Validation of FlowVision CFD on ICCS2015 Test Case: Application of Gap Model\textsuperscript{TM} and SGGR\textsuperscript{TM} for Leakage Flow Prediction in a Dry Screw Compressor

U Ozturk\textsuperscript{1}, S Soganci\textsuperscript{1}, V Akimov\textsuperscript{2}, O Tutkun\textsuperscript{1}, A Aksenov\textsuperscript{2}

\textsuperscript{1}Capvidia NV, Leuven, Belgium
\textsuperscript{2}TESIS, Moscow, Russia

E-mail: deniz.ozturk@capvidia.com

Abstract. Clearances and leakages through them play a vital role in determining the efficiency of positive displacement machines, especially of screw compressors. It is of interest to accurately predict the overall machine behavior including the leakage flows, even during the early design phases.

CFD (Computational Fluid Dynamics) methodologies can potentially address this problem however, the computational efficiency is limited by the well-known issue: dimensionality. This issue arises due to the fact that rotors are sized in tens of centimeters whereas the clearances are in the order of (sub) microns.

Here we present a reduced order model called the Gap Model, embedded in FlowVision CFD software package to overcome this problem. Our approach is based on Poiseuille flow assumption and intends to evaluate fluid flows within very thin channels without resolving these regions with traditional fine mesh discretization. In such cases when two wall boundary conditions, such as rotor-to-rotor or rotor-to- chamber in a screw compressor, are close to each other less than mesh resolved dimensions, only one row of computational cells are constructed between the walls and an extra source term is added to the momentum equation, describing the frictional force produced by clearance walls.

The presented model is applied to ICCS2015 Test Case Screw Compressor developed by Centre for Compressor Technology of City, University of London. CFD predictions using this leakage model are in agreement with experimental results within 6.3 percent maximum error. The calculation times are reduced down to less than a day for obtaining stable and converged results.

1. Introduction

Screw machines are one of the most commonly used machines in compressing gases. The advantage of these positive displacement-type machines is their compact nature with relatively lower quantity of moving parts, making the suitable for various industries such as oil & gas, refrigeration, food industry etc. These type of machines can operate over a wide range of flow rates and pressures.

Apart from some exceptional cases, they consist of two or more helical rotors. These rotors mesh into each other with the idea of creating an inter-lobe volume which increases first to a maximum and decreases back. The increase in the inter-lobe volume creates a suction, then as the fluid travels it is isolated from the suction port (normally this occurs at the position
corresponding to maximum inter-lobe volume) and is subjected to a decrease in volume, resulting in increased pressure. There are reports focusing on the design and design optimization of screw machines which have been accumulating since 60s [1, 2, 3, 4]. Early calculations to assess flow properties in a screw compressor relied on the assumption of no leakage, reducing the calculations to ideal gas laws. Later, more computational power enabled designers to implement somewhat detailed models to be used [5, 6, 7, 8]. A brief review of these approaches is reported by Stosic et al. [9] and Yan et al. [10].

Furthermore, with the newly developed highly accurate metrological systems and manufacturing techniques the manufacturing accuracy has been increasing at a good pace. This makes it possible to attain lower clearance levels which is a key step to increase efficiencies. In parallel to this, the now well-known technology of Computational Fluid Dynamics (CFD) and their extensions (FSI and optimization) have been being implemented (or have potential to be implemented) in the design process to resolve the internal and external flow fields, distribution of flow properties (such as pressure and temperature), thermal changes etc. both in time an space.

In CFD of positive displacement machines, the grid generation process usually requires specialized tools. Furthermore, it is important to perform simulations based on the actual CAD geometries.

To attack this problem, a grid generator is developed for screw machines by Kovacevic [11] which has evolved to be an extremely useful tool for CFD engineers in this field. Later, other similar tools have been developed as well such as spline based method proposed by Moller et al. [12].

Challenging flow physics, changing fluid properties, the existence of complex flow passages and grid generation within these volumes, in overall, makes the CFD a hard task for positive displacement machines.

In this text, a CFD software package, which consists of automatic grid generator, pre- and post-processor is used to solve an example screw compressor case. An already existing reduced order method called Gap Model\textsuperscript{TM} to overcome the dimensionality issues is applied and comparison of the results with experimental measurements is presented. This text intends to highlight the accuracy, computational efficiency and processing time of the used method.

