Conjugate Heat Transfer Simulation and Cooling Optimization of the Flow inside a Screw Compressor

J F Willie
Compressors and Vacuum Pumps Systems (CVS) Engineering GmbH, Großmattstraße 14, D-79618 Rheinfelden, Germany
E-mail: james.willie@cvs-eng.de

Abstract. Computational Fluid Dynamics (CFD) as a predictive tool is playing an increasing role in the design and optimization process at Compressors and Vacuum Pumps Systems (CVS) Engineering GmbH. Design optimization using Design of Experiment (DoE) and continuous improvement of our products has led to an increase in the use of CFD and other predictive tools. This paper investigates the use of CFD combined with Conjugate Heat Transfer (CHT) to optimize the flow and the cooling inside a rotary screw compressor. Due to the small clearances in these machines, thermal expansion of the rotors can lead to contact between the rotors and between the rotors and the housing. Determining the exact hot gap size will help reduce leakages, increase the volumetric efficiency, and help prevent contact due to thermal expansion. Using CHT to determine the thermal loads on the rotors and CFD to determine the pressure load on the rotors it is possible to find the correct hot gap sizes inside these machines using Fluid-Structure Interaction (FSI). Other critical phenomena like over and under-compression can be prevented with the help of CFD when running off design. Having localized over-compression, for example, can lead to sudden drop in pressures when the discharge port opens and this can induce shock waves that can lead to noise and pulsation. It can also lead to an increase in the power consumption of the compressor and hence to an increase in the temperature. Optimizing the suction and the discharge can lead to an increase in the volumetric efficiency and improve the cooling of the compressor. This paper presents an example of how CFD and CHT can be used in the development and optimization of screw compressors. Similar work has been reported in [1, 2, 3, 4, 5, 6]. In all but one of them, the flow considered was without heat transfer and as a result the wall temperatures were imposed as boundary conditions. In the case that considered heat transfer validation was not done with measurement. In this paper, it is shown that with the use of CFD and CHT it is possible to predict the cold and hot flows that includes heat transfer to account for the losses in the bearings, gears and sealing inside a screw compressor.
The paper begins with the description of the measurement setup. This is followed by the case setup including the description of the geometry for both the cold and hot cases. This is proceeded by the results and discussion and finally, the conclusions and future work are given.

2. Measurement setup

A picture showing the setup of the measurement test stand is depicted in figure 1. The following are measured: Volume flow rate, inlet/discharge temperature and pressure, compressor power, pressure drop ($\Delta p$) and the temperatures of the gear casing, compressor housing and the bearing carrier as well as the bearings temperatures of the main and gate rotors.

![Figure 1. Picture of measurement setup in the laboratory](image)

In real life application, the compressor is mounted on a truck and driven via a Power Take-Off (PTO) drive shown in the schematic in figure 2(a). To mimic the setup on the truck, a prop-shaft with an angle similar to that on the truck is used to connect the electric motor to the compressor as shown in figure 1. Torque is measured using the setup in figure 2(b). It is done by mounting a torque flange from the company HBM onto the compressor shaft and the signal input as a voltage to the Bruel & Kjaer (B & K) pulse noise and vibration analysis system. Microphone and accelerometers are used to correlate the noise and the vibration to the torsional response synchronized by a tachometer signal on the prop-shaft drive coupling. More
information on this setup can be found in [7, 8, 9]. Dynamic pressure is measured by using high temperature pressure transducers as reported in [5].

2.1. Operating point simulated
The simulated point reported in this work is for the maximum speed of the main rotor at an input speed of 1800 rpm. The measurement point simulated is shown in table 1. In this table, the volume flow rate is normalized with the maximum designed value and the main and gate rotors speeds are normalized with the main rotor maximum design speed.

