City Research Online

City St George's, University of London

Citation: Arjeneh, M., Kovacevic, A., Manolis, M. & Stosic, N. (2015). Numerical and Experimental Investigation of Pressure Losses at Suction of a Twin Screw Compressor. IOP Conference Series: Materials Science and Engineering, 90(1), 012006. doi: 10.1088/1757-899x/90/1/012006

This is the published version of the paper.

This version of the publication may differ from the final published version. To cite this item please consult the publisher's version.

Permanent repository link: https://openaccess.city.ac.uk/id/eprint/13764/

Link to published version: https://doi.org/10.1088/1757-899x/90/1/012006

Copyright and Reuse: Copyright and Moral Rights remain with the author(s) and/or copyright holders. Copies of full items can be used for personal research or study, educational, or not-for-profit purposes without prior permission or charge, unless otherwise indicated, provided that the authors, title and full bibliographic details are credited, a hyperlink and/or URL is given for the original metadata page and the content is not changed in any way. For full details of reuse please refer to City Research Online policy.

City Research Online: http://openaccess.city.ac.uk/ publications@citystgeorges.ac.uk

IOPscience iopscience.iop.org

Home Search Collections Journals About Contact us My IOPscience

Numerical and Experimental Investigation of Pressure Losses at Suction of a Twin Screw Compressor

This content has been downloaded from IOPscience. Please scroll down to see the full text.

2015 IOP Conf. Ser.: Mater. Sci. Eng. 90 012006

(http://iopscience.iop.org/1757-899X/90/1/012006)

View the table of contents for this issue, or go to the journal homepage for more

Download details:

IP Address: 138.40.68.28
This content was downloaded on 17/03/2016 at 14:34

Please note that terms and conditions apply.

Numerical and Experimental Investigation of Pressure Losses at Suction of a Twin Screw Compressor

M Arjeneh, A Kovacevic, S Rane, M Manolis, N Stosic

Centre for Positive Displacement Compressor Technology, City University London, London, United Kingdom, EC1V 0HB

E-Mail: Mohammad.Arjeneh.1@city.ac.uk

Abstract. Rotary twin screw machines are used in the wide range of industrial applications and are capable of handling single and multiphase fluids as compressors, expanders and pumps. Concentration of liquid in the inlet flow can influence the performance of the machine significantly. Characteristics of the multiphase flow at the suction of a screw compressor depend on the local flow velocities and concentration. Local flow velocity measurements inside the screw compressors are difficult to obtain. However other flow properties such as local pressures are easier to attain. It is therefore useful to carry out experiments with local pressure variations in the suction which can be used to validate the 3D numerical Computational Fluid Dynamic (CFD) models that could help in studying the single and multiphase flow behaviour in screw compressors.

This paper presents experimental efforts to measure the local pressure losses inside the suction plenum of the screw compressor. Pressure variations are measured at 23 locations in the suction port at various operating conditions and compared with 3D CFD model. The grid generator SCORG^{TM} was used for generating numerical mesh of rotors. The flow calculations were carried out using commercial 3D solver ANSYS CFX. It was found that the local pressure changes predicted by the CFD model are in the good agreement with measured pressures. This validated the use of CFD for modelling of the single phase flows in suction of screw machines.

1. Introduction

Screw compressor developers, manufacturers and users are constantly challenged to broaden the scope of application of such machines and operate them more efficiently than ever. One of the objectives is use of screw compressors for handling of multiphase flows. Chamber models such as the one embedded in SCORPATH, as discussed in [1], provide very good accuracy in predicting standard oil injected and oil free compressor performance. However, despite of the good performance predictions with high gas volume fraction in a multiphase suction operation, it was found out that the prediction of the low gas volume fractions is less accurate [2]. The reason is that, although the thermodynamic models are computationally efficient and reasonably accurate they do not capture fully the characteristics of the fluid transport through the suction and discharge plenums. Therefore more detailed characterization of flow is required and it is believed that 3D CFD modelling can offer accurate prediction of the flow in the suction and discharge plenums of screw machines.

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd

A Number of reviewed research publications showed that CFD can effectively be used for analysis of the flow in the suction port of a screw compressor. Authors reported in [3] the efforts to visualize the flow by high speed camera and compare this with the CFD model results. Similar approach was used previously for predication of flow losses [4]. More examples on use of CFD in analysis of the flow in the suction plenum to determine potential loss areas [5], [6] showed possibilities to improve efficiency by the simple change in the design of the suction plenum.

