CFD analysis on the effect of discharge port geometry of the hook and claw vacuum pumps

K Theofanidis\textsuperscript{1}, Y Lu\textsuperscript{2} and A Kovacevic\textsuperscript{3}
\textsuperscript{1,2,3}Center for compressor technology, City University London, EC1V 0HB, UK

E-mail:\textsuperscript{1} kyriakos.theofanidis24@gmail.com, kyriakos.theofanidis@city.ac.uk

Abstract. Due to the sharp discontinuities in the profile of hook and claw pumps which make it difficult to produce a good quality body fitted mesh, their CFD simulations are challenging. However, the cut-cell Cartesian meshing method is capable to handle the very complex moving geometries and is easy to setup. However it may lead to conservativeness issues. The aim of this paper is to investigate the influence of discharge port geometry of the hook and claw pump on its performance by using commercial software ANSYS Forte, which utilizes automatic mesh generation using the cut-cell Cartesian method. Firstly, five discharge ports with different geometries were designed. Secondly, the pumps with these different port configurations were set up and ran in the ANSYS Forte. Finally, the power and volume flow rate were compared. It was noticed that the geometry with the extended discharge port tip and the preliminary discharge port opening result in performance and reliability improvements. The simulation results can be used to further optimize the discharge port during the design of these machines.

1 Introduction
The procedure of analyzing claw vacuum pumps involves rotor profile design, using various mathematical principles and coding, CAD design and CFD simulations. Since the use of vacuum pumps is increasing rapidly over the years, properly executed analysis and innovation will be important for the evolution of them. They consist of two rotating rotors and a casing. The rotating rotors trap and move the air that is coming from the inlet towards the outlet discharge gate. Previous researches have gone very deep into the understanding of their working principle using many software packages such as SCORG, ANSYS Fluent, Star CCM+ and others. For this case another leading CFD package was used: ANSYS Forte. It is utilizing the cut-cell Cartesian mesh generation method which is not used a lot to investigate rotating machines. Thus, the accuracy and stability of this method was examined. The main component of study is the port designs, which is a not much previously researched area. The geometries of the inlet and discharge ports play a significant role in the operation of the pump, and no major investigation showcased that until now. In order to conduct this analysis, three different discharge port configuration models have been modeled and compared in terms of performance, efficiency and reliability.

![Figure 1: Claw pump model for CFD analysis](image-url)
2 Background

2.1 Vacuum pumps overview

Vacuum pumps and compressor systems are rotary positive displacement machines that include only a few rotating and moving parts. Kovacevic et al. (2006) [1] presented the operation of positive displacement rotary machines and their principles of operation while it is one of the few books that facilitate an extensive study in CFD analysis. The pair of the moving parts are the rotors, that mesh with each other utilizing their helical groove design. Some pumps, like the one that is being investigated, do not have grooves and the rotors are pretty short and straight, while still being able to pump a significant amount of gas. These pumps are known as claw pumps. This is achieved due to the meshing geometry of them, which utilizes the mathematical approached envelope method. The two rotors are firmly surrounded by a casing [2]. In between the rotors and the casing, a radial clearance exists, in order to ensure smooth operation and minimal wear. The vacuum pumps have a working chamber that the meshing rotors form by rotating and they move it while it is filled with air from the inlet towards the outlet discharge port. The two rotors are synchronized by timing gears which are also transmitting the motor’s power to move them [3][4][5]. Besides the inlet, the compression, and discharge processes, claw vacuum pumps form a special kind of process which is called the mixing process. The mixing process, is observed at the carry-over chamber which is formed in between the two rotors. The chamber accommodates gas after the discharge process and gas from the inlet process (Figure 1).

Stosic et al. (2005) [2] presented the main advantages of screw machines. Screw machines only consist of rotating parts and not translating ones and thus they can run on higher speeds. The contact forces are low and therefore wear is minimized. Also, the pressure inside rises and this results in less gas leakage. Furthermore, Tuo et al. (2018) [6] and Wang et al. (2015) [7] showcased some more advantages including: high reliability, stable medium conveying, small vibration, oil-free usage, and compact structure.