Table 1. Screw compressor test parameters used in the CFD and CHT simulation.

| Parameter                           | Measured value |
|-------------------------------------|----------------|
| Inlet pressure ($p_1$)              | 1.00 bar       |
| Inlet temperature ($T_1$)           | 21° C          |
| Discharge pressure ($p_2$)          | 3.5 bar g      |
| Discharge temperature ($T_2$)       | 219° C         |
| Normalized volume flow rate ($\dot{V}$) | 0.855          |
| Normalized female rotor speed ($n_f$) | 0.67           |
| Normalized male rotor speed ($n_m$)  | 1.0            |
| Compressor power (P)                | 70.87kW        |

Figure 3. Picture showing temperature measurement points

To enable the simulation and the validation of the CHT case the temperatures at MP1, MP2, MP3, MP4 and MP5 in figure 3 are measured. In addition, the oil sump temperature is measured until steady state is reached, when the temperature of the oil is no longer changing. The oil used is Anderol FG XL 100 food grade oil.
3. Case setup
This section presents the case setups for the baseline (benchmark) geometry of the cold flow and coupled flow with heat transfer. This is important for the DoE part of this work because it would provide a reference for comparison when the optimized geometry of the housing is proposed. Both the cold and the coupled flows with heat transfer are simulated and validated with measurement data. The baseline geometry is denoted as geometry A in this paper. In the CFD setup, monitoring points are defined at each of the measurement points shown in figure 3. The screw compressor simulated includes: Inlet manifold, rotors (main and gate) including the rotor shafts, discharge flange, screw housing, gear casing, bearing carrier (at the non-drive end) and the oil sump as depicted in figure 4. To limit the computational cost due to the many DoE simulations performed and because the oil sump is detached from the rotor housing, the oil sump is not included in the DoE simulations. The cold flow considers only the fluid and the CHT simulations account for both the fluid and the solids. The working fluid is air and the solids are made of the following materials: The gear casing, compressor housing, intake and discharge manifolds are made from Cast Iron while the oil sump and the bearing carrier are made from Aluminum and the rotors and the rotor shafts are made from steel.

For the pre-processing CONVERGE Studio are used. It is important to note that the cartesian mesh used in this tool is automatic and adaptive using Automatic Mesh Refinement (AMR). Therefore, only the boundary conditions, the materials and the numerical parameters have to be specified. The simulation uses a pressure inlet and a pressure outlet. The Reynolds Average Navier Stokes (RANS) and the Pressure Implicit with Splitting Operators (PISO) solver implemented in the CONVERGE is used. To account for compressibility the equation of state for ideal gas is chosen because it depicts the behavior of air very well for moderate pressures. The mesh size was influenced by the base grid. Adaptive mesh refining based on temperature and velocity gradients and fixed embedding for the geometry are used in the setup of the AMR. For the cold flow, 8 mm base grid is used. This was decided by performing grid sensitivity analyses of the flow to ensure that the results obtained were grid independent. The mass flow rate, velocity and the volume flow at the inlet and outlet were checked for mesh independence. The gaps (Radial gap between the rotors, housing gap and axial gap) in the geometry correspond to the cold gaps. In reality and during operation, these will change when the rotors and the casing heat up and expand. To account for this in CONVERGE, sealing and porous media are used to reduce the gaps to mimic the hot gaps in the machine during operation. The sealing and porous media are varied until the predicted volume flow rate matches the measured volume.

Figure 4. Baseline model (Geometry A) used for benchmarking the simulation and optimization.
flow rate. Grid sensitivity analysis is also repeated for the CHT case. In this case, the setup is done such that losses are imposed at the bearings and gears as heat fluxes using:

$$P_{shaft} = P_{isen} + \dot{Q}$$

where $P_{isen}$ is the compressor work given by $P_{isen} = \frac{\kappa}{\kappa - 1} p_1 V [(\frac{p_2}{p_1})^{\frac{\kappa}{\kappa - 1}} - 1]$ and $\dot{Q}$ denote the losses from the gears, bearings, and sealing. $P_{shaft}$ is the motor power, $\kappa$ is the ratio of the specific heats taken as 1.4, $V$ is the volume flow rate and $p_1$ and $p_2$ denote the suction and discharge pressures, respectively. Thus, using equation (1), the losses can be estimated since $P_{shaft}$ is measured and $P_{isen}$ can be determined from measured values. With these losses imposed the CHT case is simulated. The details of the setup for the CHT case can be seen in the CONVERGE User Manual [10]. Because the cooling oil is not included in the model, its effect is not accounted for. Thus, the temperature monitored at the 5 points are exaggerated when compared to measurements. The losses are therefore tuned by reducing their values and the simulation repeated until the temperatures monitored are matching those measured within a reasonable degree of accuracy ($\pm 10\%$ of the measurement). Fixed embedding is used in the solid upto level 3 and level 3 AMR is used on the velocity-Sub-Grid Scale (SGS). The CHT tuning is done for two fluid base grids 16 mm and 24 mm and two solid base grids (2 & 4mm) with 5 fluxes applied at the bearings and the gear location and tuned. A $\dot{Q}$ value of 2kW gave acceptable results when compared to the temperature measured at the 5 monitoring points as shown in figure 5. The temperatures are normalized using the maximum simulated temperature of 451°C located at MP5 in the baseline geometry. The same normalization is used in figure 7. The highest deviation in figure 5(a) is 5.4% and it occurs at MP4 while the maximum deviation in figure 5(b) is 7.3% occurring at MP4. The values reported here were taken at an Crank Angle (CA) 1900°. Based on these result it was decided to use a base grid of 24mm in order to limit the mesh size and reduce the computational time and cost. The results of this base grid when compared to measurements is within good agreement. 4 mm was used as base grid for the solid. For the CHT setup interfaces are defined between the solids and between the fluid and the various solids to account for heat transfer. The heat transfer coefficient is determined implicitly by the CONVERGE solver. For the heat transfer between the solids and the surrounding air a value of 10 W/(m²K) is used for the heat transfer coefficient ($\alpha$). This value is acceptable for the lab conditions because the surrounding air is almost stagnant. Based on results of the mesh sensitivity analyses it can be said that the results reported in this paper are mesh independent and therefore physical. The simulations are done with the CONVERGE solver in parallel using mswitch with 8 cores on a Windows Workstation with an Intel (R) Xeon (R) Gold 6142, 2.6 GHz processor and a RAM of 128 GB. For turbulence modeling, the $k-\varepsilon$ model is used with the law of the wall using the standard wall function [10, 12]. Post-processing is done using Tecplot.

4. Results and discussions

This section presents the cold flow results of the baseline geometry validated with measurement data. Doing this gave us the opportunity to identify potential areas for improvement in the compressor. This is followed by the presentation of the CHT results of the baseline and the optimized geometries. Based on the results the optimized geometry was fabricated using rapid prototyping and tested to check the veracity of our model prediction. The test results are in line with what the model predicted.

4.1. Cold flow

The cold case simulated is compared and validated with measurement data as shown in table 2. The values are normalized using the maximum values for each parameter. CFD is reporting a
Figure 5. CHT simulation of the baseline case and validation with measurement

Table 2. Comparison of CFD prediction with measurement data

| Parameter                                | CFD | Measured/computed | % Error |
|------------------------------------------|-----|--------------------|---------|
| Normalized mass flow rate (-)           | 1.0 | 0.93               | 7.3     |
| Normalized volume flow rate (-)         | 1.00| 0.95               | 5.9     |
| Normalized compressor power(-)          | 1.0 | 0.97               | 2.86    |
| Axial load on male rotor (N)            | 1943| 1732.3             | -       |
| Axial load on female rotor (N)          | 220 | 95.64              | -       |

larger volume flow and hence the compressor power computed in CFD is larger than measured. This is because in the measurement there is a pressure drop at the suction and this drop was not included in the CFD. The suction pressure was taken as atmospheric (See table 1). The same is true for the main and gate rotor torques. The compression work is done by the main rotor while
the gate rotor transports the fluid. Hence, the main rotor torque from CFD is 48.59Nm and the gate rotor torque is 3.35Nm. The reason for the difference between the measured and predicted axial load is because the axial load is not measured but calculated using a simplified analytical model that does not consider the 3D effects like turbulence and over and under-compression. The CFD predicted axial load consider these effects. The trend is however as expected with the main rotor axial load much larger than the gate rotor.

