Multiphase-flow simulation of a rotating rectangular profile within a cylinder in terms of hydraulic loss mechanisms

H Vasuthevan¹, A Brümmer¹

¹TU Dortmund University, Chair of Fluidics
Leonhard-Euler-Str. 5, 44227 Dortmund, Germany

E-mail: hanushan.vasuthevan@tu-dortmund.de, andreas.bruemmer@tu-dortmund.de

Abstract. In wet running screw compressors, liquid (oil or water) is injected into the machine. The advantage of liquid injection is cooling of the compressed fluid, sealing of clearances, lubrication of rotors and bearings, and reduced noise and vibration. The disadvantage of wet running machines is the loss in internal mechanical efficiency from the hydraulic loss caused by the liquid. In this paper, multiphase flow simulations are performed using the Volume of Fluid method to investigate hydraulic loss mechanisms. The 2D geometric model consists of a rectangular profile rotating in a cylinder with a clearance height that represents the screw machine housing clearance. In this investigation, the distribution and flow pattern of the liquid are examined with regard to hydraulic loss. Furthermore, the influence of the amount of liquid and the circumferential tip speed is determined through calculation of the torque on the rectangular contour. The hydraulic losses increase with increasing liquid mass in a working chamber and with increasing circumferential tip speed. For the simulations points examined, the tip speed has a greater impact on the hydraulic loss than does the amount of liquid.

1. Introduction
Twin-screw machines are widely used as compressors in industry and are the most commonly applied compressor type. Conventional applications are for compression of air, process gases or refrigerants. Furthermore, screw machines can be operated as expanders, which are typically used for waste heat recovery systems. Therefore, an increase in efficiency of screw machines leads to a positive contribution to the reduction of CO2 emissions in electrical power generation.

A crucial factor for the efficiency of screw machines is leakage flow through geometric clearances in the machine. These flows exist due to pressure differences between chambers and relative motion between rotors and housings. The efficiency of a screw machine can be improved by means of wet-running or liquid flooded machines. Here, an auxiliary liquid is brought or injected into the machine to partially seal the clearances, thus reducing gas leakage mass flow. Further beneficial effects due to the presence of liquid in the machine in terms of efficiency are cooling (compressor) or heating (expander) of gas by the liquid, lubrication of rotors, and reduction of vibration and noise. The disadvantage is the increase in internal mechanical losses due to friction and interactions between liquid and rotors or housings, referred to as hydraulic losses. Important factors affecting hydraulic losses that have been identified are: injected liquid mass, clearance heights, circumferential tip speed, and liquid viscosity [1, 2]. The influence of injected liquid for a screw compressor with oil injection [1] and for a screw expander with water injection [2] are analysed by means of indicator diagrams. It is shown in both cases that injection of liquid improves the performance of the machine due to clearance sealing, while
hydraulic losses increase with increasing circumferential tip speed, liquid mass and viscosity. Furthermore, it is suggested in [1] that a liquid surge is pushed in rotational direction at the rotor tip causing additional hydraulic losses. This hypothesis of a liquid surge is examined further in [3], where an analytical model for the distribution of injected oil in the working chamber and in the clearances is developed. The model is compared to experimental results, where the distribution of oil in a screw compressor is examined by means of optical access. An essential result was detection of an oil surge at the rotor tip. Nevertheless, the author described the restrictions of the optical analysis and proposed further investigation with improved optical resources. In a further work, distribution of water and oil in a screw expander is investigated [4].

An analytical model to determine hydraulic losses is developed in [5]; three loss mechanisms are identified: frictional loss, momentum loss, and acceleration loss. Frictional loss describes the viscous shear loss of the liquid. For the momentum loss, it is assumed that an attached liquid film on the housing is periodically accelerated by the passing rotor surface. The third loss mechanism is the acceleration from the injected liquid velocity to the circumferential speed of the working chamber. Further, in a fundamental experiment, one rotor is rotated in a liquid flooded cylinder to measure hydraulic losses, which consist of the frictional and the momentum loss. Comparisons of simulation results with the models are made for a screw compressor with regard to the integral hydraulic loss, but not for the individual loss mechanisms; thus, these analytical models are not validated explicitly. Empirical models for the prediction of hydraulic losses are given in [6]. Further works where the integral hydraulic losses of experimental wet – running screw machines are represented by a mechanical efficiency can be found in [7, 8, 9]. In [10, 11, 12] frictional losses in clearances due to liquid are examined with simplified numerical models, which consist of the superposition of turbulent incompressible Couette and Poiseuille flows. With these analytical models, chamber model simulations are performed to compare the behaviours of dry – and wet – running machines with the auxiliary liquid oil and water.

