

Article

Advanced 2D Computational Fluid Dynamics Model of an External Gear Pump Considering Relief Grooves

Nikolay Nikolov ¹, Alexander Mitov ^{2,*}  and Ivan Kralov ¹

¹ Department of Mechanics, Technical University of Sofia, Kliment Ohridski 8 Boulevard, 1000 Sofia, Bulgaria; nyky@tu-sofia.bg (N.N.); kralov@tu-sofia.bg (I.K.)

² Department of Hydroaerodynamics and Hydraulic Machines, Technical University of Sofia, Kliment Ohridski 8 Boulevard, 1000 Sofia, Bulgaria

* Correspondence: a_mitov@tu-sofia.bg; Tel.: +359-886208937

Abstract: The article presents an advanced two-dimensional (2D) computational fluid dynamics (CFD) model of an external gear pump which considers relief grooves. Relief grooves are limiting design features for the flow process of this type of pump, and their influence in existing studies is considered by a three-dimensional (3D) model only. The structural modification proposed by the authors is beyond the possibilities of real implementation, but it gives the possibility to precisely model the pump's design features. In contrast to the existing studies (using 3D CFD), the proposed advanced 2D model requires significantly fewer computing resources. Numerical experiments were carried out using the 2D model at different pump operating modes depending on the rotation frequency (950–1450 min^{−1}) and pressure load (5–150 bar). The numerical results were validated by a real-world experiment for the same pump operating modes using an existing laboratory experimental setup. An analysis of the CFD model and real experiment results was carried out by determining a quantitative index of match (FIT), which varies in the range of 97.93–99.82%. This proves the performance of the proposed CFD model, which can be further used as a part of more complex hydraulic systems models.

Keywords: external gear pump; CFD; volumetric flow rate; relief grooves



Citation: Nikolov, N.; Mitov, A.; Kralov, I. Advanced 2D Computational Fluid Dynamics Model of an External Gear Pump Considering Relief Grooves. *Appl. Sci.* **2024**, *14*, 4299. <https://doi.org/10.3390/app14104299>

Academic Editor: Marek Krawczuk

Received: 12 April 2024

Revised: 14 May 2024

Accepted: 15 May 2024

Published: 19 May 2024



Copyright: © 2024 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction and Motivation

One of the most commonly used pumps in hydraulic drive practice is the external gear pump. In systems with velocity throttle control of the hydraulic motors, without high requirements regarding the flow rate ripple and the need for continuous operation at high pressure, this type of pump is most preferred. Their relatively low cost and availability in the market also contribute to their wide application not only for industrial but also for mobile applications, especially when a pump with a small displacement volume is needed [1,2]. There are many disadvantages of external gear pumps. The most prominent disadvantages are the relatively large level of flow rate ripple, the fixed displacement volume, high noise levels, the impossibility of continuous operation at a pressure above 20 MPa, non-repairability, and others. But the relatively simple construction of these pumps motivated researchers years ago to look for structural modifications to overcome the listed disadvantages. To date, the development of modern software products with an engineering application makes it possible to solve complex tasks requiring not only constructive experience but also process knowledge. With the advent of computational fluid dynamics (CFD) ANSYS Fluent® 2019 R3 software, design modifications' influence on working characteristics of hydraulic machines has become easier to study. Of course, this is only useful if the numerical results are experimentally validated [3].

The literature review shows that the research on external gear pumps using a fluid–structural approach can be summarized in several main groups. One group is aimed at constructive improvements related to gaps in order to reduce internal volume losses

(leakages) and consequently increase volumetric efficiency [4–9]. Another significant part of the research is aimed at changing the number or geometric profile of the teeth in order to reduce the flow rate ripple [10–14]. Often, these studies also contribute to reducing the level or intensity of generated noise [15–24]. Not infrequently, the goal is to create modifications for continuous work at high pressure [25–27]. This necessitates the investigation of directly related parameters—trapped volume, pressure in the inter-teeth zone, and distribution of the radial force on the gears [28–33]. In summary, these are studies on the influence of various design parameters on the performance and universal characteristics of the machine. There is research aimed at developing specific applications based on gear pumps [34–38]. The cavitation phenomena are also a current research problem, which is valid not only for this gear type but also for other types of hydraulic displacement rotary machines [8,13,27,39–44].

Creating a CFD model of newly designed or existing hydraulic machines (volumetric or turbo) is not an easy task and depends directly on the purpose for which it will be used [45,46]. The main question to consider is whether to create a two-dimensional (2D) or three-dimensional (3D) model [47–51]. The answer depends on whether it is possible to create a 2D model that gives results very close to the 3D model while using significantly less computational resources.

One of the main problems of external gear pumps' CFD modeling is the influence of so-called relief grooves (channels) located on the two bearing bodies (balancing plates) [52]. This has motivated the authors to design a 2D model solution beyond the possibilities of the construction, which would enable these grooves to be modeled in such a way as to properly consider their influence on the numerical results.