The work presented in this paper is the part of the research program which aims to determine multiphase flow characteristics at the suction of a screw compressor as described in [7]. Modelling of multiphase flows in screw compressors using 3D CFD is still not part of a conventional design procedure due to the difficulties in obtaining solution and the speed of simulation. Previously, the authors [3] attempted to measure the multiphase flow characteristics (flow velocity) by means of Mie-Scattering technique. This technique proved to be very useful for evaluation of a single phase flow but posed challenges in measuring the multiphase flow due to formation of liquid film on the transparent test sections. Therefore, no quantitative and meaningful information were obtained for validation of CFD results. Local flow velocity measurements inside the screw compressors are difficult to obtain. However other flow properties such as local pressures can be measured with relative ease by using pressure measurement devices. For that reason, before the multiphase flow experiment is performed, it was proposed to carry out the measurements of pressure fluctuations with the single phase flow in order to obtain the pressure distribution in the suction chamber and validate the data obtained with CFD. This will demonstrate whether the local pressure measurement is a suitable method for validating the CFD results and determine suitability of the experimental setup for such measurements.

The screw compressor used for evaluation of pressure losses in the suction port was oil free compressor with the rack generated "N" rotor profiles and the internal two stage gearbox of 7.197:1 gear ratio. The main rotor has 3 lobes, 127.45 mm outer diameter and wrap angle of 285º. The gate rotor has 5 lobes, outer diameter 120.38. The compressors rotors are with length over diameter ratio of 1.6 and 93 mm centre distance. The manufactured interlobe and radial clearances are 180 microns. The compressor has axial suction port and combination of axial and radial discharge port, as shown in **Figure 1** Screw compressor configuration and sub-domains**Figure 1**.

In this paper the results obtained from the local pressure measurements of a single phase flow within the suction port at various locations and operating conditions are compared with the results obtained from the CFD model.

2. Grid generation and pre-processing for CFD calculation

Prerequisite for the successful full 3D CFD simulation of screw compressor is to obtain a numerical grid of rotating and stationary domains which will preserve conservativeness of space and equations and allow fast and reliable calculations. The decomposition in sub-domains and the topology of the numerical mesh used for CFD analysis of this oil free screw compressor research is shown in **Figure 1**.The mesh consists of the moving main and gate rotor sub-domains which are connected to the suction port and the discharge port mesh. The suction port is extended by a circular pipe. The rotor domains were meshed using SCORG^{TM} (Screw COmpressor Rotor Geometry Grid generator) [8], while the stationary parts were meshed using standard ANSYS mesh generators ICEM.

Two sets of meshes were generated for this research; one with the radial and interlobe clearances of 60 microns and the other with interlobe and radial clearances of 40 microns. Axial clearances were not taken in consideration. The reason for providing two sets of the mesh is to determine the most likely size of the leakage gaps in the operation of this machine. The size of mesh was just over 2 million cells. Table 1 shows breakdown of mesh size for all sub-domains.

Table 1 Size of the mesh for individual sub-domains

Component	No of cells in sub-domain
Main Rotor	451008
Gate Rotor	459360
Inlet Pipe	14650
Suction Port	568126
Discharge Port	590925
Total	2084069

