Flow visualization at suction of a twin screw compressor

A. Kovacevic, M. Arjeneh, S. Rane, N. Stosic, M. Gavaises,

City University London

Abstract

Rotary twin screw machines are commonly used for handling single and multiphase fluids in industry as compressors, expanders and pumps. Liquid concentration in the admitted gas can influence the performance of the machine significantly. The flow characteristics of the liquid phase depend on its concentration and may also affect the ability of the machine to handle it without getting damaged. Experimental techniques enable single and multiphase flows to be observed and measured but are difficult to control. It is therefore useful to develop models to predict the flow and performance of multiphase screw machines.

The £D Computational Fluid Dynamics (CFD) is used in this paper to investigate the effect of water injection into the air stream at the inlet of an oil free screw compressor. The CFD results were compared with measurements of the flow pattern obtained from a high speed camera. The agreement between them is sufficient to justify the use of CFD for further investigation of multiphase flow in the suction of the compressor.

Introduction

Multiphase fluid flows are challenging to model. Most investigations of multiphase flow regimes are limited to flow in vertical or horizontal tubes of constant cross section [1]–[5]. Only few reports are identified in an open literature which attempt to evaluate flows in passages with complex geometries. However, many industrial processes involve mixing of different fluids or phase change of a single fluid. Many of these processes could be enhanced if the understanding and modelling of such flows were improved. Publications on CFD modelling of multiphase flows inside oil flooded screw compressors are rare. In one of these [6], a concentration model was used to predict the distribution of oil injected into the working chamber. However, no literature sources are found on modelling multiphase flows in the suction of screw compressors yet its variation can affect the compressor performance significantly.

The multiphase suction flow to a screw compressors has been studied numerically by means of 3-D Computational Fluid Dynamics (CFD) and the flow patterns, thus obtained, were compared with images obtained using a high speed camera. This was carried out in four stages as follows:

a) An oil free compressor with “N” profile rotors was selected. This had a male rotor diameter of 128 mm, a 3/5 lobe configuration and a length to diameter ratio (L/D) of 1.6. The distance between the centre line axes was 93 mm and the wrap angle was 285 deg. Assuming nominal clearances of 160 micrometres, the compressor geometry was defined by a combination of structured and unstructured numerical meshes for use in a CFD solver. The mesh was generated by use of SCORG© which allows accurate representation of screw compressor domains by use of boundary fitted rotating, sliding and stretching numerical meshes [7]. Good quality of hexahedral meshes was achieved by the use of the recently developed method for smoothing of sliding interfaces and for local mesh refinement.
b) Using this numerical grid, a 3-D numerical solution for the compression of dry atmospheric air was obtained using commercial CFD package Ansys CFX. This was compared with measurements obtained in the test rig equipped with pressure probes in the compressor. The flow visualisation predicted for the inlet pipe and suction domain was compared with photographs obtained with a high speed camera by use of the Mie-Scattering method. This required a high speed camera to capture images of the flow, illuminated via continuous source of light using a smoke machine.
c) A 3-D CFD analysis was then performed for the multiphase flow of air and water in the suction pipe and port using a stationary numerical grid for these compressor parts and applying transient boundary conditions, obtained from the full domain single phase calculations.
d) Finally, the flow in the pipe and suction domain was visualised by the Mie-Scattering method. In this case, water was injected at various mass rates into the fast moving stream of air flowing through a perplex pipe attached to the suction port of the compressor. The transparent suction port and pipe were designed to enable full visual access to the area of interest.

Numerical simulation of the single phase flow using of full compressor model

The oil free twin screw compressor used for this research was operated at the speed of 8000 rpm to compress air from atmospheric conditions to 2 bar discharge pressure. In this setup the single phase dry air was used as the working fluid. Fig. 1 shows the CAD model of the compressor with locations for transparent Perspex windows on the top and sides. It also shows the extended suction pipe.