2. Approach

A C++ implemented commercial named FLOWVISION CFD (versions 3.09.05 and 3.10.01) was used. This solver is a general purpose CFD software package based on finite-volume method, covering 2D/3D inviscid and Navier-Stokes formulation for laminar and turbulent flow regimes accompanied with various physical modules such as heat and mass transfer, phase interactions, chemical reactions and ablation etc.

This CFD solver is a segregated, implicit/explicit all-Mach number solver, which uses split algorithms solving Navier-Stokes equations based on Belotserkovsky’s approach [13], an accurate approximation scheme for convective impulse transport. An implicit approach introduced in 2014 is an efficient and more stable extension of the previous implicit methods, allowing computation of supersonic flows (M>1) with large (CFL>10) time steps [14].

The CAD resolution and automatic grid generation in this software is based on non-staggered Cartesian grid with adaptive local refinement and a Subgrid Geometry Resolution Method (SGGR\textsuperscript{TM}) for description of curvilinear complex boundaries. This approach was first reported by Aksenov et al. in 1996 and 1998 [15, 16].

In simulations such as the current one, the use of original CAD geometry without any simplifications or extra operations is extremely important both for accuracy and to cut the engineering time to prepare the CAD. The SGGR\textsuperscript{TM} allows bringing the original CAD to the CFD simulation and creates the basis for the automatic grid generation, which is also a time
consuming step especially in CFD simulations of positive displacement machines. SGGR\textsuperscript{TM} is an advanced analogy of the primitive-cut cell methods. After local adaptations, surfaces of the Computer Aided Design (CAD) model or Finite Element (FE) mesh (for FSI studies) were used as a boundary for the fluid domain. These facets are formed by a set of external faces of the CAD or FE mesh. An example boundary cell is shown in Figure 1.

Figure 1. An SGGR\textsuperscript{TM} boundary cell. Green surface is the boundary of the fluid domain defined by the original CAD geometry and/or FE mesh. Defining the borders through SGGR\textsuperscript{TM} allows for boundary resolution to be independent of grid density. This creates a basis for automatic grid generation and also correct functioning for the Gap Model\textsuperscript{TM} through accurate calculation of clearance height.

Through SGGR\textsuperscript{TM} any arbitrary motion (either based on FEA or prescribed motions) in 6-DOF can be implemented without the necessity of manual grid generation. If there are objects moving/rotating in a simulation, or entering/exiting from CFD domain, the grid is automatically regenerated according to the user definitions (adaptations etc.).

In order to efficiently calculate the leakage flow behavior without fully resolving the flow field, a feature named Gap Model\textsuperscript{TM} is used. This approach is capable of eliminating dimensionality issues with the order of $10^6$, which means down to micron level clearances in the current case. This approach results in a significant decrease in total number of cells and computational effort without sacrificing significant accuracy. Gap Model\textsuperscript{TM} allows for calculation of forces and fluxes in thin channels between solid surfaces by analytic expressions instead of fully resolving the gap flow. The surfaces by which the gap is to be formed are identified if they are either the surfaces belong to different “wall” or “connected” boundary conditions; or if the angle between the normals to the surfaces is between 120 and 180 degrees. Finally, SGGR\textsuperscript{TM} allows accurate calculation of the clearance height, which is necessary for the correct functioning for the Gap Model\textsuperscript{TM}.

Standard Gap model model implemented in FlowVision assumes that a laminar Poiseuille flow occurs in a clearance. Then the bulk friction force exerted on the fluid is given by the Eq. 1:

$$ F = -\frac{12\mu VS}{\delta \Omega} $$

where $\Omega$ is gap cell of volume, $\delta$ and $S$ is the channel width and half-bounding area respectively. The velocity profile across the channel is assumed to be parabolic and the average
velocity to be \( V \).

In the case of heat transfer, if temperature on both gap forming walls are specified, heat flux is given based on Eq. 2:

\[
J_{q,1} = -\frac{2\lambda}{\delta} (3T - 2T_1 - T_2) \mathbf{n}
\]  

(2)

where \( \lambda \) is molecular thermal conductivity, \( \mathbf{n} \) is normal to the wall, \( T_1 \) and \( T_2 \) are temperatures of the gap forming walls 1 and 2 respectively. For further details and generalized cases, readers should refer to Ref. [17] and Ref. [14].

As an example, the auto detected gap cells for one of the cases solved within this work is shown in Fig. 2.