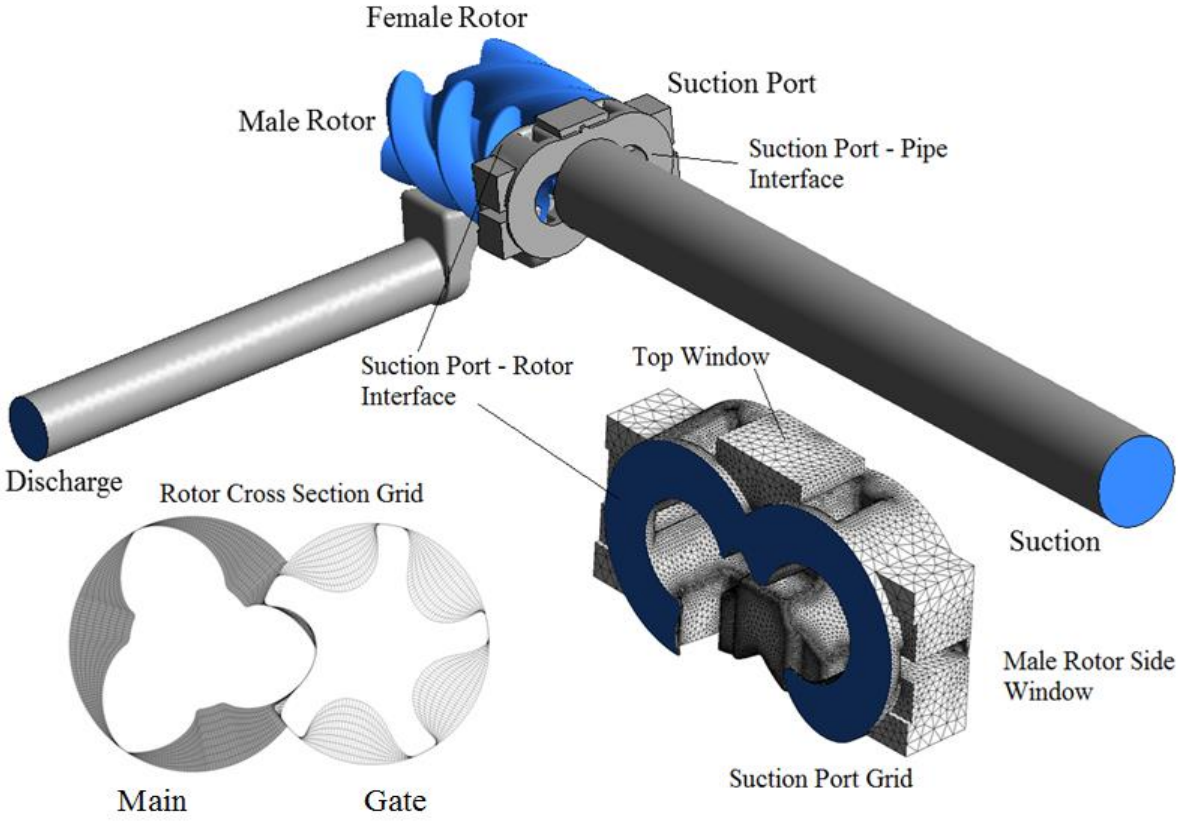

Figure 1 Screw compressor configuration and sub-domains

The CFD calculation was performed using commercial coupled pressure based solver ANSYS CFX. The working fluid is air with at 300 K suction. The suction pressure is 1.0 bar and discharge pressure is 2.0 bar. Convergence criteria were set at 1e-3RMS. The solver was allowed to run until satisfactory cyclic repetitions were achieved across all the monitored points. Selected turbulence model was SST K-Omega proved to be suitable for both main flow domains and in clearances.

3. Experimental Setup

Experimental investigation of the pressure fluctuations and performance of a test screw compressor was carried out in the air compressor test rig at City University London. **Figure 2** shows the layout of the measurement setup, the main test rig components and measurement points. In order to allow visual access into the suction port it was modified and Perspex windows were machined in it. Additionally, a transparent 1 meter long pipe was attached to the port in order to allow visualisation of the flow up stream of the suction plenum. The compressor is driven by a variable speed 75kW motor and has an

internal synchronizing gear box with the gear ratio 7.197:1. The speed of the motor is adjusted with a speed controlled through the variable frequency converter. The torque meter is installed on the motor shaft while the digital encoder for the speed measurement is mounted on the main rotor shaft. The pressure and temperature of the gas are measured at the inlet, the discharge and upstream of the orifice plate. In addition, three pressure transducers are used for recording the interlobe pressures and are located in the working chamber through the compressor casing on the main rotor side. The air flow through the compressor is measured by use of an orifice plate installed in the discharge pipe system. The discharge pipe system contains the control valve for regulation of the discharge pressure.

Measured data from transducers are collected by a data acquisition system. It consists of a National Instrument Compact-RIO (CRIO-9022) Real-Time with an 8 slots chassis CRIO-9114. It has an industrial 533 MHz real-time processor for reliable real-time applications. Compact _RIO has 256 MB of DDR2 RAM and 2 GB of a non-volatile storage for holding programs and logging data. The unit also permits the acquisition of signals from the measurement transducers simultaneously during the experiment that are inputs to the unit itself and connected to a computer via an Ethernet cable.

The programming of the data acquisition system was by LabVIEW FPGA (Field Programmable Gate Array) which is suitable for high frequency data acquisition. The measurements obtained from the transducers are acquired on the cRIO (NI Compact RIO) and send to the PC where the latter instantaneously calculate values of the compressor power consumption, air flow rate, specific quantities and efficiency. All required information is displayed on a computer monitor in real time.

