Use of CFD to predict trapped gas excitation as source of vibration and noise in screw compressors

James Willie
Gardner Denver Schopfheim GmbH, Johann-Sutter-Straße 6+8, 79650 Schopfheim, Germany
E-mail: james.willie@gardnerdenver.com

Abstract. This paper investigates the source of noise in oil free screw compressors mounted on highway trucks and driven by a power take-off (PTO) transmission system. Trapped gas at the discharge side is suggested as possible source of the excitation of low frequency torsional resonance in these compressors that can lead to noise and vibration. Measurements and lumped mass torsional models have shown low frequency torsional resonance in the drive train of these compressors when they are mounted on trucks. This results in high torque peak at the compressor input shaft and in part to pulsating noise inside the machine. The severity of the torque peak depends on the amplitude of the input torque fluctuation from the drive (electric motor or truck engine). This in turn depends on the prop-shaft angle. However, the source of the excitation of this low torsional resonance inside the machine is unknown. Using CFD with mesh motion at every 1° rotation of the rotors, it is shown that the absence of a pressure equalizing chamber at the discharge can lead to trapped gas creation, which can lead to over-compression, over-heating of the rotors, and to high pressure pulsations at the discharge. Over-compression can lead to shock wave generation at the discharge plenum and the pulsation in pressure can lead to noise generation. In addition, if the frequency of the pressure pulsation in the low frequency range coincides with the first torsional frequency of the drive train the first torsional resonance mode can be excited.

1. Introduction
Simulation technique is playing an increasing role in the development process of screw compressors at Gardner Denver (GD). This reduces development time by reducing expensive tests in the laboratory. The availability of computing power at a relatively lower cost also makes modeling larger cases using finer meshes possible in industry. This paper presents one such example of how CFD can be used in industry to help in the development process of screw compressors. Using this technique in the modeling of turbo-compressors is commonplace but in the case of screw compressors, which are positive displacement machines, this is not the case. Some of the few and most recent cases that report the modeling of screw compressors can be found in [1, 2, 3, 4]. The reason is due to the complex nature of the screw profile that must be simulated and the very small gaps/clearances in these machines that must be modelled. The screw mesh in this paper was generated using TwinMesh from CFX-Berlin together with the CFX-solver. The Junction Box routine is used to enable mesh motion of the screw. Because of the complicated nature of the screw, the rotation angle for generating each mesh is 1°. The fine mesh generated makes it possible to capture the complicated physics taking place inside the compressor. For example, it was possible to capture the over-compression at the discharge.
in the absence of the axial gap/equalizing chamber. It is shown that in this case, a trapped gas region is created that leads to over-compression at the discharge. To mitigate the effect of the pulsation created by this trapped gas a pressure equalizing chamber or groove in the discharge side cover is used. It is seen that using this groove or chamber reduces the pressure pulsation significantly. This is because the axial discharge port has a delay in closing and this can create a residual volume, $V_r$, when the port closes. As a result, $V_r$ is still significant when the discharge port closes, especially when the number of male rotor lobes are few and as the rotor rotation continues it leads to a very high pressure of the fluid contained in it. When left unchecked, it can lead to increase in compressor power and a loss of efficiency. The prediction by CFD reported here is validated with measurement data. The CFD model is further validated by comparing the compressor torque/power and the pressure and mass flow rates to those determined in measurements.

One of the main findings reported in this paper is the possibility that the low frequency torsional modes observed in screw compressors that are mounted on trucks and driven via the truck PTO could be excited by the pulsation of the trapped gas reported in this work. The prediction and measurement of torsional resonance in screw compressors has been reported in [5, 6, 7]. In them, the authors showed that based on sensitivity analysis, the compressor behaves as a rigid body and that the most sensitive element responsible for the first torsional mode excitation is the prop-shaft in the case of the electric motor drive and the thin PTO shaft in the case of the truck PTO drive. One manifestation of this torsional resonance is the presence of side bands at all the gear meshing frequencies and their harmonics and the presence of pulsating noise that was found to be due to amplitude modulation and not frequency modulation [6, 7]. The element providing the excitation within the machine was not reported. Based on the current work, it is shown that when trapped gas is present, over-compression is seen in the pressure data and Fast Fourier Transform of the pressure time series data showed clear side-bands at the lobe passing frequencies and harmonics of the compressor. When the pressure equalizing chamber is used, this is not the case. Furthermore, the amplitude at the low side-band frequency is much higher in the pressure time series data without the pressure equalizing chamber than with the pressure equalizing chamber.