The general motive for the present study is that the classical 2D models [3,5,8,36,40,43,44] do not consider the influence of relief grooves, although it is possible. Omitting relief grooves causes the following negative effects:

- Inter-teeth zone pressure obtained using the classical 2D model is not realistic because this zone is, most of the time, connected by such grooves. The pressure in the connected inter-teeth zones is usually equal to that in either the suction or the discharge port. This disadvantage is illustrated in Figure 1, where the relief grooves are superposed on a pressure field obtained with a classical 2D model. It can be seen that in the case of two contact points between the teeth, the pressure in the trapped zone is colored neither orange nor blue but has a much higher value (in red). Also, the inter-teeth zones colored in yellow should have a pressure value equal to that in the outlet zone (in orange).

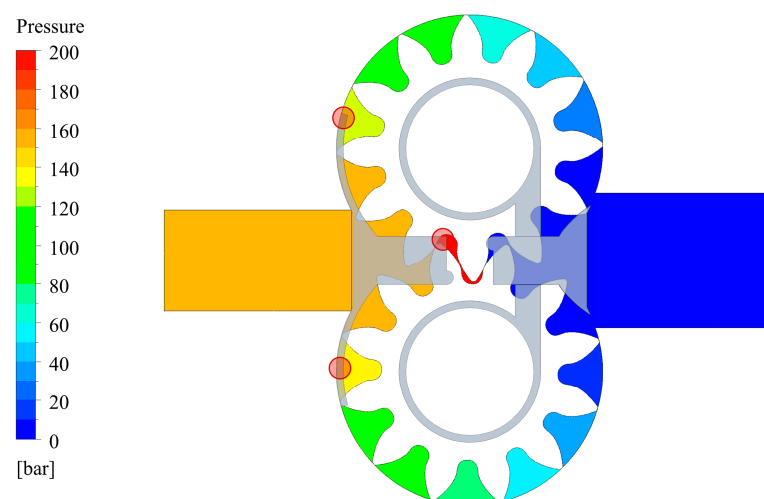


Figure 1. Pressure distribution with a classic 2D model at an output pressure of 150 bar. The relief grooves are superposed, and the unconsidered connections are marked with red circles.

- In the presence of two contact points, the fluid volume trapped between them is completely transferred to the low-pressure zone. This reduces the volumetric flow rate below the theoretically and experimentally determined values and increases the pump flow rate ripple. In reality, this closed volume is much shorter than the classical model shows since most of the time (during pump operation), it is connected by the relief grooves to either the high-pressure or the low-pressure zone. Figure 2 shows a technique to compensate for the reduced flow rate q obtained with a classical 2D model by increasing the gear width b . This technique has been shown in [3] to perform well over a wide range of modes. However, the expected average flow rate value of the studied pump, also shown in Figure 2, at the considered operating mode is nearly 27 L/min, while the classic 2D model gives 26.13 L/min. This disadvantage can be avoided by considering the relief grooves' influence.
- The classical 2D model gives an unrealistic volumetric flow rate variation, which cannot be used for further studies of flow and pressure pulsations in the discharge pipeline, causing noise and vibrations. The law shown in Figure 2 is strongly dependent on the number and position of the contact points during the mesh of both gears, while at the same time, it does not depend on the shape and location of the relief grooves, which have a significant influence on it.

The literature review showed that relief grooves are considered using 3D models only [14,26–30,41,42]. This creates certain straits since these models require too much computing resources and/or the use of specialized software that is purchased separately. No attempts were found in the literature to add relief grooves to the significantly simpler and more resource-efficient 2D models, which still have their application in some studies of external gear pumps. This motivates us to prove whether and to what extent it is possible to account for the influence of relief grooves using a 2D model.

The main goal of the article is to design an advanced 2D CFD model of an existing external gear pump considering the relief grooves' influence on the flow process.

This article is organized as follows: Section 2 includes a detailed description of geometric model development; Section 3 presents the numerical results and discussion; Section 4 shows the experimental study and model validation; and in Section 5, some conclusions are given.