From the evaluation of the available literature it can be said that there are no validated, generally applicable, detailed methods or models for analysis of hydraulic loss mechanisms in a liquid flooded screw machine. A first step in this direction is made with two-dimensional, multiphase flow simulations to provide a better understanding of the hydraulic loss mechanisms. For a generic examination, a model of a rotating rectangular profile in a cylinder is used in this paper, whereby the cylinder is filled with air and a prescribed amount of water. The rotating rectangular profile can be developed further toward geometries characteristic of actual rotor profiles; it is intended that this be realised in future works. Most of the liquid will be accelerated out to the region of the rotor tip, where a liquid surge is formed [3]. In this paper, the hydraulic losses will be analysed in the steady state condition where the liquid surge is already formed at the contour tip.

First, common fundamentals and influencing parameters for hydraulic losses are described. Then the model and the boundary conditions of the two-dimensional multiphase flow simulation with the Volume of Fluid method (VOF) is presented. Subsequently, simulation results are presented and discussed.

2. Hydraulic losses
In this chapter essential fundamentals and influence factors for the examination of hydraulic losses in screw machines are described. The indicated power from a pressure-volume diagram can be calculated by equation (1); the result is the conversion of energy between the working fluid and the rotors due to the change of volume of a closed working chamber.

\[ P_i = n \int pdV \]  

The effective power \( P_e \) is the power which is measured on a rotor shaft by means of rotational speed and torque. Equation (2) shows the effective power, the sum of the indicated power and the mechanical power losses. The mechanical power losses can be divided further into two parts: internal \( \Phi_{m,i} \) and outer \( \Phi_{m,o} \) mechanical power losses.
\[ P_e = P_i + \Phi_m = P_i + \Phi_{m,i} + \Phi_{m,o} \]  \hfill (2)

While the internal mechanical power losses \( \Phi_{m,i} \) describe the losses caused due to friction and interactions between fluid and rotors in the machine, the outer mechanical losses consider the losses of external parts such as bearings and seals.

For dry running screw machines, the internal mechanical power losses \( \Phi_{m,i} \) are relatively small and therefore not considered, while for liquid-flooded machines the internal mechanical power losses cannot be neglected. Internal mechanical losses in liquid – flooded machines are called hydraulic losses. In the simulations, a rectangular profile rotating within a cylinder is considered; pressure-volume work is not performed and there are no outer mechanical losses. Therefore, the internal mechanical power loss can be analysed by means of the torque acting on the contour and the angular frequency, as shown in equation (3).

\[ \Phi_{m,i} = M \cdot \omega \]  \hfill (3)

In the left side of figure 1, a rectangular profile with the surface \( S \) is shown to illustrate the calculation of the torque \( M \). Forces acting on the surface in combination with the lever arms of the forces lead to the torque. The force acting on the surface can be divided into two parts: force in the normal direction \( \vec{n} \) and force in tangential direction \( \vec{t} \). The force normal to the surface is caused by fluid pressure while the force tangential to the surface is caused by the wall shear stress. The calculation of the torque \( M \) for the entire contour is given in equation (4).

\[
M = \oint_S dM = \oint_S \left[ -p(\vec{r} \times \vec{n}) + \tau_w(\vec{r} \times \vec{t}) \right] dS
\]  \hfill (4)

**Figure 1.** Left: Illustration for the calculation of torque on a rectangular surface \( S \)  Right: Female rotor tooth with liquid film attached on the housing and a liquid surge at the rotor tip.

