2012

Some Aspects of Estimating Geometric Characteristics of Screw Compressors

David Buckney
david.buckney@howden.com

Ahmed Kovacevic

Elvedin Mujic

Nikola Stosic

Some Aspects of Estimating Geometric Characteristics of Screw Compressors

David BUCKNEY[12]*, Ahmed KOVACEVIC[21],
Elvedin MUJIC[3†], Nikola STOSIC[2]

[1]Howden Compressors Ltd., Research and Development,
Glasgow, UK, Corresponding Author
Contact Information (david.buckney@howden.com)

[2]City University London, Centre for Positive Displacement
Compressor Technology, London, UK

[3]BITZER Kühlmaschinenbau GmbH, Research and Development,
Rottenburg, Germany

ABSTRACT

The methods used today for the calculation of screw compressor thermodynamic and fluid flow processes, from quasi one-dimensional thermodynamic models to three-dimensional computational fluid dynamics (CFD) procedures, require accurate identification and quantification of geometric parameters, such as volume, gradient and cross-section, leakage flow and blow-hole areas. Historically, some of the geometric characteristics have been neglected, or approximated when calculation accuracy was not essential. However, more sophisticated models may lose some of their advantages if accurate geometric characteristics are not included in them. The work presented in this paper is aimed to enhance the accuracy of geometric calculations. The usual geometric calculation procedures are given and modified where necessary. The results are compared with models generated by 3D Computer Aided Design (CAD), software packages, from which improvements in the accuracy of predictions can be demonstrated.

1. INTRODUCTION

Pioneering work in the accurate presentation of screw compressor geometry was given in the comprehensive screw compressor textbook by Sakun (1960) and the screw compressor handbook by Amosov (1977), while Rinder (1979) presented a similar work, given on the basis of, the most popular screw compressor profile of that time.

Not long after, Singh *et al.* (1984, 1988 and 1990) presented a series of three papers dealing more or less successfully with screw compressor geometry, while Sangfors (1984) for the first time published the principles of differential modelling of the screw compressor thermodynamic process. Stosic *et al.* (1986), published a comprehensive application of such a differential model, previously widely used in the modelling of internal combustion engines and reciprocating compressors, to calculate screw compressor performance. The ability to quickly and accurately estimate the performance of screw compressors, as confirmed by many authors, for example in Fujiwara and Osada (1995) has revolutionised the field of the screw compressor design and optimisation and identified them as a 'success story of the twentieth century', as stated by Fleming *et al.* (1998). During a fairly long history of compressor modelling, this has been convincingly achieved by the use of time dependent models, which assume homogenous properties within the compressor control volumes while neglecting their spatial distribution, as presented, for example by Hanjalic and Stosic (1997). This, as further shown by Stosic and Hanjalic (1997), did not require a spatial and three-dimensional representation of the full complex geometry.

[†] Contributions were made by Dr. Elvedin Mujic while at City University London.

Applying initial conditions in these papers was sufficient to solve the differential equations of conservation of energy and continuity and further to calculate the thermodynamic properties, like pressure, temperature and density in the compressor suction, compression and discharge control volumes within the compressor cycle, as a function of time, or shaft angle. The application of these methods, improved by the growing power of computers and the continuous efforts of researchers throughout the world, led to great improvements in compressor design. This work has been well described in the published literature as, for example, by Feng *et al.* (2001), who presented their work in screw rotor pumps. More information on the development and application of such methods can be found in Stosic *et al.* (2005).

In these works, more details are generally provided on the thermodynamic models rather than on the calculation of the main geometric characteristics. For example, the interlobe leakage area, sealing line shape and shape or size of the blow-hole were accurately calculated using analytical or numerical methods, because it was considered these have a substantial effect upon the performance. However, some of the geometric parameters of screw compressors were only approximated which was sufficient at the time of their development.

More recent work, for example by Mujic *et al.* (2008), which describes developments in screw compressor acoustic predictions, demonstrated the need for the shape and size of the suction and discharge ports and their areas to be calculated more precisely. Subsequent efforts have recently been made to introduce improvements in this area and these have been used as the basis of this paper.

The geometric characteristics in chamber models are regularly represented as a function of time, or compressor shaft angle within the compression cycle. Various flow paths exist during the different phases of the compressor cycle for particular working chambers: suction and discharge port flow paths, as well as, the two types of leakage paths, one for the inflow into the chamber with the lower pressure from the chamber with higher pressure, or from the compressor discharge and one for outflow from the chamber with higher pressure to the lower pressure chamber or to suction.