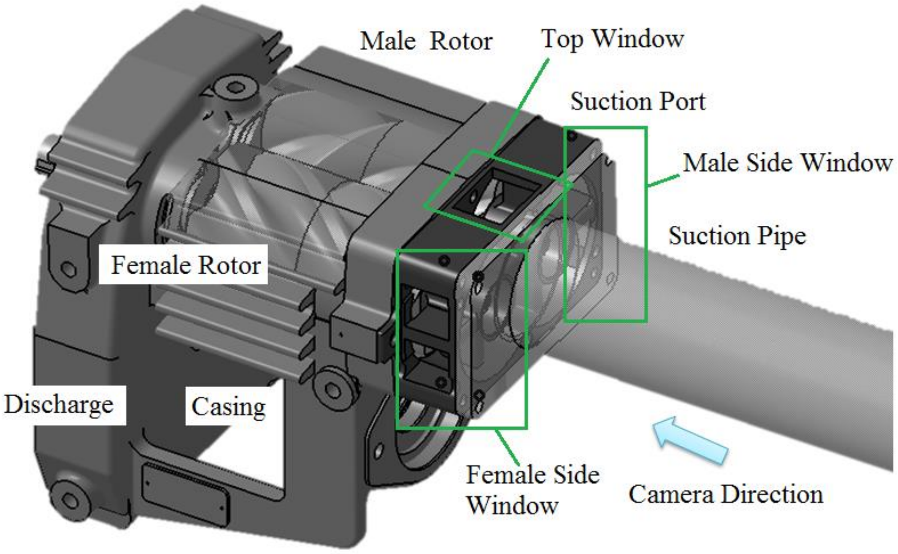

Fig. 1 Screw compressor with Suction port windows and extended suction pipe

The complete numerical mesh for the CFD analysis of the full model is shown in Fig. 2. In the full model, all elements of the compressor are simultaneously calculated by use of commercial CFD solver, ANSYS CFX.

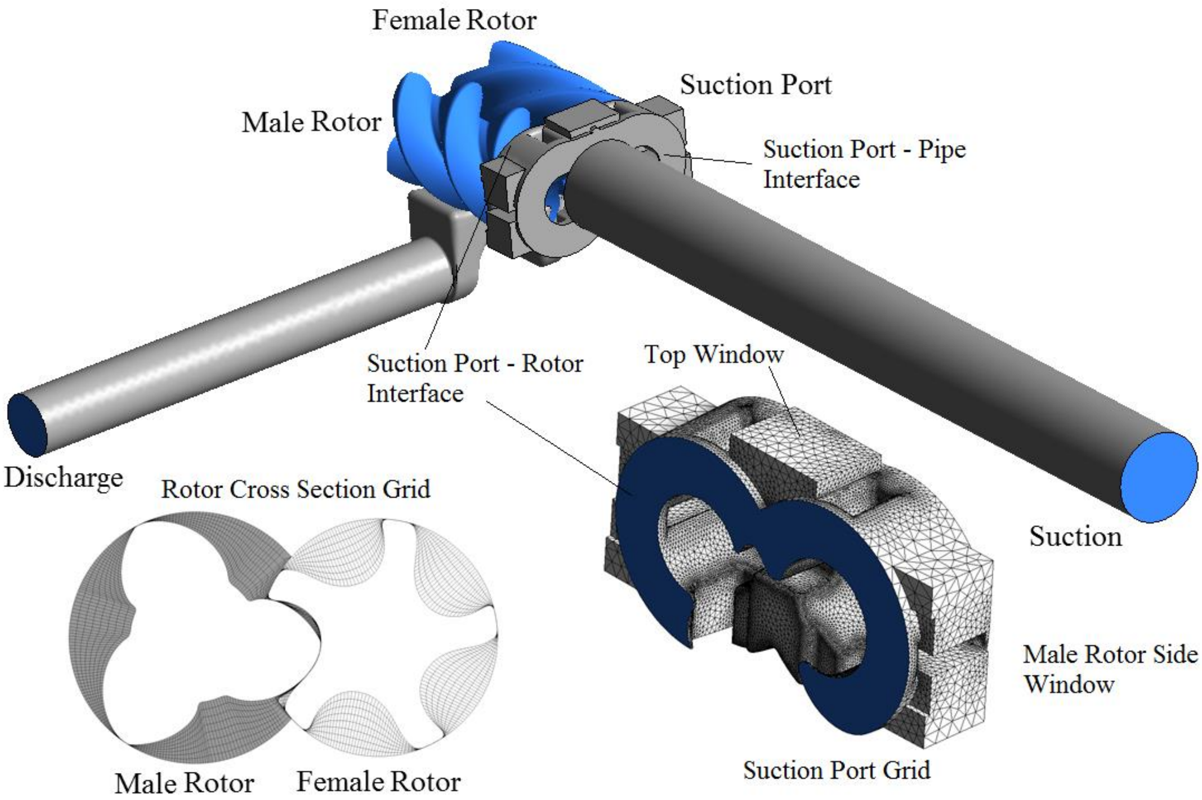

Fig. 2 Full 3D model for analysis of the single phase flow in the test screw compressor

This setup is used to calculate both single phase flow of the dry air and the multiphase flow of the mixture of air and dispersed water. The full solution with the single phase flow is obtained and results are used for comparison with the test results which are described later in the text. However, due to sliding and stretching interfaces between the rotor domains [7] the multiphase case had difficulty to converge and produced nonphysical errors.