This paper begins with the description of the case that is simulated, including the measurement point that is modeled and the machine layout. After this, the geometry and the mesh of the test case simulated are presented and this is followed by the presentation of the case setup in CFX-pre and the computer hardware on which the case is run. This is proceeded by the presentation of the results and finally, the conclusions are given.

2. Measurement and layout of the machine

The schematic showing the setup of the measurement and a picture of the measurement setup is depicted in Figure 1.

In addition to performance measurement, waterfall plots are generated using the torque run-up test described in [7] to check if any torsional mode is excited. The Bruel & Kjaer (B&K) pulse noise & vibration analysis system for torque measurement is used. The torque is measured by mounting the torque flange on the compressor input shaft. In Figure 1, the axial, profile and housing gaps are sketched. One key aspect in designing screw compressors for transport is noise. To limit noise, the suction and discharge ports are sized in such a way that their velocities are low. This is critical because based on wind tunnel test, the sound pressure level, $SPL \sim (Ma)^n$, where $Ma$ is the Mach number and $n$ is a real number. After finding the inlet duct diameter based on the volume flow rate to limit the velocity to a low value, conservation of mass and continuity equation are used to determine the size of the discharge port. The inlet and discharge ports are also aerodynamically optimized in order to limit the pressure drop in the machine. One
of the key findings is that during the operation of screw compressors, most of the compression work is done by the male rotor while the female rotor transports most the fluid. Hence, the direction of rotation of the female rotor dictates the flow direction of the air exiting at the discharge. The fluid leaves the rotor with a given angular momentum that is given by $mvr$, where $m$ is the fluid mass, $v$ is its velocity and $r$ is the tip radius of the female rotor and this dictates the flow direction. This information is helpful in deciding on where to place the discharge and how to optimize the flow at the discharge.

The $L/D$ of the rotors profile used is 2.06 and the Center Distance (CD) is 100mm. To limit noise, the maximum male rotor speed is low while the input speed range is 1000-1800 rpm. The female rotor has more lobes than the male rotor as in most screw machines. The rotors used are large enough to limit any large torsional response (angular deflection) and the profile backlash are designed to ensure no torque fluctuation (torque reversal) at the rotors.

### 2.1. Measurement point used in the simulation
The case corresponding to the maximum male rotor speed at input speed of 1800 rpm is selected and used in the CFD simulation. The measurement point simulated is shown in Table 1. In this table, the volume flow rate is normalized with the maximum design value and the female and male rotor speeds are normalized with the male rotor maximum design speed.

| Parameter                        | Measured value |
|----------------------------------|----------------|
| Inlet pressure ($p_1$)           | 1.002 bar      |
| Inlet temperature ($T_1$)        | 21° C          |
| Discharge pressure ($p_2$)       | 2.2bar g       |
| Discharge temperature ($T_2$)    | 181°C          |
| Normalized volume flow rate ($\dot{V}$) | 1.041     |
| Normalized female rotor speed ($n_f$) | 0.6       |
| Normalized male rotor speed ($n_m$) | 1.0       |
| Compressor power (P)             | 69.26kW        |
3. Geometry and mesh generation using TwinMesh

The screw compressor simulated is comprised of the following key parts used in the simulation of the air from the suction to the discharge: inlet manifold, rotors (male and female), discharge cover and the discharge plenum and duct as shown in Figure 1. The mesh is divided into two parts: the rotor and the stator. The rotor mesh is made of the screws. The stator mesh is made of the suction fluid and the discharge fluid and it is generated using Ansys Meshing. The two regions are imported into CFX-pre together with the defined interfaces that makes transfer of information between the static and moving regions of the geometry possible. Two cases are simulated. In the first case, denoted as Case 1, the axial gap and equalizing chamber or groove at the discharge are omitted in the geometry and in the second case labelled as Case 2, the axial gap and equalizing chamber are included at the discharge. In both cases, the stator mesh at the suction side and rotor mesh remained the same.

