Doing the above eliminates surprises between engineering design, manufacturing and assembly throughout the design phase and at project completion. In addition to design, there are several supplemental benefits to electronic communications/data transfer. With global sourcing, design assignments can be transferred domestically or offshore in the evening and the project can be worked on throughout the night in preparation for the designer to continue with the design in the morning, thus, reducing the detailing phase of the design time. A bit of caution at this point, because of the ease of design changes with solid modeling, it was not uncommon to have continuous design changes being made throughout the day. Changes must be managed, focusing on the specific objective of the system. Once this is accomplished, it is suggested that the engineering department lock-in and freeze the design or potentially miss the design time commitment.

![Diagram of Interaction of departments in new product development](image)

**Figure 3: Interaction of departments in new product development**

Going beyond the design benefit of transferring solid models and moving on to manufacturing; it was found that the solid model could be transferred to various machine shops and casting pattern shops. This was invaluable; the casting vendor would now be able to quote the project quickly, in addition to making immediate suggestions as related to his process, reducing the cost and lead time. Turning a request for quote went from 24 days to 2. The machine shop vendor was able to quote off the model and was also able to build off the solid model, generating a program, to run his tool path.
With the new modular design philosophy it was determined that there was considerable benefit to manufacturing and assembly if each of the standard modules incorporated all of the typical VOC operation requirements and controls, creating a single part number for a high percentage of the sales. Many options were now included as part of the standard product. Additional options were limited, but where required, could be installed easily. During the design phase of the module various vendors were contacted who specialized in that specific system. It was expected that the vendor participate in the design process so that part performance and cost reduction could be maximized. It is suggested that prior to meeting with the specialty vendor, (again applying 6-sigma practices), the engineering staff should understand the breakdown of each module to include material and the necessary labor to install it. Several vendors were evaluated for each module and were rated on performance, quality, cost, engineering relationship and the ability to transfer data. As a result of this interaction, the final module design was a combination of ideas between the vendor and the engineering staff.

4.1 Specific design recommendations
When performing a new product design from scratch it is important to brainstorm and develop several conceptual illustrations of the new product; all of which comply with the VOC direction. It is imperative that the VOC collection represents the best sampling available in order to create a marketing specification for the design team to work from. At this point of the design, time should not be an overriding factor since a good base design simplifies the rest of the project.

Modularize and tabulate the discreet systems, (controls, oil, gear box, etc.), of the selected design. Evaluate each of the systems for performance, material cost, labor cost and risk. Transform final conceptual sketch into solid modes, in preparation for transfer to support functions. Design and modify each system working off existing proven designs and/or standard engineering practices. All new conceptual designs/developments will be spun off to R/D for evaluation and testing. On a parallel path apply a standard back up in the event the new development falls short in performance or cost, thus; still being able to comply with time to market commitment. Select which of the modules is the core competence to the company along with identifying the core competence of selected vendors with the remaining modules. Only work on the company’s core competency and outsource the rest. The design of the product needs to be engineered around the capabilities of the vendor while also keeping in
consideration the impact to assembly. Continuously monitor the design so as to simplify and reduce parts as it will significantly impact transactional costs. In addition to reducing parts, eliminating assembly processes is just as effective in reducing cost.

For example in the picture above, the previous assembly procedure had the unit painted after assembly and test. Due to all the required steps in painting it was very time consuming, in addition to being only marginally effective in controlling rust. With the new plant air compressor all parts from gear box castings to oil tanks are phenolic coated, (off site), prior to receiving. This eliminated field rust problems, environmental paint issues, and the associated painting, labor and material on the shop floor. This step alone eliminated 5 days of the product cycle.

### Module Identification and Evaluation

<table>
<thead>
<tr>
<th>Module #</th>
<th>Module Name</th>
<th>Vendor</th>
<th>Material Cost</th>
<th>Labor Cost</th>
<th>Total Cost</th>
<th>Lead Time</th>
<th>Risk</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Compressor Area</td>
<td>CTC</td>
<td>$</td>
<td>$</td>
<td>$</td>
<td>time</td>
<td>%</td>
</tr>
<tr>
<td>2</td>
<td>Inlet Assembly</td>
<td>A</td>
<td>$</td>
<td>$</td>
<td>$</td>
<td>time</td>
<td>%</td>
</tr>
<tr>
<td>3</td>
<td>Control Center</td>
<td>B</td>
<td>$</td>
<td>$</td>
<td>$</td>
<td>time</td>
<td>%</td>
</tr>
<tr>
<td>4</td>
<td>Drive Motor</td>
<td>C</td>
<td>$</td>
<td>$</td>
<td>$</td>
<td>time</td>
<td>%</td>
</tr>
<tr>
<td>5</td>
<td>Oil System</td>
<td>D</td>
<td>$</td>
<td>$</td>
<td>$</td>
<td>time</td>
<td>%</td>
</tr>
<tr>
<td>6</td>
<td>Option Group</td>
<td>E</td>
<td>$</td>
<td>$</td>
<td>$</td>
<td>time</td>
<td>%</td>
</tr>
<tr>
<td>7</td>
<td>Hardware</td>
<td>F</td>
<td>$</td>
<td>$</td>
<td>$</td>
<td>time</td>
<td>%</td>
</tr>
<tr>
<td>8</td>
<td>Test</td>
<td>G</td>
<td>$</td>
<td>$</td>
<td>$</td>
<td>time</td>
<td>%</td>
</tr>
</tbody>
</table>

*Figure 8: Compressor engineering by module*

After the resources and vendors were identified complete the above table per module. At this point not only will the vendor be selected per module, it will be a starting point, identifying Module: design, required performance, data transfer, interfacing, assembly, lead times, risk, cost, etc. It should be apparent at this time the relationship with the vendor and the manufacturer is solid with a common goal. Remember, the manufacturer is responsible for the final product and VOC commitment, having an alternative source for all critical components is wise. It was found that following the above guideline optimizes the design; performance and cost along with minimize time to market.
Generic New Product Development Procedure Outline

- Identify engineering design team, supported by marketing, manufacturing, procurement, aftermarket services, vendors and out source services
- Confirm and audit marketing’s scope of supply of new product, modifying as required, from cost to desired scope.
- Brainstorm, by hand sketching several conceptual illustrations of full package and review with potential customers.
- Compare each conceptual design package with a 6 Sigma evaluation, identifying each system with part count, cost, efficiency, risk, etc.
- Structure each of the product systems into a discreet functional module.
- Create solid models of selected design package and individual module.
- Identify and keep in house modules which are the company’s core competence.
- Identify support vendors, capabilities and their core competence. Confirm vendor solid model capabilities.
- Match specific modules to be outsourced to selected vendors. Link each vendor and internal functions electronically through, solid models, spreadsheets, etc. where required.
- Confirm that a continuous dynamic link exists between all functions, from marketing to manufacturing to procurement, to insure milestones are tracked and followed.

5. SUMMARY

Today’s global economy will continue to put pressure on all facets of the business from sales to marketing to engineering to procurement to assembly and final delivery to the customer. Being able to recognize market needs and quickly react with innovative yet cost effective designs is imperative. By incorporating a module design approach, the team was able to design and deliver a new plant air compressor within a lead time that meets or exceeds customer expectations. On the TA2020, the product development team reduced lead time from 8 weeks to 2 weeks while being able to offer a competitively priced product through the module concept. This was accomplished by utilizing the core competencies of their own engineering staff while relying on the supply chain to help design innovative solutions that enables global sourcing while facilitating the use of low cost labor for assembly. The success of this project will significantly impact future plant air compressor designs where pricing and lead times are paramount.
REFRIGERATION COMPRESSORS
Lubrication quality assessment and viscosity measurements in AC/refrigeration compressors

Ali T. Herfat
Emerson-Copeland Corporation, USA

ABSTRACT

Viscosity control for product quality and process control is crucial in a variety of manufacturing operations. As manufacturers and suppliers focus on customer satisfaction and consistency of product, lubrication quality becomes more important because it directly influences cost efficiency and products quality in various operational applications. In oil condition monitoring (which includes machine, contamination and lubrication condition), viscosity measurement is a form of machine condition monitoring. Because viscosity can be greatly affected by physical variables, such as temperature, load, oxidation and contamination, viscosity measurement has challenged process engineers, Reliability engineers, and quality control departments for years. Additionally in original equipment manufacturer (OEM) applications, there is a desire by the solution providers to deliver reliability-enhancing features to their products. Monitoring of viscosity is used, for controlling of lube oil blends, or in compressors for monitoring of the oil-refrigerant mixture lubricant fluid quality. Viscosity management can result in significant savings for life, performance, and maintenance costs of equipments and machines that is monitored using a condition-monitoring program. In this research, viscosity measurements of the refrigerant vapor/liquid lubricant equilibrium viscosity reduction of a POE (synthetic polyolester based oil, grade 32 ISO VG) with the refrigerant R-410A, white napthenic mineral oil (46WMO,46 centistoke,) with the refrigerant R-22, and AK Mobil Oil and R-404A have been studied at compressor sump (bottom shell). The viscosity and composition of the solubilized gas mixture in solution with the lubricant has been obtained with the constant gas vapor at a controlled temperature. Viscosity of refrigerant/oil mixture and oil dilution percentage at compressor sump has been measured during systems operation using: 1) Electro-Mechanic technique, and 2) Acoustic technique.

1. INTRODUCTION

The primary function of lubrication is to prevent metal-to-metal contact. In order to perform this function, oil needs to be relatively free of contamination and within a specified viscosity range. Viscosity, of course, decreases with an increase in temperature. Condensers in poor heat transfer condition will have very damaging effects on the air conditioning system. The higher temperature can lower oil viscosity to the point where metal-to-metal contact occurs in the
compressor. Charging the system with extra refrigerant can cause the reduction of the lubricant viscosity by flooding back the liquid refrigerant to the compressor sump. These events will result in rapid wear, or even compressor seizure in extreme cases. This condition is often referred to as “lack of lubrication” or “lubrication breakdown.”

The most important parameters that shall be monitored in lubrication oil analysis include:

- Moisture (ppm)
- Acid Number indicates level of corrosive acids in the oil (may be caused by moisture contamination or high operating temperatures).
- Dark Color: a high number or dark color indicates higher than normal operating temperatures.
- Oxidation: an “oxide” due to moisture contamination.
- Monitoring of the lubricant quality by measuring the lubricant viscosity at different operating conditions of systems or machines.

2. CLASSIFICATION LIQUID LUBRICANT FILMS

Fluid films can be provided by:

1. Surface tension: If a drop of liquid lubricant is placed upon a flat surface and then another flat surface is laid upon the wetted surface, some liquid will be squeezed out, but not all of it. Surface tension force makes complete leaving out of liquid very difficult, or withhold the lubricant between two surfaces.

2. Hydrostatic lubrication: Two sliding surfaces can be separated by pumping a lubricant fluid into the contact region at a sufficient pressure to separate the surfaces. A large volume of fluid will separate the sliding surfaces a great distance, thereby producing a low resistance to sliding motion. Hydrostatic lubrication is effective over all sliding speeds, but its reliability is influenced by the reliability of the required external oil pump.

3. Hydrodynamic lubricant: If one surface slides along another at moderately high speed, and if the shape of the leading edge of the moving surface is such that fluid can be gathered under the sliding surface, the two surfaces will be separated and slide easily.

Bearings life is extended by having right oil managements and required viscosity at different operational conditions of machines or various load levels. Fluid film lubrication naturally divides into two categories. Thin film lubrication is usually met with in counter-formal contacts, principally in rolling bearings and in gears. The thickness of the film in these contacts is of order of 1µm or less, and the conditions are such that the pressure dependence of viscosity and the elastic deformation of the bounding surfaces must both be taken into account. Thick-film lubrication is encountered in externally pressurized bearings, also called hydrostatic bearings, and in self-acting bearings, called hydrodynamic bearings. Of the latter, there are two kinds: journal bearings and thrust bearings. It should be realized, however, that bearings never operate under truly isothermal conditions, and under near isothermal conditions only in exceptional cases. Viscous dissipation and consequent heating of the lubricant are always present, and the change in viscosity must be accounted for when analyzing thick-film lubrication problems. In restricted cases, where design and operating conditions are such as to suggest "uniform" temperature rise of the lubricant, the "effective viscosity" might be employed. In other, again very limited, cases, where heat conduction into the bearing surfaces can be neglected, the "adiabatic theory" might be useful. But in the great majority of practical cases, particularly under turbulent flow conditions, full thermohydrodynamic theory, including thermal/elastic deformations of the bearing surfaces, must be employed.
The design of the journal bearing needs to be sized properly for its loading and allowable temperature limits. Only when an oil film between the bearing housing and the shaft exists can reduce the wear rate during machine operations. The rotational motion places the shaft in an eccentrically position in respect to the bearing housing, the hydrodynamic pressure forces acting on the oil film will place the shaft into an equilibrium position.

3. THERMAL EFFECTS ON LUBRICANT VISCOSITY AND BEARING PERFORMANCE

Classical lubrication theory predicts bearing performance on the assumption that the viscosity of the lubricant is uniform and constant over the whole film. As the bearing performance is strongly dependent on lubricant viscosity, and as the viscosity of common lubricants is a strong function of temperature the results of classical theory can be expected to apply only in cases where the lubricant temperature increase across the bearing pad is negligible.

In many applications (small bearings and/or light running conditions) temperature rise across the bearing pad, although not negligible, remains small. It is still possible in these cases to calculate bearing performance on the basis of classical theory, but in the calculations one must employ that specific value of the viscosity, called the *effective viscosity* that is compatible with the average temperature rise in the bearing. Boswall (1928) calculated the effective viscosity on the basis of the following assumptions:

1. All the heat generated in the film by viscous action is carried out by the lubricant.
2. The lubricant that leaves the bearing has the uniform temperature.

The value of the viscosity is a function of the bearing geometry, bush and shaft materials, lubricant physical properties, bearing load and speed, and the thermal boundaries.

It is clearly demonstrated that the variation of lubricant viscosity with temperature and flow regime shall be taken into account. The flow regime shall specify appropriately to predict bearing performance. The maximum bearing temperature and lubricant fluid flow is mainly dependent on the Reynolds Number (Re) the Prandtl Number (Pr), the dissipation number ($\Lambda$), and the eccentricity ratio ($c/R$). Where $c$ is radial clearance and $R$ is journal radius.

The lubricant is an agent for heat removal. If heating occurs faster than does removal then a thermal spiral has begun, the lubricant degrades, and surfaces contact each other. Research in hydrodynamic lubrication focuses on the properties of fluids at high pressures, but particularly at high shear rates.

4. THERMOHYDRODYNAMIC THEORY

Thermohydrodynamic (THD) theory calculates point wise variations of temperature and viscosity in the lubricant film. Though there are instances of temperature rise across bearings operating in the laminar flow regime, THD theory becomes even more important in turbulent lubrication.

The Energy Equation: The energy equation is the mathematical statement of the Principle of Conservation of Energy or the time rate of energy increase in a body equals the time rate of energy supplied to it.
The mechanical energy consists of work done by the surface and the body forces

\[ w = \int (\tau \cdot v) dS + \int \rho (f \cdot v) dV \]  

(1)

For total energy input, we write

\[ Q = \int q \cdot ndS + \int \rho \Psi dV \]  

(2)

where \( q \) is the heat flux across the closed surface and \( \Psi \) is the distributed heat source per unit mass of the body. Assume there is no internally distributed heat source, \( \Psi = 0 \), then

\[ \int \rho \left( v \frac{dv}{dt} + \frac{de}{dt} \right) dV = \int (\tau \cdot v) dS + \int \rho (f \cdot v) dV + \int q \cdot ndS \]  

(3)

Where, \( v, \rho \) are the fluid velocity and density respectively, \( V \) is volume, and \( e \) is the internal energy.

Cope Model: To further simplify matters, we assume that the lubricant has constant thermal properties, \( c_v \) and \( k = \text{const} \). For incompressible fluids using continuity equation\(^\ddagger\) yields the dilatation work \(-p \text{ div } v = 0\), \( c_v = c_p = c \). Let \( e = c_v T \) for internal energy density and use the symbol \( \Phi \) for the dissipation function, then

\[ \rho c \left( \frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right) = k \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) + \mu \Phi \]  

(4)

For an incompressible fluid the dissipation function is

\[ \Phi = 2 \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right] + \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right)^2 \]  

(5)

Cope’s model is based on the assumptions of (1) negligible temperature variation across the film and, therefore, (2) negligible heat conduction into the neighboring solids. All the generated heat is carried out by the lubricant under Cope’s assumption (adiabatic theory).\(^*\)

\(^\ddagger\) Continuity equation \( \frac{\partial \rho}{\partial t} + \text{div}(\rho \cdot v) = 0 \)

\(^*\) conservation of energy for incompressible fluid \( \rho (d/dt)(c_v T) + p \text{ div } v = \mu \Phi + \text{div}[k\text{grad}\Phi] \). For compressible fluid and assuming a perfect gas ( \( p = \rho RT \) ), it can be presented as \( \rho (d/dt)(c_p T) = dp/dt + \mu \Phi + \text{div}[k\text{grad}\Phi] \).

Where \( \mu \) is the oil viscosity.
5. SHAFT LUBRICATION

The magnitude of viscous drag force for a fluid film between two parallel surfaces that are separated by a lubricant fluid film of thickness \(h\) and viscosity \(\mu\) can be calculated with the following equation. The force \(F\) required to slide the upper block with velocity \(v\) is:

\[
F = \frac{\mu A v}{h}
\]

Petroff calculated friction force, \(F\), in lubricated bearings as the viscous drag of fluid in the (nonuniform) radial clearance space, \(c\), between a shaft rotating in the center of a bearing, with a surface velocity of \(U\). The wetted area is \(\pi DL\) where \(L\) is the length of the bearing, and \(D\) is the diameter of the shaft. Then,

\[
F = \frac{\pi \mu DL U}{c}
\]

from which \(\mu\) can be calculated as \(F/W\) (where \(W\) is the applied load). This is Petroff's Law.

The fluid film pressure builds behind the location of the minimum separation between the shaft and the bearing, taking the shaft surface as the reference. The following formulation can be used to define the journal bearing lubrication phenomenon:

\[
\frac{\mu N}{p} \left[ \frac{D}{c} \right]^2 \left[ \frac{L}{D} \right]^2 = \frac{(1 - \varepsilon^2)^2}{\varepsilon \pi \sqrt{(1 + .62 \varepsilon^2)}}, \quad \text{or} \quad \frac{\mu N}{p} \left[ \frac{L}{c} \right]^2 = \frac{(1 - \varepsilon^2)^2}{\varepsilon \pi \sqrt{(1 + .62 \varepsilon^2)}}
\]

Where: \(\varepsilon\) is eccentricity of the center of the shaft from the center of the bearing (defined as \(\varepsilon = 1 - h/c\)), \(p=W/LD\), and \(N\) is the rpm of the shaft. The term on the left side of this equation is one form of Sommeneld's number or the bearing characteristic. Bearings with the same characteristic will operate with the same eccentricity. This value is significant since it was found that for efficiency, \(h/c\) (which equals \(1 - \varepsilon\)) should be about 0.3. The consequence of this recommendation would be a particular set of values for the adjustable variables \(\mu N/p\) for a given bearing.

The same equation, with small variation, can be used to analyze bearings in which an unbalanced shaft rotates. If the static (vertical) load, \(W\), on a horizontal shaft, is small as compared with an unbalanced force, the point of minimum lubricant film rotates with the shaft along the inner surface of the bearing. In this case the fluid wedge is ahead of the location of minimum film thickness. An interesting situation develops when an unbalanced shaft has a slightly larger and irregular vertical load applied. The shaft will oscillate during rotating, therefore the transition from balancing to unbalancing conditions leads to a thinner average lubricant fluid film and higher friction than for stable and balance condition.

There was a good analytical explanation of the bearing friction at higher values of \(\mu N/p\) in Petroff's law, namely, it is due to viscous drag between well-separated solid surfaces. The McKee brothers located the minimum friction for a number of bearings by experiment. It was widely agreed that at values of \(\mu N/p\) less than that which produced minimum friction the lubricant film is thinner than the height of the asperities on the opposing metal surfaces. This
condition is now referred to as “boundary lubrication. Typical data for a wide range of $\mu N/p$ and friction coefficient $f$ are shown in Figure 1.

![Figure 1. Strubeck-Gumbel or Mckee-Petroff curves](image)

Additives can convey certain characteristics that may be desirable in some cases. Extreme pressure (EP) and antiwear additives are the most common, with sulfur, phosphorus, zinc, and antimony being among the most popular.

### 6. JOURNAL BEARING LUBRICANT FILM CALCULATION USING REYNOLDS EQUATION

The lubricant film thickness, $h$, inside the journal bearings can be changed due to rigid body translation of the journal center relative to the bearing along the bearing radius, and/or rotation of the unbalanced journal (alternative changes). This phenomenon can be defined using Reynolds equation as

$$
\frac{\partial}{\partial x} \left( \frac{h^3}{\mu} \frac{\partial p}{\partial x} \right) + \frac{\partial}{\partial z} \left( \frac{h^3}{\mu} \frac{\partial p}{\partial z} \right) = 6U_0 \frac{\partial h}{\partial x} + 6h \frac{\partial U_0}{\partial x} + 12V_0
$$

(9)

If the bearing is stationary, then $U_0 = U_2$ for journal bearings, and $U_0 = U_1$ for thrust bearings (3)

### 7. EXPERIMENTAL OIL VISCOSITY AND DILUTION MEASUREMENTS DURING HVAC SYSTEMS OPERATION

The Oil Dilution Measurements have been accomplished for various HVAC systems in cooling, heating, and refrigeration modes at Emerson Climate Technologies-Solution Center, Copeland Global Reliability Engineering Testing Center. The refrigerant migration and oil dilution measurements have been investigated using the Electro-mechanic and the Acoustics technologies.

Due to corporate confidentiality issue, some of the preliminarily measurements results have been presented in this paper. The oil dilution measurement hardware and software technologies are in the development stages. Imposing the measurement results of these optimized technologies to the public needs the Copeland Corporation permission.
This paper presents the evaluations of POE lubricant with R-410A, White oil with R-22, and AKMobil oil with R-404A. It is significant in that it accurately represents viscosity and gas solubility of refrigerant vapor in the lubricant and evaluates the ratio of gasses dissolved in the lubricant when liquid/liquid miscibility differs. Plotted against temperature and pressure provides meaningful values for a dynamic system using thermodynamic cycle theory. The experimental data illustrates that various levels of fractionation of blended gasses occur with specific refrigerants and lubricants at various pressures and temperatures.

Oil dilution has fundamental effects in the HVAC systems. Domestic HVAC compressors have generally performed well except for a few issues connected with their high solubility. Mineral oils, being hydrocarbons, have high solubility in hydrocarbon refrigerants. High solubility can lead to: 1) foaming, 2) excessive lubricant dilution leading to potential wear problems, and 3) high oil carryover leading to slugging. Foaming and wear have been overcome by the use of traditional antifoaming and anti-wear additives. However, high oil carryover has lead to oil slugging issues in certain systems and this can lead to markedly reduced energy efficiency over time.

It has been found that the more soluble refrigerant gas component is responsible for compressor lubrication oil viscosity reduction at lower temperature. Therefore, it is obvious that the refrigerant/lubricant miscibility does not mainly influence viscosity reduction of the lubricant in HVAC applications. Oil dilution rate increases due to media temperature reduction at sump of compressor. Oil dilution fundamental effects can be summarized as

- Reduction of viscosity
- Reduction of pressure-viscosity coefficient
- Reduction of effective film thickness in bearing
- Higher maximum pressure in squeezed film
- Higher friction force in mixed-film lubrication
- Higher normal and shear stress between two mating surfaces
- Higher wear rate because of higher stresses
- Shorter wear life for the bearing

7.1 Experimental Oil Dilution Measurements Analysis and Results:
I. Refrigerant 410A, a high-pressure zeotropic blend, is currently being used in some new HVAC Systems specifically designed to operate at higher pressures. Refrigerant R-22 which is currently used in majority of unitary HVAC systems specifically designed to operate at medium pressure operations.

Viscosity and Density of mixture are function of temperature and pressure. Solubility of the refrigerant and lubricant also depends on temperature and pressure. Therefore, in general thermodynamic properties of lubricant and refrigerant are function of temperature and pressure.

At a constant pressure when temperature increases, the solubility of refrigerant decreases which ultimately results in thicker oil by separating refrigerant from oil. The maximum seems to take place at approximately 15% dilution level, at this point the influence of refrigerant on the mixture is still small and oil properties are dominant. At a certain temperature (i.e. depends on pressure) viscosity reaches to its maximum and after that by increasing the temperature no more refrigerant can be separated from the oil and viscosity decreases by temperature, Figures 2.
II. A 3-Ton AC system with refrigerant R-22 and White Oil was tested and oil dilution also simulated by increasing and decreasing the refrigerant charge weight percentage during the cooling mode operation in the Copeland controlled load room. Thermodynamic properties of lubricant and refrigerant are presented as function of refrigerant charge variation and in terms of the mixture temperature at compressor sump, and sump overhead pressure. Results show the elevation of refrigerant charge causes increasing the oil dilution despite of elevating the mixture temperature. Increasing the sump overhead pressure also contributes in the oil dilution rate, Figure 3. Electro-Mechanic technology is used in this test.

Figure 3: Viscosity measurements of R22 in White Oil (46 cst) oil mixture during the AC system with TXV

Figure 2: Viscosity of R410A in 32 ISO VG (3MAF POE), 32 centistokes, lubricant is a synthetic polyolester based oil) in terms of pressure and temperature mixture (4)
III. A heat pump (HP) system with R-410A and POE oil was tested in the Copeland environmental (indoor and outdoor) controlled load room. The compressor was instrumented and run for both cooling and Heat heating modes. The refrigerant and oil mixture viscosity was measured inside the compressor at different system operation conditions, Figures 4. Acoustic Technology was used in this test.

![Figure 4: Viscosity measurements of the mixture POE oil and R-410A at compressor sump during the system operation, 30-40 F OD ambient](image)

IV. Viscosity and oil dilution percentage measurements of a compressor in an Ice Machine system have been done in the Copeland Refrigeration Lab during actual operation. The instantaneous dilution percentage values (time history data) have been evaluated using also the numerical modeling, Figure 5. Electro-Mechanic technology was used in this test.

![Figure 5: Viscosity and oil dilution % (AK Mobil Oil and R404A) measurements, ambient and water temperature 110F, 100F (43.3 C, water 37.8 C) respectively](image)
8. CONCLUSIONS

Ideally, the lubricant/refrigerant mixture has sufficient miscibility or mutual solubility to allow the lubricant to flow with the liquid refrigerant and return to the compressor. Even if the lubricant/refrigerant pair are not miscible (two liquid phases form) in the evaporator, they may still have some degree of solubility. Solubility of refrigerant in lubricant lowers lubricant viscosity, which helps it flow through the evaporator and return to the compressor. This is why many refrigeration systems can operate properly, even though the lubricant and refrigerant are immiscible enough (yet partially soluble) at evaporation temperatures. Other factors, such as refrigerant vapor velocity and system geometry as well as system TXV, play key roles in lubricant return. Overall, it is important to note that lubricant/refrigerant miscibility is helpful, but not necessarily essential for proper system operation.

The R-410A and R-22, R-404A refrigerants vapor solubility in liquid POE Oil, White Oil, AK-Mobile Oil has been studied at various HVAC thermodynamic cycle operating conditions in this paper. The results show that the vapor solubility is high for higher temperatures, and and higher refrigerant charges. Reducing the refrigerant charge of the system or/and the temperature of a vapor in the compressor sump with minimizing the liquid refrigerant present in system dramatically raises the viscosity. However, many hermetic system applications frequently have excess refrigerant returned to the compressor sump. Under these conditions, one would expect to have excess refrigerant charge for a fixed lubricant charge in the system. Some of the lubricant may also be trapped somewhere in the system (such as accumulator). Therefore, the viscosity of mixture in the compressors will reduce and affect on the life reduction of the compressor due to lack of lubrication. However, the physical and miscibility properties of lubricants change when at high refrigerant concentrations at both ends of the temperature scale. It means at low temperature end the sump overhead pressure is small value, but at high temperature end the sump overhead pressure is large value. This is an important factor in the selection of systems operation conditions.

9. REFERENCES

Implementation for inverter controlled domestic refrigerator/freezer with brushless DC reciprocating compressor

Wen-Ruey Chang*, Der-Yeong Liu, Jian-Yuan Lin, Yu-Choung Chang
Energy and Resource Laboratories (ERL), Industrial Technology and Research Institute (ITRI), Republic of China

ABSTRACT

The novel inverter-controlled household refrigerator/freezer (HRF) has a significant energy-efficiency potential and also has benefit of low noise operation at a slow rotating speed. This paper describes the development of an inverter-controlled domestic fridge, sold in Taiwan, with a reciprocating compressor of refrigerant HFC-134a that is driven by a brush-less DC motor. The refrigeration capacity of the BLDC compressor is ranged from 100 to 300W, corresponding to 1500 to 4200 rpm respectively under the specified freezing temperature condition of -18°C. Some advantages of inverter-controlled HRFs are discussed here in details and these were finally implemented by the application of control methodology for this kind of refrigeration system and its auxiliary components, including compressor, inverter, fan motor, and air circulation damper of refrigeration compartment. The energy consumption and compartment temperature variations at different locations of this studied fridge were measured to evaluate the volumetric efficiency following the testing standard of CNS-2062 used in Taiwan with fixed ambient temperature of 30°C. One fridge of fixed frequency was tested as performance comparison, which had all the same specification, e.g. inner volume, refrigerating circuit, thermal insulation, other than using an AC compressor of almost the same piston displacement. Under some the specification testing procedure, the inverter-controlled fridge has over 35% improvement of energy efficiency to that of the fixed frequency product. There is also a smaller temperature variation for the cabinets of the inverter-controlled fridge.

1. PREFACE

To increase energy utilization and minimize carbon dioxide emission, some governments around the world are tending to establish increasingly strict regulation on energy consumption baseline for household refrigerators/freezers (HRF). Some methods are useful to reduce energy consumption for HRF, including with increasing thermal insulation thickness for walls, improvement of thermal insulation material or applying vacuum insulation panels, using a high-performance compressor, and optimization of refrigeration system etc. To reduce energy
consumption greatly for HRFs, the novel inverter-controlled technology for variable speed compressor with brush-less DC motor has been paying attention in recent years, especially in Japan. The energy-saving potential was evaluated about 30 to 40% in some published literatures and press medium. The DC motor inside the variable-speed compressor is driven by a digital signal processing (DSP) controller which is combined with high level micro-processor, sensor-less power electronic circuits, six-step driving transistors for pulse width modulation (PWM) regulation, switching power supplier, and some peripheral communication circuits. The compressor is driven by the previous-described printing circuit device, named as an inverter, integrated with other auxiliary electrical parts, like fan-motor, door switches, relays, lamp, damper, defrosting electrical heater etc. The start-up and sequent control commands are carried out following the pre-verified ROM built in this inverter. This product is generally named as variable frequency, or inverter-controlled HRF’s in Asia area. The refrigerant circulating flow rate of the compressor is regulated with magnetic intensity of the brush-less DC motor through six-step MOS transistors and the other circuits, sometimes these six transistors are implemented commercially as an integral power module used in high power home appliances over 1 kW, like air-conditioners. The thermal load requirement of a HRF is always changed with environmental variation. This type of variable speed compressors can regulate the optimized refrigeration capacity easily, and also guarantees the stable temperature distribution in the storage compartments to keep fresh for food.

A traditional HRF uses a reciprocating compressor with AC induction motor and start-up/running capacitances. The electric current impulse and sharp noisiness during the start-up period of an AC HRF always bother the customers and manufactures. These phenomena are accompanied with the compressor to overcome the friction and inertia from stationary state. The inverter-controlled compressor can be running up without heavy power load through continuous increment of magnetic intensity of the DC motor to avoid these problems. For an AC HRF, the temperatures of storage compartments are maintained to stable by turning on/off for the compressor and the thermostat with mechanical air-circulation damper, for what the variation is sometimes too large to assure of freshness of food. Within an acceptable range of setting temperature, the compressor turns on and off following the signal of thermostat relay inside refrigeration/freezer cabinet. For this on/off control method by using AC compressor, the HRF’s temperature fluctuates and the electricity energy is consumed high than those by inverter-controlled type generally. Inverter-controlled HRF’s can adjust refrigeration capability through compressor speed regulation; so the main power turning on/off frequency could be greatly reduced, and the energy consumption could be also decrease due to increase of refrigeration cycling efficiency.

From the second law of thermodynamics, the coefficient of performance for one refrigeration system will be improved as by continuous running, if possible, compared to the intermittent on/off operation under the same condition. The cumulative energy consumption of each cycling loss could occupy a large percentage for the whole-day energy usage of a HRF. The storage temperature of a fixed-frequency refrigerator/freezer is maintained by turning on/off of its AC compressor. Such a control method causes thermodynamic cycling loss and reduces its energy efficiency. At starting state of the compressor, the system needs to re-establish the pressure potential between its condenser and evaporator, to reach the normal conditions for refrigeration. Coulter et al. [1] studied the starting and steady state cycling losses of refrigerators/freezers experimentally at room temperature from 60 to 100°F. The starting operation caused refrigeration loss of system by 3 to 17%, the energy consumption increased by 1 to 9% and the COP reduced by 5 to 25%. Wicks [2] indicated that the variable speed control of compressor would reduce temperature variation than that by on/off control for a refrigerating system. For
one system with refrigeration capability rated by 12000 Btu/hr operating under partial load of 6000 Btu/hr, variable speed control would save more energy than that by on/off control by 41% from the analysis of thermodynamic second law. On the other hand, energy consumption of refrigerators/freezers would be decreased by reduction of pressure potential cross compressor’s discharge and suction lines. From the study by Woodall et al. [3], the energy efficiency of refrigerators/freezers improved by 10.5% due to decrease of temperature difference between the condenser and evaporator, as the rotation speed of compressor decelerating from 3600 rpm to 2400 rpm under steady-state operation. Recently, Liu and Chang et al. [4] studied the effect of door openings of refrigerators/freezers on the energy consumption and compartment temperature variation by experimental approach. The energy consumption of refrigerators/freezers with door opening was found to increase by 10%. Without door opening, the static test indicated that the variable frequency HRF saved more energy than the fixed frequency HRF by 22%. With door opening test, the variable frequency HRF saved more energy than the fixed frequency HRF by 25%. In 2004, Chang et al. [5] discussed some control methods for variable frequency refrigerators/freezers, including the temperature control method, defrosting heater control method and door opening control method. Depending on additional functional requirements, more control methods can be established and flashed as machine codes in the DSP micro-controller.

2. IMPLEMENTATION

Following the previous article [5] for the control strategy of inverter-controlled HRFs, this paper will focuses on the implementation and performance testing results of one variable frequency HRF developed in Taiwan, and also shows comparison with a fixed frequency with the same inner storage volume and geometry specification made by the same manufacturer. In this paper, we will use “VF” and “FF” standing for the variable and fixed frequency HRFs respectively. The rated gross volume is 560 liters for these two HRFs with the same total freezing volume of 133 liters, refrigerated volume of 309 liters, and vegetable compartment of 118 liters, as shown in Figure 1(a). The high-pressure polyurethane foaming of HCFC-141b was used for the wall thermal insulation. Three doors for these three compartments were designed with a top-mounted freezer/evaporator, and a drawer type was used for the vegetable-fruit compartment at the bottom of the HRFs. The cold air is brought to the refrigeration compartment from the evaporator through one baffle-type damper, which may be driven by a spring-mechanism or by step-motor tracing the thermostat. The required power source is 110V/60Hz for commonly household usage in Taiwan and the electric power for defrosting heater is specified as 200W for AC current. Two type of reciprocating compressors with the same displacement volume of 7 cubic centimeters and all for refrigerant HFC-134a were applied in the studied HRFs, for VF and FF types. The VF type of the compressors is driven by a brush-less DC motor, which the variable speed control was implemented by using MOS-FET transistors, sensor-less circuiting, the DSP MCPU coding, and the other auxiliary components, as shown in Figure 1(b). A fixed frequency HRF is cooled by AC compressors with on/off turning control for maintaining the storage temperature to some specified values, e.g. 3°C and -18°C for refrigerating and freezing compartments. Following the previous description, the FF HRF has some features inferiorly to the VF HRF, including the storage temperature variation, noisy during compressor intermittent running, and the power consumption.

The photograph of the VF hermetic reciprocating compressor is shown in Figure 1(c). In this study, the VF compressor was tested by a caloric meter system of refrigerant HFC-134a before
the implementation of the VF HRF, following the test standard [7] for refrigerant compressors under different rotation speed and suction pressures. The refrigeration capacity increases proportionally to the rotation speed and also to the suction pressure as shown in Figure 2. The energy efficiency ratio (EER) was also evaluated from the previous refrigeration capacity dividing by the instant power capacity under the specified condition. The EER values range from 1.1 to 1.27 kcal/hr/W under the suction pressure of 1.2 kgf/cm². The power capacity of the inverter controller was included in the measurement of the power of the VF compressor.

On the other hand, the refrigeration capacity of the FF compressor is rated as 202.7 kcal/hr with power capacity of 185.6 W, EER of 1.092 kcal/hr/W, refrigerant mass flow rate of 5.009 kg/hr, and discharge temperature of 97.3°C under the conditions of condensation temperature T_{cond} of 43.3°C (P_{cond}=11.3 kgf/cm²) and evaporation temperature of -23°C (P_{evap}=1.2 kgf/cm²), in the same caloric meter system supplied with 110V/60Hz power source. From the comparison of EER values, the VF compressor has a wide-range application and retains higher efficiency than an AC compressor with potential increase of 6% to 15%.

![Figure 1. The variable frequency HRF developed in Taiwan.](image)

To reach the best efficient with user-friendly function for customers, the inverter-controlled refrigerator/freezer can be implemented by hardware design for the printed circuit board and program coding for the specified control strategy. Hardware design is including with the power converter from AC source to DC, power transistors for DC motor, sensor-less circuit, and some auxiliary circuits for thermostat, fan, doors, defrosting heater, lamp, etc., used commonly in refrigerators/freezers. The detail control strategy for inverter-controlled household refrigerators/freezers (HRFs) can be referred to Chang et al. [5].

The energy efficiency potential and storage temperature performance were studied experimentally in this paper for the VF and FF refrigerators/freezers, which were made in 2003 by local manufacturer in Taiwan. The testing procedure used here is following CNS-2062 standard [6] used in Taiwan, similar to ISO 8561 [8] and ANSI/AHAM HRF-1 [9]. The room ambient temperature is fixed at 15 and 30°C, standing for the winter and summer conditions respectively. The relative humidity is controlled at 75%, and the storage temperature is set at -18±0.5°C for frozen food compartment and ±0.5°C for fresh food compartment. The VF and FF samples were tested under the same conditions concurrently in an environmental controlled...
room for several days. Several T-type thermocouples with pre-calibrated uncertainty of 0.2°C and also the electric power consumption uncertainty of 3% were applied in this study. The measurement data were logged per every 5 seconds a time by a hybrid recorder and power meters. From the time tracing results, one can evaluate the product performance or modify the control commands in the system controller.

![Figure 2. Refrigeration capacity of the VF compressor for different rotation speeds.](image)

![Figure 3. Energy efficiency ratio of the VF compressor for different rotation speeds.](image)

Some important specifications and the thermal performance are tabulated in Table 1 from the experimental results, where the VF and FF samples are represented for the studied HRFs and the average storage temperatures for freezing and refrigeration compartments were kept for the requirement of CNS-2062 [6]. The temperature variation for storage compartments was evaluated from the difference of the maximum and minimum temperature values of thermometers located in some specified positions during the testing period. The temperature variation of freezer of ±2.8°C for the VF sample is much smaller than that of FF sample, and similarly for the smaller temperature variation of refrigeration compartment for the VF HRF. The electric power consumption was measured in a whole 24-hour period including one defrosting operation. The daily energy consumption for the VF HRF is 1.66 kWh lower to the FF HRF of 2.255 kWh. There is an energy efficiency parameter defined in CNS-2062 [6] as the

---

287
energy factor EF. The energy factor of the VF HRF is 13.3 liters/(kWh/month) with about 36% efficient potential higher to that of the FF HRF under the same close-door testing procedure. Another parameter for refrigerators/freezers is the running percentage of the compressor, can be defined as the running time period divided by the operating period for the compressor during one defrosting cycle, where the operating period is the sum of the turning on and off period for the compressor. The running percentage for VF HRF is larger than the FF HRF, and the larger this running percentage will increase the energy efficiency for the brush-less DC compressor.

Table 1. The specifications and performance characteristics of the HRF samples in this study.

<table>
<thead>
<tr>
<th>Items</th>
<th>Fixed-Frequency Type (FF HRF)</th>
<th>Variable-Frequency Type (VF HRF)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rated Gross Volume (liters)</td>
<td>560</td>
<td>560</td>
</tr>
<tr>
<td>Freezing Storage Volume (liters)</td>
<td>133</td>
<td>133</td>
</tr>
<tr>
<td>Refrigeration Storage Volume (liters)</td>
<td>427</td>
<td>427</td>
</tr>
<tr>
<td>Equivalent Gross Volume (liters)</td>
<td>667.7</td>
<td>667.7</td>
</tr>
<tr>
<td>Average Temperature for Freezer Compartment, ( T_{F,\text{avg.}} ) (°C)</td>
<td>-18.1</td>
<td>-18.1</td>
</tr>
<tr>
<td>Variation for transient ( T_{F} ) (°C)</td>
<td>±4.1</td>
<td>±2.8</td>
</tr>
<tr>
<td>Average Temperature for Refrigeration Compartment, ( T_{R,\text{avg.}} ) (°C)</td>
<td>3.2</td>
<td>3.4</td>
</tr>
<tr>
<td>Variation for transient ( T_{R} ) (°C)</td>
<td>±2.6</td>
<td>±1.7</td>
</tr>
<tr>
<td>Energy Consumption (kWh/day)</td>
<td>2.255</td>
<td>1.660</td>
</tr>
<tr>
<td>Energy Factor value, EF (liters/(kWh/month))</td>
<td>9.81</td>
<td>13.30</td>
</tr>
<tr>
<td>Energy Efficiency Potential</td>
<td>1</td>
<td>1.36</td>
</tr>
<tr>
<td>Running time/Operation Time in percentage for the compressor during one defrosting cycle</td>
<td>54.37 %</td>
<td>72.57 %</td>
</tr>
</tbody>
</table>

Remarks:
1. The uncertainty was 0.2 °C for temperature, 3% for electric power consumption, 4.7% for energy factor, EF of standard CNS-2062 used in Taiwan. All the testing procedures and the evaluation of corresponding uncertainties were following the quality management for laboratory requirement by ISO-17025.
2. The energy factor, EF is defined as the equivalent gross volume divided by the energy consumption, which is obtained by testing result of 24-hour electric power consumption under the standard procedure of CNS-2062 multiplied by 30 days.

3. PERFORMANCE COMPARISON

Temperature variation for storage compartments of HRFs is an important index for consumers, for what will affect the storage quality of food. Figure 4 shows the temperature variations of freezer(\( T_{F} \)) as well as the refrigeration(\( T_{R} \)) compartments for the fixed frequency type of HRF during a 24-hr testing period. There was encountering one time of defrosting operation for the evaporator coil in a whole daily period. The ambient temperature for test was set at 30°C, and the relative humidity at 75% in this and also the coming cases. The average temperature for the
refrigeration compartment, $T_{R\text{, avg.}}$, is 3.2°C with variation of ±2.6°C, and the average temperature for the freezing compartment, $T_{F\text{, avg.}}$, is -18.1°C with variation of ±4.1°C. So, the highest temperatures for the refrigerator and freezer would be above 5.8°C and −14.0°C respectively, for what maybe give taint to the food for long-term storage. There would be reaching freeze point as the refrigerator being cooled down to the lowest temperature of 0.6°C. Usually, there is not any auto-controlled compensation mechanism for the storage temperature of a fixed frequency HRF sold in Asia. The vegetable and fruit in the cabinet would be frozen sometimes during the wind chill of winter season.

Compared to the traditional fixed frequency (FF) HRF, the variable frequency (VF) HRF performs with stable storage temperatures as shown in Figure 5. The average temperature for the refrigeration compartment, $T_{R\text{, avg.}}$, is 3.4°C with variation of ±1.7°C, and the average temperature for the freezing compartment, $T_{F\text{, avg.}}$, is -18.1°C with variation of ±2.8°C. The highest temperatures for the refrigerator and freezer are about 5.1°C and −15.3°C respectively. The temperature variation of a DSP inverter-controlled HRF can be shrunk to the lowest after several tests and modification for the control commands, chill air-circulation, operation of the refrigeration damper etc.

From the transition of storage temperatures, saw-toothed variant curves are detected to describe the truth that whether VF or FF refrigerators/freezers, unstable thermo-state exists especially for the FF HRF. To reduce this storage temperature variation, soft-switch design for the AC compressor should be considered for FF one which would increase the cost. For VF HRF, this problem has been reduced and can be minimized by using modification of control codes without addition of mass-production cost. We know the advantage from the previous discussion that the energy efficiency has been improved by 36% as the best choice for a variable frequency HRF. The additional cost for VF HRFs would be overcome through all the benefits for consumers.

As the studied refrigerators/freezers started from the stationary state with all the ambient and cabinet temperatures being of 30°C, what named as a pull-down test, it took about 1hr42min to reach the setting storage temperature of CNS-2062 [6] for the VF HRF as indicated in Figure 6.
The discharge temperature of the VF compressor reached the maximum temperature about 95°C, and then decreased to the temperature below 70°C under a normal operation. That has several benefits for the compressor as well as its corresponding refrigeration system, especially to increase the lifetime of the product. For the fixed frequency type, it took about 2hr09min to pull-down to the setting temperature, and the discharge temperature of the compressor was almost round 100°C. As the refrigeration system reached to a normal state, this discharge temperature reached above 90°C, for what would be imperfective anywise as contrasted with the VF type. For the suction line temperature, the VF and FF HRFs had almost the same value under the pull-down operation, and the amplitude of temperature variation for the FF type is much larger than that of the VF type. The details of temperature variations for these compressors are shown in Figure 6, where the \( T_{\text{comp, out}} \) and \( T_{\text{comp, in}} \) represent the discharge and suction temperatures respectively. The logging period of the data acquisition system was also maintained as 5 seconds for each datum point.

![Figure 5 Temperature variations of the VF HRF during a 24-hr testing period.](image)

![Figure 6 Discharge/suction temperature variations of the compressors for VF and FF HRFs.](image)
4. CONCLUSIONS

In recent years, the price for digital signal processing micro-controller has been gradually dropping and gaining market competitive advantage. Therefore, home appliances can realize the idea of digital control. With the same mechanical and electrical configuration, many complicated control methods only need to expand software program and memory to process peripheral signals or drive key components in appliances, such as BLDC motor in compressor, fan motor and electric heater, so the need of optimization can be satisfied. Even by software upgrade, the newest control program can be copied to appliance through communication port. Consumers can enjoy updated and more convenient services. Development for inverter-controlled refrigerator can be considered as a revolutionary advancement for white appliances. It not only operates with lower noise, stable storage temperatures, fast freezing, freshness preservation and energy saving. Variable frequency control technology even makes truth for the smart refrigerators/freezers. It adjusts compressor speed and puts control over other components according to various conditions. A variable frequency refrigerator/freezer needs some proper control methods to obtain good performance. It also needs integration among all components to achieve optimization in speed and temperature control under different input conditions.

This paper introduced some benefits of inverter-controlled refrigerators/freezers and also described some implementation experience from the performance tests and their illustration for AC and DC compressors as well as the close-door test of domestic refrigerator/freezer. The advantage of DC compressors can be explored from the comparison of the refrigeration capacity with excellent efficiency over large application range by contrast with the AC compressor. Due to the caloric testing procedure of refrigeration compressors, the brush-less DC compressor behaves with higher efficient potential of 6 to 15% compared to the AC type one under the same testing conditions. As implementation for an inverter-controlled refrigerator/freezer, the energy factor, EF, of the VF HRF is higher than the FF type by up to 36%, much more than the potential comparison of compressors. The thermal stability is another advantage for the development of VF HRFs confirmed from the performance testing results in this study. The temperature variation of the refrigeration and freezing compartments are all reduced obviously for the change from the FF HRF to the VF type. The running percentage of the compressors was also evaluated and the larger of this running percentage will increase the energy efficiency for the brush-less DC compressor. Finally, for the pull-down test, the discharge temperature of the compressor of the VF HRF was much lower than that of the FF HRF under the same ambient condition, and the suction temperature behaved with near the same magnitude.

5. ACKNOWLEDGEMENT

The authors would like to thank the Bureau of Energy, Ministry of Economic Affairs, Taiwan, Republic of China, for the support of this study under the Energy R&D funding.
6. REFERENCES


Thermal and fluid dynamic behaviour of a trans-critical carbon dioxide small cooling system: Experimental investigation

G. Raush, J. Rigola, C. D. Pérez-Segarra, A. Oliva
Centre Tecnològic de Transferència de Calor (CTTC)
Laboratori de Termodinàmica de Catalunya (UPC), Spain

J. M. Serra, M. Escribà, J. Pons, J. Jover
ACC Spain, S.A., Spain

A first trans-critical carbon dioxide hermetic reciprocating compressor prototype has been designed, numerically optimised, built and experimentally tested. Different heat exchangers have been specially developed to be used under trans-critical carbon dioxide cycles. An experimental unit has been designed and constructed to work with carbon dioxide as fluid refrigerant allowing the experimental study proposed and the validation of the numerical results obtained from the companion paper. After the numerical parametric results and the experimental comparative data obtained, a semi-hermetic reciprocating compressor has been designed, tested and compared, locating the suction plenum and cylinder head in contact with ambient.

The first objective of this work has been to test inside the trans-critical cycle, carbon dioxide hermetic and semi-hermetic compressors, taking into account a special shell, a different configuration muffler, an specific stroke to bore ratio in the compressor chamber, and an increased and reinforced cylinder head. In both cases, the valve plate, the suction and discharge valves, the motor and the crankshaft connecting rod mechanism has also been necessary modified. The heat exchangers have also been adapted to carbon dioxide properties and high gas cooler pressures.

The second objective is to experimentally validate the numerical simulation results obtained in the companion paper, in order to assure a real virtual laboratory prototype when numerical simulation tool developed and adapted to carbon dioxide is used.

The following aspects presented in this paper are: i) the optimal gas cooler pressure under the desired working conditions; ii) the experimental validation of the double pipe water counter flow heat exchangers (gas cooler and evaporator); iii) the global transcritical cycle comparison between the numerical study and the experimental data under different evaporation temperatures for the hermetic compressor; and iv) the experimental comparison between both hermetic and semi-hermetic CO₂ reciprocating compressors.

1 INTRODUCTION

Trans-critical carbon dioxide cycles had a revival after 1992 as a natural fluid refrigerant as heating and cooling alternative systems [1] [2] and [3]. In 2004, Kim et. al presented the most important review of fundamental process and system design issues in carbon dioxide systems [4], where the important trends in carbon dioxide technology in refrigeration, air-conditioning and heat pumps applications are provided.
Since 1998, different experimental carbon dioxide refrigerating systems in general [5] [6] and commercial refrigeration in particular [7] have been presented in the technical literature. Other experimental systems like mobile air-conditioning [8] [9], heat pump water heaters [10] [11] and heat pumps for residential applications [12] have also been presented. Together with this, different hermetic reciprocating compressor prototypes are also experimentally referenced [13] [14] [15] and [16].

The aim of this work is to present a hermetic reciprocating compressor carbon dioxide prototype numerically studied in the companion paper, and its experimental behaviour in a whole trans-critical cycle to work in similar conditions as a small cooling application, in comparison with a new semi-hermetic compressor prototype. The optimum gas cooler pressure under specific working conditions is also experimentally studied. The heat exchangers behaviour and the whole cycle are also experimentally validated in comparison with the numerical results of the companion paper. Finally, both hermetic and semi-hermetic compressors are experimentally compared.

2 CARBON DIOXIDE CYCLE EXPERIMENTAL SET-UP

An experimental set-up has been specially designed to evaluate the thermal and fluid dynamic behaviour of carbon dioxide trans-critical cycles and to validate the numerical simulation results of the numerical model presented in the companion paper.

A schematic diagram and a general view of the refrigeration system are depicted in Figures 1 and 2, respectively. The experimental unit is made up of the following elements: a one stage carbon dioxide hermetic reciprocating compressor prototype, dual heat transfer coil gas cooler and evaporator together with a metering valve. The auxiliary fluid used in the gas cooler and the evaporator annuli is water. Tables 1 and 2 show the cycle components and instrumentation elements parameters, respectively. Two thermostatic heating and cooling units control the inlet auxiliary water temperature in the condenser and evaporator auxiliary circuits, respectively. The volumetric flow in these secondary circuits is controlled by two modulating solenoid valves and measured by means of two magnetic flow-meters, with an accuracy of ± 0.01 l/min. from 0 to 2.5 l/min., and ± 0.5 % F.S. from 2.5 l/min. to 25 l/min. Compressors HCL15 and SHCL15 are the CO2 compressor prototypes.
3 OPTIMUM GAS COOLER PRESSURE CARBON DIOXIDE TRANS-CRITICAL CYCLE

The compressor cycle has been designed for a cooling capacity around 600W, under 0°C of evaporation temperature and inlet compressor gas temperature and outlet gas cooler temperature of 35°C. Once the main working conditions are decided, it is necessary to know the optimal gas cooler pressure, whose value presents a maximum COP depending on compression pressure ratio.

Figure 3 presents the experimental results obtained from hermetic compressor experimental test of section 2, under the working conditions detailed above, considering three different evaporation temperatures of -10, 0 and +10°C. Experimental results show how in all cases presented the COP has a maximum value around 90 bar of gas cooler pressure. It is interesting to remark that all global experimental values tend to decrease after the maximum value, although the slope increases when evaporation temperature decreases.
4 CARBON DIOXIDE HEAT EXCHANGERS COMPARATIVE RESULTS

Based on the numerical simulation model of the thermal and fluid dynamic behaviour of double pipe heat exchangers, different comparative results have been obtained when the numerical simulation model is compared with the experimental data obtained, considering only the heat exchangers phenomena through the experimental set-up described in section 2.

The results of Figure 4 are the numerical results of gas cooler and evaporator working under the same conditions of the results obtained in Figure 3, with three different evaporation temperatures of -10°C, 0°C and +10°C, and different gas cooler pressures that range from 75 to 110 bars.

The numerical results show how differences on CO_2 outlet gas cooler and evaporator temperature are lower than 5% in all studied cases, equal than water outlet auxiliary fluid temperatures. It is interesting to highlight that correlations used always under-predict the experimental data.
In order to numerically compare and experimentally validate the numerical results and the experimental data, it is necessary to obtain a numerical parametric study of the hermetic compressor behaviour to be used in the whole simulation system. Figure 5 shows the numerical results of the compressor behaviour under different working conditions. The numerical information obtained is used in the simulation of the whole refrigerating cycle. It is interesting to highlight the influence of the evaporation temperature in the different evaluated parameters (e.g. volumetric efficiency, isentropic efficiency and mechanical-electrical efficiency). This influence is different than conventional compressors working with R134a, which non-dimensional parameters are approximately only a function of pressure ratio.

Together with Figure 5 results, it is necessary additional information about the heat transfer shell losses. This additional parameter is defined as $\eta_{Q_{sh}} = 1 - \dot{Q}_{sh}/\dot{W}_e$. More detailed information about this non-dimensional parameter and all detailed thermodynamic characterisation of these compressors is shown in [17]. In the studied cases, the heat transfer shell losses are around 42, 59 and 74% for evaporation temperatures around -10°C, 0°C and 7.2°C, respectively.

Table 3 shows the numerical results vs. experimental data of the most relevant refrigerating cycle variables (pressure, temperature and mass flow rate). The values in brackets are the numerical boundary conditions, compared with the experimental data. Five temperatures are shown in Table 3, which represent outlet compressor temperature, inlet and outlet gas cooler temperature, inlet evaporation temperature, and inlet compressor temperature, respectively, together with the inlet evaporator vapour quality ($x_{ev}$).

<table>
<thead>
<tr>
<th>results</th>
<th>$P_{gc}$ (bar)</th>
<th>$P_{ev}$ (bar)</th>
<th>$T_2$ (°C)</th>
<th>$T_3$ (°C)</th>
<th>$T_4$ (°C)</th>
<th>$T_6$ (°C)</th>
<th>$T_8$ (°C)</th>
<th>$x_{ev}$</th>
<th>$m$ (Kg/h)</th>
</tr>
</thead>
<tbody>
<tr>
<td>numerical</td>
<td>89.71</td>
<td>25.43</td>
<td>136.17</td>
<td>128.71</td>
<td>35.23</td>
<td>-11.37</td>
<td>34.35</td>
<td>0.489</td>
<td>6.54</td>
</tr>
<tr>
<td>experimental</td>
<td>89.71</td>
<td>25.48</td>
<td>127.29</td>
<td>115.39</td>
<td>35.24</td>
<td>-11.30</td>
<td>35.38</td>
<td>0.471</td>
<td>6.49</td>
</tr>
<tr>
<td>numerical</td>
<td>90.37</td>
<td>32.52</td>
<td>130.49</td>
<td>126.01</td>
<td>35.35</td>
<td>-2.53</td>
<td>34.51</td>
<td>0.488</td>
<td>10.42</td>
</tr>
<tr>
<td>experimental</td>
<td>90.37</td>
<td>32.97</td>
<td>126.00</td>
<td>118.45</td>
<td>35.23</td>
<td>-2.03</td>
<td>35.62</td>
<td>0.433</td>
<td>10.41</td>
</tr>
<tr>
<td>numerical</td>
<td>90.14</td>
<td>45.00</td>
<td>105.56</td>
<td>103.75</td>
<td>36.40</td>
<td>10.03</td>
<td>34.44</td>
<td>0.415</td>
<td>18.45</td>
</tr>
<tr>
<td>experimental</td>
<td>90.14</td>
<td>44.40</td>
<td>96.87</td>
<td>94.03</td>
<td>36.45</td>
<td>9.48</td>
<td>35.42</td>
<td>0.412</td>
<td>18.42</td>
</tr>
</tbody>
</table>

Differences are lower than 10% for all compared variables. These differences are the same as in all general cases previously compared when numerical simulation model of hermetic reciprocating compressors and refrigerating cycles were studied using R12, R134a, R600a, etc. It is interesting to highlight that compressor outlet temperature is always over-predicted, while differences in mass flow rate are lower than 1% for the three studied cases.
Table 4 shows the global comparative values under the three experimental cases considered. Results of power consumption \( (W_e) \) and cooling capacity \( (Q_{ev}) \) shows a good agreement with lower differences in the global parameters than the differences obtained in Table 3.