Additional features that may need to be considered for precise performance prediction include accurate position, shape and size of the injection ports, part load recirculation passages, liquid injection ports and economizer ports, as well as sliding or poppet valves. For all of these, a detailed breakdown of the flow areas may vary, but the principle of their representation remains the same. Furthermore, in many cases the geometry of rotors or port openings is obtained as a set of measured, or approximated points rather than generated analytically or numerically by use of software packages. In such cases, existing programs used to calculate geometric characteristics based on conventional methods might sometimes fail to produce accurate results.

It was suggested fairly early that multidimensional analysis of geometric characteristics might be required for more accurate calculation. For example, Singh and Schwartz (1990) proposed that a full three-dimensional surface representation might be required for calculation of the blow-hole area. Only later, when computational fluid dynamics was used, together with efficient post-processing facilities, to evaluate the full three-dimensional flow field of a screw compressor, as presented in Kovacevic *et al.* (2003), a detailed insight into the internal behaviour in the screw compressor flow was obtained. It indicated, for example, that the flow through a blow-hole area could be accurately approximated only by a three-dimensional representation and that the flow rate depended on the pressure ratio rather than the pressure difference. Although the use of computational fluid dynamics may be prohibitively expensive and time consuming, due to the highest level of expertise required, it gives much needed clear understanding of the internal compressor flows, however it imposes new requirements upon the further development of the compressor geometric characteristics.

To find a compromise between the accuracy of the expensive multidimensional calculations and fast, but less precise chamber models, which is adequate for evaluation and general performance prediction and optimisation, a hybrid model was proposed by Mujic *et al.* (2010) which used chamber models for analysis of flow within the rotor domains and a three-dimensional approach at the compressor ports. This provides a quicker solution with the level of detail required for optimisation of the shape of flow domains, and shows that tools such as computational fluid dynamics can be efficiently used to improve particular aspects of chamber models.

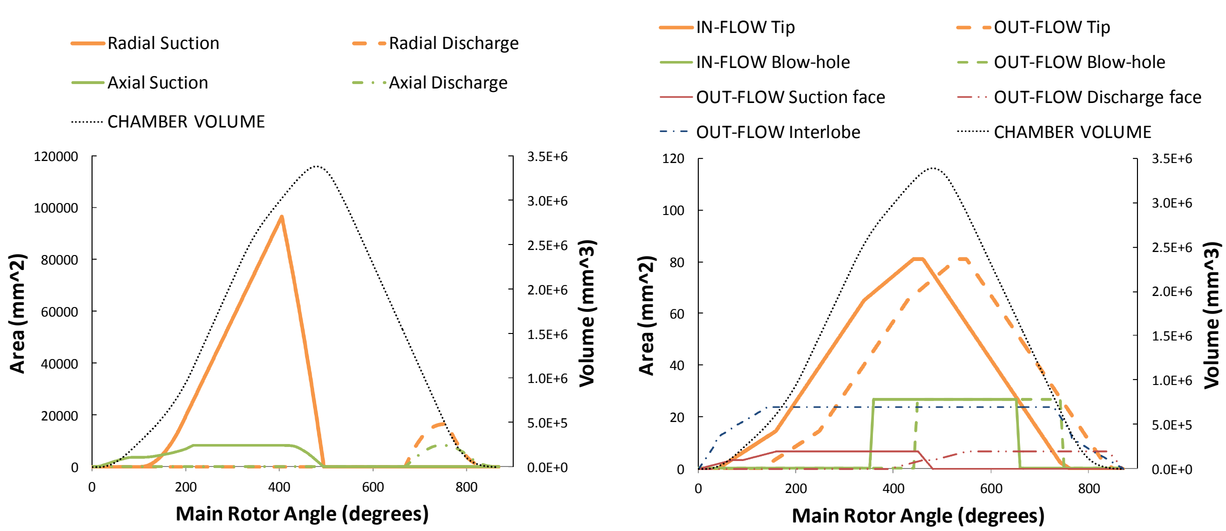
The short calculation time of the chamber model makes it ideal for use in optimisation algorithms. The accuracy and flexibility of recent manufacturing methods for screw compressor rotors is continually improving, which makes it economically feasible to produce small batches of rotors, which justifies the use of rotor and compressor optimisation for various applications, even if the number of compressors produced is limited. Together, these tools facilitate a competitive advantage in practical implementation, for example, the ability to offer a standardised compressor casing with specialised rotor profile and ports.

