

Turbulence Capture in CFD for Screw Machines

Prof. **A. Kovacevic**, Ms. **M. Kethidi**, Prof. **N. Stosic**, Dr. **E. Mujic**, Prof. **I.K Smith**

Centre for Positive Displacement Compressor Technology, City University London, United Kingdom.

Abstract

Previous publications show that Computational Fluid Dynamics (CFD) can be readily used for the flow prediction and analysis of screw machines. In order to validate the accuracy of the CFD calculations, results from measurements obtained by Laser Doppler Velocimetry (LDV) are compared with the CFD simulations. The results not only confirm the viability of the developed methodology but also show the potential for further improvement in modelling of screw machines.

One of the areas for improvements in accuracy of results from CFD models is the use of an appropriate turbulence model. Therefore, extensive activity is being undertaken to review available turbulence models for use in the analysis of internal flows in screw compressors. It is shown that the flexibility of the method used for CFD calculation together with appropriate turbulence modelling may increase the scope of application of CFD in twin screw machines.

INTRODUCTION

Screw compressors are today commonly used for handling air, process gases or refrigerants. Accuracy in modelling of such machines is therefore paramount for competitive application of new compressors and improvements in existing machines. Among others, numerous reviews and publications over the past 30 years have summarized various levels of approach already in use for the study of flow in screw compressors. Only recently some publications include 3-D numerical analysis of screw compressors [1], grid-generation in screw compressors [2], 3-D numerical performance estimation [3], solid-fluid interaction in screw machines [4], prediction of flow generated noise in screw machines [7], cavitation modelling in gear pumps [9], and flow in multiphase pumps.

An important requirement for the successful design of all types of compressors is an ability to predict accurately the effects of changes in design parameters on performance. In order to optimize the design it is necessary to understand the flow in the suction, discharge and working chambers and especially, through the clearance gaps, so as to characterize the whole sequence of processes that occur within a compressor.

In recent years there has been a steady growth in the use of Computational Fluid Dynamics (CFD) as a means of calculating 3-D internal and external flow fields [12], [17], [3]. It is widely used today for estimating flow in screw compressors and specialized codes have been developed for this to increase the speed of calculation. But there is still a need to reduce computational time and increase the accuracy of the results. This requires the development of even more specific procedures. Comparison of flow measurement results obtained by Laser Doppler Velocimetry (LDV) in screw compressor flow domains with predicted values suggested that errors in modelling exist and that improving the turbulence modelling can provide more accurate and faster CFD calculations [11].

In spite of the numerous publications in the field of screw machines, very few authors have analyzed the effects of turbulence. Examples are the work of Vimmr 2006 [12], [13], 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 in that region 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 [1], who later confirmed that further validation of full 3-D CFD calculation results could not be obtained by the use of simplified numerical or experimental methods. However, it was also shown that the use of alternative differencing schemes and turbulence methods influences local velocity and pressure values in certain machine regions. Although these differences have a low impact upon the overall performance [1], their influence upon flow development needs further investigation. For this, a full understanding of the effects of turbulence in the machine suction, compression and discharge chambers is needed. The material presented in this paper is a part of the long term research project aimed to investigate, develop and validate suitable turbulent models for accurate CFD calculation in screw compressors.

LASER DOPPLER VELOCIMETRY IN SCREW COMPRESSORS

The instrumentation for measuring complex flows in a screw compressor must be robust to withstand the unsteady aerodynamic forces and oil drag, must have a high spatial and temporal resolution and most importantly may not disturb the flow. Point optical diagnostics, such as LDV [10] can fulfil these requirements. In order to measure flow velocities inside a screw compressor, an experiment, using this technique, 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, [11].

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 1. After machining, the internal and external surfaces of the window were 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 1. Details of the used equipment are given in [11]

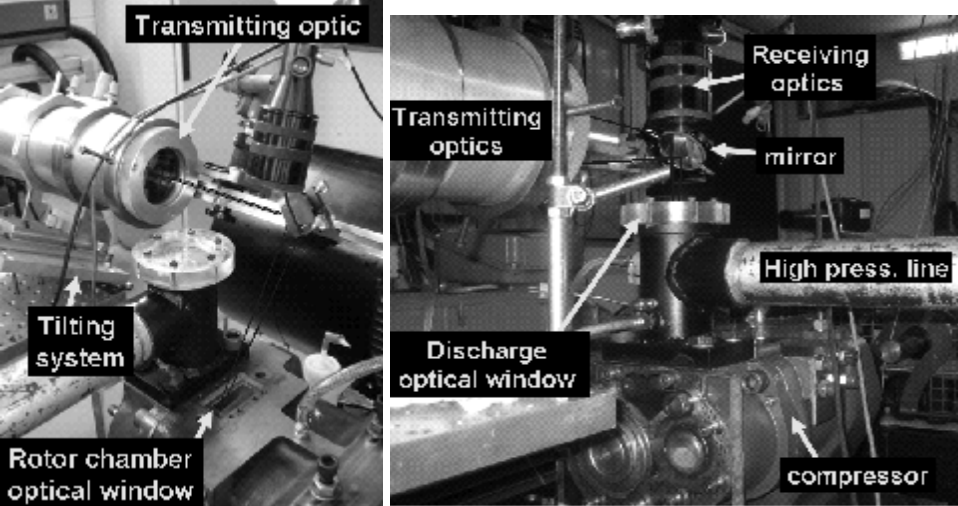


Figure 1 Optical compressor (left), LDV optical set for discharge chamber (right)

In order to measure velocities within the rotor chamber, two coordinate systems were defined one for the male and the other for the female rotor. The female rotor coordinate system is shown in Figure 2(a). Measurements were obtained at $R_p=48, 56, 63.2\text{mm}$, $\alpha_p=27^\circ$ and $H_p=20\text{ mm}$ for the male rotor, and at $R_p=42, 46, 50\text{ mm}$, $\alpha_p=27^\circ$ and $H_p=20$ for the female rotor.