<table>
<thead>
<tr>
<th>results</th>
<th>( P_{gc} ) (bar)</th>
<th>( P_{ev} ) (bar)</th>
<th>( T_{ev} ) (C)</th>
<th>( m ) (Kg/h)</th>
<th>( W_e ) (W)</th>
<th>( Q_{ev} ) (W)</th>
<th>COP (bar)</th>
</tr>
</thead>
<tbody>
<tr>
<td>numerical</td>
<td>(89.71)</td>
<td>25.43</td>
<td>−11.37</td>
<td>6.54</td>
<td>319.02</td>
<td>348.46</td>
<td>1.092</td>
</tr>
<tr>
<td>experimental</td>
<td>(89.71)</td>
<td>25.48</td>
<td>−11.30</td>
<td>6.49</td>
<td>326.47</td>
<td>351.65</td>
<td>1.077</td>
</tr>
<tr>
<td>numerical</td>
<td>(90.37)</td>
<td>32.53</td>
<td>−2.53</td>
<td>10.42</td>
<td>340.57</td>
<td>531.10</td>
<td>1.559</td>
</tr>
<tr>
<td>experimental</td>
<td>(90.37)</td>
<td>32.97</td>
<td>−2.03</td>
<td>10.41</td>
<td>343.65</td>
<td>535.11</td>
<td>1.557</td>
</tr>
<tr>
<td>numerical</td>
<td>(90.14)</td>
<td>45.00</td>
<td>10.03</td>
<td>18.45</td>
<td>343.49</td>
<td>818.82</td>
<td>2.383</td>
</tr>
<tr>
<td>experimental</td>
<td>(90.14)</td>
<td>44.49</td>
<td>9.48</td>
<td>18.42</td>
<td>335.11</td>
<td>832.70</td>
<td>2.485</td>
</tr>
</tbody>
</table>

Comparative results show a COP around 2.4 and a cooling capacity of 800W under an evaporation temperature of +10°C. The COP is around 1.55 and a cooling capacity of 530W under an evaporation temperature of 0°C. Although the results present lower COP if compared with conventional R134a compressors, the comparative values are promising results taking into account that the compressor can be improved if the clearance ratio and mass flow leakage are reduced under new prototypes.

6 CO\(_2\) HERMETIC AND SEMI-HERMETIC COMPRESSORS EXPERIMENTAL COMPARISON

Once the carbon dioxide hermetic reciprocating compressor has been numerically analysed, optimised and experimentally tested, a semi-hermetic compressor has been built following the same strategy, taking the advantage to design a direct suction, and a cylinder head in contact with the ambient. Both changes have allowed to obtain the improvements detailed in this paragraph.

Table 5 shows the main CO\(_2\) semi-hermetic compressor prototype parameters of the Fig. 6. The hermetic reciprocating compressor compared has been detailed in the companion paper and numerically validated above.

<table>
<thead>
<tr>
<th>Table 5: Crankcase compressor main parameters.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>SHCL15 (semi-hermetic reciprocating compressor)</strong></td>
</tr>
<tr>
<td>( D_{inlet} ) (5.0 mm)</td>
</tr>
<tr>
<td><strong>Suction line</strong></td>
</tr>
<tr>
<td>Shell ( 3350 \text{ cm}^3 )</td>
</tr>
<tr>
<td><strong>Crankcase</strong></td>
</tr>
<tr>
<td>bore diameter ( 14.0 \text{ mm} )</td>
</tr>
<tr>
<td>clearance volume ( 5.42% )</td>
</tr>
<tr>
<td>suction stop ( 0.8 \text{ mm} )</td>
</tr>
<tr>
<td>diameter (1) ( 3.2 \text{ mm} )</td>
</tr>
</tbody>
</table>

Fig. 6: Carbon dioxide semi-hermetic compressor.
Table 6 shows the experimental comparative results between the hermetic reciprocating compressor numerically tested and experimentally validated, with the new semi-hermetic compressor designed taking into account all optimised studies developed for the hermetic reciprocating compressor and taking advantage of the external ambient exchange in the direct suction plenum and discharge cylinder head.

The three experimental comparative cases have been carried out considering a gas cooler pressure around 90 bar, an inlet compressor temperature and outlet gas cooler temperature of 35°C and evaporation temperatures around -10°C, 0°C and 7.2°C.

<table>
<thead>
<tr>
<th>type</th>
<th>$P_{gc}$ (bar)</th>
<th>$T_{gc}$ (°C)</th>
<th>$T_{evap}$ (°C)</th>
<th>$m$ (Kg/h)</th>
<th>$W_c$ (W)</th>
<th>$Q_{evap}$ (W)</th>
<th>COP</th>
</tr>
</thead>
<tbody>
<tr>
<td>HCL15</td>
<td>89.71</td>
<td>-11.30</td>
<td>127.1</td>
<td>6.49</td>
<td>346.5</td>
<td>351.6</td>
<td>1.077</td>
</tr>
<tr>
<td>SHCL15</td>
<td>89.19</td>
<td>-9.05</td>
<td>103.8</td>
<td>9.30</td>
<td>369.2</td>
<td>499.2</td>
<td>1.352</td>
</tr>
<tr>
<td>HCL15</td>
<td>90.37</td>
<td>-2.03</td>
<td>126.0</td>
<td>10.41</td>
<td>343.6</td>
<td>535.1</td>
<td>1.557</td>
</tr>
<tr>
<td>SHCL15</td>
<td>89.58</td>
<td>0.11</td>
<td>98.2</td>
<td>14.04</td>
<td>380.0</td>
<td>714.0</td>
<td>1.881</td>
</tr>
<tr>
<td>HCL15</td>
<td>90.14</td>
<td>7.12</td>
<td>115.0</td>
<td>15.30</td>
<td>352.0</td>
<td>716.0</td>
<td>2.034</td>
</tr>
<tr>
<td>SHCL15</td>
<td>89.96</td>
<td>7.72</td>
<td>90.5</td>
<td>18.39</td>
<td>367.6</td>
<td>883.1</td>
<td>2.402</td>
</tr>
</tbody>
</table>

Based on linear interpolation/extrapolation between Table 6 results to obtain closer values at exactly -10°C, 0°C and 7.2°C, it is possible to show that that outlet compressor temperature decreases considerably and the mass flow rate increases between 7 and 20% at 7.2°C and -10°C evaporation temperatures when the semi-hermetic compressor is considered. Under the same considerations, the power consumption increases between 8 and 10%, while the cooling capacity increases between 14 and 20%. Thus, the Coefficient of Performance of the semi-hermetic compressor increases from 6 and 12% for evaporation temperatures from 7.2°C to -10°C, respectively.

7 CONCLUSIONS

An experimental study of a trans-critical carbon dioxide cycle has been presented. The experimental data has shown the optimal gas cooler pressure values under the working conditions of the trans-critical cycle, when a small cooling system is considered. The experimental results have allowed to validate the numerical simulation model of hermetic reciprocating compressors and the whole trans-critical cycle evaluation. The numerical results have presented a reasonable good agreement with the experimental data, not only when compressor or heat exchangers are evaluated separately, but also when the elements are considered as a whole. The comparative experimental results have shown an important efficiency improvement when semi-hermetic compressor is considered, with a values closer than conventional R134a hermetic compressors. Then, results show promising perspectives for carbon dioxide trans-critical cycles under the specific application considered in this work, and numerically analysed in the companion paper.

8 ACKNOWLEDGEMENTS

The authors gratefully acknowledge the financial support provided by the 'Comisión Interministerial de Ciencia y Tecnología' (ref. no. TIC2003-07970).
REFERENCES


Thermal and fluid dynamic behaviour of a trans-critical carbon dioxide small cooling system: Numerical analysis

J. Rigola, C. D. Pérez-Segarra, G. Raush, A. Oliva
Centre Tecnològic de Transferència de Calor (CTTC)
Laboratori de Termotècnia i Energètica, Universitat Politècnica de Catalunya (UPC), Spain

A detailed numerical simulation of the thermal and fluid-dynamic behaviour of a transcritical vapour compression refrigerating unit has been developed, and specially adapted for carbon dioxide small cooling systems, taking into account the use of internal heat exchangers between gas cooler outlet and compressor inlet.

The compressor is modelled, considering transient or steady state cyclic conditions, on the basis of global mass and energy balances in the whole system. The empirical information needed to solve the compressor (volumetric efficiency, isentropic efficiency and heat transfer shell losses efficiency) is obtained by means of an advanced simulation model, specially adapted to be used under carbon dioxide fluid refrigerant.

The heat exchangers and capillary tube, if it is used, are modelled solving the governing equations of the flow (continuity, momentum and energy) in one dimensional and transient form.

The global software allows the analysis, in transient or steady state, of a wide range of situations, taking into account different working fluids, geometries and boundary conditions. The experimental comparison is presented in a companion paper.

The following aspects presented in this work are: i) detailed numerical behaviour of the carbon dioxide hermetic reciprocating compressor models depending on boundary conditions; ii) numerical results of the heat exchangers under different working conditions; and iii) numerical study of the whole system and its performance depending on evaporation temperatures.

1 INTRODUCTION

The Montreal protocol [1] stipulated the phasing out of CFCs and HCFCs as refrigerants that deplete the ozone layer (ODP); while the Kyoto protocol [2] encouraged promotion of policies for sustainable development and reduction of Global Warming Potential (GWP), including the regulation of HFCs. Therefore, the ecological problems caused by refrigerating units need an urgent solution. Sustainable development in the refrigeration and air-conditioning fields implies the unavoidable use of "natural" available substances: ammonia or hydrocarbons (toxic and/or flammable), air (poor efficiency), water (limited range of applications) and carbon dioxide. Thus, the investigation and use of new and natural refrigerants is an important goal. During the last decade, the investigation indicates that the use of carbon dioxide has an important interest as a natural fluid refrigerant [3] [4] [5] [6].
The use of carbon dioxide as a non-toxic, non-flammable, harmless and environmentally neutral fluid refrigerant must not only be an attractive alternative, but also a competitive solution. Total Equivalent Warming Impact (TEWI) is the parameter that must be taken into account. TEWI evaluates the sum of the direct contribution to GWP due to refrigerant escape, and the indirect contribution produced by carbon dioxide emissions resulting from the energy required to operate the system over its normal life. In conclusion, the way to obtain reliable cleaning systems puts up with the use of ‘natural refrigerants’, e.g. carbon dioxide, and the obtaining of more energy efficient and emission reduction new alternative refrigeration systems.

For these reasons, the obtaining of general and flexible methods to simulate the thermal and fluid-dynamic behaviour of vapour compression refrigerating systems is a change in the way of doing the design, the applications and the optimisation purposes of these units in order to take into account different aspects such as: the specific geometric characteristics of each component of the system, the consideration of thermal loadings, the use of new and non-contaminant refrigerants like carbon dioxide, etc.

Different works have been presented in the literature, focusing their attention on modeling typical vapour compression refrigerating systems, their components, the overall refrigeration cycle and their experimental comparison. Different authors have modeled the transient performance of single stage heat pumps [7], domestic refrigerators [8], on the basis of global balances. The authors developed a numerical simulation model of the thermal and fluid dynamic behaviour of single stage vapour compression refrigerating systems [9] [10] [11], which was improved and adapted to be used under carbon dioxide fluid refrigerant. More recently, other papers [12], [13] and [14] show the performance of reciprocating refrigeration systems taking into account variable speed, different working conditions and carbon dioxide properties.
The modelling presented in this paper consists of a main program that sequentially calls different subroutines until convergence is reached. The unit studied consists of a double-pipe condenser and evaporator, a capillary tube or expansion device, a reciprocating compressor, and different connecting tubes. Figure 1 presents the refrigerating unit scheme, while Figure 2 presents the pressure enthalpy carbon dioxide diagram.

The heat exchangers and capillary tube are solved on the basis of a control volume formulation of the governing equations (continuity, momentum and energy), considering transient and one-dimensional flow. The creation entropy is considered in the capillary tube in order to detect the limitation of the physical process produced under critical flow conditions. The compressor has been modelled by means of global balances between its inlet and outlet cross-section. This compressor model also needs some additional information: the volumetric efficiency, the isentropic efficiency and the heat transfer losses through the shell. This information has been obtained using an advanced numerical simulation model of hermetic reciprocating compressors, numerically verified and experimentally validated under a wide range of compressor capacities and working conditions.

2 CARBON DIOXIDE HERMETIC RECIPROCATING COMPRESSORS

The possibility to include the numerical simulation model of the compressor in the whole cycle resolution would excessively increase the CPU time needed to solve the cycle. In this case, the model has been carried out on the basis of global balances of mass and energy between the inlet and outlet cross-sections of the compressor considering cyclical steady state. This formulation requires additional information for the evaluation of volumetric efficiency, isentropic efficiency, and heat transfer losses efficiency. This information is experimentally obtained or parametrised from the advanced simulation model mentioned below, and specially adapted to be used with carbon dioxide. The numerical verification and the experimental validation of hermetic reciprocating compressors have been extensively analysed [15].

2.1 Numerical simulation model of hermetic reciprocating compressors

The numerical simulation model solves the thermal and fluid-dynamic behaviour of hermetic reciprocating compressors in the whole domain. The one-dimensional and transient governing equations of the fluid flow are discretised using an implicit control-volume formulation and a SIMPLE-like algorithm extended to compressible flow. Effective flow areas are evaluated considering a multidimensional model based on model analysis of fluid interaction in the valve. The complete set of discretised momentum, energy and pressure correction equations is solved by the direct method TDMA (Tri-Diagonal Matrix Algorithm). Parallel circuits and extra elements (double orifices, resonators, etc.) are also considered in the formulation. The motor torque equation system is linearly independent, thus it is solved directly by means of the inverse matrix system LU resolution. The thermal analysis of the solid elements is based on global energy balances at each macro-volume considered, which is also directly solved in the same way as motor torque. The theoretical basis of the numerical simulation model summarised above for hermetic reciprocating compressors is described in detail in [16].

The numerical results obtained are: local mean instantaneous pressure, temperature and mass flow rate at each fluid flow control volume, together with instantaneous piston position, angular velocity and acceleration, solid temperatures, etc. The compressor behaviour is characterised by the following parameters: volumetric efficiency \( \eta_v = (\dot{m}/\rho_1)/G_{\text{c}}^{v=0} \); isentropic efficiency \( \eta_s = w_s/w_{cp} \) and heat transfer losses efficiency \( \eta_{Q_{sh}} = 1 - (q_{sh}/w_e) \), which are obtained from the numerical results detailed above. The compressor parameters are a function of the compressor geometry, fluid refrigerant, compression ratio, inlet compressor temperature, etc.
2.2 Numerical illustrative results of CO₂ compressors

Table 1 shows the main CO₂ compressor prototype parameters used to simulate compressor behaviour. Figure 3 shows a detailed numerical description of the compressor prototype considering a gas cooler pressure of 90 bars, an inlet compressor temperature of 35°C and evaporation temperatures of -10°C, 0°C and +10°C, respectively. Figure 3 shows the pV diagram, compression chamber pressure and temperature evolution together with suction and discharge valves displacement. The model also allows the possibility to obtain the instantaneous motor torque, angular compressor velocity and acceleration, together with instantaneous compression work or mass flow leakage. It is interesting to highlight the differences of carbon dioxide behaviour on pV diagram or mass flow leakage in comparison with conventional R134a hermetic reciprocating compressors.

Table 1: Crankcase compressor main parameters.

<table>
<thead>
<tr>
<th>HCL15 (hermetic reciprocating compressor)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>D_{inlet}</strong></td>
<td>5.8 mm</td>
</tr>
<tr>
<td><strong>Suction line</strong></td>
<td></td>
</tr>
<tr>
<td>chambers</td>
<td>5.1/1.6 cm³</td>
</tr>
<tr>
<td>plenum</td>
<td>1.0 cm³</td>
</tr>
<tr>
<td>Shell</td>
<td>3350 cm³</td>
</tr>
<tr>
<td><strong>Crankcase</strong></td>
<td></td>
</tr>
<tr>
<td>bore diameter</td>
<td>14.0 mm</td>
</tr>
<tr>
<td>clearance volume</td>
<td>5.42 %</td>
</tr>
<tr>
<td>suction stop</td>
<td>0.8 mm</td>
</tr>
<tr>
<td>diameter (1)</td>
<td>3.2 mm</td>
</tr>
<tr>
<td><strong>D_{outlet}</strong></td>
<td>5.0 mm</td>
</tr>
<tr>
<td><strong>Discharge line</strong></td>
<td></td>
</tr>
<tr>
<td>chambers</td>
<td>9.0/6.5 cm³</td>
</tr>
<tr>
<td>plenum</td>
<td>6.5 cm³</td>
</tr>
<tr>
<td>clearance ratio</td>
<td>5.42%</td>
</tr>
</tbody>
</table>

Fig. 3: Top row: CO₂ pV and compression chamber pressure and temperature. Middle row: valves displacement, mass flow leakage and compression work. Bottom row: instantaneous motor torque, angular velocity and acceleration.
3 HEAT EXCHANGERS

The trans-critical carbon dioxide heat exchangers (gas cooler, evaporator, and possible internal heat exchangers (IHX)) are directly solved in the whole numerical simulation cycle, although these equipments are able to be modelled independently solving the governing equations of the flow (continuity, momentum and energy) in a one-dimensional and transient form. The heat exchangers analysed and simulated in this paper are counter flow double pipe gas coolers and evaporators, where water is used as an auxiliary fluid. The numerical simulation model developed, its numerical verification and experimental validation is explained in detail in [17] [18].

3.1 Numerical simulation model of heat exchangers

The numerical simulation model has been developed by means of the finite volume technique based in a one-dimensional and transient integration of the conservative equations of the fluid flow (continuity, momentum and energy). The governing equations of the flow inside the internal tube and the annulus, together with the energy equation in the internal tube wall, external tube wall and insulation, are solved iteratively in segregated manner. The discretised governing equations of the fluid flow are numerically integrated using a fully implicit numerical scheme and are solved by means of a pressure-based method (SIMPLEC). The empirical information needed in the governing equations are the convective heat transfer, shear stresses, and the void fraction. The general correlations implemented and adapted for gas coolers and evaporators when CO$_2$ is considered as fluid refrigerant have been taken from [19] [20] [21] [22] [23] [24] and [25].

All the flow variables are evaluated at each point of the grid at which the domain is discretised. Depending on the case, inflow and/or outflow conditions, and/or wall temperatures are taken as boundary conditions. The governing equations of the flow are also used to solve the single phase flow in the annular duct, where a counter current water flows and a double-pipe is considered.

The one-dimensional heat conduction equation in the solid element has been discretised considering one-dimensional phenomena (in the longitudinal directions), on the basis of a central-difference numerical scheme. The set of discretised equations has been solved using a line by line algorithm. The fluid temperature and local heat transfer coefficient distribution (inside the tube) are taken as a result of the heat exchanger subroutine.

3.2 Numerical illustrative results of CO$_2$ heat exchangers

Table 2 shows the main CO$_2$ double pipe counter flow water heat exchangers geometry parameters used to simulate gas cooler and evaporator behaviour. Figure 4 shows the detailed numerical results of CO$_2$ and water fluid temperature evolution during gas cooler heat exchanger considering two different heat transfer correlations [21] and [22] for three different mass velocities of 154, 247 and 437 kgs$^{-1}$ m$^{-2}$ respectively. Figure 5 shows the detailed numerical results of CO$_2$ and water fluid temperature evolution during evaporator heat exchanger considering three different heat transfer correlations [23] [24] [25] for the same mass velocities considered in Figure 4.

Heat transfer coefficient of gas cooler heat exchanger is similar under the different correlations considered, although heat transfer has a maximum when gas cooler temperature tends to outlet gas cooler value. The heat transfer increases when carbon dioxide mass flow increases.

Differences on heat transfer coefficients around evaporator behaviour are important depending on heat transfer coefficient considered. Thus, a more accurate information on heat transfer and fluid flow temperature must be taken into account in these cases. However, in all cases heat transfer decreases when vapour quality increases.
Table 2: Heat exchangers parameters.

<table>
<thead>
<tr>
<th>Sample Tube</th>
<th>Annulus</th>
<th>Length</th>
<th>Insulation</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/4 OD</td>
<td>1/2 OD</td>
<td>4.5 m</td>
<td>20 mm</td>
</tr>
</tbody>
</table>

**Dual HTC gas cooler and/or evaporator**

<table>
<thead>
<tr>
<th>Tube Length (m)</th>
<th>Fluid Temperature (C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>30</td>
</tr>
<tr>
<td>1</td>
<td>40</td>
</tr>
<tr>
<td>2</td>
<td>50</td>
</tr>
<tr>
<td>3</td>
<td>60</td>
</tr>
<tr>
<td>4</td>
<td>70</td>
</tr>
<tr>
<td>5</td>
<td>80</td>
</tr>
<tr>
<td>6</td>
<td>90</td>
</tr>
<tr>
<td>7</td>
<td>100</td>
</tr>
<tr>
<td>8</td>
<td>110</td>
</tr>
<tr>
<td>9</td>
<td>120</td>
</tr>
</tbody>
</table>

**CO₂ gas cooler**

- p=90 bar
- G=154 Kg/sm²
- T CO₂ Yoon
- T H₂O Yoon
- T CO₂ Pitla
- T H₂O Pitla

**CO₂ evaporator**

- p=44.4 bar
- G=154 Kg/sm²
- T=-11.3°C
- CO₂ evaporator P=44.4 bar G=437Kg/sm² T=9.5°C

**Fig. 4:** Top row: Fluid flow CO₂ gas cooler and secondary water annul temperatures under three different working conditions. Bottom row: CO₂ gas cooler heat transfer coefficients under same working conditions of top row.

**Fig. 5:** Top row: Fluid flow CO₂ evaporator and secondary water annul temperatures under three different working conditions. Bottom row: CO₂ evaporator heat transfer coefficients under same working conditions of top row.
4 CARBON DIOXIDE TRANS-CRITICAL CYCLE

The single stage vapour compression system is the cycle numerically analysed in this paper and experimentally validated in the companion paper. The thermal and fluid dynamic behaviour of these systems is comparable to the trans-critical cycle when internal heat exchanger is used, considering point 5 and 8 of Figure 2 corresponding to outlet gas cooler and outlet evaporator respectively.

4.1 Numerical simulation model of single stage vapour compression units

The numerical resolution consists of a main programme composed of different subroutines. The mathematical formulation of these subroutines has been carried out to solve the single phase and two-phase flow inside a characteristic duct control volume, together with the conduction heat transfer along solid tube control volume. The different elements of the equipment (evaporator, compressor, gas cooler or condenser, expansion device and auxiliary connecting tubes) are solved by means of the mentioned subroutines called in a convenient way.

The compressor is numerically parametrised defining the volumetric efficiency, isentropic efficiency and heat transfer shell losses efficiency evolution. The numerical results are obtained from the numerical simulation model of section 2. The heat exchangers are directly evaluated from the subroutines detailed on section 3, while expansion device is evaluated in a similar subroutine when capillary tube is considered, and taking into account that in this case, all the inflow conditions cannot be simultaneously input data, as the critical mass flow rate is fixed for a given capillary tube. In the case herewith presented a commercial valve as expansion device, have been used and selected to provide an accurate desired control flow rates. In this case, the numerical model is based on considering the flow through the valve as a sudden contraction along the tube. The contraction of the tube is function of the $C_v$ which represents the coefficient of flow, obtained from the characteristic behaviour of the valve. The $C_v$ is function of the number of turns open of the valve.

The algorithm solves the global equations system using the successive substitution method. Thus, at each time step, the subroutines which solve all the different elements are called sequentially, transferring adequate information to each other until convergence is reached. Transferred information depends on whether transient or steady state is considered. The boundary conditions for the simulation of the whole system are the inlet temperature, pressure and mass flow rate of the secondary flow in the gas cooler and evaporator, the compressor speed, the ambient temperature and pressure, and the opening position in the valve. A more detailed information about global algorithm resolution is shown in references [11] [26].

4.2 Numerical illustrative results of CO$_2$ trans-critical cycle.

Figure 6 shows the numerical results of three different whole trans-critical carbon dioxide cases. The compressor is numerically analysed under geometrical parameters of Table 1. The heat exchangers considered are detailed in Table 2. In all three cases, the gas cooler pressure is 90 bar, with an outlet gas cooler temperature and inlet compressor temperature of 35°C. The three cases present evaporation temperatures of -10°C, 0°C and +10°C, respectively.

Numerical results, indicate that inlet evaporation vapour quality is higher than qualities compared with sub-critical R134a cycles for similar applications, cooling capacities or evaporation temperatures. The use of an internal heat exchanger is the solution under the real cycle application. In the companion paper the numerical results of whole cycle presented are experimentally validated with a reasonable good agreement.
5 CONCLUSIONS
A numerical analysis of trans-critical vapour compression cycle behaviour has been presented. A detailed numerical simulation model of hermetic reciprocating compressors in general and carbon dioxide numerical prototypes in particular has been used to evaluate the thermal and fluid dynamic behaviour of a trans-critical CO\textsubscript{2} compressor prototype. The numerical results of compressor model, together with the numerical simulation of heat exchangers has allowed the calculation of carbon dioxide cycle results under specific working conditions considering small cooling capacity systems. The numerical results are experimentally validated in the companion paper.

6 ACKNOWLEDGEMENTS
The authors gratefully acknowledge the financial and technical support provided by ACC Spain, S.A. and the ‘Comisión Interministerial de Ciencia y Tenología’ (ref. no. TIC2003-07970).

REFERENCES


REFRIGERATION
Perspectives on the performance of carbon dioxide compressor in a light commercial refrigeration appliance

Ricardo A. Maciel, Reinaldo Maykot, Gustavo C. Weber
EMBRACO S/A – Empresa Brasileira de Compressores, Brazil

ABSTRACT

HFC refrigerant has been identified as a contributing factor with regard to global warming. This fact has put pressure on most of the refrigeration industry to replace the HFC refrigerant fluids currently employed in vapor compression systems. In this search for an environmental friendly technology, Carbon Dioxide (CO₂) has emerged as a leading candidate to be a replacement for HFC’s. Today there is a strong focus on CO₂ potential for beverage cooling application although it is still in the preliminarily stages. A comparison between a CO₂ compressor under development and a standard HFC compressor has been performed in both calorimeter and application tests. Preliminary data have showed that a CO₂ can perform competitively to the HFC systems in the field.

1. INTRODUCTION

1.1. Background

The design of a refrigeration system involves many considerations. Design invariably requires a critical evaluation of the solutions proposed by considering factors like economics, reliability, safety and environmental impact. The vapor compression cycle has dominated the refrigeration market to date because of its advantages: high efficiency, low cost, and simple mechanical embodiments. Despite those advantages, one of the concerns regarding the use of these systems comes from the fact that the refrigerating effect is produced by making a volatile fluid boil at a suitably low temperature. Most of the classes of volatile fluids currently in use aggravate the ozone layer, promote the global warming of the Earth or present unfavorable properties to the human health and/or safety.

In recent years environmental aspects are increasingly becoming an important issue in the design and development of refrigeration systems. In vapour compression systems, the banning of CFCs and HCFCs because of their negative environmental impacts has made way for other refrigeration technologies. Until now there has been no motive for the refrigeration market to find alternatives. As environmental concerns grow, alternative technologies which use either inert gases or no fluid at all become attractive solutions with regard to the environmental issue. Regulations prescribe that CFCs and also HCFCs should no longer be used as refrigerants in a few years from now and HFCs seem to be only an interim solution. Looking for final choices and taking into account regulations for greenhouse gas emissions, natural fluids become a
promising alternative as refrigerant fluids. Some of these refrigerants like the hydrocarbons and ammonia come with safety risks. If non-toxicity and non-flammability are required, the focus comes to CO$_2$, which has been considered for low capacity applications operating in transcritical refrigeration cycle.

1.2. CO$_2$ transcritical cycle
Transcritical cycle is that one in which the compressor discharge pressure is above the refrigerant critical point and the suction pressure is below it. In other words, condensation does not take place at the high side heat exchanger and the refrigerant vapour and liquid phases co-exist in only one distinguishable phase. Figure 1 depicts a typical CO$_2$ transcritical cycle in a pressure-enthalpy diagram. Since usual ambient temperatures can exceed 31°C (CO$_2$ critical temperature), a CO$_2$ refrigerating system will not exhibit condensation at the high side heat exchanger, as shown by the heat rejection process 2-3 in the figure, and temperatures and pressures will not be linked as in saturation (dome) region. In a refrigeration cycle where conditions range from subcritical to supercritical, CO$_2$ reaches high pressures, as much as 100 bars (10 MPa) or more.

![Figure 1: CO$_2$ pressure-enthalpy diagram.](image)

2. DEVELOPMENT OF CO$_2$ COMPRESSORS

2.1. Market demands
The light commercial refrigeration market is getting more and more sophisticated creating a new demand for high efficiency, long-life, and environmental friendly refrigeration systems.

Some European countries have regulated or are about to regulate on the HFCs phase-out. The remaining European countries are waiting for the EU regulation which should be sanctioned likely in two years from now.

The first step towards the HFC-free refrigeration in the light commercial segment has been the HFC replacement in industrial facilities. Important refrigeration end-users like Coca-Cola, Unilever (Ice Cream), Nestle and McDonald's are looking towards an HFC-free refrigeration future [1]. The demand for low-cost, trouble-free, high efficiency refrigeration compressors by
those end-users as well as by other players in the same markets has been leading to in-depth, time-intensive R&D programs worldwide on CO₂ refrigeration technology. Embraco is a company committed to study and evaluate the potential of this technology.

2.2. Methodology and comparison basis
Since January 2004 Embraco has been researching and developing CO₂ compressors. Based on the demands required by CO₂ as a refrigerant fluid, a completely new compressor platform was developed. Many CO₂ compressor prototypes have been tested to date in order to assess the performance of the concept.

All the results obtained in this work were compared to current HFC technology in the field. Neither the improvement on current HFC refrigeration compressors nor the improvement on the appliance were considered despite the opportunities for that. CO₂ is applied as a drop-in, keeping the refrigeration system technology at the same level it is today. Regarding calorimeter tests, the comparison basis is the volumetric and the compressor external isentropic efficiencies while the energy consumption in a monthly basis constitutes the comparison parameter for the appliance tests. A T.E.W.I (Total Equivalent Warming Impact) analysis is performed in order to assess the potential for CO₂ emissions reduction from the refrigerating system and thus for the reduction of its global warming potential. T.EWI is the sum of the direct (chemical-related) and indirect (energy consumption-related) emissions of greenhouse gases from an equipment along its lifetime. A summarized introduction to the T.E.W.I. analysis is provided in the Appendix 7.3.

3. CO₂ COMPRESSOR TESTING PROGRAM AND RESULTS
The facilities for the CO₂ compressors performance evaluation were based in a hot cycle calorimeter in which the testing procedure consists of imposing the pressure and temperature at the compressor inlet. The discharge pressure and the ambient temperature are also imposed and the parameters measured are the refrigerant mass flow rate and the compressor power consumption. Figure 2 depicts an outline of the test rig for CO₂ compressor evaluation.

Calorimeter tests were performed at different evaporating temperatures and discharge pressures for the CO₂ compressors. Evaporating temperature was varied from –5°C to –15°C while the discharge pressure ranged from 83 bar to 95 bar. Ambient temperature was kept constant and at 32°C. The compressor inlet temperature was also kept constant and equal to the ambient temperature.

Figures 3, 4, and 5 show the performance curves for the CO₂ compressor, namely cooling capacity, isentropic efficiency, and volumetric efficiency. In the same plot the respective curves for an standard HFC compressor can be seen. For the CO₂ runs, the cooling capacity was calculated considering an approach of 4.6K in the gas cooling process. For the HFC curve, the condensing temperature considered was 43°C and the sub-cooling as 0.4K. The isentropic and volumetric efficiencies were calculated as described in the appendix of this work.
Figure 2: Test rig for CO₂ compressors performance evaluation.

Figure 3: CO₂ compressor cooling capacity map.
Figure 4: CO₂ compressor isentropic efficiency map.

Figure 5: CO₂ compressor volumetric efficiency map.
4. APPLIANCE TESTING PROGRAM AND RESULTS

The compressor performance comparison presented in section 3 is very useful to understand the improvements expected on compressor performance when moving from the HFC-134a to CO2.

However, when comparing only compressor performance in a calorimeter test some variables are not contemplated, and some assumptions such as the approach temperature at the gas cooler outlet for the CO2 cycle or the subcooling degree at the condenser outlet for the HFC cycle, and isenthalpic expansion process, have to be made.

Therefore, an additional comparison between CO2 and HFC-134a technologies was proposed based on a final application. A 405 cans storage capacity beverage vendor cabinet was chosen for that purpose. The tests were performed at 32°C ambient temperature and 65% relative humidity for both HFC-134a and CO2. The overall system energy consumption was measured and the contributions of the compressor and other system components (fans, lights, electronic controls) were evaluated apart. Figure 06 shows the energy consumption results for the baseline system with HFC-134a as well as with CO2.

![Figure 06: Beverage vendor energy consumption (32°C / 65%).](image)

In order to have a fair comparison between both technologies, no significant changes were made on the system components design. The system was tested with the same heat exchangers (evaporator and condenser/gas cooler) for HFC-134a and CO2. The capillary tube was redesigned for CO2 but keeping the original length in order to not change the capillary tube/suction line heat exchanger geometry. Original capillary tube with 1.2 mm internal diameter was replaced by a new one with 0.79mm when setting up the system for CO2. Refrigerant charge was also adjusted in conjunction with the capillary tube in order to obtain the desired CO2 discharge pressure. A discharge pressure in the range of 85 bar was set, allowing for an operation close to the optimum performance point for the CO2 transcritical cycle. Refrigerant charge went up from original 400g of HFC-R134a to 600g of CO2.
By testing the cabinet with the previous modifications it was observed an approach temperature (difference between gas cooler outlet temperature and the ambient temperature) of 4.6°C for the CO₂ run. For the HFC-134a run it was observed condensing temperature of 43°C with 0.4°C subcooling. Evaporating temperature, air temperature at evaporator inlet (return air) and average cans temperature were at equivalent values for both CO₂ and HFC-134a.

Based on the results presented in Figure 6, a TEWI analysis was performed considering the assumptions referred to in a previous work [2], namely annual refrigerant leakage rate of 5%, 10 years system life time, 75% refrigerant recycling factor at the end of life and β factor of 631 gCO₂/kWh (North America), 457 gCO₂/kWh (Europe) and 688 gCO₂/kWh (Asia). The TEWI analysis results can be seen in Figure 7.

![Figure 07: Beverage vendor TEWI analysis (32°C / 65%).](image)

5. CONCLUSION

Results for isentropic and volumetric efficiency were shown for a CO₂ compressor prototype and for an standard HFC compressor currently in the field. It was found that CO₂ compressor prototype can deliver better isentropic efficiency than an HFC compressor, regardless the discharge pressure and the evaporating temperature within the ranges tested. The same conclusion could be observed for the volumetric efficiency. The gains observed with the CO₂ compressor prototype varied from 45% to 50% in terms of isentropic efficiency, and from 32% to 44% in terms of volumetric efficiency. Superior compression and volumetric efficiencies can make the CO₂ application feasible, overcoming the intrinsically low CO₂ transcritical cycle efficiency.

Energy consumption test results performed at 32°C ambient temperature and 65% relative humidity were disclosed. The overall energy consumption of the HFC system was reduced by 10% when the HFC compressor was replaced by a CO₂ prototype and the capillary tube adjusted for the new fluid. Considering only the compressor energy consumption, that is, not
accounting for condenser and evaporator fans, lights, and electronic controls, the reduction observed was 11.8%.

Based on the results of the energy consumption tests, a TEWI analysis was performed for three different regional markets: North America, Europe and Asia. For all of them it was noticed a potential reduction of the CO₂ emissions from the refrigerating system. While for North America the reduction observed in the TEWI index was 12.9%, for Europe and Asia it was found, respectively, 14.1% and 12.7%. The differences are explained by the energy sources mix of each market which affect the β factor.

Replacing current HFC refrigerant fluids by CO₂ is meaningful only if the TEWI index is at least at the same level of the respective baseline. The preliminary results obtained in this work pointed to a better performance in terms of TEWI for a CO₂ refrigeration system when compared to an HFC one. However, further research is required to make a final statement on the potential of CO₂ as a replacement for HFC refrigerant fluids.

6. REFERENCES


7. APPENDIX

7.1. External isentropic efficiency
The external isentropic efficiency was calculated by the following equation:

\[ \eta_{\text{sen, tropic}} = \frac{\dot{m} \cdot \Delta h_{\text{sen, tropic}}}{W} \]  \hspace{1cm} (1)

where:
- \( \eta_{\text{sen, tropic}} \), is the external isentropic efficiency, [-]
- \( \dot{m} \), is the mass flow rate, [kg/s]
- \( \Delta h_{\text{sen, tropic}} \), is the difference between the enthalpy at the compressor discharge considering an isentropic process and the enthalpy at the compressor inlet tube, [kJ/kg]
- \( W \), is the compressor power consumption during the test, [W]

7.2. Volumetric efficiency
The volumetric efficiency was calculated by the following equation:
\[ \eta_{\text{volumetric}} = \frac{m \cdot \nu}{Q} \]  

where:  
- \( \eta_{\text{volumetric}} \) is the volumetric efficiency, [-]  
- \( m \) is the mass flow rate, [kg/s]  
- \( \nu \) is the specific volume at the compressor inlet, [m\(^3\)/kg]  
- \( Q \) is the compressor swept volume, [m\(^3\)/s]  

7.3. T.E.W.I. – Total Equivalent Warming Impact  
TEWI is the sum of the direct (chemical) and indirect (energy) emissions of greenhouse gases from an equipment for its useful life. TEWI is calculated in accordance with the equation (1):  
\[
\text{TEWI} = \text{GWP} \cdot L \cdot n + \text{GWP} \cdot m \cdot (1 - \alpha) + n \cdot E \cdot \beta
\]

- GWP - Refrigerant Global Warming Potential (equivalent to CO\(_2\)) [kg/CO\(_2\)]  
- L - Leakage rate per year [kg/year]  
- n - System operating life time [years]  
- m - Refrigerant charge [kg]  
- \( \alpha \) - Recycling factor of the greenhouse gas at the end of the system lifetime [%]  
- E - Energy consumption per year [kWh/year]  
- \( \beta \) - CO\(_2\) emissions on energy generation [kg CO\(_2\)/kWh]

When a fuel is burnt, energy is produced and carbon dioxide (CO\(_2\)) and other chemicals, mostly water, are produced. The ratio of energy produced to CO\(_2\) differs according to the type of fuel used. Electricity is generated from a range of fuels including nuclear, gas, oil and coal and in some cases waste. Besides energy generation by burning fuels there are several other alternatives to produce electricity like hydroelectric plants, wind power, geothermal energy sources, tidal power, wave energy, photovoltaic panels, etc.

Depending on the approach that is considered to estimate the amount of CO\(_2\) generated to produce a certain amount of electricity these values can vary significantly. Some sources consider on the calculation a Life Cycle approach what means that not only the amount of CO\(_2\) generated during the energy conversion is considered on the calculation but also the amount of CO\(_2\) generated on the construction of such installation. When burning a fuel, not only CO\(_2\) generated during the burning process is considered but also equivalent CO\(_2\) generated during the transportation of the fuel up to the installation is taken into account in some methodologies.

Besides the factors listed before each region inside each country has its own mix of primary sources used to produce electricity. This mix can change significantly from country to country and even from one state to another into the same country, depending on the primary sources of energy available per region.

Due to the above reasons it is very important when identifying the factor “\( \beta \)” (CO\(_2\) emissions on energy generation expressed in gCO\(_2\)/kWh) for a certain region to understand the methodology and considerations being applied. Different sources of information using different methodologies can supply completely different values for the same factor.
Compression efficiency in transcritical CO₂ applications

Jürgen Süss
Central Compressor R&D, Danfoss A/S, Denmark

ABSTRACT

The use of carbon dioxide (CO₂) as a refrigerant is being considered for an increasing number of applications including low capacity transcritical processes. Due to the fluid properties of CO₂, the pressure ratio of the refrigeration process is rather low compared to common refrigeration processes while the pressure difference is high. Furthermore, the volumetric capacity of CO₂ is higher than for traditional refrigerants. These facts result in special demands regarding the design of components and especially compressors to be able to meet the demands regarding overall system performance. It is found that different strategies for suction and discharge processes can be applied for transcritical CO₂. This paper describes the development and resulting performance characteristics of such a small compressor working with CO₂ as the refrigerant.

Keywords: CO₂; compressor, compressor design, compression process, efficiency

1. INTRODUCTION

The increasing interest in CO₂ as refrigerant in light commercial applications has created a demand for small compressors. For CO₂ cooling at ambient temperatures above 30 °C, the thermodynamic properties of CO₂ require the transcritical cycle, which has gas cooling instead of condensation. This implies, not only a different control strategy compared to normal systems, but also a compressor capable of operating with high differential pressures. As found earlier this implies that a favorable compressor choice is a reciprocating type with piston rings, as any leakage of the cylinder needs to be minimized to achieve a high efficiency /1,2/. The high volumetric efficiency of CO₂ in combination with the need for good cylinder sealing seems to limit the possibility to design the compressors for low refrigeration capacities.

As mentioned /3/, a development of a small CO₂ compressor for cooling capacities in the range of 400 W to 1,2 kW refrigeration capacity at -10 °C was launched. The development has progressed in sequential steps since the first initiative in early 2001.

This paper focuses on the concepts used in the latest development step of the CO₂ compressor and describes the solutions to some of the main design parameters influencing efficiency of the machine.
2. COMPRESSION PROCESS AND DESIGN REQUIREMENTS

Due to the thermodynamic properties of CO₂ the effects influencing the compressor efficiency are different compared to those of conventional refrigerants like HFCs. Static pressure losses and heat transfer losses in the cylinder are of minor importance for the efficiency /2/. However, pressure losses due to non-stationary flow such as pulsations need to be considered and resulted in relatively large suction and discharge plenums. Also the leakage is known to be a driving factor in the determination of a suitable type of compression mechanism. Due to the large difference between the suction and the discharge pressure, cylinder leakage is critical for the compression process performance. This requirement led to the choice of a single piston compressor with piston rings. Another important factor was found to be the heat transfer outside the cylinder- especially the suction gas heating inside the suction plenum. The efficiency of the present compressor was - compared to earlier versions - significantly increased by the reduction of heat transfer between the cold suction gas and the compressor.

The first feasibility test of a small compressor for transcritical CO₂ applications was based on an existing HFC compressor platform and showed satisfactory results. As described /1,2/, leakage of the working chamber, and especially between the piston and cylinder is a major parameter, which determines the overall success of the compressor development. To further investigate this impact, 3 different types of pistons were produced having none, one and two piston rings. The piston/cylinder clearance of the piston without a ring was machined to an accuracy, as being applied in today’s HFC/HC piston ring free compressors with the same piston diameter. The gaps of the pistons with one or two rings were a magnitude larger. These three pistons were tested on the same compressor of the first type to investigate the impact of cylinder liner leakage on the compressor’s performance. Figure 1 shows the results of the measurements for the different piston arrangements as a function of the discharge pressure. Suction conditions were kept constant throughout all testing at 30 bars and 15 °C.

Figure 1:
Impact of cylinder leakage on the isentropic and volumetric compressor efficiency

■ no piston ring and minimum gap; ▲ one piston ring; ● two piston rings

Limiting leakage through the piston/cylinder gap by applying a minimum gap, results in the lowest isentropic and volumetric efficiency for the compressor. One piston ring offers already significant performance improvements while two rings could further reduce the harmful leakage, which results in a further performance increase /1/.

The obvious efficiency reduction with increase of the discharge pressure results mainly from suction gas temperature increase, which goes up together with the pressure ratio of the com-
pression process. The amount of suction temperature increase depends on the compressor design, and was more significant for this compressor than for the later compressor version, where the suction gas intake was more direct.

3. THE NEW COMPRESSOR CONCEPT

The compressor, as being developed so far is shown in Figure 2. It is a more compact design compared to earlier prototypes. The drive mechanism, the piston and cylinder arrangements as well as the valves are basically kept unchanged from the earlier compressor versions /3/ reusing the experiences regarding lifetime and reliability. Various strokes in 3 different models presently offer a capacity range from 400 W to 1200 W refrigeration at -10 °C evaporation and 32 °C compressor return and high side heat exchanger outlet temperatures.

As shown in Figure 2 the present compressor is of a semi-hermetic type. A large thread connects the compressor block and the shell. The power feed through and the lid on top of the compressor are also held to the block by applying a thread. Furthermore, the valve plate and its cap are bolted to the compressor block.

Apart from this, the overall compressor design and especially the drive and lubrication concept correspond to the design of traditional HFC/HC hermetic type compressors for light commercial application. A standard single-phase motor, as used today in HFC/HC hermetic type compressor, is applied. Motors for different voltage and frequency are available.

As mentioned before, suction gas heating is, besides cylinder leakage, a major source of energetic and volumetric losses of the CO₂ compression process /2/. Not considering valve losses, the pressure drop in the suction chamber is negligible due to the high suction pressure and therefore hardly contributes to volumetric process losses. On the other hand, heat transfer resulting in suction gas heating, may reduce the energetic and volumetric efficiency by 10-20 %, especially when the temperature difference between the suction gas and the surrounding is large. Therefore, they have to be considered and heat transfer needs to be minimized.
In the applied valve plate design, the suction gas spends as little time as possible in the suction plenum before entering into the cylinder. This was implemented as shown in Figure 3. The suction gas enters directly a chamber with a volume of only a few times the cylinder volume. Oil returning from the process gets separated and drained to the compressor housing through a connection in the bottom of the suction plenum. The housing volume, which contains suction gas, acts as a buffer. When the suction valve opens, only the gas close to the suction hole gets sucked into the cylinder. During this process the pressure in the suction chamber decreases and gas flows to the chamber from the suction line and from the compressor shell through the connection line. During this a significant amount of gas is sucked directly through the cold suction line into the cylinder. By the end of the suction process, the refilling of the suction chamber is still in progress, but now with colder gas out off the suction line. Since the housing is a fixed volume with no other connections, a backflow occurs into the shell - draining separated oil from the suction chamber back to the oil sump.

The same concept is chosen for the discharge chamber as the gas gets directly guided towards the discharge line passing only a small volume. Depending on the system design, this may result in pressure pulsations, which will propagate into the discharge line and further on to the high-pressure side of the system. Any noise implication due to these pulsations is, however, quite small due to the robust nature of the compressor and the system tubing.

4. PERFORMANCE EXPERIENCES

So far, more than two hundred compressor samples were assembled and various independent performance tests were undertaken. Furthermore, the compressors were integrated in various applications mainly in beverage coolers, vending machines and heat pumps with satisfactory results. In addition, a calorimetric test rig was set up to investigate the performance of the compressor. Energy balance and direct mass flow measurement to quantifying the refrigerant mass flow were used to base the compressor performance data on independent measurement techniques.

The isentropic and the volumetric efficiency of the present compressor version as shown in Figure 2 is given in Figure 4. The isentropic efficiency is defined as the ratio of an isentropic compression from certain suction conditions to a high pressure related to the measured electric power input to the compressor at the same working point. The volumetric efficiency indicates the ratio of the compressed mass related to the theoretical mass through the compressor, which is calculated from the compressor’s stroke volume and the current frequency at the certain running condition. All data were recorded at 230V and a frequency of 50 Hz. The data refer to the biggest compressor presently available with 1200 W refrigeration at -10 °C / 32 °C / 32 °C.
Figure 4: Isentropic and volumetric efficiency of the present compressor version

The plotted data range covers a suction pressure ranging from 20 to 50 bar and a discharge pressure ranging from 50 to 110 bar. All shown performance data are based on mass-flow measurements and confirmed by mass-flow calculations from energy balances of the high- and low side heat exchangers. The isentropic efficiency increases with the suction pressure and reaches maximum values of around 60 %. The volumetric efficiency reaches, especially at small compression ratios, rather high values of up to 90 %. This high efficiency is possible, as
suction gas heating inside the compressor is minimized, suction gas pressure drop has almost no impact and leakage does almost not occur.

The drop of volumetric efficiency with increasing compression ratio is mainly due to re-expansion from the clearance volume of the cylinder. Leakage - which could also cause the drop in volumetric efficiency, is again only affecting the compression process at a negligible degree, as the isentropic efficiency is hardly depending on the compression ratio. The minor efficiency change is rather due to valve losses.

Finally, the compressor performance is not strongly depending on the suction gas temperature, but a slight efficiency increase with rising temperature can be noticed.

5. CONCLUSION

A small CO₂ compressor for cooling capacities in the range of 400 W to 1,2 kW refrigeration capacity at -10 °C was development. The development has progressed in sequential steps since the first initiative in early 2001. The present design showed promising results regarding performance and reliability, so that an industrialization process is started.

REFERENCES

/1/ Süß, J.: Untersuchungen zur Konstruktion moderner Verdichter für Kohlendioxid als Kältemittel, DKV-Forschungsbericht Nr.59, Stuttgart 1998
/3/ Süß, J.: Low Capacity Hermetic Type Compressor for Transcritical CO₂ Applications. Proceedings of the International Purdue Compressor Technology Conference 2002, Purdue, USA,
Compressors for Carbon Dioxide refrigeration systems

A.B. Pearson
Star Refrigeration Ltd, UK

The last ten years have seen a remarkable change in the industrial refrigeration market in Europe, with the introduction of large, low temperature systems using carbon dioxide as refrigerant. The first of these systems used the CO2 as a “volatile secondary”, with no compressor on the low side of the cascade, but recent developments in compressor technology have produced several plants with two or more operating temperature levels. This paper reviews the types of compressor currently available for industrial CO2 systems. It considers the properties of carbon dioxide and the compressor operating parameters in comparison with traditional refrigerants to illustrate the implications for compressor designers and it looks at the additional development required to make this environmentally-friendly alternative more universally applicable.

1. INTRODUCTION

Carbon Dioxide was first used as a refrigerant in 1866, but it gradually fell out of favour at about the time that the chlorofluorocarbons were introduced, in the 1930s. In the last decade the decline of CFCs has seen the reintroduction of CO2 as one of the so-called “natural refrigerants”, and as it is also non-toxic and non-flammable it has proved to be very popular in a wide range of applications. The most attractive feature of carbon dioxide as a refrigerant, its high volumic refrigerating capacity, stems from its greatest disadvantage, the high pressures at which the system runs in comparison with other refrigerants. Another benefit of using carbon dioxide is that it remains at positive pressure, even in low temperature systems (below –40°C), but this range is firmly limited by the triple point at –56°C, below which liquid does not exist. The gradient of the pressure-temperature characteristic for carbon dioxide is flatter than all other refrigerants, which makes carbon dioxide systems relatively efficient in operation as the effect of suction line pressure loss is reduced. For example a loss of 100kPa in an ammonia suction line at –30°C is equivalent to a drop of 2K in the saturated temperature, but the same loss in a carbon dioxide system will only drop the saturated suction by 0.2K. The loss in Carnot efficiency in the system, if it condenses at 35°C is given by

\[
\Delta \eta_c = 1 - \frac{T_e'(T_c - T_e)}{(T_c - T_e')T_e}
\]  

(1a)
For the ammonia system this gives a loss of

\[ \Delta \eta_c = 1 - \frac{241(308 - 243)}{(308 - 241)243} = 3.8\% \]  

(1b)

and for the carbon dioxide system it is only

\[ \Delta \eta_c = 1 - \frac{242.8(308 - 243)}{(308 - 242.8)243} = 0.4\% \]  

(1c)

The remarkably low viscosity of carbon dioxide makes it particularly suitable as a low temperature heat transfer fluid, and the low gas/liquid density ratio improves distribution in evaporators and liquid return in suction pipes, even in risers. Heat transfer coefficients are also generally better than for fluoro-carbons, although not as good as ammonia. (Stoecker, 2000).

The other main disadvantage of using carbon dioxide, apart from higher than usual pressures, is the very low critical temperature. This means that in order to reject heat to atmosphere, carbon dioxide must either be used in cascade with another refrigerant, or it must operate above the critical temperature, using a gas cooler instead of a condenser. This transcritical cycle is unfamiliar to most refrigeration designers, and is not in common use in commercial and industrial systems. Strange things happen above the critical point: the specific heat capacity of the fluid can rise to over twenty times its normal value, the speed of sound in the fluid can drop to zero and system performance can be improved by increasing the temperature lift, not reducing it. This means that a reappraisal of all traditional aspects of refrigeration system design is worthwhile.

2. EARLY CARBON DIOXIDE SYSTEMS

The first recorded refrigeration system using carbon dioxide was constructed by Thaddeus Lowe in the United States of America in 1866, and patented in the following year in the United States and Great Britain. Lowe’s system used a modified hydrogen pump, and must have achieved pressures of 50 or 60 bar, as it used river water in a sub-critical condenser. His system was used to make block ice, so probably had a suction pressure of about 30 bar. It is shown in the drawings in his United States Patents as a vertical single cylinder with a belt drive. It probably had a long piston to minimise leakage, and with a pressure ratio of less than 2 it did not need a particularly small clearance volume. Lowe’s main business interests lay in gas works and he did not develop his ice making system further (Pearson, 2003).

Some further development work on carbon dioxide compressors was done by Wilhelm Raydt and Carl von Linde in Germany in the early 1880s, without commercial success, but in 1886 Franz Windhausen patented a development of the “carbonic acid pump” which made the machines more reliable, and this patent was licenced by Everard Hesketh of J&E Hall, Dartford in 1887 (Thévenot, 1979). Hesketh, who was a graduate mechanical engineer and had introduced Halls to refrigeration through air cycle machines a decade earlier, went on to produce many more technical developments, helping to establish carbon dioxide as the preferred refrigerant for marine applications with Halls as the preferred supplier. These developments included a safety relief valve for the cylinder head, a copper gasket for the cylinder head, a double acting compressor and a precision ground piston rod seal. Halls
installed their first marine installation in 1890, and within six years had over 400 machines in operation (Miller, 1985).

As system discharge pressures edged towards the critical point, through increased use of atmospheric condensers on land and refrigerated cargoes sailing in tropical waters, the carbon dioxide systems became progressively less and less efficient than ammonia ones. Many technical innovations were made in the first quarter of the twentieth century, including a novel economiser design for a double-acting reciprocating compressor patented by the Haslam company in 1923, shown in Figure 1. Even this was not enough to prevent the economic benefits of ammonia from displacing carbon dioxide in most markets, and the advent of CFCs in 1929 provided a range of suitable alternatives for the few applications for which ammonia was not acceptable. By the beginning of the second World War carbon dioxide systems were no longer being installed, and by the 1950s they had virtually disappeared.

Figure 1 – Haslam’s economised reciprocating compressor, 1923
3. RECENT CARBON DIOXIDE SYSTEMS

The first response in the mid-1980s to the proposed reduction in quantity of CFC in use was to develop alternative hydrofluorocarbons such as R-134a, which contains no chlorine and hence has no ozone depletion potential. By the early 1990s it was accepted that a total ban on CFCs was more likely, and the high cost of HFCs and their associated lubricants, polyol esters and polyalkyl glycols, had prompted an evaluation of many “not-in-kind” solutions. At this time carbon dioxide was proposed as a secondary fluid because it offered much lower viscosity than water/glycol or water/brine solutions at low temperatures. This would permit the use of small ammonia systems in cold stores and freezers, with the carbon dioxide circulated to the points of use. One apparent disadvantage of such a system was the need to keep the carbon dioxide liquid under pressure to prevent it from boiling, but this was turned to a significant advantage by controlling the pressure and allowing the liquid to evaporate at the load, just as in a pumped circulation refrigeration system. As the carbon dioxide circuit had no compressor it was not a true cascade plant, and it became known as a volatile secondary system (Pearson, 1993). This system was mainly used in supermarkets in the early to mid 1990s (Rolfsmann, 1996) but a few industrial plants were also installed.

The next step in the reintroduction of carbon dioxide to industrial refrigeration was the installation of a low temperature cascade plant by Nestlé UK at their freeze dried coffee plant at Hayes in Middlesex (Pearson, 2000). This plant runs at an evaporating temperature of –50°C, giving a suction pressure of 5.8 bar and uses old-fashioned double-acting oil-free reciprocating compressors very similar in concept to the basic Haslam design illustrated earlier. The oil-free design was selected because it was not known how the evaporator coils in the vacuum chambers of the freeze drier would perform with a non-miscible oil. The nominal discharge of the compressors is at 20 bar which equates to –18°C saturated. As part of the trial work conducted before the main project a small pilot plant was connected to an air cooler in the freezer tunnel. This plant, with a duty of 100kW refrigerating capacity used a standard twin screw compressor running on polyol ester oil. This test plant proved that many common refrigeration components could be used on carbon dioxide systems provided that a means to limit the pressure rise was provided. The commercial screw compressor, which has a swept volume of 100m³/h at 3000rpm, was not able to handle start-ups with high suction pressure as it had no means of unloading, so the demonstration plant was designed to run continually. It did so from November 1998 until it was decommissioned early in 2001 when the evaporator was replaced as part of the main installation, apart from a brief spell around the end of 1999 when it was shut down due to some Year 2000 controls testing. The model of compressor used was also available in a semihermetic variant but the compressor used was an open drive type, because a 55kW drive motor was required. The largest motor available in the semihermetic version was rated at 26kW. This discrepancy is a consequence of the high volumic refrigerating effect of carbon dioxide (kJ/m³ rather than kJ/kg), and typifies the difficulty of using hermetic designs on larger compressors. During the period from 1998 to 2001 while the project at Hayes was under development many other low temperature freezer plants were installed using a range of more or less standard refrigeration compressors. These included lubricated reciprocating machines and oil injected screws, but in both cases the discharge pressures were limited. The reciprocating compressor was designed for an allowable pressure of 40 bar, for use on ammonia heat pumps, and early systems used standard ammonia machines as the low stage of the carbon dioxide cascade. Standard screw compressors were more restricted, typically with an allowable pressure of 32 bar or less. This limited them to cascades with a carbon dioxide condensing temperature of about –10°C, allowing for relief valve and high pressure cutout settings.
A more recent development in industrial systems has been a combination of the volatile secondary and the cascade technologies to provide carbon dioxide cooling at frozen and chill temperatures from a central plant. This has required compressors capable of slightly higher pressures (screws) or larger capacities (recips). The oil injected screw with a modified shaft seal, suction port and oil circuit has now been developed by several manufacturers for an allowable pressure of 40 bar. This has been used in distribution warehouses which have one large chamber at –25°C for frozen goods and several marshalling areas at various temperatures above 0°C for fresh produce. A typical configuration would have carbon dioxide evaporating at –32°C for the cold store and compressed to about 30 bar so that it can be condensed at –5°C by an ammonia system evaporating at –10°C. This system requires a high pressure cutout at 33 Bar and relief valves set at 36 Bar or higher – it is now possible to design the high pressure side of the carbon dioxide circuit for 40 bar. Comparative analysis of two similar sites has shown that this design uses 10%-15% less energy over a year than an ammonia – glycol chill system, principally because the pump power is greatly reduced (Pearson, 2005).

Even with a compressor rated for 40 bar operation it is not possible to provide hot gas for evaporator defrost systems. To achieve a quick, clean and efficient defrost compressor discharge gas can be condensed inside an evaporator, using the latent heat of condensation to heat the tube wall and fin block. However a temperature of about 10°C is required for this, which equates to a pressure of 44 bar. This has been successfully achieved with a modified version of the heat pump reciprocating compressor mentioned above, which was rated for an allowable pressure of 50 bar, using a modified gudgeon pin and connecting rod arrangement. However to provide high pressure cutout and relief valve settings with a full 10% tolerance would require an allowable pressure of 52 bar. This has been recognised by Danfoss Industrial Refrigeration who have rated their range of Industrial Control Valves for this pressure. The lack of compressors for this pressure has led to the development in the meantime of a number of alternative defrost systems including an interlaced warm glycol loop in the evaporator, the use of evaporator pods which enclose the evaporator during defrost, and the use of hot gas generation from pressurised liquid. In warm climates ambient air can be used, even for cold stores.