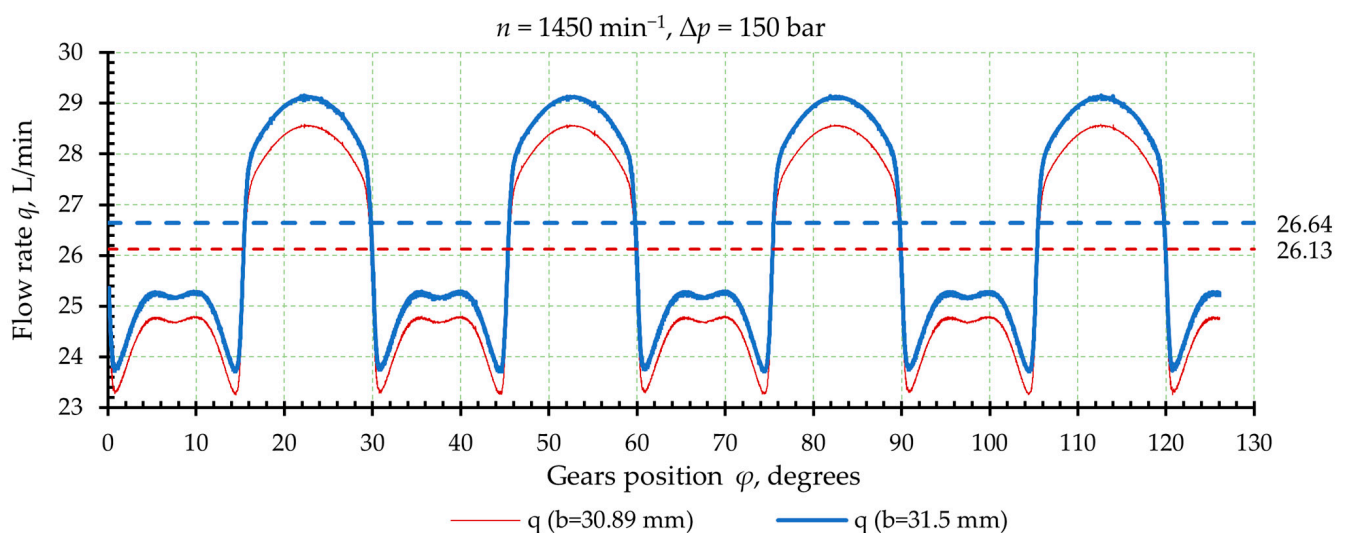


Figure 2. Flow rate correction by increasing gear face width from 30.89 to 31.5 mm (q —flow rate; b —gear width, φ —gears position), as proposed in [3].

2. Geometric Model Considering Relief Grooves

The object of the study is a certain specimen of an external gear pump with a displacement volume of 19 cm^3 . The gears teeth number is 12, and their profile is involute. Figure 3 shows a disassembled state of the pump 3D model created by the authors. The creation of this model is described in [3].

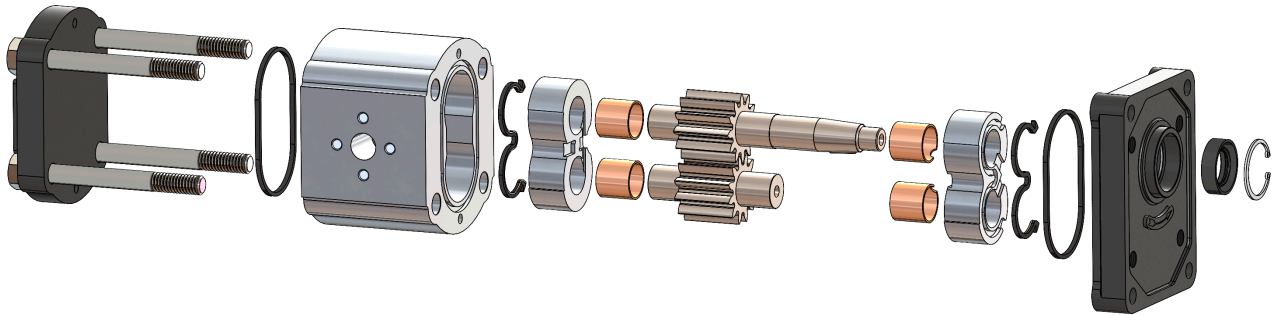


Figure 3. The 3D model of the external gear pump—disassembled state.

In order to model the influence of the relief grooves on the gear pump flow process, it is necessary to create an accurate CAD geometry first. This makes it possible to precisely determine and study the internal geometric volume, which can subsequently be modeled using appropriate 2D shapes. Figure 4 presents a different layout of geometric volume. Figure 4a shows a general view of the entire fluid volume and the plane of symmetry. A section through the plane of symmetry that forms the geometry of a classic 2D model is presented in Figure 4b. The relief grooves in the section parallel to the plane of symmetry are shown in Figure 4c. The challenge is to consider the influence of the grooves shown in Figure 4c, making changes to the section shown in Figure 4b.

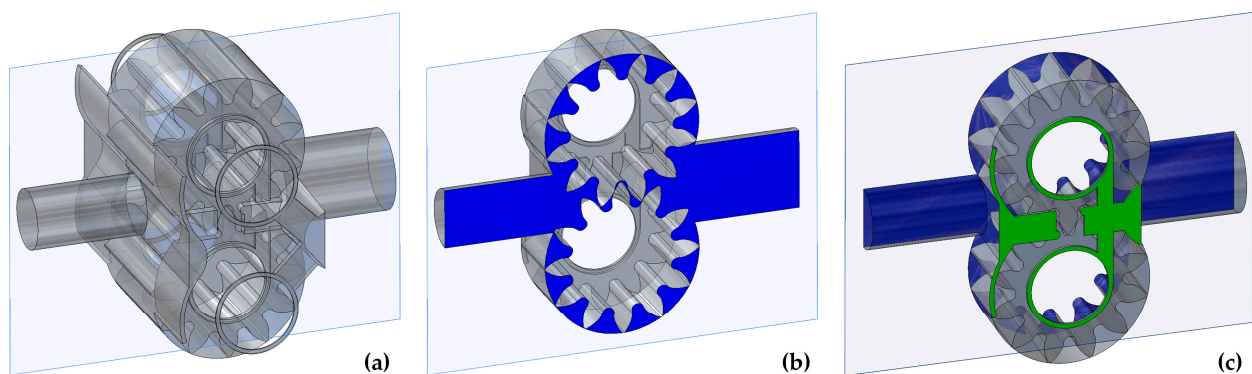


Figure 4. Geometric volume: (a) general view of the fluid volume and the plane of symmetry; (b) a section through the plane of symmetry that forms the geometry of a classic 2D model; (c) relief grooves in a section parallel to the plane of symmetry.