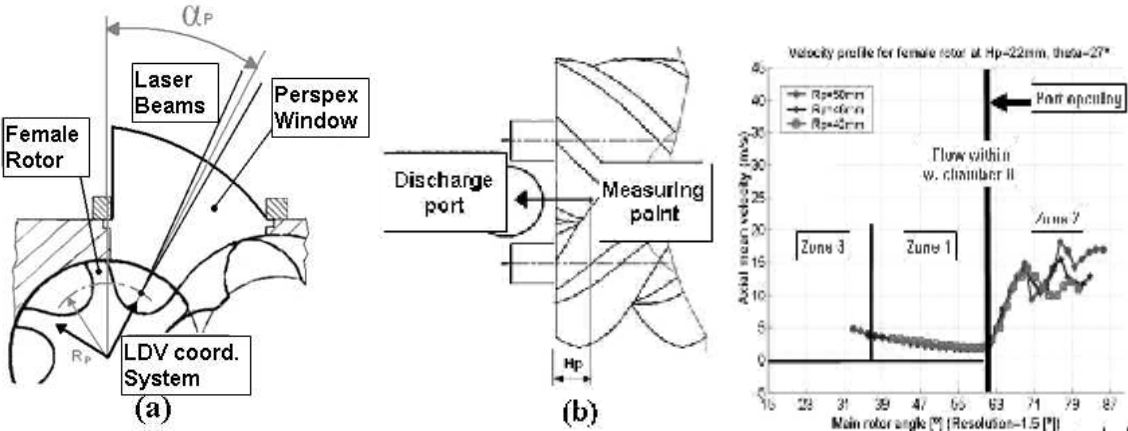


Figure 2 (a) Coordinate system and the window; (b) Axial plane view, (c) LDV measurements

Typical velocity values measured in the working chamber are shown in Figure 2(c). Three zones are identified. Zone (1) is the trapped working domain with fairly uniform velocities. Zone (2) is the opening of the discharge port. The velocities and turbulence in this zone are much higher than in Zone (1). The flow in this zone is driven by the pressure difference between rotors and the discharge chamber. Zone (3) is the radial leakage flows. Velocities here increase to values higher than in Zone (1) but the flow is not as turbulent as in Zone (2). Conclusions derived from the measurements are explained in more detail by Guerrato, 2007 [11], and are summarised here as follows:

- Chamber-to-chamber velocity variations are higher near the leading edge of the rotor.
- 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
- The effect of the 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.

A schematic arrangement of the measurement points in the discharge chamber is shown in Figure 3. The chamber is physically divided into the discharge port domain and the discharge cavity. The coordinate system in Figure 3 identifies the location of the measured CV. Measurements were made at $X_p=5.5\text{mm}$, $Z_p=13\text{mm}$ and $Y_p = -8$ to 13mm .

Typical measured results obtained by LDV in the discharge chamber are shown in Figure 3(b).

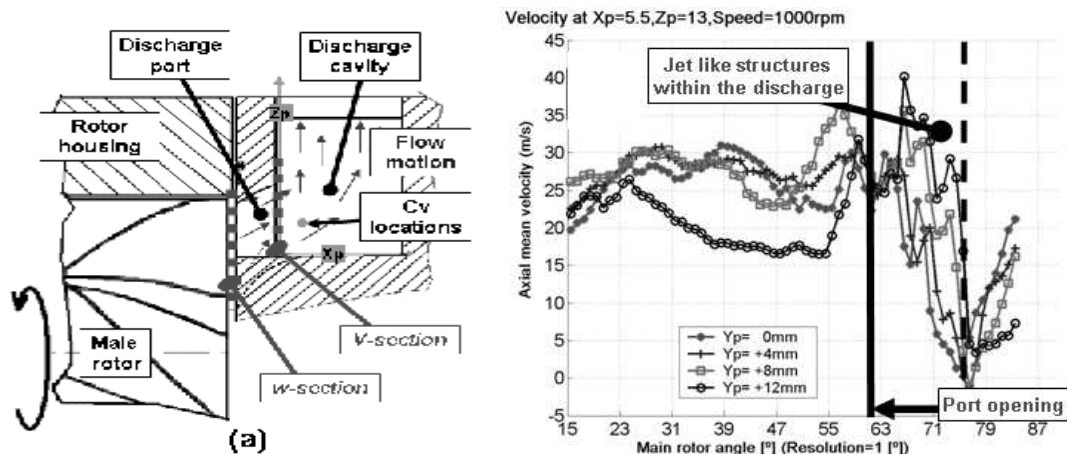


Figure 3 Measurement points in the discharge chamber (a)

LDV measured axial velocity component inside the discharge chamber (b)

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.

- Velocities are higher than in the compression chamber due to fluid expansion in the port between sections W and V.
- 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 than one working chamber is connected.
- The jet flows create velocity peaks making the flow in that region highly turbulent.

COMPARRISON OF CFD RESULTS AND LDV MEASUREMENTS

In order to identify differences between the CFD calculations and the measurements, a numerical mesh was generated consisting of 935000 numerical cells. Standard K- ϵ turbulence model was used. One full rotation of the male rotor consisting of 300 time steps was sufficient to obtain a converged solution. Each time step took approximately 25 minutes to calculate on a standard PC. This comprehensive study of the validation of the CFD results by LDV measurements is described in detail in [6]. In this paper, only the most important findings related to the turbulence modelling are presented.