Regarding the disadvantages, Lu et al. (2017) [8], presenting the working process in the twin screw vacuum pumps, denoted the most important one of them: Very high rotational speeds can significantly decrease the discharge pressure whereas low speeds can lead to over consumption. Thus, speed affects the performance dramatically and may lead to failures. Except that, Kovacevic et al. (2007) [9] mentioned that cavitation and erosion at the rotor shafts and gaps also lead to damage.

2.2 Design analysis

The main focus in the design process of a claw vacuum pump is the generation of the rotors. The shape of the rotors defines the main performance characteristics of the pump which are related to the displacement and the leakage flows. The rotor design is achieved usually within a coding software like MATLAB or FORTRAN with the use of derived geometrical equations that define the curve plot of each rotor. Stosic et al. (2018) [10] provided a clear mathematical approach on rotor generation which used the envelope method to acquire the meshing points and meshing conditions of the rotors. Later on, a set of coordinates and their derivatives were obtained generating a variety of primary curves. The rest of the curves were calculated automatically using numerical series. For the optimization of the rotors, an investigation of each rotor feature must be conducted separately. Many studies have focused in the analysis of different aspects of the rotors with impressing results. For example, Yan et al. (2017) [11] noted that the use of involute-cycloid profiles leads to increased efficiency that results in larger mass flow rate while also larger mass flow pulsation. Also, Lu et al. (2015) [12] added that by combining a large flow cross section area with a small sealing line and a small blow hole area will lead in higher efficiency. Lastly, Stosic et al. (2011) [13] highlighted that despite the level that rotor generation has reached, there is always more space for improvement in making lighter and more efficient rotors. Moving on from the rotors, other characteristics regarding the casing also affect the performance of the pump. These include: The port design, the axis distance, the axial clearance and the radial and interlobe gaps.
2.3 **Computational fluid dynamics (CFD)**

Computational fluid dynamics (CFD) is a way of applying numerical methods to analyze the heat transfer and fluid dynamics governing equations in order to acquire fluid flow problem solutions. In order for this to be achieved, modern computers require algebraic equations, that have been discretized from continuous differential ones, in order to calculate a solution. Under the hood of a CFD package like ANSYS Fluent and ANSYS Forte, there are a series of governing equations that tackle the procedure of calculations. These equations are basically the conservation equations which include: The continuity equation (1), the momentum conservation Navier-Stokes equation (2), the space (3) and energy (4) conservation equations.

\[
\rho \left( \frac{\partial \bar{u}}{\partial t} + \bar{u} \cdot \nabla \bar{u} \right) = -\nabla P + \nu \Delta \bar{u}
\]

\[
\rho \left( \frac{\partial \bar{u}}{\partial t} + \bar{u} \cdot \nabla \bar{u} \right) = -\nabla P + \nu \Delta \bar{u}
\]

\[
\frac{\partial}{\partial t} \int_{V} dV = \int_{S} \bar{V}_b dS
\]

\[
\rho \frac{DV}{Dt} = -\nabla q - p(\nabla \cdot V) + \rho \dot{Q}
\]

The above equations are used to form the 2 basic approaches for a study of a flow which are: The simulation and the modeling techniques. The simulation ones, include the direct numerical simulation (DNS) and the large Eddy simulation (LES). DNS and LES, have very specific applications and their computational cost is very high. Thus, the industry has only started to use them the latest years, that the computational power has been increased, and only as a separate optional feature in CFD solvers. Nowadays, the most common approach in numerical flow studies is the modeling technique called Reynolds Averaged Navier Stokes (RANS) method. RANS is the oldest method of CFD analysis. It is very simple, it has a low computational demand and it is very efficient for turbulent flow analysis. ANSYS Forte, which is the solver used for this investigation also follows the RANS approach, aiming to simulate the averaged flow field. This approach utilizes the RANS equations themselves (5), (6) and the Reynolds stresses equation (7) presented below [14][15]:

\[
\frac{D\bar{u}}{Dt} + \frac{1}{\rho} \nabla \bar{p} - (\nabla \cdot \nu \nabla) \bar{u} = \nabla \cdot \tau
\]