Throughout this development of industrial systems there has also been substantial use of carbon dioxide in smaller equipment. This however has followed a different path, with the development of many small compressors capable of operating at much higher pressures (Fagerli, 1996). In Japan several varieties of small scroll and reciprocating compressor are used in domestic heat pumps for tap water heating (Shimada, 2004), in Norway larger heat pump systems have been laboratory tested (Nekså, 2004), and in Denmark a family of reciprocating compressors for use in commercial chilling applications has been developed (Süss, 2004). Larger reciprocating compressors, with swept volume up to 12m³/h have been developed in Germany and Italy for the truck and bus cooling market. These varied developments followed the early work by Lorentzen at the SINTEF research institute in Trondheim which is summarised in his first transcritical patent application of 1989, granted in 1993. This patent showed that the efficiency of a transcritical cycle can be improved by keeping the discharge pressure well above the critical pressure because the effect of reduced enthalpy at the gas cooler outlet far outweighs the additional work input required at the compressor. Lorentzen went on to develop a range of control devices to provide better control and part load performance.
4. IMPLICATIONS FOR COMPRESSOR DESIGN

The benefits of transcritical operation in the small commercial systems described are overall system efficiency and simplicity. Industrial refrigeration systems need compressors in the range 50m$^3$/h to 500m$^3$/h with a 40 bar suction operating at pressures up to 100 bar to offer the same advantages on a larger scale. The volumetric efficiency of reciprocating compressors is principally affected by the pressure ratio because the gas left in the cylinder at the end of the exhaust stroke is re-expanded and at large ratios can fill the chamber, leaving no volume remaining for inlet gas. A secondary effect which also reduces the volumetric efficiency is leakage past the piston. This is dependent upon pressure difference, not pressure ratio. In a typical ammonia compressor on water chilling duty the pressures P1 and P2 will be 4.8 bar abs. and 17.8 bar abs. respectively, giving a ratio of 3.71 and a difference of 13 bar. For a transcritical carbon dioxide compressor the values are 37.7 bar abs. and 90 bar abs., giving a ratio of 2.39 and a difference of 52.3 bar. Thus rather unusually the clearance volume in a transcritical reciprocating compressor will be rather less important and the piston leakage will be more important. Valve design is also rather less critical than with ammonia and fluorocarbons because of the flat pressure-temperature characteristic mentioned earlier. To achieve 500m$^3$/h in an eight cylinder machine running at 1450rpm would require a gross swept volume per cylinder of 0.718 x 10^{-3}m$^3$. This would be given by a cylinder bore and stroke of 100mm, which is typical of the dimensions of an industrial refrigeration reciprocating compressor. However the machine would look substantially different to traditional models in detail. The pressure difference of 52.3 bar would produce a force of 41kN on the piston at top dead centre, and it would be difficult to lubricate a conventional gudgeon pin, where the bearing area may only be 20% of the piston area. One possible solution would be to flare the lower end of the piston to create a ball and socket joint so that it does not enter the cylinder. The surface area of the load bearing hemisphere will be twice the area of the cross-section, so even if the hemisphere has the same diameter as the piston the load bearing area is doubled. This follows from the surface area of a sphere, which is given by $A_s=4\pi r^2$, so the area of a hemisphere is $A_h=\pi d^2/2$ whereas the area of a circle is of course $A_c=\pi d^2/4$. This arrangement would also allow for a longer piston to give improved sealing, but it would result in the compressor being physically larger than a standard 500m$^3$/h 8 cylinder model.

By contrast, a screw compressor volumetric efficiency is primarily determined by pressure difference owing to gas leakage across the rotor tip seals and through the “blowhole”. The built in compression ratio of the compressor, which is determined by the size of the discharge chamber versus the size of the suction chamber also affects the point at which the adiabatic efficiency is maximal, with lower peak values at higher volume ratios. The lowest value of volume ratio which can be achieved in a standard compressor is 2.6 because of the geometry of the discharge port although lower values, down to 2.1, are possible by increasing the size of the discharge port, usually by machining part of the slide valve or casing. The index of compression, $\gamma$, for carbon dioxide at 37.7 bar abs. saturated is 1.28, so the optimal volume ratio for a carbon dioxide machine working from 37.7 to 90 bar abs. is derived in the following way:

$$P_1V_1^\gamma = P_2V_2^\gamma \Rightarrow V_i = \left(\frac{V_1}{V_2}\right)^{\frac{1}{\gamma}} = \left(\frac{P_2}{P_1}\right)^{\frac{1}{\gamma}} = \left(\frac{90}{37.7}\right)^{0.781} = 1.97$$

(2)

Thus for a transcritical screw compressor working on an air conditioning duty the ideal volume ratio is less than can be realistically achieved in a typical twin screw compressor. This does
not mean that the machine is not possible to build, just that it does not run at the most efficient pressure ratio. It also means that, unlike a reciprocating compressor, it will be less beneficial to drop the discharge pressure below the critical point in low ambient temperatures, as the pressure ratio will then be far below the optimal, resulting in significant over-compression losses. Other considerations in the design of larger transcritical screw compressors will be that the L/D should be kept as short as possible to minimise rotor flexing, and the bearing area should be as large as possible. Even with a balance piston the thrust bearing load is significantly higher than in a typical refrigeration screw compressor, which suggests that a Zimmern type single screw compressor might be a more appropriate configuration for these high pressure differences. A further advantage of the single screw is that the radial loads are balanced, so provided the gate rotor teeth can be made stiff enough to stand the large pressure difference without deflecting and leaking the compressor efficiency will be high.

![Fig 2a Recip compressor efficiencies](image)

Typical plots of volumetric efficiency (solid lines) and isentropic efficiency (dashed lines) are shown in Fig 2a for reciprocating compressors and Fig 2b for screw compressors. The heavy lines show approximately what the efficiencies would be at the suction pressures found in an ammonia compressor and the light lines show the equivalent at the much higher pressures found in a carbon dioxide compressor. In Fig 2a it can be seen that the higher pressures of carbon dioxide give an improvement in isentropic efficiency but a reduction in volumetric. For the screw compressor in Fig 2b, both volumetric and isentropic are reduced at a given pressure ratio by raising the suction pressure.
It has been found that carbon dioxide compressor performance is greatly improved even in subcritical screws by greatly increasing the oil flow and viscosity, as this provides better rotor tip sealing (Klidonas, 2004). However there is a limit to the extent to which this can be used, as the excess oil requires power to move it and will therefore impair the volumetric and adiabatic efficiencies if taken to extremes. It might be appropriate for higher pressure machines to create a double tip seal on the rotors, perhaps with a smaller quantity of oil fed through internal channels in the rotor to the tip in order to provide a better seal. To minimise the blowhole losses a small diameter rotor running at high speed would be preferred, provided the shaft is stiff enough to carry the high torque required.

As Haslam found in 1923 there is a substantial advantage in economising transcritical carbon dioxide machines, but it is difficult to reconcile the large suction port required to get a suitable volume ratio with the need to position the economiser port as close to the suction end of the machine as possible. One possible solution would be to use a series of small diameter holes from an economiser “chamber” to the compression space instead of a single larger hole. An alternative would be to adopt the “variable economiser port” developed by Stal AB in the early 1990s, where the intermediate gas is fed through holes in the capacity slide rather than through the compressor casing. The ultimate success of the transcritical screw compressor is far more dependent upon a successful means of economising the system than for most other refrigerants because the efficiency improvements made possible by this technique are far greater, although the same development could usefully be applied to some HFC blends such as R-404A and R-410A. Economising is equally important for transcritical reciprocating compressors, and can be achieved by running the cylinders of a multistage compressor at two or more suction conditions with a common discharge (Bell, 2004). For larger systems it might be appropriate, whichever compressor type is used, to have one or two machines providing the economising duty and the rest handling the suction gas.

In smaller sizes scroll compressors have been considered for carbon dioxide, and the balanced forces and large bearing surface make them seem at first sight quite suited, however there are
some drawbacks which might prevent their development for this application. They are configured as hermetic machines, so the usual problem of fitting a large enough motor to drive the carbon dioxide compressor applies. As the leakage on the rotors is a function of pressure difference as in a screw compressor it might be appropriate to adopt permanent magnet motors giving high speed operation and high power in a very small framesize. It is also difficult to make the scrolls stiff enough without increasing the tip leakage. The high pressures would predicate tall, thin gas passages, so that the clearance at the tip is a smaller percentage of the swept volume, but the stiffness required to withstand the high pressure requires a short, fat passage. Similar motor technology might also be applied to centrifugal compressors. Traditionally these are not used for carbon dioxide in lower pressure applications because the molecular weight is so low, at about one-third that of R-12. However at high pressures the gas density could make a centrifugal compressor feasible, and the low pressure ratio would also suit a single stage machine, although if two impellers were used it would be possible to economise the cycle and gain the advantages described earlier. With a hermetic, oil-free design an economised carbon dioxide centrifugal compressor would be a very compact yet efficient machine.

5. CONCLUSIONS

The compressors to facilitate the adoption of transcritical carbon dioxide cycles in industrial refrigeration do not yet exist in the mass market, but they are not far off. Central plant for air-conditioning of large commercial buildings is potentially a large volume market, so the development would seem worthwhile. For systems with varying discharge pressure a modified reciprocating compressor will be very efficient, but if the system incorporates high grade heat recovery screw compressors are also attractive. Either way the incorporation of some form of economiser into the system is essential to good efficiency. If the market demand is proven there is also scope for a range of machines based on scrolls at the smaller end and centrifugal compressors for larger duties.

6. REFERENCES


Klidonas, G., “Screw Compressor Packages: Considerations for use on CO2/NH3 Cascade Systems” Frigair, SAIROC, Cape Town 2004

Lorentzen G., “Method of Operating a vapour compression cycle under trans- or supercritical conditions”, European Patent EP0424474, 1993

Nekså, P., “CO2 as refrigerant for systems in transcritical operation”, “Natural Refrigerants Conference, AIRAH, Sydney, 2004


Shimada G., “Introduction of Rolling Piston Type 2 Stage Compressor for CO2 and Application for Refrigeration Cycle”, ASHRAE Winter meeting, Anaheim, 2004


Süss J., “A Small Compressor for Transcritical CO2 Applications”, ASHRAE Winter meeting, Anaheim, 2004

Thévenot, R., “A history of refrigeration throughout the world”, IIR, 1979
An economizer cycle for A/C applications

Michael F. Taras
Carrier Parkway, USA

NOMENCLATURE

\( P \) – pressure, (Pa)
\( T \) – temperature, (ºC)
\( c_p \) – specific heat at constant pressure, (J/kg·ºC)
\( c_v \) – specific heat at constant volume, (J/kg·ºC)
\( h_{fg} \) – vapor-liquid phase change enthalpy, (J/kg)
\( \Delta h_{ref} \) – refrigerant enthalpy change, (J/kg)
\( \dot{m}_{ref} \) – refrigerant mass flow rate, (kg/sec)
\( U_{ax} \) – overall heat transfer coefficient, (Wt/ºC·m²)
\( A_{eff} \) – effective heat transfer area, (m²)
\( \delta T_{log-mean} \) – log-mean temperature difference, (ºC)
\( \dot{Q}_{ax} \) – heat flux, (Wt)
\( Pr \) – pressure ratio, (dimensionless)

\( \left( \frac{dP}{dT} \right) \) – pressure gradient with respect to temperature, (Pa/ºC)
\( \left( \frac{c_p}{c_v} \right) \) – specific heat ratio, (dimensionless)
\( \left( \frac{dP}{P} \right)^{\frac{1}{2}} \) – leakage parameter, (dimensionless)

ABSTRACT

A ban on HCFC-22 (effective 2010) and the introduction of alternate refrigerants may have a negative impact on conventional system performance. In particular, R410A refrigerant is becoming increasingly popular on the market but presents new challenges to system designers. This paper evaluates the advantages offered by the economizer cycle for air conditioning and heat pump applications and suggests that such an approach to enhance system performance has
significant potential. The benefits of the economizer cycle are especially evident in the environment in which conventional methods reach a plateau of rapidly diminishing returns on investment or become physically impossible.

1. INTRODUCTION

Life-cycle cost of air conditioning equipment recently became one of the foremost concerns for original equipment manufacturers and end customers. Newly introduced legislation and industry regulations concerning minimum efficiency standards have increased the focus on this subject. Since a well-defined set of system performance characteristics represents an essential component in the equipment cost matrix, unit efficiency becomes particularly important. Furthermore, a ban on HCFC-22 (effective 2010) and the introduction of alternate refrigerants may have a negative impact on system performance, since these alternate refrigerants have thermo-physical properties that are considerably different from those of HCFC-22 (see for instance [1]). In particular, the R410A refrigerant is becoming increasingly popular in commercial air conditioning applications. This market segment is drastically different from the residential sector, where R410A is developing into a refrigerant of choice. The major differences lay in operating environments in terms of ambient temperature and ventilation requirements as well as equipment size, controls and optional features. On the commercial side, although several design enhancement options are feasible, R410A exhibits a performance deficiency at certain environmental conditions in comparison to the R22 refrigerant that is being phased out. These issues present new challenges to system designers and must be addressed in the most comprehensive and cost effective manner.

These developments have prompted the introduction of non-conventional methods to significantly boost system efficiency, as well as incremental improvements to existing performance augmentation techniques. Consequently, new systems may incorporate novel design approaches, such as an economizer cycle, that in the past would not have been considered for similar applications. The choice of a particular system design and configuration is defined by essential performance characteristics, component reliability, applied cost and specific application considerations. This paper evaluates the advantages offered by the economizer cycle for air conditioning and heat pump applications and suggests that such an approach to enhance system performance has significant potential. The benefits of the economizer cycle are especially evident in the environment in which conventional methods reach a plateau of rapidly diminishing returns on investment or become physically impossible.

2. HFC REFRIGERANTS

Since introduction of non-ozone depleting refrigerants has a profound effect on essential operational characteristics, it is practical to execute a zero-order analysis to identify potential sources of system performance degradation. Azotropics (or near azotropics) blends are exclusively considered as R22 replacements in this paper, since they don’t present major serviceability concerns and simplify the assessment. Potential alternatives for R22 can be divided into two categories, namely high-pressure and low-pressure substitutes. R410A refrigerant is a widely recognized choice for the former category, and R134a is the most logical selection for the latter group. In this chapter, R410A is analyzed to a greater extent, followed by a few comments regarding R134a.
On one hand, R410A refrigerant has superior evaporation thermo-physical properties, promoting relatively high heat transfer coefficients and elevated saturation temperatures. On the other hand, it has slightly inferior condensation heat transfer coefficients and a relatively low critical temperature, leading to a reduction of refrigerant enthalpy difference in the evaporator and increased throttling losses and compressor power consumption. Furthermore, since R410A is a high-pressure refrigerant, its high density allows for more forgiving component design with respect to pressure drop, that is approximately 40% lower in comparison to R22 for equivalent mass flow rates and operating conditions. In other words, the \( \left( \frac{dP}{dT} \right) \) factor is higher for R410A, allowing for longer circuits in the heat exchangers and smaller connecting lines.

As indicated by equation (1), evaporator capacity, on one hand, is a product of refrigerant mass flow rate and enthalpy difference and, on the other hand, a product of air-to-refrigerant log-mean temperature difference, overall heat transfer coefficient and effective heat transfer surface area.

\[
\dot{Q}_{hx} = \dot{m}_{ref} \cdot \Delta h_{ref} = U_{hx} \cdot A_{eff} \cdot \delta T_{log-mean}
\]  

Evaporation heat transfer coefficients for R410A are approximately 30% higher, in comparison to R22, directly translating into 10% enhancement in the overall heat transfer coefficient in equation (1) in accordance to a split of thermal resistances (since external heat transfer surface area is larger than internal as well). This result suggests that, in order to obtain an equivalent capacity level, refrigerant-to-air temperature difference is to decrease by an identical percentage, corresponding to a saturation suction temperature elevation of 1ºC. Consequently, refrigerant mass flow rate, being proportional to refrigerant density (or pressure) at the compressor suction, increases by about 3.5%.

Turning our attention to the condensation process, it can be shown that the enthalpy span \( h_{fg} \) over the two-phase dome is smaller for R410A than for R22 by 3-8%, depending on the corresponding saturation temperature. Such a phenomenon occurs due to closer proximity of the R410A condensation process to the critical point, promptly reflecting a reduction in the evaporator enthalpy difference in equation (1), since for an ideal vapor compression cycle the enthalpy of refrigerant exiting condenser and entering evaporator are identical. For the higher efficiency systems (systems with high energy efficiency ratios - EER and low life-cycle cost conventionally achieved by enlarging the condenser coil), the product of refrigerant mass flow rate and evaporator enthalpy difference in equation (1) practically doesn’t change, and equivalent capacity should be expected for these commercial systems at the ARI rating point. On the other hand, for the lower efficiency units, capacity degradation of about 5% will take place. Although minor system readjustments of saturation temperatures and refrigerant flow rate occur in the latter case, due to imbalance of the left- and right-hand sides and interrelation of the parameters in equation (1), these effects are of second-order, don’t alter the analysis and are neglected for simplicity.

Similarly, condenser performance can be described by equation (1) as well. Condensation heat transfer coefficients for R410A are approximately 15% worse in comparison to R22, translating into 5% reduction in the overall heat transfer coefficient in equation (1). This result implies that, in order to attain equivalent condenser capacity (and subsequently subcooling), refrigerant-to-air temperature difference is to be increased in an identical proportion, corresponding to a saturation temperature elevation of about 0.5ºC. This phenomenon, along
with the increased refrigerant mass flow rate and reduction in the enthalpy span over the two-phase dome, negatively affects compressor power and condenser performance.

The compression process is influenced by physical properties of the refrigerant, namely the specific heat ratio \( \frac{C_p}{C_v} \) (or a slope of the saturation vapor line) and the pressure ratio \( Pr \), corresponding to the operating conditions, and by the compression elements design. The \( \frac{C_p}{C_v} \) values are within 3\% and the pressure ratios - within 2.4\% for the two refrigerants under consideration, and no noticeable deviation in the compression process is anticipated. Furthermore, regardless of the fact that the pressure differences for the R410A are approximately 1.6 times higher than for the R22 refrigerant, somewhat similar refrigerant leakage losses, on a relative scale, loosely defined by the \( \left( \frac{dP}{P} \right)^{1/3} \) parameter (since refrigerant density is almost proportional to its absolute pressure) are expected. Hence, with the compression elements geometry optimized and positioned within the manufacturing process capability, no deterioration in volumetric efficiency, defined by the refrigerant leakage loss, and isentropic efficiency, affected by the friction and compression losses, should be observed, promoting similar compressor performance characteristics for both refrigerants.

Table 1 presents a high-level summary of the “order-of-magnitude” estimates for the R410A efficiency/deficiency sources in comparison to the R22 refrigerant. As was discussed above, superior evaporation and lagging condensation thermal-physical properties of R410A along with relatively close proximity of the condensation process to the critical point and adverse effect of the saturation liquid line slope are the primary factors responsible for such an outcome. The last line in the table displays an increase/reduction of the refrigerant flow circulating through the system required to attain an equivalent cooling load, and reflects a typical tradeoff between capacity and efficiency. Needless to say, that direct system simulations are to be performed in order to obtain more precise results. Since all original equipment manufacturers face similar performance degradation issues, standard enhancement techniques, such as compressor or coil size adjustments, or non-conventional approaches, like the economizer cycle, need to be employed to reestablish desired performance levels. Obviously, a path to improve system characteristics is at the designer’s freedom and involves usual performance-cost tradeoffs.

### TABLE 1: Performance comparison summary - R410A vs. R22

<table>
<thead>
<tr>
<th>Component</th>
<th>Higher Efficiency System [%]</th>
<th>Lower Efficiency System [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Capacity</td>
<td>EER</td>
</tr>
<tr>
<td>Evaporator</td>
<td>3.5</td>
<td>0</td>
</tr>
<tr>
<td>Condenser</td>
<td>-3.0</td>
<td>-0.5</td>
</tr>
<tr>
<td>Compressor</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>System</td>
<td>0.5</td>
<td>-0.5</td>
</tr>
<tr>
<td>Equal capacity system*</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

* assumed that only compressor enlargement is feasible
It can be easily seen, even from Table 1 by comparing high and low efficiency systems, that performance degradation issues for the equipment containing R410A refrigerant become more profound at relatively high ambient temperatures, where the system performance is valued and needed the most. This fact alone can drive technology innovation and justify its insertion. As known, the commercial applications are drastically different from the residential installations with respect to the operating environment. Residential units are ARI-rated and operated at lower ambient conditions, far away from the critical temperature, and don’t experience the critical point phenomenon. This is exactly why no performance degradation is observed and no additional enhancement measures are required for the residential R410A systems. Consequently, the aforementioned issues are primarily related to the commercial equipment.

Let’s turn our attention to the low-pressure replacements for R22, and R134a in particular. R134a stands out as a substance with zero ozone depleting potential (ODP) and a relatively high critical temperature of 101.1°C (for reference, the critical temperature for R22 is 96.1°C and for R410A – 72.5°C). R134a has comparable evaporation heat transfer coefficients with R22 and 15% higher condensation heat transfer coefficients but, due to lower density, 50% higher pressure drop for similar mass flow rates and operating conditions. Obviously, R134a is a viable candidate for R22 substitution and should not be overlooked, however the equipment size concerns, directly related to the pressure drop characteristics and addressed at the component level with respect to the compressor, piping and heat exchanger design, become crucial considerations and major obstacles for the implementation of R134a.

3. ECONOMIZER CYCLE

The economizer cycle is well justified for high compression ratio conditions, such as refrigeration applications, where a 30% increase in performance (capacity in this case) provides a substantial advantage. Until recently, the economizer cycle was not considered a cost-effective solution for air conditioning installations, where conventional technology was pushed to the limit. Oversized heat exchangers (condensers and evaporators) reached their effectiveness plateaus and compressor volumetric and isentropic efficiencies approached the respective maximums. Now, however, the implementation of environmentally sound refrigerants has negatively affected system performance at certain conditions, and industry standards and government regulations have raised the bar for the minimum efficiencies of the air conditioning equipment, a trend that will continue in the future. This course of events has opened the door to various kinds of alternate technologies, encouraging a more serious evaluation of the economizer cycle.

3.1. Principle of operation

Although the economizer cycle schematics may differ from one another with respect to specific system configuration and implementation details, two basic design solutions exist, one of which is shown in Figure 1. The other scheme can be easily deduced from Figure 1 by relocating an economizer branch diversion point downstream of an economizer heat exchanger (see Figure 1a). The latter scheme is typically utilized to ensure a subcooled state of the refrigerant entering an auxiliary expansion device, but potentially at the expanse of a larger economizer heat exchanger. The economizer cycle is more complex than the conventional system, integrates additional components, and entails more sophisticated control algorithms. When the economizer cycle is in operation, a portion of liquid high-pressure refrigerant is diverted from the main circuit and rerouted through the economizer loop, where its temperature and pressure are reduced to some intermediate level in an auxiliary expansion device.
Therefore, the colder economizer flow can be utilized, through the heat transfer interaction in the economizer heat exchanger, to further boost subcooling of the main refrigerant. This extra subcooling allows for the enthalpy difference increase in the evaporator and subsequent system performance augmentation (see the P-h diagram in Figure 2). Downstream of the economizer heat exchanger, the auxiliary flow is returned to the intermediate compressor port, usually at the superheated state controlled by the auxiliary expansion device. When the colder economizer flow is mixed with the partially compressed suction vapor, temperature of the latter is reduced, benefiting the compression process. Furthermore, the bypass loop between the economizer and suction compressor ports, also exhibited in Figures 1 and 1a, can be employed for the system unloading strategy. As known, the flash tank can be utilized in place of the economizer heat exchanger. As known, the flash tank, separating liquid and vapor phases and essentially representing a 100% effective heat exchanger, can be utilized in place of the economizer.

Although the economizer cycle advantages are obvious, the full benefits of the concept are slightly diminished by three phenomena concurrently occurring in the system. First, the evaporation temperature is slightly reduced, negatively impacting refrigerant flow rate at the compressor suction port and subsequently system capacity. This side effect of extra subcooling and system rebalancing to maintain adequate refrigerant superheat at the evaporator exit is of a second-order significance and, since it equally affects capacity and power, no change in the system efficiency is registered. Second, the refrigerant flow rate in the condenser is increased, promoting condensation temperature elevation. Since the economizer flow is relatively small in air conditioning applications while the condenser coils are typically of sufficient size, this concern can easily be addressed through proper condenser design. Lastly, additional compressor power consumption is expected, as extra economizer flow must be compressed from the intermediate to the discharge pressure. These three phenomena moderate, but do not completely overshadow, the performance benefits achieved in the “ideal” economizer cycle, at most operating conditions. The economizer cycle performance diminishes with the decrease in operating pressure ratio and subsequent decrease of the available thermal potential in the economizer heat exchanger, approaching the conventional system characteristics at the point where the aforementioned losses become equal to the performance gains.

The economizer cycle advantages can be utilized in two different ways. On one hand, an equivalently sized economized compressor can be used to deliver more capacity to the system. In another approach, the compressor size reduction by about 8 to 10% will lead to the system thermodynamic efficiency boost (since the heat exchanger size remains the same), while preserving the capacity characteristics at the ARI design point. Management of all system losses, especially in the compressor and economizer heat exchanger, are crucial for the economizer approach to be successful in air conditioning applications. Hence, it is not uncommon for the economizer heat exchanger effectiveness to reach 90%, promoting extremely close temperature approaches of nearly 0.5°F. The compressor injection scheme is of high significance as well. Therefore, the intermediate compression pockets must have an optimal and restricted exposure to the economizer ports, limiting cyclic parasitic compression and expansion losses associated with the pressure difference in the economizer loop and in the aforementioned compression pockets. For instance, it was observed that the economizer effectiveness reduction by 10-15% or utilization of the conventional compressor injection scheme leads to a decline in the economizer cycle performance gains by 30 to 50%, making the approach extremely difficult to justify.
3.2. Performance

3.2.1. Standard rating conditions
As discussed above, the two strategies, namely equal capacity approach and equal efficiency tactic, are feasible. In each scenario, it is prudent to determine the performance enhancement minimum for the economizer cycle, particularly at the ARI standard rating conditions\(^1\) for commercial equipment. A midsize packaged scroll compressor system of premium efficiency would serve as a logical contender for such a study. For the purpose of this analysis, the condenser and evaporator dimensions and circuiting were preserved, although circuiting optimization could provide a performance enhancement potential. The test results closely matched by the analytical model predictions are summarized in Table 2. As shown in Table 2, the R410A economizer cycle provides a boost in performance of at least 4 - 5% at the ARI conditions. Also, it becomes obvious that the economizer cycle may not be required to match the R22 performance for some premium efficiency systems. However, if further performance enhancement is desirable or a number of systems of various capacities share an identical chassis size, the economizer cycle becomes a useful technique to achieve a desired target. In such cases, twice as much enhancement (in comparison to the numbers exhibited in Table 2) can be obtained from the economizer approach, while the conventional technology will experience rapidly diminishing returns on investment or become physically impossible. One of the side benefits of the economizer cycle is its augmented dehumidification capability, due to lower evaporation temperatures, that potentially can be utilized in hot and humid environments to replace the reheat coil methodology.

![Table 2](image)

<table>
<thead>
<tr>
<th>Design Option</th>
<th>Identical Compressor Size</th>
<th>Reduced Compressor Size</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Capacity, [%]</td>
<td>EER, [%]</td>
</tr>
<tr>
<td>Design basis (R22)</td>
<td>100</td>
<td>100</td>
</tr>
<tr>
<td>R410 economized</td>
<td>105</td>
<td>100</td>
</tr>
<tr>
<td>R410A conventional</td>
<td>100</td>
<td>100</td>
</tr>
</tbody>
</table>

3.2.2. High ambient temperatures
When a life-cycle cost analysis of any air conditioning equipment is performed, system operation at a wide spectrum of environmental conditions must be examined. It is not unusual that a high ambient temperature region attracts particular attention, since at those conditions the system performance is most often demanded and valued by end customers. Any refrigerant system suffers performance degradation in high ambient environments, particularly in applications where the operation requirements are stretched to temperatures as high as 57.2°C. The downward trend for the equipment containing the R410A refrigerant is more profound than the respective decline for the R22 systems. In some cases, the performance parity at the full-load ARI conditions (and at the 26.7°C /19.4°C -26.7°C part-load conditions) is not a sufficient criterion to obtain identical life-cycle cost characteristics over the entire system operation envelope. Distinct design points of the equivalent R410A performance must be chosen to adequately represent a region of high ambient temperatures for various classes of applications. Some R410A systems must have superior performance at the ARI conditions as compared to the equivalent R-22 equipment, in order to have parity at high ambient temperatures.

---

\(^1\) The ARI standard rating conditions are 26.7°C dry bulb and 19.4°C wet bulb indoor air temperatures, and 35°C outdoor air temperature.
The packaged commercial unit of premium efficiency that we used in this study serves here as well to determine minimum performance degradation for the R410A systems at high ambient temperatures. Table 3 displays performance characteristics of the R410A systems as compared to the equivalent R22 systems operating at the performance parity at the ARI conditions. The 51.7°C ambient temperature limit reflects the absolute minimum demanded by a vast majority of commercial applications today. The R410A economizer cycle reveals superior performance and is lagging the R22 system by only 5%, while the conventional R410A system exhibits twice as much performance degradation. Obviously, the gap between the R410A and R22 conventional systems (as well as the R410A conventional and economizer systems) becomes much deeper for less efficient equipment and at higher ambient temperatures. As a result, the equipment will accumulate more hours at less efficient operation, in order to satisfy external load demands at those conditions.

<table>
<thead>
<tr>
<th>Design Option</th>
<th>35°C Outdoor Temperature</th>
<th>51.7°C Outdoor Temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Capacity, [%]</td>
<td>EER, [%]</td>
</tr>
<tr>
<td>Design basis (R22)</td>
<td>100</td>
<td>100</td>
</tr>
<tr>
<td>R410A economized</td>
<td>100</td>
<td>104</td>
</tr>
<tr>
<td>R410A conventional</td>
<td>100</td>
<td>100</td>
</tr>
</tbody>
</table>

For R134a, similar critical temperatures and somewhat analogous shapes of the two-phase region in the vicinity of the critical point suggest that both R134a and R22 refrigerants exhibit comparable performance degradation at high ambient temperature conditions. Furthermore, the R134a refrigerant has a lower compressor discharge temperature (by 12 to 15°F), promoting higher reliability and translating to a wider compressor operation envelope. The relative performance results for all three refrigerants are presented in Figures 3 and 4, with the R134a indeed revealing less performance reduction than the R410A at high ambient temperatures.

Lastly, we can conclude that it requires significantly less effort and investment to reach performance parity at high ambient for the R410A economizer systems, while any conventional approach may become thermodynamically prohibitive.

### 3.2.3. Unloading

A conventional multi-circuit system unloads by turning circuits on and switching them off to satisfy external load demands and to control temperature and humidity within the conditioned space. For the economizer system, multiple additional steps of unloading can be accomplished naturally, reducing life-cycle cost of equipment and augmenting part-load performance characteristics. More particularly, the economizer cycle capacity can be decreased by bypassing a portion of the refrigerant between the economizer and suction compressor ports (see Figure 1) as well as by operating the system in the economizer and conventional modes. This approach offers four operation modes for each independent circuit within the system: 1) conventional mode, 2) economizer mode, 3) conventional mode with bypass and 4) economizer mode with bypass. These four modes offer numerous opportunities for the unloading strategy. It should be noted that unloading through the bypass between discharge and economizer ports is also feasible. Furthermore, adjustable flow control devices can be utilized for the economizer expansion and bypass valves, offering an infinite number of unloading steps through modulation or pulsation techniques and tremendous flexibility in control of various operational parameters. Such parameters may include (but are not limited to) system capacity, compressor discharge temperature and power, and conditioned space temperature and humidity. For instance, for abnormally high ambient temperatures, a sequence of small unloading steps can
limit compressor power while preserving system performance, extending the operation envelope, and preventing nuisance shutdowns. Thus, continually changing load demands are satisfied with greater precision, keeping the conditioned space parameters within the comfort zone, eliminating temperature and humidity variations, and enhancing system reliability and efficiency through a reduction of start-stop cycles.

4. HEAT PUMPS

Heat pumps simultaneously serve cooling and heating markets, covering broader applications and growing in their importance. Such a trend offers a tremendous opportunity for the economizer cycle, since the heat pump equipment usually operates at the elevated pressure ratios (in comparison to the traditional air conditioning installations), while having sufficiently high performance targets. The economizer cycle has only recently entered the air conditioning arena and most likely will penetrate the heat pump market in the near future. Heat pump designs incorporating an economizer loop are more complex than the traditional systems and respectively include an additional four-way valve (see Figure 5), an additional economizer heat exchanger (see Figure 6) or an extra main expansion device–check valve assembly (see Figure 7). In the schematic exhibited in Figure 7, the heating cycle expansion device has to be closed while the system is operating in the cooling mode, and visa versa. Obviously, many variations of these basic designs (primarily related to the economizer loop connections and insertion of additional components such as an accumulator) are foreseeable. Although extra system complexity, additional hardware, and increased cost are to be justified by the benefits obtained from the economizer cycle performance, a vacuum in potential contenders will further promote the technology.

5. CONCLUDING REMARKS

It is apparent that the economizer cycle offers numerous benefits for air conditioning applications, but at the same time introduces several design, and application challenges that must be properly addressed. In particular, the economizer cycle provides a competitive advantage, where other methodologies fail to be economically viable or become physically impossible. A typical performance augmentation for the economizer system is expected to be approximately 8%. Such an improvement can easily compensate for the R410A performance deficiency at the ARI rating point for commercial equipment. Furthermore, the economizer cycle demonstrates its superiority at high ambient operation, where it outperforms conventional systems on average by 10%. Although this is a significant improvement, issues associated with R410A performance degradation in such environments must be addressed further, potentially by a combination of the economizer cycle and traditional technology. Other benefits of the economizer cycle are related to its flexibility in the unloading strategy, improved dehumidification capability and superior operation in the heat pump installations. The economizer concept can be employed in a multi-circuit system configuration as well, where economizer heat exchangers associated with each circuit can be combined into a single unit.

Although strong arguments can be presented, based on performance, for R134a to be the refrigerant of choice for packaged commercial equipment, the cost concerns develop into a major obstacle, even for the economizer cycle.
REFERENCES


FIGURE 1 ECONOMIZER CYCLE

FIGURE 1a ECONOMIZER CYCLE
(ALTERNATE CONFIGURATION)

FIGURE 2 P-h DIAGRAM FOR ECONOMIZER CYCLE
Minimum Capacity Degradation

![Graph showing Minimum Capacity Degradation across different ambient temperatures for R134A Conventional, R134a Conventional, R410A Economized, and R410A Conventional refrigerants. The graph indicates capacity ratio (relative to R22) as a function of ambient temperature (in °C).]

Minimum EER Degradation

![Graph showing Minimum EER Degradation across different ambient temperatures for ARI Rating Points, R134a Conventional, R410A Economized, and R410A Conventional refrigerants. The graph indicates EER ratio (relative to R22) as a function of ambient temperature (in °C).]
Determination of the thermodynamic feedback of the shell of a small hermetic piston compressor

R. A. Almbauer, Z. Abidin, A. Burgstaller
CD Laboratory for the Thermodynamics of Reciprocating Engines, Institute of Internal Combustion and Thermodynamics, Graz University of Technology, Austria

ABSTRACT

The thermodynamic performance of a piston compressor mainly depends on the temperature level of the refrigerant and its temperature development during the compression phase. In addition the four major losses (a) pressure drop and (b) temperature gain in the suction system, (c) top dead centre volume and (d) pressure loss in the discharge system are important. Concerning a hermetic piston compressor the shell produces a second thermodynamic system boundary in addition to the cylinder – piston system. So each cooling of compressor parts in the shell leads to an increase of temperature level of the shell. The temperature level in the shell itself influences the temperature level of the refrigerant in the suction part, the cylinder and the discharge line. This thermodynamic feedback of each cooling heat reduces or vanishes the positive effect on the COP. The paper deals with the simulation and the experimental verification of the feedback for a 12 ccm piston compressor for isobutane (R600a). The improvement of the COP for a 9 K lower refrigerant temperature at the exit of the suction muffler is evaluated. The results show a considerably smaller COP gain compared to the same temperature reduction for an open compressor.

1 INTRODUCTION

The energy efficiency of hermetic compressors for household refrigeration will become more and more important in the future. This is due to the fact that in the EU-directive 2003/66/EC the classes A+ and A++ are introduced on top of the existing energy efficiency class A. Energy efficiency class A+ is defined to use energy between 99% and 71% of class A (100%). Class A++ must be under 71% of class A for the same cooling capacity. The most important influences on the compression work are the pressure ratio and the temperature level of the gas. Applying these two principles to the suction muffler, it must keep the heat transfer and the flow resistance as low as possible. The gas movement in the muffler is highly transient. During the suction period the gas is sucked through the volumes and pipes of the muffler resulting in velocities of up to 60 m/s. In addition the muffler has to catch as much of the cold gas coming from the evaporator as possible. Therefore the flow around the muffler entrance is of major importance for higher efficiency. The flow can be simulated using commercial CFD tools. Fagotti and Possamai [1] assess the application of CFD in the development process of compressors and approve its applicability and usefulness. 1D and 3D CFD simulation tools must in any case be capable of calculating the flow of the refrigerant which behaves as a real

© The Authors 2005
gas. Choi et al. [2] showed the CFD model application for the development of a smart suction muffler and its validation by measurements. Chikurde et al. [3] investigate the temperature distribution using 3D CFD analysis in combination with a conjugate heat transfer calculation. The results of the flow field and the gas temperature increase in the muffler are comparable to the results which are published by Abidin and Almbauer [4]. The simulation shows an increase of approximately 33 K for a 12 ccm piston compressor for isobutane (R600a). The calculated pressure loss is around 3100 Pa.

In the frame of a research project a new design of a suction muffler has been created. In a CFD study a reduction of the gas temperature in front of the inlet valve of 9 K has been found. The 3D-simulation included only the suction line. Therefore the influence on the COP was assessed by a 1D CFD simulation tool. This commercial simulation model has been adopted for the calculation of the piston compressor and has been validated with extensive measurement data for the base case [5]. The adoption includes also the determination of temperature boundary conditions for the muffler walls, the cylinder, the valve plate and the piston. The use of the temperature boundary conditions of the base case results in an overestimated COP increase. The negative feedback from the shell increases the temperature of all boundaries. This will be elaborated in the following sections, which include the model description, estimation for the change in boundary conditions and experimental results, and finally conclusions. In figure 1 the most important parts of the piston compressor are shown.

![Diagram of the compressor model and its components](image)

**Figure 1: The compressor model and its components**

2 **NUMERICAL SIMULATION USING A 1D MODEL**

The numerical simulation aims at the creation of a model which is able to describe all thermodynamic and physical processes in the hermetic piston compressor. For the 1D simulation the software BOOST (AVL) [6] which has been developed for internal combustion engines is applied. Therefore several adoptions have been carried out to be able to simulate piston compressors for the refrigerant isobutane (R600a). The validation for the real gas behavior has been made for special cases like isobaric cooling, isentropic compression and comparison with fast pressure measurements from a pressure indication test bed. The results show a sufficient consistency in the required pressure and temperature range.
2.1 Theoretical Basis of the 1D software BOOST

2.1.1 High pressure cycle

The high pressure cycle is determined by the first law of thermodynamics in combination with the equation of state for real gases. Some of the derivatives are made with respect to crank-angle \((d\alpha)\). The others are made with respect to time \((dt)\). Nevertheless both are coupled by the rotational speed.

\[
\frac{d(m \cdot e)}{d\alpha} = -p \frac{dV}{d\alpha} - \sum \frac{dQ_w}{d\alpha} - h_l \cdot \frac{dm_i}{d\alpha}
\]

Eq. 1

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(m)</td>
<td>mass</td>
</tr>
<tr>
<td>(p)</td>
<td>pressure</td>
</tr>
<tr>
<td>(V)</td>
<td>volume</td>
</tr>
<tr>
<td>(e)</td>
<td>internal energy</td>
</tr>
<tr>
<td>(d(m \cdot e)/d\alpha)</td>
<td>change of internal energy in the cylinder</td>
</tr>
<tr>
<td>(-p \cdot dV/d\alpha)</td>
<td>piston work</td>
</tr>
<tr>
<td>(\sum \frac{dQ_w}{d\alpha})</td>
<td>sum of all wall heat loses (piston, liner, and valve plate)</td>
</tr>
<tr>
<td>(-h_l \cdot \frac{dm_i}{d\alpha})</td>
<td>enthalpy flow due to leakage</td>
</tr>
</tbody>
</table>

\[p \cdot V = m \cdot Z \cdot R \cdot T\]

Eq. 2

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(R)</td>
<td>gas constant</td>
</tr>
<tr>
<td>(Z)</td>
<td>real gas factor</td>
</tr>
</tbody>
</table>

2.1.2 Gas exchange process in the cylinder

The gas exchange process is also governed by the first law of thermodynamics.

\[
\frac{d(m \cdot e)}{d\alpha} = -p_e \cdot \frac{dV}{d\alpha} - \sum \frac{dQ_w}{d\alpha} - \sum h_e \cdot \frac{dm_i}{d\alpha} + \sum h_i \cdot \frac{dm_e}{d\alpha}
\]

Eq. 3

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(h_i)</td>
<td>sum of all enthalpy flows into the cylinder</td>
</tr>
<tr>
<td>(h_e)</td>
<td>sum of all enthalpy flows out of the cylinder</td>
</tr>
<tr>
<td>(d(m \cdot e)/d\alpha)</td>
<td>change of internal energy in the cylinder</td>
</tr>
<tr>
<td>(-p_e \cdot dV/d\alpha)</td>
<td>piston work</td>
</tr>
<tr>
<td>(\sum \frac{dQ_w}{d\alpha})</td>
<td>sum of all wall heat loses (piston, liner, and valve plate)</td>
</tr>
<tr>
<td>(\sum h_e \cdot \frac{dm_i}{d\alpha})</td>
<td>enthalpy flow due to leakage</td>
</tr>
</tbody>
</table>

The mass balance in the cylinder is described by the following equation:

\[
\frac{dm}{d\alpha} = \sum \frac{dm_i}{d\alpha} - \sum \frac{dm_e}{d\alpha}
\]

Eq. 4

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(i,e)</td>
<td>into, out</td>
</tr>
</tbody>
</table>

The mass flows in the inlet and outlet valves are approximated by the following equations for isotropic jet flow:
\[
\frac{dm}{dt} = A \cdot p_1 \cdot \sqrt{\frac{2}{R \cdot T_1}} \cdot \psi 
\]

\( A \) ... free flow area

For any pressure ratio the flow function \( \psi \) can be calculated with:

\[
\psi = \left[ \frac{\kappa}{\kappa - 1} \left( \frac{p_2}{p_1} \right)^{\frac{2}{\kappa}} - \left( \frac{p_2}{p_1} \right)^{\frac{\kappa+1}{\kappa}} \right] 
\]

\( \kappa = c_p / c_v \) ... ratio of specific heat for constant pressure to specific heat for constant volume

Indices 1, 2 ... thermodynamic conditions at locations 1 and 2

### 2.1.3 Cylinder heat transfer

In order to calculate the heat transfer coefficient between the gas and the cylinder the following equation from Hohenberg (BOOST [5]) is used:

\[
\alpha = 130 \cdot V^{-0.06} \cdot \rho^{0.8} \cdot T^{-0.4} \cdot (u_m + 1.4)^{0.8} 
\]

\( u_m \) ... mean flow velocity in the cylinder

### 2.1.4 Plenum

The gas dynamics in plenums are calculated with the first law of thermodynamics (Eq. 3) without a change of the volume. The heat transfer coefficient is calculated in BOOST using:

\[
\alpha = 0.018 \cdot \frac{\kappa}{\kappa - 1} \cdot R \cdot \rho \cdot u_m^{0.8} \cdot L_{ch}^{0.2} \cdot 0.2 \cdot (0.127 + 1.3 \cdot T \cdot 10^{-4}) 
\]

\( L_{ch} = \frac{\sqrt{V}}{u_{ch}} \) ... characteristic length

\( u_{ch} \) ... characteristic velocity

### 2.1.5 Pipe flow

The pipe flow is determined by the continuity equation:

\[
\frac{\partial \rho}{\partial t} = -\frac{\partial (\rho \cdot u)}{\partial x} - \rho \cdot u \frac{1}{A} \frac{dA}{dx} 
\]

\( \rho \) ... density

the momentum equation:

\[
\frac{\partial (\rho \cdot u)}{\partial t} = -\frac{\partial (\rho \cdot u^2 + p)}{\partial x} - \rho \cdot u^2 \frac{1}{A} \frac{dA}{dx} - \frac{F_R}{V_{cell}} 
\]

\( F_R \) ... friction force

and the energy conservation equation:
\[
\frac{\partial E}{\partial t} = -\frac{\partial \left[ u \cdot (E + p) \right]}{\partial x} - u \cdot (E + p) \cdot \frac{1}{A} \cdot \frac{\partial A}{\partial x} + \frac{Q_w}{V}
\]
Eq. 11

\[
E = \rho \cdot c_v \cdot T + \frac{1}{2} \cdot \rho \cdot u^2 \quad \text{... energy content of the gas}
\]

2.1.6 **Heat flow in the pipe**
The convective heat transfer is estimated by standard Nusselt number theory.

2.1.7 **Numerical Solver**
The above mentioned coupled equations are numerically solved using an iterative solver for compressible flow. The converged solution fulfills all equations in each of the finite volumes for each time step. An ENO scheme is used for the solution of the convection terms. The set of non-linear differential equations described above are solved in a coupled way. The calculation for a steady state result needs several cycles in order to damp out initial conditions.

2.2 **BOOST model**

![BOOST model diagram](image)

**Figure 2: BOOST model of the compressor**

The pre-processor of BOOST allows the graphical assembling of the piston compressor (Figure 2). All pipes and plenums in the suction and the discharge line are represented by lines and rectangles. The cylinder with the attached valves is represented by the circle. The system boundaries on both ends represent the inlet and outlet conditions.

3 **VALIDATION OF THE MODEL**

3.1 **Test case**
The validation of the simulation model has been carried out for a commercial 12 ccm piston compressor for isobutane. The test case is described by the ASHRAE conditions with a surrounding temperature of 32 °C, an evaporation temperature of -23.3 °C and a condensation temperature of 55 °C. The temperature of the refrigerant at the inlet is kept constant at 32 °C. The COP of the standard compressor is in the range of 1.65 for the ASHRAE test case.
3.2 Validation

The validation of the BOOST model has been carried out using pressure measurements in the cylinder, in the area in front of the inlet valve and in the cylinder top cover close to the outlet valve (Pischinger [5]). The valve movement has been evaluated using inductive sensors (Figure 3). Six thermocouples have been applied to measure surface temperatures and gas temperatures. Finally the indicated work per cycle has been assessed using efficiency factors for mechanical friction and the losses of the electric motor. As an example the measured and simulated pressure history for the whole cycle is given in Figure 4. The experimental setups for temperature and pressure measurements are shown in Figure 5. Table 1 shows the global comparison between experimental and numerical results.

![Figure 3: Measured valve lift curve](image)

**Table 1: Global comparison: experimental data vs. numerical simulation**

<table>
<thead>
<tr>
<th>Gas temperature</th>
<th>Measurement standard muffler</th>
<th>Simulation standard muffler</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet of muffler</td>
<td>48.85 °C</td>
<td>48.85 °C (Boundary Cond.)</td>
</tr>
<tr>
<td>Trumpet of muffler inlet</td>
<td>49.0 °C</td>
<td>50.29 °C</td>
</tr>
<tr>
<td>Muffler neck before inlet valve</td>
<td>59.0 °C</td>
<td>58.63 °C</td>
</tr>
<tr>
<td>Cylinder head</td>
<td>123.2 °C</td>
<td>140.91 °C (max. ( T_{gas} ))</td>
</tr>
<tr>
<td>outlet of serpentine</td>
<td>85.6 °C</td>
<td>84.45 °C</td>
</tr>
</tbody>
</table>
4 DEVELOPMENT OF A NEW MUFFLER

A 1D simulation of the compressor which includes the muffler entrance is not appropriate due to 3D effects in that area. So a new muffler design has been evaluated using 3D CFD simulation. The simulation results show a reasonable decrease of the gas temperature at the entrance of the inlet valve in the order of 9 K compared to the standard muffler. The boundary conditions for the 3D calculation have been kept constant for both simulations.

4.1 Theoretical Basis for 3D- Simulation

The simulation in 3D was calculated using the software FLUENT and is described in [4]. The basic equations are given by:

continuity equation:
\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \cdot \mathbf{u}) = 0 \tag{Eq. 12}
\]
momenum equation:
\[
\frac{\partial (\rho \cdot u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (\tau_{ij} + \rho S_i) \tag{Eq. 13}
\]
\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) \quad \text{... rate of increase of momentum per unit volume}
\]
\[-\frac{\partial p}{\partial x_i} \quad \text{... pressure force per unit volume}
\]
\[
\frac{\partial \tau_{ij}}{\partial x_j} \quad \text{... viscous force per unit volume}
\]
\[
S_i \quad \text{... body forces per unit volume}
\]
\[
i, j \quad \text{... indices for Einstein’s summation convention}
\]

energy equation:
\[
\frac{\partial (\rho h)}{\partial t} + \frac{\partial (\rho u_i h)}{\partial x_i} = \frac{\partial}{\partial t} \left( \rho \frac{\partial T}{\partial x_i} + \frac{1}{2} k \frac{\partial T}{\partial x_i} \right) + W_h + S_h \tag{Eq. 14}
\]
\[
\frac{\partial (\rho h)}{\partial t} + \frac{\partial (\rho u_i h)}{\partial x_i} \quad \text{... rate of increase of enthalpy per unit volume}
\]
\[
\frac{\partial}{\partial x_i} \left( \rho \frac{\partial T}{\partial x_i} \right) \quad \text{... heat conduction per unit volume}
\]
\[
W_h \quad \text{... viscous work per unit volume}
\]
\[
S_h \quad \text{... energy source per unit volume}
\]

For turbulence modeling the standard k-ε model is used. This is two-equation model which solves the conservation equations of turbulent kinetic energy (k) and dissipation rate (ε).

4.2 Comparative 3D results for the muffler simulation

![Figure 6: Gas heating through both mufflers](image-url)
In Figure 6 the gas heating through the suction muffler is given as a comparative presentation between the standard muffler and the new developed muffler. Both of them are calculated using the 3D Software FLUENT.

4.3 Numerical simulation and analysis

In order to assess the gain in COP the 1D model has been modified and applied to the new test case. The modification has been carried out in a simplified way. It was not able to create a model which represents the design modification. Therefore the inlet temperature at the system boundary has been reduced by 10 K, which resulted in a 9 K lower gas temperature at the inlet valve. The 1D model simulated a COP gain of 2.5%. This was the reason to produce the new muffler type and to test it. The expected COP gain of the simulation results of 2.5% were not reached in the measurements. The COP gain was approximately 1.55% for the prototype. The reason for the reduced COP gain lies obviously in the negative feedback of the shell. Evaluating the results of Abidin and Almbauer [4] the original muffler design produces a temperature increase of approx. 15 K from the entrance of the shell to the inlet of the muffler. Assuming a fresh gas temperature of 35 °C at the entrance of the shell and a shell temperature of 60 °C, this reflects a mixing of cold fresh gas and warm shell gas in the mass proportion of 40% to 60% in this area. Only 40% of the fresh gas can be directly sucked into the muffler. The new muffler concept is able to catch 80% of the fresh gas. This reduces the cooling of the shell volume by 40% of the fresh gas. From simulations and measurements the heat transfer coefficient for the shell is known. Taking the heat transfer over the shell into account the reduced cooling (approximately 11 W) should result in a temperature increase of approximately 6 K in the shell volume.

Table 2: Results of 1D simulation of different cases

<table>
<thead>
<tr>
<th>Parameter</th>
<th>standard muffler</th>
<th>modified muffler without influence of temperature</th>
<th>modified muffler with influence of temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean gas temperature at the muffler trumpet</td>
<td>50.29 °C</td>
<td>38.10 °C</td>
<td>42.62 °C</td>
</tr>
<tr>
<td>Mean gas temperature at muffler neck</td>
<td>58.63°C</td>
<td>50.38 °C</td>
<td>54.46 °C</td>
</tr>
<tr>
<td>Gas temperature in the cylinder head</td>
<td>140.91 °C</td>
<td>138.42 °C</td>
<td>139.70 °C</td>
</tr>
<tr>
<td>Gas temperature in first discharge chamber</td>
<td>137.35 °C</td>
<td>133.8 °C</td>
<td>135.62 °C</td>
</tr>
<tr>
<td>Gas temperature in second discharge chamber</td>
<td>132.12 °C</td>
<td>127.07 °C</td>
<td>128.84 °C</td>
</tr>
<tr>
<td>Outlet gas temperature of serpentine</td>
<td>84.45 °C</td>
<td>83.72 °C</td>
<td>89.25 °C</td>
</tr>
<tr>
<td>Calculated mass flow</td>
<td>0.560633 g/s</td>
<td>0.576374 g/s</td>
<td>0.572108 g/s</td>
</tr>
<tr>
<td>Electrical power</td>
<td>116.65 W</td>
<td>117.06 W</td>
<td>117.31 W</td>
</tr>
<tr>
<td>COP (calculated)</td>
<td>1.614</td>
<td>1.654</td>
<td>1.639</td>
</tr>
<tr>
<td>COP (measured)</td>
<td>1.62</td>
<td>--</td>
<td>1.64</td>
</tr>
<tr>
<td>COP relative</td>
<td>100 %</td>
<td>102.5 %</td>
<td>101.55 %</td>
</tr>
</tbody>
</table>

This has several influences on the COP: (a) The mixing of 80% fresh gas with 20% gas from the shell volume results in a 1.2 K warmer mixing temperature. (b) All temperature boundary
conditions for the calculation of the heat transfer are increased by 6 K. (c) It can be assumed that all wall temperatures of the cylinder, the valve plate and the piston are increased by 6 K. Applying the new boundary conditions in the 1D model results in a considerably reduced COP. Table 2 shows the results for the 1D- simulation and measurements for the same case.

5 CONCLUSION

The design of a new modified suction muffler reduces the mixing in front of its inlet. This results in a reduced gas temperature at the inlet valve and leads to an increased mass flow per cycle and a COP rise. In order to evaluate the COP improvement simulation and measurement results have been compared for the ASHRAE test case. There some discrepancies have been found out.

Simulation results depend very much on temperature boundary conditions. One of the most important boundary conditions is the shell temperature as it influences many heat transfer situations. Vice versa all heat transfers in the shell influence the temperature distribution. The shell forms a second system boundary, where all heat fluxes are balanced. This is the reason for a feedback of these heat fluxes on temperatures.

Concerning the modified muffler the reduced mixing of fresh (cold) gas reduces the cooling of the shell volume. This is the reason for an increase of the shell temperature, which has a negative effect on the COP. Generally simulation results and measurements show, that the COP increase due to the reduction of the gas temperature at the inlet valve dominates the COP decrease due to a higher shell temperature. Concluding this findings it is still valuable to reduce the temperature level of the gas in the compression phase although there exists a second system boundary, where a negative feedback might occur.

6 ACKNOWLEDGEMENTS

This research was supported by Christian Doppler Gesellschaft Austria and ACC Austria.

7 REFERENCES

FEA aided discharge tube design for hermetic reciprocating AC/R compressors  
Part 1: Determination of discharge tube FEA boundary conditions

Jianxiong Chen
Technical Services, Emerson Climate Technologies, USA

ABSTRACT

The finite element analyses (FEA) for determining internal discharge tube stresses during compressor startups and shutdowns are required in FEA aided internal discharge tube design. To determine the FEA boundary conditions, seven linear variable displacement transducers (LVDT’s) were used to measure compressor body motions of compressor startups and shutdowns. The time-dependent displacement boundary conditions of the internal discharge tube were calculated from compressor body motions using an optimization procedure for transient finite element analyses based on a 3-D rigid body motion model of compressor body.

1. INTRODUCTION

An air conditioning and refrigeration (AC/R) hermetic reciprocating compressor is sealed inside the hermetic shell; and an internal discharge tube is required to provide a passage for refrigerant between the internal compressor running gear and the outside system. One end of the internal discharge tube connects to the compressor discharge port through a muffler, while the other end of the internal discharge tube exits the compressor shell and is welded to the shell. Usually the discharge tube is designed after the surrounding parts; space limitations add difficulties to the discharge tube design. A discharge tube generally needs several bends to be fitted into the limited space without contacting shell and compressor body.

The test-only discharge tube design is a trial-and-error process. It physically changes the shape and dimensions of the discharge tube. Each new design has to be prototyped, strain gauged, and compressor tested. This method is apparently very time consuming and costly. A new design approach, FEA aided discharge tube design, is thus introduced in this paper. The FEA aided design method virtually simulates the test-only design method on a computer. Since developing and analyzing discharge tube FEA models are relatively cheaper, the new design method reduces developing time and cost. Furthermore, since more trails can be evaluated through FEA, the FEA aided design method can usually reach a better design than the test-only design method.
Compressor tests were performed to determine the FEA boundary conditions of the discharge tube. The compressor body motions during startups and shutdowns were measured by seven LVDT’s and modeled as a 3-D rigid body motion in space. The discharge tube displacement boundary conditions were calculated from the compressor body motion measurements for transient FEA using an optimization procedure. Besides the displacements of the compressor body, the stresses and the first modal frequency of discharge tube were also experimentally determined for verifying the FEA model. Weibull distribution analyses were used to handle the statistical variability of compressor startups and shutdowns.

2. COMPRESSOR TESTS FOR MEASURING COMPRESSOR BODY MOTION

The major objective of the compressor tests was to obtain compressor body displacement data that could describe the rigid-body motion of the compressor body in 3-D space, and then, from the compressor body motion, the displacement boundary conditions of the discharge tube could be determined for FEA aided discharge tube design. Seven LVDT’s were used: two in X direction (perpendicular to the piston motion direction), three in Y direction (parallel to the piston motion direction), and two in Z direction (parallel to the crankshaft axial direction). Since a 3-D rigid body motion has 6 degrees of freedom, a minimum of six LVDT’s are needed. At least one LVDT is required for each axis to catch all three translational degrees of freedom of compressor body motion. The LVDT’s were positioned away from each other to increase the differences of LVDT readings for higher accuracy of calculations. Using more than six LVDT’s is desirable to improve accuracy by reducing noise via an optimization procedure discussed later in this paper. The internal discharge tube used for the compressor tests and the fixed coordinate system are shown in Figure 1. A compressor with a bolted shell was used. The LVDT armature was sealed in a housing welded to the shell, see Figure 2. The LVDT rod was connected to the compressor body through a joint. The joint and the radial clearance between the LVDT rod and the armature provided sufficient rotational degrees of freedom needed when the location of measurement on the compressor body moves away from the LVDT axial centerline.