Figure 4 shows a comparison of the axial mean velocities in the compression chamber close to the discharge port.

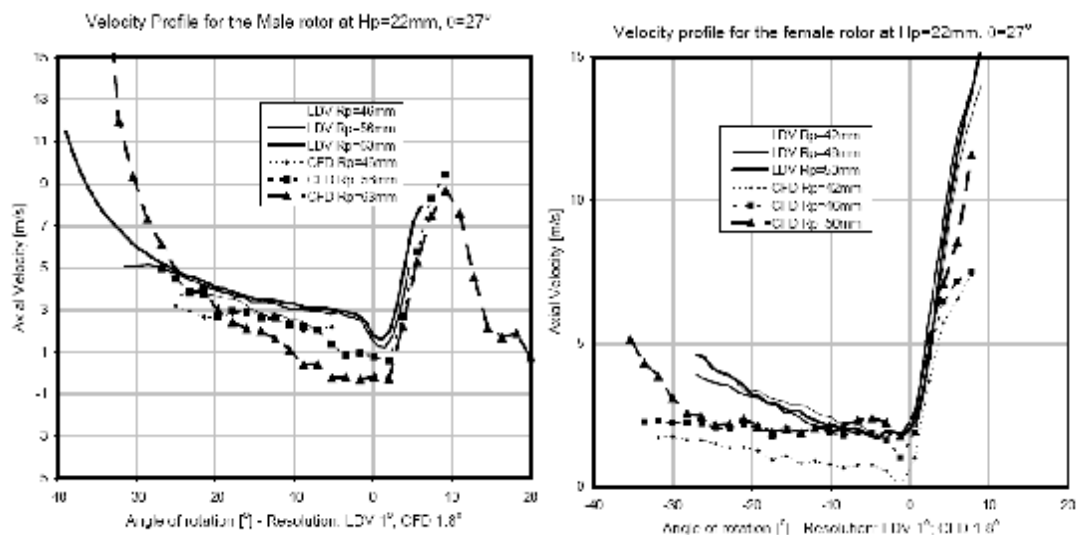


Figure 4 Comparison of the LDV and CFD axial velocities in the compression domain

Good agreement between measured and calculated data is observed for zones (1) and (2). 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-ε turbulence model to cope with near wall flows in the large numerical cells [1]. The negative velocities measured in the transition between Zone (2) and Zone (3) are due to backflow to the compressor. These are not fully captured by CFD. Comparison of the axial velocities in the discharge port is shown in Figure 5. The differences seem to be rather large but trends and mean values appear to be similar.

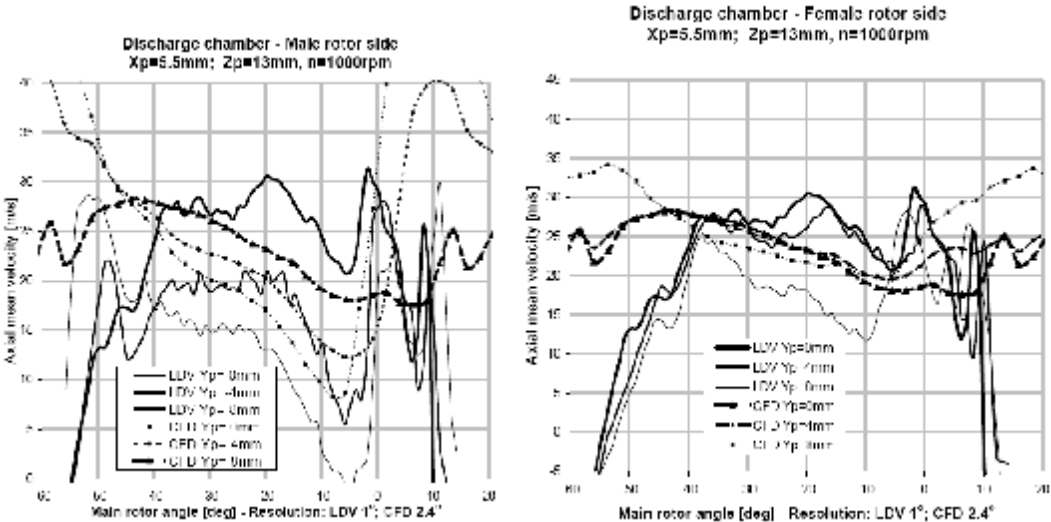


Figure 5 Comparison of the measured and calculated axial velocities in the discharge chamber

The measurements suggest that turbulence plays a 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 properly cope with the near wall velocities seems to be the main reason for differences in the CFD results and measurements.

Both of the presented cases indicate that further research into turbulence modelling for internal flows in a screw compressor is necessary.

INFLUENCE OF TURBULENCE SCHEME ON THE CFD RESULTS

Following the findings of the validation of the CFD results presented in the last section, a study was performed to look into the application of readily available, well known turbulence models for CFD calculation with a commercial CFD code and evaluate differences between the results. These initial results will then be used to decide which of the more complex turbulence model should be evaluated further and give clear directions for further investigation into the effects of turbulence modelling on the results of CFD calculations in screw compressors.

This study was performed on an oil-free screw air compressor with 4/5 lobe configuration of N^o rotor profile rotors. The male rotor outer diameter is 234 mm and the rotor length is 363

mm. The compressor was set to operate between 1 bar suction and 2.15 bar discharge pressures. By studying an oil free compressor the effects of different turbulence models can be analysed avoiding other effects, such as oil injection.