Attention in this paper is focused on some aspects of calculation of geometric characteristics of screw machines by flexible and generic methods with an aim to assess their accuracy and identify limitations. The adequacy and accuracy of the calculated geometry is, where possible, validated by use of the three-dimensional computer aided modelling. This procedure can be applied for any shape of rotors or port shapes obtained analytically, numerically or experimentally. It is flexible enough to introduce any customised feature in the machine, for example, additional liquid injection ports.

The procedure used in this work is generalized from a number of known methods widely available in the open literature, as well as from one presented in Mujic *et al.* (2008) which was the method used for the calculation of arbitrary shapes of the discharge port.

2. REQUIREMENTS IMPOSED UPON GEOMETRY CALCULATION

The basic requirements defined for the calculation procedure presented in this paper are; - to accurately calculate required geometric characteristics of a screw compressor independently, prior to thermodynamic calculations, - to represent them as a function of the main rotor angle, θ, - to give output of all rotor geometry parameters in a single matrix, - to accept rotor coordinates of any type and from any source for calculation, allowing flexibility and independent rotor comparisons, and - to support the use of arbitrary ports. A selection of some of the typical results obtained has been shown in figure 1. In practice, the data in figure 1a and 1b are available in a single output file. Additional features not shown in the diagram but which may need to be considered for accurate performance prediction include injection ports, part load re-circulation passages, liquid injection ports, economizer ports etc. All of these would be presented in the same manner.



a. port areas b. leakage areas

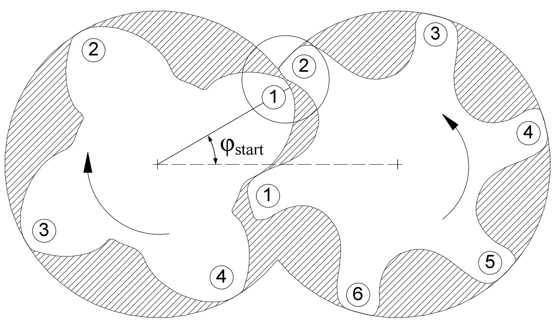
Figure 1: Volume and area curves

Rotor coordinates may be provided either by use of an existing generation tool, for example as proposed in Stosic (1998) or potentially by another source such as manufacturing or measurement files. The ports may be given in either analytical form or as their point coordinates from the design source, or measurements. The output matrix may then be used for thermodynamic calculations to estimate compressor performance and will provide a high degree of control and flexibility when optimising the rotor profile for specific applications.

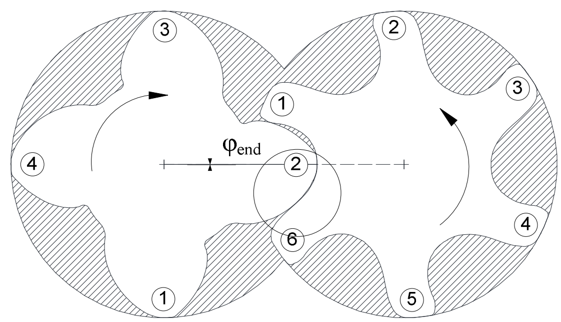
In order to validate the accuracy of the procedure used here for calculation of geometric properties, like rotor cross-sectional areas, rotor volume and its gradient, sealing lines and their lengths and leakage flow areas, all as a function of the rotation angle θ, the calculation results are compared with a three-dimensional CAD model. The compressor in question has asymmetric rotor profile; equal rotor diameters of 255mm; L/D = 1.65; Wrap angle = 300 degrees; and Lobes = 4/6.

3. DEFINITION OF A ROTOR CYCLE

The angle that defines the rotor cycle, θ, is set to start at the point where the volume of the chamber has zero value, just before the start of the filling process. The corresponding main rotor start angle, φ_{start}, in the suction plane of the rotors is measured from the axis connecting the rotor centres to the tip of the main rotor on the suction plane as shown in figure 2a. This angle is sometimes neglected and the filling process starts at $\varphi_{start}=0$. However, for typical asymmetric rotor profile the main rotor will be rotated backwards by the start angle as the initial area is formed by an undercutting action. The start angle is equal to the meshing angle at the point on the sealing line closest to the casing cusp, the sealing line points will be discussed in more detail later. The main rotor end angle, φ_{end}, measured on the discharge plane, is also defined using the sealing line but is typically zero if there is no undercutting on the main profile leading side as shown in figure 2b.



a. start of cycle at suction face



b. end of cycle at discharge face

Figure 2: Rotor start and end positions

4. SEALING LINES AND INTERLOBE CEARANCES

The sealing line is usually calculated by using the rotor point coordinates x and y and their pressure angle defined by -dx/dy. A full derivation of these can be found in Stosic (1998).

