Purdue University
Purdue e-Pubs

International Compressor Engineering Conference

School of Mechanical Engineering

2010

Advances in Numerical Modelling of Helical Screw Machines

Ahmed Kovacevic
City University London

Nikola Stosic
City University London

Ian Keneth Smith
City University London

Elvedin Mujic
City University London

Advances in Numerical Modelling of Helical Screw Machines

Ahmed KOVACEVIC*, Nikola STOSIC, Elvedin MUJIC, Ian K Smith

City University London, Centre for Positive Displacement Compressor Technology,
London, United Kingdom
Phone: +44 207040 8780, Fax: +44 20 7040 8566, e-mail: a.kovacevic@city.ac.uk

ABSTRACT

Screw compressors are mature products used broadly in industrial applications. Similar technology is applied for industrial expanders, pumps, vacuum pumps… Mathematical models of various complexities have been developed through decades. Computational Fluid Dynamics (CFD) is regarded the most modern technique which proved suitable for fundamental and applied research of these machines. This paper is review of the work in this area. Case studies show the scope and applicability of CFD in screw machines. Examples include prediction of flow generated noise in screw machines, cavitation modelling in gear pumps, and flow in multiphase oil and gas pumps. Additionally, test programme carried out with Laser Dopler Velocimetry to measure velocity distribution in screw compressor flow domains is presented. It provided data for verification of CFD predictions and suggested areas of research including evaluation of turbulence modelling can provide more accurate and faster CFD calculations.

1. INTRODUCTION

Early designs of screw compressors were based on the assumption of an ideal gas in a leak proof working chamber undergoing a compression process which could reasonably be approximated in terms of pressure-volume changes by the choice of a suitable value of the exponent “n” in the relationship pV^n = Constant.

The advent of digital computing made it possible to model the compression process more accurately and, with the passage of time, ever more detailed models of the internal flow processes have been developed, based on the assumption of one-dimensional non-steady bulk fluid flow and steady one dimensional leakage flow through the working chamber. Together with suitable flow coefficients through the passages, and an equation of state for the working fluid, it was thus possible to develop a set of non-linear differential equations which describe the instantaneous rates of heat and fluid flow and work across the boundaries of the compressor system. These equations can be solved numerically to estimate pressure-volume changes through the suction, compression and delivery stages and hence determine the net torque, power input and fluid flow, together with the isentropic and volumetric efficiencies in a compressor. In addition, the effects of oil injection on performance can be assessed by assuming that any oil passes through the machine as a uniformly distributed spray with an assumed mean droplet diameter. Such models have been refined by comparing performance predictions, derived from them, with experimentally derived data. A typical result of such modelling is the suite of computer programs described by *Stosic et al, 2005*. Similar work was also carried out by many other authors such as *Fleming and Tang 1998* and *Sauls, 1998.* Despite the speed and relatively accurate results, these models neglect some important flow effects mainly in the suction and discharge ports which could influence compressor performance.

Screw compressor performance can be estimated more precisely by a three dimensional Computational Fluid Dynamics (CFD). Nonetheless there are few publications available that describe its successful application in this field. *Kovacevic et al* published a number of papers between 1999 and 2005 which described 3D numerical analysis of the entire machine domain. These were followed by a monograph on CFD applied to screw machines by *Kovacevic et al, 2006,* which gives a comprehensive overview of the methods and tools used for the analysis of flow in these machines.

A number of commercial CFD software packages are currently available which can both analyse the flow through screw machines and easily integrate the results with CAD systems. However the moving, stretching and sliding mesh required for mapping the working chamber cannot be produced within their grid generator packages. Additionally, the time required for calculation of the flow through the entire machine by use of these codes is excessively long. Therefore development of the grid generation method proved to be the key for success for application of CFD in analysis of screw machines.

A prerequisite for success in the highly competitive market of screw machines is the ability to design, analyse and produce machines quickly. These activities need to be automated for use by design engineers in industry. A management suite like that elaborated in *Kovacevic et al, 2006* was developed to integrate tools for the design and manufacture of screw machine components in a user friendly environment suitable for industrial use. It manages both geometric and non geometric information transfer between several software components and has been given the name DISCO (Design Integration for Screw Compressors). This suite provided the platform for integration of CAD and CFD of screw machines. There is still need to reduce computational time and increase accuracy of results which requires specific procedures for such calculations to be developed.