3D Numerical mesh

Figure 6 shows the 3D numerical mesh of the screw compressor domains used in this study. Most of the parts of the compressor flow domains, including those around the moving rotors, suction port, and suction and discharge receivers are mapped with a hexahedral block structured mesh obtained by the in-house grid generation software called SCORG (Screw Compressor Rotor Geometry Grid generator) [1]. This software enables numerical mapping of both, the moving and stationary parts and their direct integration in commercial CFD or Computational Continuum Mechanics (CCM) codes. The numerical mesh of the discharge domain was generated directly from the 3-D CAD model by the use of a commercial grid generator [18]. The mesh of the discharge port contained 32,719 grid elements while the entire numerical mesh had 1,254,511 cells.

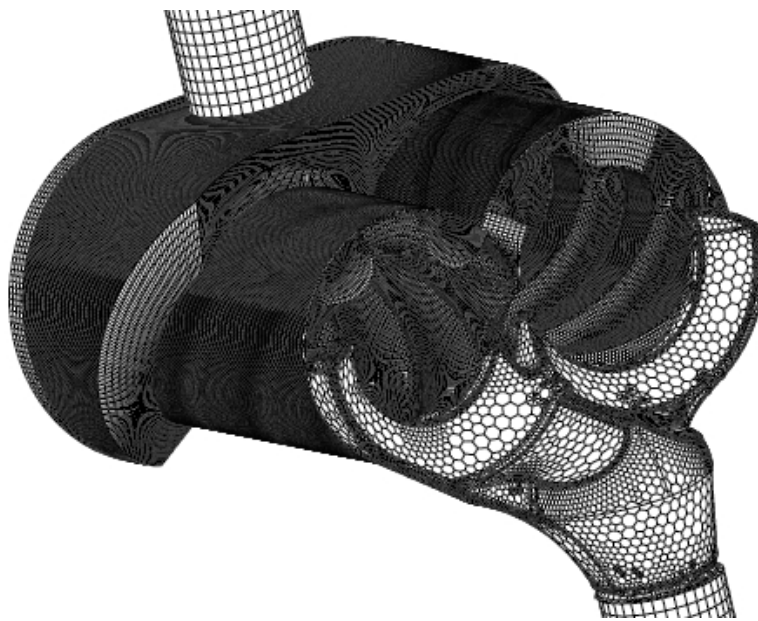


Figure 6 Numerical Mesh of flow domains within a Screw Compressor

CFD Calculations

The calculations were performed with a commercial CCM software COMET of the Adapco-CD group [18]. The solution was obtained for a stationary case at a fixed rotor position in order to avoid uncertainties of the rotor movement. Four cases were calculated, namely 1) assuming laminar flow, 2) with standard k-e turbulence model, 3) with a Wilcox k-w turbulence model, 4) with an RNG model of turbulence. All three mentioned turbulence

models are often used in practice, are readily available in the CFD software and are evaluated in the open literature in detail [14], [15], [16]. A converged solution in all cases was obtained after no more than 75 iterations.

The calculation of the laminar case on the numerical mesh produced by the current methodology showed some inconsistencies in the results in some regions of the flow domain in suction and discharge ports. The discharge port is shown in Figure 7.

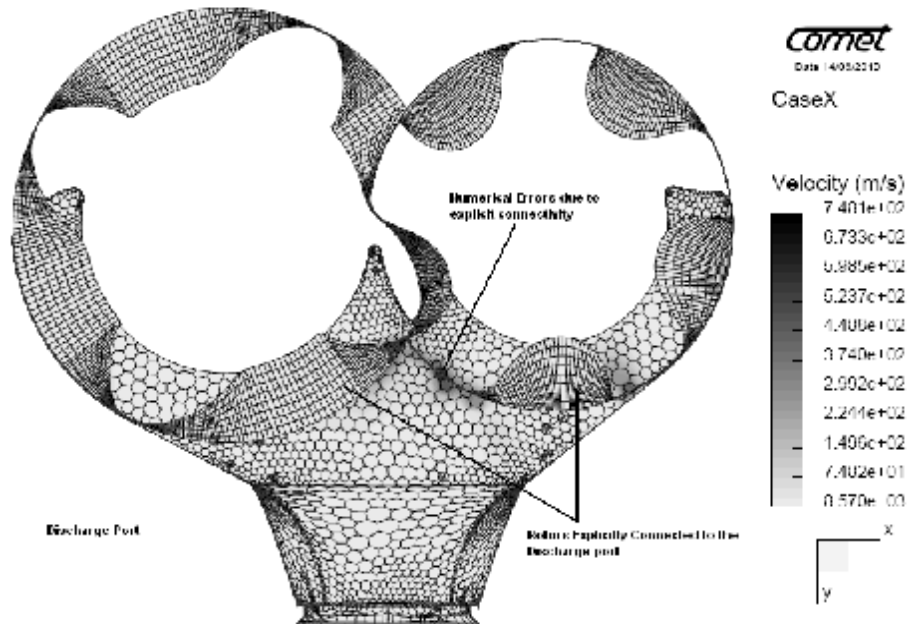


Figure 7 Velocity magnitude in the axial section on the discharge sliding interface

These can be attributed to numerical errors and are summarized as follows.