The results from the simulation shown in this paper are all time – averaged within a quasi steady state range (from \( t \) to \( t + \Delta t \)) and referenced to one tooth \( M_{z} \) with a length of 1 m, as shown in equation (5), where \( z \) is the number of teeth on the rotor. It is assumed that the time step \( \Delta t \) is great enough that fluctuation of the time dependent torque can be neglected. Furthermore, the torque on one tooth, \( M_{z} \), can only be evaluated if the same force conditions exist at every tooth.

\[
M_{z} = \frac{1}{z} \cdot M = \frac{1}{z} \cdot \frac{1}{\Delta t} \int_{t}^{t+\Delta t} M(t)dt
\]  \hfill (5)
On the right side of figure 1, a possible condition of liquid surge at the tip of a female rotor in a typical screw machine is shown to identify the influencing parameters. Essential influence factors to describe the torque of a predetermined rotor tooth geometry are presented in table 1. It should be noted that further characteristic geometric parameters can be added as influencing parameters. Here, for a common consideration only the housing radius and the housing clearance height are mentioned as essential geometry parameters. The parameter liquid volume is added for analyses of three dimensional cases. For a two dimensional case, the liquid phase is represented by its surface area.

### Table 1. Influencing parameters for the rotor torque.

| Parameter                     | Symbol | Unit   |
|-------------------------------|--------|--------|
| Circumferential tip speed     | \( u \) | \( \text{m s}^{-1} \) |
| Viscosity liquid              | \( \nu_{\text{liq}} \) | \( \text{m}^2 \text{s}^{-1} \) |
| Density liquid                | \( \rho_{\text{liq}} \) | \( \text{kg m}^{-3} \) |
| Liquid volume                 | \( V_{\text{liq}} \) | \( \text{m}^3 \) |
| Housing radius                | \( r_{\text{housing}} \) | \( \text{m} \) |
| Housing clearance height      | \( h_{\text{hc}} \) | \( \text{m} \) |
| Chamber pressure              | \( p_1; p_2 \) | \( \text{N m}^{-2} \) |

Using these influencing parameters in application of the Buckingham Pi theorem, a dimensional analysis can be performed and dimensionless characteristic numbers can be derived. Five independent dimensionless numbers are needed to describe the torque on a rotor tooth if liquid is present in the chamber. For every additional geometry parameter in table 1, an additional characteristic number can be developed to describe the geometry. The characteristic numbers so derived are given in table 2.

### Table 2. Characteristic numbers for the torque of a rotor tooth in case of liquid presence.

| Characteristic number                     | Formula                                                                 |
|-------------------------------------------|-------------------------------------------------------------------------|
| Circumferential Reynolds number           | \( \text{Re} = \frac{uh_{\text{hc}}}{\nu_{\text{liq}}} \)              |
| Moment coefficient                         | \( C_M = \frac{\overline{M_z}}{V_{\text{liq}}\rho_{\text{liq}}u_{\text{tip}}^2} \) |
| Pressure ratio                             | \( \Pi = \frac{p_1}{p_2} \)                                           |
| Liquid volume ratio                        | \( \varepsilon = \frac{V_{\text{liq}}}{V_{\text{chamber}}} \)         |
| Geometry number                           | \( G = \frac{h_{\text{hc}}}{r_{\text{housing}}} \)                   |

3. Modelling and boundary conditions
For the generic examination of hydraulic losses, a rotating rectangular profile in a closed cylinder is considered; this represents a coarse estimation of a female rotor tooth. For this case, two dimensional multiphase simulations are carried out with the Volume of Fluid (VOF) method using Ansys Fluent 17.1. Details of the mesh, boundary conditions, and solver settings are presented in the following section.
3.1. Geometrical model
As shown in figure 2, the fluid zone of the geometric model was meshed using a structured grid with hexa elements. The geometric model of a complete closed cylinder is reduced to the two-dimensional model shown based on assumptions of point symmetry of the flow field. In this case the flow entering through one periodic interface is equal to the flow exiting the other periodic interface.