2. PRINCIPLES OF 3D CFD CALCULATION IN SCREW COMPRESSORS

Initially, many attempts to model screw machines by CFD methods were unsuccessful due to inability to generate an appropriate numerical grid for complex moving domains. The breakthrough was made when an analytical transfinite interpolation method with adaptive meshing was used to establish an automatic numerical mapping method for arbitrary screw compressor geometry. It is explained in detail by *Kovacevic, 2003*. This method follows the procedure for rotor generation, fully elaborated in detail in *Stosic, 1997*, and was later regularly used for grid generation in analysis of the processes in screw compressors. The interface grid generation program is called SCORG - Screw COmpressor Rotor Geometry Grid generator. This software suite enables numerical mapping of both, moving and stationary parts of the machine and direct integration in commercial CFD or CCM codes. Thus, authors published a number of papers between 1999 and 2008 presenting feasible 3-D numerical analysis of fluid flow and stress analysis in screw compressors by use of computational Continuum Mechanics (CCM). A monograph on CFD in screw machines (*Kovacevic, 2006*), gives a more comprehensive overview of the methods and tools used. These are applicable to the majority of commercial CFD software packages and can accommodate use of a variety of CAD systems. A typical arrangement of a numerical mesh for the CFD calculation of screw compressors is shown in Figure **1**. The moving parts of the flow domain are mapped with a hexahedral block structured numerical mesh while the remaining stationary parts are modelled by an unstructured polyhedral mesh, produced directly from a CAD system , by the proprietary commercial grid generators suitable for non moving geometries. Although mainly used for CFD in screw machines, the same concept may be utilized for a variety of other applications, for example, in the grid generation of the flow paths in a rotary heat exchanger (*Alagic et all, 2005*)

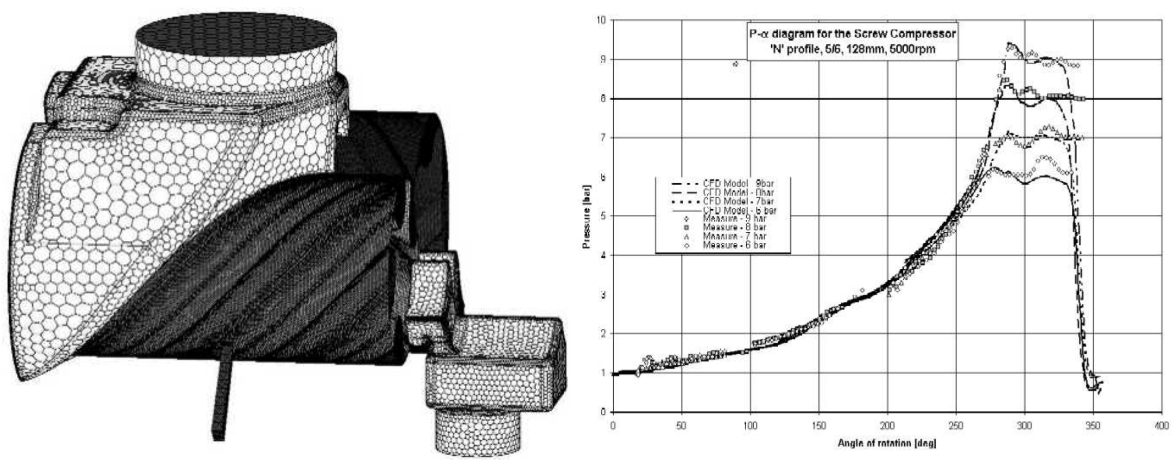


Figure **1** Numerical mesh for CFD calculation of screw compressor – left
The comparison of measured and calculated pressures - right

The first experimental verification of the numerical results was obtained in 2002 and reported in *Kovacevic,2003*. This study was performed on an oil injected screw compressor of a 5/6 lobe configuration with a male rotor outer diameter of 128 mm of ‘N’ type profile. The numerical mesh used, contained just over half a million grid cells, of which about 200,000 were used to map the moving parts of the grid, the rotors and space between them. A converged solution was obtained on an office PC after 120 time steps each requiring approximately 15 minutes of computing time.