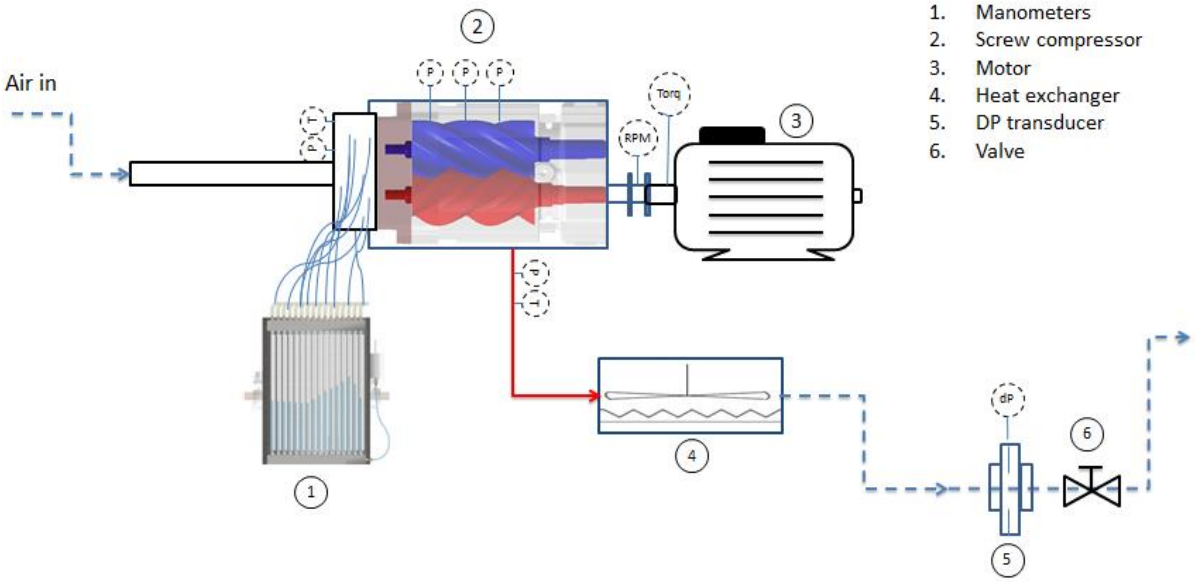

Figure 2 Experimental layout

Furthermore, the array of inclined manometers was used to map the pressure losses in the suction port. Pressures were measured in 23 places conveniently positioned within the suction plenum. Figure 3 shows the image of experimental setup while **Figure 4** shows location of the pressure probes. Measured pressures are averaged at each location over a number of revolutions.

