ADVANCES IN NUMERICAL AND EXPERIMENTAL INVESTIGATION OF SCREW COMPRESSORS

A Kovacevic, N.Stosic, I K Smith, E. Mujic and D. Guerrato
Centre for Positive Displacement Compressor Technology,
City University, London EC1V OHB, U.K.
t*el +44 20 7040 8780, Fax +44 7040 8566, e-mail:* a.kovacevic@city.ac.uk

ABSTRACT

Industrial requirements for the rapid development of screw machines and accurate prediction of their performance constantly generate the need for further improvement in existing design procedures and the development of new modelling techniques. Advances in numerical methods and the exponential increase in computer power make three dimensional Computational Fluid Dynamics (CFD) into a most promising technique for the fine improvement of such machines and for evaluating their use in new applications. However, further validation of the accuracy of such procedures is required for them to be used for the design of novel screw machines. Moreover, if alternative methods are developed, which employ 3-D CFD, further validation is needed. Modern experimental methods such as Laser Doppler Velocimetry offer new insight in the flow within screw compressors. Fair agreement between predicted and measured flow patterns was obtained but the results show that more work in turbulence modelling is needed to improve this in defined regions of the compressor.

Key words: Computational Fluid Dynamics, Laser Doppler Velocimetry, Validation

INTRODUCTION

Due to their simple design and pure rotary motion screw machines can operate reliably at relatively high speeds. Consequently they are compact and have become the most common type of positive displacement compressor used in industry. In addition their range of application is continually being increased and currently they are also used as liquid and gas expanders, multiphase pumps and combined compressor-expanders. There is a continuing demand for them to deliver greater flow rates at higher pressures and this results in changed rotor profiles and larger machines running at higher speeds. Moreover, new environmental protection legislation requires reduction in noise levels as well as improved efficiencies. More refined analytical procedures are required to help the screw compressor designer meet these demands and three dimensional computational Fluid Dynamics (CFD) offers possibilities for analysis which may help in achieving them. Since the very beginning of computer aided analysis, this technique has attracted the interest of many investigators and is already, used for a wide range of applications. It is based on the numerical simulation of the conservation laws of mass, momentum and energy, derived for a given quantity of matter or control mass. Its early development was driven mainly by the needs of the automotive and aero industries and consequently was first developed for the prediction of external flows. A comprehensive overview of available methods for its utilisation is given by *Ferziger and Perić, 1995*

Of particular importance for the analysis of screw compressors is the calculation of unsteady flow with moving boundaries, by use of the finite volume method. Contributions from several researchers such as *Demirdžić, Perić, Muzaferija, (1998, 1990, 1992, 1995)* enabled a variety of grid topologies to be used for this purpose and to calculate both flow in complex geometries and the simultaneous calculation of fluid flow and solid body stress. All these procedures are required for the analysis of flow in screw machines.

Although the number of papers on the use of CFD is very large, only a few deal with its application to screw compressors. Several attempts have been made to model screw machines by CFD methods but many of these were not successful due to inability to generate an appropriate numerical grid for the complex moving domains. *Kovačević, Stošić and Smith* published a number of papers between 1999 and 2008 in which they introduced 3-D numerical analysis of fluid flow and stress analysis in screw compressor to the world. The breakthrough was made by the use of analytical grid generation through transfinite interpolation, with adaptive meshing, to develop an automatic numerical mapping method for any arbitrary screw compressor geometry, as explained by *Kovacevic, 2005.* This was later regularly used for the analysis of processes in screw compressors by means of an interface program called SCORG (Screw COmpressor Rotor Geometry Grid generator). This also enables a grid generated by that program, to be directly transferred to a commercial CFD or CCM code. Although mainly used for computational fluid dynamics in screw machines, these concepts developed to generate numerical grids may be used for a variety of other applications.

Recently, the authors published a series of papers related to 3-D numerical performance estimation, *Kovacevic et al, 2003 and 2005*., fluid-solid interaction in screw machines, *Kovacevic et al, 2004.* These were summarised in a monograph on CFD in screw machines by *Kovacevic et al, 2006,* which gives a comprehensive overview of the methods and tools used. These can be applied to a variety of commercial CFD software packages currently available, which are capable of coping with complex flows and which can be integrated with CAD systems. More details are described by *Kovacevic et al, 2007*.

A complete report on earlier experimental verification of the numerical procedure was given by *Kovacevic, at al, 2003*. This analysed flow in an oil injected compressor, shown in Figure **1**, which has 'N' type rotors with a 5/6 lobe configuration. The rotor outer diameters are 128 and 101 mm for the male and female rotors respectively, and their centre distance is 90 mm. The rotor length to diameter ratio is 1.65. The numerical mesh for that test case comprises 513,617 cells of which 189,144 cells map the fluid parts between the rotors while the rest specify the suction and discharge ports and oil openings. A cross section through the mesh for the rotors and their fluid paths is presented in Figure 2. The calculations were carried out on an office PC. A converged solution was achieved with 120 time steps in approximately 30 hours of computing time.

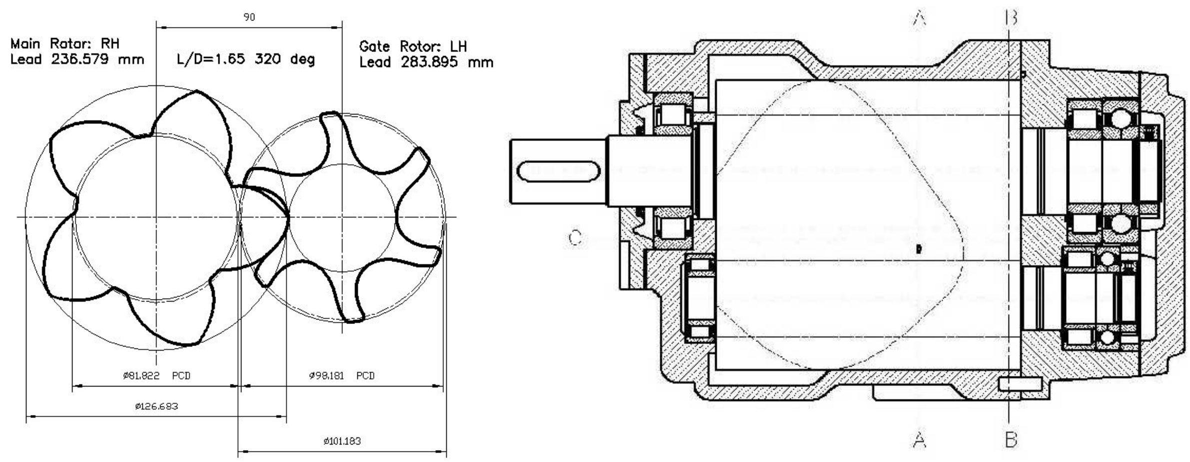

Figure 1 Cross sections through screw compressor rotors (left) and the compressor (right)