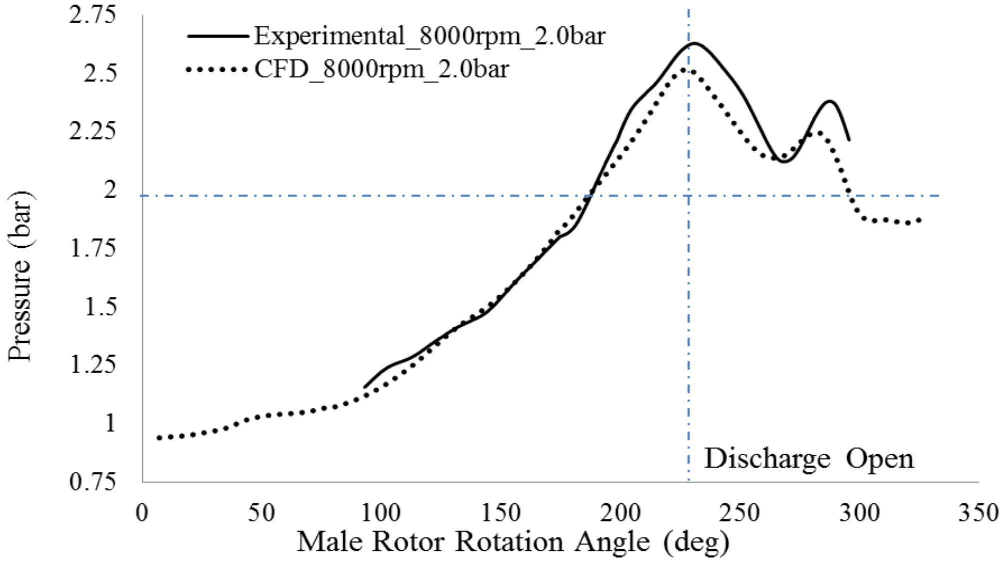

Fig. 3 Pressure-angle diagram obtained by CFD and Measurements

The comparison of pressure diagrams calculated by CFD and measured by pressure probes inside the compression chamber is presented in is presented in Fig. 3. The experimental pressure curve presented here was obtained by three pressure transducers embedded into the casing of the compressor. The results indicate that at the 2 bar discharge pressure, the compressor is operating in over compression mode.

Experimental setup

Experimental investigation of the compressor performance was carried out using the air compressor test rig at City University London. The layout of the rig with the main components and measurement points is shown in Fig. 4. 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. The compressor was 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 using a variable frequency drive. The torque meter is installed on the motor shaft while the digital encoder for the speed measurement is mounted on the male 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 were used for recording the interlobe pressures and were located in the working chamber through the compressor casing on the male rotor side. The air flow through the compressor was measured by use of an orifice plate installed in the discharge line of the system. The discharge line contains a control valve for regulation of the discharge pressure. The data acquisition was carried out using CompactRIO from National Instruments and Labview. The measurements were taken for discharge pressures up to 2.5 bar and main rotor speeds at 8000rpm.