The relief grooves can be represented in 2D by imaginary radial and tangential channels, which make the same connections at the same moments as the real ones—Figures 5 and 6. The radial channels connect the inter-teeth zone to the tangential channels, which makes the same connection between the inter-teeth zones as the actual grooves. The radial channels rotate with the gears while tangential channels remain stationary. A minimum clearance of 0.01 mm is set between the stationary and rotation walls outside the tangential channels. This value is small enough to prevent leakages and pressure loss but sufficient to ensure finite elements mesh continuity throughout the solution. A similar gap value (0.0092 mm) was used for gear meshing, while the gap above the gear tips was 0.02 mm.

The length and width of the radial channels should preferably be minimal so as to not significantly affect the inter-teeth zone dimensions. Two sizes of these grooves were used in

the present study: 2×0.25 mm and 0.5×0.25 mm. The simulations showed that reducing the radial channel length from 2 to 0.5 mm did not affect the characteristic flow rate as a function of the time $q(t)$ in any way—the differences in the instantaneous and mean values of q are below 0.02%, so a length of 0.5 mm is preferred. The width of 0.25 mm is chosen so that at each moment of the simulation the channel is meshed by at least two rows of finite elements, thus improving the solution quality. Too small a width runs the risk of an unnatural increase in hydraulic resistance, which is not observed in actual grooves.

The length of the tangential channels, as well as the position of their beginning and end, is strictly dependent on the 3D geometry of the studied pump and on the radial channels' width. To find the necessary angles α_1 – α_4 , defining channels begin and end, the drawings shown in Figure 5 were used.

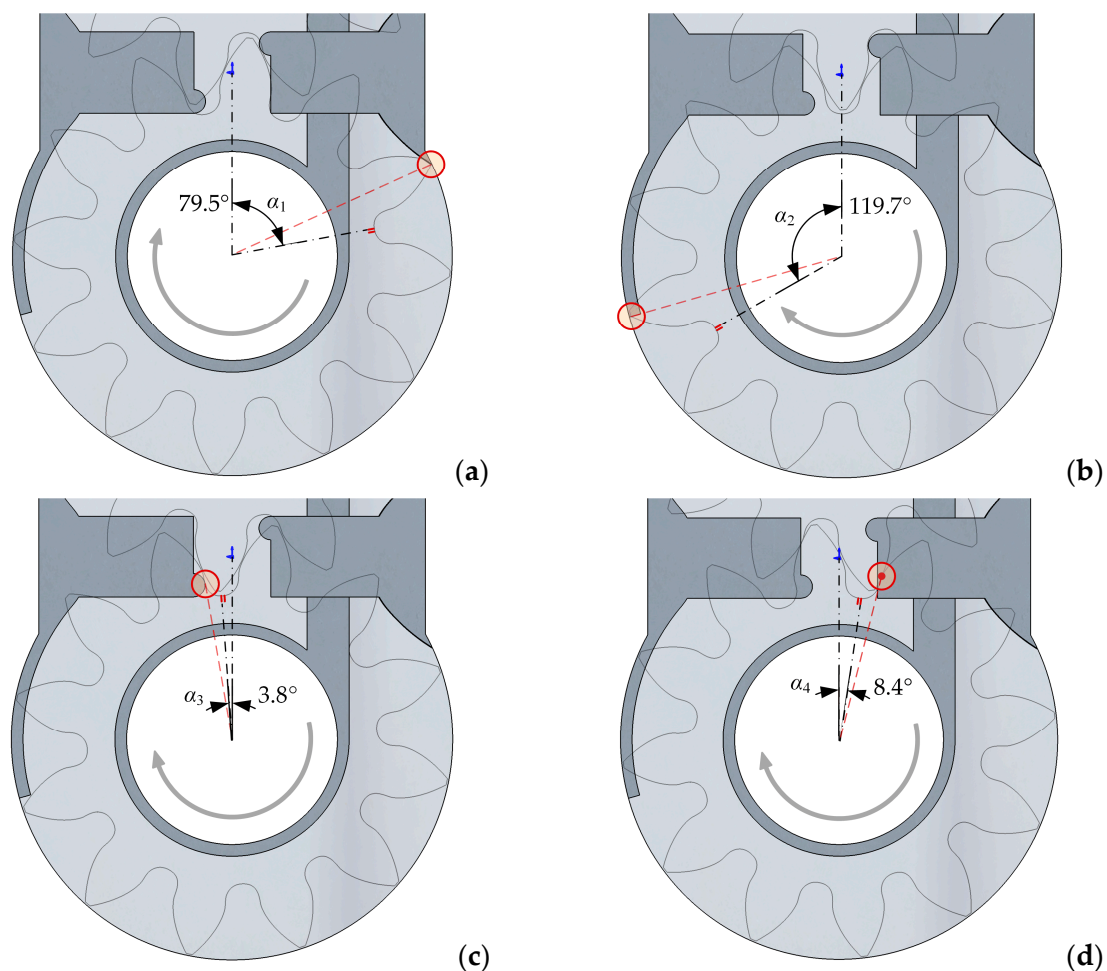


Figure 5. Relief channels geometry determination (connection zones are marked with red circles): (a) the inter-teeth zone comes out of connection with the inlet zone; (b) the inter-teeth zone comes into connection with the outlet zone; (c) the inter-teeth zone comes out of connection with the outlet zone; (d) the inter-teeth zone comes into connection with the inlet zone (α_4 enlarged with 1.4°).