![Figure 2. Meshed geometric model of rotating rectangular profile in cylinder.](image)

The mesh was created with the ICEM 17.1 software. In the region of the clearance flow and the solid walls, the mesh contains finer elements, due to higher rates of change of the fluid states in this region. Geometric and mesh information are given in table 3. Skew and aspect ratio are used in assessing mesh quality. Higher aspect ratios in a smaller number of elements (1.7% of the total number of elements) occur away from the clearance region at the housing. Thus, high aspect ratios in these regions are of minor importance. Overall, the mesh quality is in an acceptable range.

| Table 3. Geometrical and mesh information. |
|------------------------------------------|
| housing radius | \( r_{\text{housing}} \) | 0.07 | m |
| housing clearance height | \( h_{\text{hc}} \) | 0.0001 | m |
| housing clearance length | \( l_{\text{hc}} \) | 0.005 | m |
| element number | \( n_{\text{element}} \) | 935961 | - |
| mesh - determinant | - | range: 0.95 -1.0 (100 %) | - |
| mesh - skew | - | min range: 0.5-0.8 (4.5 %) | - |
| mesh - aspect ratio (Fluent) | - | max range: 70-270 (1.7 %) | - |

3.2. Boundary conditions and solver settings
Transient simulations are performed with the explicit VOF method, where a closed cylinder is filled with air and an amount of liquid water. The air and liquid water are assumed to be incompressible fluids with constant density (air: 1.225 kg m\(^{-3}\) and water: 998.2 kg m\(^{-3}\)). Furthermore, the surface tension between air and water is asset to the constant value of \( \gamma = 0.072 \) N m\(^{-1}\). The dynamic viscosities are: air \( \eta_{\text{air}} = 1.789 \cdot 10^{-5} \) kg m\(^{-1}\) s\(^{-1}\) and water \( \eta_{\text{water}} = 1.003 \cdot 10^{-3} \) kg m\(^{-1}\) s\(^{-1}\). As a boundary condition, the temperature of air and water are set at the same constant value of 300 K, thus the energy conservation equation of is not used.
The fluid zone and the rectangular contour are calculated in a rotating coordinate frame with a constant angular velocity in clockwise direction, while the housing is rotating in the counter clockwise direction. The simulations are performed with a formulation of the conservation equations in the relative system, where centrifugal and Coriolis forces are considered.

Furthermore, for this generic examination, laminar flow is assumed. It is supposed that laminar flow is sufficient to examine the hydraulic loss mechanisms although a turbulent flow probably is existent. For the consideration of water in the closed cylinder, the amount of liquid is initialised. Due to the examination of a stationary or quasi-stationary solution the location of the water initialisation is not essential for the solution. Nevertheless, the water is assumed to be near the rotor tip and housing. Water is initialised in this region to achieve the stationary solution faster. The absolute pressure in the cylinder in both chambers is initialised with $p_{init} = 0.1$ MPa; thus only a pressure ratio of $\Pi = 1$ over the chambers is examined in the simulations. In table 4 essential solver settings for the simulations with Fluent 17.1 are shown.

### Table 4. Solver settings with Fluent 17.1.

| Pressure-Velocity-Coupling | SIMPLE |
|----------------------------|--------|
| Gradient                  | Green-Gauss Node Based |
| Discretization - Pressure | Body Force Weighted |
| Discretization - Momentum | Second Order Upwind |
| Discretization – Volume Fraction | Geo-Reconstruct |
| Transient Formulation     | First Order Implicit |

A variable time stepping method was chosen for the calculations with a minimum time step of $t_{min} = 10^{-9}$ s, a maximum time step of $t_{max} = 10^{-6}$ s, and a global Courant number of $\text{CFL}_G = 0.8$. The maximum number of iterations per time step was set to 30. When all residuals fall below 0.001, the next time step is calculated. Furthermore, the torque on the contour is analysed to observe if a steady or quasi-steady solution is reached. An additional condition for every simulation point is at least one revolution of the rectangular contour.

### 4. Results

Simulation results are presented in this section. First, a detailed view of the flow pattern for the case of a simulation without liquid is given. Subsequently, the influence of the liquid volume ratio and the circumferential tip speed will be analysed with regard to their effects on hydraulic losses. Finally, results are presented depending on the characteristic numbers in a characteristic diagram. It should be noted that the results of the two-dimensional simulations are all referenced on one tooth with a tooth width of $l = 1$ m.