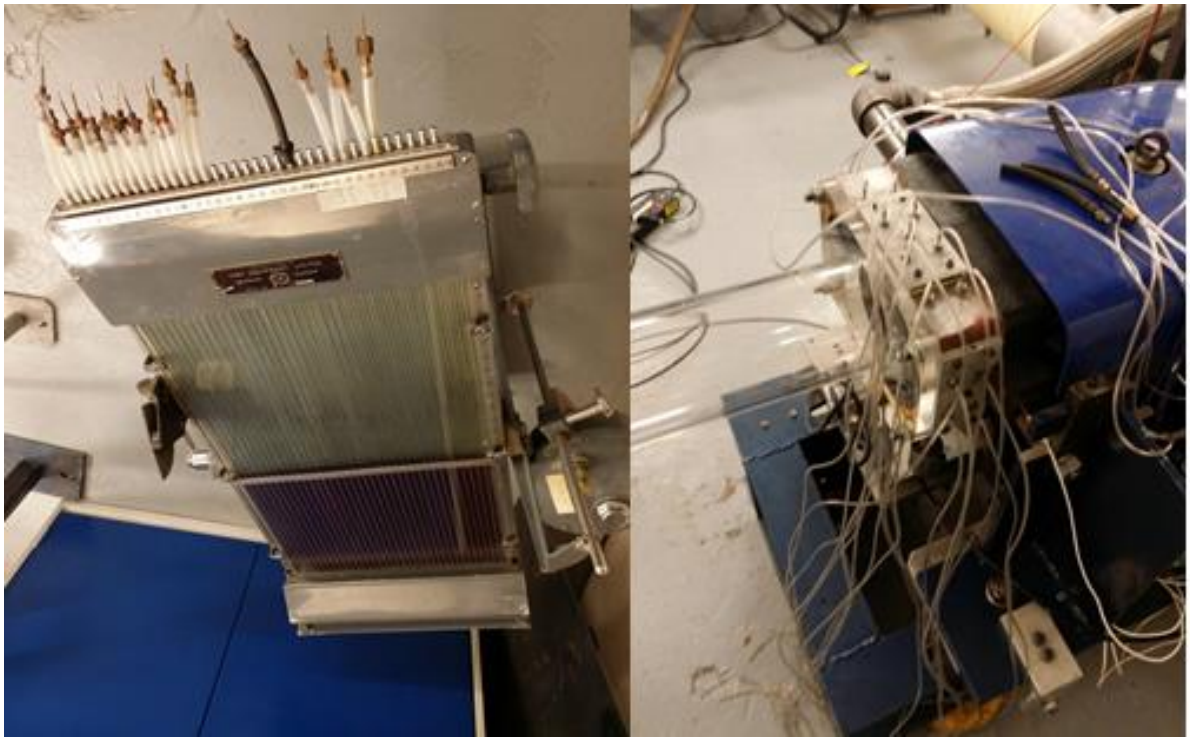

Figure 3 Suction pressure measurement setup

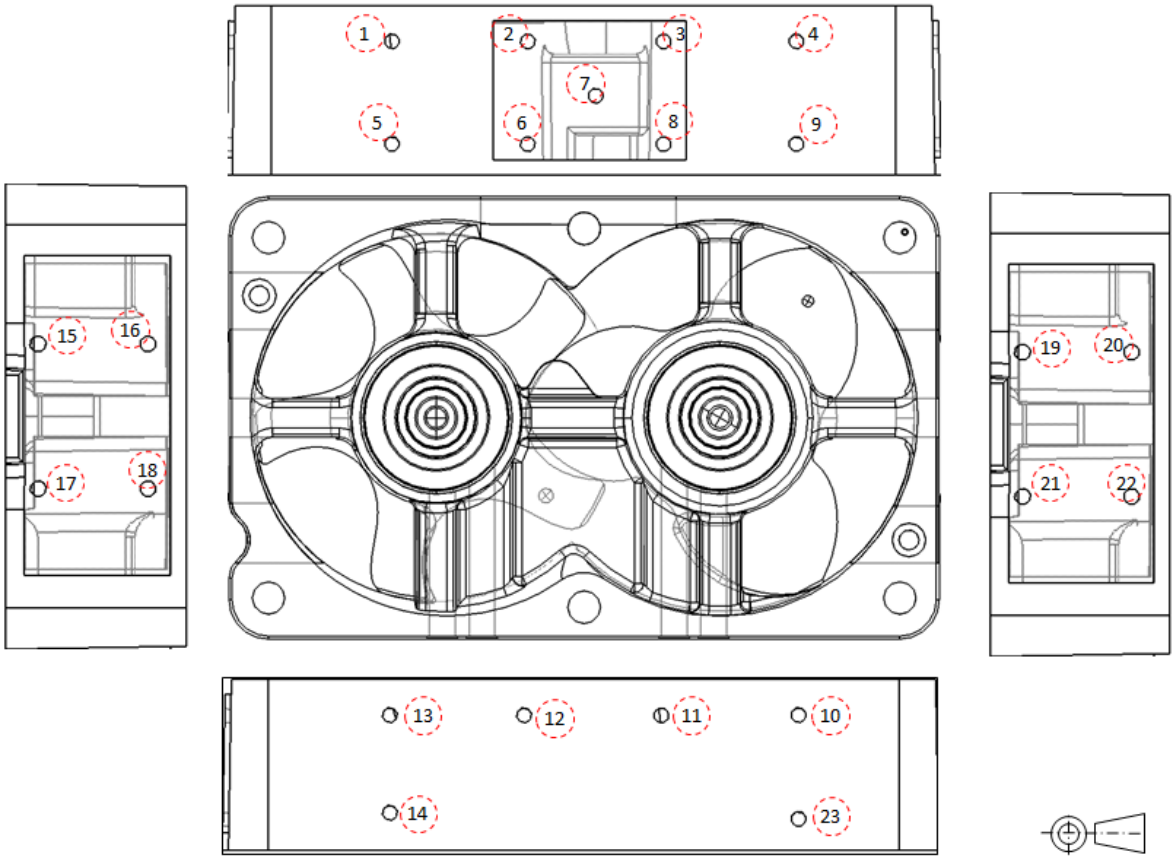

Figure 4 Location of pressure measurements in the suction plenum

4. Results and Discussion

The measured results are compared against 3D CFD simulations for the compressor main rotor speeds of 4000, 6000 and 8000 RPM. The CFD solver was allowed to run until the satisfactory repetitions of all the monitoring parameters were achieved in two consecutive rotor rotations. The CFD results presented in this paper are the averaged over twofull revolutions of the main rotor.

4.1. Internal Pressure and Compressor Performance

The comparisom of the compressor power and mass flow rate between the predictions and measurements is shown in Figure 5 for different compressor speeds. Two CFD results are shown in the diagram, one with 40µm and another with 60µm clearances. In case of 60µm clearance, the predicted mass flow is lower than the measured mass flow. However, for clearances of 40µm the predicted mass flow appeared to be almost identical to the emeasured flow rate. The predicted power is almost identical for two different clearance settings but it differs from the measured power.

The power measured in the test rig includes mechanical losses of the internal gearbox, bearings and seals while the power extracted from CFD calculations is the indicated power integrated from the pressure volume diagram which does not include any mechanical losses. The difference between the two increases with the increase in rotational speed as it would be expected.