Tangential channels are two per gear—Figure 6. The beginning and end of each of them are determined using separate drawings (Figure 5a–d). The angles α determine the position of the corresponding end relative to the axis passing through the centers of the gears. A careful analysis of the created 3D geometric model showed a gear position, presented in Figure 5d, where the high- and low-pressure zones are connected through the inter-teeth zone. To avoid this, during the tangential channels design, the angle α_4 slightly increased from its original value of 7.04° to the safer value of 8.4° .

The width of the tangential channels can be chosen arbitrarily as long as it is greater than that of the radial grooves. A large width unnecessarily increases the number of finite elements and complicates the model; therefore, a value of 0.3 mm was chosen, which also fits the finite element mesh parameters well.

The final shape and dimensions of the advanced 2D model are shown in Figure 6.

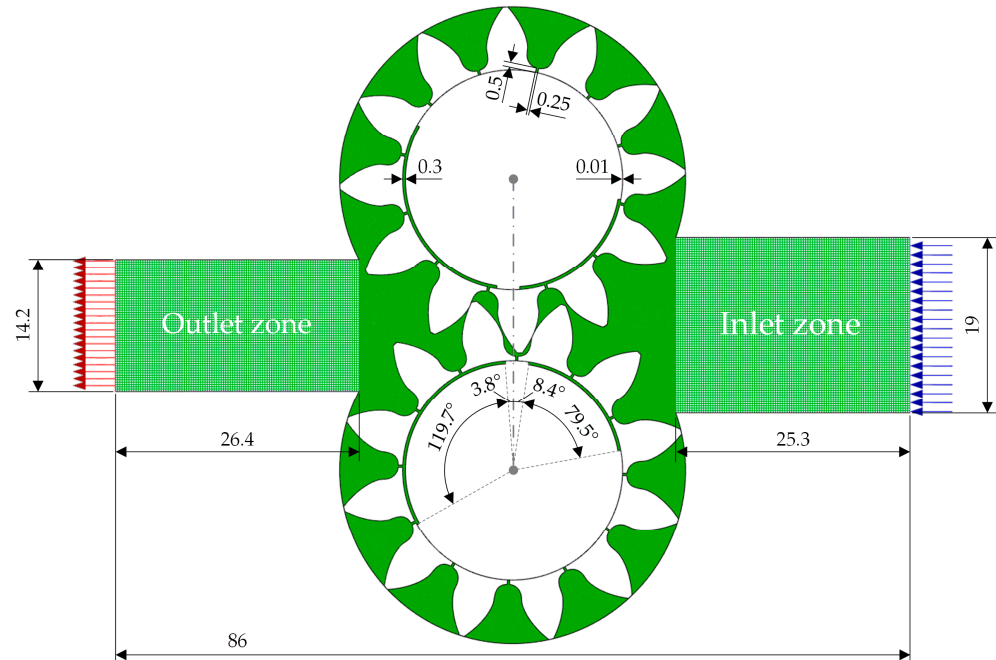


Figure 6. Advanced 2D model with relief grooves composed of radial (0.5×0.25 mm) and tangential (119.7° and 79.5°) channels. Lengths are given in millimeters.

3. CFD Model

3.1. Theoretical Details of the CFD Model Development

The following assumptions were made when creating the model:

- The flow process is 2D;
- The working fluid is *Newtonian*;
- The working fluid is incompressible;
- The Body forces are negligible;
- The viscous heating is not considered.

The CFD model realization is based on the main terms from computational fluid mechanics [53,54] integrated in ANSYS Fluent®.

The *Continuity equation* for compressible working fluid can be present in the following form:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0, \quad (1)$$

where ρ is a density of working fluid, ∇ is divergence and $\vec{V}$ is a velocity vector. For the steady state of compressible working fluid $\frac{\partial}{\partial t} = 0$ for any variables, from which it follows that

$$\nabla \cdot (\rho \vec{V}) = 0. \quad (2)$$

However, the working fluid is incompressible, $\rho = \text{const}$, therefore $\frac{\partial \rho}{\partial t} \cong 0$. In this case, the *continuity equation* can be expressed as

$$\nabla \times \vec{V} = 0. \quad (3)$$

For the incompressible fluid, the formal form is

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0, \quad (4)$$

where u , v , and w are the components of the velocity.

In a *Cartesian* coordinate system, the *Navier–Stokes* equations for incompressible fluid have a main vector form:

$$\rho \left(\frac{\partial \vec{V}}{\partial t} + \left(\vec{V} \cdot \nabla \right) \vec{V} \right) = -\nabla P + \rho \vec{g} + \mu \nabla^2 \vec{V}, \quad (5)$$

where P is the pressure, g is the gravity, and μ is the dynamic viscosity. It is valid for the second assumption (*Newtonian* fluid), which can be expressed for each of the coordinate axes:

The X component of the equation is

$$\rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = -\frac{\partial P}{\partial x} + \rho g_x + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right). \quad (6)$$

The Y component of the equation is

$$\rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = -\frac{\partial P}{\partial y} + \rho g_y + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right), \quad (7)$$

where g_x and g_y are the x and y directions of gravity. The Z component of the *Navier–Stokes* equation is not applicable to the first assumption (2D model).

