Numerical evaluation of an Oil free Screw Compressor's Suction Port design using Ansys CFX and SCORG

S Tankhiwale1*, A Dagwar1, S Rane2, A Kovacevic2, A Birari1, S Abdan1,2 and N Asati1

1Kirloskar Pneumatic Company Limited, Pune, India
2Centre for Compressor Technology, City, University of London, London, U.K.

E-mail: shubhankar.tankhiwale@kirloskar.com

*Corresponding author

Abstract. Quasi one-dimensional chamber models and experimental methods are commonly adopted by researchers to predict and validate the performance of the oil free screw compressors. However, to understand the scope for improvement in performance as impacted by compressor ports, one needs to know the details of physical processes happening in compression chamber. This can be done by understanding flow field such as internal temperature and pressure with the help of CFD model. With this CFD setup, comparison of specific power consumption of oil free screw compressor between the axial and radial suction flange is presented in this paper. A hexahedral numerical mesh is used to represent flow field which is generated by SCORG™. Flow solver ANSYS CFX has been used for study with these single domain deforming rotor grids. This paper presents the methods used for setting up the CFD model and analysis of specific power consumption for two different suction port geometries, an axial and the radial suction flange. At the design operating condition, although axial suction flange seems to be more favourable in terms of lower suction pressure drops at high operating speed, the radial suction flange when simulated using CFD model predicts almost the similar performance. However, radial suction flange gives an added advantage for ready adaptability with the suction elements like the suction filter. The results of the simulation show that the radial suction flange can be adopted for better compatibility with the oil free package and the CFD model used for the analysis can be used for optimization of the air-end at various operating conditions.

Keywords: CFD, Suction port, CFX, SCORG, Oil free screw compressor

1. Introduction.
The flow analysis of Positive displacement (PD) machines is highly complex mainly due to 3D flow structure involving turbulence, complexity in mesh due to micron size gap and unsteadiness. In recent years, a growing availability of computational resources and progress in the accuracy of numerical methods brought PD machines Computational Fluid Dynamics (CFD) methods from pure research...
work into the competitive industrial markets. Computational Fluid Dynamics (CFD) can provide a useful means of modeling the oil free machines and study the physical phenomenon in detail in order to understand the complete behavior of oil free twin screw compressors.

The numerical computation of screw machines requires accurate grid generation because the fluid flow is transient and depends on the rotor position. Kovačević [4] [5] made a breakthrough to successfully use an algebraic grid generation method with boundary adaptation and transfinite interpolation which has been implemented in the program called SCORG. This aims to generate high quality grids in a short time as a result of which CFD can now be used very frequently in industrial applications of PD machines.

An oil-free twin screw compressor the working chamber is free from oil. This is opposite to the oil injected machines, where the oil is used for lubrication, temperature control and reduced internal leakages in the working chamber. Oil-free machines are typically used in industries where the working fluid must be free of any containment. Eliminating oil from the rotor chamber adds in own complexities and challenges, such as sealing arrangements, increased operating temperatures etc. Additionally, as oil is absent from the working chamber, the induced efficiency losses that occur due to oil shear are avoided, meaning the oil-free screw compressor can operate at significantly higher tip speeds.

The suction port is a part of the suction element which influences the compressor's performance. The use of CFD allows the analysis of the flow inside the suction port which is impossible to investigate experimentally but it helps influencing the performance of a screw compressor. The Suction port's design optimization is quite difficult to take place until and unless the exact boundary conditions are known. Hence incorporating the gap between rotors and its casing in the simulation is must to improve the suction port design. VandeVoorde et al. (2005) [2] used an algorithm for generating block structured mesh from the solution of the Laplace equation for twin screw compressors and pumps using differential methods. Reports on analysis of oil free air screw compressors and twin screw expanders with real gas models are available in literature using these techniques (Paper et al., 2013) [1]. Recently, Arjeneh et al. (2014) [3] have presented the analysis of flow through the suction port of a screw compressor with water injection. With both deforming rotor domains and multiphase models it was reported to be difficult to stabilize the solver in a full 3D analysis. Although several attempts have been made in the recent past to extend the CFD technology to oil injected compressors, it has proven to be difficult to achieve the desired grid structure and the modelling conditions that can provide stability to the numerical solvers. An extensive study undertaken by Rane [6] [8] presents a high level of accuracy between CFD modeling and test data is achievable when using a single domain rotor grid. An attempt has been made in the presented work to utilize the latest developments in CFD technology for design optimization of suction port of an oil free twin screw compressor.