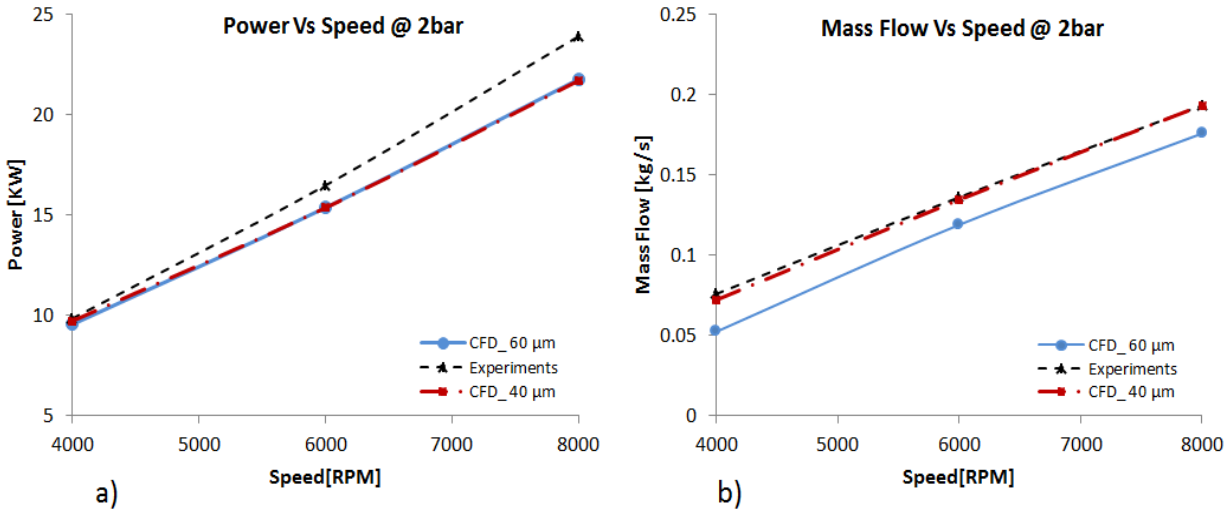

Figure 5 CFD and experiments comparison a) Power b) Mass flow rate

Pressure variations in the compression chamber as function of the main rotor rotational angle are shown in **Figure 6**Error! Reference source not found.. Measured pressure diagram matches well with pressure diagram extracted from CFD results at all speeds. The Distinctive difference between the measured and estimated internal pressure is observed in the peak pressure at low compressor speed of 4000 RPM. This could be due to the fact that at 4000 RPM the flow losses cause local temperatures to increase and the clearance changes affects mass flow rate which is marginally lower than measured mass flow rate at this speed. The peak pressure inside the working chamber increases as speed of the compressor increases. The peak pressures in both predicted and measured data are greater than the discharge pressure which indicates over compression.

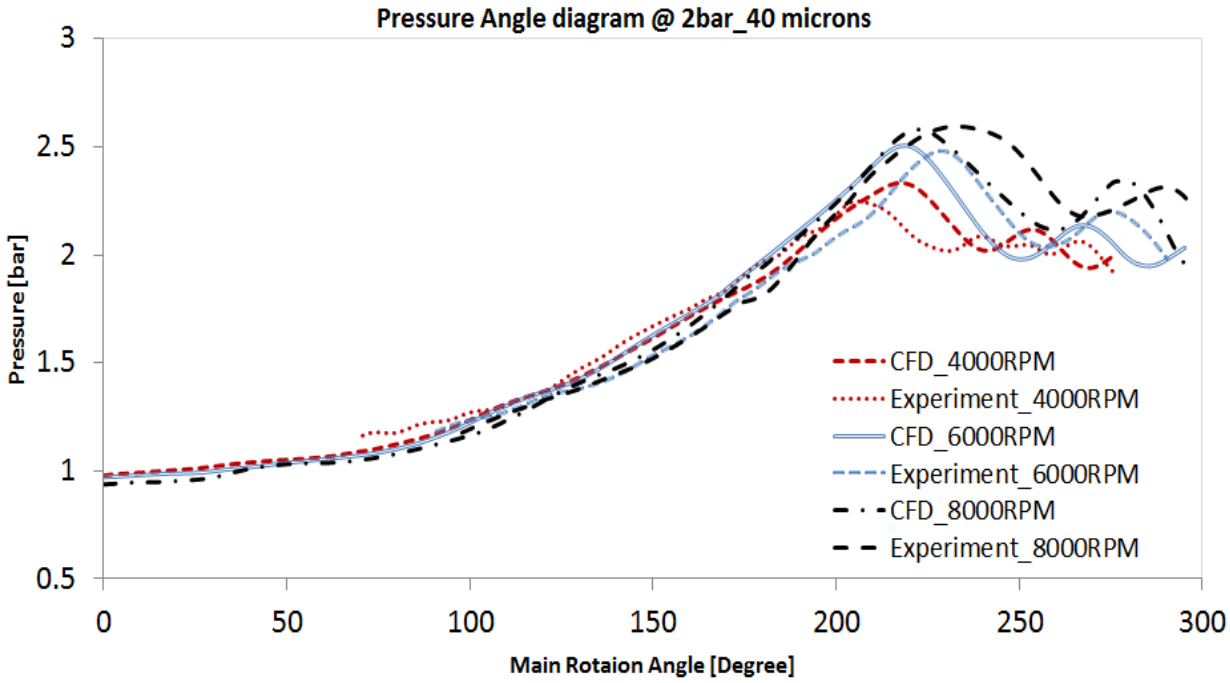

Figure 6 Pressure angle diagram with radial and interlobe clearances of 40 microns