In figure 3 the interlobe sealing line, seen from a side projection, is highlighed for a single chamber which shows how the sealing line is related to the defined cycle. Point A is identified as the point on the sealing line which is closest to the high pressure cusp at the flat flank of the rotor lobes. Point B is the same as point A but offset by one rotor lobe. Point C is identified as the maximum deviation along the z-direction between the limits of points A and B. When point C moves past the discharge face the volume of that chamber becomes zero and the cycle draws to its conclusion so this meshing angle defines φ_{end}. The lengths of segements A-C and B-C are treated seperately allowing the change in length of the sealing line due to the axial translation to be considered.

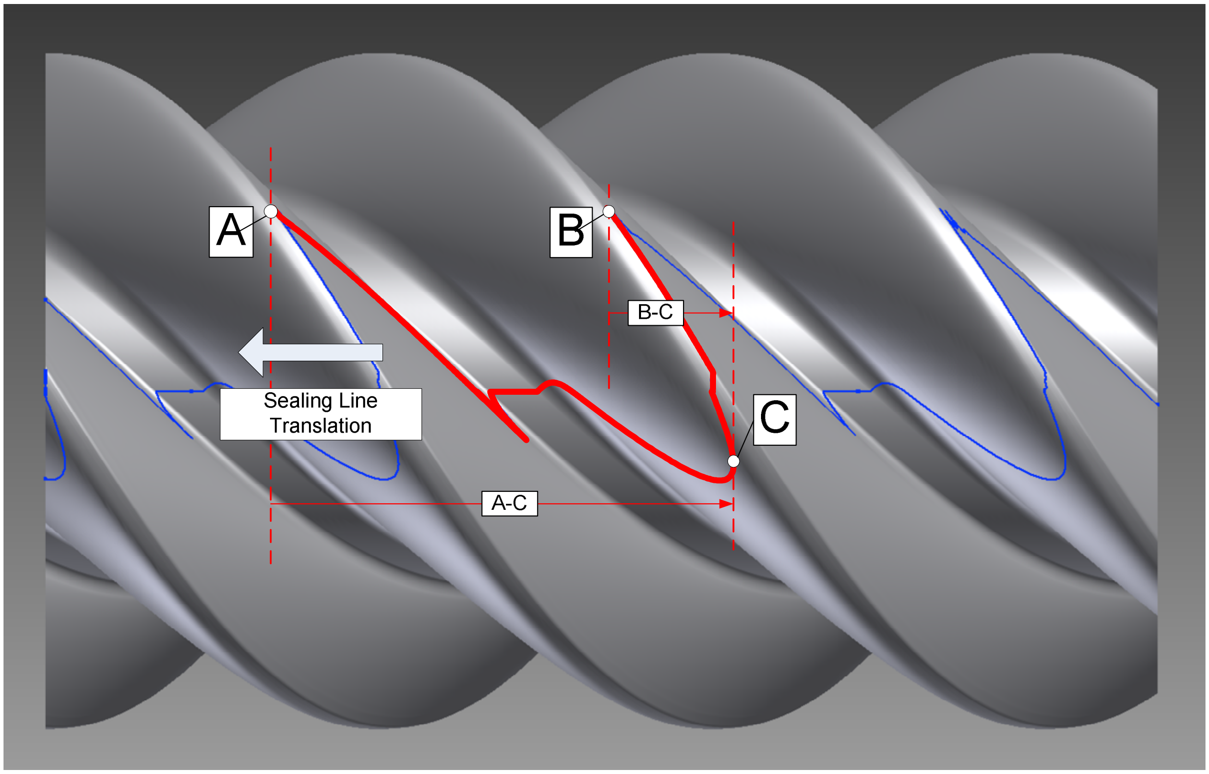
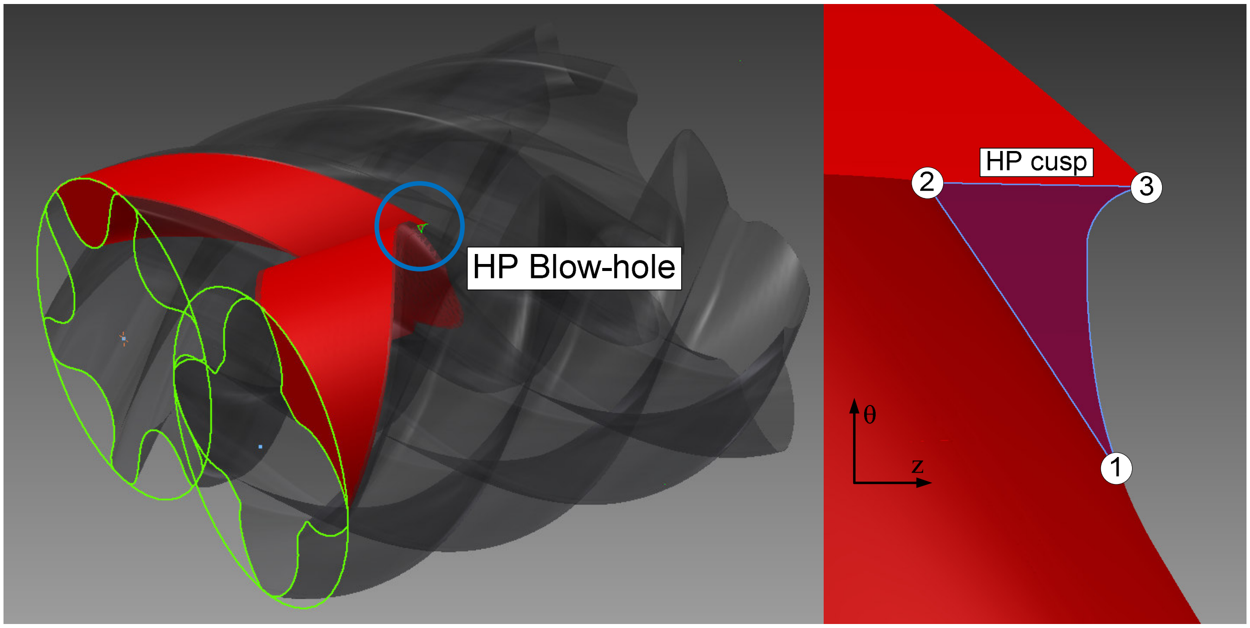