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

\[
\tau_{i,j} = \rho (u_i u'_j) = \rho (u_i u_j) - \rho (u_i) (u_j)
\]

CFD provides a deep insight into the flow and thermodynamic behavior of positive displacement machines. As the computational technology advances, major steps forward are being made in the numerical analysis methods. Information like the mass flow rate, the rotors torque and power, the pressure distribution and velocity field can all be obtained from a CFD solver [9][11]. One of the most important aspects of a CFD analysis is the mesh quality. A good mesh will ensure accurate, efficient and robust results. A lot of CFD studies have been conducted focusing on the mesh generation and analysis. Rane et al. (2018) [16], mentioned that due to high magnitude of deformation of the domain and the geometrical complexity of the rotors, many commercial flow simulation packages are not suitable for screw machine analysis. ANSYS Forte has a capable model for positive displacement machine analysis and the integrated mesh
generation utility is useful and powerful for flows like that. The most common used CFD solvers, utilize the simple body-fitted structured mesh generation approach. A structured grid is defined as a mesh that its nodes have the same number of elements around them. It follows a specific shape pattern that is being repeated in all directions in the investigated geometry. Structured grids are most commonly seen in regular bodies with no complex geometries that allow the reproduction of the same grid (usually polygonal) shape. The rotors of claw pumps though, have very sharp and pointy orientations. This complexity of the shape, makes the meshing almost impossible when trying to use the body-fitted structured mesh approach [15]. This is where ANSYS Forte grid generation shines. ANSYS Forte uses a special kind of structured not body-fitted meshing technique called the cut-cell Cartesian approach. The cut-cell Cartesian method, as its name suggests, divides the examined area using a normal Cartesian grid which adapts to the geometry. A valuable benefit of it is the excellent near wall area treatment resulting in high accuracy.

Despite all the above characteristics, the cut-cell Cartesian method has some disadvantages. The one with the most impact, is the fact that the mesh is not generated densely enough at the leakage areas and therefore additional models are required to cover this gap. This means that for the claw pump case, the clearances between the casing and the rotors need additional mesh models and this results in a more expensive computational cost and also it might lead to inaccurate results. This is why, for some cases this method is longer and more difficult to be conducted with meshing errors often appearing. Also, it is not fully conservative making it less popular for simpler designs[17][18].

2.4 Port design

The ports of a pump are very important elements affecting the performance. They are responsible for bringing gas in and out of the working chamber of the pump. Yan et al. (2016) [19] in the research of numerical modeling of twin-screw pumps mentioned that the port geometry has little effect on the mass flow rate but has significant effect on power. Also, it was noted that the port shape has possible effect on pressure distribution and shaft power. Other than that, no major investigations have been made for the port geometries of a pump. Regarding the inlet, a circular shape is most commonly used. In regards to the discharge though, many alternations of the shape exist. The geometry can differ in elements that affect the angle of the rotor rotation that the discharge process begins and the discharge volume of the carry-over chamber.

3 Rotor profile design

The cycloid is a widely used curve in the pump rotor profile design. The rotor profile used in this case study consists of straight lines, cycles, cycloids, polynomial curves, and corresponding involute curves as demonstrated in Figure 2 below. The main rotor is defined as the rotor connected with the motor. It is defined in the global coordinate system $X_1, Y_1, Z_1$ while the gate rotor is defined in the local coordinate system $X_2, Y_2, Z_2$. The first straight line curve $A_1B_1$, the second curve sector $B_1C_1$, the third cycloid curve $C_1D_1$ and the fifth curve $E_1F_1$ are defined in the global coordinate system while the fourth curve $D_2E_2$ is defined in the local coordinate system. The rest curves can be obtained using the theory of involute gears [20].
To demonstrate the function of every curve, the design parameters are shown in Table 1. The inner and outer circle radius are \( r_i \) and \( r_o \) individually. The axis distance is \( A_c = r_o + r_i \) and the pitch circle radius is \( r_p = (r_o + r_i)/2 \).