![A snapshot of automatically detected Gap Cells at an instant. Example is from the current simulation talk.](image)

**Figure 2.** A snapshot of automatically detected Gap Cells at an instant. Example is from the current simulation talk.

### 3. Case Description

The test case presented here is an oil-free 3/5 lobed twin screw compressor with \( N \) rotor profiles and a built-in volume ratio of 1.8. The rotor diameters are 127.45 and 120.02 mm for male and female respectively, and center to center distance is 93.00 mm. Male rotor operation range is from 6 to 14k rpm with a wrap angle of 285.0 deg. The model designed and operated by Centre for Positive Displacement Compressor Technology of City University and CAD geometry used in the simulation are shown in Fig. 3.

The reported clearance values are 160, 180 and 120 microns for interlobe, radial and high pressure end respectively. Furthermore it is reported as well that running clearances are expected to vary 40 to 100 microns from this values [18].

In this work, experimental measurements of 6000 and 8000 RPM speeds of this compressor are compared with the CFD calculations.

### 4. Simulation Setup

Figure 4 summarizes the Boundary Conditions (BC) and turbulence settings. Inlet temperature and total pressure are set to 300 K and 1 bar, respectively. Free opening to 2 bars at 129.45
Figure 3. Design model and initial CAD geometry used in simulations.

Figure 4. Overview of CFD setup.

Inlet:
- Total pressure = 0
- Temperature = 26.85 (absolute temperature = 300K)
- Turbulent intensity (pulsations) = 0.03
- Turbulent scale = 0.01*R_{male_rotor}

Free outlet:
- Pressure = 100000
- Temperature = 129.45
- Turbulent intensity (pulsations) = 0.03
- Turbulent scale = 0.01*R_{male_rotor}

Housing walls:
- No slip,
- Equilibrium wall functions

Rotor walls:
- No slip,
- Equilibrium wall functions,
- Rotation as moving body
- Rotation speed = 6000, 8000 rpm

*Initial conditions are chosen as equal to Inlet conditions*

is set for outlet BC. Standard k-ε turbulence model [14] is used with intensity of 0.03 and scale corresponding to 1% of male rotor radius. For all walls equilibrium wall function settings are used [14].

As one can observe, an inlet duct is placed in order to have a settled inflow to the compressor. This duct is created by extruding the original inlet cross-section.

As mentioned above, the actual operating clearances may differ from the prescribed values. In this work, instead of Conjugate Heat Transfer (CHT) calculations, a primer analysis is conducted at 6000 RPM using two different clearance values i.e. 60 and 90 µm. In order to obtain different clearance values, the rotors are scaled in the CFD software, without using a CAD tool. From these, the one which correlates better with the experimental volumetric flow rate, outlet
temperature and male rotor torque was chosen for further analysis and analysis of 8000 RPM.

Figure 5 shows the initial comparison of experimental values against calculation with different clearance values. Accordingly, overall match is observed with 90 µm and further analysis is performed with this clearance setting. This is expected as smaller clearance would result in higher flow rates and vice versa.

Figure 5. 6000 RPM, volumetric flow rate, outlet temperature and male torque compared with CFD for two different clearance values: 60 and 90 µm.

As mentioned in the Approach section, the embedded automatic grid generator of FlowVision, based on SGGR™ is employed for grid generation. The overview of the grid used during the calculations is given in Fig. 7. As observed, a slight increase in density is applied from the inlet towards the main body. On top of this, only surface adaptations are applied up to 3 levels (each level corresponding to division of a cell to 8 equal cells). While in some problematic cases, adaptation to solution or solution gradients, and adaptation to arbitrary geometrical shapes in spaces of interest can be used; in the current case this is not seen as a necessary action. Figure 6 shows the sensitivity of volumetric flow rate and outlet temperature to the adaptation levels. According to this, two level of adaptation is seen sufficient for calculations which corresponds to the minimum cell size of 3 mm (cubic elements in the main body region) and total cell count of 1.25 million. It is very important to highlight that as the computational grid adaptations (surface, solution or volume driven adaptations) may take place or disappear during the simulation, the usual methodology to solve problems such as the current one is first starting with low level adaptations and as the full solution is being approached switching to higher level adaptations. For challenging cases, the inlet or outlet boundary conditions can be set as time dependent; however, this was not necessary for the current simulation task.

At every time step dictated by surface CFL number, the rotation will take place and the adaptations will be regenerated. In the current calculation Merge Cells option is kept active meaning that if the object to which an adaptation is assigned is displaced, cells remaining far from this surface are merged.