The turbulence k - ε (kinetic energy) equations take the following general form:

$$\frac{\partial}{\partial t}(\rho k) + \nabla \cdot \left(\rho k \vec{V} \right) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k, \quad (8)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \nabla \cdot \left(\rho \varepsilon \vec{V} \right) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon, \quad (9)$$

where k is the kinetic energy, σ_k is the turbulent *Prandtl* number for k , ε is the kinetic energy dissipation rate, σ_ε is the turbulent *Prandtl* number for ε , G_k is the generation of turbulence kinetic energy (mean velocity gradients), G_b is the generation of turbulence kinetic energy due to buoyancy, Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, S_k is the user-defined source terms, and $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$, and C_μ are constants.

The dilatation dissipation term Y_M is

$$Y_M = 2\rho \varepsilon M_t^2, \quad (10)$$

where M_t is the turbulent *Mach* number.

The turbulence (*Eddy*) viscosity μ_t can be expressed by combining k and ε :

$$\mu_t = \rho C_\mu \frac{k}{\varepsilon}. \quad (11)$$

The present study is a continuation of study [3], where a rationale for choosing the k - ε turbulent model, main parameters, and settings of the model are described in detail. The new and specific moments are only commented on here. The CFD software used is ANSYS Fluent® ver. 2019 R3.

3.2. Finite Element Mesh

The used finite element mesh is shown in Figure 7. A basic mesh size of 0.14 mm was used for the initial meshing. The mesh is refined in the areas with gaps, with sizes between 0.013 mm and 0.025 mm for the involutes and contacting walls of the tangential grooves; 0.034 mm—for rounding; 0.05 mm—for the teeth tips; and 0.08 mm—for the walls. The static zones (inlet zone and outlet zone in Figure 6) have a structured mesh of rectangles with a size of 0.25 mm. During the solution, the mesh is dynamically changed. The remeshing parameters are set to minimum length scale 0.0007 mm; maximum length scale 0.32 mm; and maximum mesh skewness 0.67. Thus, after the first steps of the solution, some of the mesh cells become larger, and the general number of cells decreases from 79,821 to 65,401; however, the cells remain smaller near the walls that come into contact.

The chosen mesh settings have a significant impact on the overall solution. To a large extent, they are the result of a mesh-independent study carried out in our previous research [3]. The described mesh is fine enough to provide a detailed and accurate solution, while being economical in terms of computational resources. Besides the number of cells being relatively small for the modeled geometry, the mesh allows the time step to be set in the interval from 1×10^{-6} s to over 4×10^{-6} s (at a nominal rotation frequency of 1450 min^{-1}). An average of 1024 steps were used to turn the wheels by 30° .

Due to the large lengths and small gaps of the contact areas, as well as due to the presence of sharp transitions between the narrow and wide sections that constantly change their position, the solution is significantly more unstable than the classical 2D model. It often crashes due to Floating Point Exception. To deal with the problem, the following measures were used: varying the time step (from 2 to 4.2×10^{-6} s), activating the High Order Term Relaxation option with Relaxation Factor 0.75, and increasing the number of iterations from 20 to 35 for every step. The latter increased the computation time by about 25%. The mesh layering method is also enabled with the default settings (height-based, split factor 0.4, and collapse factor 0.2).

The computing time for the described model is of the order of 40 min/ 30° on a workstation with an Intel® Core™ i9-14900K (Intel Corporation, Santa Clara, CA, USA) processor and 64 GB DDR5 RAM.