Two strain gages were laid on the discharge tube near the shell end. The purposes of measuring discharge tube stresses were for verifying the FEA model, determining discharge tube resonant frequency, and studying the variability of startups and shutdowns. Both LVDT data and strain gage data were recorded by a data acquisition system. The first modal frequency of the discharge tube was 83 Hz, determined by a frequency sweep test. Only the first modal frequency of discharge tube was searched. Both standard shell foot mounting using rubber grommets and the solid (rigid) mounting using steel spacers were examined. Compared to the standard mounting, the average maximum stress range with the solid mounting was about 3% higher. The difference was not statistically significant, indicating that the added shell masses due to a bolted shell and LVDT armature housings did not significantly alter the compressor body motion (or the discharge tube boundary conditions).

Inspection of discharge tube stress data revealed that both startup stresses and shutdown stresses exhibited random scatter, which was predominantly due to the different crank shaft positions at the beginnings of startup and shutdown. A number of startup and shutdown tests were thus recorded to investigate the variability. The variability of test data was dealt by statistical distribution analyses; see the Weibull analyses of maximum discharge tube stress ranges for startups and shutdowns in Figure 3. The 80th percentile maximum stress range was used as the equivalent constant-amplitude fatigue stress range that caused the same fatigue

362
damage as the actual variable-amplitude stress range. The percentile used for the equivalent constant-amplitude fatigue stress range is dependent on the stress distribution and the slope of S-N curve. Using 80th percentile tends to be slightly on the conservative side. Accordingly, the LVDT data at the 80th percentile maximum stress range were used to determine the discharge tube displacement boundary conditions, see Figures 4 and 5.

3. DETERMINATION OF DISCHARGE TUBE FEA BOUNDARY CONDITIONS

The compressor body motions were measured relative to the shell as all LVDT’s were attached to the shell. Since the shell and compressor body deformations are negligible compared to the displacement measurements, it was reasonable in this case to consider the shell and compressor body as rigid bodies. The compressor body motions could be thus modeled as rigid body motions relative to a rigid shell in a 3-D space. The discharge tube stresses primarily depended on the relative displacements between its two ends. As the shell end of the discharge tube was fixed to the shell, the displacements at the discharge tube muffler end determined from the compressor body motions were the displacements relative to the shell end. Therefore, when the compressor body motions are determined from the LVDT measurements, the relative displacements at the discharge tube muffler can be calculated from the compressor body motions and used as time-dependent displacement boundary conditions for transient FEA.

In the actual calculations, the displacement boundary conditions were calculated at the muffler inlet. The reason of calculating the displacement boundary conditions at the muffler inlet instead of at the discharge tube muffler end (or the muffler outlet) was to make provision for the future analyses that would include the muffler into FEA. In fact, if only six LVDT’s are used, the LVDT data can be directly applied to the FEA model in which all LVDT measuring points and the discharge tube muffler end can be rigidly connected together to simulate a rigid compressor body. However, when more than six LVDT’s are used, an optimization procedure has to be used to find the six degrees of freedom displacement boundary conditions.

A 3-D rigid body motion has three rotational and three translational degrees of freedom. The position of a moving rigid body at any given time in space can be precisely defined by a 3-D coordinate system transformation for rotation and translation. The coordinate system transformation does not introduce theoretical errors. The possible source of errors could be from LVDT displacement measurements. These errors can be reduced by the optimization of the solution with redundant measurements, or more than six LVDT measurements. Two right-handed Cartesian coordinate systems are involved in the optimization processes:

- The fixed coordinate system \((X, Y, Z)\). Its origin is defined at the center of discharge tube shell end as shown in Figure 1. This coordinate system does not move.
- The compressor body coordinate system \((x, y, z)\). This coordinate system is attached to the compressor body, and moves and rotates with it.

At \(t = 0\) (right before the compressor body starts to move), two coordinate systems are parallel to each other. The origin of the compressor body coordinate system is located at the center of muffler inlet; its coordinates in the fixed coordinate system are \((X_{bo}, Y_{bo}, Z_{bo})\). The location of the \(i^{th}\) LVDT measurement is defined at the center of the joint that connected the LVDT rod to the compressor body. It is denoted by \((X_{s,i}, Y_{s,i}, Z_{s,i})\) in the fixed coordinate system, and by \((x_{s,i}, y_{s,i}, z_{s,i})\) in the moving coordinate system, where, \(i = 1, 2, \ldots, 7\) for seven LVDT’s. \((x_{s,i}, y_{s,i}, z_{s,i})\) can be calculated by the initial conditions at \(t = 0\) by:
\[ x_{s,i} = X_{s,i}(t = 0) - X_{bo}(t = 0) \]
\[ y_{s,i} = Y_{s,i}(t = 0) - Y_{bo}(t = 0) \]
\[ z_{s,i} = Z_{s,i}(t = 0) - Z_{bo}(t = 0) \] (1)

\((x_{s,i}, y_{s,i}, z_{s,i})\) are kept constant during the compressor body motion, which automatically ensures the condition of a rigid body. The location of the \(i^{th}\) LVDT rod tip inside the armature is denoted as \((X_{L,i}, Y_{L,i}, Z_{L,i})\) in the fixed coordinate system. The following relations exist between \((X_{s,i}, Y_{s,i}, Z_{s,i})\) and \((X_{L,i}, Y_{L,i}, Z_{L,i})\):

\[ X_{L,i} = X_{s,i} \pm \sqrt{L_i^2 - (Y_{L,i} - Y_{s,i})^2 - (Z_{L,i} - Z_{s,i})^2} \text{ for LVDT in X direction} \]
\[ Y_{L,i} = Y_{s,i} \pm \sqrt{L_i^2 - (X_{L,i} - X_{s,i})^2 - (Z_{L,i} - Z_{s,i})^2} \text{ for LVDT in Y direction} \]
\[ Z_{L,i} = Z_{s,i} \pm \sqrt{L_i^2 - (X_{L,i} - X_{s,i})^2 - (Y_{L,i} - Y_{s,i})^2} \text{ for LVDT in Z direction} \] (2)

The proper sign for “±” is determined based on the relative positions of the rod joint and tip in the fixed coordinate system. The rod length is calculated by:

\[ L_i = \sqrt{(X_{L,i} - X_{s,i})^2 + (Y_{L,i} - Y_{s,i})^2 + (Z_{L,i} - Z_{s,i})^2} \quad i = 1, 2, ..., 7 \] (3)

Only axial displacements of the LVDT rod tip inside the armature were considered. The radial displacements of the rod tip due to the clearance between rod and armature were neglected.

The compressor body coordinate system can be completely described in space by its position and orientation with respect to the fixed coordinate system. Therefore, the rotation and translation of the compressor body can be defined by transformation of coordinate systems as:

\[
\begin{bmatrix}
X_{s,i} \\
Y_{s,i} \\
Z_{s,i}
\end{bmatrix} =
\begin{bmatrix}
l_1 & l_2 & l_3 \\
m_1 & m_2 & m_3 \\
n_1 & n_2 & n_3
\end{bmatrix}
\begin{bmatrix}
x_{s,i} \\
y_{s,i} \\
z_{s,i}
\end{bmatrix} +
\begin{bmatrix}
X_{bo} \\
Y_{bo} \\
Z_{bo}
\end{bmatrix}
\] (4)

where the transformation matrix for rotation is calculated from three Euler’s angles as:

\[ c_1 = \cos \theta, \quad c_2 = \cos \psi, \quad c_3 = \cos \phi \]
\[ s_1 = \sin \theta, \quad s_2 = \sin \psi, \quad s_3 = \sin \phi \]
\[ l_1 = c_2 \cdot c_3 - c_1 \cdot s_2 \cdot s_3, \quad m_1 = s_2 \cdot c_3 + c_1 \cdot c_2 \cdot s_3, \quad n_1 = s_1 \cdot s_3 \]
\[ l_2 = -c_2 \cdot s_3 - c_1 \cdot s_2 \cdot c_3, \quad m_2 = -s_2 \cdot s_3 + c_1 \cdot c_2 \cdot c_3, \quad n_2 = s_1 \cdot c_3 \]
\[ l_3 = s_1 \cdot s_2, \quad m_3 = -s_1 \cdot c_2, \quad n_3 = c_1 \] (5)

The locations of LVDT probe tips at \(t>0\) can be determined from LVDT measurements, on the other hand, and can also be predicted from the model developed above. The objective function measured the agreement between the measurements and the model predictions with a particular set of variables. In this case, the chosen variables are three Euler’s angles and \((X_{bo}, Y_{bo}, Z_{bo})\), which fully describe the coordinate system transformation. Based on the measuring direction, the measured and predicted LVDT tip locations and the differences between them can be determined by following equations:
• Case 1: the LVDT measuring direction is in $X$ direction:

$$ X_{L,i}(measured) = X_{L,i}(t = 0) + \Delta X_{L,i} $$

$$ X_{s,i} = l_1 \cdot x_{s,i} + l_2 \cdot y_{s,i} + l_3 \cdot z_{s,i} + X_{bo} $$

$$ X_{L,i}(predicted) = X_{s,i} \pm \sqrt{I_1^2 - (Y_{L,i} - Y_{s,i})^2 - (Z_{L,i} - Z_{s,i})^2} $$

$$ d_i = X_{L,i}(measured) - X_{L,i}(predicted) $$

(6)

• Case 2: the LVDT measuring direction is in $Y$ direction:

$$ Y_{L,i}(measured) = Y_{L,i}(t = 0) + \Delta Y_{L,i} $$

$$ Y_{s,i} = m_1 \cdot x_{s,i} + m_2 \cdot y_{s,i} + m_3 \cdot z_{s,i} + Y_{bo} $$

$$ Y_{L,i}(predicted) = Y_{s,i} \pm \sqrt{I_1^2 - (X_{L,i} - X_{s,i})^2 - (Z_{L,i} - Z_{s,i})^2} $$

$$ d_i = Y_{L,i}(measured) - Y_{L,i}(predicted) $$

(7)

• Case 3: the LVDT measuring direction is in $Z$ direction:

$$ Z_{L,i}(measured) = Z_{L,i}(t = 0) + \Delta Z_{L,i} $$

$$ Z_{s,i} = n_1 \cdot x_{s,i} + n_2 \cdot y_{s,i} + n_3 \cdot z_{s,i} + Z_{bo} $$

$$ Z_{L,i}(predicted) = Z_{s,i} \pm \sqrt{I_1^2 - (X_{L,i} - X_{s,i})^2 - (Y_{L,i} - Y_{s,i})^2} $$

$$ d_i = Z_{L,i}(measured) - Z_{L,i}(predicted) $$

(8)

where $\Delta X_{L,i}$, $\Delta Y_{L,i}$, and $\Delta Z_{L,i}$ were the LVDT readings in $X$, $Y$, and $Z$ directions respectively. Finally the objective function is:

$$ F(\theta(t), \psi(t), \phi(t), X_{bo}(t), Y_{bo}(t), Z_{bo}(t)) = \sum w_i \cdot d_i^2 $$

(9)

where $w_i$'s are weighting functions. The objective function is arranged so that small values represent close agreement. Therefore, minimization of the objective function yields the best-fit solution. The minimization, also called optimization, was a minimizing process of the objective function in six dimensions due to six degrees of freedom. The Powell’s method of function optimization was used.

FEA displacement boundary conditions also include three rotational degrees of freedom and three translational degrees of freedom, but defined in a different way. The three rotational degrees of freedom defined in ANSYS (FEA software used in the analyses) are three successive rotations in the order of rotation $ROT_X$ around $X$, rotation $ROT_Y$ around $Y$, and rotation $ROT_Z$ around $Z$, where $ROT_X$, $ROT_Y$, and $ROT_Z$ are three orientation angles describing the relative orientation of compressor body coordinate system to the fixed coordinate system. The three orientation angles are defined using the fixed coordinate system as zero references. With the three orientation angles, the transformation of coordinate systems for both rotation and translation can be expressed by:
Comparing Equations (10) and (11) to Equations (4) and (5), the three rotational degrees of freedom for FEA can be calculated by following equations:

\[
\begin{align*}
\cos(ROTX) &= c_x = \cos(ROTX), \\
\sin(ROTX) &= s_x = \sin(ROTX), \\
\cos(ROTY) &= c_y = \cos(ROTY), \\
\sin(ROTY) &= s_y = \sin(ROTY), \\
\cos(ROTZ) &= c_z = \cos(ROTZ), \\
\sin(ROTZ) &= s_z = \sin(ROTZ)
\end{align*}
\]  

Comparing Equations (10) and (11) to Equations (4) and (5), the three rotational degrees of freedom for FEA can be calculated by following equations:

\[
\begin{align*}
ROTY &= \sin^{-1} (-n_1) \\
ROTX &= \sin^{-1} \left( \frac{n_2}{\cos(ROTY)} \right) \\
ROTZ &= \sin^{-1} \left( \frac{m_3}{\cos(ROTY)} \right)
\end{align*}
\]  

The three translational degrees of freedom are the displacements of the origin of compressor body coordinate system \((X_{bo}, Y_{bo}, Z_{bo})\) in the fixed coordinate system.

4. SUMMARY

Compressor body displacement and internal discharge tube stresses measurements of an operating compressor were performed. The test setup has been presented in this paper. The tests were successful in obtaining the seven LVDT compressor body displacement data and discharge tube stress data. The displacement data were adequate to describe the 3-D rigid body motion of the compressor body.

The first modal frequency of the discharge tube was 83 Hz, determined by a frequency sweep test. Weibull analyses were used to handle the random variability of startup and shutdown discharge tube stresses. The 80th percentile maximum stress range from Weibull distribution analysis was used as the equivalent constant-amplitude fatigue stress range that caused the same fatigue damage as the actual variable-amplitude stress range. Accordingly, the LVDT data at the 80th percentile maximum stress range were used to determine the discharge tube displacement boundary conditions.

Finally, an optimization algorithm was successfully carried out to calculate the six degrees of freedom displacement boundary conditions of the discharge tube for transient FEA from seven LVDT compressor body displacement measurements, based on the model of 3-D rigid body motion of compressor body.
The modeled compressor body motions can be used for other dynamic analyses (i.e., compressor suspension system analysis). The calculated displacement boundary conditions of discharge tube can also be used for other similar reciprocating compressors after calibrated for displacement amplitudes by one single strain gauged compressor test (no LVDT’s needed).

5. ACKNOWLEDGEMENT

I am pleased to thank Gordon Lis for his valuable comments and contribution to the compressor tests for displacement measurements of compressor body motions. I would also like to thank the reciprocating compressor engineering for the support of this work.

6. REFERENCES


![Figure 1 Discharge tube used in compressor tests. The origin of the fixed coordinate system was defined at the center of discharge tube shell end.](image-url)
Figure 2  The upper half shell of a bolted shell compressor with two LVDT armature housings

Figure 3  Weibull distribution analyses of maximum stress ranges for startups and shutdowns
Figure 4  Compressor startup test data: displacement and stress data of a startup with the 80th percentile maximum stress range.

Figure 5  Compressor shutdown test data: displacement and stress data of a shutdown with the 80th percentile maximum stress range.

Figure 6  Calculated discharge tube FEA boundary conditions for compressor startups.
Figure 7  Calculated discharge tube FEA boundary conditions for compressor shutdowns.
FEA aided discharge tube design for hermetic reciprocating AC/R compressors
Part 2: Design and evaluation of discharge tube based on FEA

Jianxiong Chen
Technical Services, Emerson Climate Technologies, USA

ABSTRACT

The finite element analyses (FEA) for designing internal discharge tubes include both transient and modal analyses. The transient analyses are used to determine discharge tube stresses during compressor startups and shutdowns; and the modal analyses are used to determine discharge tube modal frequencies for compressor stable running conditions. The transient FEA boundary conditions were determined from seven LVDT (linear variable displacement transducer) measurements of compressor body motions. The FEA modeling approaches were verified against compressor test results. The evaluations of the discharge tube designs are based on the FEA generated startup and shutdown stresses, the modal frequencies, and the tube displacements.

1. INTRODUCTION

High discharge tube fatigue stresses usually occur during compressor startups and shutdowns. Under stable operating conditions, the discharge tube stresses are low as long as the discharge tube does not resonate with compressor running frequencies. Therefore, the evaluations of a discharge tube design require examining discharge tube stresses during startups and shutdowns and the modal frequencies. Besides, the discharge tube displacements need to be checked to make sure that there are no contacts between the discharge tube and the surrounding parts. As a result, to evaluate a discharge tube design, both transient and modal analyses are required. The transient analyses for startups and shutdowns are performed in the same way. Hence, only the transient analyses for compressor startups are presented in this paper.

To certain extent, simplifications and assumptions were included in the analyses. Therefore, verification of FEA results against the compressor test results before using FEA to design new discharge tubes becomes necessary. For verification purpose, the discharge tube used in the compressor tests was first modeled and analyzed by the finite element method with the calculated displacement boundary conditions from seven LVDT data; and the FEA results were verified against compressor test results.

© IMechE 2005
2. FEA MODEL AND VERIFICATION

The discharge tube used in the compressor tests was modeled and analyzed by the finite element method using the calculated six degrees of freedom displacement boundary conditions in Figure 1. The finite element software ANSYS was used for the analyses. The FEA modeling approach was verified before being used to design new discharge tubes. This section focuses on FEA models and the verification of FEA models using the measured discharge tube stresses and the first modal frequency.

Figure 2 shows the FEA model of the discharge tube used in compressor tests. The discharge tube solid model was meshed with shell elements (SHELL93) to save the computational time. For the same reason, the muffler was not included in the FEA model. Instead, the muffler was replaced by stiff beam elements. The mid surface across the discharge tube thickness was modeled and meshed with shell elements. To simulate the local stiffness of either shell or muffler, two circular plates, a shell plate and a muffler plate, were attached to the discharge tube shell and muffler end, respectively. Another advantage of using the circular plate at each discharge tube end was that the boundary conditions did not need to be directly applied to the discharge tube. All degrees of freedom of the shell plate edge were fixed. The muffler plate edge was connected to the center of muffler inlet using multiple stiff beam elements (BEAM4). In transient analyses, the time-dependent boundary conditions of six degrees of freedom in Figure 1 were applied to the center of muffler inlet. The boundary conditions in Figure 1 are relative to the fixed discharge tube shell end. In modal analyses, the displacements of the muffler inlet center were fixed. The damping in transient analyses was essentially not considered. In terms of the discharge tube displacement boundary conditions, the discharge tube stiffness changes due to different discharge tube designs are negligible compared to the total stiffness of compressor body supports (including suspension system stiffness and discharge tube stiffness). Careful considerations of mesh density and time step size are very important to obtain good FEA results while maintaining reasonable computational time. Having mesh density and time step size benchmarked, they should be used for all similar designs.

The FEA generated stresses were read at the same locations where the strain gages were placed. The comparisons between FEA generated stresses and the measured stresses are shown in Figure 3; the agreements between the FEA generated stresses and the measured stresses were very good. The FEA simulated first modal frequency of discharge tube was about 4% higher than the measurement (86.4 Hz versus 83 Hz). Solid elements (SOLID95) were also used to confirm the FEA results with shell elements, as listed in Table 1. The discrepancy between FEA results and the measurements mainly resulted from the exclusion of muffler stiffness from the modal analysis. However, the first modal frequency from FEA was still a very good indication of the first modal frequency of discharge tube. Selected FEA results of stresses and displacements are shown in Figure 4. The modal analysis results of the first and the second modal frequencies are shown in Figure 5.

Since the static analysis was much cheaper to perform than the transient analysis, static analysis was also performed to calculate the discharge tube stresses. As shown in Figure 6, stresses obtained with static analyses are less accurate because of the neglect of inertial forces, but still fairly close to the stresses from transient analyses in this case. However, extreme caution should be taken when deciding to use the stresses from static analyses because they are only acceptable when the mass of the discharge tube does not significantly affect the transient results.
3. EVALUATION OF NEW INTERNAL DISCHARGE TUBE DESIGNS

The reliability requirement for transient stress ranges is the fatigue life expectation at a given probability of failure. For the discharge tube to be acceptable, its transient stresses have to be below the allowable stress. Different discharge tube materials have different fatigue strengths and different allowable stresses.

The discharge tube performance at stable operating conditions is assessed according to the discharge tube modal frequencies. The design criterion for the modal frequencies is basically to avoid two major vibration excitations: motor running frequency and the piston running frequency. The piston running frequency is simply the motor running frequency times the total number of pistons. Note that the motor running frequency is slightly lower than the frequency of motor line power supply due to a motor slip. The first and second modal frequencies of the discharge tube should be away from both motor and piston running frequencies.

As mentioned above, the discharge tube was designed after the surrounding parts were designed. Therefore, the discharge tube transient displacements are also examined to ensure there are no contacts between the discharge tube and the any other surrounding parts. Four examples of new discharge tube designs are shown in Figure 7.

The final compressor tests with a strain gauged discharge tube are still recommended to examine the discharge tube resonant frequency, startup and shutdown stresses, and stable running stresses.

4. SUMMARY

A new design approach, FEA aided discharge tube design, has been presented in this paper to design new internal discharge tubes. The new design process uses FEA to optimize discharge tube design based on the discharge tube startup and shutdown stresses and the modal frequencies. The FEA generated stresses and the first modal frequency were in good agreements with the measurements, validating the FEA modeling approaches for discharge tube design. Compared to the test-only design method, the new approach has advantages in cutting down developing cost and in reaching a better design. The whole design process can be summarized into the following steps:

- Set up the compressor with a strain gauged prototype discharge tube and at least six LVDT’s to measure discharge tube stresses and compressor body displacements during startups and shutdowns. A frequency sweep test is required to determine the first modal frequency of discharge tube.
- Determine the FEA displacement boundary conditions from compressor motion measurements based on a 3-D rigid body motion modal.
- Verify the FEA model by comparing the FEA generated discharge tube stresses and the first modal frequency against the measurements.
- Having the FEA model verified, use FEA to design new discharge tubes. Evaluate the new designs according to the FEA generated discharge tube startup and shutdown stresses, displacement, and modal frequencies. Pick out the best from these designs that satisfy the design criterion for the final strain gauged compressor test.
5. REFERENCES

(3) D. Kececioglu, Reliability and Life Testing Handbook
(4) D. Crowe & A. Feinberg, Design for Reliability, CRC Press
(5) ANSYS, INC., ANSYS Release 8.1 Documentation.

Figure 1 Calculated discharge tube boundary conditions of compressor startups from compressor body motions

Figure 2 FEA model of the discharge tube used in the compressor tests
Figure 3 Comparisons between FEA generated and measured stresses for the discharge tube used in the compressor tests

Table 1 Modal frequency comparison between FEA and the measurement for the discharge tube used in the compressor tests

<table>
<thead>
<tr>
<th></th>
<th>First mode frequency (HZ)</th>
<th>Second mode frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>FEA (SHELL93)</td>
<td>86.4</td>
<td>123.1</td>
</tr>
<tr>
<td>FEA (SOLID95)</td>
<td>86.9</td>
<td>123.5</td>
</tr>
<tr>
<td>Measurement</td>
<td>83</td>
<td></td>
</tr>
</tbody>
</table>
Figure 4 Selected FEA results of stress and displacements for the discharge tube used in compressor tests

Figure 5 FEA results of discharge tube modal frequencies (top picture - undeformed, left picture - deformed shape of the first modal frequency, right picture - deformed shape of the second modal frequency
Figure 6: Comparison between the stresses generated from transient and static FEA for the discharge tube used in the compressor tests.

Figure 7: Four examples of the new discharge tube designs investigated.
SCROLL COMPRESSORS
Novel vapor injection method for scroll compressors

Alexander Lifson
Carrier Corp., Division of United Technologies, Carlyle Compressor Division, USA

ABSTRACT

A vapor injection into an intermediate compression pocket coupled with the use of an economizer heat exchanger or flash tank can be a cost effective solution for improving efficiency and boosting capacity of refrigeration units. However, the use of vapor injection for air conditioning applications is still not widespread. This is due to a low pressure ratio operation, typical of air conditioning applications, which causes limited efficiency gains when vapor injection cycle is utilized. A novel injection technique alleviates this problem by improving the cycle efficiency and minimizing the throttling and pumping losses of a conventional vapor injection port design. This is accomplished by machining the injection ports through the tips of a fixed scroll and providing corresponding indentations on the floor of an orbiting scroll instead of using the old conventional technique where the ports are machined through the floor of the fixed scroll. In the new design, we can precisely control the opening and closing of the injection ports and shorten the duration of the port opening as compared to the conventional design. Results show that the new injection technique is roughly five percent more efficient and delivers six percent more capacity than the conventional injection scheme. Also presented is a simplified analysis to help improve the understanding of how the interaction between the internal compressor injection port design and system operation affects the economized cycle performance.

NOMENCLATURE

\( m \) – mass flow rate
\( h \) – enthalpy
\( P_s \) – compressor suction pressure
\( P_d \) – compressor discharge pressure
\( P_{ec} \) – pressure in injection line
\( P_{c}(t) \) – instantaneous pressure in compression pocket
\( dP_{c}(t)/dt \) – rate of pressure change during compression
\( A_{ec}(t) \) – time varying area of injection ports
\( V_s \) – suction volume
\( V_c(t) \) – time varying volume of compression pocket
\( dV_c(t)/dt \) – rate of change of compression volume

© IMechE 2005
\( n \) - polytrophic exponent for compression process
\( u_{ec}(t) \) – instantaneous refrigerant velocity through injection port
\( \alpha \) – port and connecting line flow resistance coefficient
\( \rho \) – refrigerant density

1 BACKGROUND

As shown in References (1) and (2) the effectiveness of the vapor injection (economized cycle) for boosting capacity and efficiency is more pronounced for high pressure ratio applications\(^1\). As a result, the economized cycles were mainly applied in refrigeration systems. However, recent developments have extended the potential use of the economized cycle to lower pressure ratio application typical of chillers and A/C units using scroll compressors (see Reference (2) and (3)). Some of these new developments include:

- Increased proliferation of zero ozone layer depletion refrigerants such as R410A and R407C, that may have inferior cycle efficiency than R-22 refrigerant; thus augmenting the efficiency deficit of these newer refrigerants economized cycles started to be considered for the low pressure ratio applications.
- Recognition by the industry and end user that the unit capacity boost coupled with efficient unit operation is mostly needed at high ambient temperature, where the economized cycle is most effective and where both R410A and R407C under-perform as compared to R22 refrigerant (see Reference (2)).
- Introduction of high displacement scroll compressors – making it more cost effective to introduce the economized cycle into the scroll compressor unit designs.
- Positive industry experience with vapor injected compressors in refrigeration applications.
- Use of injection ports for a by-pass (unloading) operation (see Reference (4)) thus providing an additional incentive for incorporating vapor injection into the compression cycle.
- More emphasis on enhancing cycle efficiency to alleviate the effects of global warming.
- Recent rise in energy and electricity cost accelerating the need for efficiency improvement.

Even though the above mentioned factors make the use of the economized cycle in contemporary air conditioning applications more beneficial than in the past. The efficiency gain offered by the vapor injection for relatively low pressure ratio applications, may still not be sufficiently high to justify the additional expense of the economizer circuit. A new vapor injection technique, described below, offers additional enhancement by improving the efficiency and capacity of the cycle an additional five to six percent\(^2\) over the conventional injection scheme, thus offering additional incentive in applying the economized cycle.

\(^1\) A typical commercial refrigeration system has a much colder evaporator coil and thus operates at a lower suction pressure than an air-conditioning system. The discharge pressure though would not differ substantially between these two systems, therefore the refrigeration system runs at higher pressure ratio (ratio of discharge pressure to suction pressure) than the air conditioning system.

\(^2\) Both the efficiency and capacity gain increase with the rise in pressure ratio, thus the benefits are higher for operation at high ambient temperature and for lower efficiency units with undersized evaporator and condenser coils. Conversely, the expected benefits are less at lower ambients and for higher efficiency units.
2 DESIGN OF NEW ECONOMIZER INJECTION SCHEME

The vapor injection into a scroll compressor occurs through an injection line that passes the refrigerant from an economizer heat exchanger or flash tank, into the intermediate injection point within the scroll compressor. Inside the compressor, the injection flow splits into two streams. The flow from each stream is injected into a separate compression pocket through an independent injection port -- one for each pocket. Two of the major obstacles in achieving an efficient vapor injection are pumping (sloshing) and throttling losses. The sloshing of the refrigerant in and out of the compression pocket leads to unrecoverable flow losses. In the prior scroll compressor designs, the injection port opening was located on the fixed scroll floor. In this case, the injection port would be exposed to almost a full range of pressure variation within the intermediate scroll compression pocket. Thus, at the beginning of the cycle when the pressure in the scroll compression pocket is low, the injected refrigerant would fill the compression pockets. However, toward the end of the compression cycle within this pocket, the refrigerant would be driven back into the injection line – this results in high sloshing losses as refrigerant moves in and out of the compression pocket. The throttling caused by pressure drop in the injection line and through the injection ports would also contribute to the loss mechanism.

The novel patented vapor injection method addresses both of these problems by precisely timing the injection, and more importantly, drastically reducing the amount of time the injection ports are open. This minimizes both refrigerant sloshing (no reverse flow out of the compression pockets) and throttling losses (ports of optimum size are much larger and open for shorter period than in the old design). Reference (5) describes this method in details.

Figure 1 shows the photographs of the fixed and orbiting scroll for the new injection scheme design. In this new design, the economizer injection ports are machined through the wrap tips of the fixed scroll. Corresponding indentations are also machined on the floor of the orbiting scroll. In this arrangement, the vapor injection takes place during a limited period only when the corresponding indentation on the floor of the orbiting scroll uncovers the injection port. However, once the orbiting scroll has moved away, the facing orbiting scroll base plate (floor) closes off the ports. In this way, a scroll compressor designer is able to easily control the "on/off" time for the refrigerant injection into the intermediate compression chamber. The new injection scheme is especially effective for the so called ‘hybrid wraps’ whose thickness allows large injection ports to be formed through the wraps. Figure 2 shows a photograph of the fixed and orbiting scroll for the old injection scheme design, where the injection takes place through the injection ports machined in the floor of the fixed scroll (as explained earlier for this design there are no indentations on the floor of the fixed scroll).

3 ANALYSIS

We analyzed and tested three scroll compressor configurations:

- Non-economized operation, where the economizer injection was turned off
- Old economized scheme with injection through the floor of the fixed scroll
- New economized scheme with injection through the fixed scroll wrap.
Figure 1: Fixed and orbiting scroll with new injection scheme

Figure 2: Fixed and orbiting scroll with old injection scheme

Figure 3 shows a typical economized (vapor injection) compression cycle and Figure 4 shows a corresponding P-h diagram. The refrigerant entering from the evaporator is compressed in the scroll compressor from point 5 to point 7”, additional refrigerant is added from the economizer injection line entering the compressor at intermediate compression point 7’’’, where it mixes with already partially compressed refrigerant of point 7’”; the refrigerant of these two combined streams is then further compressed from point 7 to point 1.
The refrigerant leaves the compressor and rejects heat in the condenser from point 1 to point 2. At point 2, the refrigerant splits into a main flow path and secondary flow path. The main flow is additionally sub-cooled from point 2 to point 3 by a secondary stream. This secondary flow steam, after expanding to lower temperature and pressure, accepts heat from main flow from point 6 to point 7'. The main flow passes through an expansion device and enters the evaporator at point 4. If it were not for the economized cycle, the refrigerant would have entered the evaporator at higher enthalpy (corresponding to the enthalpy of point 2). The enthalpy difference $\delta h_{sc-2-4}$ represents amount of refrigerant subcooling that is responsible for additional capacity gain $m_{evap} \cdot \delta h_{sc-2-4}$ of the economized cycle. As both the primary and secondary stream pass through the economizer heat exchanger, under ideal conditions of no heat loss to ambient, the amount of heat added to the secondary stream (vapor injected flow) must be equal to the amount of heat removed from the primary stream (evaporator flow):

$$m_{evap} \cdot \delta h_{sc-2-3} = m_{ec\_average} \cdot \delta h_{ec\_7'-6} \quad (1)$$

The ratio $\delta h_{sc-2-3}/\delta h_{5-6}$ is the relative increase in the cycle capacity over the non-economized cycle, where $\delta h_{5-6}$ is the enthalpy rise through the evaporator if the vapor injection were turned-off (non-economized mode of operation). The purpose of the economizer cycle is two-fold, first to increase the system capacity, and more importantly, to increase the system efficiency. Figure 4 shows that the system operating at lower $P_{ec}$ (pressure along the line $6'\_6\_7$) will have higher capacity because of higher value of subcooling $\delta h_{sc-2-3}$. Locating the economizer injection ports immediately after the start of compression always results in lower $P_{ec}$ and thus higher cycle capacity (see Reference (6)). However, if the goal is to optimize the

---

3 We assume same suction pressure and temperature, and discharge pressure for all cases under consideration, ignoring the “real” system effects where re-balancing takes place: condenser pressure increases slightly and evaporator pressure decrease slightly due to additional refrigerant injection as the coils become more loaded.

4 The lower value of economizer pressure $P_{ec}$ results in the lower enthalpy of the intersection point 6', and thus larger value for the enthalpy difference $\delta h_{sc\_2-3}$. 

---

385
cycle efficiency, depending on the unit operating conditions, the injection ports may need to be located farther into the compression process. The overall cycle efficiency is increased, if the increase in the system capacity were higher than the increase in compressor power due to additional work required to compress the injected refrigerant from point 7 to point 1. An additional benefit in using a compressor with vapor injection is ability to use the same injection ports for unloading operation (see Reference (4)). In this case, we can improve the cycle efficiency by engaging an optional by-pass unloader valve that provides the communication between the injection and suction line. The valve opens when the cooling demand is low, this avoids inefficient system operation by minimizing the unit cycling. This mode is especially effective for the new design with early vapor injection cut-off.\(^5\)

Neglecting the compressibility effects\(^6\) for the flow entering or exiting the injection port, the following equation describes pressure change in the compression pocket:

\[
dP_c(t)/dt = [A_{ec}(t)\cdot u_{ec}(t) + dV_c(t)/dt] \cdot n \cdot P_c(t)/V_c(t) \tag{2}
\]

In equation 2, the first term in the bracket, represents the volumetric flow rate in the injection line. If the \(u_{ec}(t)\) term is positive then the refrigerant is injected into the compression pocket and if it is negative then the refrigerant leaves the pocket. The second term in the bracket represents the rate of compression volume change that is defined by the scroll compressor geometry and operating speed. This term is always positive as compression volume of the pocket is always reduced during the compression process. In this equation, the polytrophic exponent \(n\) is assumed constant throughout the compression cycle\(^7\).

The pressure drop\(^8\) through the injection port is defined as:

\[
|P_{ec}(t) - P_c(t)| = \alpha_{ec}\cdot \rho_{ec}(t) \cdot u_{ec}(t)^2/2 \tag{3}
\]

If the volume of the economizer (injection) line\(^9\) is several times of the scroll compression volume, then we can assume that the economizer line pressure \(P_{ec}(t)\) and density \(\rho_{ec}(t)\) in the above equation are constant values. For incompressible flow, for example, the flow resistance coefficient \(\alpha_{ec}\) through an orifice-like opening is equal to 2.7. If the injection line has additional flow losses (line friction or other restrictions), the total flow resistance coefficient \(\alpha_{ec}\) can be calculated as shown in References (1) and (7).

We find the average mass flow \(m_{ec\_average}\) over time \(T\) through the economizer injection line by integrating the instantaneous value of the mass flow \(m_{ec}(t)\):

---

\(^5\) Large ports mean lower throttling losses and the location of the port early in the compression cycle minimizes the amount of work required to compress the refrigerant before it is by-passed back to suction.

\(^6\) For the case of the flow through economizer injection under normal circumstances, we can assume the flow to be incompressible since the pressure ratio between the flow in the economizer line and the corresponding pressure in the intermediate compression pocket would not exceed the value of 1.5. Reference 1 and Reference 7 provide a detailed description of how compressibility effects are taken into account for higher-pressure ratio.

\(^7\) More precise analysis accounts for a slight change in the polytrophic exponent due to real gas effects and non-uniform addition or removal of heat throughout the compression cycle. Normally these effects would be of a second order, especially in view of evaluation of relative differences between the models.

\(^8\) It should be noted that in the equation above the pressure difference is assigned an “absolute” value to account for a sign change if the flow direction is reversed if the refrigerant to leave the compression pocket.

\(^9\) This volume includes connecting lines as well as the free volume of economizer heat exchanger or flash tank.
\[ \dot{m}_{ec\_average} = \frac{\int_0^T (\rho_{ec} \cdot \dot{u}_{ec}(t) \cdot A_{ec}(t)) dt}{T} \]  

(4)

4 SIMULATION RESULTS

In this section, we will compare the system performance under three configurations.

- Non-economized operation, where the vapor injection ports are blocked off.
- Economized mode of operation with vapor injection through the floor of the fixed scroll.
- New economizer injection scheme with vapor injection through the fixed scroll wrap tips.

We will optimize the two vapor injection schemes, each with its own design constraints, for most optimum injection port size and location, to deliver the best possible cycle efficiency. For all three configurations we will use the same basic compressor geometry, and for the non-economized case we assume that there are no injection ports. We assume idealized system using the same conditions for suction and discharge pressure for all three cases.\(^{10}\) For both vapor injection schemes we will allow injection line pressure and refrigerant mass flow to “float” to satisfy the heat balance equation 1 above.

Considering the above assumptions and solving a set of equations 1 through 4, and substituting the varying port opening area \(A_{ec}(t)\) of Figure 5 (representative of a \(10^{-4}\) m\(^3\) displacement compressor) into equation 2, we determine the injection line pressure \(P_{ec}\), injection mass flow rate \(\dot{m}(t)\), and \(PV\) compressor diagram as shown in Figures 6 and 7 respectively\(^{11}\). Using a well known equation we then calculate the cycle efficiency:

\[ \text{COP} = \frac{\dot{m}_{evap} \cdot \delta_{evap\_5\_4}}{W_{compr}} \]  

(5)

Where \(W_{compr}\) is computed from a known compressor speed and area enclosed within the \(PV\) diagram. Figure 5 and 6 show that the new injection technique shortens the injection time moving the economized cycle to lower vapor injection pressure. In the old design, the injection flow reverses, while in the new design the flow is always into the compression pocket. The \(PV\) diagram, as expected, shows that there is a noticeable increase (about 10\%) in the consumed power during the economized vapor injection operations either for the old or new design as compared to the non-economized mode. However, the power consumption for both injection schemes is almost identical to each other\(^{12}\), while at the same time there is a 6\% beneficial capacity increase for the new injection technique over the old one. Table 1 summarizes the performance difference between these three designs, using the non-economized cycle as a basis.

---

\(^{10}\) For the system under consideration we chose a typical R410A unit operating at pressure ratio equal to 3.9 where suction pressure and discharge pressure are respectively equal to 3.7 bars and 14.5 bars for all three configurations. In other words we neglect the system re-balancing effects caused by vapor injection that would affect refrigerant pressure at the condenser and evaporator. These effects are considered in details in Reference 2.

\(^{11}\) We would also use a technique developed in reference 1 for calculating flow through the compressor discharge port.

\(^{12}\) As seen from the \(PV\) diagram and Table 1, there is only 1\% difference in the power consumption between the new and old vapor injection scheme, this slight increase is caused by additional amount of injected refrigerant when the new injection technique is applied.
Figure 5: Port area standard and novel port

Figure 6: Mass flow through

Figure 7: PV diagram for standard, no injection, new and old injection scheme
**5 CONCLUDING REMARKS**

- Technique for calculating power and capacity of the economized cycle in relation to injection port size, location, and configuration is presented.
- In the new design, the injection ports are machined through the fixed scroll wrap. These ports selectively open/closed by indentations on the floor of the orbiting scroll.
- The novel vapor injection technique limits the exposure time of the vapor injection ports to the compression cycle, which minimizes the throttling and pumping losses.
- Injection line pressure is reduced and mass flow rate is increased in the new injection scheme; driving the improved performance of the new injection scheme over the old.
- The new vapor injection improves the efficiency over the conventional vapor injection scheme by roughly 5% and over the cycle with no vapor injection by 12%, while the capacity is respectively increased by 6% and 23%. We analyzed idealized system, ignoring the system re-balancing effects on changes in condenser and evaporator pressure due to vapor injection. We considered the system under high ambient temperature, where the capacity and efficiency increase are needed the most.
- The application of the new injection technique is especially beneficial for systems with increasingly popular R410A and R407C refrigerants that have a diminished performance at high ambient temperature as compared to R22 refrigerant.
- Capacity unloading scheme, accomplished by turning off the refrigerant injection and engaging the by-pass valve, is more effective for the new injection scheme.
6 REFERENCES

A system for the documentation of feature variation and its effect on scroll compressor design

J. Sauls
Trane, USA

ABSTRACT

This paper presents a system used for documentation of scroll compressor feature variation and subsequent use of the information in the design of a compressor for air-conditioning applications. The system consists of a database for storage of data, a “scorecard” to collect data and document results of analyses and a set of assembly, performance and reliability analysis tools. The approach described is an implementation of the concepts of Critical Parameter Management (CPM). The CPM concept is introduced and elements of the system are briefly reviewed in the paper. An example of how the system is used to study the effects of manufacturing capability on design choices concludes the report.

1. INTRODUCTION

Designers of compressors for air-conditioning systems are required to define appropriate configurations in a complex environment of interactions at several levels. Even what might seem to be the relatively simple task of selecting the proper overall configuration and sizing details to satisfy the rating condition requirements for capacity and efficiency is complicated by the variety of conditions that are imposed on the compressor by the systems in which they are used [1]. This is especially true of general purpose compressors such as commercial scrolls and screws which are applied in a variety of systems such as rooftop units, air-cooled water chillers and water-cooled water chillers.

In addition, the designer must design for robustness. This means that the compressor must be capable of providing its required functions in the face of stresses from the two environments in which it lives – the manufacturing environment and the application environment. Application stresses are measured by variation in imposed operating conditions (e.g. inlet and discharge pressures) from the nominal values for which the compressor configuration and sizing details were chosen. The manufacturing environment imposes stresses in the form of variation in dimensions of important features which in turn create variation in the key physical characteristics leading, finally, to variation in the functional outputs of the compressor.
This paper addresses the manufacturing environment issue. For all of the complexity and interactions found in a modern compressor, the underlying fundamental principle of design is that:

...a designer can only manipulate the functional characteristics of the compressor through specification, in terms of linear dimensions, angular dimensions, and/or material properties of the individual features on the separate parts that comprise the design.

The important functional characteristics of scroll compressor capacity and efficiency are affected by numerous parameters such as physical displacement, leakage path length and clearances. In most cases, these parameters exist as a result of the assembly of several parts. The parts in turn are defined by collections of individual features whose actual dimensions will vary to an extent determined by the capabilities of the processes used to manufacture them. Critical parameter management (CPM) is the process of understanding, accommodating and controlling features – mean and variation – in order to control key subsystem parameters and ultimately to control the user-sensible outputs of the compressor.

Section 2 of this report provides a brief overview of the principles of CPM as used in the design of a scroll compressor for application in air-conditioning systems. The generation, storage and flow of information in the scroll design process are discussed in Section 3. Here, we review the types of data, the use of databases and the various design and analysis tools employed to use the data to create a robust design meeting the basic functional requirements. An example of the principles applied to the particular question of scroll involute meshing is given in Section 4 with summary remarks provided in Section 5. Reference citations are noted in the text by numbers in brackets []. The references and all figures are collected in sections at the end of the report.

2. CRITICAL PARAMETER MANAGEMENT

Creveling [2] defines critical parameter management (CPM) as follows:

“CPM is the disciplined and focused attention to the design’s functions, parameters and responses that are critical to fulfilling the customer’s needs.”

This is, of course, another statement of what is the approach to every design: select and manage the details in order to produce the required product outputs sensible by the user.

CPM is one method of defining a disciplined approach to achieving this end. In the design phase – after customer requirements have been converted into appropriate product functional characteristics – the emphasis is on feature level data and the flow of this information into parameter characteristics and ultimately to the primary metric in CPM, the product’s “functional responses.” Metrics chosen to watch during the design are part of the CPM discipline. “A dominant rule of CPM is to refrain from measuring…forms of quality that are distant from the function of the design...” [2].

Another important requirement for execution of CPM is the need for accurate “transfer functions.” Transfer functions convert data from lower levels, starting with individual features, to the higher levels such as assembled static and functional characteristics of subsystems and,
finally, to the compressor functional outputs. The transfer functions can be analytical or empirical. In the design phase, many of these functions will be design models and rules.

We can summarize the discussion so far by identifying three tenets for CPM’s role in the compressor design phase:

1. Metrics of customer sensible functional characteristics – mean and variation – are the most important measures of design quality and thus are the drivers in all design decisions. This connects the design to the user’s requirements.

2. Customer sensible functional characteristics are related to individual feature characteristics, defined by their mean and variation values. Thus, we need data or accurate estimates for the feature dimensions. This connects the design and manufacturing processes.

3. Connecting the features to product function requires transfer functions for accurate representation of the conversion of feature dimensions to product operating characteristics.

A process built around these tenets is illustrated in Figure 1 (Figures are collected in one section at the end of this report). We show some selected feature dimensions as underlined entries. Moving to the left, features are associated with a part if the flow is to an italicized item or to a parameter. Parameters become more complex as we move to the left. For example, Lower bearing clearance is the simple difference of the bearing and journal diameter dimensions. This parameter, along with Upper bearing clearance and the Upper housing and Lower housing features (only four of which are shown), determines the relative location of the orbiting and fixed scrolls – the OS/FS relative location parameter. In this case, the assembly of the individual features and parameters is quite complex and is carried out in the first of the three major transfer functions shown in the figure – the Alignment Model.

As we move farther to the left, we assemble more features and previously computed parameters through the second transfer function, the Mesh Model, to arrive at functional characteristics of the fixed/orbiting scroll interaction: contact force and/or flank clearance. These and many other factors then flow through the final transfer function, the Thermodynamic Model, which computes functional characteristics for the various compressor subsystems (Motor, Bearings, etc.) and assembles these into overall compressor functional characteristics – in this illustration the isentropic efficiency.

One very important impact that this approach has on execution of the design process is that virtually all calculations are carried out using Monte Carlo analyses. This is because feature variation is an important input and control of variation in the compressor’s functional characteristics is a key design goal. For the example shown in Figure 1, the Alignment Model and Mesh Model are used in a Monte Carlo analysis to determine a distribution of flank mesh characteristics, the combination of clearance and contact force for each of 10,000 trial assemblies. This distribution is then input into the Thermodynamic Model along with distribution data for other factors to create the distribution of the compressor functional characteristics. An illustration of the data flow follows in Section 3 and an example of the process applied to scroll compressor design is presented in Section 4.

393
3. INFORMATION FLOW

The task at hand is execution of analyses for optimizing a design to meet functional requirements throughout the range of noise that comes from manufacturing variation in feature dimensions. This means that we need to move all the way from features defined by linear or angular dimensions and/or material properties through the design process, adding information and translating until we arrive at functional characteristics. Optimization is carried out to achieve a design that meets target functional requirements with the simplest, lowest cost configuration.

The design is then built and qualified in a test laboratory. During this phase, the compressor is qualified against its critical application requirements. Results of the design analyses and these tests become the basis for specifications delivered to users to guide them in proper application of the compressor in their systems.

Analyses also reveal the most important features in the design. These features are those to which the functional characteristics of the compressor are most sensitive and which will generally have the most stringent manufacturing control plans.

A simplified picture of the sources and flow of information and the plans and tools that are used to execute this process is shown in Figure 2. Four key elements in the process are noted with the “cloud” boxes in the Figure. These are: Design Optimization, Qualification, Consistent Manufacturing (of the qualified design) and Dissemination of Application data.

At the center of the process are the databases that collect and distribute the various data required. The important factor in this structure is that the same data is used in executing all elements of the process. Data flows in and out of the databases and these databases grow and change during the course of the project.

The design optimization process, illustrated with a single “work process” box in the figure is actually a complex process using all of the analysis tools available to the team. Here is where details of the flow from feature to function, as shown in Figure 1, are executed. During this phase, the relationship between features and function are defined and the most critical features identified. This process is shown as “Feature Assessment” in Figure 2. As shown in the figure, there is a connection from the qualification tests via the “Project Design and Technical Reviews” process to “Feature Assessment” as actual running experience adds to the assessments made during the analytical phase of the design process. The feature assessment leads to decisions on manufacturing processes and to “Control Plan Development”.

The qualification of the design is carried out according to the “Qualification Plan” and test results flow into the “Test Data” database. As information is collected and processed, the functional characterization and application documentation are generated, illustrated in the single work process labeled “Generate Application Data” in Figure 2.

A “Scorecard” is shown in the center and near the top of the chart in Figure 2. This is an important element in the process. The scorecard can accumulate data and report on the current state of the relationship between the features with their manufacturing variation characteristics and the resultant compressor functional mean and variation values.
4. APPLICATION TO SCROLL COMPRESSOR DESIGN

This section presents a summary of the results of a design study documented in more detail in [3]. The problem is one of setting an appropriate throw in the face of manufacturing variation of features that affect the involute flank mesh functional characteristics: flank clearance and flank contact force. Throw is defined as the radial offset of the orbiting scroll relative to the fixed scroll, as illustrated in Figure 3. The theoretical perfect throw is the orbit radius computed from the specified involute shape and thickness. This value for the throw positions the involutes such that there is contact without contact force. This contact point (or line) spirals inwardly, with one point on each side, as the crank rotates. This situation represents the highest efficiency condition for flank mesh characteristics.

The design goal for involute meshing is to maximize performance while meeting reliability criteria. In our example, we simplify the reliability criteria to setting a limit on the average contact force. Due to the complexity of the assembly, analyses show that clearance and contact force both vary during the course of one rotation of the crankshaft; the average values for clearance and contact force used in this example are the averages over one revolution for each assembly analyzed.

If we start with the throw set at a relatively small value, the involutes are separated relative to the theoretical ideal and we have clearance between the flanks. Performance is improved when the throw is increased and the flank clearance is reduced, resulting from lower levels of internal leakage. However, as average clearance is lowered, we begin to see periods of operation in which there is flank contact. The resulting frictional losses begin to moderate the efficiency gained from the leakage effect. Eventually, we reach a situation where increases in throw have little effect on average clearance, but result in large increases in force. Efficiency begins to fall off from its peak value. Finally, as we push to even higher values for the throw, the average contact force rises to a level exceeding the reliability limit for some assemblies.

In our example we explore the effect that manufacturing process variation has on our choice of the nominal throw dimension. This analytical study uses the assembly, mesh and thermodynamic models introduced in Section 2. The assembly models accept data describing the feature dimensions and variations from the nominal dimensions produced by the manufacturing processes. Data from manufacturing operations relative to a particular feature will define the expected average value of the feature dimensions produced over a period of time and the variation in these dimensions, the process mean ($\mu$) and standard deviation ($\sigma$), respectively.

Variation characteristics of feature dimensions can be specified in terms of a manufacturing process capability index, Cpk. This index is a ratio of the location of the mean value within the tolerance range specified for the feature to a reference range based on variation generated by the manufacturing process that produces the feature. The index is computed as follows:

$$Cpk = \min \left( \frac{USL - \mu}{3 \cdot \sigma}, \frac{\mu - LSL}{3 \cdot \sigma} \right)$$

where USL and LSL are the upper and lower specification limits, respectively.

The manufacturing process reference range is defined as three times the process variation. Tolerance ranges that are large compared to process variation have high values for Cpk.
Conversely, process variations that are large compared to the specification limits are represented by low values of Cpk. A feature whose tolerances were equal to $3\sigma$ range would have a $C_{pk} = 1$ and would be made with actual dimensions between the USL and LSL 99.87% of the time.

In reality, the assessment of process capability and use of capability data in the analyses is a complex proposition. The data may represent short term studies of the processes, requiring judgment as to the applicability of the data to long term production. While the capability index is based on a normal distribution of feature dimensions, actual distributions can be quite different.

For our case study, we will assume that the mean value for all features is equal to the nominal value defined on the drawings and that the actual process variation is described by a normal distribution. In addition, we will set process capabilities indices for all features to the same value. In this study, we will carry out analyses for process capability levels of 2.0, 1.0 and 0.6.

The first requirement for our design is that we meet the reliability requirements. We will assume that our budget for the involute mesh is three in 10,000 assembles (0.03%) having an average flank contact force greater than our criteria. The problem is one of finding a throw that maximizes performance while meeting this target. To do this requires a series of Monte Carlo analyses. A nominal throw is assumed and 1,000 trial assemblies are computed. Using the computed distribution for the contact force, we calculate the defect rate relative to our criteria. If the rate is below the target, we increase the throw for the next trial; if the rate is too high, we reduce the throw. As we accumulate cases, we determine the average efficiency and contact force defect rate as functions of the nominal throw. From this we can determine the throw that provides the maximum efficiency at our target reliability level.

An example of this process is shown in Figure 4. Results here represent the case where all process capability indices are equal to 1.0. Results for three levels of throw – small, medium and large – are shown in Figures 4a, 4b and 4c, respectively. The dashed line in the figures shows the location of the contact force reliability criteria. Assemblies whose relative efficiency vs. clearance data points fall below this line fail the criteria test. When a nominal throw is proposed that results in a population having more than 0.03% of its assemblies below this line, it fails the reliability and test and the throw cannot be accepted as the target nominal.

These charts illustrate some of the interesting characteristics of the complex scroll assembly leading to the involute meshing. At low levels of throw (Figure 4a) the manufacturing variation produces average mesh clearance values ranging from $35\mu m$ to $75\mu m$. Performance variation is along a single straight line which defines the effect of leakage through the flank mesh clearance.

As throw is increased (Figure 4b), we see deviation of the performance from this line. This is a result of introducing flank contact in some assemblies. The resulting frictional power is an additional loss. The leakage-only effect is visible – points along the line identified in Figure 4a – as there are still a large number of assemblies with no contact between the flanks.

For the largest throw (Figure 4c) we see more assemblies with contact. By virtue of the lower values of efficiency in the extreme cases, we can also see that we are generating higher levels of contact in some assemblies.
In both cases 4b and 4c, there are some assemblies with contact force exceeding the reliability criteria – those cases in the figures that fall below the dashed line. The process illustrated in Figure 4 continues until a throw is found that maximizes efficiency and results in the design meeting the reliability criteria. Results of this entire process for the case where Cpk = 1 are illustrated in Figure 5. Here, the average performance of all assemblies is plotted against the mean value of the average clearance for the population computed. The solid circle symbols show the average values of cases a, b and c (reading right to left along the curve) whose details are given in Figure 4. The open diamond symbol in Figure 5 shows where the analysis predicts we can achieve the highest performance while meeting the reliability criteria.

This entire process is repeated at levels of Cpk=0.6 and Cpk=2. Results are shown in Figure 6. Here we can see the effect that manufacturing capability has on the design choice and resultant operating characteristics. At low levels of capability, variation in all of the features is large and so we must move the nominal to large average clearance in order to meet the reliability criteria. As we improve capability, we can push the nominal throw to values resulting in smaller average clearance. At Cpk=2, we see we have pushed the average performance up to levels approaching the theoretical best case (relative efficiency =1).

5. SUMMARY

The basic tenet of Critical Parameter Management is the connection of functional requirements of the complete compressor system to characteristics of the manufacturing processes used to make the individual features. Here we have reviewed the concepts of CPM as applied to a refrigeration scroll compressor and introduced a roadmap of information flow that supports the practical application of these concepts. Results are shown with an example of selecting the critical feature of involute throw, showing how the process looks during the design phase as we optimize complex subassemblies for multiple targets.

There are two environments that impose extreme stress on the compressor – the application environment (e.g. a reversible heat pump application) and the manufacturing environment (feature dimensions vary from their target values). The focus here has been on dealing with the stress of manufacturing variation during the selection of the target nominal dimensions. The process requires selection of the appropriate functional characteristics to design for, availability of manufacturing process variability data or estimates and capable, verified design analysis tools.

The example illustrates the powerful effect that the manufacturing environment stress has on the design. This fact speaks to the need to include consideration of manufacturing variation in the critical design decisions. Making the right choices requires that the designer have access to the best data or estimates available so that the effects can be included during the design definition phase.

ACKNOWLEDGEMENTS

I would like to thank Trane for the opportunity to present this study. I must also acknowledge Mr. Mike Benco and Mr. Rod Lakowske for their groundbreaking work in the development of the complex assembly analyses that are used to generate the mesh information input into the thermodynamic simulation. Their contribution is the foundation for the results presented here.
REFERENCES


© American Standard 2005
FIGURES

Figure 1
Chain from Features to Parameters to Sensible Functional Response

Figure 2
Information Flow
Figure 3
Scroll Compressor (a) with Detail of Crankshaft (b) Definition of “Throw”

Figure 4
Monte Carlo Results for Cpk = 1.0, Varying Throw
Figure 5
Averages for Cpk = 1.0, Varying Throw

Figure 6
Effect of Manufacturing Capability
Gas leakage in CO$_2$ and R22 scroll compressors and its use in simulations of optimal performance

Tatsuya Oku$^{(1)}$, Keiko Anami$^{(2)}$, Noriaki Ishii$^{(3)}$, Charles W. Knisely$^{(4)}$, Kenji Yasuda$^{(1)}$, Kiyoshi Sawai$^{(5)}$, Kiyoshi Sano$^{(5)}$, Takashi Morimoto$^{(5)}$

$^{(1)}$ Graduate School of Engineering, Osaka Electro-Communication University, Japan
$^{(2)}$ Dept. of Mechanical Engineering, Ashikaga Institute of Technology, Japan
$^{(3)}$ Dept. of Mechanical Engineering, Osaka Electro-Communication University, Japan
$^{(4)}$ Dept. of Mechanical Engineering, Bucknell University, USA
$^{(5)}$ Air-Conditioning Research Lab., Matsushita Electric Industrial Co. Ltd., Japan

ABSTRACT

This study presents empirical values of the friction factor for leakage flow through small axial and radial clearances between the orbiting and fixed scrolls of scroll compressors. Leakage flow experiments were conducted using both CO$_2$ and R22 to permit comparison. The refrigerant flowed from a pressurized vessel to the atmosphere through thin rectangular cross-sectional openings with small clearances. The pressure decay in the pressurized vessel due to leakage was measured using a maximum pressure of 3 MPa for CO$_2$ and 0.6 MPa for R22. The Darcy-Weisbach equation for incompressible, viscous fluid flow through the thin rectangular cross-section was applied to calculate the leakage mass flow rate, thus simulating the pressure decay characteristics, where the empirical friction factors were determined and plotted on a Moody diagram. As a result, it was shown that the empirical friction factors for both axial and radial clearance leakage flows take on essentially the same value for both CO$_2$ and R22, despite the significantly different working pressures. The friction factor was strongly dependent on the relative roughness of leakage channel surface. Subsequently, the empirical friction factor was incorporated into computer simulations for both CO$_2$ and R410A scroll compressors to determine their optimal performance. As a result, it was shown that CO$_2$ scroll compressors can achieve high performance levels, comparable with those of R410A scroll compressors.