The results were compared with measurements obtained from a laboratory air screw compressor. Four piezo-resistive transducers were positioned in the housing to measure pressure fluctuations across the compressor. The numerical and experimental results were compared for discharge pressures of 6, 7, 8 and 9 bar. Good agreement was obtained both for the integral performance parameters, as well as for the instantaneous pressure values, as shown in Figure **1**. The report by Kovacevic et all, 2004 also discussed the effects of various factors that influenced the calculation accuracy. These included variations in the mesh size, different turbulence models and differencing schemes. It was concluded that these variations did not affect the overall calculation accuracy significantly. Therefore the method was recommended as a reliable procedure for performance calculations in industry. However it was also shown that use of different differencing schemes and turbulence methods significantly influences predictions for local velocity and pressure values in certain machine regions. Although these differences have a low impact upon the overall performance, their influence upon flow development needs further investigation. Very few authors have analysed local effects in screw compressors. Examples are the work of *Vimmr, 2006*, following *Kauder et al , 2000*, who analysed the flow of a single leakage path through a static mesh at the male rotor tip to conclude that the rotor relative velocity does not affect flow velocities significantly and that none of the turbulence models used change the modelling outcome significantly. This agreed with the findings of (*Kovacevic et al, 2006*), but also confirmed that further validation of full 3-D CFD calculation results could not be obtained by the use of simplified numerical or experimental methods. For this, a full understanding of the local velocities in the machine suction, compression and discharge chambers was needed.

3. LASER DOPPLER VELOCIMETRY IN SCREW COMPRESSOR

In order to measure flow velocities inside a screw compressor, an experiment, using Laser Doppler Velocimetry (Durst, 2000, Albrecht 2003, Drain, 1986), was set up at City University and an extensive study was performed to measure velocities in the compression domain and in the discharge chamber of an air screw compressor, as reported by *Guerrato et al, 2007*.

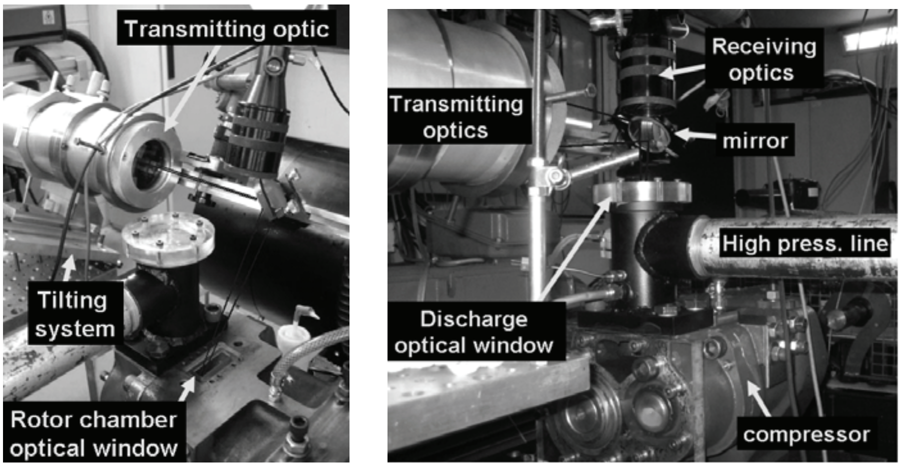


Figure **2** Optical compressor (left), LDV optical set for discharge chamber (right)

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 on the pressure side of the compressor near the discharge port, as shown in Figure **2**. After machining, the internal and external surfaces of the window were fully polished to allow optical access. Optical access to the discharge chamber was arranged 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 transmission of a laser beam and its traverses, as shown to the right of Figure **2**. The laser Doppler Velocimeter operated 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 optical system, 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. The signal from the photomultiplier was processed by a processor interfaced to a PC and led to angle-averaged values of the mean and RMS velocities.

Flow measurements within the compression chamber

Two coordinate systems were defined within the rotor chamber of the compressor, one for the male and the other for the female rotor. The female rotor coordinate system is shown in Figure 3. Each of them was 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 the male rotor, and at R_p=42, 46, 50 mm, α_p=27º and H_p=20 for the female rotor.