Figure 3: Sealing line and its presentation in three-dimensional CAD model

The sealing line gap is often represented by its length even though the ultimate input to the thermodynamic calculation program is the cross-sectional area along this line through which fluid leaks. This approach is advantageous as it allows modification to the operating clearance and consequently the operating leakage areas. Variable clearance distributions can be considered by mapping local clearances onto the sealing line. By this, the effect of the round and flat flank contact can be investigated, and its advantage was explained in detail by Stosic *et al.* (2005).

5. BLOW-HOLE AREA

The blow-hole represents a cross-sectional area filled with fluid between two consecutive working chambers having two different pressures at each side, thus causing fluid flow from higher to lower pressure. Highlighted on figure 4a is the position of the blow-hole, which gives a feeling about the relative size of this leakage path. The blow-hole behaves similarly to the tip leakage path. Since it connects two working chambers of not significantly different pressure, it does not affect the compressor performance much, unless it is unreasonably large. Since the blow-hole is present on both sides of the working chamber, the leakage gain and the leakage loss are somewhat balanced, however, the excess of work needed to recompress the leakage recirculation is relatively large. Therefore, a blow-hole problem can be detected by its small influence upon the compressor flow and large influence upon the compressor power.



a. fluid volume during discharge b. blow-hole

Figure 4: Blow-hole area

The blow-hole area is a three-dimensional triangle, with non-linear edges, positioned between the rotors and the housing cusp. The vertices are presented as the points, 1, 2 and 3 in Figure 4b, where the rotors first contact and the points where the main and gate rotors meet the housing cusp. The distance, dz, between the curves '1-2' on the main rotor and '1-3' on the gate rotor is calculated for a number of positions, θ (see figure 4b).

In the first instance, the blow-hole area was defined as in Figure 4, by creating a cutting surface to intersect with the fluid volume. The cutting surface was a cylinder on the main rotor bore diameter. The accuracy of the blow-hole area calculation relies on the success of determining the position of points 1 to 3 in figure 4b. This is compared in table 1 for each method on the row labelled 'Cylindrical Surface'.