Nomenclature

- $d$: Equivalent diameter, m
- $G$, $G_0$: Mass of refrigerant gas, kg
- $G$: Gravity acceleration, m·s$^{-2}$
- $H$: Channel height, m
- $L$: Leakage channel length, m
- $m$, $m_0$: Hydraulic mean depth, m
- $M$: Leakage mass flow rate, kg·s$^{-1}$
- $n$: Polytropic exponent
- $P$, $P_0$, $P_1$: Absolute pressure of refrigerant gas, Pa
- $P_\infty$: Atmospheric pressure, Pa
- $R$, $r$: Radius of equivalent channel model for radial clearance, m
- $Re$: Reynolds number
- $u_m$, $u_0$: Average flow velocity, m·s$^{-1}$
- $W$: Leakage channel depth, m
- $\alpha_a$, $\alpha_r$: Coefficient of friction factor
- $\beta_a$, $\beta_r$: Index of friction factor
- $\delta_a$, $\delta_r$: Leakage clearance, $\mu$m
- $\lambda$, $\lambda_a$, $\lambda_r$: Friction factor
- $\mu$: Coefficient of viscosity, Pa·s
- $\rho$, $\rho_0$: Density of refrigerant gas, kg/m$^3$

© The Authors 2005
1. INTRODUCTION

In recent years, scroll compressors, because of their low vibration, low noise and high efficiency, have become an increasingly popular choice for compressors not only in conventional refrigerant air-conditioning systems, but also in systems using CO₂ as the refrigerant. These scroll compressors in air-conditioners have small axial and radial clearances between the orbiting and fixed scrolls. The leakage of the compressed refrigerant gas through these small clearances has a strong detrimental effect on the volumetric efficiency. In order to carry out accurate performance simulations of scroll compressors for efficiency optimization, it is necessary to establish a reliable method of calculating the leakage flow through these axial and radial clearances.

From this perspective, experiments for the leakage flow of R22 and R410A through axial and radial clearances were conducted by Ishii *et al.* [1], [2], in which it was shown that the leakage flow through the small clearances in scroll compressors can be calculated by the Darcy-Weisbach equation for incompressible, viscous fluid flow, despite the fact that the working fluid itself is compressed in the compressor. However, when CO₂ is used as the refrigerant, the operating pressure is comparatively higher, possibly as high as 9 MPa at maximum. It is therefore necessary and prudent to re-examine the applicability of the Darcy-Weisbach equation for CO₂ refrigerant leakage flow.

In the present study, leakage flow experiments were conducted using both CO₂ and R22 to permit comparison. The refrigerant flowed from a pressurized vessel to the atmosphere through thin rectangular cross-sectional openings with small clearances. The refrigerant gas was initially contained in a large reservoir at a maximum pressure of 3 MPa for CO₂ and 0.6 MPa for R22. The decay of the upstream reservoir pressure due to gas leakage through the thin rectangular cross-section with a known surface roughness was measured at a variety of initial pressures. Subsequently, the Darcy-Weisbach equation for incompressible, viscous fluid flow was applied to the leakage flow through the axial and radial clearances, in order to calculate the leakage mass flow rate. The measured pressure decay rates were carefully modeled by assuming a polytropic process and adjusting the friction factor. As a result, the empirical friction factors were determined for the leakage flows of both CO₂ and R22. These friction factors were then plotted on a Moody diagram to document clearly the effect of the surface roughness of leakage flow channel. Finally, the empirical friction factors were incorporated into computer simulations of both CO₂ and R410A scroll compressors, and the volumetric, mechanical and compression efficiencies and the resultant overall efficiency were calculated both for CO₂ and R410A scroll compressors, to determine their optimal performance.

2. EXPERIMENTAL SET-UPS

In a scroll compressor, the refrigerant leaks through the axial clearance \( \delta_a \) and the radial clearance \( \delta_r \), due to the pressure difference between the compression chambers, as shown in Fig. 1. In order to ascertain the leakage characteristics, two test models shown in Fig. 2 were

![Figure 1. Refrigerant leakage flows in scroll compressors: (a) cross-sectional view; (b) leakage flow through radial clearance; (c) leakage flow through axial clearance.](image-url)
constructed. Both were covered with a thrust plate. Fig. 2(a) shows the model for the leakage flow through the axial clearance, where the streamwise length and transverse depth of the axial clearance were fixed at 4.0 mm and 15.0 mm, respectively. Fig. 2(b) shows the model for the leakage flow through the radial clearance, where the involute curves of orbiting and fixed scrolls were represented by two circular arcs with differing radii. The radii were fixed at 14.4 mm and 11.8 mm, respectively, and the transverse depth of the radial clearance was fixed at 15.0 mm. Both the axial and radial clearances were carefully adjusted to 10 µm by using thickness gauges with a width of 2.5 mm as shown in Fig. 2(c). Therefore, the leakage transverse depth is reduced from 15 mm to 10 mm. Special attention was given in the design of these experimental models to maintain adequate strength against the high pressure CO₂ gas. In addition, as shown in Figs. 2a and 2b, an O-ring was attached between the test piece and the thrust plate, and a liquid gasket was also used on the contact surface to entirely eliminate any unintended leakage.

The high pressure chamber on the left hand side of the test section was directly connected to a supply tank with a volume of 860 cm³, in order to avoid any disturbances in the leakage flow. The low pressure chamber on the right hand side was open to atmospheric pressure through a discharge valve. With the discharge valve closed, both the high pressure and low pressure chambers were initially pressurized at a specified pressure. Then the low pressure chamber was suddenly vented to the atmosphere by opening the discharge valve. The pressure in the high pressure chamber decreases due to refrigerant leakage from the test chamber clearance, and its time-dependent pressure drop was measured by a semiconductor pressure transducer.

3. LEAKAGE TEST RESULTS

The experimentally determined pressure decay in the high pressure chamber due to leakage through a 10 µm axial (left side) and a 10 µm radial clearance (right side) are shown by the solid lines in Fig. 3, where Fig. 3(a) is for CO₂ gas leakage and Fig. 3(b) is for R22 gas leakage. For the CO₂ gas leakage, 5 different values of initial pressure were used, ranging from 1.0 MPa to 3.0 MPa. For the R22 gas leakage, 4 different initial pressures were used ranging from 0.3 MPa to 0.6 MPa. These values of initial pressure were chosen as being representative of the pressure difference between adjacent compression chambers in scroll compressors. The data in Fig. 3 show that upon the sudden connection of the low pressure chamber to the atmosphere at time t=0, the pressure in the high pressure chamber decays due to refrigerant leakage, approaching atmospheric pressure (about 0.1 MPa). Since both the CO₂ and the R22 leakage tests were made under the atmospheric exit conditions, the initial temperature was between 16°C to 21°C.
The clearance height in both the axial and radial tests was 10 µm. However, the rate of pressure decay through the radial clearance is about 3-times faster than through the axial clearance. This tendency is recognized for both the CO2 and the R22 leakage tests.

4. CALCULATIONS OF PRESSURE DROP AND EMPIRICAL FRICTION FACTORS

Once a theoretical method for calculating refrigerant gas leakage flows through the axial and radial clearances is established, computer simulations of the scroll compressor can be made to optimize its performance. However, it is noted here that the theoretical method for the refrigerant gas leakages should be kept as simple as possible, since the computer simulations of the resultant performance include many complicated procedures to calculate the mechanical, compression and volumetric efficiencies. The following analysis presents a very simple method for calculating the refrigerant gas leakage flows through the axial and radial clearances in a scroll compressor based on the Darcy-Weisbach equation for incompressible, viscous fluid flow through the thin rectangular cross-section.

4.1 Leakage through the axial clearance channel

When the axial clearance leakage flow through the thin rectangular cross-section is assumed to be an incompressible viscous flow, the following relation can be derived from the conservation of momentum principle:

$$\frac{P - P_a}{\rho g} = \lambda_a \frac{L}{4m} \frac{u_m^2}{2g}$$

(1)

where the frictional force acting on the wall outside the leakage passage was neglected, since it is far smaller than the frictional forces on the leakage passage area. This expression is known as the Darcy-Weisbach equation when written for a pipe flow, and indicates that the pressure drop (P-P_a) through the leakage channel with a length of L is basically determined by the friction factor \( \lambda_a \) of the channel surface. The average leakage flow velocity is represented by \( u_m \). Since the axial clearance height, \( \delta_a = 10 \) µm, is very small compared with the channel depth \( W=10 \) mm, the hydraulic mean depth, \( m \), for the axial clearance with a rectangular cross section is given by

$$4m = \frac{4\delta_a W}{2(\delta_a + W)} \rightarrow 2\delta_a$$

(2)

The hydraulic diameter \( d \) of a circular pipe with a pressure drop equivalent to that of the rectangular cross-section channel is given by this quantity \( 4m (\equiv d) \).
Given values of the friction factor $\lambda_a$ and the pressure difference $(P-P_a)$, the mean leakage flow velocity $u_m$ can be calculated from Eq. (1), and the mass flow rate $M$ can be calculated by

$$\dot{M} = \rho \delta_a W_u_m$$

(3)

This flow produces the following pressure decay $\Delta P$ in the high pressure chamber over a small time $\Delta t$:

$$\Delta P = \frac{P_0}{G_0^n} \cdot n \cdot G^{n-1} \cdot \dot{M} \cdot \Delta t$$

(4)

where the pressure decay is assumed to be a polytropic process with exponent $n$. $P_0$ represents the initial pressure. $G$ represents the residual refrigerant mass in the high pressure chamber, which can be calculated by subtracting the total leakage mass from the initial refrigerant mass $G_0$:

$$G = G_0 - \int_0^1 \dot{M} dt$$

(5)

Using Eqs. (4) and (5), the pressure $P$ in the high pressure chamber can be calculated successively.

The friction factor $\lambda_a$ is generally a function of the Reynolds number $Re$. Here the following expression for the friction factor is assumed:

$$\lambda_a = \alpha \cdot Re^{-\beta}$$

(6)

which corresponds to fully turbulent flow. The Reynolds number $Re$ is defined by

$$Re = \frac{4mu_m}{\mu / \rho} = \frac{2\delta u_m}{\mu / \rho}$$

(7)

where the equivalent diameter $d (= 4m)$ is chosen as the representative length. The viscosity coefficient $\mu$ is given in Table 1, as a function of pressure at 18°C. Using a polytropic polytropic exponent $n$ of 1.30 for CO$_2$ and 1.32 for R22, the density $\rho$ can be calculated as

$$\rho = \rho_0 \left( \frac{P}{P_0} \right)^{\frac{1}{n}}$$

(8)

When the coefficient $\alpha$ and exponent $\beta$ in Eq. (6) for the friction factor $\lambda_a$ are assigned the values given in the leftmost plot in Fig. 4(a), the pressure decay simulated for CO$_2$ gas shows close agreement with the measured decay, as shown by the dotted lines in the first diagram in Fig 3a. Using the mean values of the plotted data for $\alpha$ and $\beta$, the friction factor $\lambda_a$ for CO$_2$ gas leakage flow through the axial clearance can be given by

$$\lambda_a = 3.54 Re^{-0.43}$$

(9)

For R22 gas, similar analyses were carried out. Close agreement between the calculated and the measured pressure decay, as shown by the dotted lines in the first diagram of Fig 3b, was obtained when the values of $\alpha$ and $\beta$ shown in the first diagram of Fig. 4(b) were used. Taking the average values of the data for $\alpha$ and $\beta$, the friction factor $\lambda_a$ for R22 gas leakage flow through the axial clearance can be given by

$$\lambda_a = 3.50 Re^{-0.45}$$

(10)

Table 1. Coefficient of viscosity of pressurized CO$_2$ gas and R22 gas at 18°C.

<table>
<thead>
<tr>
<th>Pressure [MPa]</th>
<th>CO$_2$ [\mu Pa·s]</th>
<th>R22 [\mu Pa·s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01</td>
<td>-</td>
<td>12.51</td>
</tr>
<tr>
<td>0.05</td>
<td>-</td>
<td>12.51</td>
</tr>
<tr>
<td>0.1</td>
<td>14.49</td>
<td>12.50</td>
</tr>
<tr>
<td>0.5</td>
<td>14.73</td>
<td>12.41</td>
</tr>
<tr>
<td>1.0</td>
<td>14.97</td>
<td>-</td>
</tr>
<tr>
<td>1.5</td>
<td>15.20</td>
<td>-</td>
</tr>
<tr>
<td>2.0</td>
<td>15.49</td>
<td>-</td>
</tr>
<tr>
<td>2.5</td>
<td>16.20</td>
<td>-</td>
</tr>
<tr>
<td>3.0</td>
<td>16.92</td>
<td>-</td>
</tr>
</tbody>
</table>

As shown in the center diagrams of Figs. 4(a) and 4(b), the leakage flow velocity $u_m$ is significantly smaller than the sonic speed for CO$_2$ (about 250 m/s) and R22 (about
170m/s), yielding a Mach number less than 0.3 and justifying the treatment of the flow as incompressible. The tested maximum Reynolds numbers are 5800 for CO₂ and 1400 for R22.

4.2 Leakage through the radial clearance channel

A model of the radial clearance channel is shown in Fig. 5, where CO₂ gas flows through a small clearance between two circular arcs with the radii of r and R, from left to right, due to the pressure difference (P - P₀). The minimum height of the radial clearance is represented by δ. The clearance height at any counter clockwise angle φ from the minimum clearance position is represented by h, which can be approximated as:

\[ h = R - (R - r - \delta_r) \cos \phi - \sqrt{r^2 - (R - r - \delta_r)^2 \sin^2 \phi} \]  

Continuity of the radial leakage flow can be used to find the flow velocity \( u_\phi \) at the angle \( \phi \) in terms of the leakage flow velocity at the minimum clearance, \( u_m \):

\[ u_\phi = \frac{\delta}{h} u_m \]  

The head loss in the radial clearance results from the frictional loss due to the fluid viscosity. The head loss can be calculated by integrating the local differential frictional loss over the whole leakage channel:

\[ \frac{P - P_a}{\rho g} = \int_{\phi_0}^{\phi} \lambda_r \frac{R d\phi}{4m_\phi} u_\phi^2 \]  

where the angle \( \phi_0 \) represents the integral region, and the hydraulic mean depth \( m_\phi \) is given by

\[ 4m_\phi = \frac{2hW}{h + W} \]  

The Reynolds number \( Re \) can be defined by

\[ \frac{P - P_a}{\rho g} = \int_{\phi_0}^{\phi} \lambda_r \frac{R d\phi}{4m_\phi} u_\phi^2 \]
Assuming the friction factor $\lambda_r$ can be written in the same form as $\lambda_a$ in Eq. (6), the time-dependent pressure $P$ in the high pressure chamber can be calculated successively by Eqs. (13) and (3) to (5). If $\alpha$ and $\beta$ in the formulation of the friction factor $\lambda_r$ are given by the data plotted in the first diagram of Fig. 6(a), the pressure decay calculated for CO$_2$ gas shows close agreement with the measured decay, as shown by the dotted lines in the rightmost plot in Fig 3(a). As a result, taking the mean values of the plotted data of $\alpha$ and $\beta$, the friction factor $\lambda_r$ for CO$_2$ gas leakage flow through the radial clearance can be given as

$$\lambda_r = 3.70 \text{Re}^{-0.46}$$  \hspace{1cm} (16)

A similar analysis of the pressure decay was made for R22 gas, confirming a close agreement with the measured pressure decay, as shown by the dotted lines in the rightmost plot in Fig. 3(b). The values of $\alpha$ and $\beta$ are plotted in the leftmost plot in Fig. 6(b). As a result, the friction factor $\lambda_r$ for R22 gas leakage flow through the radial clearance can be given by

$$\lambda_r = 3.63 \text{Re}^{-0.45}$$  \hspace{1cm} (17)

As shown in the center and rightmost diagrams of Figs. 6(a) and 6(b), the representative leakage flow velocity $u_m$ is low enough relative to sonic speeds of CO$_2$ and R22 to justify incompressible analysis, and the tested maximum Reynolds number is 13561 for CO$_2$ and 2482 for R22.

### 5. EMPIRICAL FRICTION FACTORS ON MOODY DIAGRAM

The empirical friction factors for a leakage clearance height of 10$\mu$m are denoted by the thick solid lines on the Moody diagram in Fig. 7. The empirical values over the tested range of Reynolds number are denoted by the solid line, while the
extrapolated values for higher Reynolds numbers are shown as dashed lines. As shown in Table 3, the absolute surface roughness $\varepsilon$ of the leakage channels took on values ranging from 0.23 to 0.25 $\mu$m for three tests (CO$_2$ radial, R22 axial and radial clearances) with a corresponding relative roughness $\varepsilon/d$ from 0.011 to 0.013. As a result, the values of the empirical friction factors for these three tests are very close to each other. It is emphasized here that very similar empirical friction factors were found for the two refrigerants (CO$_2$ and R22), despite large differences in working pressures. Similarly, the two types of leakage channels also have similar friction factors. In contrast, a different surface roughness of 0.80 $\mu$m was found for the test with CO$_2$ through the axial clearance, resulting in a relative roughness of 0.04. This value is about 3 to 4 times larger, resulting in a slightly larger friction factor, as shown on the Moody diagram.

The empirical friction factor values found in these tests are quite large relative to the expected values for laminar pipe flow. Therefore, it is suggested that the leakage flows through the narrow rectangular leakage channels were in a fully turbulent state. The surface roughness of the leakage passage was comparatively large, and the initial leakage flow velocity was also large. The initial sudden release of compressed gas through the check valve may have produced a large enough disturbance to initiate turbulent flow that persisted throughout the leakage tests.

6. CALCULATIONS OF COMPRESSOR EFFICIENCIES

Computer simulations of CO$_2$ and R410A scroll compressors were undertaken to optimize compressor performance (see [3], [4]). The friction factors used in the simulations were all assumed to follow the relationship given in Table 4, in which the constant coefficients are the numerical averages of the values found in the four test cases. Also shown in Table 4 are the major specifications for the two compressors. The suction volume and compression ratio were 4.25 cm$^3$ and 2.07 for CO$_2$, and 11.4 cm$^3$ and 2.4 for R410A, resulting in the same cooling capacity of 727 kJ/h at the rated values of mean crankshaft speed, suction temperature, suction pressure and discharge pressure given in Table 4. The suction and discharge pressures were 3.5 and 9.0 MPa for CO$_2$, and 0.81 and 2.46 MPa for R410A. The scroll thickness and cylinder diameter, based on the current scroll compressor designs, were kept at 3.0 mm and 67.54 mm, respectively. The clearance between the orbiting and fixed scrolls was kept at 3.0 $\mu$m for the axial direction and 6.0 $\mu$m in the radial direction. The crankshaft moment of inertia, $I_0$, was adjusted depending upon the necessary driving shaft power, and the orbiting scroll mass $m_o$ depended on the scroll height B. The coefficients of Coulomb friction at each pair of moving compressor elements were kept at from 0.055 to 0.0013, as measured by friction tests.

| Table 4. Major specifications and mechanical constants of scroll compressor. |
|---------------------------------|---------------|---------------|
| Suction volume $V_s$ [cm$^3$]   | CO$_2$ 4.25   | R410A 11.4    |
| Cooling capacity $Q_s$ [kJ/h]   | 727          | 727          |
| Operation speed $n$ [rpm]       | 3498         | 3498         |
| Involute base circle radius $r_0$ [mm] | 1.4 ~ 2.8 | 1.4 ~ 2.8 |
| Scroll height $B$ [mm]          | 9.7 ~ 3.6    | 26.0 ~ 9.8   |
| Scroll thickness $t$ [mm]       | 3.0          | 3.0          |
| Cylinder diameter $D$ [mm]      | 67.54        | 67.54        |
| Volume ratio $\lambda_v$        | 2.07         | 2.40         |
| Pressure ratio $\lambda_p$      | 2.57         | 3.04         |
| Specific heat ratio $\kappa$    | 1.30         | 1.074        |
| Suction temperature $T_s$ [°C]  | 10.5         | 10.45        |
| Suction pressure $P_s$ [MPa]    | 3.5          | 0.81         |
| Discharge pressure $P_d$ [MPa]  | 9.00         | 2.46         |
| Axial clearance $\delta_a$ [$\mu$m] | 3.0, 0     | 3.0          |
| Radial clearance $\delta_r$ [$\mu$m] | 6.0          |              |
| Empirical fric. factor $\lambda_e$ | axial       | 3.61Re$^{-0.45}$ |
| Friction coefficients $f_i$ at oldham ring | 0.055         |              |
| Friction coefficients $f_i$ at thrust bearing | 0.011         |              |
| Friction coefficients $f_i$ at crank journal | 0.011         |              |
| Friction coefficients $f_i$ at ball bearing | 0.0013         |              |
| Moment of Crankshaft $I_0$ [kg-m$^2$] | 0.107~0.114  | 0.139~0.147  |
| Orbiting scroll mass $m_o$ [kg] | 0.116~0.112  | 0.118~0.170  |
| Oldham ring mass $m_o$ [kg]     | 0.037        |              |
| Crankshaft radius $r_s$ [mm]    | 8.0          |              |
| Crankpin radius $r_o$ [mm]      | 8.0          |              |
| Fric. coef. at oldham ring       | 0.055        |              |
| Fric. coef. at thrust bearing    | 0.011        |              |
| Fric. coef. at crank journal     | 0.011        |              |
| Fric. coef. at ball bearing      | 0.0013       |              |
Calculations were made for the involute base circle radius \( r_b \) from 1.4 mm to 2.8 mm, which results in a scroll height \( B \) ranging from 9.7 mm to 3.6 mm. First, the leakage flow velocity and leakage mass flow rate were calculated for both the axial and radial clearances using the empirical friction factor. Then the compressed gas pressure was calculated. From the pressure, the net leakage mass \( \Delta G \) during one revolution of the crankshaft was calculated, as shown in Fig. 8(a), where the solid line is for CO2 compressor and the dashed line is for R410A compressor. The abscissa is the involute base circle radius \( r_b \). The friction factor was almost identical for the CO2 and for the R410A, while the pressure difference driving the gas leakage was significantly larger for CO2 than for R410A. Thus, the net leakage mass \( \Delta G \) for CO2 is about 2 to 4 times larger than that for R410A. As a result, the volumetric efficiency \( \eta_v \) was reduced by about 8.9% for CO2 when compared with that for R410A, as shown in Fig. 8(b).

However, the volumetric efficiency \( \eta_v \) for both CO2 and R410A exceeds 80%, indicating that only a small percentage of the initial suction mass leaks from the compressor. Therefore, the compressed gas pressure is not seriously affected by the gas leakage, and the compression efficiency \( \eta_c \) for both CO2 and R410A exceeds 90%, as shown in Fig. 8(c). The mechanical efficiency \( \eta_m \) also exhibits levels in excess of 80% for both CO2 and R410A, as shown in Fig. 8(d). As a result, the overall efficiency \( \eta \) is clearly dominated by the volumetric efficiency, indicating lower performance for the CO2 compressor than for the R410A compressor, as shown in Fig. 8(e). The optimal performance was found at \( r_b = 2.2 \) mm, where the difference in performance between the two machines was about 6.9%.

In the hypothetical limit, now assume that the axial clearance is reduced from 3 \( \mu \)m to 0 \( \mu \)m in the CO2 compressor. The calculated results for such a condition are shown by the dotted lines in Fig. 8. \( \Delta G \) is significantly decreased, as shown in Fig. 8(a), resulting in a significant improvement in the volumetric efficiency for the CO2 compressor. As a result, the overall efficiency for CO2 takes on an optimum value of about 80%, only 2.0% lower than for R410A, as shown in Fig. 8(e). In addition, if the radial clearance also is reduced from 6 \( \mu \)m to 5 \( \mu \)m in the CO2 compressor, the optimum efficiency of the CO2 compressor approaches the same 82% level found for the R410A compressor.

### 7. CONCLUSION

Leakage flow experiments were conducted for two refrigerants, CO2 and R22, flowing through the axial and radial clearances with a height of 10\( \mu \)m, between the orbiting and fixed scrolls of scroll compressors, where the pressure drop due to leakage from 3 MPa for CO2 and 0.6 MPa for R22 was measured over a range of Reynolds numbers up to \( 1.2 \times 10^4 \) for CO2 and \( 2.5 \times 10^7 \) for R22. The Darcy-Weisbach equation for incompressible, viscous fluid flow through the thin
rectangular channels was used to calculate the pressure drop of the leakage flow. Empirical values for the friction factor were determined by comparing the calculated pressure drop with the measured values. The empirically determined friction factors were then plotted on a Moody diagram. It was shown, regardless of refrigerant, that the empirical friction factors for both axial and radial clearance leakage flows take on essentially the same values, suggesting that the leakage flows were fully turbulent. The friction factor $\lambda$ was found to be strongly dependent on the relative roughness of leakage channel surface, and for relative roughnesses from 0.011 to 0.013 was given by $\lambda = 3.61Re^{-0.45}$. The present leakage tests were made for specified pressure differences. However, since the pressure difference between adjacent compression chambers in scroll compressors does not significantly change during compression processes, the test results can be effectively applied to scroll compressors under operating conditions. In addition, the tests were made at a lower temperature than those typically found in scroll compressors, thus resulting in higher test condition Mach number than would be found in operating compressors. Therefore, the tests presented here would be expected depend more strongly on the compressibility of the working fluid. Since no compressibility effects were found in these leakage tests, none should be expected in the leakage flows inside an operating compressor.

Subsequently, the empirical friction factor was incorporated into computer simulations for both CO$_2$ and R410A scroll compressors with the same leakage height of 6$\mu$m for both the axial and radial clearances. The objective was to determine the optimum performance and the involute base circle radius $r_b$ at which it occurred. The optimal overall efficiency was 74.5% at $r_b=2.0$ mm for the CO$_2$ compressor, which was 6.9% lower than the corresponding value for the R410A compressor. Further computer simulations showed that if the axial and radial clearances were reduced to 0$\mu$m and 5$\mu$m, respectively, CO$_2$ scroll compressors can achieve a relatively high 82% efficiency, comparable with that of R410A scroll compressors.

**ACKNOWLEDGEMENT**

The authors would like to express their sincere gratitude to Mr. Masanobu Seki, Senior Counselor, Air Conditioning Devices Division, Matsusita Home Appliances Company, the Matsusita Electric Industrial Co., Ltd., and Mr. Shuichi Yamamoto, Director, Air-Conditioning Research Laboratory, Corporate Engineering Division, Matsusita Home Appliances Company, the Matsusita Electric Industrial Co., Ltd., for their cooperation in carrying out this work and their permission to publish this study. The authors extend their thanks to Mr. Yohei Tada, graduate student, Graduate School of Osaka Electro-Communication University, who assiduously conducted the difficult experiment of leakage for the present study.

**REFERENCES**


Development of R-410A scroll compressor used with brushless DC motor control

Yu-Choung Chang, Ann Huang, Kun-Yi Liang
Compressor Lab., Energy & Resources Laboratories, ITRI, Taiwan
Ching-Huan Tseng
Department of Mechanical Engineering, National Chiao Tung University, Taiwan

ABSTRACT

To develop an HFC-410A (R-410A) scroll-type compressor (STC) from original HCFC-22 (R-22) model with least change, and combine DC brushless motor and inverter control technologies, are the objectives in this study. A systematic design method with optimization has been proposed, and the developed R-410A STC presents better performance. The detail results of the STC performance that include cooling capacity, power consumption and coefficient of performance (COP), are measured by the calorimeter, meanwhile, the performance curve of the developed STC has also been obtained. At rated operating conditions with various speed operations, the COP of this new R-410A STC model, is higher than 3.0 and meet the requirements of commercial products.

1. INTRODUCTION

Among the various compressors, the scroll type compressor (STC) is known for its have higher isentropic and volumetric efficiencies, lower noise and more vibration free, are employed in the applications of room and packaged air-conditioners widely. Specially, the STC used with inverter-controlled and applied on air-conditioners has been popular gradually in Asia area.

For refrigerant alternative issue, the new refrigerants HFC-410A (R-410A) is considered to be the major substitutes for R-22, that was the major refrigerant used in room and packaged air-conditioners during past years. Note that the vapor pressure of R-410A is about 60% higher than that of R-22. As a result, in order to achieve competitive performance relative to the original R-22, the new STC design of R-410A need to make more efforts.

In these connections, it is important to investigate the characteristics of the STC subjected to the alternative refrigerant R-410A and combined with inverter control technology.

The major target of this study is to built-up an optimization design algorithm and to develop the R-410A STC associated with brushless DC motor and inverter control. Thereafter, the prototypes of the R-410A model have been implemented and the performance tests have also been carried out to verify the design results.
2. THE OUTLINE OF STC DESIGN STRUCTURE

Figure 1 shows the cross section of a hermetic STC that is designed for room air-conditioner. The major components include a scroll set (fixed and orbiting scroll with scroll wraps), a set of backpressure mechanism, an Oldham coupling ring, three bearings (driving bush, main journal bearing, lower journal bearing), a crankshaft and a driving motor. The examined structure of the developing STC is a low-pressure-shell design and combined with a solid axial compliance mechanism. The solid axial compliance mechanism is defined as the fixed scroll is urged by the solid force from the pressing members to move axially and to remain in a close contact with orbiting scroll so as to overcome the leak that takes place in the end surface between two scroll members. The pressing members may be one solid ring or multi-pins and located on the circumferential planar surface of the back of the fixed scroll. This innovation mechanism has presented good performance in other case studies [1].

To shorten the developing time for a new model design, a better computer simulation package is needed. In this study, a practical computer tool that is developed by ITRI [2] has been utilized for estimating the performance of STC. Figure 2 shows the simulation flowchart of this paper used with the STC design tool.

Figure 1: The cross section of a hermetic STC used in this study
3. DESIGN PROCESS FOR THE R-410A STC DEVELOPING

3.1 Define the basic design requirements

Table 1 depicts the basic specifications of a R-410A STC product requirement, which will be used for the specified R-410A air-conditioner with inverter control. The objective is to get the minimum COP (coefficient of performance) of the developing STC must higher than 3.0 at rated operating conditions and the cooling capacity is limited to the range of 3000W ± 100W.
Table 1: Basic specifications of the R-410A STC used in this study

<table>
<thead>
<tr>
<th>Refrigerant</th>
<th>R-410A</th>
</tr>
</thead>
<tbody>
<tr>
<td>Input power</td>
<td>220V, single phase</td>
</tr>
<tr>
<td>Lubricants</td>
<td>POE VG32</td>
</tr>
<tr>
<td>Shell type</td>
<td>Low pressure shell</td>
</tr>
<tr>
<td>Outward dimensions</td>
<td>$\leq 120 \phi \times 320 H$ mm</td>
</tr>
<tr>
<td>Motor type</td>
<td>4-Pole DC brushless motor</td>
</tr>
<tr>
<td>Operating speed range</td>
<td>3600–5400 rpm</td>
</tr>
<tr>
<td>Cooling Capacity at rated operating condition</td>
<td>3000W ±100W</td>
</tr>
<tr>
<td>Required COP at rated operating condition</td>
<td>$\geq 3.0$</td>
</tr>
</tbody>
</table>

The rated operating conditions

<table>
<thead>
<tr>
<th>Condensing Temp.</th>
<th>Evaporating Temp.</th>
<th>Subcooling Degree</th>
<th>Superheating Degree</th>
<th>Room Temp.</th>
</tr>
</thead>
<tbody>
<tr>
<td>54.4°C</td>
<td>7.2°C</td>
<td>8.3°C</td>
<td>27.8°C</td>
<td>35.0°C</td>
</tr>
</tbody>
</table>

The specified range of operating conditions

<table>
<thead>
<tr>
<th>Condensing Temp.</th>
<th>Evaporating Temp.</th>
<th>Subcooling Degree</th>
<th>Superheating Degree</th>
<th>Room Temp.</th>
</tr>
</thead>
<tbody>
<tr>
<td>40.0–55.0°C</td>
<td>5.0–15.0°C</td>
<td>8.3°C</td>
<td>30.0–20.0°C</td>
<td>35.0°C</td>
</tr>
</tbody>
</table>

3.2 Design variables decisions

The first step is to identify the STC design variables, design constraints and the objective function. Four major parameters are used to define the basic dimensions of the scroll set, which are the pitch $p$, thickness $t$, height $h$ and extending angle $\phi_e$ of scroll wraps, have been selected as the design variables in this study. Figure 3 shows the geometrical definitions of these four relevant design variables.

The initial data of these four design variables are come from the existing dimensions of STC used with refrigerant R-22 that is operated at constant speed of 3450rpm and the rated cooling capacity is 2900W.

![Figure 3: Four major design parameters of scroll wrap](image)

Based on the least change requirement from existing R-22 STC, this study selects to use the same crankshaft between these two STC models. It means the developing R-410A STC need design to use with the same orbiting radius as original R-22 STC. Therefore, the four
design variables $P$, $t$, $h$, $\phi_E$ can be reduced to three as $t$, $h$, $\phi_E$, as for using with the same orbiting radius:

$$r_{ob} = \frac{p - t}{2},$$

which gives one constraint between $P$ and $t$.

Meanwhile, the finite element analysis used with SolidWorks 2003 and COSMOS/WORKS has been worked out. Figure 4 and Table 2 show the stress and strain results of the fixed and orbiting scroll members under specified loadings. The safety factors are higher than 4.5 and the maximum displacements are small than 13 µm. The results all can be accepted by experienced engineers. Therefore, the wrap rigidity of scroll and the cutter tool rigidity can be defined as:

$$G_W = \frac{h}{t} \leq 6.0 \quad \text{and} \quad G_t = \frac{h}{(p - t)} \leq 2.5,$$

respectively. It clearly defines the other constraints of $P$, $t$, $h$ for the developing STC design.

![Figure 4: Stress and strain analysis of scroll members in this study.](image)

Table 2: Stress analysis results of scroll members

<table>
<thead>
<tr>
<th>Items</th>
<th>Max. Von Mises stress (Mpa)</th>
<th>Max. Shear stress (Mpa)</th>
<th>Min. Safety factor</th>
<th>Displacement (µm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Orbiting scroll</td>
<td>21.73</td>
<td>11.85</td>
<td>4.5</td>
<td>7.6</td>
</tr>
<tr>
<td>Fixed scroll</td>
<td>17.76</td>
<td>10.04</td>
<td>5.3</td>
<td>12.9</td>
</tr>
</tbody>
</table>

417
3.3 Optimization process

An optimizing sequence for the STC subjected to change search direction between related design variables has been evaluated [3]. The objective function is COP and the required minimum value is 3.0. Figure 5 describes the optimum design process in this study detailedly.

![Diagram of optimization process]

**Identify:**
1. Design variables: \( p, t, h, \phi_E \)
2. Objective function E.E.R. to be maximization
3. Design Constraints: cutter rigidity, \( h/t \), etc.

Collect data to describe the R-410A STC system
1. Motor efficiency
2. Friction coefficient of each contact surface of STC
3. Suction superheat temperature & leakage clearance estimation from experimental measurement
4. Oil viscosity vs. temperature

Estimate initial design

Analyze the system with STC simulation software package developed

Check the constraints

Does the new design satisfy convergence criteria?

Interactive session

Change the search direction and step size of design using manual method with practical experience

Figure 5: The optimum design process for R-410A STC developing in this study

Based on the R22 STC original dimensions, the initial design data have been selected as \( p = 12mm, \ t = 3mm, \ r_{ob} = 3mm \). Figure 6 shows the evaluation results of the first phase iteration, every scroll extended angle can subject one height value of scroll wrap to obtain a cooling capacity under design constraints of this study required. But the feasible region to meet the 2900W~3100W of cooling capacity constraint occurred at \( 930^\circ \leq \phi_E \leq 980^\circ \) and the maximum COP is on \( \phi_E = 930^\circ \).
Thereafter, searching the optimum value of $t$ is carried out. Figure 7 presents to increase $t$, the COP is decreased, in the mean time, the operating compression ratio is increased. Due to the manufacture limit and the R-410A STC model with inverter controlled would be operated at lower compression ratio for light-loading or high-speed conditions, thus $\phi_e = 930^\circ, h = 12.3, t = 2.7$ is selected as final design. The evaluated performance of the developed R-410A model has shown as Figure 7, the predicted COP is 3.03 at rated operating condition. Table 3 shows the comparisons of design data between the original R22 and the developing R-410A STC. In practical application, only three parts has been design change from original R22 STC in this study. They are fixed scroll, orbiting scroll and back-pressure mechanisms.

![Figure 6: Evaluated results of first phase with optimum design process](image)

Max. COP occurs at $\phi_e = 930^\circ$

Feasible region

![Figure 7: Evaluative results for searching optimum thickness of scroll wrap](image)

Selected point $t = 2.7\text{mm} & h = 12.3\text{mm}$
Table 3: The comparisons between R-410A and original R22 STC

<table>
<thead>
<tr>
<th>Refrigerant</th>
<th>R-22</th>
<th>R-410A</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required cooling capacity at rated conditions (W)</td>
<td>2900</td>
<td>3000</td>
</tr>
<tr>
<td>Height of scroll wrap (mm), $h$</td>
<td>14.5</td>
<td>12.3</td>
</tr>
<tr>
<td>Pitch of scroll wrap (mm), $p$</td>
<td>12.0</td>
<td>11.4</td>
</tr>
<tr>
<td>Thickness of scroll wrap (mm), $t$</td>
<td>3.0</td>
<td>2.7</td>
</tr>
<tr>
<td>Extending angle of scroll wrap (mm), $\phi_E$</td>
<td>1050.0</td>
<td>930.0</td>
</tr>
<tr>
<td>Suction volume (cc)</td>
<td>14.21</td>
<td>9.69</td>
</tr>
<tr>
<td>Volumetric ratio</td>
<td>2.75</td>
<td>2.30</td>
</tr>
<tr>
<td>Compression ratio at rated operating conditions</td>
<td>3.43</td>
<td>3.39</td>
</tr>
<tr>
<td>Calculated capacity at rated conditions (W)</td>
<td>2962.51</td>
<td>3097.98</td>
</tr>
<tr>
<td>Calculated COP</td>
<td>3.06</td>
<td>3.03</td>
</tr>
</tbody>
</table>

4. EXPERIMENTAL RESULTS

Figure 8 shows the prototyping products, every component included the brushless DC motor, was manufactured in Taiwan. In otherwise, the sensorless controller for driving brushless DC motor built in the R-410A STC was also developed by ITRI simultaneously.

The prototype of the new R-410A STC model has been verified at calorimeter of compressor. Detailed results of the compressor performance are measured. Figure 9 describes the performance data of the developing STC operated at variable speed and rated conditions. The COP all is higher than the target of 3.0 of the predefined objective. The deviation between evaluation and measurement is very close. At the range of operating conditions with different condensing and evaporating temperatures, the performance curves of the developed STC that operated at 4800rpm, has been present in Fig. 10.

5. CONCLUSIONS

This study demonstrated a practical process of STC design optimization used with alternative refrigerants R-410A and brushless DC motor. The efficiency of the developed R-410A STC is over the objective requirements after optimization. The prototype of this new model has been implemented and verified in performance at calorimeter. Some important results in this study are summarized as below:

(1) Following on the practical design requirements, the STC model used with original R22 STC model has been transferred to the new R-410A STC model easily.
(2) A sequence of optimizing method subjected to change search direction between major design parameters, has been used for the case study of R-410A STC product development and the results meet the objective requirements.
(3) The prototype of R-410A STC combined with brushless DC motor controller has implemented and verified the performance. At rated operating conditions and 3600rpm, the cooling capacity is within 3000W±100W, the COP is 3.03 and over the design objective. At various operating speeds and different operating conditions, the new R-410A STC model presents better performance and robust characteristics.
Figure 8: Prototyping products in this study

Figure 9: The performance data of the developed STC with various speeds and operated at rated conditions.
Figure 10: The performance curves of the developed STC operated at 4800rpm.

ACKNOWLEDGMENT

The authors would like to express gratitude for financial support from the Energy R&D foundation funding provided by the Energy Commission of the Ministry of Economic Affairs in Taiwan.

REFERENCES


Comparative study of the impact of the dummy port in a scroll compressor

M. M. Cui
TRANE Air Conditioning, American Standard Companies, USA

ABSTRACT
A dummy port plays an important role in the porting process and the improvement of the performance of a scroll compressor. This paper documents an investigation on the working mechanism of the dummy port in a scroll compressor. To characterize the dummy port effects on the different parts of the scroll compressor, two scroll compressors, one with and the other without dummy port, are studied comparatively. The flow through the dummy port is examined in the background of an integrated compressor working process. The compressor studied includes upper bearing housing, scrolls, check valve, and discharge plenum. The Navier-Stokes equations with a $k-\varepsilon$ turbulence model are solved at the standard operating conditions of a scroll compressor. Refrigerant-22 is used as the working fluid. The thermodynamic and transport properties of the refrigerant gas are modeled by the Martin-Hou equation of state and power laws, respectively. Global flow physics is investigated first to lay a foundation to understand the working mechanisms that control the porting process before averaging techniques are applied. The behavior of the gas pockets in the porting process is characterized in both geometric and dynamic nature. The time-dependent variation of volume, mass, energy, and volume-averaged field quantities inside the gas pockets are studied throughout the porting process. The impact of the dummy port on the compressor performance is defined.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>area</td>
</tr>
<tr>
<td>$b$</td>
<td>flank gap width</td>
</tr>
<tr>
<td>$m$</td>
<td>local mass flow rate</td>
</tr>
<tr>
<td>$M$</td>
<td>total mass flow rate</td>
</tr>
<tr>
<td>$P$</td>
<td>pressure</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
</tr>
<tr>
<td>$U$</td>
<td>velocity</td>
</tr>
<tr>
<td>$u$, $v$, $w$</td>
<td>component of the velocity</td>
</tr>
<tr>
<td>$V$</td>
<td>gas pocket volume</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>$\Phi$, $\Psi$</td>
<td>arbitrary physical quantity</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Subscripts</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$l$</td>
<td>leakage</td>
</tr>
<tr>
<td>$m$</td>
<td>mass averaged quantity</td>
</tr>
<tr>
<td>$in$</td>
<td>value at inlet</td>
</tr>
<tr>
<td>$out$</td>
<td>value at outlet</td>
</tr>
<tr>
<td>$max$</td>
<td>maximum value</td>
</tr>
<tr>
<td>$p$</td>
<td>pocket</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Superscripts</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>-</td>
<td>average</td>
</tr>
</tbody>
</table>
1 INTRODUCTION

A dummy port is typically located on the orbiting scroll. The shape and size are the same as the discharge port. The basic function of the dummy port is connecting two neighboring gas pockets to equalize the pressure in the two pockets connected. In this regard, the dummy port is an integrated part of the internal flow path of a scroll compressor. The overall flow path of a scroll compressor consists of a series of gas pockets in crescent-shaped volumes. These gas pockets move along a spiral profile and are connected by the thin flank gaps. The volume and shape of the gas pockets change with their locations. The variations of pressure and temperature inside the gas pockets generate leakage flows between the gas pockets. These gas pockets are eventually squeezed out periodically downstream through the discharge port. As a part of the discharge system, the dummy port provides an additional flow passage for pressured gas from gas pockets to discharge port. Eventually, the gas flow passes through the discharge passage and reaches the discharge plenum, a cylindrical open space. The flow through the dummy port is transient and three-dimensional. The complex, continuously moving and deforming of the scroll compressor geometry pose a challenge for numerical simulation of the flow and heat transfer at the dummy port. Although the dummy port has been used in scroll compressor designs, there are few attempts being made to analyze the work mechanism of the dummy port and predict its impact on compressor performance.

In the current study, a scroll compressor has been numerically simulated in an integrated fashion. The dummy port as one link in the flow path from the inlet to outlet of the scroll compressor is included in the simulation (Figure 1). The asymmetric structure of each component and interaction between them are analyzed. The inlet of the compression chamber is located at the bottom of the structure (Figure 1(a)). The Figure 1(b) shows the shape of the scrolls and surrounding structure inside the compressor. The dummy port is located on the bottom of the geometry at the center of the scroll compressor.

![Figure 1. Flow domain of the scroll compressor: (a) compressor; (b) scroll involute.](image)

2 METHODOLOGY

The overall feature of the scroll compressor and the numerical techniques used in the simulations have been described previously [1,2,3]. The current paper focuses at the work associated with the dummy port. Two scroll compressors are simulated comparatively to characterize the behavior of the dummy port. The compressors are identical except that the first compressor has a dummy port while the second compressor does not. The compressor
with dummy port is used as the baseline case. The impact of the dummy port is evaluated by analyzing the differences between the baseline compressor and the compressor without the dummy port. Both compressors operate with the same mass flow rate through their system. The inlet temperatures are the same. The inlet flows are turbulent with an intensity of 5%. The compressors are operated at 3500 rpm.

The strategy is to provide the fundamental understanding for the working mechanism of the dummy port first and then define the link between the working mechanism to compressor design parameters. Therefore, the engineering simplifications are limited in the analysis. The simulations are conducted in a general format with no simplification on geometry and operating conditions. After the detailed flow physics are obtained, the result is processed systematically by using averaging algorithms to show the leading order phenomena. The local effects are then obtained in proper post processing techniques. The main objective of the analysis is to build the connection between compressor performance with the fundamental physics. The flows through two compressors are described first through the field quantities. The pressure and velocity are shown to define the overall flow features. To illustrate the gas pockets behaviors, the field quantities such as density, pressure and velocity, are averaged inside the individual pockets and plotted as a function of the crank angle. The life cycle of these gas pockets can be observed through these indicator profiles. After the gas pockets on the two sides of the compressor merge into a single pocket at the center of the compression chamber, the flows are characterized by calculate area-averaged field quantities to illustrate the time-dependent features of the discharge flow. Throughout the process, the two compressors are investigated comparatively. The distinguishable features of the gas flows are defined in a relative fashion. Some surface, volume, and mass averaging algorithms are listed in Table 1.

The data is recorded after the inlet and outlet gas properties show good agreement with experimental observation. For each time step, the basic field quantities are recorded as functions of time and location. The other field quantities can be calculated from these basic field quantities. The statistic quantities can be obtained following the proper averaging procedures. The overall design and performance parameters are obtained by integrating the field quantities over the domain of interest.

### Table 1

<table>
<thead>
<tr>
<th>Volume averaged properties</th>
<th>Area averaged properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \bar{V} = \iiint_V dv )</td>
<td>( A = \iint_A dA )</td>
</tr>
<tr>
<td>( M = \iiint_V \rho dv )</td>
<td>( \dot{m} = \int_A \rho \bar{V} \bullet \bar{n} dA )</td>
</tr>
<tr>
<td>( \bar{\Phi} = \frac{1}{V} \iiint_V \Phi dv )</td>
<td>( \bar{\Psi} = \frac{1}{A} \iint_A \Psi dA )</td>
</tr>
<tr>
<td>( \bar{\Phi}_m = \frac{1}{M} \iiint_V \rho \Phi dv )</td>
<td>( \bar{\Psi}_m = \frac{1}{m} \iint_A \rho \bar{U} \bullet \bar{n} dA )</td>
</tr>
</tbody>
</table>

3 OVERALL FEATURES OF THE FLOW FIELD

The impact of the dummy port on the overall features of the flow field is assessed first at the system level. Since the flow field inside each compressor is connected from inlet to outlet, the changes in the flow field propagate in the compressors. The altered design features in the scroll compressor results in new state of equilibrium when the compressors are operating.
When the mass flow rate is fixed, the pressure, temperature, and velocity distribution adjust themselves to fit in the new environment provided. Figure 2 to Figure 7 show the pressure and velocity distributions on the z-planes. To generalize the results, the field quantities are non-dimensionalized using compressor inlet conditions.

Figure 2 and Figure 3 show the pressure and velocity distributions for the crank angle of 135°. The difference of the two compressors exists in comparison to one another. The overall pressure level is higher in the case without the dummy port. The range of the velocity is also wider for the same case. Since the dummy port provides an additional flow passage downstream, the resistance to flow through the compressor is smaller compared to the compressor without the dummy port. For the same rotating speed of the crankshaft, the higher inlet pressure is required to pump the same amount of gas through the system. As the higher pressure increases the leakage flow for the same flank gap, the more velocity activity is observed (Figure 3) in the compressor without the dummy port.

The second feature observed in the compressor is the asymmetric flow. The asymmetric features of the flow fields are induced by two mechanisms. The first is the geometry-induced asymmetric features. The upper bearing housing casting and the involutes are not designed symmetrically. In this particular design the shape of the casting and the tips of the fixed and orbiting scrolls can be observed in Figure 1 and Figure 2. The second mechanism is the motion–induced asymmetry. Since the orbiting scroll moves around the center of the compressor, the discharge port is not connected to both sides of the gas pockets at the same time. The schedules of the through flow areas for the two sides of the involutes are different. This asymmetric nature of the discharge process induces distortion of the flow field inside the compressor. Figure 3 shows the velocity distortion near the center of the compressor for both cases.

To study the asymmetry of the flow fields inside the scroll compressor, the two sides of the scrolls are designated as side 1 and side 2. There are three gas pockets on each side of the involutes in the scroll compressors. They are labeled as pocket 1_1 through 1_3 and pocket 2_1 and 2_3, respectively. This name convention is kept consistently throughout the current investigation.

Figure 2. Pressure distribution at the crank angle = 135° for (a) with dummy port and (b) without dummy port.
Figure 3. Velocity distribution at the crank angle = 135° for (a) with dummy port and (b) without dummy port.
The distortion is larger in the case without a dummy port. The gas pocket connected first to the discharge port has lower pressure. The other gas pocket that connects to the discharge port later has a higher value of pressure. The two gas pockets also merge with the center pocket with different flow area schedules. It further aggravates the flow field distortion.

The velocity also shows more activities at the center of the compressors. In both cases the asymmetric flow velocity is observed during the merging process. The magnitude of the merge velocity is significantly higher in the compressor without the dummy port. Since there is no passage to allow gas to go under the scroll vanes as in the dummy port case, the gas flow has to squeeze into the center pocket through the gap at the tip of the scroll vanes. The smaller cross-section area of the gaps requires higher speed to allow the compressor to provide the same through flow as in the case with the dummy port.

At the crank angle of 180°, the similar phenomenon can be observed (Figure 4 Figure 5). The overall pressure is still higher in the compressor without dummy port. Asymmetric velocity and pressure patterns exist in both cases. However, the asymmetric activities near the center of the compressor are stronger than in the compressor without the dummy port. Comparing Figure 3 and Figure 5, the irregularities at the center of the compressor are intensified at the crank angle of 180°.

Figure 6 shows the pressure distribution at the center of the scroll compressors when the crank angle equals 180°. An important phenomenon observed here is that the center gas pocket connected to the downstream component does not have the lowest pressure. To illustrate the details of the flow field, only the two gas pockets next to the center pockets are shown. The gas pocket on the side 1 has the lowest pressure. The gas pocket on the side 2 has the highest pressure while the pressure of the center gas pocket has an intermediate pressure value. The merge flow path among the three gas pockets is from side 2 to Side 1 through the center gas pocket. When a scroll compressor is operating, the relative positions of the gas pockets are changing as the orbiting scroll moves around the rotating axis of the crankshaft. It is an inherent feature of the scroll compressors that two sides are asymmetric during the merge and discharge process. This geometric asymmetry induces physical asymmetry in the scroll compressors.

Figure 4. Pressure distribution at the crank angle = 180° for (a) with dummy port and (b) without dummy port.
Figure 5. Velocity distribution at the crank angle = 180° for (a) with and (b) without dummy port.
Figure 6. Pressure distortion at the center of the compressor at the crank angle $= 180^\circ$ for (a) with and (b) without dummy port.

4 CHARACTERISTICS OF GAS POCKETS

Inside the scrolls, the gas pockets change their shapes and volumes continuously. These shape and volume changes and associated pressure and temperature rises are the fundamental working mechanisms of the scroll compressors. The forms of the shape and volume changes control the forms of the pressure and temperature changes. The dummy port changes the flow path at the center of the compressor and impacts the pressure and velocity distributions. These geometry-induced pressure and velocity changes affect the leakage flows and overall mass distribution inside the compressors. To quantify the impact of the dummy port on the behavior of the gas pockets, the total mass inside a gas pocket is plotted as a function of its volume (Figure 7). To generalize the results obtained, the mass inside the gas pockets are non-dimensionalized by the maximum value of the mass inside a pocket. Qualitatively, the overall profile of the total mass inside a gas pocket of the two compressors are similar. The two features that differentiate the two compressors are shown in Figure 7 are maximum mass locations and asymmetry of the mass distribution among the gas pocket on the two sides of the same compressor.

The compressor without a dummy port reaches its maximum mass value earlier at the volume ratio of 0.62 compared to the compressor with the dummy port that reaches its maximum at the volume ratio of 0.52. The pockets also keep the mass at its high value longer. The restriction of the flow path downstream makes the gas accumulate in the pockets and stay there longer.

The second feature shown is the asymmetry of the mass inside the gas pockets on the two opposite sides of the discharge port. Although both compressors have the asymmetry of mass distribution, the one without the dummy port shows a significant larger difference between the two sides.

The volume-averaged pressure history of the gas pockets is shown in Figure 8. The overall value is higher in the compressor without the dummy port. The asymmetric profiles can be observed in both compressors. The basic geometry-induced features are the same in the pressure plots. The discharge angle is at the crank angle of $486^\circ$. The over-compressions are shown in both compressors.
Figure 7. Volume and mass changes in the gas pockets for (a) with (b) without dummy port.

Figure 8. Volume-averaged pressure in the gas pockets for (a) with and (b) without dummy port.

Figure 9. Volume-averaged temperature history of gas pockets for (a) with and (b) without dummy port.

The magnitude of the over-compression is stronger in the compressor without the dummy port. Since the downstream flow passage is more restrictive, the results seem to be a natural consequence of the design. The higher overall pressure value magnifies the asymmetry of the
pressure distribution on two sides of the compressor. The interactions between the mechanisms controlling the compressor performance illustrate the necessities to improve the compressor design in an integrated fashion.

The patterns of the volume-averaged temperature history of the gas pockets are shown in Figure 9. Since the temperature is linked with pressure through the equation of state, the temperature history is similar to the pressure profiles.

5 DISCHARGE PROCESS

The dummy port is a part of the porting system in the scroll compressors. The change on the dummy port has a direct impact on the porting process and its performance. The flow field shows the most complex features during porting in the compressors. On the background of overall changes induced by the dummy port, the local flow field near the dummy and discharge ports experiences drastic changes during the porting process. The changes are both spatial and temporal. Some detailed discussions about the porting process can be found in [2,3]. In this paper, the analysis is focused at the comparison of the two compressors with and without a dummy port. The physical quantities involved in this analysis are velocity, pressure, and mass flow rates. The pressure and velocity components are area-averaged over the open area of the discharge port. The mass flow rate is obtained by integrating the local density and velocity over the open area of the discharge port at the given time instance (Table 1). These quantities then can be calculated for a series of instantaneous values and plotted in Figure 10 through Figure 12.

The two compressors studied have the different velocity profiles at the discharge port (Figure 10). The velocity profiles show dominant 60-Hz fluctuations in both compressors for all three components of the velocity. The difference is mainly at the higher frequency components caused by the details of discharge flow field. The u- and v-components are tangent velocity components. Their profiles are mainly determined by the scanning of the orbiting scroll vane through the discharge port. There is some change on the velocity profiles between two compressors with and without the dummy port. The gas flow inside the discharge port is actually a swirling flow. However, the large-scale features are the same for both cases.

The w-component of the discharge velocity is mainly caused by the pressure difference. When the compressor with dummy port is connected to the discharge port at the crank angle of 486°, the pressure inside the gas pockets equals the pressure in the discharge plenum. There is no additional mass flow going downstream. The w-component of the discharge velocity does not show the distinguishable peak in the velocity profile. When the center gas pocket is connected to the discharge port, the scroll vanes keep compressing the gas inside the pocket, the pressure becomes higher. The normal discharge flow stream is formed. This gas flow downstream generates the only peak in the w-component profile.

For the compressor without the dummy port, the pressure is higher in the gas pocket. When it connects to the discharge port, the flow accelerates downstream. This flow generates the first peak in the w-component profile before the center gas pocket starts to port. The main peaks for both compressors are generated by the center gas pockets. The mechanisms are the same [2].
Figure 10. Area-averaged velocity distribution at discharge port for (a) with and (b) without dummy port.

Figure 11. Area-averaged pressure at discharge port for (a) with and (b) without dummy port.

Figure 12. Mass flow distribution at discharge port for (a) with and (b) without dummy port.

The pressure profiles for both compressors are shown in Figure 11. The two peaks exist in the discharge pressure profiles for both compressors. These two peaks are generated by two gas pockets, the center gas pocket and its neighboring gas pockets on the Side 1. The differences of the pressure values in the two gas pockets are induced by the motion of the orbiting scroll.
The gas in the center pocket is compressed more since it stays in the compression chamber longer.

The discharge mass flow rates for the two compressors are shown in the Figure 12. The mass flow profiles are similar to the w-component of the velocity in Figure 10. The compressor without a dummy port shows the two peaks in the mass rate profile. The first peak is smaller than the second one. This discrepancy between the velocity profiles and mass flow profiles is caused by the pressure increase during the time interval between the two peaks.

6 CONCLUDING REMARKS

Adding a dummy port in a scroll compressor changes the overall performance of the compressor. The entire flow field inside the compressor has to adjust to fit the new feature of the compressor. The dummy port is not a simple local design feature for the scroll compressor in regard of the compressor performance.

The averaged properties of the gas pockets are altered by the dummy port. The dummy port affects both upstream and downstream pockets. The additional flow passage provided by the dummy port makes gas going through compressor easier. The asymmetry of the gas pockets on the two sides of the scroll compressor is reduced. In this particular design, the degree of over-compression is decreased.

The temporal profiles of discharge flow are affected by the dummy port when the discharge port is kept same. The difference can be observed in velocity, pressure and mass flow rate at the discharge port. The impact of the dummy port propagates downstream and is not limited to the local quantities.

ACKNOWLEDGEMENTS

The author would like to thank TRANE Air Conditioning, American Standard Companies, for the permission to publish this paper. Jack Sauls of TRANE Air Conditioning, American Standard Companies, reviewed the manuscript.

REFERENCES


SCREW MACHINES
Clearance management in multifunctional screw machines

Ahmed Kovacevic, Nikola Stosic, Ian K. Smith, Elvedin Mujic
Centre for Positive Displacement Compressor Technology, City University, London, U.K.

ABSTRACT

When expansion and compression are performed together in a single oil free machine of the twin-screw type, changes occur in the high pressure clearances in both the compressor and expander sections, caused by differential expansion between the rotors and the casing. These are more difficult to control than when the two functions are carried out in separate machines. The clearance changes affect both the performance and reliability of the machine but can be controlled by using different materials of construction for each section. Clearances, predicted by the assumption of linear expansion of the components, were included in a well-proven software package for performance estimation of both screw compressors and expanders and the results compared with experimental data. It was found that the clearance in the machine, being dependent on the temperature, could be estimated fairly accurately by matching measured discharge temperature and the temperature obtained by the estimation model. Therefore, a simple expansion analysis of the main machine clearances appeared to be an adequate tool for use in the design of these machines in order to optimise performance and prevent the machine seizing as a result of differential thermal expansion during operation.