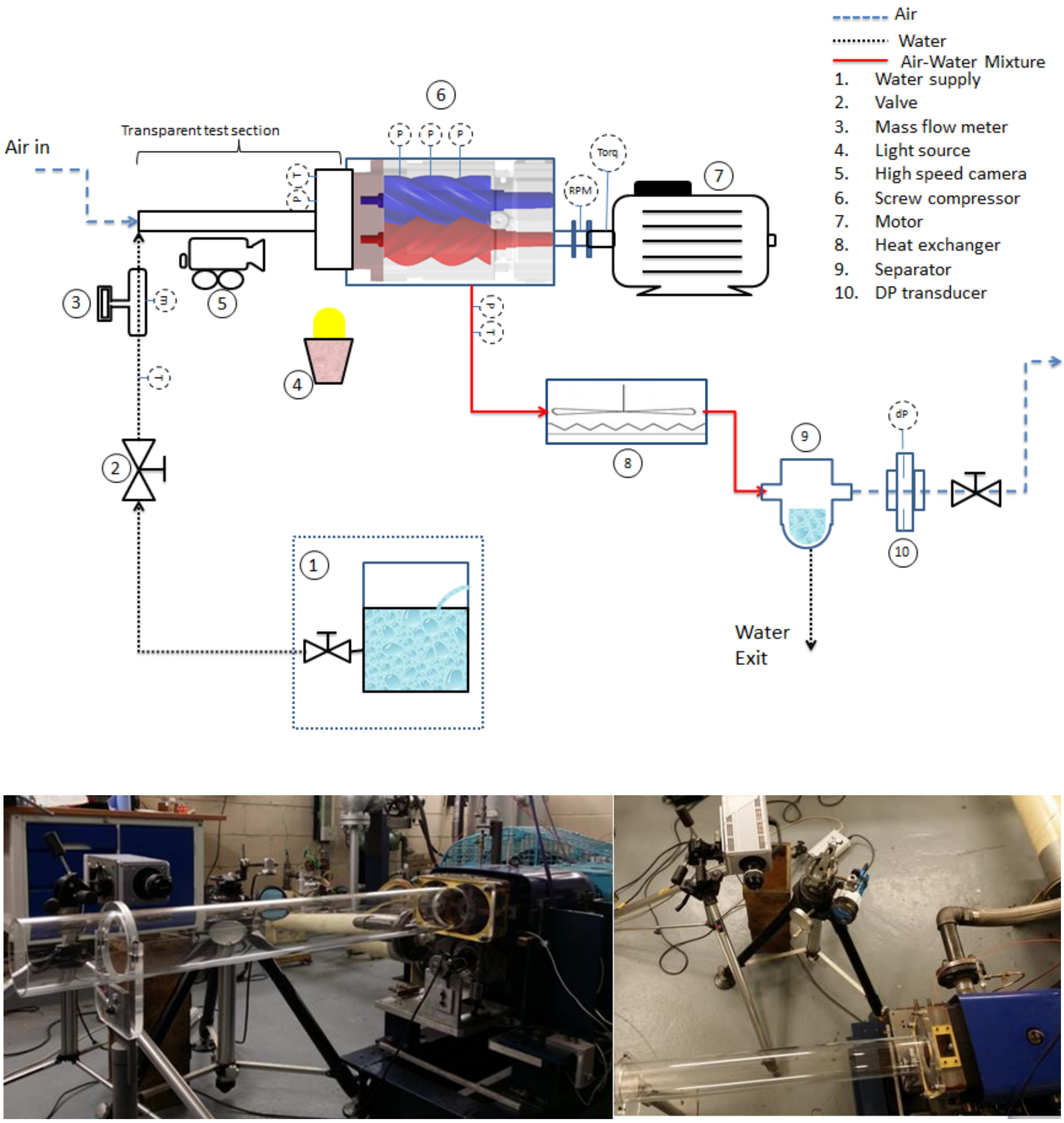

Fig. 4 Experimental setup for testing and visualisation of suction flow

Visualisation of the single phase flow by high speed camera

The comparison of high speed imaging at 10000 fps and 8000 rpm and CFD results at the suction port is shown in Fig. 5 for the gate rotor side in Fig. 6 for the main rotor side window. The discharge pressure was 2.0 bar. High speed visualisation of smoke shows formation of swirl at suction port which is comparable to the CFD results.