The results of the 3-D calculations were compared with measurements obtained from an experimental compressor of identical dimensions. The pressure fluctuations within the machine were measured with piezo-resistive transducers, positioned in the male rotor side of the housing to cover as much as possible of the whole process. Measured results were obtained for suction conditions of 1 bar abs pressure and 20^{o}C temperature with discharge pressures of 6, 7, 8 and 9 bar. Good agreement was obtained both for the integral parameters and the instantaneous pressure values, as shown in the right diagram of Figure 2. The effects of various factors on the accuracy of the calculated results are discussed. These include the size of the numerical mesh, the influence of applied turbulence models, the influence of the differencing schemes used and other factors. It was concluded that the developed methods give reasonably accurate results and can be applied in industry.

However it was also recognised that differences exist in local values at some regions of the machine. Although these have a low impact on the overall performance, their influence on phenomena which may happen locally had to be further investigated. Very few authors have analysed these aspects. For example *Vimmr 2006*, following on *Kauder et al 2000*, analysed flow through a static mesh of the single leakage flow at the tip of the male rotor to conclude that rotor relative velocity does not affect flow velocities significantly and that neither of the turbulence models they used significantly change the outcome of modelling. That was in agreement with the findings published of *Kovacevic et al 2005*, but also confirmed that the need for further validation of full 3D CFD results could not be obtained by simplified numerical or experimental analysis. Instead, a full understanding of the local velocities in the suction, compression and discharge chambers of the machine was needed to further validate the existing methods to develop additional models, if needed.

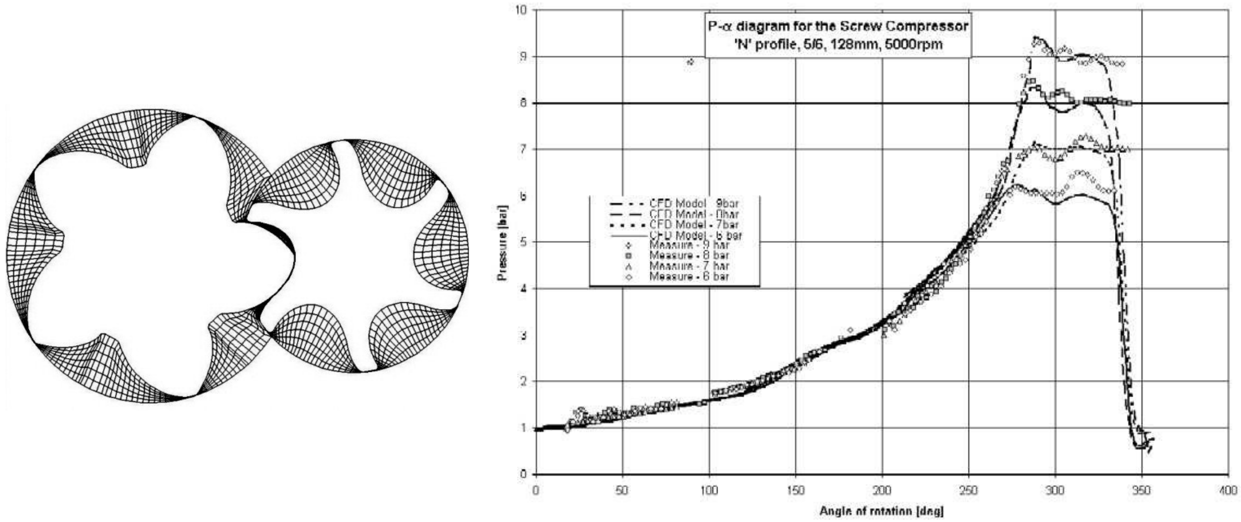

Figure 2 Numerical mesh in rotors cross section (left), the pressure comparison diagram (right)

FLOW MEASUREMENTS IN SCREW COMPRESSOR BY USE OF LASER DOPPLER VELOCIMETRY

Flow in a screw compressor is complex, three-dimensional and strongly time dependent, similar to that in cylinder flows of gasoline and diesel engines, centrifugal pumps, or in a turbocharger. This implies that the measuring instrumentation must be robust to withstand the unsteady aerodynamic forces and oil drag, must have a high spatial and temporal resolution and most importantly must not disturb the flow. Point optical diagnostics, like LDV, can fulfil these requirements (*Nouri et al, 2007*). The experimental results with this technique were obtained in an extensive study of the discharge part of the working chamber as well as in the discharge chamber, *Guerrato et al, 2007*.

A transparent window for optical access into the rotor chamber of the test compressor was machined from acrylic to the exact internal profile of the rotor casing and was positioned at the pressure side of the compressor near the discharge port, as shown in Figure 3. After machining, the internal and external surfaces of the window were fully polished to allow optical access. Optical access to the discharge chamber was through a transparent plate, 20 mm thick, installed on the upper part of the exhaust pipe. The optical compressor was then installed in a standard laboratory air compressor test rig, modified to accommodate the transmitting and collecting optics and their traverses, as shown to the right of Figure 3. The laser Doppler Velocimeter operates in a dual-beam near backscatter mode. It comprised a 700 mW argon-Ion laser, a diffraction-grating unit, to divide the light beam into two and provide frequency shift, and collimating and focusing lenses to form the control volume. A Fibre optic cable was used to direct the laser beam from the laser to the transmitting optics, and a mirror was used to direct the beams from the transmitting optics into the compressor through one of the transparent windows. The collecting optics were positioned around 25° of the rotor chamber and 15° of the discharge chamber to the full backscatter position and comprised collimating and focusing lenses, a 100 µm pin hole and a photomultiplier equipped with an amplifier. Although the crossing region of the laser beams is an ellipsoid more than 0.5 mm long, the size of the pinhole defines the effective length of the measuring volume so that it can be represented as a cylinder 100 high and 79 µm in diameter. The fringe spacing is 4.33 µm. The signal from the photomultiplier was processed by a TSI processor interfaced to a PC and led to angle-averaged values of the mean and RMS velocities. In order to synchronise the velocity measurements with respect to the location of the rotors a shaft encoder that provides one pulse per revolution and 3600 train pulses, with an angular resolution of 0.1°, was used and fixed at the end of the driving shaft. Instantaneous velocity measurements were made over thousands of shaft rotations to provide a sufficient number of samples. In the present study the average sample density was 1350 data per shaft degree. Since the TSI software is provided by 4 external channels, one of them was used to collect the pressure signal coming from the high data rate pressure transducer via an amplifier.