NOTATION

R Rotor
H Housing
C Clearance
L Characteristic length
T<sub>m</sub> Machine temperature
T<sub>o</sub> Ambient temperature
β Coefficient of thermal expansion

Subscripts
b Bearing
c Compressor
e Expander
h High pressure side of machine
l Low pressure side of machine
sp Separating plate

© IMechE 2005
1. **INTRODUCTION**

The efficiency of a twin-screw compressor or expander is greatly influenced by leakage within it and, for this reason, all internal clearances must be minimised. Oil injected machines are kept cool by the oil, which also partially blocks the gaps. Thermal expansion of the rotors and their casing is therefore small and the permissible clearance gaps within them are thus determined largely by attainable manufacturing tolerances. However, when such machines operate oil free, due allowance has to be made for differential thermal expansion between the rotors and the casing and hence, the clearances must be much larger. Internal leakage rates are therefore much greater. This is compensated by far lower viscous drag on the rotors, which enables them to run at much higher tip speeds. Consequently, the gas flow rate is much greater and the adverse effect of leakage, which is virtually independent of rotational speed, is thus reduced. Nonetheless, correct design procedures must be followed to minimise the effects of leakage in all cases. These are dependent on a proper understanding of the relative significance of the various leakage paths in a twin-screw machine and how they can be controlled.

The leakage paths in a twin-screw machine, either compressor or expander, are shown in Fig 1. As shown in [1], the most significant is the clearance gap along the sealing line between the rotors. Next is the clearance gap between the rotor high pressure ends and the housing high pressure end wall. These both form direct leakage paths between the working chambers at the highest and lowest pressures. Leakages through the blow hole, the radial clearances between the rotor tips and the end clearance between the rotors and the housing are driven by smaller pressure differences and are therefore less important.

![Figure 1 Leakage pathways in a screw compressor](image)

The size of the clearance gaps is dependent on the dimensions assigned to the rotors and their housing and the fluid temperature rise during the machine operation. These are determined during the design process. However, those at the suction and discharge ends of the machine can be altered during the machine assembly by adjusting the position of the locknuts on the discharge side of the rotor, which determines the position of the axial thrust bearings, located at that end.
A study was previously conducted on a screw compressor with the oil free compression of air at 1 bar and 20°C to a discharge pressure of 3 bar. This included a full 3-D numerical analysis of fluid flow and thermal deformation of the compressor structure for such a machine, by means of Computational Continuum Mechanics (CCM) [1]. It was shown that the air temperature in the compressor discharge port reaches as much as 200°C while at the same time the rotor temperature does not increase above 135°C. Despite large temperature gradients in the fluid, the temperature of the rotors, due to cyclic exposure to high and low temperature regions, appears to be constant across any cross section and decreases uniformly towards the suction end. The deformation of the compressor rotors associated with this is presented in Figure 2. As can be seen from the right hand figure, both rotors grow at the discharge end, where the fluid temperature rises substantially. Therefore, the control of the high pressure end clearance, by fixing the position of axial bearings at rotors and at that end of the casing, makes it virtually independent of any thermal expansion of the machine.

![Figure 2 Temperature distribution and displacement vectors for a dry screw compressor](image)

The authors [4,5] have already reviewed a variety of possibilities for the use of screw machines comprised of a compressor and expander in a common housing. This study is concerned with their use to supply compressed air to a fuel cell, where recovery of power from expansion of the steam formed in the cell, together with the compressed nitrogen in the air supplied, is essential if the overall efficiency of the system is to be acceptable [5].

As shown in [4], dependent on its application, there are a number of rotor and port configurations possible for such a dual function machine. A schematic view of a compressor-expander considered to be most suitable for use with a fuel cell is shown in Figure 3. In this case there are four ports with a separating plate placed between the compressor and expander and the high pressure ports located on either side of the separating plate near the centre of the casing.

This arrangement allows for almost complete balancing of the axial forces and an approximately 20% reduction in the radial forces. By this means, the mechanical losses can be reduced and the radial and axial bearings, located at each end of the machine, can be smaller than in an ordinary screw compressor or expander. However, since the axial forces can change direction because of imbalance in the machine, it is necessary to ensure that axial loads can be sustained in both directions. The arrangement by which the axial bearings are locked in their housings on both sides of the machine is shown in Figure 3. This is opposite to the mode
fixing of axial bearings in ordinary screw compressors or expanders, where the bearing is locked on the rotor.

![Diagram of Compressor-expander for fuel cell application](image)

**Figure 3 Compressor-expander for fuel cell application**

2. **THERMAL EXPANSION OF THE MACHINE ELEMENTS**

When compression and expansion is performed in one unit, the axial bearings must be fixed to both the casing and the rotors at one end, but fixed only to the rotors at the other end, to allow for differential expansion. However, the critical axial clearances of both the expander and the compressor sections are on either side of the separating plate, near the centre of the casing. Therefore differential expansion will change them. This occurs during the machine operation for the following reasons.

Firstly, as the rotors revolve, the temperature of the gas, trapped between them changes, as it is compressed. Thus, at any cross section, the rotors are subjected to cyclic changes in the temperature of the gas with which they are in contact. Because the rate of thermal conduction in the rotors is high, relative to the rate of convective heat transfer between the rotors and the gas, the rotors maintain a uniform average temperature across any cross section. On the other hand, each section of the housing remains in contact with a flow of gas, the temperature of which varies both circumferentially and axially, but which is virtually time independent. The expansion of the casing is therefore not uniform and varies both along its length and around its circumference.

Secondly, the various components of the machine are only rarely made of the same material and each of these may have a different coefficient of thermal expansion.

Both of these effects have to be taken into account in the design of the machine. For this purpose analytical and numerical procedures, based on one-dimensional flow assumptions, as described in [6], were used to predict the effects of thermal expansion on the performance of a combined screw compressor – expander. The results were compared with experimental data obtained from a machine running at different loads and in which the materials of construction were varied.
3. ANALYTICAL MODEL OF THE CLEARANCE ANALYSIS

Figure 3 shows the basic dimensions considered in the calculation of rotor and housing deformation due to temperature changes. The significance of the terms used can be understood by reference to the Notation provided. Using the main dimensions thus described, machine end face clearances can be obtained by summation of these dimensions, after taking account of the initial axial clearances, which have to be set up during the assembly of the compressor. These are: Ccl – compressor low pressure clearance gap; Cch – compressor high pressure clearance gap; Ceh – expander high pressure clearance gap; Cel – expander low pressure clearance gap.

Once these geometrical relations are established it is possible to calculate both axial and radial clearance changes caused by thermal expansion of the rotors and housing in a machine, as shown in Fig 4, since all are obtained from the general relationship, \( \Delta L = \beta \cdot L \cdot (T_m - T_a) \), bearing in mind that the rotors, separating plate and housings may be of differing materials.

![Figure 4 Combined compressor-expander](image)

**Figure 4 Combined compressor-expander**

*Left – Main parts  Right – In the test cell*

Figure 4 shows the second build of a machine, in which the rotors were made of stainless steel, both the compressor and expander housings were made out of aluminium and the separating plate was bronze. In addition, two other combinations of materials were tried. Thus, in the first build, all parts were made of stainless steel. In a third build, the compressor housing was made of grey cast iron, while the remaining materials were the same as in the second built. All three builds are listed in Table 1 and were analysed.

The most important dimensions of the machine are as follows:

- Male rotor diameter 68 mm,
- Centre line distance between the rotor axe 48 mm,
- Compressor relative length 1.1
- Expander relative length 0.8
Table 1 Description of compressor-expander builds

<table>
<thead>
<tr>
<th>Build</th>
<th>Compressor</th>
<th>Separating Plate</th>
<th>Expander</th>
<th>Rotors</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Stainless Steel</td>
<td>Stainless Steel</td>
<td>Stainless Steel</td>
<td>Stainless Steel</td>
</tr>
<tr>
<td>2</td>
<td>Aluminium</td>
<td>Bronze</td>
<td>Aluminium</td>
<td>Stainless Steel</td>
</tr>
<tr>
<td>3</td>
<td>Cast Iron</td>
<td>Bronze</td>
<td>Aluminium</td>
<td>Stainless Steel</td>
</tr>
</tbody>
</table>

The results of the calculation of the changes in axial and radial clearances on the compressor high-pressure side are presented in Figure 5. The combination of a grey cast iron compressor housing, bronze separating plate and aluminium expander housing gives the minimum axial and radial clearance. In this case, both decrease as the discharge temperature increases. Since the discharge temperature is proportional to the discharge pressure this means that for higher discharge pressures all major clearance gaps in the compressor side of the machine reduce and the volumetric efficiency increases. This, in turn, reduces the discharge temperature.

Figure 5 Axial and Radial compressor clearance gaps as the function of temperature

It can be shown that the use of an aluminium compressor housing in build 2, as described in Table 1, will produce the opposite effect. Here, the clearance increase at higher pressures and temperatures is substantial. Thus the axial clearance increases by up to 120 µm for the compressor discharge temperature of 200°C while even the radial clearance increases by up to approximately 100 µm. Since such large clearances increase the internal leakage in the machine significantly, when the compressor housing is made of aluminium, the initial clearance of the machine must be changed. Thus, the axial clearance must then be adjusted to virtually zero at the assembly of the machine. Even then, at a discharge temperature of 220°C, the compressor clearance grows to approximately 120 µm. Hence the machine performance will deteriorate as the discharge pressure increases.

More significantly, this is not the case only when the machine components are of the different material. Even if all the machine components are made of the same material, the clearances still increase as the discharge temperature rises, especially so when aluminium is used for machine components, rather than steel, as shown in Figure 5. However, as can be shown from
this model, only the compressor working cycle and the temperature at the compressor discharge cause deformation of the compressor elements while the expander temperature does not play really a significant role in the deflection of the compressor rotors and housing.

Figure 6 Axial clearances on the expander high-pressure side

For the expander, axial clearances at its high pressure end are affected differently due both to the material used for the separating plate and the expander inlet temperature. As shown in Figure 6, the highest increase in that clearance is experienced for the build 3 in which the compressor housing is made of grey cast iron. Although the compressor becomes more efficient, the expander clearance increases and its performance thus deteriorates. In this case, the expander performance would be improved by making the compressor casing of aluminium, as shown in the top right diagram of Figure 6. However, the expander high-pressure end clearance can become negative if the compressor initial end face clearance is set very low. Therefore, the compressor will still be inefficient and it is then, highly likely that the machine will seize on the expander side.
4. MAPPING THE NUMERICAL AND EXPERIMENTAL RESULTS

Measurements of the working parameters for the combined compressor – expander machine were conducted for two cases, namely; the machine built with both the compressor and expander housing made of aluminium and subsequently with the compressor housing made of cast iron. In both cases, the machine rotors were made of stainless steel and the separating plate of bronze. Figure 7 shows two sets of measurements. The dashed line shows the relative mass flow for all measured rotor speeds when using aluminium casings. It is clear in this case the relative mass flow decreases with temperature equally for all speeds. This is due to the sharp increase in the clearances as the temperature rises. Since higher rotational speeds correspond with higher fluid velocities within the machine, increasing the compressor speed has the effect of increasing the pressure drop in the connecting passage between the compressor discharge port and the expander inlet port. Thus the ratio of expander inlet volume flow to compressor discharge flow increases with speed and this creates an additional restriction to the compressor discharge, which therefore raises the compressor discharge pressure and temperature.

The grey cast iron compressor housing clearances are slightly reduced with increase in temperature. This reduces leakage flow and therefore increases the volumetric efficiency at all compressor speeds, thus reducing the discharge temperatures. Therefore, different trends of the relative mass flows in the function of the discharge compressor temperatures are recorded at different speeds for that compressor-expander arrangement. In both cases, all initial assembly clearances were kept the same.

Since the clearance during the working process could not be measured directly, the influence of discharge temperature on the compressor clearance and performance has been deduced from a combination of measurements and calculations. Firstly, the machine performance was predicted by means of the SCORPATH [6] software package. This is based on a well verified mathematical model developed on the assumptions of quasi-one dimensional flow through the machine and no change in clearances due to thermal distortion. Performance predictions were therefore made for different but invariant sets of axial clearances starting from 40 µm up to 150 µm. These were then matched with the experimental results for cases where the measured and predicted discharge temperatures were equal. The axial clearances were then deduced from this.

There are some limitations to this method, since the estimated machine clearances, which are dependent on flow and temperature distribution that are essentially three-dimensional, are derived from a one-dimensional simulation model. Firstly, thermal expansion requires knowing the temperature distribution along the rotors and housing. This has been assumed to change uniformly and linearly both through the rotor and housing. Such an assumption was made, based on the preliminary 3D flow and solid structure calculation within the combined machine [4]. However, this assumption may not always be accurate, especially during transient processes, such as start-up or shutdown. Additionally, the estimated temperatures of the compressor structural elements take account only of the heat generated and transferred from the working chamber. However, the integral motor and compressor bearings generate heat which is transferred through the rotor shafts and housing. This may also affect the clearances. Therefore, differences between the predicted and measured discharge temperatures may, under some circumstances, be significant.
Figure 7 Experimental results for the second build of the combined compressor-expander

Figure 8 shows the results obtained. In the machine in which the compressor housing is made of aluminium, the axial high pressure clearance gap increases with increase in temperature while for the grey cast iron compressor housing that gap stays almost the same. This confirms the estimates made by the simple analytical method, as shown in Figure 5.
5. CONCLUSIONS

The problems associated with differential expansion in oil free machines of the twin-screw type, in which expansion and compression are performed within a single casing, have been investigated. Estimates of clearance changes, based on simple linear expansion effects showed how these could be controlled to optimise performance while avoiding seizure of the machine. One method is to use different materials for the compressor and expander casing. By assuming the machine clearances derived from this simple analysis, in a software package derived to estimate the compressor and expander performance, the performance predictions, thus obtained, agree well with those obtained from tests on a machine of this type. The use of this simple expansion model as a design tool for combined screw compressor-expander units therefore appears to be valid.

6. REFERENCES


2. Kovacevic, A.; Stosic, N.; Smith, I.K.; “The Influence of Rotor Deflections upon Screw Compressor Performance”, Conference on Screw Type Machines VDI-Schraubenmachinen, Dortmund, Germany, September 2002, 17-28 (b)


Noise prediction in screw compressors

E. Mujic, A. Kovacevic, N. Stosic, I. K. Smith
Centre for Positive Displacement and Compressor Technology, City University, London, U.K.

ABSTRACT

Both established and proposed procedures for the analysis of noise in screw compressors are reviewed and the main conclusions that can be derived from them are given. This includes a general introduction to the acoustic models, which can be used for screw compressor noise calculation and prediction and a review of recently developed software, which can be used to trace the sources of noise. A proposed strategy for further development work to include noise prediction as part of the screw compressor design process is given in outline.

1. INTRODUCTION

Screw compressors are used today in construction engineering, refrigeration and air conditioning, in vehicles, pneumatic transport and in process and food industry. The majority of these applications require low compressor noise levels. Therefore, either the compressor must not generate noise, or the noise generated must be contained within the compressor system. Unfortunately in any compressor system, as shown in Fig 1, part of the energy transfer required to drive it is dissipated during the operational cycle in the form of flow and mechanical disturbances, which cause not only vibrations of the system but also generate pressure waves with a range of frequencies and intensity levels called noise. Pressure waves, that propagate through the air that surrounds the noise source are classified as air-borne. Those transmitted through the working fluid are classified as fluid-borne and those transmitted through the machine components are described as structure-borne. Regardless of the manner in which these disturbance travel through the system, they finally reach a receiver which can either, feel, register or measure them. The level of noise reaching a receiver can be decreased by means of insulation, interruption of the noise propagation path or by trying to eliminate the source of the noise. The compressor designer should be involved in the all of three above mentioned steps in order to achieve noise reduction. This paper considers ways how to reduce noise generation in screw compressors. Since noise generation is implicit in the compressor mode of operation, it is impossible to remove the noise source completely, but some improvements are possible if efforts are directed towards the reduction of disturbances created during the working process. Therefore, the first stage to reducing noise level is to distinguish
between the different mechanisms of source generation and to classify them according to both their significance and the possibility of their reduction. These are as follows:

- Mechanical sources of noise generated as the consequence of periodic contact of moving parts in the screw machine,
- Fluid sources of noise related to gas pulsation in the suction and discharge ports,
- System vibration as a source of noise, which is actually a consequence of the two previously explained mechanisms.

It is also important to determine what other sources of noise are encountered in a typical screw compressor system and to compare their level of generated noise with that arising from actual compressor. To illustrate this, the compressor system shown in Figure 1 has been taken as an example. The main components of the screw compressor system are:

1. The Screw Compressor and Oil Separator,
2. The Drive Motor,
3. The Compressor Drive with Torque meter,
4. The Drive Motor Fan.

Figure 1: Typical screw compressor system

In this case measurements were taken of a compressor with a male rotor of 120 mm diameter and a 4/5 lobe configuration running at 3000 rpm. Figure 2 shows the sources of noise, which cover the frequency spectrum, through the compressor system. By conducting a frequency analysis of the captured sound signal it is possible to separate some of the main noise sources in the compressor system. In this case, these sources are:

1. The Compressor,
2. The Drive Motor,
3. Vibration of the Compressor Drive,
4. The Drive Motor Fan.
The more important comparison is that between the noise generated by the screw compressor and that obtained from the other noise sources. It can clearly be seen from Figure 2 that the screw compressor generates a higher level of noise than the other components. It follows that to reduce the system level of noise, it is most important to concentrate on reduction of noise generation in the screw compressor itself. One possible means of reducing the mechanical source of noise is to optimise the distribution of the torque transfer between the rotors so that contact forces are minimised with no reversal of torque in any rotational position. However, the fluid source of noise is a consequence of the compressor working process and therefore it cannot be removed.

Figure 3 shows a typical sound pressure pulsation frequency spectrum in the suction and discharge ports. Basic signals, which represent measured fluctuations of the gas pressure, have been transformed by use of Fourier transforms from the time domain to the frequency domain. The result, as shown, indicates that the gas fluctuations are higher in the discharge port than in the suction port. Higher gas fluctuations produce a higher sound pressure level on the discharge side. Accordingly, investigation of the gas pulsations in the discharge port is of prime importance from the point of view of noise suppression. Many recent investigations of noise in compressors have concentrated on gas pressure pulsation in the discharge port and the various parameters, which affect it. These, together with the main conclusions, are given in the following section.
2. REVIEW OF PREVIOUS WORK

There have been several studies on compressor noise which describe its sources, the influence of various working and geometrical parameters on it and some mathematical models for noise calculation. The most significant of these are given in refs [1, 2, 10 and 14].

According to Sangfors [10] the main source of noise in screw compressor is gas pulsation caused by the successive opening and closing of the inlet and discharge ports. In his research, Sangfors obtained values of gas pressure pulsations as a function of screw compressor working parameters, geometric characteristics and the physical characteristics of the working fluid. His analytical results showed good agreement with measured data.

Tanntari, [14] conducted similar studies, which generally confirm Sangfors work. This showed that the sound pressure in the discharge port increases as the discharge pressure is raised and as the oil to gas ratio is reduced.

Koaki and Soedel [1, 2] also identified gas pulsation in the suction and discharge ports as the main source of noise in screw compressors. They were concerned with the determination of instantaneous fluid flow rates through the ports, and with the interpretation of unsteady periodic pressure changes within them. The basic fluid flow parameters through the ports were calculated by use of known geometric characteristics and compressor working parameters. Using this data Koaki and Soedel developed both one-dimensional and three-dimensional acoustic models based on the finite element method. Their experience is that by use of a three-dimensional finite element numeric model, more precise results can be obtained. The main reason for the higher precision of the 3D model is its ability to include complex fluid flow geometry in the numerical model and the calculation process.
The following summarises the main conclusions of these studies, which a screw compressor should take into account in the design process.

- In general, the greater the number of rotor lobes in the screw compressor rotors, the lower is the sound pressure level generated [10].
- Decreased leakage increases noise generation at the discharge port, due to increased flow through the screw compressor, but this effect is small [1, 10].
- Changing the wrap angle by 50 degrees will change the discharge noise level by approximately 1 dB. [10].
- The Length/Diameter ratio of the rotors has no practical influence on the generated noise level [10].
- Oil has an attenuating influence on the noise generation process, but this affects only the high-level harmonics, from the 3rd level upwards [1, 10, 14].
- According to Sangfors, sound pressure level, expressed as a function of discharge pressure, has a minimum. Tanntary reported that the minimum is local and weak. Generally, the sound pressure level increases with the discharge pressure.
- The sound pressure level is affected by the working chamber length. This influence is significant and the sound pressure level expressed as a function of chamber length also has minimum [10].
- The pressure difference between the compressor outlet and the discharge chamber has a strong effect on gas pulsation. Generally, a lower the difference reduces the gas pulsation. A negative pressure difference or “underpressure” gives a lower flow pulsation than a positive difference or “overpressure” [1].

3. FUTURE RESEARCH

Previous investigators have developed mathematical models for the estimation of noise generation in screw compressors. However, in order to solve them, many simplifying assumptions were necessary leading to the omission from these models of many of the factors believed to be significant.

As a result of recent developments in numerical methods and computer capabilities, new and more comprehensive models can be used, based on finite volume numerical calculations. By this means, advances have been made in developing acoustic models for other internal and external applications. These new acoustic models use the method of ‘acoustic analogy’ to predict the sound pressure level. This approach is now widely used in aerodynamics and aero acoustics. It is based on first modelling the basic fluid flow parameters such as pressures and densities. These, in turn, are then input to the sound field model [8,15]. Basically, solutions to the conservation equations rearranged in the wave equation form, provide the sound pressure level and the power of a sound source at a distance from the machine. The following methods that have been used to obtain them:

- **Direct methods** solve the compressible Navier-Stokes equations and simultaneously calculate fluid flow and acoustic contributions [8].
- **Hybrid methods** based on decoupling the flow and acoustic parts. The fluid flow is calculated first in order for the acoustic sources to be defined. The far field radiation can then be obtained by the use of calculated source values [8].
In the hybrid approach, a wave equation is obtained for the density (or pressure) fluctuations, with three types of sound sources: monopole source, dipole sources and quadruple sources. Density or pressure fluctuations first have to be obtained from CFD modelling of flow in the compressor. These values, which are derived as functions of time, first have to be transformed into a frequency domain and after that transferred to an acoustic numerical grid. The acoustic model is widely based on a boundary element method, which is the most appropriate for this type of calculation. By this means, the compressor geometry can be simplified and the calculation process thereby made faster.

The most comprehensive model yet developed for the estimation of fluid flow in screw compressors is that of Kovacevic et al [4-7]. This includes both 1-D and 3-D analysis of screw compressors and is contained in a package that includes the following components:

- **SCORPATH** - The software generates basic screw compressor geometry and calculates thermodynamics parameters during cycle [12,13].
- **SCORG** - The software generates numerical grid for fluid and solid domain in screw compressor and prepares all boundary conditions for calculation in numerical finite volume solver [5,6].
- **DISCO** – This software manages all phases of the design and makes 3D CAD model of the screw compressor, which is later together with other input parameters used in CFD calculation [7].

Figure 4 shows schematic view of how it is proposed to incorporate proposed new acoustic studies of screw compressors within this package. The ultimate aim of this will be to attach an acoustic model to DISCO, the software management program so that prediction of screw compressor noise can become part of the design process.

---

**Figure 4: Planned screw compressor design process**
The advantages to this approach are as follows:

- The influence of oil, which was not included in earlier models, can now be included, with the thermodynamic properties already taken in account.
- Changes to the compressor geometry and their effect on sound propagation can be automated. This will speed up the generation of results.
- The model can include the effects of change in the compressor geometry which, though not affecting either the fluid flow field or gas pulsation, can greatly influence noise transmission and emission.

It potential limitations are:

- The accuracy of the results from the acoustic model is dependent on the input data obtained from CFD analysis of fluid flow. A more accurate CFD model can be obtained by better understanding both (CFD and acoustic) problems and using mesh with more cells [15].
- In order to capture an adequate number of pressure fluctuations, the CFD model needs a large number of time steps. This is very important for accuracy [15].
- Both these requirements increase size of the computer resources and the calculation time. It is hoped that continuing progress in computer hardware development will decrease the significance of this limit.
- These two models will need considerable skill by the operator in setting up input parameters for the acoustic model, since uncrical transformation of the CFD generated time dependent variables into acoustic frequent dependent variables on shared parts of grid can product inaccurate results [15].

4. CONCLUSIONS

Studies have already shown that the screw compressor itself is the main source of noise in a complete compressor system. The main process within the screw compressor that causes noise is gas fluctuations in the ports. Gas pulsation in the discharge port generates a higher noise level than gas pulsation in the suction port. Today, as noise suppression is becoming an increasingly important environmental issue, there is a growing need to take account of this during the screw compressor design process. Analytical models are therefore required which can predict noise levels with sufficient accuracy during the design stage. Models for this purpose have already been developed for other applications. However, to implement them in the screw compressor design process it is first necessary to be able to carry out a CFD analysis of fluid flow within the machine. The sources of noise can then be identified from those results and input to an acoustic model, which calculates the sound pressure level around the compressor. Software for CFD analysis of fluid flow in screw compressors has already been developed and it is proposed to use this as the basis of future development of an acoustic model. The aim is then to include this in a complete software package for screw compressor analysis and design.
REFERENCES


Charge changing in screw-type vacuum pumps –
experimental investigation and simulation

K. Kauder, D. Stratmann
FG Fluidenergiemaschinen, Universität Dortmund, Germany

ABSTRACT

Charge variation during both inlet and outlet phases seriously influences the operating behaviour of dry-running positive displacement vacuum pumps.

This report provides results from the experimental investigation of the charge variation in a dry-running screw-type vacuum pump with isochoric processing - no inner compression - working against ambient pressure. Thermodynamic procedure and fluid mechanics conditions can be predicted by pressure indications for the charging and discharging phases.

The main physical characteristics affecting the entire energy conversion of vacuum pumps as well as the final attainable vacuum (inlet) and the extremely impulsively black-flow conditions for the discharging (outlet) are determined and analysed.

Integrating simplifying models for the charging variations enables the simulation software KaSim to simulate the whole working cycle of positive displacement vacuum pumps in rough vacuum conditions. Comparison of simulated and measured machine characteristics (pressure curves, attainable final pressure and suction speed) provides a good agreement by experiment and simulation.

1 INTRODUCTION

Basically the operational area of vacuum pumps is specified by suction speed, achievable final pressure and maximum gas temperature while operating with chemical reacting gases. There are multiple machine types for the production of vacuum, whereby positive displacement vacuum pumps are the most important and widely-used type (1). Applications for these machines are mostly limited to rough (pressure $p = 1$ to 1000mbar) and fine vacuum (pressure $p = 10^{-3}$ to 1mbar). Higher standards in vacuum cleanliness, especially in the computer chip industry, have led to the banning of cooling and sealing fluids inside the working chamber and an increasing demand for and development of dry-running vacuum pumps. The absence of service liquids leads to unfavourable operating behaviour in dry-running pumps, resulting in significant thermal loading of machine components so that operating reliability is at risk, or there is a reduction in suction speed and final achievable pressure.
In connection with the main objective of simulating the operating behaviour of dry-running positive displacement vacuum pumps, the simulation software *KaSim* has been developed in recent years (2). Based on the chamber model of the machine, the simulation program calculates simultaneously the state of working fluids inside the chambers taking into account interactive mass and energy flows through chamber connections (clearances, other chamber connecting elements) for given angles of rotation and for the whole working cycle. It is assumed that the state of the working fluids inside the chambers is approximately homogeneous.

In previous studies the special conditions of clearance flows in vacuum has been researched (3) and an algorithm has been derived to quantify the clearance mass flow for continuum flow, Knudsen flow and molecular flow conditions (4). The implementation of the calculation model for clearance rates enables *KaSim* to simulate the thermodynamic process of the transportation phase.

The aim of this study is to investigate the physical mechanism of the charge variation for both inlet and outlet in dry-running positive displacement vacuum pumps and to analyse their influence on the main integral operating behaviour. For these purposes a screw-type vacuum pump with isochoric processing was experimentally examined (Fig. 1). The description of the charge variation in simplifying models then enables the simulation software *KaSim* to calculate the whole working cycle of positive displacement machines.

---

**Fig. 1: Model of a screw-type vacuum pump**

A,B rotors
C casing
LP low pressure side / inlet port
HP high pressure side / outlet port
1 working chamber inside the charging phase
2 working chamber inside the transportation phase
3 working chamber inside the discharging phase

*The test machine is a prototype screw-type vacuum pump of the company *Sterling SIHI.*

---

2 EXPERIMENTAL INVESTIGATION

2.1 Experimental set-up

The experimental investigation of the charge variation during both, the inlet and the outlet phase was carried out for an air suction, dry-running, isochoric - no inner compression, double-
threaded screw-type vacuum pump working against ambient pressure and consisting of two symmetrical rotors with rather low pitch and consequently a large wrap angle. The test machine is a prototype model-screw-type vacuum pump of the company Sterling SIHI. In contrast to series machines of this type the test machine does not an inner compression.

Pressure indications are used to acquire the transient, time-dependent as well as local pressure curves inside the working chamber during the charging and discharging processes. Measurements of the flow rate as well as inlet and outlet fluid conditions (pressure and temperature) of the working fluid allow the analyses of influence of the unsteady charge variation on the integral operating behaviour of the test machine. The test rig and the diagram of the experimental set-up are shown in Fig. 2.

Due to the rotor geometry and the solely axial-orientated inlet and outlet areas the charge variation of the test machine is extended to a theoretical angle of rotation of \( \phi = 450^\circ \). The pressure gauges are fixed positioned inside the rotor casing and distributed along the

---

Fig 2:  
\begin{itemize}  
  \item a Test rig  
  \item b Schematic experiment set-up  
  \item c Volume curve of the working chamber and position of pressure gauges inside the working cycle  
\end{itemize}
extension of the working chamber. The ascertainable scope of rotation for each pressure is limited to an angle of rotation of \( \Delta \phi = 70^\circ \) as a result of the size of the pressure-sensitive area and the height of the working chamber. For this range the sensor diaphragm lies completely within the working chamber that passes the fixed pressure gauge and measures the operating pressure inside the working chamber.

### 2.2 The test procedure

Within the experimental investigations a significant temperature dependence of the operating behaviour of the test machine was detected which is typical for this machine type. Shortly after the vacuum pump is started the operating behaviour becomes unsteady due to the rapidly increasing thermal load at the high pressure side. This is directly associated with modified clearance geometries and changing leakage mass flows as a result of thermal stressing of the machine components.

To gain directly comparable results for known and unchanged boundary conditions (clearances, temperatures of machine components), measurements were acquired for ‘cold’ operating machine conditions. To ensure cold machine conditions any operating point determined by revolution speed \( n \) and inlet pressure \( p_{in} \) was selected after a short period of time (less than 1 minute), while long cooling-down intervals between two measurements had been kept.

### 2.3 Charging

The results of the pressure indications show a characteristic pressure progression during the charging phase, shown for two operating points in Fig. 3. The partially discontinuous pressure curves appears as the result of variable accuracy of the pressure gauges and different measurement positions inside the working chamber.

The predominant charging process for rough vacuum inlet pressures indicates no significant differences between operating pressure inside the working chamber and intake pressure. Consequently, throttle influences during charging as well as gas dynamic mechanisms (gas inertia) which may be expected to cause a pressure drop inside the working chamber, are minor or negligible and are due to the comparative size of the axial inlet areas and the moderate speed of volume extension of the working chamber. Here an almost ideal chamber filling is anticipated and the influence of back-flowing leakage masses on the chamber filling can also be given low rating. On the one hand this results from the pressure levels of other working chambers in the cycle, partly also just from charging (via the casing gap\(^1\)). On the other hand leakage mass flows through other gaps (radial gap, profile meshing gap, blow hole) are expected to be minute because of the explicitly lower gap areas.

A characteristic rise in pressure is detected for the charging end typically just before real chamber closure. This sudden rise in pressure is observed at the point in time when the end of the rotor profile passes the edge of the rotor bore in the casing (rotor angle \( \phi \approx -35^\circ \)\(^2\)). The pressure rise results from a back-flow of leakage masses through clearances from working chambers ahead of the chamber being examined. These masses flow into the working chamber being charged and affect the filling process. The working chamber ahead is connected to the chamber under consideration via the casing gap, which closes at a rotation angle of

---

\(^1\) The casing gap connects two working chambers on one rotor with a difference in rotation angle of \( \Delta \phi = 180^\circ \).

\(^2\) Rotation angles are based on taking 0\(^\circ\) to signify the transition from the charging to the transportation phase (inlet) or the transition from the transportation phase to discharge phase (outlet).
ϕ = -180°, showing the same pressure development as previously. Consequently, increasing leakage mass flows into the charging working chamber are expected from this point on.

Fig. 3 Pressure progression (time-dependent) and local pressure distribution during the charging process and the transition to the transport phase, radial positions of pressure gauges to acquire local pressure gradients and selected geometric machine parameters

Annotation: 0°-angle position signifies the charging end (chamber closure)

- p: operating pressure
- 1-3: positioning of pressure gauges along the rotor circumference (rotor angle ϕ = -35° to 35°)
- AK: extension of the working chamber
- V: chamber volume
- \( V_{\text{max}} \): maximum chamber volume

Back-flowing leakage masses do not directly cause a rise in pressure inside the charging working chamber. Instead a pressure-equalizing flow from the charging working chamber into the inlet port may be assumed. The time-dependent and local pressure development inside the working chamber, as shown by signals 1-3 in Fig. 3, indicate this direction of flow as the pressure rise starts before the chamber closure, first near the inlet area and then in the middle and finally at the end of the working chamber, reaching pressure levels above the inlet pressure. After this the pressure distribution becomes homogeneous in the working chamber. Due to the reduction in the inlet area the backflow is increasingly affected and blocked so that the rise in pressure spreads across the working chamber. Depending on the operating point, the rise in pressure at the inlet area inside the working chamber is above of the subsequent homogeneous pressure level in the closed chamber. Such pressure development is followed by a temporary decrease in pressure.
The characteristic pressure development for the transition from the charging to the transportation phase is closely linked to the integral operating behaviour of the vacuum pump, dependent on suction speed as well as operating efficiency. The more the pressure increases for the transition to the transport phase, the more the delivery rate decreases, Fig. 4. For constant inlet pressure \(p_{in} = 100\text{mbar}\) an increase in rotor speed reduces the back-flow, resulting in a less strong rise in operating pressure (Fig. 4, left). The reduction in back-flow is caused by shortened operating cycles, which creates lower leakage mass flows. Furthermore, with increasing rotor speeds the pressure gradient inside the transportation phase moves in the direction of the discharge so that pressure ratios for clearances are minimized for the area near the inlet, which also results in lower leakage mass flows.

For a constant rotor speed of \(n = 4000\text{min}^{-1}\) and decreasing inlet pressures (Fig. 4, right), the delivery rate drops. This is again linked to the pressure development at the end of the charging phase, and is caused by increasing leakage mass flows through clearances. Increasing pressure ratios within the machine result in increasing leakage flows at the suction port area, although the throttle characteristic of clearances increases significantly at lower pressures (3).

2.4 Discharging

The discharging process of dry-running positive displacement vacuum pumps against ambient pressure is determined by the operating pressure inside the working chamber at the moment of opening. The time-dependent pressure development for the end of the transportation phase and the discharge is clarified in Fig. 5 for two operating points.

The operating pressure of isochoric processing vacuum pumps increases during the transportation phase to a level above the inlet pressure because of the leakage mass flows through the clearances necessary for successful operation, but it does not normally attain the ambient pressure. For this reason, during the transition from the transportation to the discharge phase, a pressure-equalizing unsteady back-flow of gas from the exhaust port into the opening discharging working chamber can be detected. Pressure equalization between the working
The local pressure change during the discharge process is analysed by several sensors along the chamber extension measuring simultaneously (Fig. 6). At the moment when the chamber opens the operating pressure inside the working chamber is clearly below the ambient pressure. The increase in pressure inside the working chamber extension starts from the outlet port side and...
develops along the chamber length. The pressure gradients are similar for each sensor with a delay due to their positioning. The time interval indicates extremely high back-flow speeds against the intended delivery direction.

Fig. 6 Pressure indications of the discharging – pressure progression as a function of the rotor angle for the commencement of the discharge (rotor speed \( n = 6000 \text{ min}^{-1} \); inlet pressure \( p_{\text{in}} = 1 \text{ mbar} \))

- \( p_\text{at} \): ambient pressure / exhaust pressure
- \( A_\text{E} \): axial outlet area
- 1-6: radial positioning of pressure sensors

### 3 SIMULATION

#### 3.1 Modelling of charging and discharging process

The charging phase is modelled by means of a pressure-balancing connection, so that for every time-step of the calculation, balanced conditions between the charging working chamber and the inlet are obtained. This arrangement is based on the assumption that the effects of incomplete charging due to the inlet throttle mechanism and gas inertia for rough vacuum inlet pressures (\( p_{\text{in}} \) between 1 to 1000 mbar) are not dominant. In any case, the influence of back-flow leakages into the charging working chamber from those ahead of it and also into the inlet port from the chamber at present under suction are taken into account. So leakage mass flow is the only effect dropping the ideal volumetric efficiency:

\[
\lambda_L = \frac{m_{\text{in}}}{m_{\text{in}}} = 1 - \frac{m_{\text{sp}}}{m_{\text{in}}}
\]

\( \lambda_L \): volumetric efficiency (delivery rate)
\( m_{\text{in}} \): intake mass (sucking mass)
\( m_{\text{th}} \): theoretical intake mass
\( m_{\text{sp}} \): leakage mass flow

The discharge phase is modelled by a simplifying throttle approach, the flow coefficient is given by \( \alpha = 1 \):

\[
\dot{m} = \alpha \cdot A \left( \frac{p_1}{p_0} \right)^{\frac{1}{\kappa}} \cdot \frac{p_0}{R T_0} \cdot \sqrt{\frac{2 \kappa}{\kappa - 1}} \frac{R T_0}{1 - \left( \frac{p_{1,\theta}}{p_0} \right)^{\frac{1}{\kappa - 1}}} 
\]

\( \dot{m} \): mass flow
\( \alpha \): flow coefficient (\( m/m_{\text{th, max}} \))
\( p_{1,\theta} \): pressure
\( R \): gas coefficient
\( T \): gas temperature
\( \kappa \): isentropic exponent
3.2 Boundary conditions
All machine gaps are modelled inside the simulations. According to the measurements acquired for ‘cold’ operating conditions of the vacuum pumps, constant clearance heights are assumed for the simulation. Machine components and the air at the inlet are set to a temperature of 293K as in the experiment. Complete heat transfer conditions are assumed, which is insignificant for the computations however. For any simulated operating point the inlet and outlet pressure are set as constant boundary conditions.

3.3 Simulation results
The calculation results are presented in Fig. 7. For constant inlet pressures the volumetric efficiency of the vacuum pump increases with increasing rotor speed, as expected. For lower rotation speed ranges $n < 4000 \text{min}^{-1}$ the suction speed as well as volumetric efficiency drops drastically at lower inlet pressures. For higher rotation speed ranges $n > 5000 \text{min}^{-1}$ and intake pressures lower than $p_{\text{in}} < 100 \text{mbar}$ this state of affairs is reserved. So decreasing inlet pressures result in higher volumetric efficiency, characteristics which approach those of an ideal machine. For a rotor speed of $n = 4000 \text{min}^{-1}$ and an inlet pressure of $p_{\text{in}} = 10 \text{mbar}$ a small increase in rotor speed causes a significant rise in the delivery rate due to the movement of back-flowing leakage in the direction of the exhaust port.

![Fig. 7 Simulated machine characteristics – volumetric as a function of rotor speed n (parameter : inlet pressure $p_{\text{in}}$; constant clearance heights)](image)

4 VERIFICATION
A quite good agreement is obtained by experiment and simulation for the analysed screw-type vacuum pump for rough vacuum conditions ($p = 1 \text{mbar}$ to $100 \text{mbar}$). The interrelationships which actually occur during operation are reproduced by the simulation. Acquired operating pressures during the transportation phase, dynamic pressure measurements for the discharge phase, final pressures attained at the inlet as well as suction speed (or volumetric efficiency $\lambda_L$, Fig. 8) all show acceptable deviations. This agreement disappears at decreasing intake pressures or higher rotor speeds (Fig. 8, $p_{\text{in}} < 10 \text{mbar}$). A possible reason for the deviation is the increasing influence of relative velocity of gap boundaries, which causes modified flow characteristics at the clearances. As simulated machine efficiency exceeds measured efficiency
significantly here, and is similar to that of an ideal machine, other external leakages or virtual leakages could be predominant in the experiment.

![Graph showing comparison of simulated and measured machine characteristics](image)

**Fig. 8** Comparison of the simulated and measured machine characteristics for ‘cold’ operation conditions – delivery rate $\lambda_L$ as a function of inlet pressure $p_{in}$ (parameters: rotor speed, constant clearance heights)

5 CONCLUSIONS

**Charging process:**

The experiment indicates that leakage mass flow into the charging working chamber results in a pressure rise inside it just before real chamber closure closely linked to the volumetric efficiency of the vacuum pump. Also backflow out of the charging working chamber into the intake port can be assumed by the pressure indications.

As the simulation only includes decreasing effects of back-flow leakage on the delivery rate and a good degree of agreement can nevertheless be claimed, this seems to be the predominant influence on the operating efficiency and suction speed for rough vacuum ($p_{in} = 1$ to 1000mbar).

**Discharging process:**

For machines with isochoric processing the discharging phase is determined by the operating pressure of the working chamber for the moment starting the discharging. As the pressure level is commonly below the ambient pressure, massive pressure-equalizing back-flow into the discharging working chamber is detected.
6 REFERENCES


Three-dimensional curvature analysis on screw rotor and its applications

H. Kameya  
Hitachi Limited, Japan

M. Aoki  
Hitachi Industrial Equipment Systems Co., Ltd., Japan

S. Nozawa  
Hitachi Air Conditioning Systems Co., Ltd., Japan

ABSTRACT

Three-dimensional differential geometry helps us to know principle curvatures and a normal curvature perpendicular to a sealing line on a screw rotor surface. We propose an analyzing method, which is calculating an approximate arc instead of a rotor profile. The curvatures are applied for evaluating an internal leak flow, a contact stress between rotors and a maximum diameter of a sphere contact probe of a coordinate measuring machine. By these evaluations we can progress accurate compressing simulation for efficiency prediction and keep enough reliability on surface strength.

1. INTRODUCTION

Male and female screw rotors shown in Fig. 1 are meshed and rotated synchronously. They contact each other on a continuous sealing line theoretically, however there is a clearance between the rotors. Even if a compressor is oil-injected type, there is a clearance on a transverse side of its lobe.

Each normal curvature radius is on a plane perpendicular to the sealing line at each point shown in Fig. 1(c). As its inverse number, normal curvature perpendicular to the sealing line is important and has some applications. For example, the curvature is as important as clearance width between rotors, because they affect leak flow as a fluid friction. In an oil injected machine, screw rotors contact directly for rotation transmitting. Therefore, at a contact point on the rotors should be evaluated to how much stress, in comparison with the rotor material fatigue strength. A calculation of Hertz contact pressure needs surface curvatures, which cross at the contact point (1).

Another application is decision of a sphere contact probe of a three dimensional coordinate measuring machine. A male rotor has sharp grooves between lobes and the grooves have minimum curvature radius. When the diameter of the probe is greater than the minimum curvature radius, we cannot measure the bottom coordinates by interference.

For replying these demands, we propose a method to calculate three-dimensional curvatures on screw rotors and show some applications.
2. CURVATURE ANALYSIS

Screw rotor profile is defined with several curves on a transverse section in general. Each curve is arc, parabola or generated line by the partner rotor or the rack. At every boundary,
the curves connect smoothly. A rotor surface consists of the envelope of the profile moving along its helix. Therefore some equations are needed to indicate a coordinate of a surface point and it is difficult to analyze these equations directly.

We propose a curvature analyzing method with an approximate arc instead of the profile itself. Every point on the profile can be approximated to an arc as a second order tangent on a transverse section. If this arc moves along the rotor’s helix, it makes the envelope shown in Fig. 2. On a transverse section, the normal line of a surface point crosses the pitch point, and the surface point meets partner rotor’s surface. We can calculate three dimensional curvatures including normal curvatures and principal curvatures at this surface point (2). The curvatures are same as the original profile.

We show this method with an existing profile (2) which has marked points a - e on the female and f - j on the male rotor shown in Fig. 1. The results are shown in Fig. 3 as non-dimensional numbers. The horizontal axis means a position on a sealing line which origin is a point h a top of the male rotor, and point c a bottom of the female rotor. Its positive number means leading side to point j or e, and its negative number means trailing side to point f or a.

The male and female approximate arc curvatures $\kappa_{tm}$ and $\kappa_{tf}$ on a transverse section are shown in Fig. 3(a). For the meshing to be clear, the vertical female axis is upsidedown to male axis. From this graph (a), the normal curvatures perpendicular the sealing line $\kappa_{nm}$ and $\kappa_{nf}$ shown in (b), and the male rotor’s principal curvatures $\kappa_{pm1}$ and $\kappa_{pm2}$ shown in (c).

3. INTERNAL LEAK FLOW

As an internal leak flow is supposed to cross the sealing line at right angle at every point on it, we made a two-dimensional model on a plane perpendicular to the sealing line shown in Fig. 4. The male and female rotor’s normal curvatures $\kappa_{nm}$ and $\kappa_{nf}$ are compounded to an equivalent curvature $\kappa_e$ (shown in Fig. 5 (a)) for to be simple without centrifugal force.

We made a supposition that the clearance between rotors is $C = 0.3*10^{-3}$ uniformly shown in Fig. 5 (b). It means 50$\mu$m clearance of 150mm diameter male rotor. When the supposed compressor is oil-injected type, there is a contact point at d and i like curve C’.

Internal leak flows per unit length along sealing line of this model is calculated by CFD (Computing Fluid Dynamics). The boundary conditions are 0.8 and 0.1MPa pressure, and the fluid is oil which kinematic viscosity is 100mm$^2$/s. These conditions make the leak flow subsonic.

As a result, the flow late is in proportion to the clearance C to the 2.5th power and a square root of the equivalent curvature radius $R_e$ approximately. Using this result, the leak flow is calculated and shown in Fig. 5(c).

As a total leak flow of one meshing length is an integral of each point leak flow on the graph (c), the area under the graph represents total leak flow. We can find out the leak of the trailing side is 3.5 times of the leading one. If the compressor is oil-injected type, the difference will be widened.
Fig. 3  Curvatures on male and female rotor
Fig. 4 Model of leak flow at sealing line and equivalent model

Fig. 5 Simulation result of internal leak flow between rotors
4. CONTACT STRESS

In order to calculate elliptical Hertz contact pressure, we need some inputs shown in Fig. 6. They are a curvature radius $R_1$ along to the sealing line, curvature radii $R_2$, $R_3$ perpendicular to the sealing line, a material properties, and normal load. The reason why there are two radii perpendicular to the sealing line is the contact point d/i is an inflection point shown in Fig. 7. Therefore the contact pressure is calculated on the c/h and e/j sides separately. From each side curvatures, the equivalent curvature radii are calculated. Otherwise the curvature radius $R_1$ along to the sealing line is approximated with the clearance line shown in Fig. 5 (d). The material properties (Young’s modulus and Poisson’s ratio) are shown in Table 1. The normal load is calculated from a transmission torque between the male and female rotors.

The modulation of these functions gives the normal load and contact pressure graphs, which have a common horizontal axis as in the approaching deformations in Fig. 8. The normal loads are indicated as half the calculated values because each line means a half field contact. By summing these lines, the composed line becomes an integrated relationship between the normal load and the approaching deformation.

Using this composed line, the given normal load 85 N indicates a deformation of 0.8 mm. Then this deformation means a contact pressure of 0.30 [S MPa] at the c/h side and 0.57 at the e/j side, as in Fig. 8 (b). The larger pressure 0.57 is important as an evaluation of the strength. These contact pressures are indicated as non-dimensional values comparing with the weakest material’s strength $S$.

The contact fatigue strength database of several materials that are used as power transmission gears (5) is available. The contact condition of gear surfaces is similar to screw rotors for contact pressure, rotational speed, slipping ratio, friction, lubrication, and materials. We think this database can be used to evaluate the screw rotor strength. We picked up some data on the materials from which the screw rotors can be made as shown in Fig. 9. They are different from each other according to their ingredients and processing. The weakest strength $S$ of material C is used as a base of this comparison.

The screw rotor’s contact pressure 0.57 [S MPa] is less than the material strength of 1.0 [S MPa]. Therefore, this comparison suggests the reliability of the rotor surface is sufficient.

5. MINIMUM CURVATURE RADIUS

The minimum curvature radius on the male rotor surface is shown in Fig. 10. The cross section including the minimum curvature radius is independent to the sealing line. The three-dimensional curvature analysis can be applied to this problem. The principal curvatures mean maximum and minimum curvatures among all direction curvatures on a point. In Fig. 3 (c), the minimum principal curvature $\kappa_{p2} = -207$ means the minimum concave radius on the male rotor surface. If the male rotor diameter is 150mm, the minimum radius is 0.725mm, and we should use more than 1.45mm diameter sphere contact probe for coordinate measuring.
Table 1 Properties of Exsample material

<table>
<thead>
<tr>
<th>Name</th>
<th>Ductile cast iron</th>
</tr>
</thead>
<tbody>
<tr>
<td>Youngs modulus</td>
<td>$E = 1.6 \times 10^{11}$ Pa</td>
</tr>
<tr>
<td>Poissons ratio</td>
<td>$\nu = 0.3$</td>
</tr>
</tbody>
</table>

Fig. 9 Contact pressure comparison with fatigue strength of material
6. CONCLUSION

We propose a three-dimensional curvature analysis method for screw rotor surface using an approximate arc on each point of its profile. The curvatures can be applied for the next examples.
(a) Internal leak flow can be evaluated on each point of the sealing line. The leak of the trailing side is more than leading side.
(b) Contact stress between the male and female rotors is calculated with Hertz contact theory.
(c) Minimum concave curvature radius on the male rotor can be calculated for a contact probe selection of a coordinate measuring machine.

REFERENCES

(2) Kameya, H., Ishii, E., Nozawa, S., Aoki, M.,
  Analysis of Rotor Contact of Screw Compressor,
(3) Fujiwara, M. Nakamura, Y., Takeda, F., Torigoe, T. and Nakagawa, K.,
  Reduction of Partial-Load Rattling Noise of an Oil-Injected Screw Compressor,
(4) Kameya, H., Aoki, M., Nozawa, S.,
  Analysis of Rotor Contact for Screw Compressor,
(5) Kishi, M., Hashimoto, M., Honobe, M., Contact Fatigue Strength of Several Material Gears,
About transient torque impacts on turbo-compressor shafting driven by induction motors or synchronous motors

Ulrich Kern, Michael Gamm
Dept. Mechanics/KMS, Atlas Copco Energas GmbH, Germany

ABSTRACT

This paper deals with the calculation of the torsional stress in the shafting of turbo-compressor drives in the MW range. The compressors may be coupled with cage induction motors or synchronous motors, as well as cylindrical rotor and salient pole machines. The shafting may be endangered when during start-up the actual speed meets one of the torsional critical frequencies. This report is restricted to direct-on-line (DOL) starting, however the calculation tools which are explained can also be used for starting via frequency converter or by use of starting aids like auto transformer, soft starter, star-delta-connection etc.

Besides starting, other electric transients like short-circuiting, supply interruption or system transfer are to be taken into account, because such incidents may result in excessive peak torques.

Two methods of computation are presented in this paper and their results are shown for typical examples of drive systems and for all transients of practical importance.

More than that, the paper depicts how to benefit from the combination of both methods in the field of torsional analysis.

1 INTRODUCTION

Cage induction and synchronous motors are commonly used in applications with turbo-compressors. Drives, in which the motor is supplied by a frequency converter, are not covered by this paper. Therefore, both kinds of drives are started asynchronously, which means, that the rotor windings of the motor are short-circuited during running up. The starting process ends, when the so-called asynchronous torque of the motor is identical with the counter torque of the compressor. The respective speed is less than the synchronous speed of the motor. In case of an induction motor, this speed is identical with the operational speed, in case of a synchronous motor the drive must be synchronized by switching D.C. voltage on the field winding of the motor; so the speed at steady-state operation is synchronous speed.

The advantage of the use of a synchronous motor is its capability to deliver reactive power when it is operated in an over-excited mode.
The disadvantage of a synchronous machine is caused by the asymmetry of the rotor. The rotor is equipped with a so-called damper winding which can be considered as equivalent to the cage of an induction motor and, in addition with the field winding, which excites the rotor field during operation as synchronous machine. During asynchronous starting the field winding must be short-circuited, either directly or via a resistance, in order to support the effects of the damper winding. The exciter winding acts like a single-phase winding which causes the asymmetry. Thus, a synchronous motor develops pulsating torques of twice slip frequency during starting, which ranges from two times line frequency at standstill to nearly zero close to synchronous speed. Consequently the torsional critical speed is excited at a specific speed by the pulsating torque of twice line frequency; this effect does not exist in case of induction motors.

Two computation methods are presented in this paper and illustrated by typical examples:

• Method 1:
  By use of a software to solve the equations of motion of the shafting which sophistically covers all elements including its mass of inertia, the linear or non-linear spring constants, the damping coefficients etc.. The relevant transient is initiated by the injection of an analytical function of the electromagnetic torque of the motor. However, an analytical equation of the time dependence of the transient motor torque can be given for constant speed only, e.g. after switching on a motor with locked rotor. The use of this time characteristic for the starting period with varying speed obviously results in a rough approximation of the truth.

• Method 2:
  By use of software, in which the voltage equations of the motor and the equations of motion of the shafting are solved simultaneously. Since this software is developed by electrical engineers, a simplification of the mechanical model is often unavoidable.

It is obvious that only the last mentioned procedure is in harmony with the physical truth, because it comprises the feed-back between the mechanical and electrical quantities. However, both methods can be combined. It can be shown, that the characteristics of elements at the far end of the compressor shaft are of little impact on the electromagnetic torque of the motor. Therefore, when the shafting is reduced to a two-mass or three-mass model, which is in accordance with the counter torque and the total mass of inertia of the real configuration and which meets the first respectively the two first torsional critical speeds, the use of method 2 results in an excellent approximation of the true time dependence of the motor torque. In the second stage of the calculation, the numerically stored time characteristic of the motor torque will be injected into the differential equations of method 1 in order to get exact figures of the torques and the angular velocity at all locations of the shafting.

2 THE MODEL OF THE MECHANICAL SYSTEM

The upper part of figure 1 shows a typical installation, consisting of the motor at the left side, the turbo-compressor on the right with a metal-elastic coupling between the two components. The bull gear of the compressor is connected to the different impeller branches via pinions. At the bottom of figure 1, this real configuration is segregated into several concentrated inertias and massless spring elements. The springs are characterized by the linear or non-linear spring constant and the damping coefficients. The damping torque is usually assumed to be proportional to the angular velocity.
3 THE EQUATIONS OF THE MOTOR TORQUE TO BE USED FOR METHOD 1

Powerful software for the modeling of the shafting is available already since the eighties. At that time, no method was customary how to link the electromagnetic and the mechanical transients at its interface i.e. the motor torque. Therefore, the motor manufacturers were asked to deliver equations for the time dependence of the motor torque. Those equations, which can be requested from the motor manufacturers up to now, are listed in figure 2. However, such equations represent rough approximations in many aspects. Analytical expressions of the motor torque can be derived for constant speed only. Amongst a lot of other neglects, the current displacement in the cage of induction motors respectively in the damper winding of synchronous motors is not covered by the equations of figure 2.

In principle, the resulting torque is composed of so-called asynchronous torques, which are constant or decay with time, and superimposed decaying pulsating torques.

In case of induction motors (IM), the pulsating torques are of line frequency for all symmetrical switching operations (e.g. starting, three-phase short-circuiting, system transfer). Two-phase short-circuiting produces additional pulsating torques of twice-line frequency. The decaying characteristics can be separated into one part with a small time constant of the leakage fields and a longer time constant of the main field.

I refrain from a detailed explanation of the more complicated equations of synchronous motors (SM) and would like to mention only, that the complexity is caused by the single-phase character of the rotor windings, especially of the field winding but depending on the kind of rotor also of the damper winding.

4 THE MOTOR MODEL TO BE USED FOR METHOD 2

Just to illustrate the fundamental procedure, figure 3 shows the equivalent circuit of a three-phase cage induction motor during transients. It is sufficient to consider a single line diagram. The ladder network on the right allows the simulation of the current displacement in the cage.

It is beneficial to use the method of instantaneous symmetrical components for the calculation. The bold-typed capital letters $U$, $I$ associated by an inverted comma, represent the complex positive sequence symmetrical component of the relevant quantity. By this way, the voltage equations for stator and rotor can be stated. These two equations contain three unknown quantities, namely the currents $I_1^*, I_2^*$ in stator and rotor and the speed $n$ respectively the angular velocity $\omega_m = 2 \pi n$. Consequently, the equation of motion of the mechanical shafting must be taken into account too. The equations are linked by the electromagnetic torque, which is proportional to the product of the two currents and contains the speed in the exponent of the formula. Hence, the system of differential equations can be solved numerically only.

An equivalent procedure is possible for drives with synchronous motors. However, at least three voltage equations are necessary, because the rotor contains two windings, and frequently different permeances in the so-called direct and the quadrature axis must be taken into account.
5 PRESENTATION OF COMPUTATIONAL RESULTS

Atlas Copco has been using a software package since 1988 serving method 1. Shortly after 1990, Atlas Copco acquired a license on the software DYN for induction motors and SVPDYN for synchronous motors to be applied in all cases when method 2 is used.

The combination of methods 1 and 2 is not demonstrated in the following examples, because the calculation results cannot be distinguished from those of method 2.

In the following diagrams, method 1 is called “approximate numerical calculation” and abbreviated by ANC. Method 2 is called “precise numerical analysis” and abbreviated PNC.

6 START-UP OF A TURBO-COMPRESSOR DRIVEN BY A THREE-PHASE CAGE INDUCTION MOTOR

Figure 4 shows some of the computation results. The motor behind the plotted curves is a deep bar rotor, the most common rotor type of high-voltage cage induction motors. However, DYN permits the calculation of all slot geometries including double-cage rotors.

Already the comparison between the calculation results of the motor torque by the two methods shows the extreme deficiencies of method 1: It results in a high peak torque directly at the instant of switching on the motor, whereas in reality the envelope curve of the pulsating torques increases with a small time constant to a maximum value which is much lesser than calculated by method 1. In addition, the red envelope curve decays with a too high time constant. The break-down torque cannot be seen from the results of method 1. The blue time characteristic according to method 2 shows an interesting feature shortly before the end of the starting interval, which is inherent to all electrical machines. Due to its electromagnetic characteristics, they develop a small electromagnetically excited spring constant. The frequency which can be seen from the plot is the torsional resonance frequency, determined by the electromagnetic spring constant and the sum of all moments of inertia of the shafting. This effect is also responsible for the over swing of the speed at the end of the starting period.

The lowest critical speed of the drive was calculated to $f_{\text{crit}} = 11$ Hz, which cannot be seen from figure 4. The rotational speed is identical to the torsional critical speed at about 10 s after starting. At this time, the curve of the motor torque is smooth, it does not contain pulsating torques of the frequency close to the torsional critical speed. Therefore, torsional vibrations of reasonable amplitudes cannot develop and therefore are not visible in the plots of speed or shaft torque (with shaft torque is meant the torque of the coupling between motor and turbo-compressor).