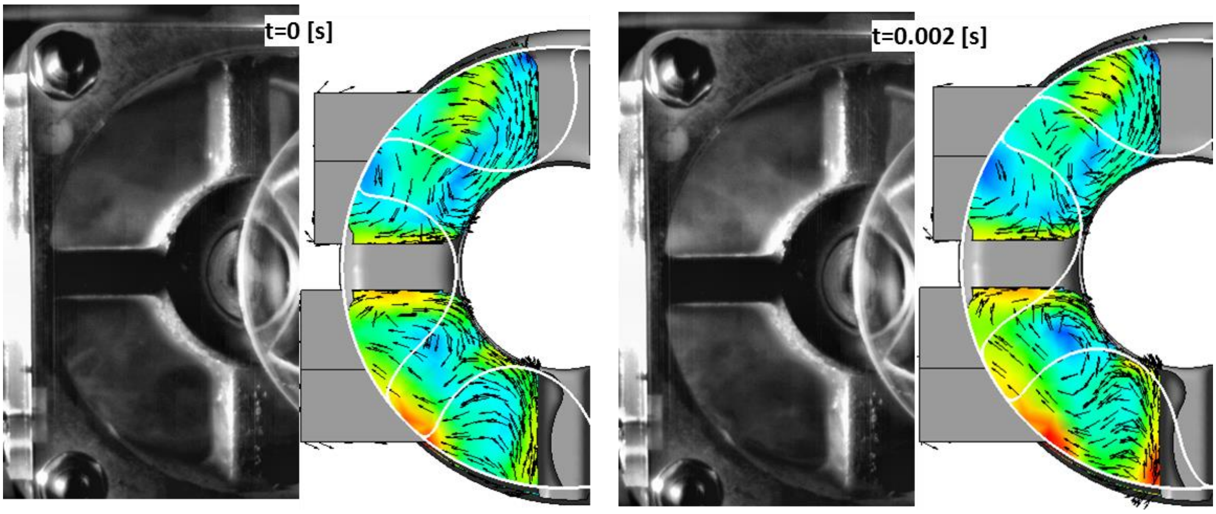

Fig. 5 High speed camera and CFD comparison for the gate rotor

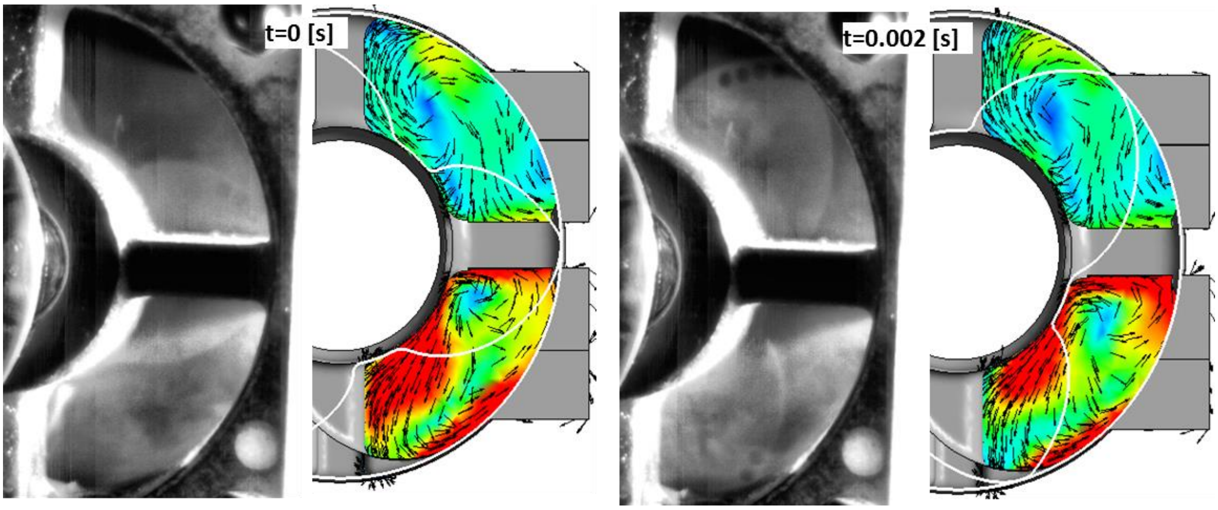

Fig. 6 High speed camera and CFD comparison for the male rotor

Both, CFD results and visualisation also show that the flow rate through the bottom opening on the male rotor side is higher than through the top opening on the male rotor side

Numerical analysis of the multiphase flow in the suction pipe and port

Due to difficulties to obtain the solution for multiphase flow in the full 3D model of the compressor, the analysis of multiphase water-air mixture was performed on the reduced model which consists of the suction port and the suction pipe only. Fig. 7 shows the numerical grid of the compressor suction pipe and port used for CFD analysis of multiphase flow. The reduced model requires additional boundary conditions on the interface between the stationary inlet port and rotors. The influence of rotors on the suction port is maintained by transient boundary conditions derived from the solution of velocity field in the single phase flow calculation with the full model. In addition, temperature and turbulence parameters were also applied on the suction-rotor interface boundary.