Table 1: Rotor parameters

| Parameter                | Value          | Parameter     | Value     |
|--------------------------|----------------|---------------|-----------|
| Rotor combination        | 1by1           | Rotor length  | 25 (mm)   |
| Axis distance (\( A_c \))| 45 (mm)        | Axial clearance| 0.08 (mm)|
| Outer circle diameter (\( r_o \)) | 65.42 (mm) | Radial gap  | 0.07 (mm) |
| Inner circle diameter (\( r_i \)) | 24.42 (mm) | Interlobe gap| 0.08 (mm) |

4 Port design

In order to do the discharge port comparison, a base claw pump model was designed. With this model as a starting point, 5 different port configurations (Figure 3) were created. Each had a different discharge port geometry. All the geometries follow the arcs and overall dimensions of the main rotor. The changes in the geometries were defined by two main aspects: The angle of discharge and the extended discharge port tip. By altering the geometry of the discharge port, the angle at which the port opens and starts to discharge changes. In this way, by rotating the main rotor at 350°, 355°, 360° and 370° and by following its lobe’s arc, 4 different configurations were created. The only difference in them is the length of the outer arc, which is the factor that defines the angle that the port starts opening. With this change, the volume ratio index \( V_i = \frac{V_{max}}{V_{discharge}} \) is also changing (Table 2). In regards to the extended discharge port tip, the case with the angle of discharge at 360° was modified so that the port’s tip is longer, enabling the carry-over chamber to be discharged in a higher rate. All the ports had as basis the port’s 1 design. In (Figure 3) the base port 1 is denoted as the yellow hatched area whereas the other ports are denoted as different colors. The port 1 geometry is shown in the foreground in all cases so the differences are clear. Lastly, the extended discharge tip of port 2 is shown with an arrow.

![Figure 2: Rotor Profiles](image-url)
Figure 3: Port configurations

Table 2: Port characteristics

| Port       | 1     | 2 (Extended tip) | 3     | 4     | 5     |
|------------|-------|------------------|-------|-------|-------|
| Discharge angle | 360°  | 360°             | 370°  | 350°  | 355°  |
| Vi (Volume ratio index) | 1.28  | 1.28             | 1.31  | 1.22  | 1.225 |

5 Simulations and results

5.1 Set-up and boundary conditions

ANSYS Forte is using the finite volume method (FVM), solving the governing algebraic equations. The boundary conditions and the set-up settings are listed in Table 3 below. The working process of the claw pump was assumed to be an adiabatic one, with the operational fluid being air.

Table 3: Simulation set-up

| Turbulence model      | SST-k Epsilon |
|-----------------------|---------------|
| Inlet boundary conditions | Inlet Pressure: 1 bar |
| Outlet boundary conditions | Outlet Pressure: 1.21 bar |
| Time step              | $10^{-5}$ sec |
5.2 Meshing

ANSYS Forte’s automatic mesh generation (AMG) does not require much input and processing. After defining the surfaces and their refinement methods, the only input required is the global mesh size. The global mesh size defines the resolution of the mesh, and therefore the detail of the results. A smaller mesh size, normally would give a more accurate result. But there is a disadvantage: Smaller mesh sizes require more computational time and power. This is quite an important factor, since time and power resources are limited. That is why, a mesh independence study was conducted, trying mesh sizes 0.2 cm, 0.3 cm and 0.4 cm on port 1. It was found that the 0.3 cm mesh is the most suitable, because it balances computational resources and accuracy, since the percentage difference of it comparing to the 0.2 cm mesh was only 1.96% (Table 4).