3.1. Mesh generation

The stator mesh is an unstructured tetrahedral mesh as shown in Figures 2 and 3. In Figure 2, a sectional view of the mesh at the suction side is shown. The mesh has a total of 1,615,484 cells.

![Sectional view of the suction stator mesh](image)

**Figure 2.** Sectional view of the suction stator mesh, 1,615,484 cells, inlet located left

In Figure 3(a), a sectional view of the stator mesh at the discharge in Case 1 is shown and in Figure 3(b), a sectional view of the stator mesh at the discharge in Case 2 is shown.

![Sectional view of the discharge stator mesh](image)

(a) Case 1, 507,470 tetra cells  
(b) Case 2, 667,222 tetra cells

**Figure 3.** Sectional view of the discharge stator mesh used in the CFD simulation

To generate the rotor mesh, TwinMesh is used. It is a template based approach used for generating structured meshes for screw rotors. It is developed by CFX Berlin GmbH [3, 4]. In the case presented here, the 2D structured mesh is generated for every 1° rotation of the rotors. The rotor curves are imported into TwinMesh as shown in Figure 4 using a cut sectional view.
Figure 4. Cut sectional view showing a cut through the rotor curves imported into TwinMesh

The cold gaps values for the housing, the profile and the axial gaps at the suction and discharge are included. After defining the mesh parameters, the 2D mesh is generated as shown in Figure 5 for a rotation angle of $0^\circ$. This is repeated for every angle up to $360^\circ$ after which the mesh is generated in 3D by projecting the 2D mesh in the axial direction along the rotor axis for the length of the rotor. The mesh information is as follows: The male rotor has 6340 quad elements in 2D and 634000 hexahedral elements in 3D. The female rotor has 5540 quad elements in 2D and 554000 hexahedral elements in 3D. For mesh quality, it is recommended a minimum angle of 24 is maintained. Rotor 1 is the male rotor and Rotor 2 is the female rotor. When viewed from the discharge size, the male rotor rotates in the counterclockwise direction while the female rotor rotates in the clockwise direction.

Figure 5. Cut sectional view showing TwinMesh 2D grid of the male and female rotors at $0^\circ$

3.2. Case setup in CFX-pre

The setup is done by importing the rotor and stator meshes into CFX-pre. The boundary conditions used are: suction inlet is taken as an Opening and under the Mass and Momentum Option, Opening Pressure and Direction is used and at the discharge an Opening is also used with Entrainment under the Mass and Momentum Option and the Turbulence Option is set to Zero Gradient in order to specify the flow direction as required. This way, the solver computes the mass flow rate based on the pressure and temperature defined from measurements. Interfaces are defined between the static and dynamic mesh regions of the suction and discharge sides. The Shear Stress Transport (SST) turbulence model is used [8]. It combines the k-$\omega$ and the k-$\varepsilon$ turbulence models such that the former is used in the inner region of the boundary layer and then switches to the latter in the free shear flow. The Total Energy option is used for heat transfer and air ideal gas is used for the fluid material since the flow is compressible and
for the mesh motion, the Junction Box Routine is used. The male and female rotors are given
the appropriate rotational speeds and directions. Heat transfer within the solid rotors are not
accounted for in the simulation.

Cases 1 and 2 are run on a HP Z820 16 Core Computer Workstation with a RAM of 64GB
and an Intel Xeon processor. During the run, 8 of the 16 cores are used and in both cases, a
minimum of three revolutions of the rotors are simulated.

4. Results and discussions
The results section begins with the discussion of the flow optimization at the discharge in order
to reduce the pressure drop inside the machine. This is followed by the discussion of the two
Cases 1 and 2 considered. Three design variations with regards to the optimization at the
discharge are considered, but only the third is presented here. One aim in the development of
turbo and screw compressors is the reduction of the pressure drop in the machine, especially
at the discharge. It has implication for noise, compressor power and in some cases for pressure
pulsation as it will be shown in this paper. In the analysis performed here, attempts are made
to eliminate flow recirculation at the discharge when the air exits at the discharge port. Figure
6 shows the velocity vectors in a plane at the discharge with and without the flow recirculation.
In screw compressors, heat generation is mainly due to gas compression and expansion at the
discharge port. Creating a flow re-circulation at the discharge can have other localized effects
like increased thermal gradients.