#### 4.1. Flow pattern

In this section, the flow pattern will be analysed for the case without liquid $\varepsilon = 0$ and for the case with a liquid volume ratio of $\varepsilon = 0.005$ at a constant circumferential tip speed of $u_{tip} = 30$ m s$^{-1}$.

##### 4.1.1. Case $\varepsilon = 0$: Only air

In figure 3, the flow field for the case without liquid ($\varepsilon = 0$) in the cylinder is shown in the relative system. It should be noted that the full cylinder is only shown here for reference and visualisation. In the relative system, the air flows towards the housing clearance and is then separated into two domains, where the upper part flows through the clearance. The other part flows tangentially at the contour towards the centre and further towards the opposite clearance, where the flow is disturbed slightly by a flow separation dead space before the flow is brought together with the stream of the lower clearance flow. This stream flows towards the upper clearance again and the process is
repeated. Due to this flow behaviour, a counter clockwise rotating vortex is formed at the centre of each chamber. The relative tangential velocity at the rectangular contour, which is given in figure 3, can be used in heat transfer modelling.

For the transient simulations with only air, the torque converged to a constant value: $M_z = 0.027 \, \text{Nm}$. The torque is mainly influenced by the pressure difference $\Delta p_{\text{max,tip}}$ which is 1250 Pa at the contour tip and which decreases with decreasing radius.

**Figure 3.** Relative flow pattern without liquid $\epsilon = 0$ for a circumferential speed of $u_{\text{tip}} = 30 \, \text{m s}^{-1}$.

4.1.2. Case: $\epsilon = 0.005$. The progression of the torque for the case with a liquid volume ratio of $\epsilon = 0.005$ is shown in figure 4. While in the case without liquid the torque tends to a constant value, the torque with liquid is fluctuating with time. In this case, the mean value of the fluctuation can be used to quantify effects. Furthermore, a liquid surge, which varies with time, is formed at the rotor tip. In figure 4, black areas represent water and grey identifies areas of air. It can be seen that the fluctuation of the torque on the contour is influenced essentially by the shape of the liquid surge at the contour tip. The time averaged torque is $M_z = 2.2 \, \text{Nm}$ and the pressure at the contour tip on the liquid surge side at a point in time $t = 0.023 \, \text{s}$ is about $p_{\text{max,tip}} = 0.23 \, \text{Pa}$, while the pressure on the opposite side of the liquid surge at the contour tip is about $p_{\text{tip,op}} = 0.102 \, \text{MPa}$.

**Figure 4.** Torque over time for a liquid volume ratio of $\epsilon = 0.005$ and a circumferential speed of $u_{\text{tip}} = 30 \, \text{m s}^{-1}$ with flow pattern of liquid surge at different moments (black – water ; grey – air).
The complete relative flow pattern is presented at time $t = 0.023 \text{s}$ in figure 5. The flow field can be divided into two regions, where the one region is similar to the flow field without liquid. In this region, the vortex at the centre of the chamber can be identified. The other region is located in the vicinity of the rotor tip and is disturbed due to the liquid. This region is more complex and consists of smaller regions. The main field of the liquid surge at the contour tip is shown in figure 5b. It should be noted that the rest of the stream lines is not shown here, which mainly consist of air and is attached to the liquid surge. The liquid distribution can be divided into two parts: a liquid surge, which is located directly at the contour tip, and a liquid film, which is attached to the housing.

Figure 5c shows the flow field of the liquid at the contour tip; this will be discussed in the following. The black streamlines show the flow of water and the grey streamlines the flow of air. Similar to the flow of air, liquid flows towards the housing clearance and is separated into two streams. One part of the liquid flows through the upper clearance and flows further as a thin liquid layer on the circumference of the housing towards the lower clearance. The other part flows tangentially on the rectangular contour in the direction of the centre of rotation. This flow is then deflected due to the centrifugal forces into circumferential and further in radial direction, where the liquid is entrained by the liquid stream which then flows towards the upper clearance. Accordingly, a rotating liquid surge is formed at the rotor tip. This rotating liquid surge is surrounded by an outer stream, which flows back away from the clearance.