*The mesh size 0.4 cm was too coarse and therefore resulted in errors.

| Global Mesh size (cm) | Inlet vol. flow rate (L/min) |
|-----------------------|-------------------------------|
| 0.2                   | 396                           |
| 0.3                   | 388.2                         |
| 0.5                   | 8.6*                          |

% difference: 0.2 vs 0.3 1.96%

5.3 Results

The results were validated by comparing the mass flow rates of the work of Gu et al. (2020) [3]. In this investigation a similar but not identical pump was considered. Therefore some small differences are expected. Also, the main objective is the comparison of the port geometries and not the absolute values. For the above reasons, a 10% discrepancy is reasonable and accepted. The validation data suggest an average mass flow rate of $6.63 \cdot 10^{-3}$ (kg/s). As observed from Table 5 bellow the average % difference obtained is 9.17% which is within the acceptable range.

| $\dot{m}$ (avg) | Validation $\dot{m}$ | % difference from validation |
|-----------------|-----------------------|-----------------------------|
| $7.3 \cdot 10^{-3}$ | $6.63 \cdot 10^{-3}$ | 9.17%                       |

In Table 6 and Figure 4 below, the results of the simulations are presented. Using ANSYS CFX-Post, the pressure, temperature and port velocity contours of all cases were plotted.

| Port | Inlet Volume flow rate (L/min) | Power (W) | Power to vol. flow rate ratio (specific power) |
|------|--------------------------------|-----------|-----------------------------------------------|
| 1    | 388.2                          | 266       | 0.685                                         |
| 2    | 365.4                          | 270       | 0.739                                         |
| 3    | 374.4                          | 281       | 0.750                                         |
| 4    | 385.6                          | 263       | 0.682                                         |
| 5    | 386.4                          | 267       | 0.690                                         |
Figure 4: Pressure/Temperature/Port Velocity Contours

As it is observed from table 6, the case with the highest power is port 3. Despite that, the fourth case has the best performance since it has the lowest specific power. Port 3 is the least efficient with the highest specific power. Moving on to the graphs (Figure 4), the time step showcased above is located mid-way the discharge phase in order to showcase the effect of the port’s geometry. The following observations were made:
The pressure distribution graphs show that overall the five cases are similar. Despite that, the fourth case seems to have slightly lower pressure build up during the discharge phase, showing that the $10^\circ$ early opening of the port actually helps in the discharge process. In contrast to that, port 5 which has $10^\circ$ late opening angle, results in the highest pressure among all the cases. Furthermore, port configuration 2 has also relatively low pressure, showing that the extended tip of the discharge port helps in the discharge process and in the carry-over chamber off-loading. A high pressure build-up can cause reliability issues over time to the pump and therefore the lower the pressure, the better the pump will perform and thus the failure possibilities will be reduced.

The temperature distribution graphs also show almost identical results with a small but significant difference. In ports 1, 3, 4 and 5 in the spot where the two rotors are very close, the temperature rapidly increases between the clearance. This shows that the extended discharge tip geometry on the second port is working effectively and therefore off-loads the carry-over chamber more efficiently. This is really important because, the high temperatures may cause the rotors to expand over time, and due to the very small clearances, it may lead to a collision and therefore a failure of the pump. Also, a high pressure and temperature loaded carry-over chamber leads to more power consumption which is not desirable.

Finally, the velocity vectors of the five cases also show that port 4 is less loaded, with lower velocity. The maximum velocity that has been reached in port 4 is significantly lower than any other case. The reason behind that is the $10^\circ$ early opening of the discharge port, showcasing again the importance of this geometry feature. Higher velocity means higher kinetic energy and thus higher power consumption.

Summarizing the results, ports 1, 3 and 5 do not seem to present any positive development. In fact, port 3 has the highest specific power, and therefore the worst performance, while port’s 5 highest pressure and velocity make it the most unreliable of them all. The ports that have showed positive developments are port 4 and port 2. Port’s 4 early discharging, ensures the highest performance in combination with good reliability and efficiency due to the low pressure and exit velocity. From the other hand, the extended tip of discharge port 2, ensures great carry-over chamber off-loading, reducing the possibilities of failure, and therefore increasing the reliability of the corresponding claw pump.