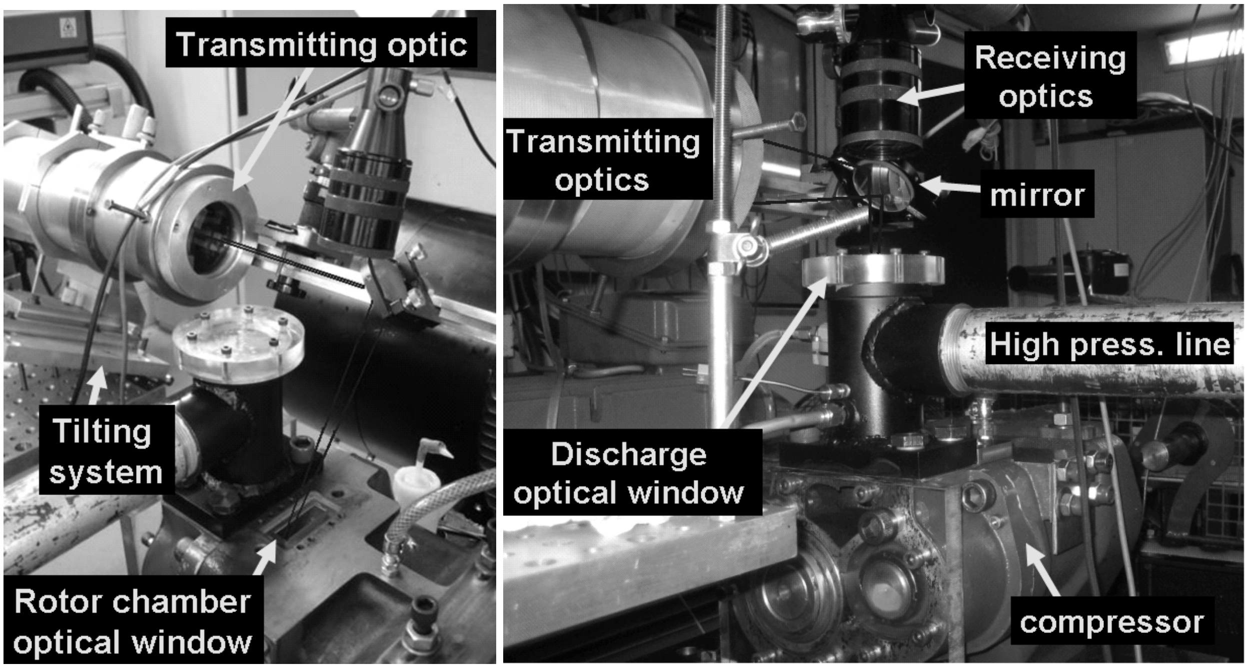

Figure 3 Optical compressor (left) and the LDV optical set for discharge chamber (right)

Two coordinate systems were defined within the rotor chamber of the compressor as shown in Figure 4 (a), (b) and (c). Each of them is applied to one of the rotors where α_p and R_p are, respectively, the angular and radial position of the control volume and H_p is the distance from the discharge port centre. Taking the appropriate coordinate system, measurements were obtained at R_p=48, 56, 63.2mm, α_p=27º and H_p=20 mm for male rotor, and at R_p=42, 46, 50 mm, α_p=27º and H_p=20 for female rotor.

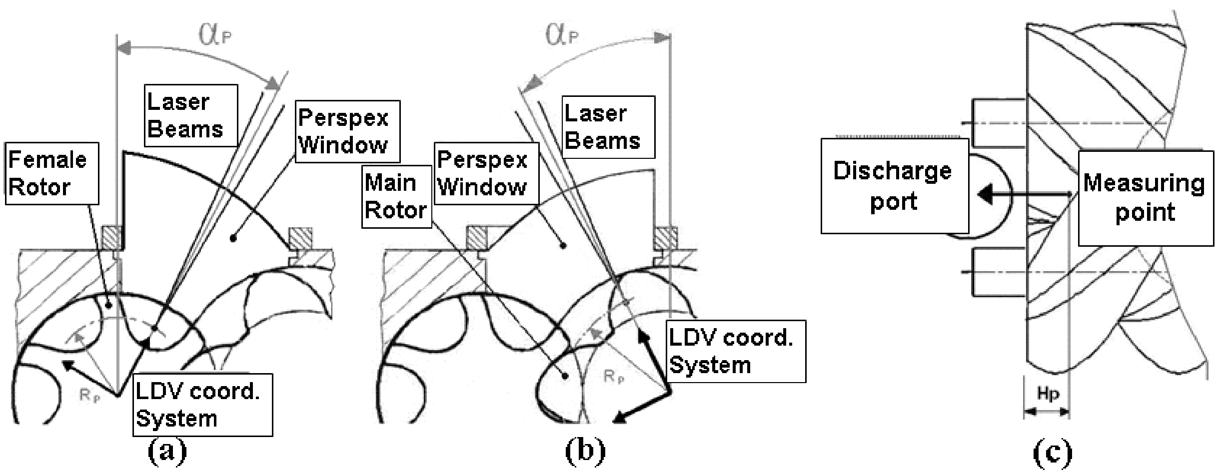

Figure 4 (a) Coordinate system and window for the female rotor; (b) Coordinate system and window for the male rotor; (c) Axial plane view

Typical velocity values measured in the working chamber are shown in Figure 5. Three zones were identified in the working chamber near the discharge port. Zone (1) covers most of the main trapped working domain with fairly uniform velocities. Zone (2) is associated with the opening of the discharge port. The velocities and turbulence in this zone are much higher then in Zone (1). In this zone the flow is driven by the pressure difference between rotors and the discharge chamber, which is especially visible in this case as the pressure in the discharge system was maintained at practically atmospheric conditions. Zone (3) is associated with the leakage flows between the rotors and the casing, where velocities increase to values higher then in Zone (1) but are not as chaotic as in Zone (2).