4.2. Coupled flow with heat transfer
The CHT results of the baseline case are presented here and compared to that of the optimized geometry D with the larger inlet. The monitoring points temperature are compared at simulation time 1900deg Crank Angle (CA). The discharge air temperatures are compared at 2800deg CA. Other performance parameters like the rotor torque and hence the compressor power are compared. This is a good indicator for pressure drop inside the compressor. The lower the compressor power the lower the pressure drop $\Delta p$.

Figure 7 shows the normalized temperatures of the baseline geometry A compared to the optimized geometry D relative to the measurements. The values are normalized using the maximum value in temperature occurring at MP5 in geometry A. The optimized geometry is showing lower temperatures at MP1, MP4 and MP5, which are crucial in the cooling of the compressor and the oil sump. At MP2 and MP3 the values are close. At simulation time 2800deg CA the discharge air temperature in the optimized geometry D is higher than in geometry A as expected because the air is first used to cool the compressor before it is compressed in geometry D as shown in table 3.
Figure 7. Normalized predicted bearings and housing temperatures

Table 3. Normalized discharge air temperature comparison between CFD and measurement

| Sim. time (CA deg.) | Measured Temp. (-) | Geometry A (Baseline) (-) | Geometry D (Optimized) (-) |
|---------------------|--------------------|---------------------------|---------------------------|
| 2800                | 1.0                | 0.9045                    | 0.9085                    |

The compressor power predicted in the optimized geometry is 70.7 kW and that predicted in the baseline case is 73.05 kW, which is an improvement since more of the power in the baseline case would end up as heat due to losses and this will in turn lead to an increase in temperature of the cooling oil.

The contour plots of the temperature of the housings and solid rotors using the same scale are compared as shown in figure 8. The trend clearly shows that the cooling in the optimized geometry D is better than in the baseline geometry A. Although the simulation times are different it is important to note that the position of the rotors are identical in all the plots.

4.3. Validation of the DoE results
To check the veracity of the DoE results, geometry D is fabricated using rapid prototyping and measured. Performance maps for the volume flow rate, temperature, efficiencies, etc. are determined and compared with the baseline geometry using the same cooling oil. The measurements are done at input speeds of 1000 rpm, 1200 rpm, 1400 rpm, 1600 rpm and 1800 rpm and at different discharge pressures: No load, 0.5 bar(g), 1.0 bar(g), 1.5 bar(g), 2.0 bar(g) and 2.5 bar(g). In all the tests the same trend is observed and so we report here the results of the worst case scenarios at input speeds of 1000 rpm and 1800 rpm as shown in figure 9. The results are corrected to the standard room temperature of 20°C.

In figures 9(a) and 9(b), the results of the discharge air temperature and oil sump temperature at 1000 rpm are shown. As expected, the discharge air temperature in the optimized geometry increased because the larger volume of air is used to first cool the oil and the rotor casing before it is compressed. At full load, the temperature difference is 10°C between the baseline case and the optimized case but the discharge temperature is still within the customer specification. The higher temperature difference is also due to the higher losses and the higher leakages occurring
Figure 8. Normalized contour plots of the temperature of the housings and the rotors inside the compressor at the lower speed. However, the oil sump temperature at steady state in the optimized housing is 20°C less than that in the baseline machine. In figures 9(c) and 9(d), the results at 1800rpm input speed are reported. Similar to the 1000rpm input speed case the discharge air temperature in the optimized geometry increases but the increase is lower at 2°C and crucially, the oil sump temperature decreases by 30°C to 123°C, which is very close to the required oil sump temperature of 120°C from our sales department. The trend seen here is in agreement with the prediction made by the CHT results.