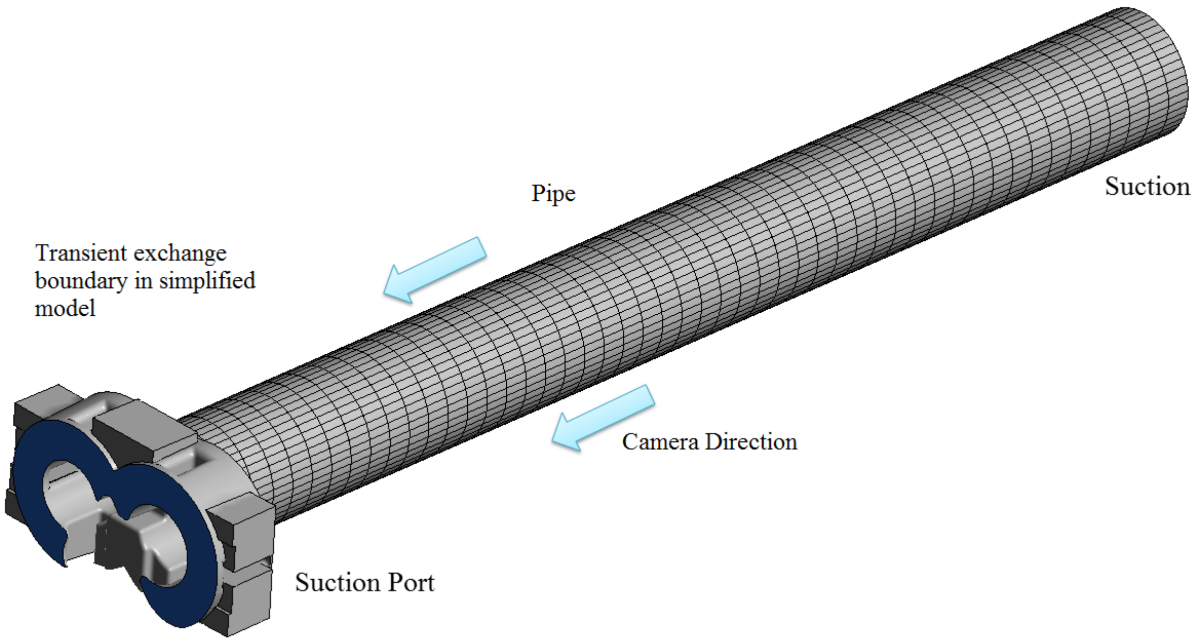

Fig. 7 Reduced Multiphase CFD model with the transient boundary interfacing with rotors

Comparison of the velocity profile between the full model and the boundary condition of the suction port model at an instant is shown in Fig. 8.

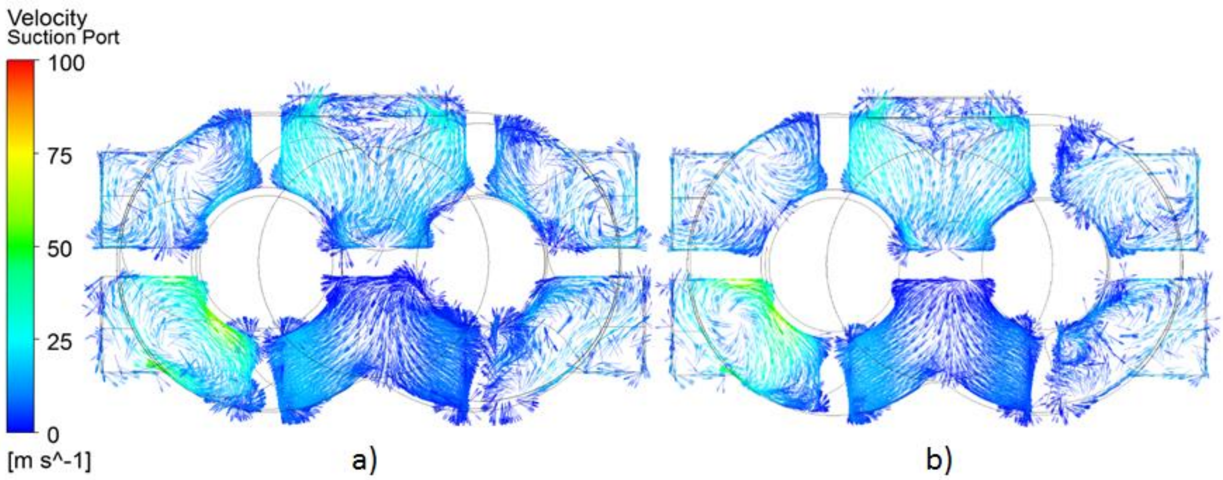

Fig. 8 Velocity vector plot at the suction a) Full case b) Suction port model

The left plot identified with a) shows the velocity field in the suction port obtained by the full model while the right hand side identified by b) is velocity field which is applied on the boundary in the reduced model of the suction port and pipe. A single phase calculation was performed in the reduced model in order to verify the mass flow rate and pressure drop. The results showed very good match of the two models used. It was therefore concluded that the reduced model could be used for accurate calculation of multiphase flows in the suction port and pipe.