Conclusions derived from the measurements, as explained in more detail in *Guerrato et al, 2007*, are as follows: (1) Chamber-to-chamber velocity variations were up to 10% more pronounced near the leading edge of the rotor. (2) The mean axial flow within the working chamber decreases from the trailing to the leading edge with velocity values up to 1.75 times larger than the rotor surface velocity near the trailing edge region (3). The effect of opening of the discharge port on velocities is significant near the leading edge of the rotors and causes a complex and unstable flow with very steep velocity gradients. The highest impact of the port opening on the flow is experienced near the tip of the rotor with values decreasing towards the rotor root.

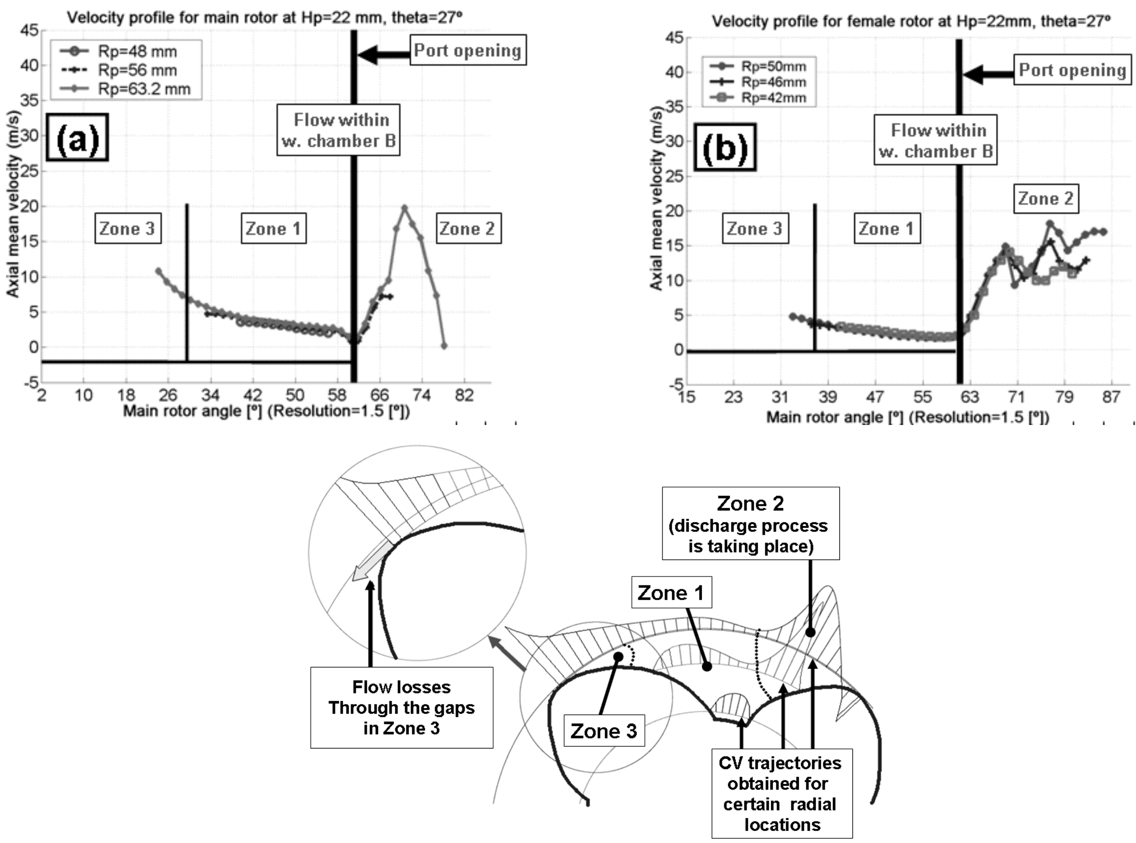

Figure 5: LDV results for the axial velocity component in the working chamber (top), Schematic representation of zones in the compressor interlobe domain

Figure 6 (a) shows a schematic arrangement of the discharge chamber divided into the discharge port domain and the discharge cavity. Figure 6 (b) and (c) show the measurement locations for two characteristic cross sections called the W and V sections. These are visible in the photograph in Figure 6 (e). The coordinate system, drawn in all of the sketches in Figure 6, identifies the location of the measured CV. Measurements were made at Xp=5.5mm, Zp =13mm and Yp = -8 to 13mm.

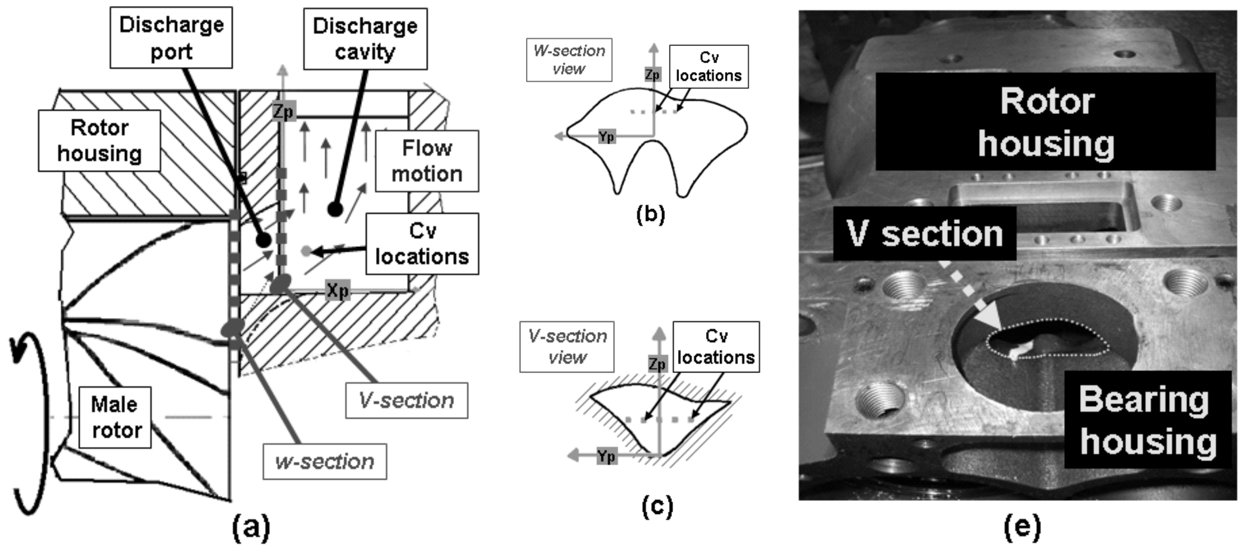

Figure 6: Position of the measured point in the discharge chamber: (a) Axial section through the discharge (b) "W" section view, (c) "V "section view; (e) Discharge chamber viewed from the flange connection