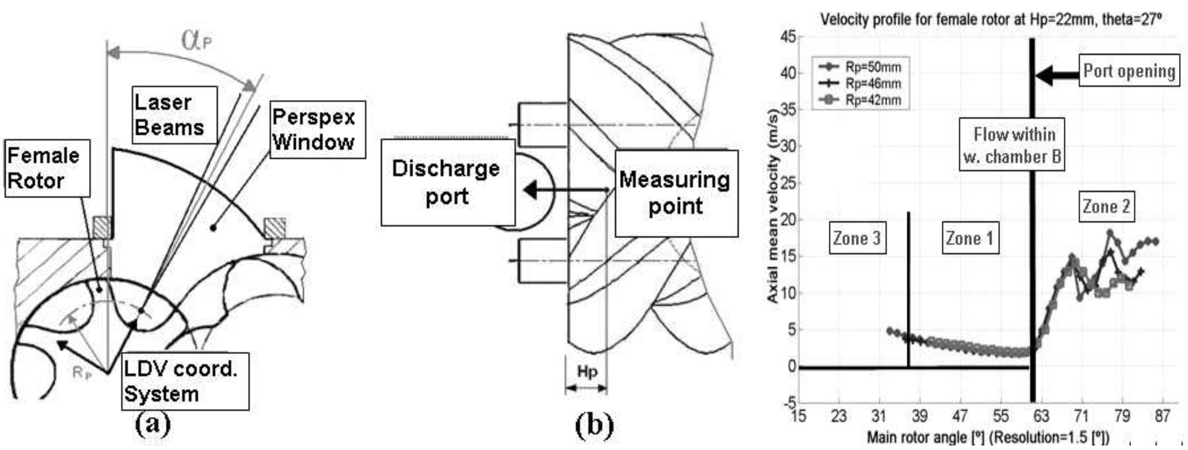


Figure 3 (a) Coordinate system and the window; (b) Axial plane view, (c) LDV measurements

Typical velocity values measured in the working chamber are shown in Figure 3. 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 are explained in more detail in *Guerrato, 2007*, and are summarised 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 on velocities of the opening of the discharge port 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.

Flow measurements within the discharge chamber

Figure 4 (a) shows a schematic arrangement of the discharge chamber divided into the discharge port domain and the discharge cavity. The coordinate system, drawn in all the sketches in Figure 4, identifies the location of the measured CV. Measurements were made at Xp=5.5mm, Zp =13mm and Yp = -8 to 13mm.

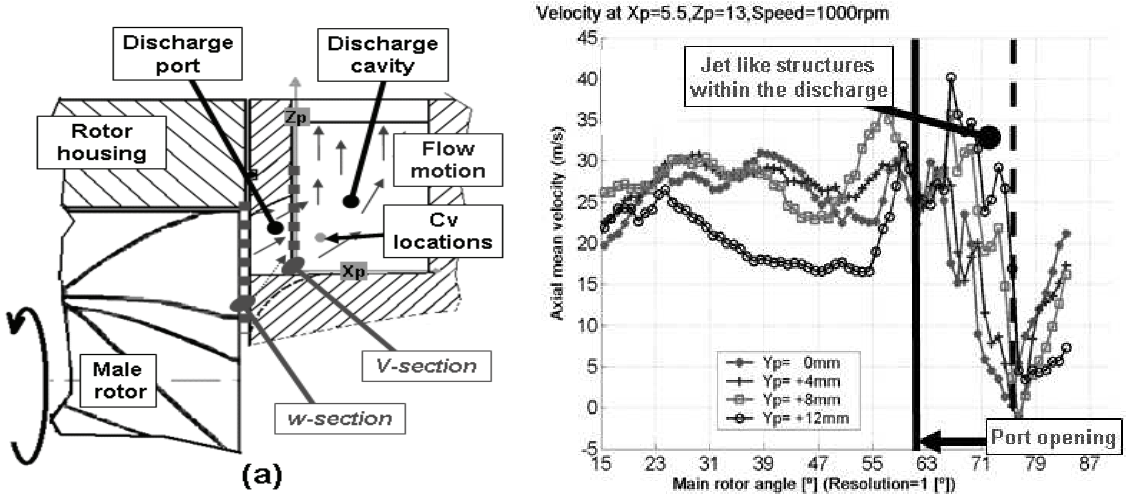


Figure 4: Measurement points in the discharge chamber (a)
LDV measured axial velocity component inside the discharge chamber (b)

Typical measured results obtained by LDV in the discharge chamber are shown in Figure 4. The axial mean flow velocities are obtained 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 related 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. (3) The jet flows create velocity peaks making the flow in that region highly turbulent.