The time stepping was driven by (Courant-Friedrichs-Lewy) CFL law for all cases. In the software used for this task, three different CFL numbers are given for convective, surface and diffusion transport. In the beginning of the calculations, CFL numbers as high as 15 were chosen. However, last half rotation (apprx.) was run with CFL=1.

5. Results and Discussion
An overview of cycle-averaged volumetric flow rate, male torque and outlet temperature as a function of male rotor speed is given in Fig. 8. As it can be seen from this, a good correspondence of experimental and simulation data exists. Experimental results for outlet temperature show a
Figure 6. Plots used to determine optimum grid for the calculations.

Figure 7. Overview of the computational grid with level 2 surface adaptation on rotor and housing walls.
thought provoking scheme, while simulation shows strong dependence of the outlet temperature on the operation speed.

Figure 8. Cycle-averaged volumetric flow rate, male torque and outlet temperature as a function of operation speed. A comparison between CFD and experiments.

Furthermore, time histories of torque, inlet/outlet mass flow and outlet temperature is shown in Figs. 9 left and right respectively. Accordingly, the mean values of mass flow rate for 6000 RPM is 0.1445 and 0.1453 kg/sec for inlet and outlet. These values are 0.2139 and 0.2143 kg/sec for 8000 RPM. Cycle-averaged torque values are 24.1832 (measurement 25.397 Nm) and 26.9994 Nm (measurement 28.204 Nm) for 6000 RPM and 8000 RPM respectively. Based on these, the overall comparison with the experiments show less than 6.3%. This maximum error which is obtained for volumetric flow rate of 6000 RPM, is in the reasonable range when the use of Gap Model™ is considered. For the outlet temperature history representation, mass flow averaged values are considered instead of area averaged, since this is more meaningful.

Figure 9. Time histories of inlet and outlet mass flow rates, and outlet temperature. Outlet temperature is mass flow averaged.

Finally, the pressure-alpha diagrams are presented in Fig. 10 for both operating conditions. These pressure trace curves are well in agreement with the measurements. The peak is well captured for both speeds; however, the higher frequency oscillations observed in the measurements are not captured by CFD. This is similar to the other similar CFD studies performed [19, 20].

5.1. Computational and Processing Time
All cases in this study are solved on a setup of Intel Xeon X5570 2.93GHz 4 cores 8 threads. For the initial stages of the calculation a low level adaptation (1) was used. Average time for
one revolution for this setting was 1.7 hour with 4 processors. After 4 full rotations, level 2 adaptation was applied and the solution is continued. Average time for one revolution was 10.0 hours with 4 processors for this setting. The second setting was kept for 1.5 rotations, giving a total time of 21.8 hours with 4 processors. The second part (adaptation level 2) of the 8000 RPM case was also run on a 32 processor setup to see the speed-up. The calculation time is observed to reduces to 11.27 hours in total.

Apart from the computational time, the man-hour dedicated to CFD pre-and post processing is important and expensive. It is not correct to assess the pre/post-processing time as it might depend on the user. However, it is beneficial to mention ranges to highlight the developments on automatic grid generators or how existing tools are applied to positive displacement machines. In the current case, from importing CAD files to the software until the start of calculation; the time spend is from 40 to 60 minutes for an average CFD engineer, depending on the experience.

6. Conclusions
A reduced order method called Gap Model\textsuperscript{TM} within FlowVision CFD software package is applied to the ICCS2015 test case. This model allows for approximating the leakage flows in an efficient manner without resolving them.

The grid generation is an important step in CFD of positive displacement machines. This is performed based on the automatic Cartesian grid generator with adaptations. Only ‘to surface’ adaptations are seen necessary in the current case.

In order to find the appropriate running clearances, two different values are tried and 90 µm is found to be appropriate for all of the interlobe, radial and high pressure clearances. An alternative to this approach would be performing CHT calculations; however this was not done for the current case.

Two running speeds; 6000 and 8000 RPM for male rotor are examined closely. Overall, the CFD calculations agree well with the experimental measurements. The maximum error in total properties was 6.3% which was of volumetric flow rate of 6000 RPM.

The proposed approach is suitable for the cases where the engineer is to calculate many possible configurations for optimization (or coupled automatized optimization) purposes where it is usually necessary to observe fluctuating values (torque, pressure etc.). As analytical expressions are used for leakage flows in Gap Model\textsuperscript{TM}, and Poiseuille flow profile is postulated (see user manual [14]) the resolution of boundary layer, shock capturing etc will not be possible. In such cases, the current approach is combined with workarounds such as Overlapping Boundary

Figure 10. Comparison of pressure-alpha diagrams with experimental results for 6000 RPM (left) and 8000 RPM (right).
Layer (OBL) or similar approaches.