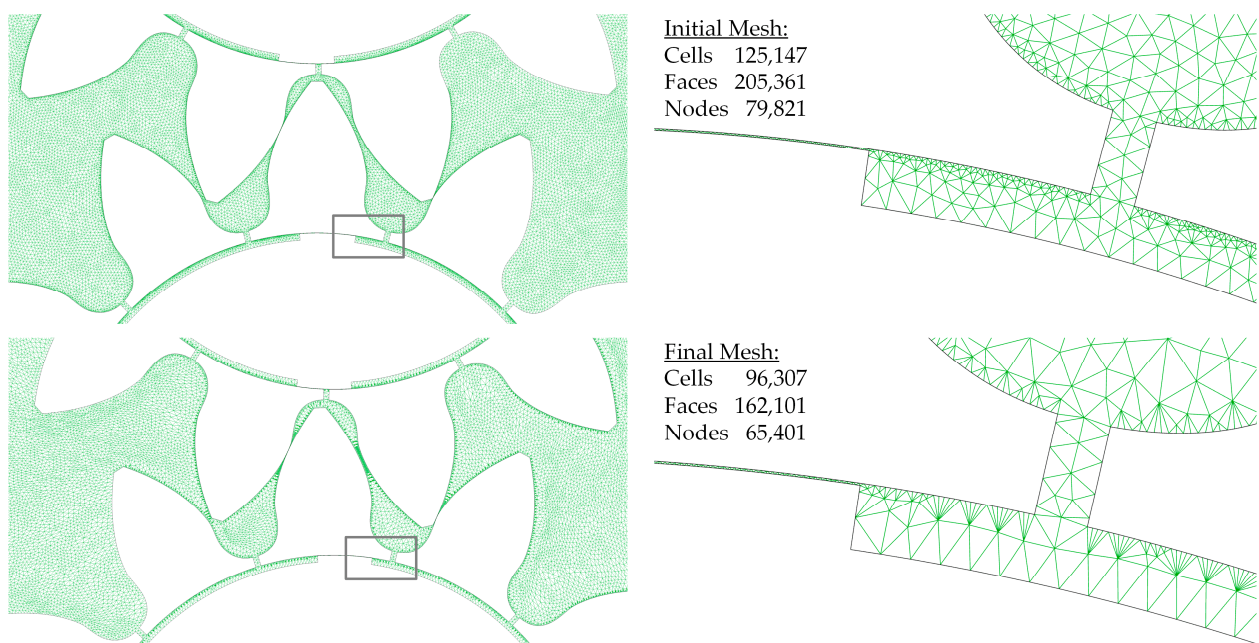


Figure 7. Finite elements initial and final mesh view. Zoomed areas are marked with rectangles.

4. Numerical Results and Discussion

The main focus of this study is on the volumetric flow rate q , L/min, and pressure Δp , bar. The first variable gives information about the flow rate ripple, which can cause undesirable pulsations, noise, and vibrations in the discharge pipeline of the pump, which the authors plan to investigate in the future. The second variable (pressure) well illustrates the benefits of the advanced 2D model. Additionally, the velocity distribution illustrates well the fluid movement through the added channels at different times of the simulation. Of course, the fluid transport through the added channels deforms the velocity field in the inter-teeth zones, so this model is not suitable for velocity analysis near the gears.

A comparison between the volumetric flow rate obtained with a classical and an advanced 2D model is shown in Figure 8. The simulation covers a 125° angle of gear rotation. Because of the established periodicity, it is sufficient to consider an interval of 30° to study the ongoing effects. The first thirty-degree interval from the beginning of the simulation is skipped to form the velocity field at the inlet and outlet zones, so the next second interval is considered. The initial and final angles of this interval (28.5° – 58.5°) are chosen in such a way that they start and end with the average value of the flow rate.

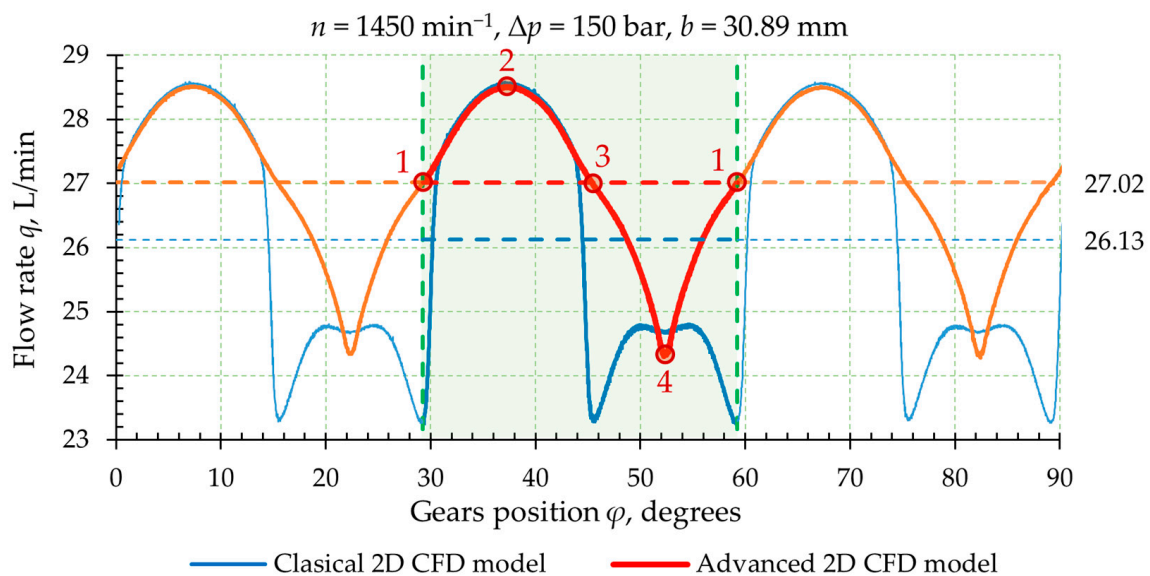


Figure 8. Volumetric flow rate obtained with classical and advanced 2D model.

Figure 8 illustrates the flow rate-related benefits of adding channels to the 2D model:

- The new shape of the $q(t)$ characteristic corresponds well to the classical theory [1].
- The average flow rate increases compared to the classic model from 26.13 to 27.02 L/min, calculated with the actual measured tooth width b of 30.89 mm. The new result is reliable and eliminates the need for experiment-based correction of b . The increased average flow rate is due to fluid flow through the channels in the direction from the inlet to the outlet, which is explained in detail below.
- Along the line 1-2-3, the two models' characteristics $q(\varphi)$ practically coincide. In this section, the gear mesh zone has a single point of contact (marked with a red circle in Figure 9, position 2), the mesh zone is opened to the surrounding spaces, and the influence of the grooves is minimal. Adding a gap between the fixed and movable walls outside the tangential channels results in a negligible decrease in the maximum value in position 2.
- On the 3-4-1 line, the gears mesh zone has two contact points (Figure 9, positions 1 and 3). The connection between the volume trapped by them and the surrounding spaces is made via the added channels. This leads to a large difference in the $q(\varphi)$ curves compared to the classical 2D model. The fluid flow through the channels is in the direction of the outlet; accordingly, the advanced model curve is located above

the classical one, which positively affects the average flow rate. The relatively small correction in the α_4 angle (shown in Figure 5d) results in a small gap leakage in the vicinity of position 4. Therefore, at position 4, the advanced 2D model flow rate is slightly lower than the classical one.