4. VALIDATION OF CFD RESULTS BY LDV MEASUREMENTS

CFD calculations are obtained on the numerical mesh shown in Figure **1**. For the purpose of obtaining the grid independent solution, three different meshes were generated, the smallest consisting of 600000 numerical cells, the mid sized mesh consisting of 935000 numerical cells and the largest with 2.7 million cells. Results used for comparison with LDV measurements in this paper are obtained on the mid sized mesh. One full rotation of the male rotor that consisting of 300 time steps was sufficient to obtain a converged solution. Each time step took approximately 25 minutes to calculate on standard PC.

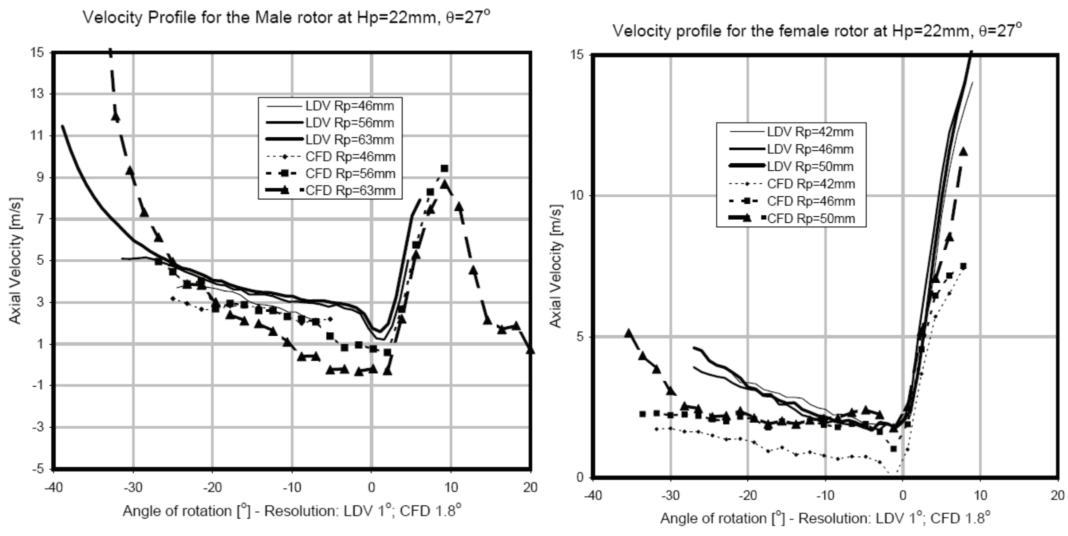


Figure 5 Comparison of the LDV and CFD axial velocities in the compression domain

Figure 5 shows a comparison of the axial mean velocities in the compression chamber close to the discharge port. This figure shows a very good agreement throughout Zone (1) and Zone (2), as specified in **Error! Reference source not found.**. In Zone (3), both the measured and calculated velocities increase but the increase in calculated velocities is larger than in the measured ones. It is believed that this difference is due to the inability of the k-e turbulence model to cope with near wall flows in the large numerical cells. Such a configuration of the numerical mesh is a consequence of the methodology used for the generation and mesh movement which is in detail explained in (*Kovacevic et all, 2006*) [0].

Figure 6 shows a comparison of the axial velocities in the discharge port. The differences appear to be rather large but trends and mean values are very similar. Calculations confirmed 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.