4.2. Suction Pressure Map

Figure 7 shows the comparison of measured and calculated pressures at 23 locations in the suction plenum. The results are shown for three operating speeds, 4000, 6000 and 8000 rpm and two sets of clearances assumed in the numerical mesh of the compressor. The pressure loss at individual locations is the consequence of the change in local velocities. It can be seen from **Figure 7**a to **Figure 7**c that pattern of this pressure loss remains same at all operating speeds while only the amplitude of pressure losses changes. The lower the suction pressure as shown in diagrams, the higher the pressure loss. The highest pressure losses are observed in the bottom and top right corners in locations 4 and 23. This indicates that the higher amount of mass flows through these regions. This also indicates the position of the highest velocities within the suction plenum. Pressure loss in position 23 is higher than position 4 indicating that more mass flows towards the male rotor than the female rotor. Pressure loss at the manometer locations 12 and 11 is relatively low clearly reflecting the fact that the flow path is obstructed in this place.

It was previously reported in [9] how the mass flow depends of set clearances in CFD calculations. Since conjugate heat transfer and solid deformations are not accounted for in this CFD calculation, it is necessary to adjust clearances in the numerical mesh prior to the CFD calculation to account for thermal expansion of the compressor elements. The results presented in **Figure 4** show the effect of reducing clearances. The case with 40µm clearance gap predicts mass flows very close to the measured values as shown in Figure 5. Consequently the prediction of pressure losses in the suction plenum appears to be agreeing well with the measured values if the clearances are set to 40µm, as shown in **Figure 7**.