This area, calculated on a cylindrical surface on the main rotor bore, does not necessarily represent the plane of the minimum flow area. An area reduction factor calculated from the helix angle at the outer diameter of the gate rotor was used in the geometry model, as suggested by Singh and Bowman (1988). This result is again compared with the blow-hole generated by a CAD three-dimensional model, this time in a plane normal to a line that is both tangent to the gate rotor bore and the gate rotor helix. A sketch of this blow hole area was created by projecting the rotor surfaces onto this plane. This comparison is shown in table 1 on the row labelled 'Normal Plane'.

Table 1: Blow-hole comparison

	Blow-Hole Area (mm^2)		*relative difference*
	CAD	Geometry Model	
Cylindrical Surface	34.7	36.6	*5%*
Normal Plane	25.3	26.6	*5%*
relative difference	*-27%*	*-27%*	

Comparing the blow-hole areas formed by a cylindrical surface, the CAD results are in reasonable agreement with those from the numerical integration used in the geometry model. The difference of 5% suggests that the numerical integration of the area is not as accurate in this case as for the chamber cross-sectional areas (discussed later). CAD area measurement will generally result in errors significantly less than 1% and it is more likely that CAD errors might be introduced by poorly defined area boundaries. The numerical model accuracy will depend on the number of divisions along the integrating cycle and in addition, the accuracy of the defined points 1 to 3. Point 1, defined by the sealing line, is sensitive to the sealing line calculation method, particularly due to the existence of interlobe clearances. When comparing the calculation of the blow-hole area in the normal plane corrected by the helix angle reduction factor, the results are again in reasonable agreement. The biggest difference in blow-hole area depends on which surface or plane the area is defined, analysing results with CAD cannot alone verify the most suitable area for thermodynamic calculation.

6. CALCULATION OF ROTOR AND PORT CROSS-SECTIONAL AREAS AND ROTOR VOLUME

6.1 Area Integration

All areas in the work presented in this paper have been calculated using a standard trapezoidal rule numerical approach. The area is divided into a number of sub-areas with the uniform width along one of the coordinates, which may be in either Cartesian or cylindrical coordinate systems.

The main feature of this procedure is that it allows the actual port area to be calculated throughout the cycle for any arbitrary port shape. This is particularly important when the port shapes and sizes deviate from their theoretical shape, for example as had to be done in the compressor noise research presented in Mujic *et al.* (2008). In the case of rotor retrofits or when the ports are optimised for a particular application, the original ports can again significantly differ from the theoretical ports so this area calculation method will provide maximum flexibility.

6.2 Chamber cross-sectional area

The cross-sectional areas of the chamber formed by the main and gate rotors are highlighted in figure 5. The full chamber cross-sectional areas are denoted by 1, while the cross-sectional overlap is denoted by 2.

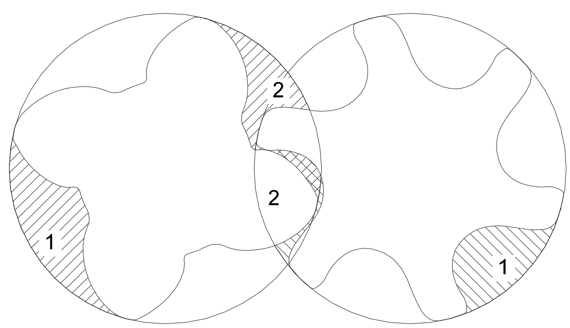


Figure 5: Chamber cross-sectional areas

Applying the numerical integration method described above, the areas defined in figure 5 can be calculated. Table 2 shows a basic check of the rotor chamber cross-sectional area by comparison with values obtained by CAD for each half of the chamber separately.

Table 2: Area Comparison

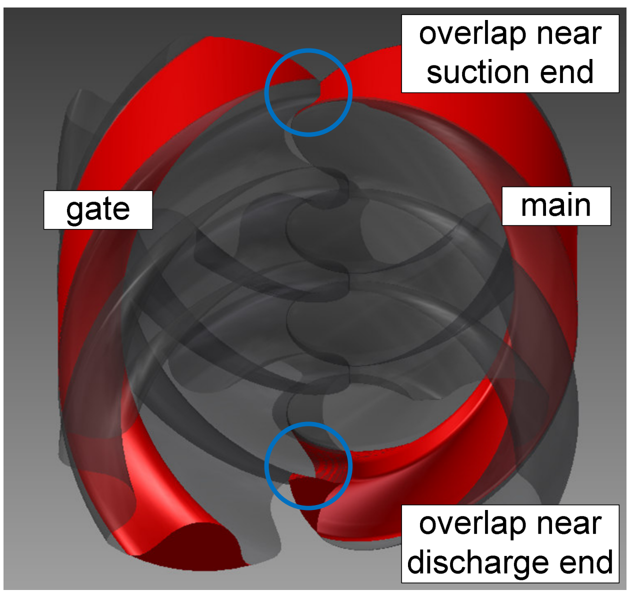
	Cross-Sectional Area (mm^2)		*relative*
	CAD	Trapezoidal Rule	*difference*
Main	4714	4705	*-0.2%*
Gate	3585	3576	*-0.3%*