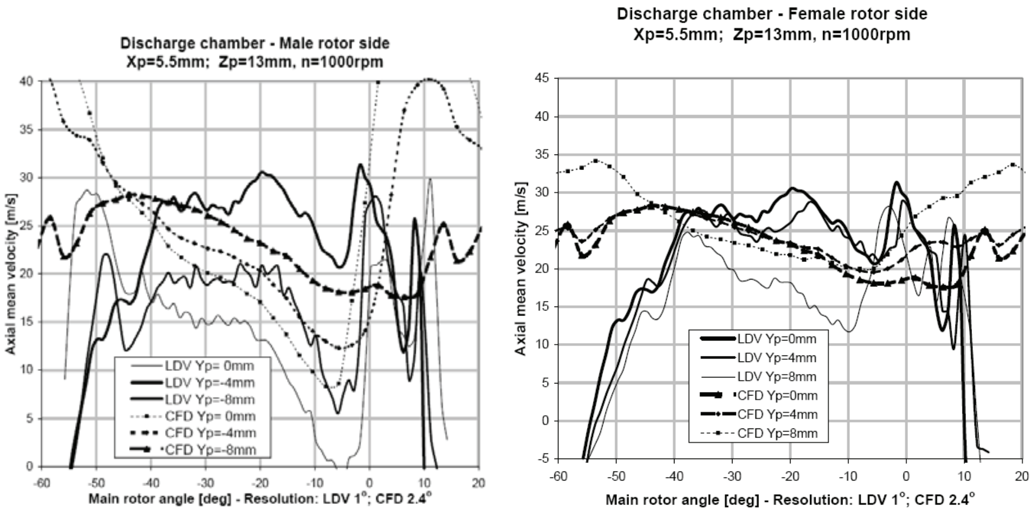


Figure 6 Comparison of the measured and calculated axial velocities in the discharge chamber

The measurements suggest that turbulence plays significant role in the discharge port where narrow passages connect the compression chamber and the discharge domain. The inability of the existing turbulence model to cope with near wall velocities properly seems to be the main reason for differences in the CFD results and measurements. Therefore further research into turbulence models for internal flow in compressor ports is necessary.

5. CASE STUDIES

Four case studies are presented to demonstrate flexibility of the method, namely 1) Estimation of pressure oscillations for noise prediction in a screw compressor discharge port, 2) Investigation of cavitation in a helical gear pump, 3) CFD modelling of a multiphase screw pump.
In all cases a numerical mesh was obtained by the in-house grid generator described earlier in this paper. However, the CFD calculations were performed by a different CFD numerical solver for each of presented cases.

CFD for Noise Prediction

Identification of sources of noise and noise attenuation has become an important issue for the majority of screw compressor applications. Pressure fluctuations in the discharge port not only generate aero acoustics in that domain but also induce mechanical noise due to rotor rattling. It was confirmed in previous studies that adequate porting can decrease the noise level and improve the machine performance. A thermodynamic model was set up to estimate pressure oscillations as a function of the discharge port shape and the cross sectional area of the connecting flange. These predictions are convenient for the estimation of the main noise harmonics. However, this model does not take into account the shape of the discharge chamber which probably plays an important role in the generation of higher harmonics. Therefore further steps were undertaken to analyse pressure fluctuations in the discharge port by use of a full 3-D CFD code, Figure 7.