- Due to the high geometric ratio of the main compressor chamber to the clearances, velocities in the clearances appeared to be highly transonic. It is believed that this is a consequence of numerical error caused by the applied differencing scheme. Calculations repeated with turbulent models showed reduced velocities values.
- The velocity values in the neighbouring cells on two sides of the sliding interface showed inconsistencies. This is particularly visible in places where small clearances in the rotor domains connected to the large flow domain of the discharge port. It is believed that the inconsistent velocity values are the consequence of the numerical error caused by the mapping method used for interface representation.

Both previous errors may be corrected either through more appropriate numerical generation in the identified regions or through development of a mapping procedure which might mitigate the problems associated with the existing procedure.

Evaluation of calculated results

The evaluation of the differences between results calculated with different turbulence models was performed for two characteristic regions in the screw compressors namely, the axial interface between the suction chamber and the moving rotors, shown in Figure 8 and the axial interface between the discharge chamber and the moving rotors, shown in Figure 10.

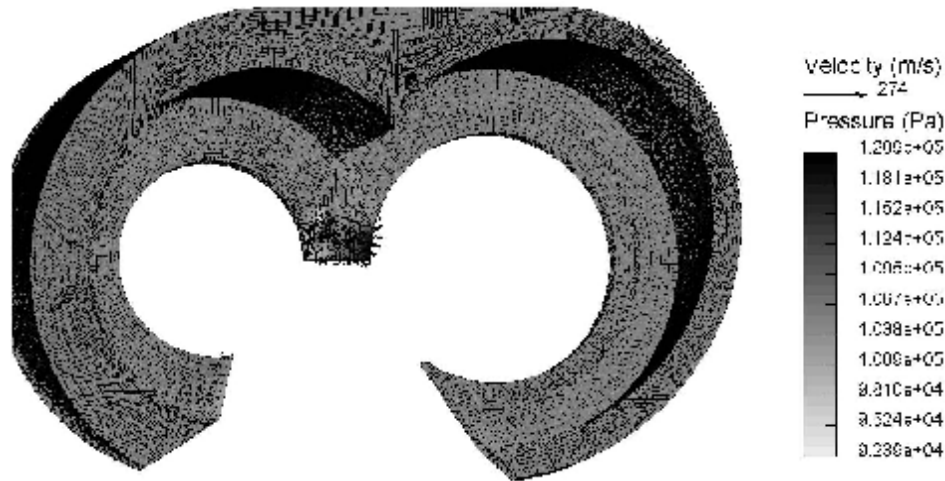


Figure 8 Axial part of the suction port directly connected to the end face of rotors

The results presented here consider only regions which are not affected by numerical errors identified in the laminar case. The velocity profiles for three analysed turbulence models are plotted in Figure 9 (a), (b) and (c). The horizontal axis represents the linear distance along the inside diameter of the axial suction port, measured from the point where the axial and radial ports of the female rotor are joined. It is observed that for all three cases the mean velocity value remains almost the same while the amplitude of fluctuations changes depending of the turbulence model applied. The fluctuation peaks are associated with the appearance of the small clearance between the rotors and the casing on the opposite side of the investigated reference plane, which represents the sliding interface. The amplitude fluctuations obtained by the k- ϵ model are higher compared to those of the other two models. The magnitude of the fluctuations increases near the clearances for all the three models. The velocity values are plotted in the vicinity of the wall around the axial part of the compressor against the relative wall distance y^+ , as shown in Figure 9d. For the k- ϵ and RNG models, the transition from laminar to turbulent flow takes place around $y^+ > 100$. The laminar layer for these models ranges approximately from $1 \leq y^+ \leq 100$. The transition layer varies approximately from $100 < y^+ < 700$ for both k- ϵ and RNG models of turbulence. The outer layer with dominant turbulent shear starts at $y^+ > 700$. These are well captured by the predictions. In contrast, the k- ω model does not predict the log wall within the y^+ range of 1 to 1000, which means that even at that distance the flow does not convert to fully turbulent.