To understand why the losses are higher at the lower input speed and why the discharge air temperature is higher in the optimized geometry, the efficiency curves are plotted for the benchmark or baseline geometry verses the optimized housing as shown in figure 10. It is observed that in the optimized housing the design point has shifted from the optimum internal volume compression of $\pi_1 = 2.1$. At 1800 rpm, the maximum point of the efficiency curve is close to the maximum point in the baseline machine. At 1000rpm the difference is more pronounced and the maximum efficiency is at $\pi = 2.5$ in the baseline machine and this has shifted to $\pi = 2.07$ in the optimized case. This means that a further optimization of the discharge contour to determine the optimum internal volume compression is required.

5. Conclusions
This paper reports the successful use of CFD and CHT to optimize and improve the cooling inside a screw compressor. It shows that CFD can be used as an invaluable tool in the design and optimization of screw compressors. This reduces the time and cost of development by limiting
Figure 9. Normalized measured discharge air temps.

Figure 10. Plots of the normalized measured compressor efficiencies

the testing and the number of prototypes required.

As future work, the optimum internal volume compression required to give the maximum compressor efficiency in the optimized geometry D will be determined using DoE and measurements and presented. FSI will be used to determine the rotors deflection and hence the hot gaps inside the machine.

Acknowledgment
The author would like to thank Timo Martschinke and Andreas Matt for carrying out the measurements. I would also like to thank Riccardo Gretter of Convergent Science for his support.
References

[1] Hauser J and Schulze-Beckinghausen P 2016 Performance Prediction of Oil-Free Screw Compressor using Computational Fluid Dynamics (VDMA Compressors Users Int. Forum 2016, Düsseldorf, Germany), September 14-15, pp. 303-312

[2] Rane S, et al 2015 Analytical Grid Generation for Accurate Representation of Clearances in CFD for Screw Machines IOP Conf. Ser.: Mater. Sci. Eng. 90 012008

[3] Spille-Kohoff A, et al 2015 CFD Simulation of a Screw Compressor including Leakage Flows and Rotor Heating IOP Conf. Ser.: Mater. Sci. Eng.90 012009

[4] Hesse J, Spille-Kohoff A, Hauser J and Schulze-Beckinghausen P 2014 Structured Meshes and Reliable CFD Simulations: TwinMesh for Positive Displacement Machines (Int. Conf. on Screw Machines 2014, TU Dortmund University, Dortmund, Germany) September 23-24 2014 (VDI-Berichte 2228) pp. 297-105

[5] Willie, J 2017 Use of CFD to predict trapped gas excitation as source of vibration and noise in screw compressors IOP Conf. Ser.: Mater. Sci. Eng.232 012021

[6] Willie, J 2018 Analytical and numerical prediction of the flow and performance in a claw vacuum pump IOP Conf. Ser.: Mater. Sci. Eng.425 012026

[7] Willie J, Asal W and Sachs R 2014 Analysis of the Noise and Vibration of a Dry Screw Compressor (Int. Conf. on Screw Machines 2014, TU Dortmund University, Dortmund, Germany) September 23-24 2014 (VDI-Berichte 2228) pp 91-308

[8] Willie J and Sachs R 2015 Structural and Torsional Vibration Analysis of a Dry Screw Compressor IOP Conf. Ser.: Mater. Sci. Eng.90 012005

[9] Willie J and Sachs R 2016 Structural and Torsional Vibration and Noise Analysis of a Dry Screw Compressor (Proc IMechE Part E. J Process Mechanical Engineering 0) vol 0, DOI: 10.1177/0954408916648989

[10] CONVERGECFD Manual Series 2019 Conjugate Heat Transfer Modeling, (CONVERGE CFD Software) pp. 521-536

[11] Fister W 2013 Fluidenergiemaschinen: Physikalische Voraussetzungen, Kenngrößen, Elementarstufen der Strömungs- und Verdrängermaschinen, (Berlin: Springer-Verlag, Band 1)

[12] Pope S B 2000 Turbulence Flows, ed 1 (Cambridge University Press)