Typical measured results in the discharge chamber are shown in Figure 7. The axial mean flow velocities are obtained using Laser Doppler Velocimetry (LDV) at a rotational speed of 1000 rpm and a pressure ratio of 1.0. The most important findings are as follows. (1) Velocities are higher than in the compression chamber due to fluid expansion in the port between sections W and V. (2) The axial velocity distribution within the discharge chamber is strongly correlated to the rotor angular position since the rotors periodically cover and expose the discharge port through which, at some point, more then one working chamber is connected. The left diagram in Figure 8 shows the case when only one compression chamber is connected to the discharge port and the flow is relatively stable. This corresponds with the domains to the left of the port opening line in diagrams in Figure 7. As another chamber with high pressure flow connects to the discharge chamber, in the right diagram in Figure 8, jet like flows near sides of the discharge chamber passage occur. These are rendered with high velocities in the domain to the right of the thick port opening line in Figure 7. (3) The jet flows create velocity peaks making the flow in that region highly turbulent.

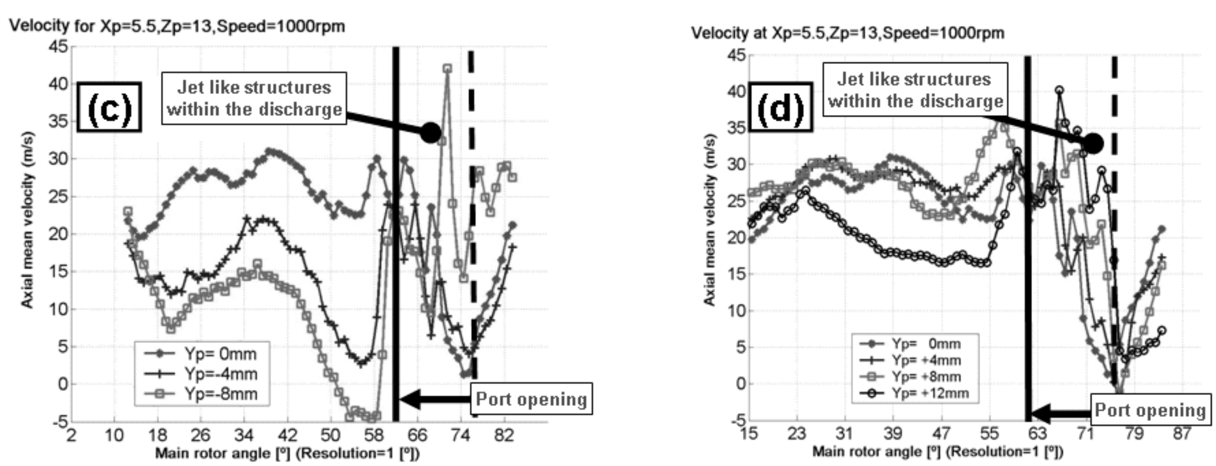

Figure 7 LDV measured axial velocity component inside the discharge chamber: male rotor side (left), female rotor side (right)

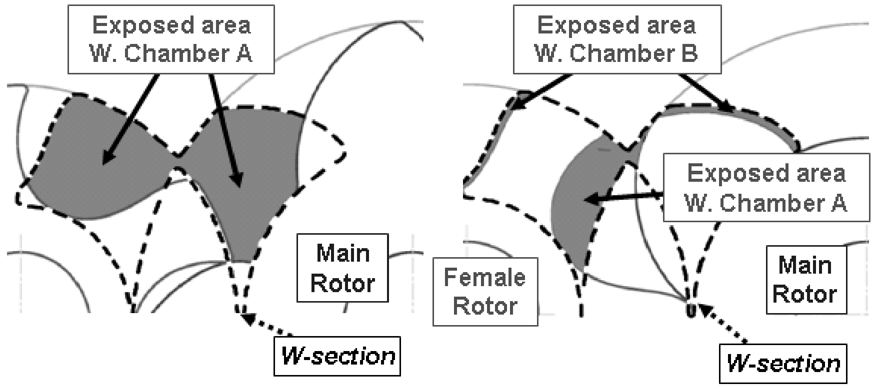

Figure 8 Schematic view of the periodic exposure of working domains to the discharge port

VALIDATION OF COMPRESSOR CFD BY COMPARISON OF MEASURED AND CALUCLATED DATA

The numerical mesh used for CFD calculation and comparison with the measured data, obtained with the LDV technique, is shown in Figure 9. The flow paths around the rotating parts of the machine are generated using the SCORG in-house software to set up the CFD calculations. The pre-processing script generated in SCORG is used to connect these with the stationary numerical mesh of compressor ports, generated directly from the CAD system, and to transfer the entire case to the CFD solver. Figure 9 shows the mid sized mesh consisting of 935000 numerical cells. For the purpose of obtaining a grid independent solution, three different meshes were generated, the smallest consisting of 600000 numerical cells and the biggest with 2.7 million cells, which was the largest possible case that could be calculated by the single processor of the computer then used.