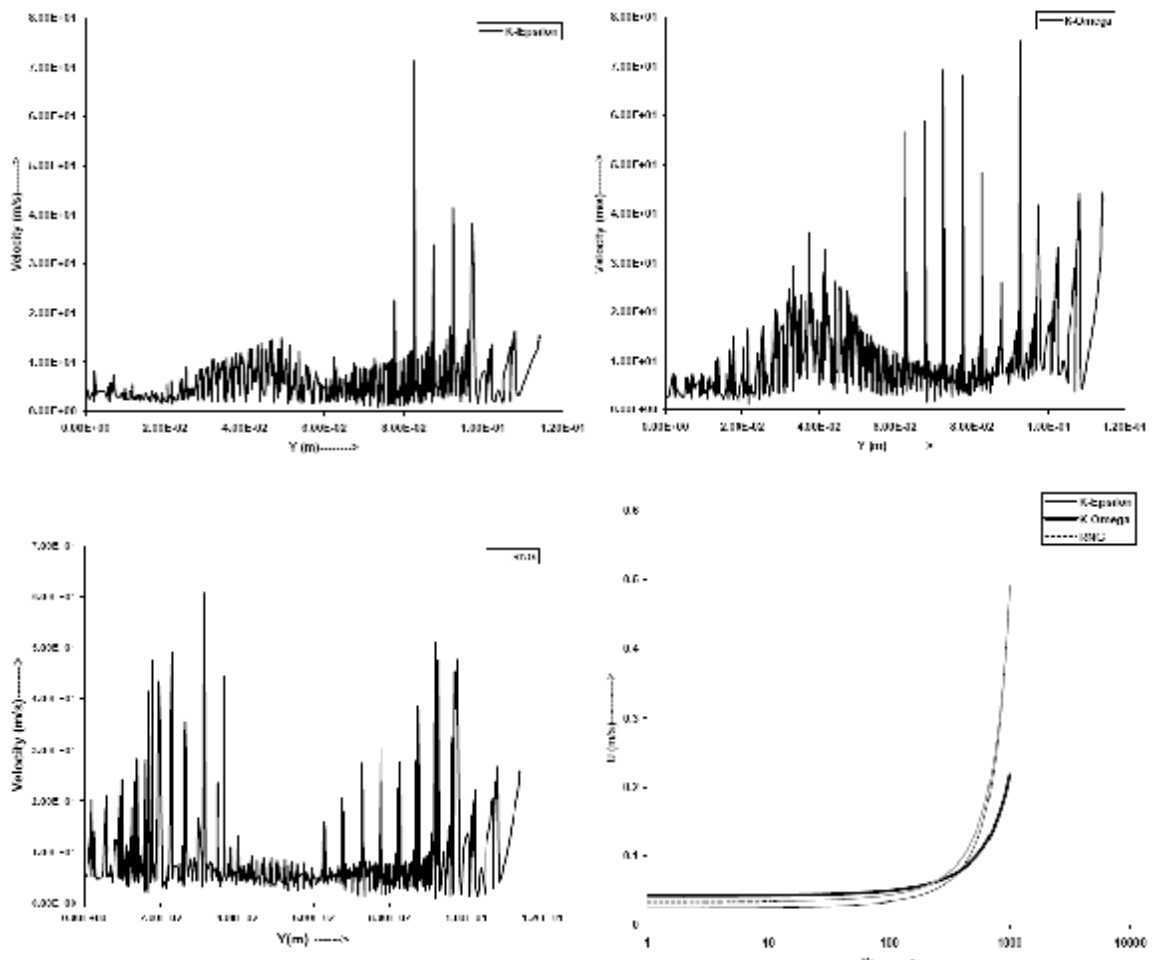


Figure 9 Velocity Distribution in the axial plane of the suction port exposed to the rotors (a) K-Epsilon (b) K-Omega (c) RNG K-Epsilon, (d) Velocity vs y^+ in the suction port



Figure 10 Axial part of the discharge port directly connected to the rotors

Figure 10 shows the sliding interface on the axial part of the discharge port, through which the flow domains around the compressor rotors are connected, to the stationary flow domain of the discharge port. High velocity values recorded there are the consequence of the leakage through small clearances from the trapped chamber with higher pressure directly to the discharge port.

Figure 11 (a), (b) and (c) show the velocity profiles for the axial part of the discharge port. The values on the horizontal axis are measured along the outer diameter of the rotors, starting from the end port point on the female side of the discharge port, shown in Figure 10. The velocities are higher than in suction port due to the reduction in the discharge port flow area. The peaks are related to the small clearances between the rotors on the opposite side of the sliding interface. The results of different turbulence models differ in amplitude of the velocity fluctuations while the general average values remain almost the same.