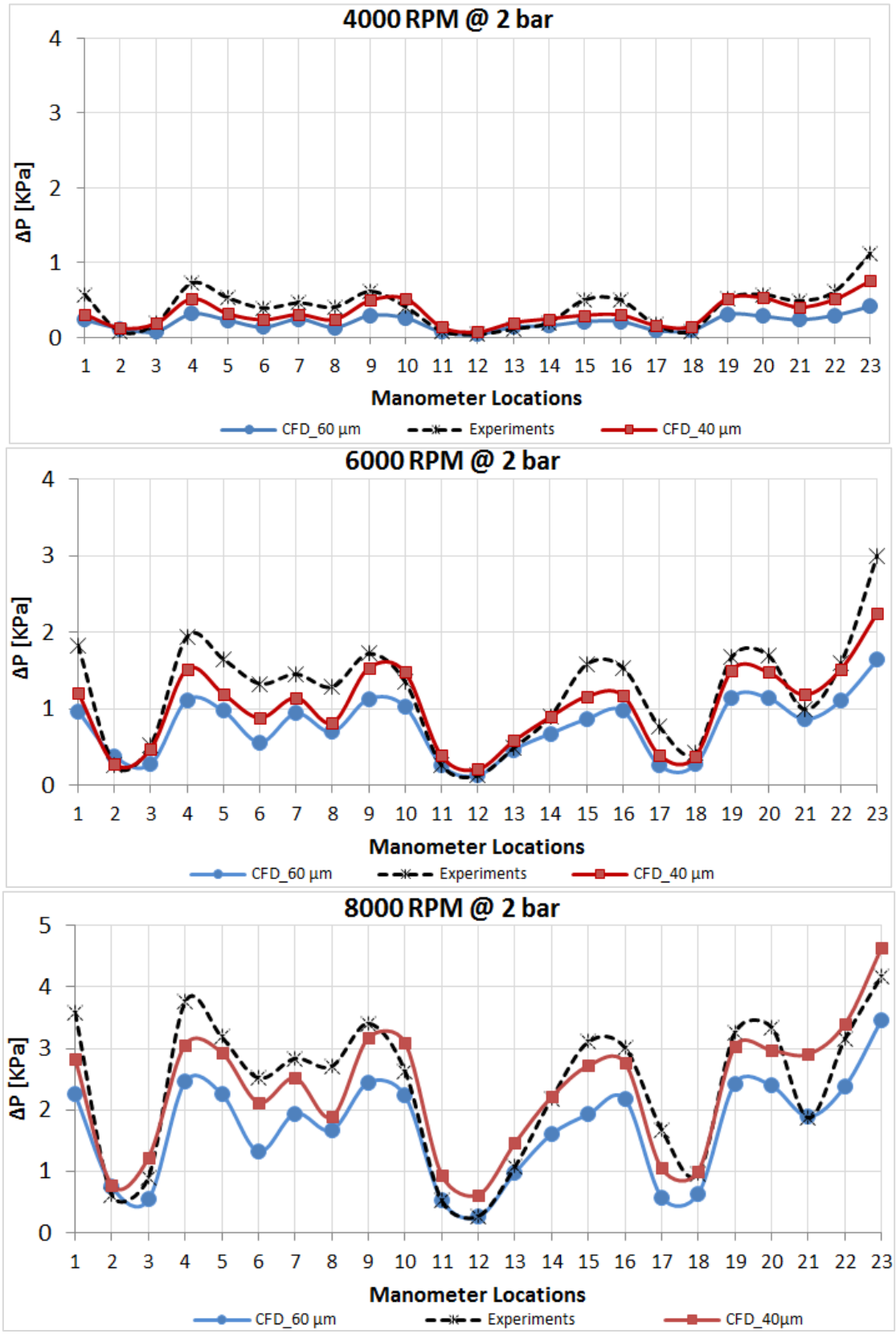

Figure 7 Pressure loss in the suction port at various locations

5. Conclusion

Measurements of pressure losses in the suction domain of an oil free screw compressor were carried out for the single phase flow of air by use of 23 inclined tube manometers. Alongside the power and flow measurements, the pressure variations within the compression chamber were recorded by use of three dynamic pressure transducers embedded in the casing on the main rotor side. The results are used to evaluate predictions of pressure losses in the suction plenum of the compressors derived from 3D CFD calculations. The following can be concluded:

- The predicted pressure fluctuations in the suction plenum have the same pattern as the experiment. At location 11 and 12 pressure losses are the lowest due to the obstruction in flow path. The highest pressure loss is measured at location 23 indicating the region with highest velocity in the bottom side of the main rotor.
- The measured and predicted chamber pressure matched well and the noticeable difference in the maximum pressure reduced substantially once the clearances of 40 μm were selected for simulations. This was due to better prediction of mass flow rate.
- The predicted values of suction pressure losses changed substantially with change in radial and interlobe clearances. The better prediction of losses coincided with more accurate prediction of mass flow rate at set clearance of 40 μm.
- It was found that CFD simulation the flow at the suction of a screw compressor using numerical grid generated by SCORG is able to predict the local flow features with high accuracy.

This study shows that 3D CFD can be used for study of the local flow features in the suction of screw compressors and can be used to optimise the shape of the suction plenum and port which is not usually considered in the current design practice. Furthermore it provides a good platform for studying effect of multiphase flows on flow characteristics at the inlet port of twin screw compressors. In the future the authors plan to continue experiments with multiphase flow at the suction of screw compressors,

6. References

[1] N. Stosic, A. Kovacevic, K. Hanjalic, and L. Milutinovic, “Mathematical Modelling of the Oil, Influence Upon the Working Cycle of Screw Compressors,” 1988.
[2] F. Cao, T. Gao, S. Li, Z. Xing, and P. Shu, “Experimental analysis of pressure distribution in a twin screw compressor for multiphase duties,” Exp. Therm. Fluid Sci., vol. 35, no. 1, pp. 219–225, Jan. 2011.
[3] A. Kovacevic, M. Arjeneh, S Rane, N Stosic, M. Gavaises, “Flow Visualisaton at Suction of a Twin Screw Compressor,” in International Screw Compressor Conference at Dortmund, 2014.
[4] N. Stosic, A. Kovacevic, I. K. Smith “The CFD Analysis of a Screw Compressor Suction Flow,” Int. Compress. Eng. Conf., p. Paper 1479, 2000.
[5] J Sauls, S Branch, Trane, “CFD analysis of refrigeration screw compressors,” Int. Conf. Compress. their Syst., 2009.
[6] M Pascu, M Heiyanthuduwage, S Mounoury and U. Howden Compressors Ltd, “Influence of the suction arrangement and geometry of the inlet port on the performance of twin screw compressors.pdf,” 8th Int. Conf. Compress. their Syst., 2013.
[7] M. Arjeneh, A. Kovacevic, M. Gavaises, and S. Rane, “Study of Multiphase Flow at the Suction of Screw Compressor,” in 22nd International Compressor Engineering Conference at Purdue, 2014.
[8] A. Kovacevic, N. Stosic and I.K. Smith. Screw compressors - Three dimensional

computational fluid dynamics and solid fluid interaction, ISBN 3-540-36302-5, Springer-Verlag Berlin Heidelberg New York, 2007

[9] A. Kovacevic, S. Rane, N. Stosic, Y. Jiang, M. Furmanczyk, and S. Lowry, "Influence of approaches in CFD Solvers on Performance Prediction in Screw Compressors," in 22nd International Compressor Engineering Conference at Purdue, 2014, pp. 1–10.