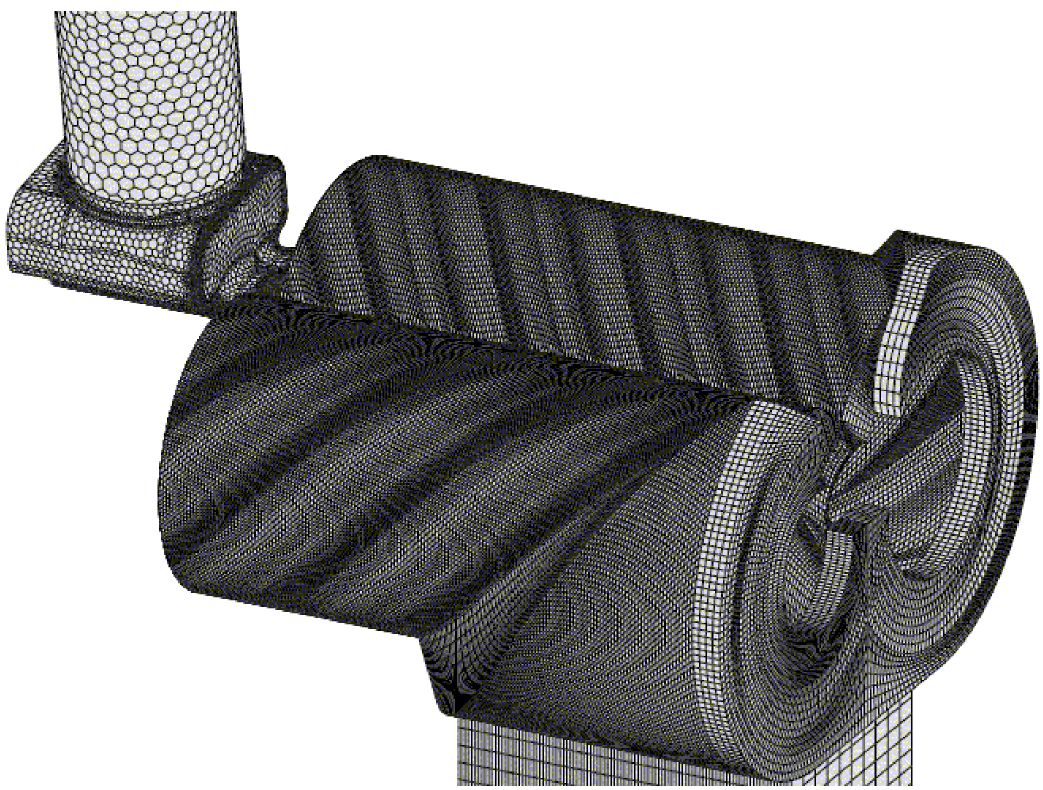

Figure 9 Numerical mesh of the test compressor flow passages

Due to space limitations, this paper only compares the CFD results extracted from the middle size model with the LDV results. The compressor working conditions and the position of the CV are identical to the LDV measurements. Figure 10 shows a comparison of the axial mean velocities in the compression chamber close to the discharge port. This figure shows very good agreement throughout Zone (1) and Zone (2), as defined in Figure 5. In Zone (3) both the measured and calculated velocities increase but the increase in velocities obtained from CFD is larger as a consequence of the inability of the k-e model of turbulence used to cope with flows near walls in rather large numerical cells. Such a configuration of the numerical mesh is the result of the methodology used for generating and moving of the numerical mesh, as explained in more detail by *Kovacevic et al, 2006*.

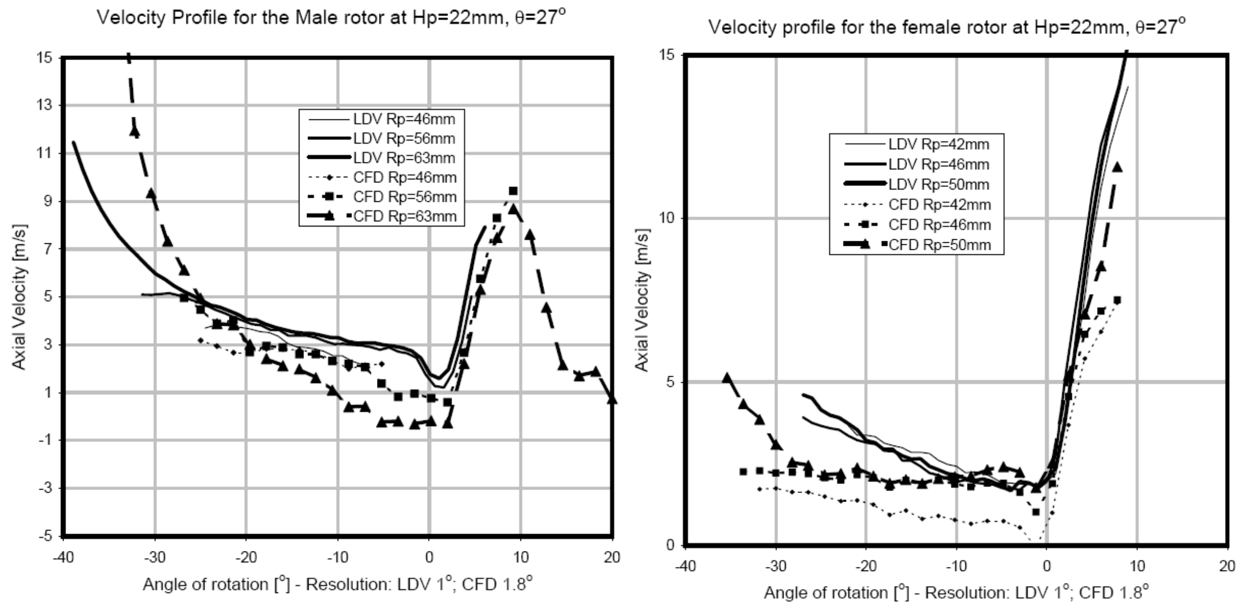

Figure 10 Comparison of the LDV and calculated axial velocity components inside the working chamber

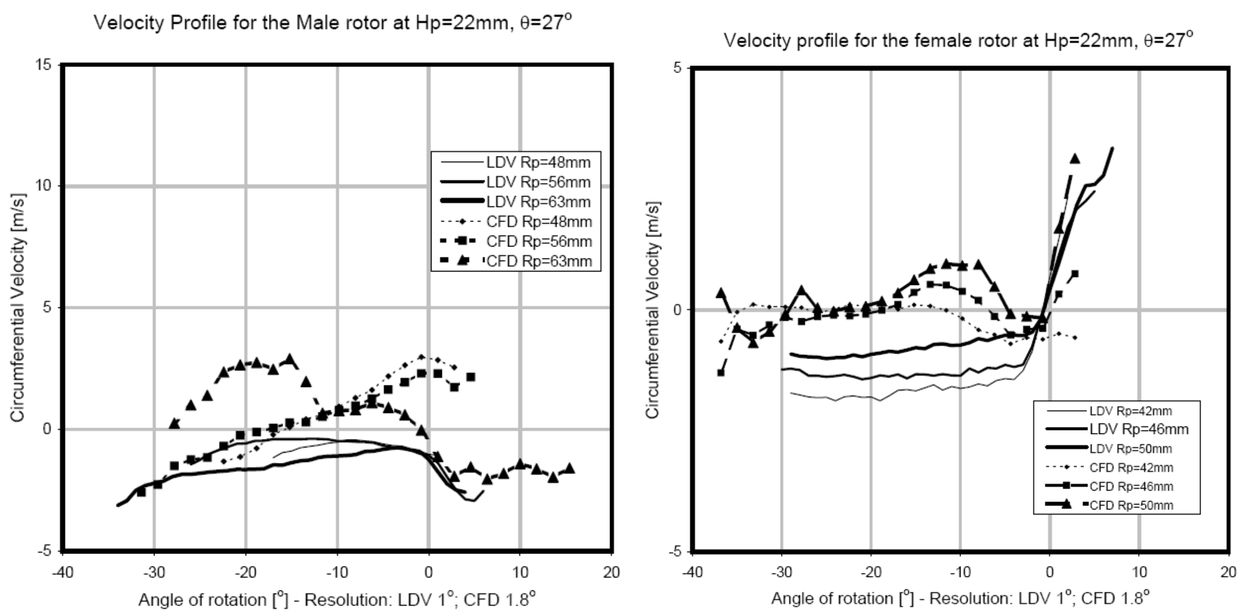

Figure 11 Comparison of the measured and calculated circumferential velocity components inside the working chamber