CFD results for the multiphase flow setup

The multiphase flow is the mixture of water and air obtained by controlled injection of water into the air stream at the suction of the compressor. Therefore in calculation of multiphase flow, air was set as the background phase and water was set as the dispersed phase with a constant droplet diameter. The size of the droplet is the parameter in the calculation of drag and volume fraction of the secondary phase [7]. For this calculation droplet size was set to 30μm. Results from the calculation are presented in Fig. 9 and Fig. 10.

Fig. 9 shows the volume fraction of water inside the suction port and the suction pipe obtained by 3D CFD modelling with the reduced model. The volume fraction of water is about 0.001 which means 99.9% of volume is occupied by air. Although this amount of water by volume is insignificant, it translates to just above 40% of the mass flow due to high density of water compared to air. It was evident from the CFD simulation that there is no slip between the two phases and the flow can be assumed homogenous. This could be due to a small droplet diameter set for this model. It was also observed that the flow is fully developed by the time it reaches the midsection of the pipe. Marginal changes in water volume fraction beyond this point indicate a fully developed flow which could be achieved with a shorter pipe.

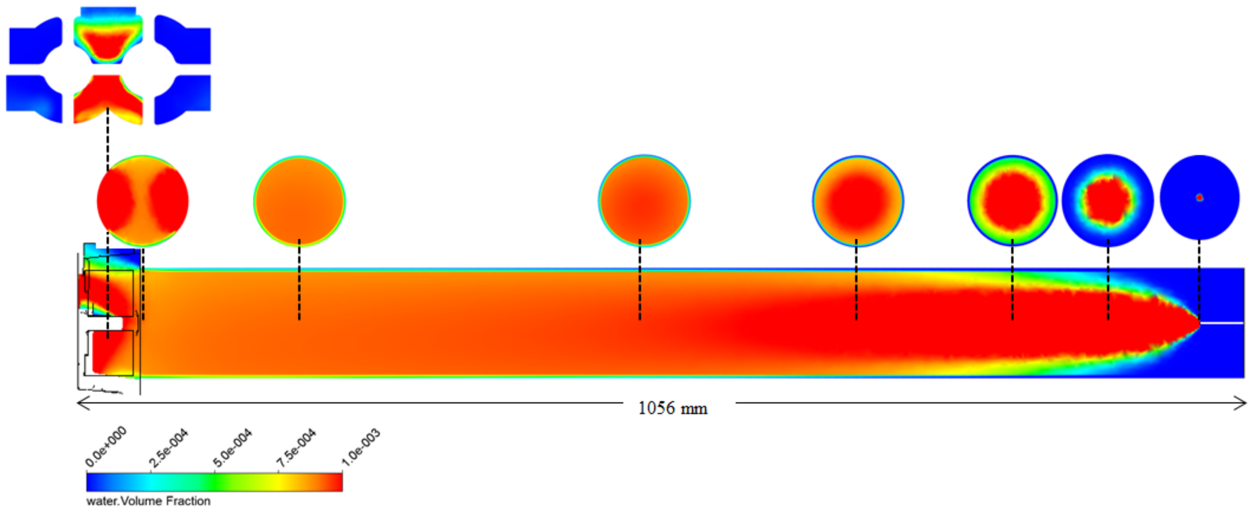

Fig. 9 Volume fraction of water across the pipe and suction port

Water accumulation on the suction port is shown in Fig. 10. Water accumulates at the bottom of the suction port. The water volume fraction in this location is above 50 %.

The majority of remaining water droplets flow through the top middle passage and there are no traces of water phase in the top right and top left windows. A small amount of water enters the compressor male rotor through the bottom right window of the main rotor due to the higher velocity. Some of the water entering the bottom left window is the water accumulated at bottom of the suction port.