![Figure 6. Velocity vectors in a plane at the discharge (Left is without flow deflector & right is
with the flow deflector)](image)

4.1. Performance comparison
The performance of Cases 1 and 2 are compared and validated with measurement data. The
following performance parameters are compared: average maximum pressure at the discharge,
the male and female rotor torques, the compressor power and the mass flow rate as shown in
Table 2. The mass flow rate is normalized using the measured value.

The pressure data shows that in Case 1, the pressure is much higher and we have over-
compression as a result of the trapped gas (See Figure 7). In Case 2, there is no over-compression
and the pressure in the trapped gas region or residual volume region is the same as the pressure
at the discharge.
Table 2. Performance comparison of cases simulated and validation with measurement.

| Parameter                        | Case 1 | Case 2 | Measurement |
|----------------------------------|--------|--------|-------------|
| Max. avg. press. at discharge (bar abs.) | 4.6    | 3.6    | 3.2         |
| Male rotor avg. torque (Nm)      | 84.11  | 79.5   | -           |
| Female rotor avg. torque (Nm)    | 1.11   | 2.997  | -           |
| Compressor power (kW)            | 73.88  | 70.88  | 69.26       |
| Norm. average mass flow rate (-) | 0.72   | 0.77   | 1.00        |

As already mentioned, the compression work is done by the male rotor as can be seen from the male rotor torque value when compared to the female rotor torque. In the two cases considered, it is seen that the male rotor torque and power are larger in Case 1. Due to the over-compression in Case 1 the rotor must do more work to overcome the over-compression. The compressor power in Case 2 is closer to the measured value than in Case 1. The mass flow rate deviates a lot more from the measured value due to leakages in the gaps. The gaps values used in the mesh generation are values for the cold flow. In reality, the hot gaps values are smaller than the cold gaps values due to the thermal expansion of the rotors. The hot gap values are however more difficult to measure when the machine is in operation. Due to the large pressure at the discharge, leakages are also possible which leads to losses in the volumetric efficiency of the machine. In both cases, mass conservation is achieved and the periodic mass flow showed statistical convergence. The flow in the gaps are however treated as stationary instead of rotating. Due to over-compression, the corresponding rotor temperature in Case 1 is higher as shown in the contour plot in Figure 8. The measured air temperature at the discharge is 454K and the average temperature at the discharge in Case 1 is 657K while that in Case 2 638K. The rotor temperatures in the CFD are larger because the rotor solid and heat transfer through the solid rotors are not accounted for in the CFD; only the surface of the rotors are accounted for. Also, the cooling of the rotors through convection are not accounted for in the CFD. During measurement, the measured temperature of the cooling oil is 10°C lower in Case 2 when compared to Case 1, with both machines always having an axial gap.

Compared to other methods like the Immersed Solid Method [3], TwinMesh is able to accurately predict the physics of the flow taking place inside screw machines. With Immersed Solid, a momentum source term is used to force the fluid to move with the rotor velocity and it does not account for mesh motion. However, it is able to reproduce the velocity trends and...
4.2. Pressure pulsations

To study the effect of pulsation on the machine in Cases 1 and 2, pressure time series data is taken at the discharge and Fast Fourier Transform (FFT) is performed and plotted as shown in Figure 9. The ordinate axes have the power spectra density (PSD) that is given by \( PSD = 20 \log_{10}(p) \), since in this case the measured pressure is not referenced to \( p_{ref} = 2 \times 10^{-5} \text{Pa} \). If referencing is done, then \( PSD = 20 \log_{10}\left(\frac{p}{p_{ref}}\right) \). The sampling rate of the pressure data is 10 kHz and 10 000 samples points are taken. In both cases presented, the machine was run at an input speed of 1800 rpm, 2 bar (g) using a 90 kW 4 pole asynchronous motor. The data shows the fundamental lobe passing frequency at around 400 Hz and the first harmonic as expected. But in the data of Case 1 we have sidebands of 30 Hz at the lobe passing frequency. When the data is zoomed to show the lower frequency range it is seen that at 30 Hz we have a peak suggesting that 30 Hz could be providing the excitation to drive torsional resonance originating from the prop-shaft as shown in Figure 10. This frequency is \( 1 \times \) input speed at 1800 rpm and therefore synchronous as previously reported by the authors [5, 6, 7]. In the paper reported in [5], it was shown that the torsional frequency occurring in the screw compressor driven by an electric motor was occurring at \( 1 \times \) input speed. In Figure 10, the peak amplitude of in Case 1 of 30 Hz is 38 dB while that in Case 2 is 28 dB.