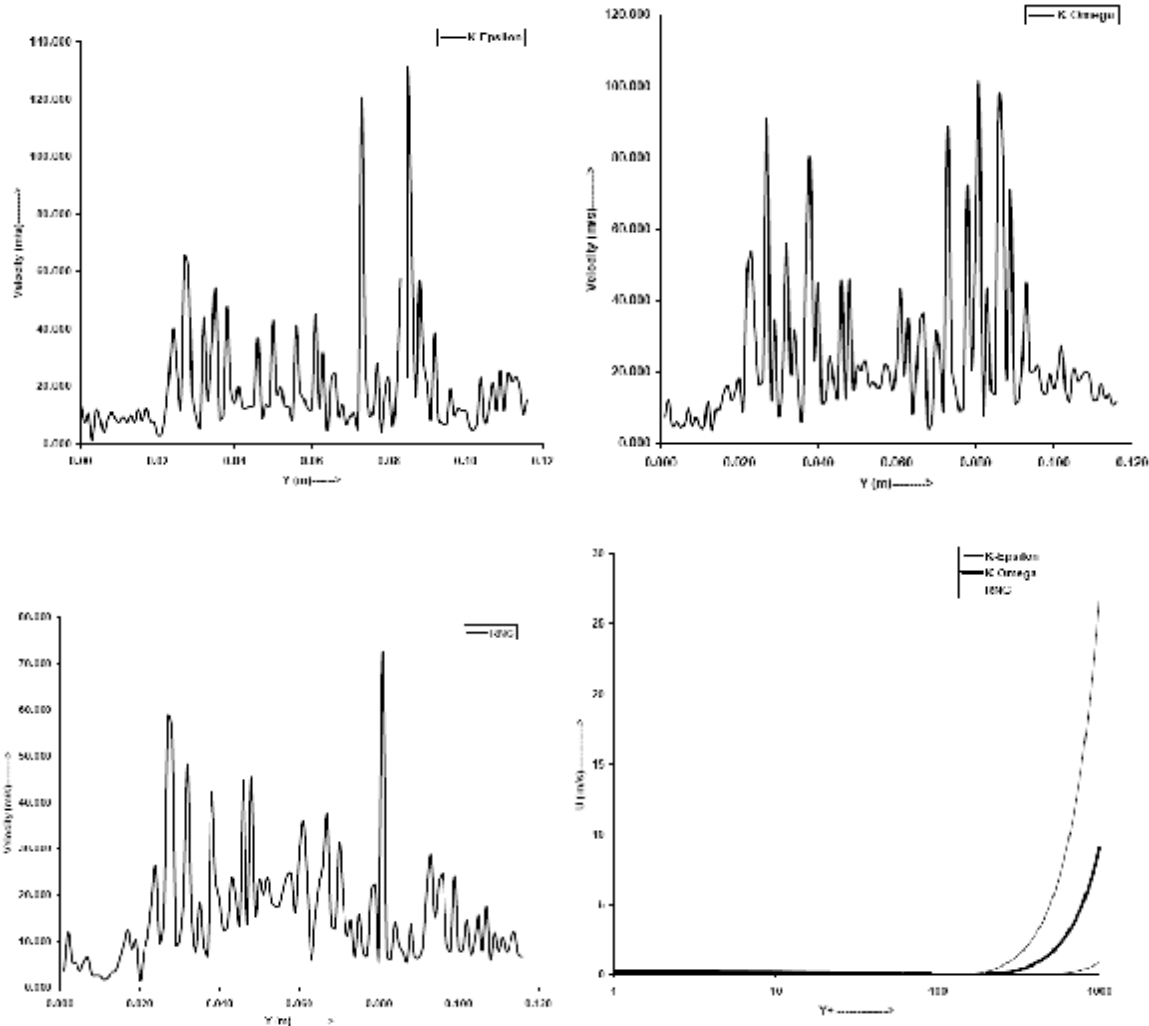


Figure 11 Velocity Distribution of the axial discharge port exposed to rotors
 (a) K-Epsilon (b) K-Omega (c) RNG K-Epsilon (d) velocity vs y+ for the discharge port

Figure 11 d) shows the velocity vs y^+ for the discharge port. Transition from laminar to turbulent flow takes place approximately around $y^+ = 90$ for all the three models. However, only the RNG model of turbulence fully predicts the log law of the wall.

Based on the findings of this chapter and particularly from **Fehler! Verweisquelle konnte nicht gefunden werden.** and Figure 11d, it can be concluded that turbulence plays a more significant role in the suction part of the compressor than in the discharge domain. Following this, it would be expected that distribution of kinetic energy of turbulence and its dissipation will lead to a similar conclusion. In previous publications [1] it was identified that the kinetic energy of turbulence and its dissipation rate are high in certain regions of the compressor while they do not play a significant role in others. Figure 12 shows the distribution of kinetic energy of turbulence at various cross sections of the flow domain for the case studied in this paper. Figure 13 gives the dissipation rate for the same rate. The presented results are obtained with the k- ϵ turbulence model.

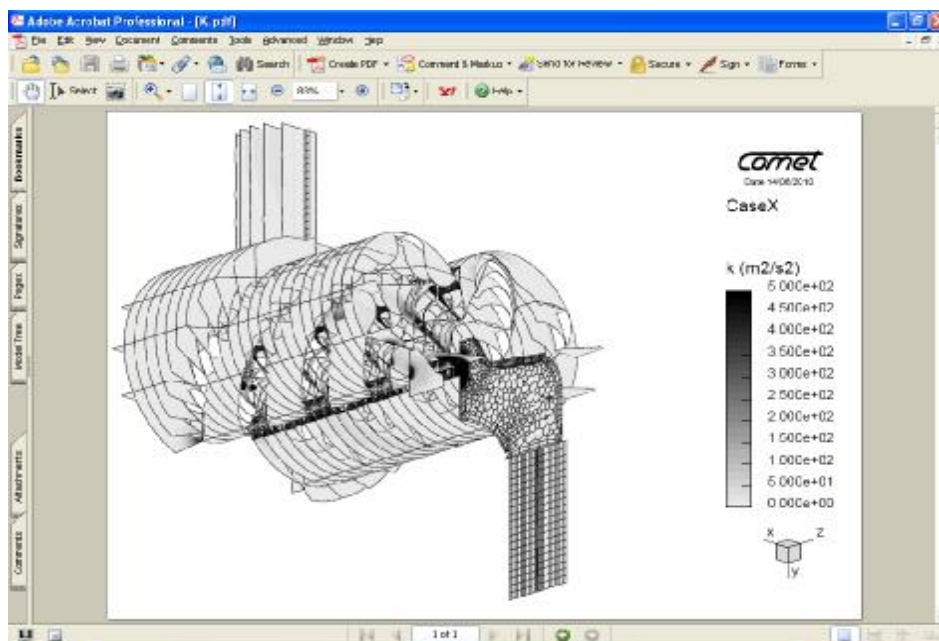


Figure 12 Kinetic energy of turbulence calculated with k- ϵ turbulence model

It is noticed that a higher dissipation rate is experienced at the discharge side of the compressor where velocity fluctuations are larger. However, based on the presented diagram, kinetic energy is generated almost equally in the suction and discharge regions but significantly more in clearances than elsewhere. It is likely that configuration of the numerical mesh used, with large boundary cells, may limit the capability of existing turbulence methods and cause irregularities and instability in calculations. More precise evaluation of these two properties which define turbulent flow is therefore required to fully evaluate the influence of

turbulence in screw compressors. However, this would require a numerical mesh with a much larger number of numerical cells and shorter time steps. The existing methodology for generation of the numerical mesh has limits due to the configuration and the relation of the size of the mesh and speed [1].