6.3 Chamber volume

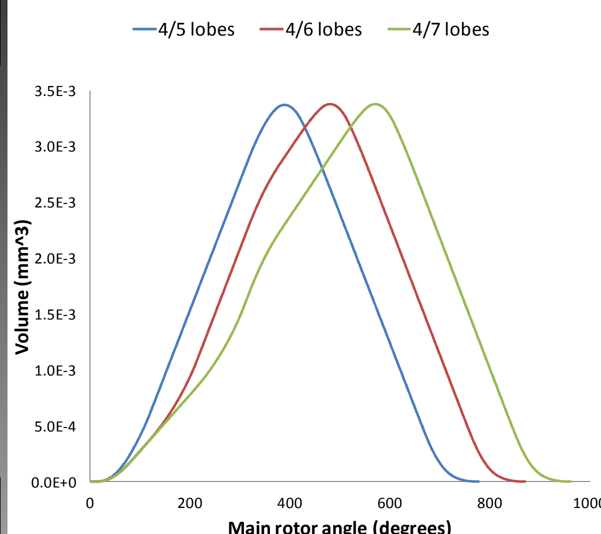
By knowing the full cross-sectional area of the main and gate chambers and the rotor length, it is simple to calculate the maximum volume displacement. This assumes that, during at least a fraction of the cycle, the cross-sectional areas do not overlap at either the suction end, or at the rotor discharge end. The maximum wrap angle, φ_{w_max}, will allow full rotor displacement

If the actual wrap angle, $\varphi_{w,}$ is greater than the maximum value, then in some planes the rotor cross-sectional area will be covered by the rotor lobes of the opposite rotor, reducing the rotor displacement. Therefore, if the wrap angle is larger, then the whole length of the rotors will not be used to form their displacement, because the overlap of the rotors will occur at both rotor ends simultaneously. Interestingly, compressor manufacturers adopted a practice that the wrap angle is regularly larger than the maximum for the free cross-sectional area. This has the consequence that the displacement is reduced by approximately 5% less than its full value. This introduces some of the benefits of a larger wrap angle, for example allowing a larger discharge port cross-section for the same built-in volume ratio than would be the case for the calculated wrap angle. If the wrap angle is smaller than its maximum value, then a so-called 'transport' phase in the rotor cycle will occur between the increasing and decreasing volume phases.

This phenomenon, which occurs in the case of the main rotor only, because the gate rotor has 'spare' lobes due to usually higher number of lobes than the main rotor, is shown in Figure 6a by use of a CAD model indicating that the main rotor volume is still reduced due to overlap at the discharge end, while the gate rotor volume is fully formed at the discharge face. In this case, the assumed convention of the overlap area belonging to the main rotor volume can affect the volume curve. The resulting volume curves for different lobe combinations are shown in figure 6b.



a. overlapping volumes of 4/6 rotors



b. effect of lobe combination on volume curve

Figure 6: Presentation of the rotor volume in three-dimensional CAD model and as volume curve

Most previous calculations show a more or less symmetrical volume curve where the maximum volume occurs approximately mid-way between the suction and compression phases. If the full cycle is defined in this way, the chamber offset during the suction phase is unlikely to affect the thermodynamic results but it provides more detailed insight into the actual cycle behaviour, which may be beneficial during port design.

7. CONCLUSIONS

Contemporary facilities used in the work presented in this paper, which compare the calculation results of geometric characteristics used in estimation of the compressor performance with the exact three-dimensional models obtained by CAD systems show considerable agreement, which offers assurance that the modern and sophisticated methods used for calculation of thermodynamic and flow characteristics are well supplied with accurate values of the compressor volume, sealing line, blow-hole area and port characteristics, all as a function of the compressor shaft angle. The adopted procedures allow quick and accurate calculation of the geometric characteristics by introducing additional flexibility such as the variation of user ports and distribution of variable clearances along the sealing line for calculation of actual suction and discharge and leakage flows. This improves the applicability of the performance calculation tools to the various real life problems that may appear in everyday practice in compressor design, manufacture and operation.