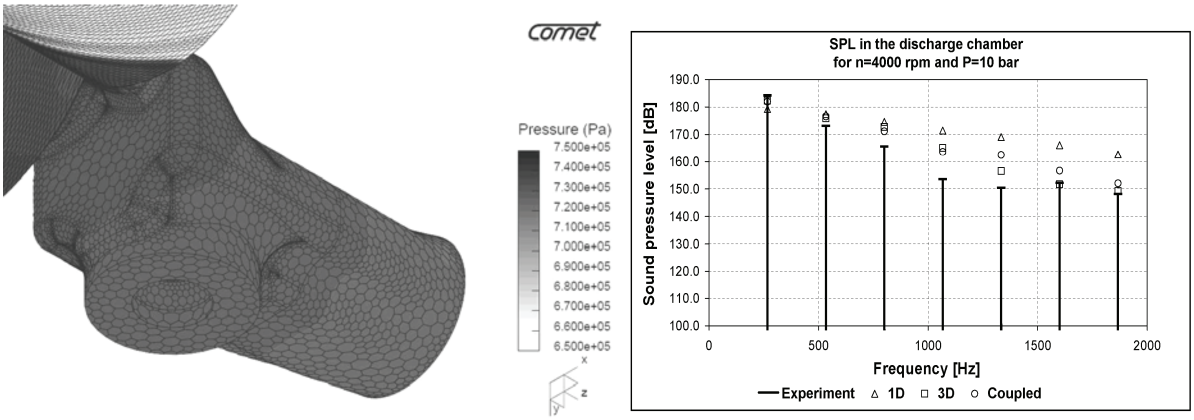


Figure 7 Pressure oscillations in the 3d CFD model of the discharge chamber

The results obtained by the 3-D model agree very well with measurements (*Mujic et all, 2008*) but the model appears to be too computation intensive for everyday industrial use. Therefore an integrated model was developed which combines the accuracy of the full 3D model and the speed of a thermodynamic chamber model (*Kovacevic et all, 2007*). A comparison of the Chamber mode, full CFD, integrated CFD and measurements is presented in Figure 7. More details are given in *Stosic, 1989* and *Mujic et all, 2009*.

Cavitation in gear pumps

Gear pumps are often used in the automotive and aero industries to supply fuel to an engine. A fuel gear pump geometry is very similar to that of a screw compressor. It has two rotors with either straight or helical lobes which are contained in a housing and connected to other flow paths of the system through the suction and discharge chambers. The left side of Figure 8 shows a numerical mesh of such a gear pump. The hexahedral numerical mesh of rotors and flow around them was generated by SCORG, while the stationary parts were meshed by ANSYS CFX and ICEM tools into a tetrahedral mesh. These two domains were connected through transient sliding interfaces, readily available in the CFX solver.

Erosion damage, caused by cavitation, was noticed on the running pump at the rotor shafts and within the gaps. The results presented here have been previously given in (*Steinman, 2006*). The authors demonstrated the capability to generate the numerical mesh in such machines by the use of the SCORG software. The CFD calculation showed that cavitation occurred in the flow through the interlobe gaps in the direction towards the suction chamber. It was outlined in [0] that the main challenge for a successful computation is a relatively complex geometry of the moving and deforming grids, as well as the transient interface. This was fully overcome by use of the SCORG software.

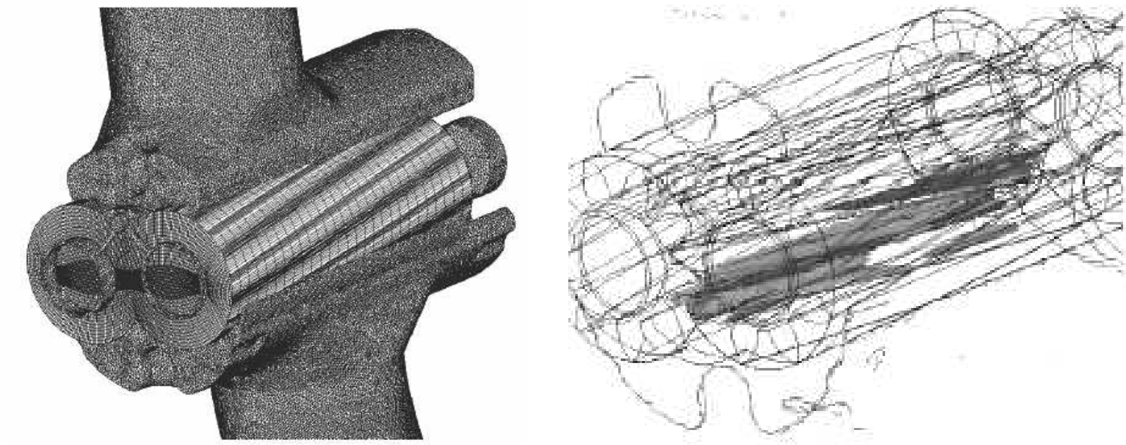
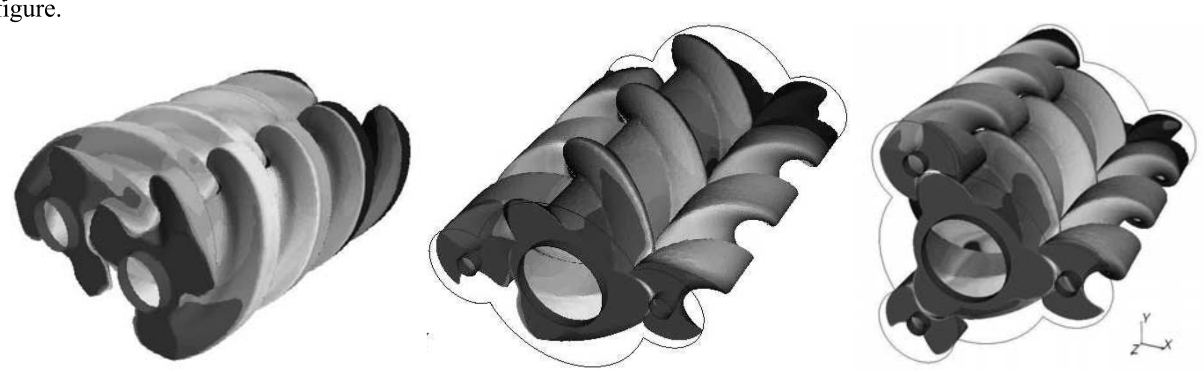