The calculation was performed by use of a damping factor of 1 % which meets the reality for use of metal-elastic couplings. If the computation would have been performed for the damping factor zero, the time characteristics of speed and shaft torque would show extreme fluctuations with a frequency of the torsional critical speed. The reason for this phenomenon is the pulsating torques of the motor with line frequency at the beginning of the start-up, which would excite torsional vibrations of the shafting not decaying because of the lack of damping. Of course, such procedure gives unrealistic results, but it is sometimes followed by an operator who is not aware of the great effect of an even small mechanical damping.

480
7 START-UP OF A TURBO-COMPRESSOR DRIVEN BY A SYNCHRONOUS MOTOR

The start-up of turbo-compressor drives with synchronous motors needs especially a very careful calculation of transient phenomena in the design stage because synchronous motors – as explained before – unavoidably produce pulsating torques of twice-slip frequency. Figure 5 is applicable to a drive having the lowest torsional critical speed of $f = 10$ Hz. At that instant of time, when the rotational speed during starting reaches a value ($1.350 \text{ min}^{-1}$ for the example under consideration), for which the double of the slip frequency coincides with the torsional critical speed, remarkable fluctuations of the torque in the coupling between motor and turbo-compressor are excited; its peak values are much higher than the amplitudes shortly after starting. The calculation was performed for a counter torque, which is proportional to the square of the speed and amounts to 27% of the rated torque at synchronism. The amplitudes depend mainly on the design of the motor (grade of asymmetry of the rotor) and the coupling (stiffness and damping).

It should be mentioned for completeness, that specific attention must be paid, when synchronous motors with pole-shoes made by solid iron are used. In this case, the common ladder network is unsuitable, but Atlas Copco is able to perform calculations also for such machines.

The explanations up to now were related to the results of method 2. The comparison with the results of method 1 demonstrates that they are so far away from the reality that the conclusion must be taken: A reliable computation of the transients during starting must be based on method 2.

The asynchronous start-up results in the steady-state operation at that speed, for which the asynchronous torque of the motor coincides with the counter torque of the compressor. This speed is below the synchronous speed and the drive must be synchronized in order to be operated as synchronous motor. The transients during synchronization are included in the diagrams of figure 5 and 6. This process is initiated by opening of the field winding, followed by connecting it to a D.C. voltage source. In the given case, the voltage of two times the rated field voltage was selected in order to ease and accelerate the synchronization. Shortly after the motor has fallen into synchronism, the D.C. voltage will be reduced to its rated value. As you can see from the plot curves, the synchronization is no harmful switching operation.

8 SUDDEN THREE-PHASE SHORT-CIRCUIT AT THE MOTOR TERMINALS

This failure can happen in practice. Therefore, all elements of the drive must be capable to tolerate the subsequent transients.

Figure 7 shows the time characteristics of the motor torque and the torque in the coupling between motor and turbo-compressor. Both traces are predominantly influenced by the pulsating torque of line frequency. Since the speed deviates not so much from the rated speed during the transient, the calculation results of method 1 differ much lesser from those of method 2 as during starting.

I refrain from the presentation of equivalent diagrams of the sudden three-phase short-circuiting of a synchronous motor, because they look very similar as for induction machines.
9 SUDDEN TWO-PHASE SHORT-CIRCUIT AT THE MOTOR TERMINALS

Also this occurrence is possible in practice. Caused by the single-phase character of the stator winding, the machine generates not only pulsating torques of line frequency, but also of twice line frequency – see figure 8.

Often the question is raised, if the two-phase or the three-phase sudden short-circuit is the more dangerous fault condition. No general answer is possible for the following reasons:

The motor torque is always approximately 30 % higher in case of the sudden two-phase short-circuit. This relation follows also from a comparison of the relevant values in figure 7 and 8. If the stator housing of the motor can be considered as nearly infinitely stiff, the fastening elements at the foundation have to withstand the motor torque.

By contrast, the excess torque of the coupling between motor and turbo-compressor is in most cases smaller for the two-phase short-circuit than for the three-phase short-circuit. This rule is fulfilled also in the given case.

Consequently, the conclusion must be taken, that calculations must be performed for both kinds of short-circuits.

10 LINE TRANSFER IN CASE OF A DRIVE WITH INDUCTION MOTOR

Customers often ask for the possibility, in case of a mains failure, in which case the supply voltage drops to zero, to reconnect the motor to another bus. The transients, which occur during such system transfer, depend mainly on two parameters:

- the switch-over time,
- the phase angle between the voltage at the motor terminals and the voltage of the new supply system at the instant of closing the circuit breaker.

The diagrams of figure 9 and 10 illustrate what happens. During the switch-over time, the stator current is zero, consequently no motor torque can be generated. Therefore, the speed decays during this interval. However, the magnetic field in the air-gap of the motor is not zero, because it is sustained by a D.C. current in the cage. The residual field induces voltages of speed-frequency in the stator winding.

It is obvious that no transient would occur, if the induced voltage by the residual field and the voltage of the new supply would be identical regarding amplitude, frequency and phase angle in that instant of time when the circuit breaker is closed. The magnitude of the transients increases with an increasing difference between the amplitudes of the two voltages. Many numerical calculations showed that a phase difference of approximately 140° between the two voltages results in the worst case for the transient torques. The calculations for figure 9 and 10 are made for this condition and the ratio $U_{\text{residual}}/U_N = 0.3$ of the voltages.

In many cases, $U_{\text{residual}}/U_N = 35$ % is the maximum tolerable figure. In practice, the circuit breaker for reconnecting the motor to the bus is blocked until the permissible ratio of the voltages is reached.
11 SUPPLY INTERRUPTION IN CASE OF A DRIVE WITH SYNCHRONOUS MOTOR

System transfer is often not allowed by the motor manufacturer in case of drives with synchronous motors, because it may result in mechanical stresses on the overhang parts of the motor exceeding its capability.

This clause deals with the less severe transients during a supply interruption, which means that the supply voltage becomes suddenly zero, caused by whatever event in the distribution network, and will be restored a few periods of line frequency later.

Figure 11 and figure 12 show the details of a supply interruption, starting from rated operating conditions for a time interval of 40 ms with voltage zero at the motor terminals. The motor torque is assumed to be the constant rated torque, which drops to zero at the beginning of the interval of interruption. During this period of time, the air-gap field is sustained by the damper winding; this can be seen from the plot of the stator voltage. The induced voltage is higher than the rated voltage; this effect is caused by the overexcited operation of the machine before the impressed stator voltage disappeared.

At the instant, when the supply voltage is restored, again a phase difference of 140° is assumed between the induced and the recovered voltage. Therefore, heavy fluctuations of the motor torque occur with maximum amplitude of approximately 16 times the rated torque. The motor torque oscillates with line frequency, whereas the torque in the coupling between motor and turbo-compressor oscillates approximately with the frequency of the torsional critical speed. The peak torque within the shafting amounts approximately to 8 times the rated torque which may be permissible. However, the motor does not fall again into synchronism. This effect can be seen from the time characteristic of the current in the damper winding, which at the end of a transient is an A.C. current of slip frequency and from the time characteristic of the speed.

Asynchronous operation is not permissible for a synchronous motor continuously, because the damper winding is not designed for this purpose. Therefore, monitoring and/or protecting devices must be installed by which either the motor is separated from the supply in any case of a voltage drop to zero or a field forcing must be initiated in order to synchronize the motor again.

CONCLUSION

The time limit for this report made it necessary to concentrate the explanations on key aspects of transient phenomena in turbo-compressor drives. The examples of typical switching and unavoidable fault conditions showed that excess torques are excited in the shafting which needs a careful and reliable forecast. It was demonstrated that only a calculation scheme for the simultaneous solution of the electromagnetic equations and the equations of motion are suitable for the intended purpose. Such software package can be organized for a complete model of the whole shafting or for a reduced model which is derived from a sophisticated mechanical modeling of all details.
Figure 1: 3-stage gear type turbo compressor
Air gap torque $T(t)$ of unsaturated run up from stand still:

**IM:**  
$$ T(t) = (1 + e^{-t/\rho_{s1}}) * e^{-t/\rho_{s2}} ) + T_s*( e^{-t/\rho_{s1}} ) * cos( \omega_s t - \phi_{s} ) + e^{-t/\rho_{s2}} * cos( \omega_s t + \phi_{s} )$$

**SM:**  
$$ T(t) = T_0(t)* (1 + e^{-t/\rho_{s1}}) * e^{-t/\rho_{s2}} ) - T_s*e^{-t/\rho_{s1}} * cos( \omega_s t - \phi_{s} ) - T_s*e^{-t/\rho_{s2}} * cos( \omega_s t + \phi_{s} ) + T_p(t) * sin(2p*( \omega_s t - \alpha ) )$$

The factors $T_0(t)$ and $T_p(t)$ are preliminary slip dependent. The motor manufacturer normally pre calculates the approximate time for start up with respect to the total mass inertia of the unit and the voltage net condition.

The run up time in conjunction with the assumption $ds/dt = constant$ gives a relationship $s = f(t)$ respectively $T_0 , T_p = f(t)$ and makes this equation usable for a time step solution.

Air gap torque $T(t)$ of a 2-phase short circuit:

**IM:**  
$$ T(t) = T_{2s1} * (1 + e^{-t/\rho_{2s1}}) * sin( \omega_s t ) + T_{2s2}*( e^{-t/\rho_{2s2}} ) * sin( 2\omega_s t )$$

**SM:**  
$$ T(t) = T_{2s1} * e^{-t/\rho_{2s1}} + T_{2s2} * e^{-t/\rho_{2s2}} ) * sin( \omega_s t ) + T_{2s3} * e^{-t/\rho_{2s3}} * sin( 2\omega_s t )$$

Air gap torque $T(t)$ of a 3-phase short circuit:

**IM:**  
$$ T(t) = T_{3s} * e^{-t/\rho_{3s}} * sin( \omega_s t )$$

**SM:**  
$$ T(t) = T_{3s1} * e^{-t/\rho_{3s1}} + T_{3s2} * e^{-t/\rho_{3s2}} * sin( \omega_s t )$$

Air gap torque $T(t)$ of an automatic reconnection:

**IM:**  
$$ T(t) = ( T_{rec1} + T_{rec2} * cos( \omega_{rc} t + \phi_{rc} ) ) * e^{-t/\rho_{rc}}$$

**SM:**  
$$ N.A $$

Figure 2: Analytical Functions
Equivalent Circuit of a Cage Induction Machine with Deep-Bar Rotor

Voltage and Torque Equations of an Induction Motor with Single-Cage Rotor

\[ U_1' = R_1 I_1' + L_1 \frac{dI_1'}{dt} + \frac{3}{2} M \frac{d}{dt} \left( I_2' \cdot e^{j\beta} \right) \]

\[ 0 = R_2 I_2' + L_2 \frac{dI_2'}{dt} + \frac{3}{2} M \frac{d}{dt} \left( I_1' \cdot e^{-j\beta} \right) \]

\[ \beta = \int \omega_m dt = 2\pi \int n dt \]

\[ T_{\text{motor}} = 3pN_2M \cdot \text{Re}\left( j I_1' * I_2' \cdot e^{j\beta} \right) \]

Figure 3: Analytical model of cage induction motors
The rotational speed is identical to the torsional critical speed $f_{\text{crit}} = 660$ CPM.

**Figure 4:** Start-up of a 3-phase cage induction motor
Figure 5: Start-up of synchronous motor
Figure 6: Start-up of synchronous motor
Figure 7: 3-phase short circuit of a 3-phase cage induction motor
Figure 8: 2-phase short circuit of a 3-phase cage induction motor
Figure 9: System transfer of 3-phase induction cage motor
Figure 10: System transfer of 3-phase induction cage motor
Figure 11: Supply interruption of synchronous motor
Figure 12: Supply interruption of synchronous motor
Numerical simulations of flow and particle dynamics within a centrifugal turbomachine

Adel Ghenaiet
Thermal power laboratory, Polytechnic School, Algiers

ABSTRACT
Gas turbines and turbochargers operating in extreme environments which are polluted by dust and sand particles are subjected to erosion damage, leading to drastic aerodynamic performance degradations of their components. In this paper, the results of numerical simulations of particle-laden flow through a small turbo-compressor are presented. The solving technique for particle trajectories is based on the Lagrangian tracking model, which accounts for turbulence effect based on the eddy lifetime concept. The flow field was solved separately from the solid phase by using the code TASCflow. On the other hand, the governing equations of particle motion were solved by using an in-house developed code. As the particle trajectories and locations of impacts through the impeller, diffuser and scroll were predicted, a semi empirical erosion correlation was used to estimate erosion contours. The initial particle sizes followed a random size distribution according to MIL-E 5007E specification, also, particle fragmentation was considered. The number of particles and their initial positions in the intake were specified by a concentration profile.

Keywords: centrifugal compressor, airflow, aerodynamic performance, particle, trajectory, impact, erosion

1 INTRODUCTION
In Saharan countries, automotive gas turbines, turbochargers, and auxiliary power units are operating in environments where the ingestion of dust is inevitable. Continued operation under particulate flow conditions results in drastic degradations of the turbomachine performance and its structural integrity. The use of experimentations to assess turbomachinery life and performance degradation under such severe operating conditions is very costly. It is therefore, vitally important to be able to use simulation procedures for predicting particle trajectories and areas prone to erosion, which are useful in health monitoring for such equipments. Particle trajectories computations through axial flow blades were firstly reported by Hussein and Tabakoff [1]. Later, they developed a procedure for solving three dimensional particle trajectories through rotating and stationary turbomachinery blade rows [2].

© IMechE 2005
Particle trajectories in centrifugal compressors are consistently different from those in axial flow machines, because of the nature of flow and the direction of centrifugal forces. Elfeki and Tabakoff [3] were the first to study particulate flow in a centrifugal compressor, and gave the following remarks: Particle collisions were distributed over the entire surface of the impeller blade pressure side, with largest impacts in the middle area. But, collisions on the suction side were mainly closer to blade leading edge, and the hub was subjected to few impacts than the casing. The prediction of particle trajectories and erosion in turbomachinery is still a very complex problem, due to many factors, including; flow conditions, initial position of the rotor blade, its material and geometry, rebound characteristics, particle fragmentation and initial positions, velocities and size of particles.

This paper presents a theoretical study for flow and particle dynamics and subsequent erosion through different components of a turbo-compressor. Firstly, the flow field was solved by a Navier Stokes solver ‘CFX-TASCflow’. Afterwards, the particle trajectories through this complex turbomachine geometry were obtained using a developed code based on a stochastic Lagrangian tracking model, including turbulence effect and particle fragmentation, in addition to random particle sizes and rebound (restitution) factors. The governing equations of particle motion were solved in a stepwise manner using the seventh order Runge-Kutta Fehlberg technique. In addition, the finite element method was used for particle tracking in different cells of the computational domain. The knowledge of points of collisions, magnitude and direction of impinging velocities was critical in predicting erosion contours. This developed trajectory-erosion code based on such methodology was validated in the case of an axial turbomachinery and gas turbine intakes [4, 5].

The particle-laden flow simulations were carried out based on an existing turbo-compressor taken from a small two-shaft gas turbine ‘Cussons Ltd’ (fig.1) used for teaching, which provides about 2 kW of electrical power. The radial compressor ‘CAV’ shown in figure 2 has 12 blades made from aluminium alloy, which rotates at a speed of 30000 - 90000 rpm. The inlet and outlet diameter and height are 22.7 mm, 38.1 mm and 5 mm, respectively. The outlet diameter and height of the pinched diffuser are 60.25 mm and 4.1 mm, respectively. The external diameter of the scroll is 145 mm with a diameter of discharge of 43.7 mm.

Simulation results of particle trajectories and erosion for this type of turbomachine revealed distinctive zones of impacts with high rates of erosion, and generally, impeller blade pressure side and shroud are more subject to erosion.

2 FLOW FIELD ANALYSES

The flow field within this turbo-compressor is solved by using the commercial code TASCflow, which uses a full discretisation for the averaged conservation equations of Navier Stokes within finite volumes. The computational domain for both flow field and particle trajectories comprises one impeller blade passage, an intake, vaneless diffuser and a scroll formed by sectors of 30 degree, (fig. 3). Five attached H type grids as depicted by figure 4 are used, with a finer meshing in the tip clearance, around blade and in wake region. The boundary conditions used for CFD simulations are; a constant total pressure at compressor intake and mass flow at outlet from scroll. In addition, periodic boundaries are applied at one pitch apart of the blade passage, and at lateral sides of vaneless diffuser and intake.
The aerodynamic performances were estimated base on mass-averaged aero-thermodynamic parameters, with respect to the inlet and outlet surface of the computational domain. The pressure rise coefficient and the total-to-total adiabatic efficiency are obtained as follows:

\[
\Pi_c = \frac{P_2}{P_1}, \quad \eta_{\text{Euler\_work}} = C_{pT} \left[ \frac{\Pi_c}{\Pi_c - 1} \right]
\]  

(1)

McCutcheon [6] pointed out that turbocharger compressors usually work over a greater range than that for gas-turbines, for which a surge range of 17% was generally specified. The surge line for this compressor was estimated according to the definition of surge margin [7].

3 PARTICLE TRAJECTORIES COMPUTATION

In the study of particle laden turbulent flows, there are generally two common approaches; The Eulerian and Lagrangian, which differ from each other in handling the dispersed phase. The Eulerian approach treats both the continuous and dispersed phase as two interpenetrating continua. On the other hand, the Lagrangian approach handles discrete particle dispersion using a stochastic method by tracking a large number of individual particle trajectories [8], which is able in handling more detailed physical description of the particle phase, such as the interaction of particles with walls. By adopting the Lagrangian model in this study, the motion of a solid particle through the turbo-compressor components is described by solving a set of differential equations along a trajectory in a compressible gas stream, with the assumption of velocity slip between gas and particle phases. The turbulent effect is included throughout the eddy lifetime concept, given by Gosman et al [9]. The turbulent velocity components are assumed to prevail as long as the particle eddy interaction time is less than the eddy lifetime. The turbulent velocities and the eddy lifetime and length are calculated based on the local turbulence properties of flow, and estimated according to Shirolkar [10].

The three dimensional equations are derived by considering; inertial forces and aerodynamic drag force (expression below) due to skin and form drag, given by Morsi et al [11]. For irregularly shaped particles, a shape factor [12] is considered in the drag expression.

\[
\bar{F}_D = \frac{\pi}{8} d_p^2 \rho_f C_{D_p} \left[ \bar{V}_f - \bar{V}_p \right] \left| \bar{V}_f - \bar{V}_p \right|
\]  

(2)

If a particle is sufficiently large and there are large velocity gradients related to fluid shearing, there will be a particle lifting force called Saffman force ‘FS’, based on linear and rotational Reynolds number [13].

The particle inertia forces (centrifugal and Coriolis forces) are derived from the second derivative of the particle vector position. Considering the drag, gravity and Saffman force as main external forces, the following set of second order non-linear differential equations is derived.

\[
\begin{aligned}
\frac{\partial^2 r_p}{\partial t^2} &= G (V_{gy} - V_{y,p}) + r_p \left[ \omega + \frac{V_{gy}}{r_p} \right]^2 - g \left( 1 - \frac{\rho_f}{\rho_p} \right) \sin \theta_p + \frac{F_{sx}}{m_p} \\
\frac{\partial^2 \theta_p}{\partial t^2} &= G (V_{gy} - V_{y,p}) - 2V_{y,p} \left( \omega + \frac{V_{gy}}{r_p} \right) - g \left( 1 - \frac{\rho_f}{\rho_p} \right) \cos \theta_p + \frac{F_{sy}}{m_p} \\
\frac{\partial^2 z_p}{\partial t^2} &= G (V_{y} - V_{y,p}) + \frac{F_{sz}}{m_p}
\end{aligned}
\]  

(3)
The aerodynamic factor is function of fluid and particle velocities.

\[
G = \frac{3 \cdot \rho \cdot C_0 \cdot d_p}{4d_p \cdot \rho_e \cdot S_p} \left[|V_{p} - V_{p1}|^2 + |V_{p} - V_{p2}|^2 + |V_{p} - V_{p3}|^2 \right]
\]  
(4)

The difficulties of providing a satisfactory step control for the ordinary Range-Kutta methods were largely overcome by the implementation of the Fehlberg R-K method [14]. The integration of the set of differential equations (3) gives the position of a particle in physical coordinates. However, for the purpose of particle tracking in different cells, the local co-ordinates are calculated by solving a non-linear set of equations using Newton-Raphson method.

The periodic boundary conditions used for the lateral sides enable a better modelling of flow and trajectories near the blade leading and trailing edge and in tip clearance. The rebound velocity and angle beyond a point of impact are computed from the restitution factors derived experimentally by Tabakoff et al [15]. The magnitude of an impact velocity and its direction relatively to a target surface constitute the essential data needed to evaluate the erosion rate.

The simulations were carried out at different compressor rotating speeds and maximum efficiency mass flow rates. Sand particles (MIL-E5007E, 0-1000 \( \mu \)m) were seeded upstream of the inlet compressor eye in order to simulate an extremely polluted atmosphere, by considering a uniform concentration of 775 mg/m\(^3\) [4].

The erosion rate expressed as the amount of material removed (milligrams) per unit of mass of impacting particles (grams) is a strong function of particle velocity, angle of impingement, target material and the mass of impacting particles. The value of erosion rate was estimated according to the correlation developed by Grant et al [16], which was based on aluminium alloys and silica sand as abrasive particles.

\[
e = K_{f} f(\beta) (V_{i0}^2 - V_{i}^2) + f(V_{i1})
\]  
(6)

The values of erosion (mg) at discrete points of an element face were summed and divided by the total mass of particles relatively to that face, in order to get an equivalent erosion rate. Then, the nodal value of erosion was evaluated by using a bilinear interpolation on four faces surrounding this node.

4 DISCUSSIONS

Flow and particle trajectories computations were carried at different turbo-compressor operating conditions, considering a large band of particle size distribution ‘MIL-E 5007E’ at a very high concentration.

Fluid velocity vectors at the higher speed of rotation of 90000 rpm near maximum efficiency mass flow rate are plotted in figures 5, which is depicting high flow acceleration over the suction side of the impeller inducer and along its straight part (fig. 5-a). Contrarily, on the pressure side, flow velocities are lower, with an evidence of a retarded flow. The existing
running clearance between the impeller blade and the casing wall allows fluid to traverse against the direction of rotation from the high to low pressure face (fig. 5-b). There is a clear evidence of reduced velocities in the shroud-suction surface corner toward the outlet, and a higher velocity on the other side of the passage near blade pressure side. There is an evidence of a Jet/wake flow pattern at the centrifugal compressor impeller outlet, causing further losses and highly distorted velocities at the impeller discharge, (fig. 5-c). This latter was generated by the rotation, channel curvature, flow separation on the suction side, and by tip leakage flow. The velocity vectors in the vaneless diffuser downstream of the impeller show changing of velocity profiles as the flow goes outward as seen from figure 6.

The figure 7 of total relative pressure is depicting big losses across the impeller channel. The resulting tip wake is squeezed down and moves closer to shroud. But, at a low flow-rate, the size of the wake increases, and moves around the blade suction side. The tip leakage flow displaces this wake from the casing-suction corner toward the middle of the passage.

The computed pressure rise coefficient and the total-total adiabatic efficiency are presented in figure 8 and 9. At a high speed of 90000 rpm, the maximum efficiency reached value of 75 % at a mass flow of 0.20 kg/s, whereas, at a low speed of 30000 rpm the maximum efficiency is 77 % corresponding to a mass flow of 0.078 kg/s. The design speed for this compressor seems to be around a speed of 60000 rpm. The predicted stalling point was based on a value of 17 % of the surge margin.

Typical trajectories of large particles (150-500 µm) are dominated by their inertia and deviate considerably from the streamlines, figures 5. The impeller rotation produce a big centrifugation imparted on particles, as a consequence trajectories are shifted toward higher radii (fig 5-b), especially in the axial part of the impeller passage, which is more obvious after blade impacts. The amount of deviation depends on the particle size, channel curvature and the rotational speed.

Figures 6 show impacts on the inducer leading edge, and over the third part of the blade. The majority of impacts are found on the pressure side, and along the straight part of the impeller, which are concentrated close to the tip. Impacts on the shroud occur mainly after the bend of the blade passage, however, on the hub, impacts are less encountered. In the vaneless diffuser particles are deviated because of changing in frame of reference, as indicated by particle paths, figure 5-a, with many impacts on hub and shroud (figure 5-b). In the volute, the impacts are mainly in the bottom and over the outer wall, thus, generating high rates of wear. Furthermore, some of particles are deflected upward in the scroll. Unlike large size particle trajectories which are dominated by their inertia, small size particles (5 µm) follow streamlines more closely and are less susceptible to surface impacts and deflections, as indicated by figures 12. Smaller particles are more affected by airflow, because of larger influence of the aerodynamic drag force on them, which can be seen from the plotted particle trajectories after collisions. The shroud impacts are concentrated near the blade pressure side, in addition to some impacts toward the suction side by particles crossing over the tip as shown by figure 12-a. Contrary to large ones, there are many impacts on the hub by small sizes particles. However, in the diffuser, the number of impacts by these particles is reduced, and particles are deviated towards the upper of scroll, generating high rates of wear in this critical enlarging area. At a reduced speed of rotation, similar trends were observed for the particle dynamics, but the intensity and rates of deviations differ from those found at high speed, because of reduced centrifugation and changing of flow patterns.
The predicted impact locations at the high rotational speed of 90000 rpm show that particle impacts on the blade pressure side are distributed over the entire surface, but, with largest impacts found in the middle area. Over the suction surface, impacts are mainly closer to the blade leading edge, and few particles were found to impact the blade suction surface beyond this region. The impeller hub is subjected to fewer collisions than the casing, owing to high centrifugation. The impacts on the casing are concentrated almost over the two thirds of the blade from the inducer. The impacting velocity levels as shown by coloured dots in figures 11 are indicating high velocities on the pressure side toward the tip and on the shroud, due to high centrifugal force. The velocities of particles were generally reduced towards the hub and around the inlet eye. The highest impact velocities are found near the tip in a strip of dense impacts over the mid third of the blade length.

The erosion rate and material removal depend on the particle size and the rotational speed, and in general, the pressure side is more eroded than the suction side. Two main areas of erosion are clear from figures 13; the first is from the leading edge to the first third of impeller extending toward the blade root, and the second from the last third of the impeller blade, with a developed erosion area near the blade root near the impeller discharge. The casing is mainly affected in its axial part and in the bend region, with higher rates of erosion toward the pressure side, (fig 13-b).

5 CONCLUSIONS

Particle-laden flow through a centrifugal compressor was studied and some of the results are presented in this paper. The particle trajectories were determined for a broad range of particle size (MIL-E5007E, 0-1000 µm), and at different compressor operating conditions. But, only results corresponding to the operating point at the maximum efficiency mass flow rate and maximum rotational speed were presented and discussed. The flow field within the turbo-compressor components and the aerodynamic performance of the compressor stage were determined first, which allowed setting the operating range and the determination of the flow information necessary in particle dynamics simulations. The particle size and speed of rotation were found to have significant effect on trajectories, locations of impacts and erosion pattern. Most of particles encounter repeated impacts with impeller pressure side, which deflect them toward the casing, but, fewer impacts were found over the suction side and on the hub. The intensity and pattern of erosion is dependent on the frequency of impacts, the magnitude and direction of impinging velocities. The blade pressure side is generally subject to more erosion than suction side, and the erosion of the casing is mainly over the radial part of the impeller, closer to pressure side. The computed erosion contours indicate that the maximum blade wear is expected on the pressure side near the casing, over the last part of blade, but, erosion of the hub is insignificant and mainly attributed to smaller particles’ collisions. The turbulence and particle fragmentation were found to influence the dynamics of finer particles and subsequently erosion patterns, especially, at the impeller discharge, near the hub and in the volute. Information from this theoretical study may be used to evaluate the associated degradation in aerodynamic performance, and to select necessary coating used for the critical regions more affected by erosion wear.

REFERENCES


**NOTATION**

- \( C_D \) drag coefficient
- \( C_p \) specific heat at constant pressure
- \( d_p \) diameter of a particle
- \( g \) gravity
- \( m \) mass
- \( r \) radial co-ordinate
- \( P \) total pressure
- \( T \) total temperature
- \( t \) time
- \( S_F \) shape factor
- \( V \) velocity
- \( z \) axial coordinate
- \( \beta \) impact angle (degree)
- \( \varepsilon \) erosion rate (mg/g)
- \( \eta \) adiabatic efficiency
- \( \omega \) speed of rotation, rotation of fluid
- \( \rho \) density
- \( \Pi \) pressure ratio
- \( \theta \) tangential co-ordinate

**Subscripts**

- \( f \) Fluid
- \( N \) normal
- \( p \) particle
- \( r \) radial
- \( tt \) total
- \( 1,2 \) at impact and rebound from a surface
- \( 1,3 \) at inlet and outlet from a compressor stage
Figure 1  The Cussons’ turbo-compressor

Figure 2  The impeller geometry

Figure 3  The impeller geometry

Figure 4  The hub to shroud grids

Figure 5  (a) Mid-height blade flow velocities; (b) tip clearance velocities; (c) impeller outlet flow velocities; compressor operating at a speed of rotation of 90000 rpm
Figure 6 Velocities at the hub-shroud plan, at speed of rotation of 90000 rpm

Figure 7 (a) Total relative pressure at mid-span; (b) in tip clearance at speed of 90000 rpm

Figure 8 Pressure rise coefficient

Figure 9 Adiabatic total-to-total efficiency

Figure 10 (a) Top view of MIL-Spec particle trajectories; (b) Side view of trajectories; (showing 24 particles of 150-500 µm), compressor is operating at 90000 rpm
Figure 11 (a) Impact velocities on the pressure side; (b) and on the casing, compressor is operating at 90000 rpm

Figure 12 (a) Top view of 5µm particle trajectories; (b) Side view of trajectories, compressor is operating at a speed of 90000 rpm

Figure 13 (a) Erosion (mg/g) on the blade pressure side; (b) and on the casing, compressor is operating near maximum efficiency mass flow rate at a speed of 90000 rpm
SYNOPSIS

Fan blades of high bypass ratio gas turbine engines experience high aerodynamic and centrifugal loads, producing the well-known phenomenon of fan blade untwist. Although manufactured to high standards some geometric variation from blade to blade is inevitable. This paper summarises the results of a computational study investigating the effect of static stagger variability on the dynamic untwist behaviour of fan assemblies. The equilibrium untwist configuration of geometrically mis-staggered fan assemblies was shown using an integrated time-domain aeroelasticity model. At certain operating conditions the influence of a single mis-staggered blade is shown to extend to the whole assembly. The stability of the system was determined at a range of operating points, giving further insight into the predicted behaviour.

1 INTRODUCTION

As an assembly of thin flexible aerofoils rotates, the centrifugal and aerodynamic forces deform the blades giving rise to the well-known phenomenon of untwist. Such assemblies are typically found in the fan system of gas turbines powering large civil airframes. It is crucial to account for the untwist of the fan at the design phase so that the blades assume the desired aerodynamic shape under running conditions. At sea level, where the aerodynamic loads are high, the deformation induced by the rotation varies significantly between operating conditions. Tip-timing measurements taken during sea-level testing of an engine suggested that over a limited operating range blade untwist becomes very sensitive to both speed and fan configuration.

Traditionally the untwist of a blade is calculated using aerodynamic pressure loads derived under the assumption that all blades are identical. In reality, manufacturing limitations will result in blades that are all subtly distinct, both aerodynamically and structurally. Geometric variability has been shown to enhance flutter stability(1), alter the vibratory response of an assembly to
aerodynamic excitation\textsuperscript{(2)}, influence aerodynamic performance\textsuperscript{(3)} and generate multiple pure tone or “buzz saw” noise\textsuperscript{(4)}. However, to the best of the authors’ knowledge, all analyses of geometric variability ignore the additional change in blade shape induced by varying pressure loads arising from geometric blade-to-blade differences. In this paper the influence of stagger variability on the untwist behaviour of a fan assembly is analysed through a computational study, addressing this shortfall.

2 AEROELASTIC FORMULATION

The computational study was performed using an integrated non-linear aeroelastic code that couples a non-linear CFD model to a linear modal model of the structure. The formulation of the code along with case studies are reported by Sayma et al.\textsuperscript{(5,6)}.

The first step in the application of the aeroelastic code to the problem of untwist prediction is to use a finite element structural code to calculate the blade deformations induced when the assembly is rotating in a vacuum. A modal analysis is also performed and the dominant low frequency vibration modes extracted, forming the basis of the structural model in the aeroelastic code.

Next, the steady-state flow solution around the centrifugally displaced blades is calculated. At this stage the blades are held fixed and not allowed to deform in response to the aerodynamic loads. Such a configuration can be viewed as modelling very low density flow around the fan assembly, in which the interaction between the aerodynamics and the structure is very weak. Once the steady-state solution is obtained, corresponding to the geometry deformed by centrifugal loads but not by aerodynamic loads, the system can then be marched forward in a time accurate fashion to find the effect of the real aerodynamic loads. In this, surface pressure loads and blade displacements are exchanged between the aerodynamic and structural models at every time-step. The aerodynamic mesh is moved at every time step using a spring analogy to accommodate the motion of the blades. When the system reaches equilibrium, that is the steady deformation with no further vibrational motion, the untwist of the blade is determined. The structure is usually heavily damped so that the untwist position can be obtained with minimum oscillation about the final position of equilibrium.

The blade surface is treated as a viscous boundary, while the flow adjacent to the end-walls is assumed inviscid for simplicity. The mesh is semi-unstructured in form, with a structured o-mesh around the blade to resolve the boundary layer and a layered unstructured mesh elsewhere. To ascertain the density of the aerodynamic grid required to produce acceptably accurate untwist results, a mesh sensitivity study was carried out using two mesh densities. It was found that increasing mesh size by a factor of two only marginally influenced the predicted untwist, hence the “coarser” mesh of five million points was adopted for the analysis.

3 UNTWIST ANALYSIS

A characteristic feature of the flow around a transonic fan is the position of the shock wave in the tip region. At low speeds, the shock forms on the suction surface ahead of the blade passage; the shock is expelled ahead of the leading edge and the flow is referred to as “unstarted”. At
higher speeds the shock moves rearward and forms in the blade passage itself; the shock is swallowed and the flow is “started”. As the shock position greatly influences the pressure load on the blade, untwist analyses were conducted at three operating points representative of each condition; unstarted, started and an intermediate case between the two regimes.

There are essentially two ways that the range of flow regimes of interest can be produced. Firstly, the fan can be moved along its working line by changing the speed of rotation at a constant downstream nozzle area. As the fan speed increases the mass-flow rises and the shock wave moves rearward. Secondly, the fan speed can be held fixed and the mass-flow varied by changing the exit nozzle area, causing the fan to traverse a constant speed characteristic. The former method shall be applied here as it is representative of the behaviour of a fan installed in an engine.

3.1 Full assembly analysis
A series of full annulus untwist calculations was carried out on fan sets consisting of mis-staggered blades. The geometric variation was achieved by modifying the stagger profile of individual blades in a linear fashion, applying zero displacement at the root and maximum stagger change at the tip. The deformations of blades rotating in a vacuum serve to reduce the stagger variation of an assembly relative to static conditions; a $\pm 0.35^\circ$ static mis-stagger translates to a $\pm 0.26^\circ$ mis-stagger at the intermediate shaft speed. In the speed range of interest, the stagger variation changes by only $0.003^\circ$ for this level of static mis-stagger. As a result all stagger variations were applied to the centrifugally displaced blades, acknowledging that this translates to a slightly larger static stagger variation. To ensure that the observed effects originate from the imposed geometric variability only, the mechanical properties of the blades were held fixed.

3.1.1 Unstarted and started flow
Calculations for assemblies containing a solitary mis-staggered blade ($+0.2^\circ$ tip stagger) were performed at both low and high shaft speeds corresponding to unstarted and started flow conditions respectively. The resultant untwist patterns are shown in figure 1(a) and 1(c), with the mis-staggered blade denoted as blade 1 and the remaining nominal blades consecutively numbered in the direction of rotation of the fan. It can be clearly seen that for both flow conditions, the presence of a single mis-staggered aerofoil in the assembly predominantly influences the untwist of the modified blade itself. The equilibrium positions of the blade immediately upstream and the first three blades downstream are marginally influenced by the mis-staggered blade ($\pm 0.04^\circ$).

Figures 2(a) and 2(c) show the predicted running tip stagger pattern for a randomly mis-staggered assembly. Once again the variation in pressure untwist is small relative to the prescribed mis-stagger, and in general the equilibrium position of the most severely mis-staggered blades deviate from the nominal level by the greatest margin. Notice that the inclusion of the pressure loads does not significantly alter the form of the tip stagger pattern. The results show that for both unstarted and started flow, the additional pressure untwist variation created by the stagger variability is small. As such the static pattern gives a good approximation to the running stagger variation of the assembly.

3.1.2 Intermediate flow condition
Calculations analogous to those discussed in the previous sections were performed at the intermediate aerodynamic speed near the transition between unstarted and started flow conditions. At this operating point the shock wave forms near the leading edge of a nominal blade in a
uniformly staggered assembly. Figure 1(b) shows the results of the untwist calculation performed where the assembly contains a single mis-staggered blade (+0.2° tip stagger). The results at this intermediate speed differ significantly from those at the started and unstarted flow conditions. Firstly, the resultant pressure untwist pattern now overwhelms the prescribed mis-stagger, with a 0.2° mis-stagger resulting in a blade to blade pressure untwist variation of ±0.54°, as opposed to ±0.074° for unstarted flow. Secondly, the region of influence of a single mis-staggered aerofoil propagates completely around the annulus, significantly moving the equilibrium of each blade from the nominal position. Alternate blades have low and high pressure untwist, forming a saw-tooth pattern. The mis-staggered blade and the first four blades downstream deviate from the approximately constant amplitude alternating pattern displayed by the remainder of the assembly. Figure 3(b) shows the equilibrium casing relative Mach number distribution of the flow. An alternating pattern is also clear in the shock structure of each passage around the annulus, with both expelled and swallowed shocks present in the passages adjacent to each blade.

A mechanism explaining the generation of the resultant untwist pattern from the CF-only steady state position, shown in figure 3(a), is as follows. The mis-staggered blade directly affects the adjacent passage geometries in the CF-only assembly. This causes the shocks in the downstream and upstream passages to move forward and rearward respectively. This shock structure is maintained as the mis-staggered blade untwists, with the region of influence of the mis-staggered blade propagating predominantly to downstream blades. The first downstream blade experiences a higher torque from the aerodynamic loads due to the expelled shock wave in the adjacent upstream passage. This blade untwists more as a result, causing the shock in the adjacent downstream passage to move further rearward. The next downstream blade untwists less due to the reduced torque applied by the aerodynamic loads, pushing the shock wave attached to the suction surface forward, starting the whole process again. The alternating untwist pattern propagates around the annulus unattenuated due to the sensitivity of the shock position at this operating point. The analysis suggests that when the fan operates at this condition the equilibrium position is very sensitive to small changes blade position.

The untwist time history shown in figure 4 shows the propagation of the disturbance when the unsteady calculation progresses from a row of blades with one blade mis-staggered via an unsteady process to a new steady solution. The disturbance propagates in the downstream direction around the annulus, pushing each blade in turn away from its nominal equilibrium position. The signal propagates fully around the assembly, reaching the mis-staggered blade once more. The shocks on both sides of the mis-staggered blade have moved away from the leading edge, attenuating the signal as observed at both started and unstarted flow conditions. This behaviour is far from what was expected, which is to expect the behaviour observed for both the started and unstarted flow conditions, where the influence of a mis-staggered blade diminishes rapidly away from the modified blade. To achieve a condition where such small change to the geometry of a single blade has such a global effect on the assembly is remarkable; it is even more remarkable that the condition where this behaviour occurs is well within normal operating limits, indeed it is where the nominal assembly is most efficient.

An interesting difference between the unstarted flow case and this operating point is the distinction between the static and running tip stagger pattern for the random mis-stagger patterns, shown in figure 2. For unstarted flow, the static and running patterns are qualitatively similar, the inclusion of the pressure loads causing only small deviations from the static pattern. At the intermediate flow condition the static and running patterns differ significantly in both the amplitude of the deviation from the uniform tip stagger and the overall pattern.
4 EQUILIBRIUM STABILITY

The previous section highlighted assembly untwist behaviour that altered significantly with operating condition. At the intermediate flow condition, the region of influence of a single mis-staggered blade extended to every blade in the assembly. It is conjectured that at this condition the equilibrium position of nominal blades is unstable.

To investigate the stability of the uniformly staggered equilibrium, a number of simplifying assumptions will be made. As the instability appears to be manifested during the transient to change to the non-uniform equilibrium in a vibration mode where adjacent blades move out of phase, only these modes shall be analysed. This allows the calculation domain to be reduced to a two-passage case with periodic boundaries, significantly reducing the model size. As the unsteady untwist calculations included an arbitrarily high mechanical damping, it is appropriate to assume that each modal force is a function of the modal displacement only, and does not depend on modal velocities or higher derivatives. Under these assumptions the equation of motion for the structure can be written as

\[ \ddot{q}_i + 2 \zeta_i \omega_i \dot{q}_i + (\omega_i)^2 q_i = F^i(q), \]

where \( \zeta_i, \omega_i \) and \( F^i(q) \) are the damping coefficient, modal frequency and aerodynamic load in the \( i^{th} \) mode respectively. The stability of the system is then determined by the aerodynamic load function in each mode, \( F^i(q) \), which can be calculated at discrete points from steady-state calculations.

By writing \( \dot{q} = r \), the second order equation of motion can be reduced to a simultaneous set of first order differential equations. The stability of the first order equations is then governed by the eigenvalues of the Jacobian, \( J \), evaluated at the nominal equilibrium, where

\[
J = \begin{bmatrix}
0 & I \\
\frac{\partial F}{\partial q} & -\text{diag} (\omega^2) \end{bmatrix}.
\]

If \( J \) has an eigenvalue with positive real part, the equilibrium is unstable. As \( \zeta_i \) and \( \omega_i \) are known for each mode, the determination of the stability of the system reduces to the calculation of \( \partial F/\partial q \). The variation of modal force in each mode with respect to modal displacement can be approximated by using a factorial experimental design approach to construct a least squares approximation, a technique described by Myers\(^8\). Including only two vibration modes in the structural model, namely first flap and first torsion, means that only nine steady-state calculations need to be performed to determine the equilibrium stability at a given operating point.

To easily produce a series of operating points, the blade speed and geometry were held fixed while the exit static pressure was varied, producing a conventional fan characteristic. Note that this is in contrast to the unsteady calculations discussed in the previous section, where the working line of the fan was traversed by varying speed to produce the desired range of operating points. The stability of the uniformly staggered equilibrium along such a characteristic is shown in figure 5. Over a small flow range at the transition between started and unstarted flow regimes, the system displays an instability, corroborating the conjecture derived from the full-assembly unsteady untwist calculations discussed above. The instability is similar to the case of wing divergence, and occurs because the modal force varies rapidly in response to small...
displacements, driving increasingly larger displacements away from equilibrium. The presence of a mis-staggered blade, in an otherwise nominal assembly, forces the neighbouring blades away from the equilibrium position, which, if unstable, causes the blades then move further away from their nominal position until a new equilibrium between the aerodynamic and structural loads is achieved. Away from the transition region, where the flow is started or unstarted, the uniform stagger equilibrium untwist is stable, explaining the attenuation of the influence of a mis-staggered blade at these flow conditions. The equilibrium stability analysis shows that the range of instability is rather small, occurring over ±0.2% of mass-flow relative to the intermediate flow regime.

5 CONCLUSIONS

The impact of stagger variability in gas turbine fan assemblies has been investigated through a computational study. For started and unstarted flow conditions, the additional stagger variation induced by the static geometric variability was found to be small. Therefore, at these conditions, static measurements can be used as a good approximation to the running configuration. At an intermediate flow condition, the influence of a single mis-staggered blade was shown to extend to every blade in the assembly. The running pattern for a randomly mis-staggered assembly at this condition was predicted to differ substantially from the static pattern. A uniform stagger system was shown to exhibit an untwist instability at the intermediate flow condition, providing further insight into the predicted behaviour.

ACKNOWLEDGMENTS

The authors would like to thank Rolls-Royce plc for both sponsoring this work and allowing its publication. They also thank Prof. N. Cumpsty and Mr. K. S. Johal for their useful comments and discussions.

REFERENCES


FIGURES

![Diagram with three graphs showing blade untwist](image)

Figure 1: Resultant untwist, blade 1 +0.2°, (a) unstarted, (b) intermediate, (c) started
Figure 2: Random mistuning resultant untwist, (a) unstarted, (b) intermediate, (c) started

Figure 3: Casing Mach number contours, intermediate speed, (a) CF-only, (b) equilibrium
Figure 4: Intermediate speed pressure untwist snapshots, blade 1 closed by $+0.2^\circ$

Figure 5: Tuned equilibrium stability
Time transient simulation model and full scale experimental verification for high speed rotor delevitation events with a 1.5 ton supercritical rotor supported by dry lubricated bushing type auxiliary bearings

Richard R. Shultz  
Waukesha Magnetic Bearings, Inc., USA

Emiliano Lucchetta  
Waukesha Magnetic Bearings, Inc., UK

ABSTRACT

A primary concern for rotating equipment end users is the assurance of safe operation of equipment during all operating conditions. This paper documents the process to verify the safe operation of supercritical rotating equipment rotors during high speed rotor delevitation events with the rotor fully supported by dry lubricated bushing type auxiliary bearings. All aspects of the verification process are described. Test results have been gathered for turbo expanders, large turbocompressors, large electric drives, and a high-speed flexible rotor test rig. The results show the dry lubricated bushing type auxiliary bearing system provides safe operation with a significant margin of safety.

1 INTRODUCTION

Magnetic bearing technology has gained worldwide acceptance for use in high-speed industrial rotating equipment. Turboexpanders, high-speed turbocompressors, and large high-speed electric drives are a partial list of successful magnetic bearing applications. There are now many established end users and rotating equipment OEMs constantly demonstrating their confidence in the technology by steadily expanding the population of magnetic bearing equipped machines. It is not surprising that these established markets are motivating other OEMs who as yet have not applied magnetic bearings to their products to design in magnetic bearings in their next generation machines.

New users of magnetic bearing system must become acclimatised to the differences between magnetic bearing systems and traditional bearing systems. Auxiliary bearings are a vital part of the overall magnetic bearing system. The auxiliary bearings form a system to protect the rotating machine internal components and the magnetic bearing components from damage during rotor delevitation events. Although the magnetic bearing system power supplies are backed up with an UPS, there are low probability events where the power for the magnetic bearing system is lost and the rotor is allowed to delevitate and subsequently coastdown on the auxiliary bearings. The auxiliary bearings also protect the bearing components and machine internals from damage if an overload condition occurs.
The majority of WMB magnetic bearing systems include an auxiliary bearing system based on a dry lubricated bushing technology. New users may be surprised that a dry lubricated bushing is used as an auxiliary bearing given their awareness of references in the technical literature about resultant rotor behaviour due to dry friction rubs. These references include Ehrich (Ref.1) and Crandall (Ref.2). The technical literature cites cases where dry friction rubs within rotating machines have precipitated backward whirl type rotor behaviour. Large rotor vibration amplitudes due to fully developed backward whirl type rotor motion can cause severe damage or even complete destruction of the rotating equipment. The concerns about backward whirl rotor motion warrant actions to demonstrate the safety of transient operation of rotating equipment where the rotor is fully supported by bearings with a dry friction characteristic.

All WMB applications in turboexpanders, high speed turbocompressors, large high speed electric drives, and high speed test rigs have completed successful auxiliary bearing rotor delevitation tests. These applications include the NAM GLT motor compressor strings shown on Figure 1 and described in Reference 3.

The rotor delevitation testing on these applications have been comprehensive. The tests have included delevitating all axes of the magnetic bearing system causing the rotor to impact all auxiliary bearings in the delevitation system, and allowing the rotor to coastdown from maximum operating speed to zero speed. It is typical for the lead machine in a product line to go through a complete set of auxiliary bearing landing and coastdown tests to verify the safety and agreed performance of the system. This paper documents this specific test and verification process for a WMB high speed test rig.

This paper describes the mechanical details of the high speed test rig used for the verification tests and design elements of the dry lubricated bushing type auxiliary bearing system. Details of the method for the time transient analysis and calculated coastdown event results are presented. A comparison is shown between the measured and calculated results.

Details are covered for the non-linear time transient rotor dynamic simulation model used to generate the analytic predictions for the coastdown event. Predictions include rotor displacement, and coastdown time. The simulation model includes detailed representations of the unique auxiliary bearing design. The model includes all key parameters having the most influence on the coastdown transient. These parameters include auxiliary bearing friction coefficient, rotating speed, imbalance distribution, foundation stiffness as well as the initial conditions.

Elements of the verification process covered in the paper include the analytical calculation results of the predicted rotor motion during the rotor delevitation events and the results of the full scale rotor delevitation tests completed with the 8000 RPM rotating plant machine.

The paper presents the coastdown event test results, which were recorded with the high-speed data acquisition system resident in the magnetic bearing system. The rotor delevitation events are initiated by switching off all magnetic bearings and the drive power to the motor. Delevitation events were initiated from several rotating speeds including the maximum rated operating speed of 8000 RPM. Rotor displacements, and coast down times were measured and compared with simulation results.

The results of the model benchmarking against the test data are presented. The benchmarked model is used to generate simulation results for the significant possible fault conditions and
key parameter variation studies. Key parameter margins to any backward whirl behaviour are presented.

2 DESCRIPTION OF TEST RIG AND AUXILIARY BEARINGS

The full-scale test article for the verification process is an 8000-RPM rotating plant machine with a 1.5-ton supercritical rotor fully levitated by magnetic bearings during normal operation. The test rig drive is a 35 kW variable speed electric motor coupled to the test rig with a flexible coupling. The electric motor drive is mounted to a separate frame to isolate any motor vibrations from the test rig instrumentation system. A photograph of the test rig is shown on Figure 2 and a drawing of the test rig showing a cross sectional view is included on Figure 3.

The magnetic bearings are housed in a pedestal type bearing housing with two pedestals mounted to a base frame. The radial magnetic bearings are identical at each end of the machine and have a stator bore diameter of 303 mm and an axial stack length of 124 mm. The axial magnetic bearing actuator is comprised of a stator and collar located at each end of the machine, inboard of the radial magnetic bearings.

The radial auxiliary bearings are identical at each end of the machine with a stator bore diameter of 260 mm and an axial length of 110 mm. The radial auxiliary bearings are located outboard of the radial magnetic bearings at each end of the machine. The radial auxiliary bearing is a dry lubricated bushing type design. The radial auxiliary bearing components are the non-rotating stator part fixed to the casing of the machine and the bearing landing sleeve fixed to the shaft of the machine. An engineered matching of the materials for the stator pad material and the rotor landing sleeve surface yields an optimum sliding coefficient of friction to maintain good rotordynamic performance.

The radial auxiliary bearing stator is comprised of multiple articulated pads. A photograph of the radial auxiliary bearing stator is shown on Figure 4. Each pad is made of a dry lubricated bushing material. The bushing material is a sintered matrix with embedded lubricants. The mixture of several types of embedded lubricants in the bushing material ensures a consistent coefficient of friction over the load and temperature operating range. The specific material composition of the sintered matrix is also matched to the required load and temperature operating range.

The articulated pad design provides features that affect the rotor dynamic performance of the machine during a landing event. The articulation provides;

- Provision for pad alignment to shaft slope
- Energy dissipation and attenuation of impact force between pad and landing sleeve due to a rotor delevitation
- Preload force applied to pad
- Adjustable stiffness of compliance travel
- Adjustable damping of compliance travel

The axial auxiliary bearing is located at the non-drive end of the machine, outboard of the non-drive end radial auxiliary bearing. The axial auxiliary bearing is comprised of a thrust collar mounted to the shaft and dual acting bushing type thrust washers fixed to the non-drive end bearing pedestal. The thrust collar outside diameter is 380 mm. The bushing material for the thrust washers is identical to the dry lubricated material used for the radial auxiliary bearing.
pads. Although there are virtually no externally applied thrust loads to the machine, the axial auxiliary bearing is generously sized. The axial auxiliary bearing fulfils two unique functions for this test rig, these are;

- Provide supplemental braking torque during auxiliary bearing coastdown tests.
- Provide sufficient lateral force coupling to benchmark analysis of these effects.

It is typical of test rigs to have less braking available than machines in industrial service. An example of this is the high working pressures for turbocompressors provide a large aerodynamic braking torque if the drive power is disabled. The test rig drive motor is sized to overcome simple rotor windage and the very low bearing losses so a fast braking time is not possible with motor braking only. To achieve representative coastdown times for machines in industrial service an additional braking torque is intentionally generated by the axial auxiliary bearing during the test coastdowns.

The supplemental braking provided by the axial auxiliary bearing is achieved by enabling the axial magnetic bearing to place an axial load on the shaft during the test coastdown events. The applied axial load causes rubbing contact at the axial auxiliary bearing and this provides a braking torque. The axial auxiliary bearing thrust collar is sized with a larger than typical outside diameter to increase the braking torque capacity.

In addition to the supplemental braking, loading the axial auxiliary bearing simulates conditions in industrial applications where external thrust loads will be applied to the auxiliary bearing system if power is lost to the bearing system. Shaft slope effects at the thrust collar combined with the axial load can cause coupling between the axial auxiliary bearing and the lateral shaft behaviour during coastdown events. The larger than typical thrust collar diameter magnifies this effect and allows a benchmarking of analytic tools to quantify the coupling to the lateral behaviour.

3 TIME TRANSIENT SIMULATION AND CALCULATED RESULTS

A full finite element representation of the rotor is used to generate the degrees of freedom for the rotor model. Additional equations of motion are considered for the radial and axial auxiliary bearings to generate the additional degrees of freedom to describe the non-linear behaviour of these components. Additional equations describing the rotational energy of the entire rotor are used with models of any external braking torques and the braking torques developed by the auxiliary bearing contact forces to calculate the rotational speed at each time step. In this way the model generates a speed versus time curve as well as the complete response of the rotor during a coastdown event.

The time integration method for the transient analysis is the Newmark beta method. The time step size is controlled by several parameters including the rate of change in the relative velocity in the friction model and the rate of change of the contact forces in the non-linear contact models.

The non-linear radial auxiliary bearing characteristics are fully considered by the model. Figure 5 shows a diagram describing these non-linearities. The contact model for the radial auxiliary bearing landing sleeve and dry lubricated pad material considers Hertzian contact theory, friction effects, and rolling resistance effects.
As Figure 5 shows, the motion of the auxiliary bearing pad is also calculated. Since the pads are compliantly mounted, additional coordinates describe the translation and rotation of the individual pads. These coordinates are required to transform the contact forces for accurate summation with all other external forces applied to the rotor.

Figure 6 shows a diagram that illustrates the elements of the axial auxiliary bearing that cause coupling to the lateral forces applied to the rotor during a coastdown event. The fixed stator part of the axial auxiliary bearing has a mass-elastic characteristic due to the mounting stiffness and the mass of the stator. Effects of static non-perpendicularity of the stator surface relative to the shaft centreline can also be modelled. The rotating shaft collar for the axial auxiliary bearing can also cause non-perpendicularity due to shaft angularity or a static misalignment to the shaft. Non-perpendicularity between the collar and the stator surface causes a non-uniform contact pressure across the collar-stator contact zone. This non-uniform contact pressure results in a corresponding non-uniformity of the tangential friction forces. The non-uniform tangential friction forces cause a resultant lateral force to be applied to the rotor. In this way, application of axial loads can cause lateral forces that affect the lateral rotor behaviour.

The time transient model also includes representation of the magnetic bearing effects. The transients are started with the rotor in the ‘levitated’ condition where the rotor is supported at the magnetic bearing actuator centrelines. The rotor is ‘operated’ in this condition for a short period of time before initiating the coastdown event to allow the rotor to settle into the static deflected shape due to gravity loads as well as the dynamic displaced shape due to imbalance loads. The postulated failures are initiated by disabling the required portions of the magnetic bearing support.

Figure 7 shows a calculated orbit plot for the non-drive end bearing location of the high-speed test rig for a delevitation transient with an initial rotating speed of 7780 RPM. Looking at Figure 7, the direction of rotation for the rotor is counter clockwise. This simulated transient is initiated by disabling all axes of the magnetic bearings simultaneously.

4 VERIFICATION PROCESS

There are many design features and performance parameters that determine the rotor behaviour during an auxiliary bearing coastdown event. Rotor design considerations include the location of the auxiliary bearing on the machine rotor determining bearing span, natural frequencies and node locations. The bearing foundation characteristics can have a significant impact on the rotor dynamics. The balance quality of the rotor has a major influence initiating rotor motion that may propagate to backward whirl. The important auxiliary bearing characteristics are coefficient of friction, coefficient of rolling resistance, clearances, and the stiffness and damping characteristics. The core of the verification process for the auxiliary bearings is to confirm the coefficient of friction and the stiffness and damping characteristics by test and analysis.

The coefficient of friction is verified in a multitude of ways; some methods are more direct than others, but all contribute to the verification of the coefficient of friction and performance of the auxiliary bearings. The analyses and tests form a matrix of results that are cross-referenced to verify a consistent characterisation of the bearing. The verification methods include;
Component rig testing: Individual component testing is completed. Test rigs were built for the axial auxiliary bearings and the radial auxiliary bearings. In these rigs, dynamics external to the auxiliary bearing are minimized. These individual component test rigs impose representative energy input, sliding velocities and loads. The coefficient of friction is determined from the measurements made on these rigs. Wear and other performance characteristics can also be measured with these rigs.

Physical torque measurements were made on the rotor with the system delevitated. For these tests the sliding speed is of course quite low. The bearing static reactions are known and the torque to turn the rotor can be measured with a high accuracy. The coefficient of friction can be backed out of this torque measurement.

Low speed coastdown of entire rotor on the auxiliary bearings without any external braking sources. The rotor is run up to a slow rotational speed (~1500 RPM) where rotordynamic effects such as gyroscopics and mode traverse effects are minimized. All external braking sources such as electric motor braking is eliminated and aerodynamic effects are minimized. The speed versus time curve is measured. The initial rotational energy is known so the coefficient of friction can be calculated from this measurement.

The landing sleeve temperature is measured directly following a coastdown event. The rotor is run-up to a high speed and the rotor is delevitated. After the landing is complete the rotor is immediately re-levitated to prevent conductive heat transfer to the fixed stator part of the auxiliary bearing. The temperature of the landing sleeve is measured at strategic points. The energy input into the sleeve is dependent on the coefficient of friction. The energy input into the sleeve can be backed out of a straightforward finite element transient heat transfer analysis of the landing sleeve and shaft. The measured speed versus time curve, measured vibration data (indication of bearing loading history), and postulated coefficient of friction are used as inputs to the finite element analysis. The match between the calculated sleeve temperature and measured sleeve temperature is verification of the coefficient of friction.

The time transient rotordynamic code calculates the complete rotor and auxiliary bearing response. The load profile on the auxiliary bearing during the landing event is determined from the response calculation results. The external braking is measured by performing a coastdown without landing the rotor. The postulated coefficient of friction, the measured external braking and the calculated auxiliary bearing loading is used to determine a speed versus time curve that can be compared to the actual.

Zero speed landings were performed to check the damping and stiffness in the system by measuring frequencies and amplitude decay of the rotor displacements measured by the magnetic bearing system position sensors. From these results, the inherent damping in the machine housings and rotor and the stiffness and damping of the auxiliary bearing compliance can be ‘tuned’ to yield more accurate analytical results.