NOMENCLATURE

θ	main rotor / meshing angle	(rad)
φ	compressor geometry angle	(rad)
A	area	(mm^2)
L	length	(mm)
D	diameter	(mm)
z1	no. main lobes	
z2	no. gate lobes	

Subscripts

start	start position
end	end position
w	rotor wrap
w_max	rotor wrap for max volume

REFERENCES

Amosov, P.E., *et al*., 1977, *Vintovie kompresornie mashinii - Spravochnik (Screw Compression Machines - Handbook)*, Mashinstroienie, Leningrad

Feng, C., Xueyuan, P., Xing, Z., Pengcheng, S., 2001, Thermodynamic performance simulation of a twin-screw multiphase pump, *Proc. IMechE. Part E*, Vol. 215, no. 2: p. 157-163.

Fleming, J. S., Tang, Y., Cook, G., 1998, The Twin Helical Screw Compressor, Part 1: Development, Applications and Competetive Position, Part 2: A Mathematical Model of the Working process, *Proceedings of the IMechEng, Journal of Mechanical Engineering Science*, Vol 212, p 369

Fujiwara, M., Osada, Y., 1995, Performance Analysis of Oil Injected Screw Compressors and their Application, *Int J Refrig* Vol 18, 4

Hanjalic, K., Stosic, N., 1997, Development and Optimization of Screw Machines with a Simulation Model, Part II: Thermodynamic Performance Simulation and Design Optimization, *ASME Transactions, Journal of Fluids Engineering*, Vol 119, p 664

Kovacevic, A., Stosic, N., Smith, I.K., 2003, Three Dimensional Numerical Analysis of Screw Compressor Performance, *Journal of Computer Methods in Applied Mechanics and Engineering*, Vol. 3, no. 2, 2003, pp. 259- 288

Mujic, E., Kovacevic, A., Stosic, N., Smith, I., 2008, The influence of port shape on gas pulsations in a screw compressor discharge chamber, *Proc. IMechE. Part E*, Vol. 222, no. 4: p. 211-223.

Mujic, E., Kovacevic, A., Stosic, N., Smith, I.K., 2010, Noise generation and suppression in twin screw compressors machines, *Proceedings of the IMechE, Part E: J. of Process Mechanical Engineering*, Special Issue, I K Smith editor.

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

Sakun, I.A., 1960, *Vintovie kompresorii (Screw Compressors)*, Mashinostroenie Leningrad

Sangfors, B., 1984, Computer Simulation of the Oil Injected Twin Screw Compressor, *International Compressor Engineering Conference at Purdue*, 528

Singh, P., Onuschak, D., 1984, A Comprehensive, Computerized Method for Twin-Screw Rotor Profile Generation and Analysis, *International Compressor Engineering Conference*, Purdue: p. 519-527.

Singh, P., Bowman, J., 1988, Calculation of Blow-Hole Area for Screw Compressors, *International Compressor Engineering Conference*, Purdue: p. 938-948.

Singh, P., Schwartz, J., 1990, Exact Analytical Representation of Screw Compressor Rotor Geometry, *International Compressor Engineering Conference*, Purdue: p. 925-937.

Stosic, N., Hanjalic, K., Koprivica, J., 1986, A Contribution to the Mathematical Modelling of Screw Compressor Working Processes, *Strojarstvo Journal*, Zagreb v28, n2, pp 95-100, 1986

Stosic, N., Hanjalic, K., 1997, Development and Optimization of Screw Machines with a Simulation Model, Part I: Profile Generation, *ASME Transactions, Journal of Fluids Engineering*, Vol 119, p 659

Stosic, N., 1998, On Gearing of Helical Screw Compressor Rotors, *Proceedings of IMechE, Journal of Mechanical Engineering Science*, London, Vol 212, Part C, 587

Stosic, N., Smith, I.K., Kovacevic, A., 2005, *Screw Compressors: Mathematical Modeling and Performance Calculation*, Monograph, Springer Verlag, Berlin, June 2005, ISBN: 3-540-24275-9

ACKNOWLEDGEMENT

Thanks to Howden Compressors Ltd. for sponsoring this research.