Figure 8 Numerical mesh of the gear pump and the occurrence of cavitation

CFD analysis of a multiphase screw pump
Multiphase screw pumps are regularly used in the oil and gas industry. A CFD analysis of the leakage flow and pressure distribution in these pumps has been calculated by Star CCM+. As an example, the pressure distribution on the first layer of cells of the 3/3 lobe combination rotors of a down hole pump passage flows for 1-10 bar pressure rise is presented in Figure 9. The leakage flow through the clearances and blow hole area is shown in the same figure.



.

Figure 9 Pressure distribution in various multiphase down-hole pumps

The numerical grid was generated by the SCORG grid generator, developed for screw compressors. In addition to the standard twin-screw arrangements, this grid generation software can generate multi-rotor arrangements as shown in Figure 9. The pressure distribution for the multi-rotor applications was calculated by StarCCM+. The machine with three female rotors shown in Figure 9achieves a smaller pressure drop between its interlobes then in the pump with two female rotors. This means that lower leakage flows are achieved in the machine with more female rotors. The integration of pressure force over the rotor surfaces shows that the load on the female rotors is more or less independent of their number.

6. CONCLUSIONS

Computational Fluid dynamics in screw compressors is advanced method used regularly for research and development purposes. Measurements of velocity field within screw compressor flow domains obtained by Laser Doppler Velocimetry confirmed validity of CFD results. 3D numerical modelling is therefore established as a means of calculating both local and bulk velocities within twin screw machines. It has been shown that the modelling accuracy may be further improved by local improvements in CFD modelling, including use of turbulence models suitable for complex pressure driven internal flows.
It has also been shown that CFD modelling can be used not only to estimate performance of screw compressors, but also for research in phenomena which could not be analysed by any other simplified models. Case studies confirm that a number of available CFD and CCM software packages can be used for this purpose if combined with the SCORG mesh generator. Such methods can further be used for predicting multiphase flows in screw and gear pumps, even in configurations that differ from classical twin screw arrangements.

REFERENCES

Alagic S., Stosic N., Kovacevic A., Buljubasic I, A Numerical Analysis of Heat Transfer and Fluid Flow in Rotary Regenerative Air Pre-Heaters, Strojniski Vestnik, Journal of Mechanical Engineering, Vol 51, 7-8, pp 411-417, 2005

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., 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., Mujic E., Stošić N, Smith I.K, An integrated model for the performance calculation of Screw Machines, Proceedings of International Conference on Compressors and their Systems, IMechE London, September 2007
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
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, 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
Mujic E, A Numerical and Experimental Investigation of Pulsation Induced Noise n Screw Compressors, PhD Thesis, City University London, 2009
Peric M.: Flow simulation using control volumes of arbitrary polyhedral shape, ERCOFTAC Bulletin, No. 62, September 2004
Steinman A, 2006: Numerical Simulation of Fluid Flow in Screw Machines with Moving Mesh Techniques in ANSYS CFX," Schraubenmaschinentagung 2006", Dortmund VDI Berichte
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.
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