Thus, two liquid layers exist at the housing moving in different circumferential directions: a liquid stream towards the clearance and an outer stream away from the clearance. During the backflow of the outer liquid stream, part of the liquid is entrained by the liquid stream, which flows towards the clearance. Furthermore, dead spaces can be identified between the two liquid streams. The liquid surge and the liquid streams cause essential shear stresses and pressure variation along the contour, which results in resistance to rotation.

A backflow against the relative rotation of the housing seems unlikely. It is assumed that the size of the rotating liquid surge is restricted due to the centrifugal forces acting in the relative system. For the steady state situation, no more liquid can be added to the rotating liquid surge; thus, liquid is forced to flow back around the surge and away from the contour.

4.2. Variation of liquid volume ratio

In this section, the liquid volume ratio $\varepsilon$ is varied from 0.25 % to 1 % for a constant circumferential tip speed of $u_{\text{tip}} = 30 \text{ m s}^{-1}$. The time-averaged torque and the power loss over the liquid volume ratio are
shown in figure 6. It can be seen that the total torque on the contour increases almost linearly with increasing liquid volume ratio. Therefore, the hydraulic power loss also increases linearly for a constant circumferential tip speed. Additionally, the time averaged torque and power loss, caused by the flow in the clearance gap, are shown in the diagrams. The torque on the contour caused by shear stresses in the clearance flow is almost constant and independent of the liquid volume ratio. Furthermore, it can be said that the influence of the liquid surge on the hydraulic power loss is greater than the effect of friction in the clearance area. The time-averaged torque and the power loss due to the liquid surge can be determined from the diagrams as the differences between the total and clearance values.

![Figure 6](image1.jpg)

Figure 6. Time averaged torque and power loss over the liquid volume ratio at a fixed circumferential tip speed \( u_{\text{tip}} = 30 \text{ m s}^{-1} \).

The essential reason for the increase of the power loss is the increasing pressure difference at the contour tip. In figure 7 the liquid distribution and the pressure at the contour tip on the liquid surge side for different liquid volume ratios are shown. The black coloured areas consist of water and the grey areas consist of air. It should be noted that the pressure at the contour tip on the opposite side of the liquid surge is about \( p_{\text{tip}, \text{op}} = 0.102 \text{ MPa} \) for the liquid volume ratios examined in the study.

With increasing liquid volume ratio, the liquid film thickness near the liquid surge increases. Further, it can be said that for low liquid volume ratios, the liquid surge contains air inclusions. The air inclusions decrease with increasing liquid volume ratio, thus the density of the liquid surge increases. This can be a possible reason for the increase of the pressure at the contour tip. Further, it seems that the height of the liquid surge at the contour remains constant for across the range of liquid volume ratio studied.

![Figure 7](image2.jpg)

Figure 7. Flow pattern of the liquid at the contour tip for different liquid volume ratios at a fixed circumferential tip speed \( u_{\text{tip}} = 30 \text{ m s}^{-1} \) (black – water; grey – air).

4.3. Variation of circumferential tip speed

Figure 8 shows the time-averaged torque and total power loss over the circumferential tip speed for a constant liquid volume ratio of \( \varepsilon = 0.0025 \). The torque increases almost quadratically and the hydraulic power loss increases almost cubically with increasing circumferential tip speed. Furthermore, the time-
averaged torque and power loss caused due to the clearance flow are shown. For increasing circumferential tip speed, the hydraulic power loss due to the clearance flow increases, but the major influence on the total hydraulic power loss is still the liquid surge at the contour tip.

\[ \frac{dp(r)}{dr} = \rho \frac{u^2(r)}{r} \]  

Equation 6, a balance of forces in the radial direction, is used to examine the qualitative progression of the pressure. This equation is only valid for simplified cases, where, for example, the radial velocity is zero. Nevertheless, the equation shows that the change in pressure in radial direction is influenced quadratically by the circumferential tip speed; this can be seen in the progression of the torque in figure 8. The pressure is influenced additionally by the high density of the liquid, which is mainly located at the contour tip.