The time transient rotordynamic code calculates the complete rotor and auxiliary bearing response. In general, increasing the coefficient of friction will cause the rotor response to increase during a coastdown event. Comparison of the magnitude, frequency content and characteristic of the rotor response presents an indication of the actual coefficient of friction.
5 MEASURED RESULTS AND COMPARISON TO CALCULATED RESULTS

Figure 8 shows a measured orbit plot for the non-drive end bearing location of the high-speed test rig for a delevitation transient with an initial rotating speed of 7780 RPM. Looking at Figure 8, the direction of rotation for the rotor is counter clockwise. This delevitation transient is initiated by disabling all axes of the magnetic bearings simultaneously.

Comparing Figure 7 and Figure 8 shows the time simulation model produces a good representation of the actual tested delevitation transient. Both figures show a 5 second time period of the transient, which contains the maximum rotor displacements occurring during the transients. Plotting of further data from the transients obscures the details of the initial impacts. The figures show the response during the initial impact and the displacement decaying with the rotor behaviour not showing any tendencies to propagate into backward whirl. The figures show rotor behaviour that is also typical for all other machines tested. Characteristics of this typical rotor motion are all rotor motion contained within the lower half of the auxiliary bearing with no tendencies to climb into the upper half region of the auxiliary bearing, the maximum displacement occurring shortly after the initial impact and rotor motion with a small horizontal displacement band extending into both sides of the bottom dead center region of the auxiliary bearing.

6 CONCLUSIONS

1. Full scale auxiliary bearing coastdown tests have been completed on a variety of high speed rotating equipment including turbo expanders, high speed supercritical turbocompressors, large high speed supercritical electric drives, and a supercritical test rig. A dry lubricated bushing type auxiliary bearing technology has been utilized in the machines for all these tests. In every case there has been no evidence of backward whirl rotor behaviour during the coastdown tests and the safety of operation during these coastdown tests has been verified by determination of a substantial factor of safety.

2. Several distinct verification tests to determine the coefficient of friction of the WMB dry lubricated bushing material have been completed. The tests have included individual component tests and full-scale supercritical rotor coastdown tests. The calculated dynamic coefficient of friction determined from the high speed tests has always been found to be less than or equal to the dynamic coefficient of friction determined from low speed tests. Therefore, the low speed test results provide an important indication of the safety of the system for high-speed delevitation events.

3. Verification tests and benchmarked time transient simulation model parameter sensitivity studies have shown that rotor imbalance levels and auxiliary bearing coefficient of friction are first order parameters in determining threshold conditions for backward whirl rotor behaviour. For the group of machines for which testing has been completed, typical threshold levels for higher probability of backward whirl rotor behaviour is a dynamic coefficient of friction twice that of the worst measured case with an imbalance distribution level of G4.0. This places the coefficient of friction for the auxiliary bearing material tested at a factor of safety greater than 2.0 for avoidance of backward whirl rotor behaviour during coastdown events.

4. A time transient simulation analysis method has been benchmarked against the test results from the high speed test rig. This analysis method considers distinct degrees of freedom for the
rotor, radial and axial auxiliary bearings, and bearing supports. The analysis method has produced representative results compared to the measured test results for the rotor behaviour during the auxiliary bearing coastdown events.

5. With applied thrust loads the motion of the axial auxiliary bearing thrust collar can generate lateral forces that influence the lateral behaviour of the rotor during auxiliary bearing coastdowns. This effect should be considered in lateral analyses of rotor behaviour during auxiliary bearing coastdown events.

These results present vital data for rotating equipment manufacturers, and end user decision makers considering the use of magnetic bearing systems in all types of high speed rotating equipment.

7 REFERENCES

Figure 2- High Speed Magnetic Bearing Test Rig

Figure 3- Cross Section View of High Speed Magnetic Bearing Test Rig
Figure 4- Radial Auxiliary Bear

Figure 5 Diagram for Radial Auxiliary Bearing Model
Figure 6- Diagram of Modeling Considerations for Axial Auxiliary Bearing

Figure 7- Simulation Result for Delevitation Transient from 7780 RPM
Figure 8- Test Result for Delevitation Transient from 7780 RPM
High-speed direct driven turbo blower

Stefan Henneberger
Atlas Copco Airpower n.v., Belgium

ABSTRACT

The efficiency level of traditional centrifugal compressors using standard induction motors is mainly penalised by the mechanical losses of the speed increasing gear sets. In order to achieve an improvement of the compressor efficiency, the elimination of the gearbox in the drive system is the logical step in the development of the future compressor generation.

A single stage low-pressure turbo compressor has been realised by implementing direct drive technology using a high efficient permanent magnet motor and magnetic bearing technology. The impeller is directly mounted to a high speed permanent magnet synchronous motor (PMSM). The PMSM is controlled by a fast switching water-cooled frequency converter and is equipped with magnetic bearings in order to guarantee a contact-free operation without any friction or vibrations. The overall drive efficiency (shaft power versus power unit input) taking into account all auxiliary losses for cooling results in 92.2 %.

Besides on the efficiency, the development focus has been put on the robustness, the reliability and the easy handling of the compressor. The machine is designed to operate at 45,000 rpm with 175 kW shaft power (24/7 cycle, at 40° C ambient and cooling water-temperature of 38° C).

1. THE UNIT

The entire unit is shown in fig. 1. The electric cubicles (fig. 1a) and the compressor / motor module are explained below. The air-flow path is extremely short in order to avoid pressure drops. The air is drawn through the noise baffling and the filters into the shroud of the compressor. The compressed air leaves the volute through the check valve to the customer installation. During the start-up and at very low consumption the air is (partly) blown-off through a silencer.

Additionally an internal cooling water system to cool the housing of the electric motor and the heat sink of the water-cooled frequency converter is provided. A solenoid controls the external cooling-water flow to the heat exchanger.
Applying the new technology to a fully integrated compressor package results in a number of advantages.

- high efficiency: by eliminating gear box losses and applying permanent magnet motor technology.
- additionally energy savings are achieved using the variable speed drive to adjust the customer demand.
- totally oil-free compressor system: there is no single drop of oil in the entire unit.
- extremely low noise level: there is no mechanical contact, air noise is limited by silencing and baffling.
- almost maintenance free: as there is no wear of moving parts, the required service tasks are limited to the exchange of filters or the verification of the cooling-water circuit.
- very compact system: the required floor space is about one third when compared to compressors delivering the same flow/pressure by using classical drive systems.
- no foundation required: the specific weight of the unit is very low, therefore the system will not distribute any vibrations to the surroundings, thanks to the magnetic bearings.
- controllability: the presence of a frequency converter gives the possibilities to apply either pressure control mode or speed (flow) control mode.
- availability: the number of starts and stops is unlimited.
2. THE COMPRESSOR / MOTOR MODULE

As the gear box is to be eliminated and the impeller is directly mounted on the shaft of the electric motor, the two components compressor and electric motor have to be regarded as one integrated compressor / motor module (figure 2).

The stainless steel backward lean impeller designed with a labyrinth seal against leakage is mounted to the vertically positioned rotor. The PMSM has a classical two pole three phase stator and a rotor with surface mounted permanent magnets. The magnetic bearings (two radial bearings, one axial bearing) are split over two identical bearing cartridges with each a radial bearing and half an axial bearing. The bearing cartridge carries the back-up ball bearings.

![Fig. 2: Compressor / motor module](image)

3. LC-FILTER

In order to achieve a high energy density of the motor the stator losses of the motor are dissipated by a very effective water-cooled housing. However, there are certain heat sources which cannot be reached by water-cooling: the mechanical air-friction losses at the rotor surface, the losses of the magnetic bearings, compression heat which is conducted from the impeller to the motor shaft and furthermore, some of the compressed air leakage is dissipated into the motor as well. These losses are dissipated by two small gas blowers which receive filtered air from the air inlet system.

Losses in the permanent magnets would further increase the temperature in the rotor rapidly because the heat is not easily dissipated out of the rotor. Therefore, the current wave form of
the stator winding has to be close to sinusoidal, in order to limit the harmonic losses in the permanent magnets. A sinusoidal current waveform could be achieved by increasing the switching frequency of the IGBT’s. However, the switching frequency is limited by the thermal capacity of the IGBT’s.

Consequently, the sinusoidal current waveform has to be achieved by using a LC-filter (sine-filter). For the C-part of the filter standard AC-voltage capacitors are used. The inductance (L-part) however is a custom made part. Due to the high switching frequencies, the core cannot be standard laminated, but has to be designed out of a special sintered iron powder composite.

The graph (fig. 3) indicates the effectiveness of the sine filter. The current wave form of the converter output is given compared to the current wave form of the motor input.

4. ELECTRIC CABINETS AND EMC

Regarding the high switching frequency of the IGBT’s, special attention of the development has been put on the conformity of the EMC standards, the EN50081-2 (1993, EMC-emission) and EN50082-2 (1995, EMC-immunity).

The electric equipment is separated over two electric cabinets, a control cabinet, the “clean” part and the power electric cabinet which is from the EMC point of view the “dirty” part.

In the control cubicle all control functions are connected to each other: the overall unit controller.
(Atlas Copco Elektronikon), the externally located control panel of the frequency converter, the magnetic bearing controller as well as switches, relays, fuses, etc.

The power electric cubicle with the drive components: RFI filter, AC choke, water-cooled frequency converter, the C-filter capacitors and the L-filter.

The RFI filter is positioned in-between the two cabinets forming the cable entry to the AC choke. The RFI filter is perfectly grounded and there are further more no significant openings to the outside of power electric cabinet. The power cables to the motor are shielded with a metal conduit.

The EMC measurements of the radiated and the conducted emission provide extremely good measurement results. Figure 5 shows the obtained measurement results for conducted and the radiated emission together with the maximum level allowed (dotted line).

5. UNIT SAFETY

The unit is controlled, monitored and protected by the overall Atlas Copco Elektronikon controller. It handles safety alarms and shut downs in function of temperature and pressure readings and the statuses which are communicated by the frequency converter and the magnetic bearing controller.

In case of an interruption of the main power supply while the motor is running at high speed, an energy buffer is required in order to ensure a safe ramp down of the rotor.

An uninterruptible power supply (UPS) would have been an expensive addition and requires some space for additional components. Instead, a wide input range (WIR) DC/DC converter is used. The WIR transforms voltages between 60 and 800 V\textsubscript{DC} into the required bearing controller power supply of 120 V\textsubscript{DC}.
The energy stored in the DC bus of the frequency converter can be used in order to provide the power which is needed to levitate the motor until it has reached a relatively low speed to full into the back-up bearings.

However, there is the theoretical possibility that the bearing controller fails. In this case the back-up ball bearings have to protect the unit from damages. They are dimensioned to catch the rotor for a certain number of unpredictable failures, like the total loss of control or power of the bearing controller while running at maximum speed.

A shut down of the main power supply while using the DC bus power is monitored in figure 7. The DC bus voltage, the rotor speed and the output voltage of the WIR are given. The graph shows that the WIR provides an output voltage for about 32 s. In this period, the rotor is slowing down from 37,000 rpm down to about 1,000 rpm before it is caught by the back-up bearings.
Fig. 7: Rotor speed, DC-bus voltage and WIR output voltage monitoring an emergency stop.

6. EFFICIENCY

In order to compare the advantages of the direct drive technology to the conventional gear drive technology of turbo compressor. The auxiliary losses of the drive system have to taken into account.

The efficiency of the high-speed motor has to be extremely high (97.5 %). Otherwise it would be impossible to make the motor as compact as it is design in this application. The efficiency of the frequency converter is regarding the additional losses due to the high switching frequency the achieved value of 96-97 % very high as well.

The auxiliary losses of the components for the direct drive are small: A cooling-water circulation pump (ca 200 W), two air cooling fans (gas blowers of each 350 W) and the power supply to the bearing controller (200 – 400 W).

The efficiency data of the conventional single stage gear drive are based on measurements on various compressors from different brands. The comparison of the drive systems is given in Sankey diagrams in figure 8.
The high-speed direct turbo has been presented. The machine set-up is indicated and the advantages of the direct drive system are explained. The main target of using direct drive technology, the improvement of the overall efficiency is shown.

**Fig. 8a:** Sankey diagram for a conventional single stage turbo compressor drive.

**Fig. 8b:** Sankey diagram for a single stage turbo compressor high efficient direct drive.

7. **SUMMARY**

The high-speed direct turbo has been presented. The machine set-up is indicated and the advantages of the direct drive system are explained. The main target of using direct drive technology, the improvement of the overall efficiency is shown.
MODELLING
Numerical and experimental analysis of counterflow vortex tube

Mladen Andrassy, Severino Krizmanic, Zdravko Virag
Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Croatia

ABSTRACT

The paper offers an analysis of the flow and thermal process in a counter-flow low-pressure vortex (Ranque - Hilsch) tube. The FLUENT software package is used for the computation of compressible laminar and compressible turbulent flow. The obtained data are compared with experiments performed for this purpose. An explanation of the obtained results is given in the light of published theories on the thermal separation in vortex tubes. Thermal separation is also recorded at laminar flow conditions, indicating that turbulence is not the only reason for it. The calculation results are in qualitative accordance with the experiment, but quantitatively they forecast somewhat lower thermal separation effects.

1 INTRODUCTION

Discovered by G.J. Ranque in 1933. (1), and first systematically investigated by R. Hilsch (2), the vortex tube represents an extremely simple device enabling the separation of a gas stream of total temperature \( \vartheta_{\text{in}} \) into two streams, one of which is cold \( (\vartheta_{\text{c}}<\vartheta_{\text{in}}) \) and the other warm \( (\vartheta_{\text{w}}>\vartheta_{\text{in}}) \). This is usually achieved in a counterflow vortex tube, Fig. 1 a (parallel flow models are also possible), where the gas stream is introduced tangentially adjacent to the diaphragm. The produced swirling flow is then separated by throttling the stream flowing from the diaphragm, thus forcing one part to emerge through the diaphragm as the cold stream, while the throttled stream is warm. The realized cooling \( \Delta \vartheta_{\text{c}} = \vartheta_{\text{in}} - \vartheta_{\text{c}} \) and warming \( \Delta \vartheta_{\text{w}} = \vartheta_{\text{w}} - \vartheta_{\text{in}} \) may be rather spectacular, depending principally on the separation ratio \( \gamma = \frac{m_{\text{c}}}{m_{\text{in}}} \) of the cold stream mass flow \( m_{\text{c}} \) and the input mass flow \( m_{\text{in}} \) and the expansion ratio \( \frac{p_{0\text{in}}}{p_{\text{c}}} \). Fig. 1 b presents the results obtained by Hilsch, who used compressed air, always at over-critical expansion ratio of inlet total pressure and cold outlet static pressure \( \frac{p_{0\text{in}}}{p_{\text{c}}} \geq \left( \frac{2}{(\kappa+1)} \right)^{(\kappa-1)} = 0.53 \). It may be seen that, depending on the input pressure, the maximum cooling of the air is between 30 and 55 °C for \( \gamma \)-values 0.3 to 0.4.
If the vortex tube is regarded as a cooling device, its COP (ratio of obtained cooling capacity and input energy) is ranging at values around 0.1, which increase with the drop of the expansion ratio (3). These are rather low in comparison to standard cooling processes.

Since the invention of the vortex effect several attempts, more or less unsuccessful, were made to explain it (1, 2). Among the interesting ones is the interpretation by Schultz-Grunow (4), who proposes that the turbulence at the boundary of the central low pressure cold flow layers and the periphery high pressure warm flow layers performs vortices that may be regarded as heat pumps expanding gas particles from the warm to the cold areas and again compressing them in the opposite direction. An explanation of this phenomenon and a proposition for modelling the turbulent diffusion was given later by Hinze (5).

A series of experiments with a low-pressure vortex tube were performed in order to prove that the temperature separation occurs even at subcritical flow conditions and to investigate its efficiency (3). The obtained results confirmed both, the mere existence of the thermal separation shown in the diagrams in Fig. 2, and better COP-s. The air cooling and warming obtained is rather low in comparison to the high-pressure tube, but the COP values were nearly doubled. The performed investigations did not yield an explanation of the physical nature of the phenomenon.

Therefore the present paper, in which commercial CFD software is used for numerical simulation of the phenomena in counterflow vortex tube is performed, and the obtained results compared with the experimental one. In addition the paper intends for a better insight in the flow and temperature patterns in the vortex tube, in order to contribute to the explanation of their origin.
2 DESCRIPTION OF THE PROBLEM

The present work analyses a low-pressure driven counterflow vortex tube, described and experimentally investigated in detail in (3). From the entire experimental arrangement, here only the vortex tube model used has been taken for the numerical simulation. It is schematically shown in Fig. 3.

The experimental vortex tube had an air inlet formed by eight nozzles, symmetrically arranged at the circumference adjacent to the orifice plate, inclined at 25° to the tangential direction. This allows the supposition of uniform flow velocity profile continuous feed for the numerical simulation, i.e. steady axially symmetric swirl flow.
3 NUMERICAL SIMULATION DETAILS

The numerical simulation of the flow in the vortex tube was performed using the commercial software FLUENT, and the results presented were obtained using the standard $\bar{k} - \bar{\varepsilon}$ turbulence model in combination with standard wall functions. Other available models from the FLUENT package, including the Reynolds Stress Model, did not yield better agreement with the measured integral parameters.

Because of the axial symmetry of the vortex tube, only one half of the tube was modelled using 23814 two-dimensional rectangle control volumes. The air was treated as ideal gas with following properties: specific heat capacity $c_p = 1006.43 \ J/(kg \cdot K)$, thermal conductivity $\lambda = 0.0242 \ W/(m \cdot K)$ and dynamic viscosity $\mu = 1.7894 \cdot 10^{-5} \ Pa\cdot s$.

According to the described model, the segregated solver for the case of steady axially symmetric swirl flow with double precision option was selected, and the SIMPLE algorithm used with the standard scheme for the pressure gradient calculation. The second order upwind scheme was chosen for the discretisation of all other equations.

The boundary conditions are:
- At the air entry to the vortex tube the total pressure and the total temperature is given. In all cases the flow direction is at 25° to the tube circumference tangent, and a 10% input turbulence intensity with a characteristic turbulence length scale of 0.1 mm is supposed.
- The tube wall and the diaphragm surface are no-slip, hydraulically smooth walls. At the cold stream outlet the zero gauge pressure is imposed.
- The warm stream outlet is of the outlet vent type, where the zero gauge pressure and the loss coefficient, controlling the total flow rate and separation ratio, are given. At this boundary the tangential velocity still exists, so the condition of radial equilibrium for the pressure distribution is applied. Using this approach the problem of reversed flow was avoided.

The calculation procedure was stopped when the mass average of total temperature at each of the outlet borders ceased to change within five significant digits.

4 RESULTS AND DISCUSSION

For the purposes of this paper the simulations of flow for a series of vortex tube diaphragm diameters according to Fig. 3 and several feed air total pressures were performed. For reasons of succinctness, here only the results for the 26.5 mm diaphragm opening and 16129 Pa feed air gauge total pressure will be shown as a characteristic example.

Table 1 gives the comparison of measured and calculated data on achieved air cooling $\Delta \vartheta_c$, heating $\Delta \vartheta_w$ and COP for four separation ratios $\gamma$. The calculated temperature differences refer to mass average total temperatures at the tube exits. Fig. 4 represents the corresponding graphical interpretation.
The data in Table 1 and Fig. 4 indicate a fair qualitative agreement of experimentally and numerically obtained results, but the thermal separation effect is quantitatively somewhat underestimated by the numerical calculation. The same trend was found at other input parameter combinations. This may be the consequence of the turbulence model, where the impact of pressure gradients was neglected when modelling turbulent diffusion of energy.

Hereinafter the obtained flow pattern obtained by numerical simulation will be analysed, in order to attempt an explanation of the thermal separation mechanism. Figure 5 represents the projection of streamlines over the field of: a - static temperature distribution $\vartheta_s$ and b - total temperature difference distribution $\Delta \vartheta = \vartheta - \vartheta_{in}$, for the axial cross section of the tube. Only the part of the vortex tube adjacent to the diaphragm and the air entrance nozzles is shown, where the physical phenomena essential for the thermal separation occur.

At the warm tube side (right to the diaphragm), in the vicinity of the feeding nozzles, a recirculation zone is seen. There is a streamline dividing the fluid stream into the part that will flow to the warm side (to the right), and the part flowing to the cold side. The gas particles that are going to exit at the cold side, first flow in the direction of the warm side on the right, and after that they turn back to the left, in the direction of the cold side exit. The static temperature distribution in Fig. 5 a clearly indicates that the particles that are to leave the tube as the cold ones are cooled both at their way to the warm side and back to the cold side. The heat exchange will be maximal in the area where the angle between the streamlines and the static temperature isotherms is minimal, which is the case in the cross sections of the recirculation zone. The total temperature difference distribution in Fig. 5 b, on the other hand, proves that the heat is transported from the areas of lower total temperature to those with higher total temperature.
Fig. 5 Streamlines, static temperature and total temperature difference at $\gamma = 0.3$ (Table 1)
From the pressure distribution profiles in Fig. 6 it may be seen that in the cross section of the recirculation zone center ($z = 0.01$ m) there exists a high pressure gradient in the radial direction, which means that the heat transport occurs from low pressure to high pressure areas. It must be noted here that the applied turbulence model did not take into account neither the impact of pressure gradients on turbulent energy diffusion, as proposed by Schultz-Grunow (4) and described by Hinze (5), nor the turbulence anisotropy, which might be significant in this problem. According to the original proposition, thermal separation is a consequence of turbulence, i.e. it occurs at the level of turbulent eddies, while the presented data are obtained at the level of the main air flow. In the accepted mathematical model the turbulence is reflected only through the increased and spatially variable effective viscosity and thermal conductivity. Therefore the question is, whether the same effect is going to exist for laminar flow, where the viscosity and thermal conductivity are constant.
\( d = 26.50 \text{ mm} \)
\( p_{0\text{ in}} = 16129 \text{ Pa} \)
\( \vartheta_{\text{ in}} = 20.85 ^\circ \text{C} \)

<table>
<thead>
<tr>
<th></th>
<th>( m_{\text{in}} )</th>
<th>( \gamma )</th>
<th>( \Delta \vartheta_{\text{c}} )</th>
<th>( \Delta \vartheta_{\text{w}} )</th>
<th>COP</th>
</tr>
</thead>
<tbody>
<tr>
<td>num</td>
<td>25.71</td>
<td>0.30</td>
<td>4.03</td>
<td>1.83</td>
<td>0.096</td>
</tr>
<tr>
<td>exp.</td>
<td></td>
<td></td>
<td>4.60</td>
<td>2.01</td>
<td>0.109</td>
</tr>
<tr>
<td>num</td>
<td>25.31</td>
<td>0.48</td>
<td>2.99</td>
<td>2.86</td>
<td>0.112</td>
</tr>
<tr>
<td>exp.</td>
<td></td>
<td></td>
<td>4.10</td>
<td>3.79</td>
<td>0.154</td>
</tr>
<tr>
<td>num</td>
<td>24.88</td>
<td>0.67</td>
<td>2.03</td>
<td>4.23</td>
<td>0.106</td>
</tr>
<tr>
<td>exp.</td>
<td></td>
<td></td>
<td>3.44</td>
<td>6.91</td>
<td>0.179</td>
</tr>
<tr>
<td>num</td>
<td>24.47</td>
<td>0.85</td>
<td>1.11</td>
<td>6.77</td>
<td>0.074</td>
</tr>
<tr>
<td>exp.</td>
<td></td>
<td></td>
<td>2.50</td>
<td>14.43</td>
<td>0.166</td>
</tr>
</tbody>
</table>

Table 2 Comparison of numerical and experimental results

Fig. 7 Graphical presentation of data in Table 2

6 LAMINAR FLOW IN THE VORTEX TUBE

In order to examine the influence of the increased thermal conductivity and its markedly non-homogeneous distribution at turbulent flow, a simulation of laminar flow was performed. Laminar flow for the same inlet conditions as in turbulent flow may be sustained only by introducing a "hypothetical air" with following properties:
\( \lambda = 13.738 \text{ W/mK} \) and \( \mu = 6.8413 \times 10^{-3} \text{ kg/ms} \). These values are obtained as mass average of the corresponding properties at turbulent flow, for given input pressure \( p_{0\text{ in}} \) and separation ratio \( \gamma \). Accordingly, in order to accomplish similar flow velocities, the zero wall shear boundary condition was imposed at the diaphragm and tube walls. From the results shown in Table 2 and Figures 7 and 8 it may be concluded that the thermal separation effect is achieved even in compressible laminar flow. Of course the obtained numerical data do not reflect reality because of the assumed unreal air properties, and unreal boundary conditions.

7 CONCLUSIONS

The paper presents a numerical flow analysis in a counterflow vortex tube. The results are compared with experimental data. The numerical results are qualitatively in good agreement with the experimental data, but they underestimate the effects of thermal separation, which may be addressed to the applied turbulence model. The described streamlines and temperature fields indicate the existence of the heat pump mechanism in the main flow, which leads to the conclusion that Schultz-Grunow’s turbulent heat pump mechanism is not the only origin for thermal separation of the flow. This is approved by the fact that the thermal separation was also found in laminar compressible flow through the vortex tube.
Fig. 8 Results of laminar flow simulation at $\gamma = 0.3$ (Table 2)
REFERENCES


The virtual compressor and the concurrent engineering environment

Fabian Fagotti, Marcos Giovani Dropa de Bortoli, Marcio Silveira, Raul Bosco Jr. Embraco - Empresa Brasileira de Compressores SA, Brazil

ABSTRACT
Considering the technical aspects to a major extent, developing a commercially successful compressor means finding the best trade-off between energy efficiency, noise level, reliability and cost, but there are a number of other issues which must be taken into account. This is specially true for the refrigerating compressor for the household appliance in the past two decades, but it has increasingly become a paradigm also for the whole refrigeration market. Achieving the optimal design is result of parallel analyses on different fields of engineering, not necessarily related to the optimum on each field. Use of computer aided engineering (CAE) has proven to be a very useful approach to speed up projects with optimum results. On the other hand it is also well known that CAE tools do not cover all the design variables and issues such as manufacture, assembly, existing equipment on each plant, production scaling-up, particular characteristics of each market, management of activities fragmented in different sites and others must be dealt with. This paper sheds some light on an approach adopted by a company with engineering resources fractionated in different global sites and devoted to the global market to optimize the design process of refrigerating compressors.

1. INTRODUCTION
In the late 80’s a significant conceptual transformation occurred in the way products were developed. The traditional “engineering cycle” used a component based optimization, focused on achieving the optimum for one single requirement considering a small set of specifications. Once this characteristic was optimized, another one was then taken into account, so the project resulted a sequence of local optima instead of a global optimum. Although some attempts to expand the integration have been made, most of them restricted to the CAE ambient, e.g., (1) and (2), some outside the boundaries of CAE but with a single objective, e.g., (3), (4), (5) and (6) a few integrating experimental and numerical analysis e.g., (7), the main goal is defining all the interfaces of the design process. In this scenario the sequence conceptualization, detailing, analysis, prototyping and testing defined the product design, which was then subject to a “cultural barrier” when transferred from product engineering to process engineering, when some characteristics could be lost due to lack or misusage of the information produced. The
process design resulted from the sequence process analysis, supply chain definition and process architecture and then the project was again subject to another “cultural barrier” when transferred from process engineering to marketing and production. Despite this approach increases the possibility of having the information corrupted due to the cultural differences among the several environments the project was subject, the main disadvantage is on dealing with changes. Due to mistakes or some restrictions not considered in the previous steps, changes are natural on a project, but the longer the chain from conceptualization to the point the change is determined, the bigger the impact on the project, thus the integration of different environments (product, process, marketing, etc.) is profitable not only to avoid barriers but also to decrease the impact of changes in the project.

Obviously the choice on this approach was not only due to a Cartesian way of thinking, but mainly to the limited power and possibilities of the available hardware and software. This sequential process was changed by a more efficient approach, called concurrent engineering, which aims, from the very beginning, the maximum integration of all the necessary fields of engineering involved in a development. Practically all the activities are started during the conceptualization phase. This concept began to be more appealing due to globalization, since most of the projects began to be developed, in an integrated way, with contributions from different sites around the world.

The kernel of this process, which in this specific case could be called “virtual compressor”, is a set of mathematical models which emulates the “real compressor” in the CAE/CAD environment. Within this concurrent engineering environment, the use of a “virtual compressor” becomes more and more important for the project’s success. The virtual compressor is strongly based on the use of CAE and CAD (Computer Aided Design) tools, allowing realistic simulations to be carried out before the construction of actual prototypes, significantly increasing the chances of having the first real compressor fully accomplishing the project goals. At a first glance, it just seems the practical translation of this scenario is that most of the work is done on computers; actually the virtual compressor works as a raw material to be modeled by people from research, design, engineering, laboratories, production, marketing and so on, playing the role of an integrator for different areas to put their weights on the optimization process. Different from the past, when the analyses were usually performed for each component alone and taking into account only one of its requirement-functionality relationships, nowadays the analyses are more focused on the system. The main available computer-based tools are related to structural analysis, electromagnetic, transport phenomena, thermodynamics and acoustic, all supported by a powerful CAD. The inter-relationship of all these tools allows the virtual modeling, where various alternatives are analyzed and optimized. Available tools as PDS (Probabilistic Design System) also allow the numerical analyses taking into account the actual dispersions in the properties of the materials, dimensions and loads, analyzing the real scene of manufacture, and not only nominal or extreme cases. Another important characteristic is such approach allows a huge number of “what-if” scenarios, which makes it easier to take into consideration features very difficult to deal with in a purely mathematical way, e.g. the ones related to cost, “manufacturability” and market driven restrictions. A general vision on the optimization process is depicted in figure (1). The main “technical” trade off is defined among structural, noise and efficiency requirements, which obviously vary a lot depending on the market.
2. COMPUTER AIDED ENGINEERING AND EXPERIMENTAL TECHNIQUES

Considering the flexibility and convenience to take the advantage of powerful optimization techniques, a complete virtual prototype would be the ideal. On the other hand, in “real world” applications it is clear that there are some drawbacks, as:

- current performance of solvers and hardware;
- robustness of the models related to ill posed problems;
- lack of deep knowledge on material properties;
- imperfect modeling when linking the results from different fields of analysis (e.g., FEM versus CFD);
- difficulty on determining how to input and balance the process and manufacture restrictions
- a virtually nonexistent precise approach on having the feedback from experiments automatically incorporated into the analysis.

In this scenario another ability required from the designer is deal with this imperfect interfaces and keep them on track to avoid the interference of this “noise” on the quality of the result, the optimum design.

Except for general application software (e.g., finite element analysis – FEM, computational fluid dynamics – CFD and acoustic analysis) all the simulation codes and testing benches and techniques henceforth mentioned have been developed in-house or through cooperation agreements with universities. Even in the last case, researchers from the company play a
important role on the development. This approach is justified considering mainly two facts. First, applications are very specific, so most of the solutions are not “drop-in”; a commercial program or workbench only becomes profitable with a deep knowledge on how it works and on the application itself. Besides, and more important, there is a common sense that aggregating research and development is the most effective way to transfer knowledge and technology, avoiding the use of “black-box like” tools.

The interest on simulation emerges from the lag between the theoretical and empirical knowledge on compressor design. For a long time, most of the design rules used were based on previous experience, try-and-cut or benchmark; respectively incomplete, tedious and limited to whom looks for leadership and continuous evolution. However, CAE tools do not allow to renounce to experimentation; the concurrent use of both approaches is clearly profitable, since one supplies the other’s deficiencies. Using CAE always leads to the doubt about how far the mathematical model is from reality, nevertheless they are much more proficient on the quality of the information engendered as well as malleable in considering different boundary conditions. It means the possibility of dealing with more scenarios to be evaluated and promptness in understanding physical phenomena. The balance between investing in proprietary or commercially available software is an important issue to be considered; benefits and drawbacks are resumed in table (1).

### Table (1) – Commercially available packages versus in-house software

<table>
<thead>
<tr>
<th></th>
<th>positive</th>
<th>negative</th>
</tr>
</thead>
<tbody>
<tr>
<td>commercial</td>
<td>- no developing lead time</td>
<td>- black-box, weak points barely known</td>
</tr>
<tr>
<td></td>
<td>- easy to maintain technology updated</td>
<td>- high cost-to-benefit relation if the number of problems is small</td>
</tr>
<tr>
<td></td>
<td>- allows to take the benefits of competition (costs, performance, etc.)</td>
<td>- robustness sometimes may lead to wrong results (garbage-in/garbage-out)</td>
</tr>
<tr>
<td></td>
<td>- other users’ experience may be advantageously used</td>
<td></td>
</tr>
<tr>
<td>in-house</td>
<td>- deep knowledge of the software deficiencies</td>
<td>- time consuming development</td>
</tr>
<tr>
<td></td>
<td>- developing a software is the best training on CFD</td>
<td>- requires a somewhat dedicated staff, which vies with a dedicated and usually greater body</td>
</tr>
<tr>
<td></td>
<td>- customized solutions, advantages in terms of CPU time and models necessary to achieve high precision</td>
<td>- “out-of-the-target” issues (programming, interfacing, debugging, model validation, etc.)</td>
</tr>
</tbody>
</table>

Late developments on CAE tools aiming the integration with CAD software have lead to huge improvements in terms of both effectiveness and flexibility in analysis involving solid models. This integration was feasible only due to the development of parametric 3D tools in some CAD packages, known as “parametric features”. The main goal is to achieve a solid model which, as per the modeling process evolves, the features integrate shaping a model which is easily modifiable and robust in terms of dimensional modifications. The integration of both CAE and CAD tools in the very same platform or in two separate platforms with an integrated two-way communication makes reasonable a unified model which satisfies the needs of the drawing and analysis interfaces, which makes superfluous the access of the original CAD model in the CAE ambient, since at the end of the analysis the final CAD model aggregates all the modifications due to the numerical analysis. Figure (2) depicts the development cycle, with and without the solid modeling; the benefit is clear in terms of both speeding up the process and avoiding mistakes.
3. EXAMPLE: INTEGRATED CAE/CAD/CAM IN CRANKCASE DEVELOPMENT

The crankcase (figure 3) is the central structure of a reciprocating compressor, and holds practically all the internal components of the compressor. In this specific case it is made by cast iron, and some specific regions, where piston, crankshaft, stator and cylinder head are assembled, require machining with a very tight finishing tolerances. In some cases, the springs, suction and discharge mufflers are assembled on the crankcase too.

The main phases of the development are: modeling (via CAD), functional analysis (via finite element analysis - FEA), manufacture of the cast tool rack (via CAM – Computer Aided Manufacturing), casting, machining, experimental tests and final approval. The whole process evolves in an integrated CAE/CAD/CAM environment. In this specific case one single commercial code was used to deal with both the CAD/CAM interface, another one to CAD/CAE interface and mesh generation, a third one to the CAE analysis and finally another one to perform the geometric measurements on the prototype. The topology is defined by the analysis of the images captioned by a laser-based system, which are analyzed through a specific software. The three-dimensional (3D) model is the basic link among all the several phases of development (figure 4), and allows a great interaction of information among the several disciplines involved in the process, preventing reworks and eliminating ambiguous information. At the end of the process, a significant increase in the quality of the component and in the productivity in the development cycle are achieved. An important aspect in the productivity gain is related to the 3D model; this approach implies no need for drawings in the initial phase of the process, since the solid model alone has all the information required for the finite element analysis, CNC programs, rapid prototyping, etc. Drawings are only required later, for activities such as: machining shop planning, dimensional control and documentation.
Figure (3) – A reciprocating compressor and its crankcase

Figure (4) – 3D Solid model: inputs for multidisciplinary tasks

Usually the functional analysis (stiffness) of the crankcase is performed via the finite element analysis (FEA). The method is used for analyzing crankcase stiffness (the main function) and deformations caused by the manufacturing process and component assembly. The main targets of FEA in the functional analysis of the crankcase are the stiffness of the cylinder and shaft bearing, natural frequencies, crankcase deformation and mass optimization. Once the model is
defined, with all the pertinent details, and the project’s numerical goals are achieved, the next phase is the casting tool rack development. The first step is defining the mold’s division and introducing the thermal expansion effect. There is also the possibility of improving the casting process through a specific software being used to define the feeders, thus improving casting performance. CNC machines manufacture the casting tool rack, and the programs are generated from the solid model of the tool rack. After the machining process, the tool rack is assembled in the casting plates, poured and production begins.

The benefits achieved by an integrated CAE/CAD/CAM system during the crankcase design can be synthesized in quality improvement and reduction in development time.

4. EXAMPLE: SUCTION MUFFLER OPTIMIZATION

The primary function of a suction muffler is conducting the intake gas from the suction tube to the suction port, with minimum heat transfer and head loss and maximum noise attenuation. Thus the trade-off between two engineering branches (acoustics and transport phenomena) lead to the optimum conceptual design. Obviously a number of experimental setups support both analysis and the designer’s experience is important to determine the shortcuts in the optimization process, as defining the weights of different objective functions (in this case, noise attenuation, heat transfer and head loss). This conceptual design is then concurrently optimized in terms of mass minimization, using a structural analysis code, and in terms of assembly process, using a parametric CAD. Figure (6) shows a general overview of the optimizing process. Some codes used are commercial: one for CAD, another for CFD, a third for FEM/BEM (finite element – boundary element methods) and the last for noise optimization; others are proprietary: one for the overall compressor simulation, another for the analysis of the flow and heat transfer through manifolds and another to deal with the hybrid
models. The approach used considered the optimization of both characteristics, efficiency and noise separately at a first step. Coefficient of performance (COP) and noise attenuation (analyzed through the Frequency Response Function – FRF) were the objective functions and the muffler main geometrical characteristics (tubes’ diameters and lengths, number of baffles and tubes and shell shape) the variables. Once a local optimum was achieved for COP the values resulted for set of variables were used as initial conditions for the FRF optimization. Testing benches were used not only to corroborate the results obtained through the numerical procedures in specific experiments, but also a design of experiments approach was used after each COP-FRF optimization cycle to validate the results. This procedure is specially important when dealing with noise; sometimes a numerical local optimum may lead to some resonance in the system which could impair the final result. Once it was observed the set of values which lead to this result was excluded from the space within which the optimization process was allowed to evolve.

Figure (6) – Optimization cycle for a suction muffler

Figure (7) depicts a suction muffler geometry and the typical results obtained via CFD (pressure pulsation and mass flow rate in the boundaries and pressure field for one specific configuration) and acoustics software (frequency function response for different configurations). Such sort of information must be analyzed together with other fields as well as other acoustic analysis to give the designer the necessary information on deciding the trade-off between performance and acoustics. The analysis must be transient for compressor operating at the normal cycling range, since a steady state simulation just give some few hints about the phenomena that take place inside the muffler. The analysis must also be coupled with the valve simulation and the overall compressor simulation program in order to take the whole picture, since the pressure field in the muffler outlet is one of the fundamental boundary condition for this problem, which is of paramount importance on compressor performance. Thus a number of other tools must be taken into account during this optimization and the development of an expert system to perform all the necessary compromises seems very difficult, becoming notable the role of the designer as the “integrator” of all information achieved. Another important aspect is the acoustic behavior of the suction muffler defined via a FRF and its correlation with the others important acoustics compressors features define the spectrum and total noise. The goal is to avoid interaction between suction mufflers and cavity, suspension and shells needs to be avoid, but considering the huge number of possible interactions the uncoupling is performed by a mix of the expert system response and the designer’s judgement.
The pressure and velocity fields are important to understand the main flow losses, which once minimized have a positive effect not only in efficiency, but also in terms of noise, since turbulence may act as a noise generation. Pressure pulsation has a strong interaction with the valve movement and a companion program is used to evaluate the compressor overall performance based on the information coming from the simulation of both pulsation and flow through the valve. The main advantage of using this approach comes from a huge reduction of the number of experiments necessary, since most of the drawbacks in terms of efficiency and noise are anticipated. Reducing the number of prototypes leads to a significant reduction in terms of costs and development time, as well as a larger number of options explored.

5. CONCLUDING REMARKS

The ever increasing pressure on speeding up the design process and achieving performance corroboration in the early stages of the development has been the main driver of using CAE/CAD/CAM tools integration. This approach makes more clear to people from different areas of expertise what are the conceptual design issues, the various sets of boundary conditions and make easier to deal with the large number of trade-off to be considered for achieving the “optimum” design. Moreover, the use of CAE tools lead to a better
understanding of experimental results, minimizes the number of expensive and time consuming experimental routines and allows the designer to achieve a clearer idea on the main issues on the design in the very beginning of the development. It does not make superfluous the use of experimental approaches, actually the main gain in this sense is avoiding the use of the ineffective, expensive and arduous “cut and try” experimental approach. The main goal is to achieve a complete scenario of the different relationship among the various branches of engineering involved in compressor design, with all the connections defined by mathematical functions, simulation procedures and/or experimental techniques. Obviously this very ambitious intent is still to be achieved and this work only shed some light on how some more restrictive instances have been solved.

Another important aspect is the integration do promote shorter lead times, but on the other hand the role of skilled work force increases, since the irresponsible use of the tool may generate results convincing at a first glance but actually useless. In conclusion the whole approach do not change some principles of “responsible” engineering, as the need of deep knowledge on both theory and practice, experimental corroboration of the models and results a global vision of the project. The main achievement is minimizing the “manual labor” in order to leave more time to the engineers to intellectual work.

6. REFERENCES

(2). Sung, K. R., Shih, Y. T. & Kim, J., “Application of the Expert System for Basic Design of Compressors”, In: International Compressor Engineering Conference at Purdue, Purdue University, West Lafayette, IN, USA, 1994
(3). de Bortoli, M. G. D., “Concurrent and Simultaneous Engineering Applied to Design Hermetic Refrigeration Compressor”, In: International Compressor Engineering Conference at Purdue, Purdue University, West Lafayette, IN, USA, 1992
(4). de Bortoli, M. G. D., “The Use of an Integrated CAE/CAD/CAM System During the Compressor Design”, In: International Compressor Engineering Conference at Purdue, Purdue University, West Lafayette, IN, USA, 2002
Compressor system technology: evolutionary potential and evolutionary limits

Darrell Mann
Systematic Innovation Ltd., UK

ABSTRACT

For the last 5 years, the world has generated an average of 550 new patents per year in relation to the design of compressors. The paper describes a programme of systematic analysis of these and earlier patents. The purpose of the research has been to establish the evolutionary status of compressor system technology across a number of the different sectors of the industry, to benchmark the capabilities of the different sectors relative to one another, to benchmark capabilities relative to a global measure of evolutionary maturity and then, most importantly, to identify future development directions and opportunities for the industry.

INTRODUCTION

This paper is aimed at setting a global context for compressor technology. Such a broad-ranging ambition requires the foundation provided by the single largest study of innovation ever conducted. That study started in the former Soviet Union in 1946 (Reference 1), and has now accumulated data from close to 3 million successful solutions in a range covering all areas of human endeavour. One of the main findings from that research has been that different scientific and engineering disciplines spend a large proportion of their time re-inventing what has already been discovered in other areas.

A large part of the focus of the more recent (1998-2004) research has been on the mechanics of system evolution (Reference 2). The research has had a particular focus on what happens when systems evolve in a non-linear, discontinuous manner from one way of doing things to another. A large part of the interest here has been on the identification of jumps that are common to all industries and disciplines. To date, the research has uncovered 35 such generically applicable discontinuous technology evolution trends. Although it is something of an over-simplification, it is useful to think of these trends as different s-curves – Figure 1.
The evolutionary s-curve drives the evolution of all systems. The research into evolution dynamics has also shown that all successful innovations possess an attraction to an ideal end-state. That end-state – typically defined as ‘Ideal Final Result’ (IFR) – is that the system delivers the functions and benefits that a customer requires, without any cost or negative impacts. While this end-state might sound somewhat theoretical, there are many examples of systems and components that have evolved to such a state (Reference 3). What Figure 1 shows is that the dynamic of evolution towards this end-state occurs through a succession of s-curves. Key to the understanding of the overall dynamic is the recognition that all systems hit fundamental limits: The flattened profile at the top of an s-curve is not an indication that the market or engineers cease to be interested in improving a system, rather that something emerges to prevent those improvements from taking place. In other words a conflict or contradiction emerges and a system hits a fundamental limit as a consequence. The only way, then, to go beyond this fundamental limit is to find a new s-curve. Finding a new s-curve means resolving the contradiction. The 35 uncovered trends (there may be more waiting to be uncovered, but 35 is the current total) in turn represents patterns describing how those contradictions have been resolved.

The paper uses these 35 trends as the global benchmark against which compressor technology can be compared. The paper describes some of the main trends and then introduces the concept of evolutionary potential as a means of comparing the absolute maturity of a system against a global standard of discontinuous system evolution. Beyond this, the paper describes the analysis of two regions of the compressor technology spectrum to demonstrate that, despite the fact that many in the industry would assume that compressors are a well-matured technology, even those designs pushing the state of the art have considerable untapped evolutionary potential left in them. Conversely, the paper also indicates areas where technologies are hitting fundamental limits. The combined implications of untapped evolutionary potential and evolutionary limits are summarised in a section of the paper discussing the best and worst places to devote future compressor technology R&D and intellectual property resources.
TRENDS OF EVOLUTION AND EVOLUTION POTENTIAL

By way of introduction to the form and content of the discontinuous trends uncovered during the innovation research, Figure 2 illustrates a trend known as ‘surface segmentation’. This trend describes the evolution of the use of the surfaces surrounding structures. According to the trend, when engineers and designers first configure a system it is likely to feature a smooth surface. Then later, 2-dimensional grooves or protrusions of some description are added. The reason why such a jump occurs changes across different industries – so that in some it will be to reduce drag, or introduce a space for a lubricant or other material, in others it will be to increase surface area to aid thermal management, and in yet others it will offer the potential to improve grip or traction. The jump from smooth to ribbed/grooved, however, is consistent across all of them. Likewise, the jumps to the next stages of the trend – 3-dimensional and then active surfaces – are common across different industries, but for shifting reasons. Also shown in the figure is a typical design of piston from a piston-type compressor. As may be seen from the figure, this design has evolved to the second of the four possible stages.

The main idea suggested by the trend, then, is that systems generally evolve in a left-to-right direction, becoming ‘more ideal’ at each stage. So, concerning the piston, which has only used two of the four possible stages, there is the suggestion that somehow the incorporation of 3-dimensional and active elements would be beneficial. Again the important idea is that the jump from one stage to the next represents a discontinuous shift from one way of doing things to another. Occasionally systems will evolve ‘the wrong way’ along a trend. There are a number of underlying reasons why this might happen – all of them so far predictable. The most common reason for a backward jump is that a backward step in along one trend is (in the short term) necessary to facilitate a forward jump along another – more important – trend.

Thinking specifically about compressor technology, as we have seen, typical piston designs have evolved to the second stage along this surface segmentation trend, while the rotors in a screw-compressor are still predominantly found at the first (‘smooth surface’) stage. The reason why the piston has made the advance is due to the demands of effective sealing and consistent lubrication. Nevertheless, according to the trend, because the screw compressor rotor is likely to be still at the first stage, it possesses considerably more untapped potential. As the demands on these compressor types continue to evolve, they are highly likely to have to use up the unused surface segmentation potential in order to achieve the desires that engineers require from them. These desires may have nothing to do with sealing or lubrication of course – in which case the evolution to 2D or 3D grooves/protrusions is likely to occur for other reasons. Like for example in this specific case to achieve lower flow resistance or improved noise properties.

Figure 3 illustrates another of the trends, this time the one known as ‘dynamization’. This is a trend concerned, as the title suggests, with the way in which things move relative to other
things. Again the trend is drawn in such a way that systems evolve in a left-to-right direction, with each stage representing a discontinuous advance on the preceding stage.

Here is another trend with direct relevance to the evolution of a wide range of different compressor types. This time, however, it is necessary to think a little more abstractly to connect between the design of a compressor and a stage on the trend. Pistons translate along their host cylinder, but yet the majority of them would be classified at the first ‘immobile’ stage of the trend. The important connection here – and an indication of the way in which the best advantage can be made of the trends – is that there is *some* aspect of a component that is immobile. A piston consisting of a single-piece would be classed as ‘immobile’ since the structure is designed to be as consistently stiff as possible. The same ‘immobile’ stage may also be seen in both screw and scroll type compressors. Examination of the valve-plates in refrigerant compressor designs, on the other hand, reveals that here is a part of the system that has evolved to the ‘completely flexible’ stage of the trend.

The fourth and fifth stages of the dynamization trend are more difficult ones as far as compressor evolution are concerned. If a compressor design is at any of the first three stages of the trend then, according to the evidence from other industries (which is where the trend has ultimately come from) it has two stages of untapped potential remaining. Both of these jumps are, however, comparatively large ones – away from mechanical to fluidic and ultimately field-based methods (where ‘field’ is intended to signify any type of field – whether it be electrical, magnetic, gravitational or any other connection that a user can make from the deliberately generic label). In both of these cases, current technological limitations prevent compressors from making the jump (but the trend has given us a good indication of where enabling technology research would be well-placed). In refrigerant compressor systems, however, we are already beginning to observe solutions that are eliminating mechanical components in favour of ‘field-based’ cooling systems – e.g. Peltier Effect based cooling devices. The jump from left-to-right in this case is occurring because field-based systems tend to be inherently more reliable, more controllable and less noisy. At least these are the reasons why the jumps have occurred in other sectors.

In reality, these two trends, and the other 33 presently known, all operate in an integrated fashion. The ‘evolution potential’ concept is a means of observing the trends together. Their basic principle of operation is very simple: In order to examine the overall evolution potential of a system, it needs to be compared against each of the trends in turn. As we do this we will soon discover that some of the trends cannot be connected to the system. Such trends are deemed to be not relevant – typically, in fact between 10 and 25 of the 35 will be useful in analyzing any one system. The non-relevant trends (which will shift from one application to another) are eliminated from the analysis. Each of the remaining ‘relevant’ trends is then compared to the system under evaluation and an assessment made of how far along the trend a system has reached.

Thus, if the valve-plate of a refrigerant compressor is under evaluation relative to the ‘surface segmentation’ trend, a state-of-the-art design will be at stage 2 of 4 possible stages. Likewise,
relative to the ‘dynamization’ trend, a state-of-the-art valve-plate will be at stage 3 of 5. Repeating this process for each trend in turn, and then arranging each trend as one spoke on a radar plot reveal a picture of the overall current evolutionary state of the valve-plate. The basic idea is illustrated in Figure 4.

Figure 4: Evolution Potential Radar Plot Structure

The perimeter of the radar plot represents the frontier of engineering knowledge against each of the known trends. Thus the empty space between the current evolutionary state of the system described by the shape at the centre of the picture and this perimeter represents the untapped potential of the system. If, therefore, there is a desire to improve some aspect of a current system, the trend lines and the untapped potential act as signposts indicating the directions that other successful systems have evolved in.

It is almost impossible to convince anyone that all of the jumps on all of the trends lead to better designs in every area, on every occasion. In twelve years of using the trends, however, this author has not found a single exception. This despite having a full-time research team of some 25 people who spend every day examining new inventions in an attempt to disprove the known trends and to find new trends. Sometimes we aren’t smart enough to work out why a jump should take place, but inevitably over the fullness of time we will see that reason becoming apparent. Whether or not people ‘believe’ the trends – and it is certainly not the purpose of this paper to attempt such a feat – at the very least they should be viewed as a potent way of focusing and directing short cuts to better solutions. In part this is where the term ‘systematic innovation’ – the emerging name of the overall methodology surrounding these trends – comes from.

In the next two sections we will explore some of the attributes of the evolution potential concept as they relate to two different compressor types. We will begin this exploration with a look at the evolutionary history of screw compressors:

EVOLUTION OF SCREW COMPRESSOR SYSTEMS

Given the basic evolution potential ‘global benchmarking’ concept and the radar plot format, it becomes possible to use the capability in a number of ways. Examining the various generations of design of screw compressor systems in the industry and overlaying the plots for each, it becomes possible to see how quickly the evolution potential is being used. Figure 5 illustrates such a composite picture highlighting the jumps that have taken place from the first screw compressors to the most recent.
The plot labeled ‘1’ describes the evolutionary state of the first generation screw compressors as defined by German patents from the latter half of the 19th Century. In simplified terms, then, stage ‘2’ represents the evolution of the first twisted rotor geometries of the Heinrich Krigar patents, also on the late 1800s. This evolution is shown as an advance along another trend of evolution known as ‘geometric evolution’ (Reference 2 provides more details on this and all other trends). We see further geometric advance in the 1930s designs of Ljungstroms Angturbin AB in Sweden when inventor Alf Lysholm developed the profile of the screw compressor and tested various configurations and rotor lobe combinations. Not only was the shape of the rotors important, he solved the problem and patented the method for accurately machining the rotors – shown in the radar plot as an advance along the ‘macro-to-nano’ scale trend. Lysholm’s 1935 patent clearly shows his asymmetric 5 female - 4 male lobe rotor design. Although the shapes have been ‘fine tuned’ over the years, the ‘modern’ screw compressor saw its birth at this point. The fourth stage highlighted on the plot represents an amalgamation of the miscellany of smaller evolution jumps that have taken place in more recent times. Shown at this fourth stage, then, are things like the incorporation of balance pistons to improve life by compensating for side loads on bearings, variable geometry porting to improve off-design performance, thrust compensation controllers (‘controllability’ trend) and assorted other secondary performance enhancing features.

![Radar chart showing evolutionary history of screw compressor technology](image)

**Figure 5: Evolutionary History Of Screw Compressor Technology**

What is perhaps most interesting about the current evolutionary state is the amount of remaining untapped potential still available. Undoubtedly screw compressor technology may be seen to be mature in terms of geometric evolution – once all of the available dimensional freedoms have been used, there is nowhere else to go. Looking beyond geometry, however, and the plot shows there is still considerable potential for future improvement along a number of other evolution trends – the previously discussed surface segmentation trend included.

It is beyond the scope of this paper to explore this untapped screw compressor potential. What can be said with some confidence though is that each unused trend represents opportunity for not only the step-change improvement in some aspect of the design of the compressor, but perhaps more importantly, in the direction of future research and the generation of new intellectual property.
Figure 6 takes the evolution potential radar plot to a different level. A single high-level plot for a typical state of the art refrigerant compressor has here been complemented by equivalent plots for some of its main constituent parts. One of the values of conducting such an exercise is that it allows R&D strategists a clear, objective picture of which components within the system have more untapped potential than others. If we find ourselves working in an environment where there is insufficient funding available to do everything, then here is a way to help focus money into those areas that are likely to deliver the biggest benefit per unit of capital invested.

A precise analysis, of course, requires rather more detail than we have the scope to describe here, but hopefully the point is made that the objectivity of the evolution potential concept certainly lies at the heart of such an analysis.

In addition to acting as a ‘global benchmarking’ tool, the more important function of the evolution potential concept is as a structured means of generating useful product evolution ideas. The information used to compile the Figure 6 plots comes from a discussion of the prior art to US patent 6,835,050, granted to Samsung on December 28 2004. The plots in this case suggest the considerable untapped potential in just about all of the components present in the design.

The motivation behind the Samsung invention was a desire to reduce noise and improve efficiency of the system. The key inventive step in achieving the inventors’ solution involves the incorporation of a groove in a gasket component. This inventive step is shown in Figure 7. Alongside the isometric view of the new gasket is the corresponding evolution potential radar plot for the new gasket overlaid onto the original plot found in Figure 6. What this figure then shows is that the essence of the Samsung invention corresponds to a single stage jump along the previously discussed surface segmentation trend. Every other aspect of the gasket is evolutionarily identical to previous designs.
Clearly there was rather more effort required by the Samsung inventors than simply adding a groove to one of their gaskets, but a key point worth thinking about here is that an invention that first appeared only a matter of months ago had been signaled by a discontinuous trend of evolution that has now been in existence for over 40 years. The surface segmentation trend – like all of the others – represents the voice of the product. Having identified that gaskets are smooth-surfaced things, the trend is trying to tell us that ‘somewhere there is a benefit in adding some kind of groove or protrusion into the design’. The trend has absolutely no idea how, where or why to do so, merely that based on the successes of others in other industries, that benefit will be present somewhere.

Figure 7: Evolutionary Comparison Between US6,835,050 And Prior Art Gaskets

The trends of evolution do not exist to replace anyone’s creativity, but instead merely to try and direct it along profitable paths. The Samsung design has made a single jump along a single trend, but as can be seen from the new radar plot, there is still a mass of untapped potential. Having made one jump, the surface segmentation trend is still suggesting the likelihood of two additional advantages by adding more grooves, making them 3-Dimensional and adding something active into them. All of the other trends are similarly suggesting a host of other currently untapped potential, not just in the gasket but, as can be seen from Figure 6, in every other component in the design too. Any one such jump may well result in the creation of a patentable solution.

ACCELERATED EVOLUTION – IMPLICATIONS AND OPPORTUNITIES

The real issue here is to think that it took until so recently to make this simple jump. Had other companies been thinking about the trends of evolution in general, and the surface segmentation trend in particular, it is interesting to speculate on whether they would have made the connection that Samsung recently did. Irrespective, though, of whether Samsung actually chose to incorporate such a design into one of their systems, an evolution potential analysis would have allowed them to at the very least recognize and protect the idea so that none of their competitors could have used it. This connection to the generation of IP is probably the most important aspect of all when the implications of the discontinuous trends of evolution uncovered during the systematic innovation research are considered.
The discontinuous trends offer the potential to considerably accelerate the evolution of systems (References 4 and 5). Every unused stage on the screw compressor or the Samsung refrigerant compressor design or any other compressor system we may care to put under the spotlight represents the opportunity to design a better system. The trends encourage designers to ask new questions when they are designing any kind of component. As suggested in Figure 8, by comparing our current system with each of the trends (in this case ‘dynamization’ has been used as an example again) and making a connection with one trend stage, all of the unused trends stages to the right of that connected stage represent potential solutions. We may not know what problems such jumps might solve yet, simply that based on what other successful inventors and problems have found, somewhere, somehow there is an advantage in moving from left-to-right.

![Figure 8: Trends As Evolution Sign-Posts](image)

One of the big ideas in systematic innovation is that someone, somewhere already solved a problem like yours. The trends represent a summation of the general directions those previously successful problem solvers have taken.

**SUMMARY**

1) The systematic innovation methodology has uncovered some of the underlying fundamentals of system evolution: successful systems evolve through a series of discontinuous jumps in a direction towards an Ideal Final Result end-state.

2) The same generic jumps may be seen to occur across different industries. By using the jumps that other industries have already made, it is possible to accelerate the pace of innovation in others. The implications of the possibilities opened up by this accelerated knowledge transfer – particularly in relation to IP issues – are potentially profound. Historically, organizations have not been very good at making step-change innovations (Reference 6). It may still be true. The difference now, however, is that at least they can see what is coming, and take the necessary avoiding steps.

3) Comparison of compressor technologies at two different points along the industry spectrum reveals that in some areas, the high-end technologies have hit some fundamental limits in relation to some possible evolution directions. At both ends of the spectrum, however, there remains considerable untapped evolution potential when comparing to the
global measure of evolutionary possibility. The implications for the industry as a whole may be expected to be significant on both counts.

REFERENCES


©2005, DLMann, all rights reserved.