2. Approach for modelling suction ports of the oil free screw compressors

2.1 Modelling Methodologies

The performance attributes of the machine can be determined numerically, through CFD modelling or the uniquely designed thermodynamic chamber model tools - such as that used in SCORGTM [1]. There are extreme differences in modelling the 1D and 3D cases. The greatest is the processing time and power required to obtain the results using the specified boundary conditions. Where the chamber model can output predictions in seconds, the CFD model would take days, ultimately due to complex and quantity of equations required to be calculated. Additionally, to perform these calculations, a large amount of high specification processing and enough number of cores is required to minimize the computational time, whereas the chamber model can operate on a single core. Further
to this, the CFD model will provide an extensive range of results, including flow phenomena that can be numerically and visually investigated, as opposite to the chamber model which outputs single calculated values relating to performance. This is not to say a particular model is superior, simply their use should be considered on a case to case basis.

2.2 Thermodynamic Chamber Model (SCORG™)

One-dimensional mathematical modelling of screw machines has been used to great effect for many years. An accurate prediction of many aspects of machine performance can be obtained from the models which simultaneously solve thermodynamic processes, based on ideal or real gas laws, with the geometric parameters of the machine, including chamber volume, port areas and sealing line lengths.

From the many modelling methodologies, that presented by Stosic et al [7] forms the foundation of the model used in this study. Described as multi-chamber thermodynamic model, the screw machine is divided into sub-domains representing each stage of the compression process. The challenge of computing time dependent flow variables within the control volumes is overcome by using a combination of differential equations to solve the rate of change of mass and internal energy, and algebraic equation to solve other internal flow behaviour such as velocity and leakages. From a user perspective, the complexities involved in the computation of the abovementioned flow variables are hidden. As much as it is recommended to have a theoretical understanding of the model, developers have excelled in creating a tool that is as much user friendly as it is reliable. This study will utilize the thermodynamic model inherent to SCORG™.

2.3 Rotor grid generation

SCORG™ is used to generate the 3D rotor fluid domain. The tool allows the user to define certain mesh parameters to calculate a suitable mesh. For that applying the principles of analytical grid generation through transfinite interpolation with adaptive meshing, the authors have derived a general, fast and reliable algorithm for automatic numerical mapping of arbitrary twin screw machine geometries. The procedure of analytical grid generation of screw machine working domain is explained in Kovačević (2002) [7]. In order to achieve a conformal single domain mesh, a new approach of background blocking has been presented in Rane (2015) [6]. In this procedure, outer boundary in each background block is defined as a combination of the rack segment and the casing circle segment. The rack segment stretches between the lower and higher cusp points and is closed by the casing as shown in Figure 1. The distribution obtained on the outer boundaries of the two blocks is used as the reference for the rotor profile distribution. Since the blocks of the main and gate rotors are obtained separately, the intersection points obtained on the common rack curve from the two blocks can be different. If different the resulting mesh has a non-conformal map between the two rotor blocks. Using the blocking approach it is possible to transform this boundary map into a conformal boundary map. The surface mesh on the interlobe surface mostly follows the axial grid lines with small transverse movements in the vicinity of the cusp points. The movements of points are along the surface of the interface and do not cause any irregularity of cells. This implementation allows for a fully conformal interface with the equal index around cusp points to ensure accurate capturing is the blow hole leakage area. The blocking approach is in more detail explained in Rane (2015) [6]. It allows both conformal and non-conformal boundary map to produce fully hexahedral 3D grid with both, the main and the gate rotor surfaces smoothly captured. The surface mesh on the casing is of the highest quality with the regular quadrilateral cells.
Table 1 Rotor grids parameters

| Parameter | Description              | Input |
|-----------|--------------------------|-------|
| A         | Radial Divisions         | 7     |
| B         | Angular Divisions        | 40    |
| C         | Interlobe Divisions      | 50    |
| D         | Circumferential Divisions| 40    |
| E         | Distribution type        | Casing to rotor conformal |

Figure 1 3D Grid parameters

With above setting shown in Table 1, the produced mesh indicates a good overall quality rotor mesh, with <1% of the cells in the 3D mesh not meeting the recommended Orthogonality and expansion factor quality criteria as shown in Table 2. Minimum acceptable orthogonality angle is 20º. Whereas an aspect ratio for 3D mesh is 100% in good category of the quality criteria. CFX uses a Fortran interface called ‘junction box routine’ to exchange external meshes with the solver.