Furthermore, it can be seen that with increasing circumferential tip speed, the height of the rotating liquid surge at the contour tip decreases. This behaviour supports the aforementioned statement that the size of the rotating liquid surge is restricted by the centrifugal forces.

\[ p_{\text{max,tip}} = 212740 \text{ Pa} \]
\[ p_{\text{max,tip}} = 123319 \text{ Pa} \]
\[ p_{\text{max,tip}} = 721376 \text{ Pa} \]
\[ u_{\text{tip}} = 10 \text{ m s}^{-1} \]
\[ u_{\text{tip}} = 30 \text{ m s}^{-1} \]
\[ u_{\text{tip}} = 60 \text{ m s}^{-1} \]

Figure 9. Flow pattern of the liquid at the contour tip for different circumferential tip speed at a fixed liquid volume ratio of \( \varepsilon = 0.0025 \) (black – water; grey – air).
4.4. Characteristic diagram

In figure 10, the time averaged torque for the liquid volume ratios studied are shown as functions of the circumferential tip speed. In case of a liquid volume ratio $\varepsilon = 0$, where no liquid is present, the torque due to internal mechanical losses is very small in contrast to the case with liquid.

![Figure 10](image)

Figure 10. Time averaged torque of the contour for different liquid volume ratios and circumferential tip speeds.

Dimensionless characteristics based on the simulation results are presented in figure 11, where the moment coefficient over the circumferential Reynolds number is shown for different liquid volume ratios. This diagram is only valid for the rectangular profile used here and for a chamber pressure ratio of $\Pi = 1$. The moment coefficient decreases with increasing circumferential Reynolds number. It is important to realise that a smaller moment coefficient does not lead automatically to lower torque or reduced hydraulic losses.

![Figure 11](image)

Figure 11. Moment coefficient over circumferential Reynolds number for different liquid volume ratios.
The torque depends mainly on the liquid mass and the circumferential tip speed. For similar cases, the moment coefficient can be determined based on the circumferential Reynolds number to determine torque on the contour. However, the characteristic diagram should be validated first with further similar simulation points, where, for example, geometry and liquid content are varied.

5. Conclusion
In this paper hydraulic loss mechanism as might occur in liquid-flooded screw machines were examined. For the examination, two dimensional transient simulations were performed with the Volume of Fluid method. The geometric model is one of a rotating rectangular profile in a cylinder filled with air and containing various amounts of water. The model is developed in a relative system, where the coordinate system of the contour rotates clockwise with a prescribed angular velocity; the relative motion of the housing is then in the opposite direction. The circumferential tip speed and liquid volume ratio were varied. First the relative flow pattern is analysed to identify the important hydraulic loss mechanisms. Subsequently, the influence of liquid volume ratio and circumferential tip speed are analysed. Finally, simulation results are presented in a dimensionless characteristic diagram, with the recommendation to have this validated in further investigations. The simulation results are analysed in a stationary state.

When there is liquid in the chamber, it is distributed to the rotor tip and casing, forming a rotating liquid surge at the contour tip and resulting in a liquid film becoming attached to the housing surface. The liquid film is transported by the housing towards the rotor-housing clearance, where part of the liquid flows through the clearance and the other part creates the liquid surge at the contour tip. Further, it can be observed that the rotating liquid surge is surrounded by a moving liquid stream which flows away from the clearance. Therefore, the liquid film consists of two liquid streams which flow in contrary circumferential directions. This motion creates additional shear stresses along the liquid film between the two streams. It is proposed that the size of the rotating liquid surge is restricted by the centrifugal forces. Therefore, the separated liquid mass at the contour tip is forced to flow back away from the clearance. During the backflow, the liquid mass is entrained by the liquid stream attached to the housing and flows towards the housing clearance.

The analysis of the torque shows a fluctuating characteristic, which is influenced by the change of the liquid shape at the rotor tip. The time-averaged value of the fluctuation can be used to characterize results. The torque increases almost linearly with increasing liquid volume ratio and quadratically with circumferential tip speed. Furthermore, the hydraulic loss due to the clearance flow between rotor and housing is analysed. It can be observed that the liquid flow at the contour tip is the major component of the hydraulic loss rather than the hydraulic loss due to the clearance flow itself.