The calculation times recorded for the current case, using 4 processors of Intel Xeon X5570 (2.93GHz, 4 cores - 8 threads) was 21.8 hours. When the case is partially solved on 32 processors the calculation time reduces to 11.27 hours.

Acknowledgements
As this study is performed based on ICCS2015 Test Case from Centre for Compressor Technology, authors would like to express our thanks to Dr. Ahmed Kovacevic and his team for providing the test case details and results and further guidance.
References

[1] Nikola Stosic, Ian Smith, and Ahmed Kovacevic. *Screw compressors: mathematical modelling and performance calculation*. Springer Science & Business Media, 2005.

[2] Nikola Stosic, Ian K Smith, and Ahmed Kovacevic. “Optimisation of screw compressors”. In: *Applied Thermal Engineering* 23.10 (2003), pp. 1177–1195.

[3] PA Andreev. “Vintovie kompressornie mashinii (Screw Compression Machines)”. In: *SUDPROM Leningrad* (1961).

[4] SE Edstroem. “A modern way to good screw compressors”. In: *International Compressor Engineering Conference Purdue*. Vol. 18. 1992.

[5] A Kh Dzhanakhmedov. “Investigation of the destructive effect of two-phase liquid on screw pump lip seals”. In: *Journal of Friction and Wear* 29.4 (2008), pp. 310–313.

[6] K Kauder and B Rau. “Auslegungsverfahren fur Schraubenkompressoren”. In: *VDI Berichte-Verein Deutscher Ingenieure* 1135 (1994), pp. 31–44.

[7] N Stosic and K Hanjalic. “Development and optimization of screw machines with a simulation model—Part I: Profile generation”. In: *Journal of fluids engineering* 119.3 (1997), pp. 659–663.

[8] N Stosić. “On gearing of helical screw compressor rotors”. In: *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* 212.7 (1998), pp. 587–594.

[9] Nikola Stosic et al. “Review of mathematical models in performance calculation of screw compressors”. In: *International Journal of Fluid Machinery and Systems* 4.2 (2011), pp. 271–288.

[10] Tian-hong YAN, Jia-lin LIANG, and Qing LI. “The Current Situation, Development and Review of New Technology for Compressors”. In: *Compressor Technology* 1 (2011), p. 018.

[11] Ahmed Kovacevic, Nikola Stosic, and Ian K Smith. “CFD analysis of screw compressor performance”. In: *Advances of CFD in Fluid Machinery Design* edited by Elder, RL, Tourlidakis, A, Yates MK, Professional Engineering Publishing of ImechE, London (2002).

[12] Matthias Möller and Jochen Hinz. “Isogeometric analysis framework for the numerical simulation of rotary screw machines. I. General concept and early applications”. In: *IOP Conference Series: Materials Science and Engineering*. Vol. 425-1. IOP Publishing. 2018, p. 12032.

[13] Belotserkovsky. *Numerical Methods in Continuum Mechanics 2nd Edition*. Moscow: Fizmatlit, 1994.

[14] Capvidia NV. *FlowVision 3.10.01 User’s Manual*. Capvidia NV. Leuven, Belgium, 2018.

[15] Andrey Aksenov, Alexander Dyadkin, Aleksey Gudzovsky, et al. “Numerical simulation of car tire aquaplaning”. In: *Computational Fluid Dynamics* 96 (1996), pp. 815–820.
[16] Andrey Aksenov, Alexander Dyadkin, and Victor Pokhilko. “Overcoming of barrier between CAD and CFD by modified finite volume method”. In: ASME-PUBLICATIONS-PVP 377 (1998), pp. 79–83.

[17] A Aksenov et al. “Oil Leakage Through a Valve Stem Seal”. In: Proc. Abaqus User Conference. 2005, p. 2005.

[18] Sham Rane. Personal communication, Test Case for 2nd Short Course on CFD in Rotary Positive Displacement Machines. 2015.

[19] Ahmed Kovacevic et al. “Influence of approaches in CFD solvers on performance prediction in screw compressors”. In: 2014.

[20] Hui Shen et al. “Investigations of Automatic Meshing in Modeling a Dry Twin Screw Compressor”. In: 2018.