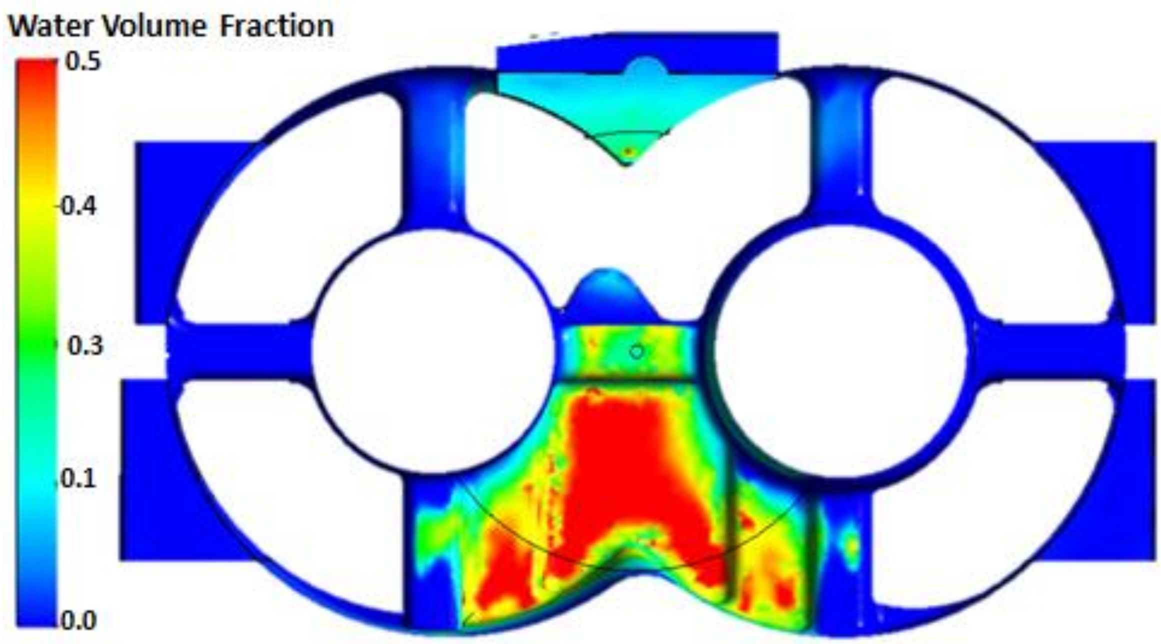

Fig. 10: Water depositions on the surface of the suction port

Conclusions

Visualization of flow by high speed camera and smoke generator was carried out for a single flow in the inlet pipe and suction domains of the oil free screw compressor. The results were compared with predictions derived from a 3-D CFD model of flow. Following conclusions can be made:

- The visualization with smoke in the suction pipe at low rotor speed shows pulsations in the flow. At higher speed the smoke was diluted and the pulsation could not be captured. CFD results in the suction pipe showed small pressure waves that correspond to these flow pulsations.
- In the suction port on both, the male and female side windows, smoke flow demonstrated a swirling flow pattern with non-uniform velocities. This pattern was well captured by the CFD model.

The multiphase flow simulation was performed on the reduced model and following conclusions are made:

- CFD model of multiphase flow of the fine spray showed a fast diffusion of the spray in the suction pipe followed by deposition on the bottom of the suction port. An unequal division of flow in different sections of the port was identified.
- Although the high speed visualization with smoke provided qualitative data for comparison with the CFD model, in future it will be useful to carry out quantitative experimental measurements like PIV and LDV to provide data for CFD model validations.
- Visualisation of water injection is still under progress and it is expected that it will confirm CFD results.

References

[1] C. Garnier, M. Lance, and J. L. Marié, "Measurement of local flow characteristics in buoyancy-driven bubbly flow at high void fraction," Exp. Therm. fluid Sci., vol. 26, no. 6, pp. 811–815, 2002.

[2] A. Jaworek, A. Krupa, and M. Trela, "Capacitance sensor for void fraction measurement in water/steam flows," Flow Meas. Instrum., vol. 15, no. 5, pp. 317–324, 2004.

[3] A. Shukla and A. Prakash, "Ultrasonic technique to determine particle size and concentration in slurry systems, " Chem. Eng. Sci., vol. 61, no. 8, pp. 2468–2475, 200

[4] P. Stahl and P. R. von Rohr, "On the accuracy of void fraction measurements by single-beam gamma-densitometry for gas–liquid two-phase flows in pipes," Exp. Therm. fluid Sci., vol. 28, no. 6, pp. 533–544, 2004.

[5] P. F. Vassallo and R. Kumar, "Liquid and gas velocity measurements using LDV in air–water duct flow," Exp. Therm. fluid Sci., vol. 19, no. 2, pp. 85–92, 1999.

[6] A. Kovacevic, N. Stosic, and I. K. Smith, "CFD analysis of screw compressor performance," Adv. CFD Fluid Mach. Des. Prof. Eng. Publ. ImechE, London, 2002.

[7] 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