Simulation results are presented in a dimensionless characteristic diagram showing the moment coefficient over the circumferential Reynolds number for different liquid volume ratios. For similar cases, the moment coefficient can be determined using the circumferential Reynolds number to determine the hydraulic power loss.

For future works, simulation of screw machine rotor profiles can be examined in a cylinder with different pressure ratios to create several characteristic diagrams. Furthermore, the characteristic diagrams should be validated with several simulation points while the characteristic numbers stay constant. These characteristic diagrams can be implemented, for example, in a chamber model simulation as a database to determine internal mechanical losses. As a matter of course, the simulation model should be validated in an experiment.
List of symbols and abbreviations

| Symbols | Abbreviations       |
|---------|---------------------|
| b       | width (m)           |
| CFL     | Courant number (-)  |
| h       | height (m)          |
| M       | torque (N m)        |
| n       | rotational speed (s⁻¹) |
| P       | power (N m s⁻¹)     |
| p       | pressure (N m²)     |
| r       | radius (m)          |
| t       | time (s)            |
| u       | circumferential tip speed (m s⁻¹) |
| V       | volume (m³)         |
| γ       | surface tension (N m⁻¹) |
| ν       | kinematic viscosity (m² s⁻¹) |
| ρ       | density (kg m⁻³)    |
| Φ       | loss power (N m s⁻¹) |
| ω       | angular frequency (rad s⁻¹) |
| i       | indicated, internal |
| e       | effective           |
| he      | housing clearance   |
| m       | mechanical          |
| o       | outer               |
| liq     | liquid              |
| init    | initialisation      |
| max     | maximum             |
| min     | minimum             |
| G       | global              |

References

[1] Kauder K 1987 Das Öl im Schraubenkompressor - ein Faktor für optimale Betriebsverhältnisse *Pumpen Vakuumpumpen Kompressoren* 87 key no. 71075 pp 38-44 (Düsseldorf)

[2] Nikolov A and Brümmer A 2016 Analysis of Indicator Diagrams of a Water Injected Twin-shaft Screw-type Expander *International Compressor Engineering Conference* (Purdue University Libraries)

[3] Harling H 1993 Untersuchungen zur Ölverteilung in Schraubenkompressoren mit Schmiermitteleinspritzung Dissertation University of Dortmund

[4] Zellermann R 1996 Optimierung von Schraubenmotoren mit Flüssigkeitseinspritzung Dissertation University of Dortmund

[5] Deipenwisch R 2000 Ein Beitrag zum Einsatz von Öl als Konstruktionsel in Schraubenmaschinen Dissertation University of Dortmund

[6] Singh P.J. and Patel G.C. 1984 A Generalized Performance Computer Program for Oil Flooded Twin-Screw Compressors *International Compressor Engineering Conference* (Purdue University Libraries)

[7] Dämgen U 1994 Wassereinspritzung in Schraubenkompressoren *VDI-Berichte - Schraubenmaschinen* 94 pp 175-194

[8] Kliem B 2005 *Grundlagen des Zweiphasen-Schraubenmotors* Dissertation University of Dortmund

[9] Nikolov A and Brümmer A 2018 Impact of different clearance heights on the operation of a water-flooded twin-screw expander – experimental investigations based on indicator diagrams *International Conference on Screw Machines* (Dortmund)

[10] Gräßer M and Brümmer A 2014 An analytic model of the incompressible one-phase clearance flow in liquid injected screw expanders *International Conference on Screw Machines – VDI Bericht* 2228 pp. 71-89

[11] Gräßer M and Brümmer A 2015 Influence of liquid in clearances on the operational behaviour of twin screw expanders *International Conference on Compressors and their Systems – IOP Conf. Series: Materials Science and Engineering* 90

[12] Gräßer M and Brümmer A 2016 Influence of water and oil clearance flow on the operational behavior of screw expanders *Journal of process mechanical engineering Part E*