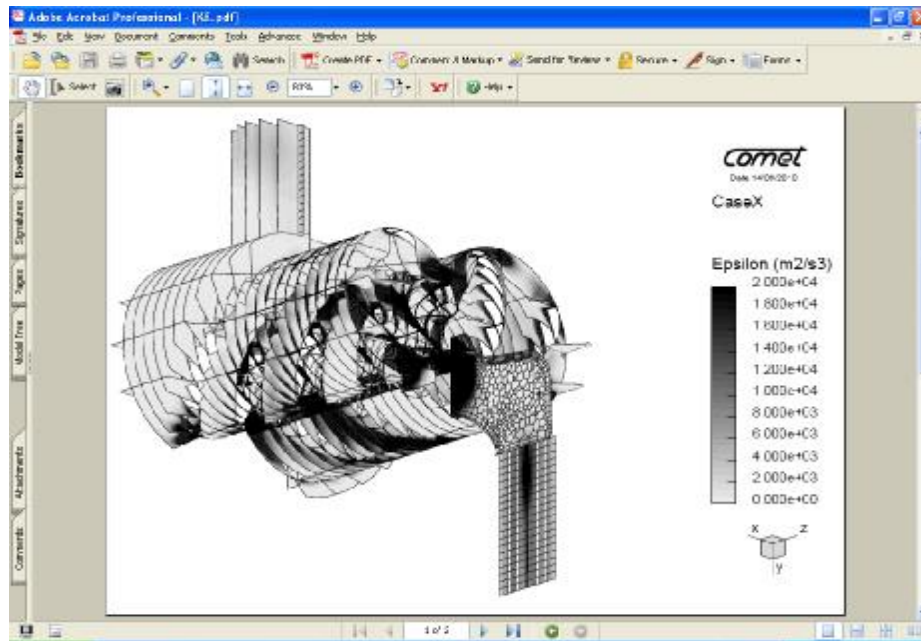


Figure 13 Dissipation rate calculated with k-e turbulence model

Based on this paper it is evident that turbulence plays a role in screw compressor performance prediction. Despite small differences in integral parameters, the used models show significant difference in local values which are strongly dependent on the turbulence treatment. This is not in full agreement with the recent investigation of Fryc and Vimmr [13] who claimed that the use of different turbulence model will not change the results significantly and therefore it requires further analysis. There is a possibility that more advanced methods such as, for example, the Menter shear stress transport turbulence model, the Nut-92 turbulence model or the Spalart-Allmaras turbulence model [15], [16] or even their combination may give more consistent results and be more suitable for internal flows in screw compressors on numerical meshes of this existing quality.

Future investigation in turbulence will be oriented towards the development of a universal or compound near-wall turbulent treatment which will combine a robust integration to the wall, for good near-wall grid resolution, and generalised wall functions for coarse near-wall grids. One or more improved Reynolds Averaged Navier-Stokes (RANS) turbulence models in conjunction with the above compound wall treatment may be used. It may even be possible

to apply one of the most promising hybrid methods of Large Eddy Simulation (LES)/RANS coupling, probably based on the 'grid detector' approach and using again the compound wall treatment where appropriate.

CONCLUSION

It is evident from measurements obtained by LDV that some effects of screw compressor flow are not always well captured by the existing $k-\epsilon$ turbulence model. Initial investigation in this problem concentrated on applying three standard turbulence models to consider difference. All of them gave different results of local velocity values in suction and discharge receivers. It is indicated that a more suitable turbulence model capable of analyzing flows in sliding and stretching domains of a screw compressor may need to be found or developed and validated.

Additionally it has been identified that some numerical issues occur in regions where very small and very large cells are connected. These are closely related to the current grid generation procedure and need to be addressed in order to allow easier application of an appropriate turbulence model.

REFERENCES

- [1] 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.
- [2] Kovačević A., Stošić N, Smith I.K, "Grid Aspects of Screw Compressor Flow Calculations", The 2000 International Mechanical Engineering Congress & Exposition, Nov 5-10, 2000, Walt Disney World Dolphin, Orlando, Florida.
- [3] 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.
- [4] 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
- [5] 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.

- [6] Kovačević A., Stošić N, Smith I.K, E. Mujic, 2008, Advances in Numerical and Experimental Investigation of Screw Compressors, International Conference on Compressor and Refrigeration, Xian, China, 2008
- [7] 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
- [8] 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
- [9] Steinman A, 2006: Numerical Simulation of Fluid Flow in Screw Machines with Moving Mesh Techniques in ANSYS CFX,” Schraubenmaschinentagung 2006”, Dortmund VDI Berichte
- [10] Nouri J.M., Guerrato D., Stosic N. and Kovacevic A., 2006, Cycle-resolved velocity measurements within a screw compressor. 18th Int. Compressor Eng. Conf., 17-20 July 2006, Purdue.
- [11] 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.
- [12] Vimmr J, Ondrey F, 2006, Numerical Simulation of Leakage Flow Between Moving Rotor and Housing of Screw Compressor, Proceedings of the Conference Modelowanie Inzynierskie, ISSN 1896-771X, 32, s. 461-468, Gliwice 2006
- [13] Fryc O., Vimmr J., “Comparison of Various Turbulence Models Applied to Leakage Flow Through a Male Rotor-Housing Gap of Twin-Screw Compressor”, Applied Mechanics 8th International Scientific Conference, 2006.
- [14] Hanjalic K., “Advanced Turbulence closure models: A View of current status and future prospects”, Lehrstuhl fur Stromungsmechanik, University of Erlangen –Nurnberg, Germany, 1994.
- [15] Ferziger J. H, Peric M, “Computational Methods for Fluid Dynamics”, ISBN 3-540-42074-6, Springer – Verlag, Berlin, Heidelberg, 2002.
- [16] Wilcox, D.C., 1993. Turbulence Modelling for CFD, DCW Industries, La Cañada, California.
- [17] 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
- [18] ICCM, 2001: COMET version 2.00 User manual, Hamburg, Germany