Figure 11 shows the circumferential velocities in the measured cross section. As the compression chamber in that area is closing very quickly and the flow is mainly driven towards the axial discharge port, the circumferential velocities are low. It should be noticed here that the numerical mesh in each cross section, as shown in Figure 2, contains 2640 cells where each interlobe in a cross section contains only 240 numerical cells. Therefore to extract results from the CFD calculations in the same position as the measurements, the velocities needed to be interpolated between neighbouring numerical cells. That inevitably led to inaccuracy and therefore, especially for the low velocities and high local gradients, the difference between the measured and calculated values is larger. Despite that, the results of the calculation show relatively good agreement with the measured values. Since the predicted integral parameters and pressure correlate well with the measured values, it is fair to assume that the small differences in the velocities in the circumferential direction do not play a significant role.

Figure 12 shows a comparison of the axial velocities in the discharge port. The differences appear to be rather large at locations where velocities are measured, although the trends and mean values are similar. It is confirmed by calculation that the highest values of axial velocity are in the middle section through the discharge port, which corresponds to the period of time when only one working chamber is connected to the discharge chamber. On both the male rotor side of the discharge port, as shown in the top diagram of Figure 12, and on the female rotor side of the port, shown in the bottom diagram, the velocities during that process decrease towards walls. However during the phase when another working chamber is connected to the discharge port, the velocities near the walls increase due to the jet like flows induced by higher pressure differences on the outside of the rotors.

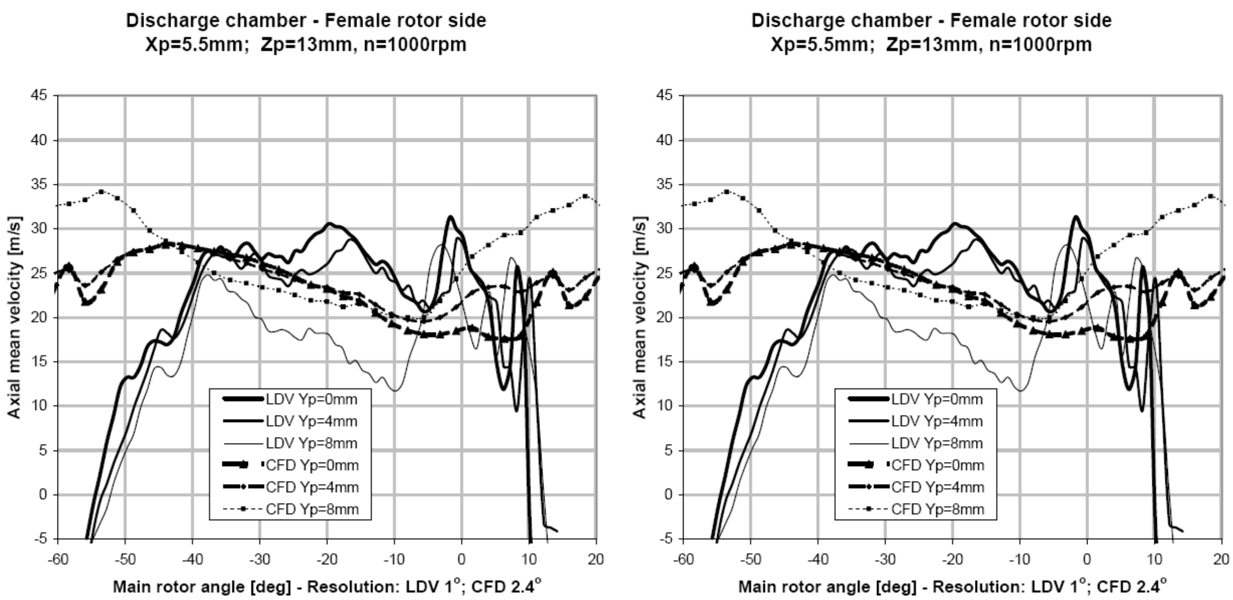

Figure 12 Comparison of the measured and calculated axial velocities in the discharge chamber

Additionally, leakage flows from the working chamber with the highest pressure, which precedes the chamber connected to the discharge port, are large, as can be seen in Figure 13. These cause an increase in velocity in the central region of the port.

The measurements confirm that turbulence plays a significant role in the chamber that connects the rotor domain to the large discharge domain. This is the probable reason why the CFD results do not replicate the measured values more exactly. Therefore further research in the turbulence models for internal flow in the compressor ports needs to be performed. Favourably, the flow on both sides of that region appears not to be so turbulent. Due to that fact and because the internal energy in positive displacement machines is significantly larger than the kinetic energy, this does not greatly affect the overall estimation of performance. Despite this, further development and improvement of CFD codes are needed.

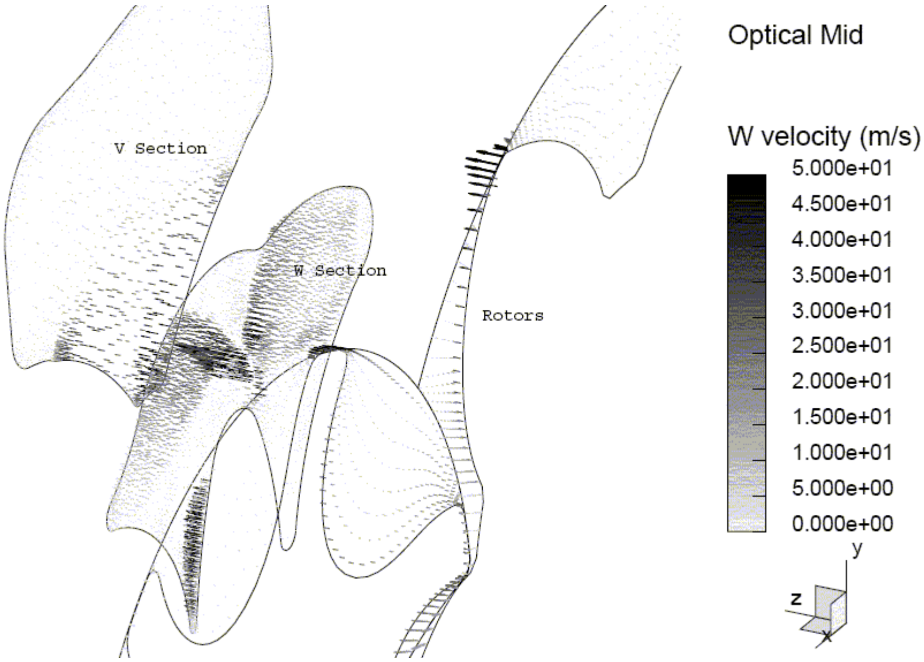

Figure 13 Axial velocities in three characteristic cross sections in the discharge region of the compressor