6 Conclusion

Overall, the cut-cell Cartesian meshing method seemed to be really consistent throughout the simulations conducted. The automatic mesh generation of ANSYS Forte was very important since it was accurate and time-saving without many errors occurring. As far as the port geometry is concerned, it seems that it does not have a huge effect, but the small alterations that some features produce are very important. In detail, the $10^\circ$ early opening of port 4 is resulting in an overall improvement in the performance, the reliability and the efficiency. Also, the extended discharge tip of the port 2, had a significant effect on the discharging process. The carry-over chamber’s off-loading, is a major feature that maximizes the reliability of the claw pumps. Therefore port 2 might not be the best performing one by a small fraction, but is the most reliable ensuring very good carry-over chamber discharging. Generally, a detailed study and optimization of the discharge port geometry of claw pumps is a worthy procedure, since interesting and significant developments in the behavior of the pumps are observed.
References

[1] Kovacevic A, Smith I, Stosic N. Screw Compressors three dimensional computational fluid dynamics and solid fluid interaction. London: Springer-Verlag; 2006.

[2] Stosic N, Kovacevic A, Smith I. Screw compressor geometry mathematical modelling and performance calculation. Berlin: Springer-Verlag; 2005.

[3] Gu P, Xing L, Wang Y, Feng J, Peng X. Transient flow field and performance analysis of a claw pump for FCVs. International Journal of Hydrogen Energy. 2020;46(1).

[4] Kisu Im, P. (1997). Screw Vacuum Pump Having A Decreasing Pitch for the Screw Members.

[5] Lu Y, Kovacevic A, Read M, Basha N. Numerical Study of Customised Mesh for Twin Screw Vacuum Pumps. Designs. 2019;3(4):52.

[6] Tuo J, Guo B, Wu R, Zhang Y, Chen X. Study on the performance prediction of dry twin screw vacuum pump. IOP Conference Series: Materials Science and Engineering. 2018;425.

[7] Wang J, Jiang X, Cai Y. Investigation of a novel circular arc claw rotor profile for claw vacuum pumps and its performance analysis. Vacuum. 2015;111.

[8] Lu Y, Fu Y, Guo B, Fu L, Zhang Q, Chen X. Numerical Simulation of the Working Process in the Twin Screw Vacuum Pump. IOP Conference Series: Materials Science and Engineering. 2017;232.

[9] Kovacevic A, Stosic N, Mujic E, Smith I. CFD Integrated Design of Screw Compressors. Engineering Applications of Computational Fluid Mechanics. 2007;1(2).

[10] Stosic, N., Kovacevic, A., Rane, S. and Smith, I. Screw Rotor Profiles of Variable Lead Vacuum and Multiphase Machines and Their Calculation Models. 2018.

[11] Yan D, Tang Q, Kovacevic A, Rane S, Pei L. Rotor profile design and numerical analysis of 2–3 type multiphase twin-screw pumps. Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering. 2017;232(2).

[12] Lu Y, Guo B, Geng M. Study on Design of Rotor Profile for the Twin Screw Vacuum Pump with Single Thread Tooth. IOP Conference Series: Materials Science and Engineering. 2015;90.

[13] Stosic N, Smith I, Kovacevic A, Mujic E. Geometry of screw compressor rotors and their tools. Journal of Zhejiang University-SCIENCE A. 2011;12(4).

[14] ANSYS, Inc, (2020). Forte Quick Start Guide, Forte tutorials, Forte Theory manual.

[15] Zikanov O. Essential computational fluid dynamics. 1st ed. John Wiley Sons; 2010.

[16] Rane S, Kovačević A, Stošić N. Grid Generation for CFD Analysis and Design of a Variety of Twin Screw Machines. Designs. 2019;3(2).

[17] Kaczmarek, R., (2013). It’S All About The Mesh – Cartesian Cut-Cell Approach. [online] Convergecfld.com. Available at: <https://convergecfld.com/blog/its-all-about-the-mesh-cartesian-cut-cell-approach>

[18] PDM Analysis, (2020). Challenges In Grid Generation And 3D CFD Modelling Of Positive Displacement Machines. [video] Available at: <https://pdmalysis.co.uk/webinar-series/webinar-11/>

[19] Yan D, Kovacevic A, Tang Q, Rane S, Zhong W. Numerical modelling of twin-screw pumps based on computational fluid dynamics. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science. 2016;231(24).

[20] Litvin F L and Fuentes A Gear Geometry and Applied Theory (New York: Cambridge university press). 2014