2.4 Stationary domain mesh
Figure 2 and Figure 3 shows the CFD model flow domain in which the mesh lines are highlighted with rendering. Table 2 summarizes the statistics of the 3D domain. All stationary domains have a combination of tetrahedral and hexahedral cell structure. Figure 3 shows an extended geometry called adapter which is connected with the axial suction flange by fluid-fluid interfaces. The rotor domain itself is a deforming grid, generated using the procedure described in Section 2.3.
Table 2 Mesh statistics

| Domain             | Cell structure | Node count | Cell count | Orthogonality angle (Min) | Expansion factor | Aspect ratio |
|--------------------|----------------|------------|------------|---------------------------|------------------|--------------|
| Rotor              | Hex            | 482636     | 426258     | 8.2                       | 586              | 389          |
| Suction Port Radial| Tetra+Hex      | 277464     | 931693     | 28                        | 388              | 7.17         |
| Suction Port Axial | Tetra+Hex      | 165342     | 504516     | 34.2                      | 365              | 10.55        |
| Adapter            | Tetra+Hex      | 43742      | 190007     | 52.4                      | 8                | 5.48         |
| Discharge Port     | Tetra+Hex      | 53337      | 156042     | 20.3                      | 54               | 11.54        |

2.5 CFD Model

CFD modeling is rapidly being adopted in all areas of modern engineering practice to solve complex flow and heat transfer problems. However, the challenges involved in modeling and computing the complex physics and transient nature inherent to the machine have proven difficult to overcome. Nonetheless, through extensive and committed research, the development of design tools and users obtaining a greater understanding of the problems, it is now possible to undertake these simulations with a certain degree of confidence that a valid solution can be obtained. To obtain this, CFX as a solver is used for this study. CFX is vertex-centred solver for which the spatial domain is firstly discretised into a mesh by using an external grid generator like SCORG™. This mesh is then used to construct virtual control volumes within the solver.

There are two CFX case setups, one is for axial suction port and other one is radial suction port. In Axial suction flange the adapter is an additional component required to connect vertically with suction filter. Whereas in radial type suction port the suction filter will be connected directly on suction flange. The male rotor has three lobes and the female rotor has five lobes with ‘N’ profiles. The built-in volume index $V_i$ of the compressor is 2.5. The wrap angle is 306°. Main and gate rotors has 3 and 5 lobes respectively. Operating speed of the machine is 18918.4 rpm.

2.5.1 CFD boundary conditions

In both the cases, air as a single phase is defined as an ideal gas state with molar mass of 28.96 kg/kmol, specific heat capacity 1.0044e03 J/kg K, dynamic viscosity 1.831e-05 kg/m s and thermal conductivity 2.61e-02 W/m K. The boundary condition is summarized in Table 3.

Table 3 Operating boundary condition for both CFD models.

| Speed (rpm) | Working Fluid | Suction Pressure (bar A) | Suction Temperature (K) | Discharge Pressure (bar A) | Discharge Temperature (K) |
|-------------|---------------|--------------------------|-------------------------|----------------------------|--------------------------|
| 18918.4     | Air           | 1                        | 298.15                  | 3                          | 393                      |

2.5.2 Solver parameters

Numerical setup for CFX is shown in Table 4. The Shear Stress Transport k-omega model is used in both the cases. k-epsilon model is used for away from wall treatment or free stream treatment whereas k-omega is used for near wall treatment of the case. The SST model compiles the best elements from both k-epsilon and k-omega models for predicting the best flow behaviour. CFX requires less number of iterations to approach the converged solution.

Table 4 Numerical setup for CFX solver.

| Criteria                | CFX-controls        |
|-------------------------|---------------------|
| Turbulence Model        | SST k-omega         |
| Inlet Boundary Condition| Opening (specified pressure and temperature) |
### Outlet Boundary Condition
- Opening (specified pressure and temperature)

### Pressure Velocity Coupling
- Co-located layout

### Turbulence Scheme
- First order upwind

### Transient Scheme
- Second order backward Euler

### Transient Inner loop coefficient
- 10 iterations per time steps

### Convergence Criteria
- $1e^{-4}$

### Relaxation Parameters
- 0.1

#### 3. Results

The results present the performance calculations, applying the boundary conditions listed above, from both the models. From here the variance are calculated and a clear comparison can be drawn.

![Figure 4](image4.png)

Figure 4 Fluid volume of a) Radial suction port and b) Axial suction port with adapter.

Results are mainly focused on two designs of the suction port shown in Figure 4. In case of radial suction port, air will flow in curvilinear passage from radial inlet to suction to rotor interface. Whereas in axial suction port design, air will flow in the same manner as radial design but here it has more area to travel between the same two surfaces.