CONCLUSIONS

Measurements of velocities within a test optical screw compressor by the use of Laser Doppler Velocimetry presented in this paper have validated estimates derived from CFD calculations. They have also shown which areas of the CFD package need to be developed further in order to make it possible to design future screw compressors without the need for expensive and time-consuming experiments and 'tuning' of computational models. They confirm that CFD can be used not only for the estimation of screw compressor performance but also for capturing phenomena not discernible by the use of other, more simplified models. They also outline the need for further research in turbulence modelling for internal flows. The most probable means of accurate accounting for this is to differentiate between areas with low and high turbulence levels and to apply different turbulence models for each flow region.

ACKNOWLEDGEMENT

This work is a part of the EPSRC project 'Experimental and Theoretical Investigation of Fluid Flow in Screw Compressors' (Compressor Project) executed in period 2006-2008 at City University London. Financial support of Engineering and Physical Science Research Council, Trane and Lotus is cordially appreciated.

REFERENCES

Albrecht, H.-E., M. Borys, N. Damaschke and C. Tropea, eds. Laser Doppler and Phase Doppler Measurement Techniques. 2003, Springer.

Durst, F., Fluid mechanism developement and advancements in the 20th century, in 10th International Symposium on Applications of Laser Techniques to Fluid Mechanics. 2000: Lisbona.

Demirdžić I, Muzaferija S, 1995: Numerical Method for Coupled Fluid Flow, Heat Transfer and Stress Analysis Using Unstructured Moving Mesh with Cells of Arbitrary Topology, Comp. Methods Appl. Mech Eng, Vol.125 235-255

Drain, L.E., ed. The laser Doppler technique. 1986, John Wiley: New York.

Ferziger J H, Perić, M, 1996: Computational Methods for Fluid Dynamics, Springer, Berlin

Guerrato D, Nouri J.M, Stosic N. and Arcoumanis, C. 2007 Axial flow and pressure characteristics within a double screw compressor. 3rd Int. Conf. Optical and Laser Diagnostics, ICOLD, 23-25 May 2007, London.

Guerrato D, Nouri J.M, Stosic N., Arcoumanis, C., Smith I.K., 2007, Flow Development in the Discharge Chamber of a Screw Compressor. Int. Conf. on Compressors and their Systems, IMEchE, September 2007, London.

Kauder K., de Araújo-Rudolph L., Sachs R.: Experimental and numerical investigation of the gas flow using a plane model of male rotor-housing gap in a screw-type machine. Schraubenmaschinen 8, Dortmund 2000

Kovačević A, Stošić N, Smith I. K, 2003: Three Dimensional Numerical Analysis of Screw Compressor Performance, Journal of Computational Methods in Sciences and Engineering, vol. 3, no. 2, pp. 259- 284

Kovačević A, Stošić N, Smith I. K, 2004: A Numerical Study of Fluid-Solid Interaction in Screw Compressors, International Journal of Computer Applications in Technology (IJCAT), Vol. 21, No. 4, 2004, pp. 148-158

Kovačević A 2005: Boundary Adaptation in Grid Generation for CFD Analysis of Screw Compressors, International Journal for Numerical Methods in Engineering (IJNME), vol. 63, 2005

Kovačević A., Stošić N, Smith I.K, 2006: Screw Compressors Three Dimensional Computational Fluid Dynamics and Solid Fluid Interaction, ISBN-10: 3-540-36302-5 , Springer Berlin Heidelberg New York

Kovačević A., Stošić N, Smith I.K, E. Mujic, 2007, CFD Integrated Design of Screw Compressors, Engineering Applications of Computational Fluid Mechanics Vol. 1, No. 2, pp. 96–108

Li, H.F., J.Y. Tu and A. Subic, The three-dimensional CFD analysis of flow behaviours in a complex twin-screw supercharger, in The 12th Biennial Computational Techniques and Applications Conference 2004: Melbourne.

Nouri J.M, Guerrato D, Stošić N, Kovačević A, 2006: Cycle Resolved Velocity Measurements Within a Screw Compressor, 18th International Compressor Engineering Conference at Purdue, Lafayette, Indiana, USA

Mujic, E., N. Stosic, A. Kovacevic and I.K. Smith, The Influence of Port Shape on Gas Pulsations in a Screw Compressor Discharge Chamber, in International Conference Compressors and Their Systems. 2007: London.

Mujic E, Kovacevic A, Stosic N, Smith IK, The influence of port shape on gas pulsations in a screw compressor discharge chamber, Proceedings of the Institution of Mechanical Engineers, Part E, Journal of Process Mechanical Engineering, 2008

Peric M.: Flow simulation using control volumes of arbitrary polyhedral shape, ERCOFTAC Bulletin, No. 62, September 2004

Stosic N, 1998: On Gearing of Helical Screw Compressor Rotors, Proceedings of IMechEng, Journal of Mechanical Engineering Science, Vol 213 (Part C): p. 587-594.

Stosic N, Smith, I.K. and Kovacevic, A, 2002: Optimization of Screw Compressor Design, XVI International Compressor Engineering Conference at Purdue, July 2002

Stošić N, Smith I.K, Kovačević A., 2005: Screw Compressors Mathematical Modelling and Performance Calculation, ISBN-10 3-540-24275-9, Springer Berlin Heidelberg New York

Vimmr J, Ondrey F, 2006, Numerical Simulation of Leakage Flow Between Moving Rotor and Housing of Screw Compressor, Proceedings of the Conference Modelowanyie Inzynierskie, ISNN 1896-771X, 32, s. 461-468, Gliwice 2006