The peak in the FFT data of Case 1 is clearer in Figure 10, where the frequency range has been narrowed to 50 Hz.

Pressure time series is also taken in the CFD and plotted as shown in Figure 11 for the two cases. The pressure is normalized using the maximum pressure Case 1. Clearly, the pressure amplitude in Case 1 is larger than in Case 2.

The small difference in the plots shown in Figure 9 and 10 is because the pressure transducer is placed at the discharge pipe and not in the trapped gas region. When the air exits the compressor, there is a sudden expansion, which leads to a decrease in pressure in the case with over-compression. The difference between them and the CFD plot shown in Figure 11 is mainly because in the CFD, the maximum pressure at the discharge is monitored, which is in the trapped gas region. Also, in Case 1 no axial gap is included in the CFD, whereas in the measurement, there is an axial gap but no equalizing chamber. The CFD plot is also done using a linear scale whereas in the measurement a non-linear scale is used. The trends between the CFD and the measurement data are however similar, as expected.
The difference in pressure pulsation in both cases has large implication for design. For example, Case 1 will demand more motor power to be able to run at 1800rpm input speed. Noise and vibration will also be more in this case and the generation of flow instability like shock waves is possible in addition to having larger thermal gradients.
5. Conclusions

This paper reports the successful simulation of the flow inside a screw compressor built for use in bulk transport on trucks. The compressor is powered using PTO on trucks and it is susceptible to noise and vibration and torsional resonance. In particular, the first torsional mode is easily excited in the PTO shaft. The response to this excitation inside the machine can lead to sidebands at all gear meshing frequencies and their harmonics and pulsating noise is possible [5, 6, 7]. Sidebands in the pressure data at the rotors lobe passing frequency and harmonics is also visible. The following conclusions can be drawn:

- Using CFD it is possible to reproduce the flow and physics taking place inside a screw compressor. This is possible due to the successful integration of TwinMesh for the structured mesh generation of the rotors and the CFX solver using the Junction Box Technique for the mesh motion.
- Flow optimization can lead to pressure drop reduction and to reduced noise and reduced thermal gradients at the discharge of such compressors. The power consumption is also improved.
- The inclusion of a groove/pressure equalizing chamber at the discharge can lead to reduced pressure pulsations and noise and thermal gradient inside the machine.
- The pulsation of the gas inside the trapped pocket may provide the necessary excitation for the first torsional mode outside the compressor, if a frequency of the pulsation coincides with the first torsional frequency of the compressor. This can lead to pulsating noise at the gears inside the compressor if care is not taken to optimize the gears at the micro-geometry level in order to ensure the right backlash and the mid-flank contact that is needed.

Acknowledgment

The authors would like to thank Francesco Maraini and Steve Munt, who carried out the measurements reported at GD Robuschi, Parma, Italy, and at GD DRUM, Bradford, UK, respectively. Their valuable contribution through discussions is also acknowledged.

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, Kovacevic A, and Stosic, N 2015 Analytical Grid Generation for Accurate Representation of Clearances in CFD for Screw Machines (International Conference on Compressors and their Systems, City University of London, London, UK) September 7-9, http://iopscience.iop.org/article/10.1088/1757-899X/90/1/012008
[3] Spille-Kohoff A, Hesse J and El Shorbagy A 2015 CFD Simulation of a Screw Compressor including Leakage Flows and Rotor Heating (Int. Conference on Compressors and their Systems, City University of London, London, UK) September 7-9, http://iopscience.iop.org/article/10.1088/1757-899X/90/1/012009
[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, 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
[6] Willie J and Sachs R 2015 Structural and Torsional Vibration Analysis of a Dry Screw Compressor (9th Int. Conference on Compressors and their Systems, City University of London, London, UK) September 7-9 2015, http://iopscience.iop.org/article/10.1088/1757-899X/90/1/012005
[7] 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
[8] Pope S B 2000 Turbulence Flows, ed 1 (Cambridge University Press)