#### 3.1 Pressure distribution.

A pressure contour is plotted at $X = 0.04$ m in YZ plane for male rotor speed at 18918.4 rpm on the flow domain shown in Figure 4 to observe the pressure distribution in both the cases. Figure 5 shows the similar pressure characteristics at the cross section. Comparatively lower suction pressure drop is observed on axial type suction port.

![Figure 5](image5.png)

Figure 5 Pressure contour plot a) Axial Suction port with adapter and b) Radial Suction Port at time step = 820 for the operating speed of 18918.4 rpm
Figure 6 shows the suction port pressure drop against male rotor rotation angle for axial and radial type suction ports. It is clear from the graph that the pressure drop for axial suction port is lower than radial type suction port design. This is due to an additional equipment called adapter is attached to the axial suction port. Adapter reduces the pressure drop due to its larger area. Whereas in case of radial suction port the pressure drop is more due to smaller area is available for air to pass from suction flange to the rotor domain as shown in Figure 4. As per standard formula of minor loss [9], the estimated loss coefficient for radial port is 28% higher than that of axial port.

![Graph of Suction port pressure drop against male rotor rotation angle for axial and radial suction port design](image)

**Figure 6** Graph of Suction port pressure drop against male rotor rotation angle for axial and radial suction port design

### 3.2 Specific Power

Specific power is nothing but the Free air (m³/min) delivered per kW of power. Overall performance results from both the simulations are averaged for each cycle to achieve performance values of specific power. Percentage difference in the specific power for both the designs is as low as below 1%.

### 3.3 Velocity distribution

Velocity vectors are plotted on the YZ plane of the radial and axial suction ports as shown in Figure 7. Higher velocities are observed at the end of both the suction ports. More recirculation can be observed towards suction to rotor interface in case of axial port which leads to the loss of free air delivery.

![Velocity vectors at YZ plane for a) Radial suction port and b) Axial suction port](image)

**Figure 7** Velocity vectors at YZ plane for a) Radial suction port and b) Axial suction port
Figure 8 shows velocity contours on the surface of the suction port. Higher velocities are observed on the suction to rotor interface.

![Velocity contour plot](image)

Figure 8 Velocity contour plot a) Radial Suction port and b) Axial Suction Port at time step = 820

4. Conclusion

This paper presents the numerical evaluation of the suction port of an oil free twin screw compressors at design operating condition. A CFD modelling approach has been used for the calculation of performance and pressure drop across the suction port. A single domain structured numerical mesh of the flow domain was generated by SCORG™ using recently developed boundary blocking and differential grid generation procedure. CFX case with Axial as well as Radial suction port design is modelled, solved and studied in this paper and comparison is made between both the cases to conclude the results.

- Cycle averaged specific power is very small between the two designs
- Despite of having a little high pressure drop at radial suction port, it can be used for its better compatibility with oil free package due to the size difference observed in Figure 2 and Figure 3.

The detailed performance characteristics have been studied with varying operating speeds of the screw compressor for further optimization of the radial suction port.

5. References

[1] Papes, I., Degroote, J. and Vierendeels, J. 2013 3D CFD analysis of an oil injected twin screw expander. Int. Mechanical Engineering Congress and Exposition, ASME IMECE at San Diego, USA.

[2] Vierendeels, Voorde Vande J. 2005 A grid manipulation algorithm for ALE calculations in screw compressors. Canada : AIAA 2005-4701, 2005. 17th AIAA Computational Fluid Dynamics Conf.

[3] Arjeneh M., Kovačević A., Gavaises M. and Rane S. 2014. Study of Multiphase Flow at the Suction of Screw Compressor at Purdue paper 1353.
[4] Kovačević, A. 2005, Boundary Adaptation in Grid Generation for CFD Analysis of Screw Compressors. Int. journal for numerical methods in engineering.

[5] Kovačević, A. 2002. Three-Dimensional Numerical Analysis for Flow Prediction in Positive Displacement Screw Machines, Thesis. City University London.

[6] Rane S. 2015 Grid Generation and CFD analysis of Variable Geometry Screw Machines. City university London

[7] Stošić N., Smith I.K. and Kovacevic A 2005 Screw Compressors: Mathematical Modelling and Performance Calculation Berlin ISBN: 3-540-24275-9.

[8] N Basha, A Kovacevic and S Rane. 2019 User defined nodal displacement of numerical mesh for analysis of screw machines in FLUENT IOP Publishing Ltd Int. Conf. on Compressors and their Systems.

[9] Rajput, R.K. (2013). *A Text book of fluid mechanics and hydraulic machine in SI Unit*. New Delhi, S. Chand and company limited.