The positions of the gears at each of the points marked in Figure 8 are shown in Figure 9. These positions have the following particularities:

- **Position 1.** The gears are located symmetrically with respect to the axis connecting their centers of rotation, which passes through the inter-teeth zone of the upper gear and the tooth of the lower gear. After this point, only one point of contact remains in the mesh.
- **Position 2.** The single contact point is located on the axis. This is the moment with maximum open inter-teeth zones, minimum fluid return to the inlet, and maximum instantaneous pump flow rate.
- **Position 3.** The gears are located symmetrically with respect to the axis, but it passes through a tooth of the upper gear and an inter-teeth zone of the lower gear. After this moment, the gears contact at two points, and the trapped fluid between them is transferred back to the inlet, which reduces the instantaneous pump flow rate. The effect is significantly more obvious in the classical 2D model, without channels.
- **Position 4.** The two contact points are equidistant from the axis; the trapped volume is not in contact with the channels, but the distance between the radial channels of the trapped volume and the tangential channels is minimal. This position is characterized both by maximum fluid transfer back to the inlet port and by maximum losses via the channels' gaps. Therefore, the instantaneous pump flow rate is minimal.

The velocity distribution shown in Figure 9 illustrates the flow through the advanced model channels. Of particular importance is the axis connecting the centers of rotation of the two gears. The role of the channels is greatest when the gears are meshed in two contact points, and the trapped fluid is transferred back to the low-pressure zone. In this case, if the trapped volume is located predominantly to the right of the axis (for example, position 1 in Figure 9), it constantly increases, drawing fluid from the inlet zone. If the trapped volume is located predominantly to the left of the axis (for example, position 3 in Figure 9), it constantly decreases, pushing the fluid from the inter-teeth zone through the channels to the outlet zone. In both cases, there is an additional fluid movement through the channels in the direction from the suction to the discharge port, which leads to a general increase in the average flow rate compared to the classical 2D model.

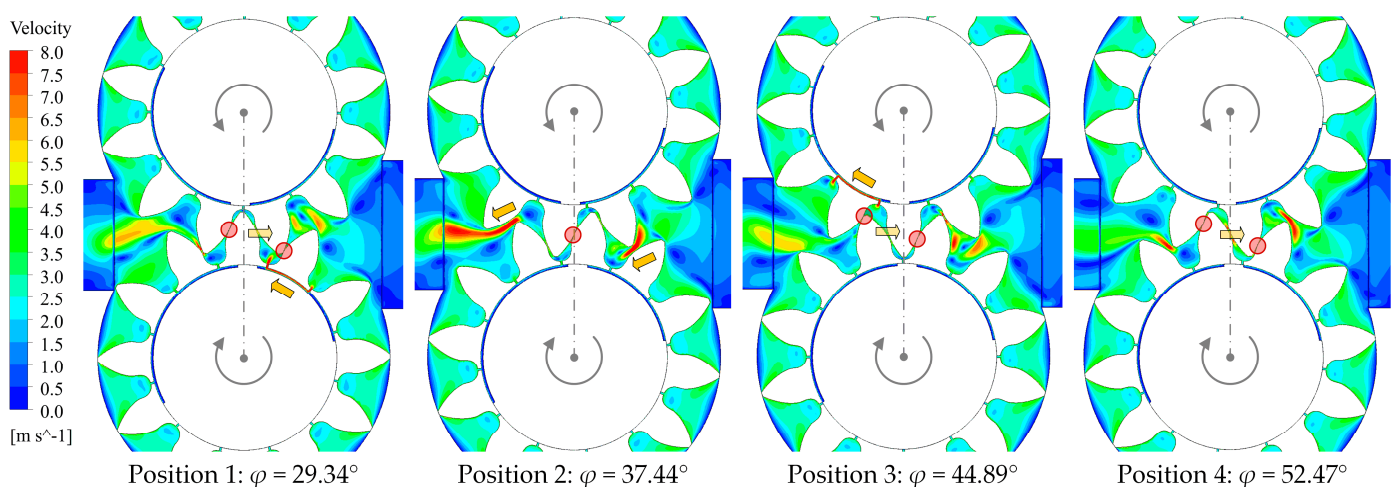


Figure 9. Velocity distribution at four rotation positions of gears. The points of contact are marked with red circles. Some important fluid flow directions are marked with yellow arrows.

Figure 10 shows the fluid pressure distribution obtained with consideration of the relief grooves. The pressure in all the inter-teeth zones is equalized with that in the contacting grooves. Figure 10a shows the position where the trapped volume does not contact the surrounding spaces through the added channels, and the pressure in it differs from that of the inlet and outlet zones. This position actually separates the outlet from the inlet. Figure 10b,c show positions very close to position 4, where, however, the pressure in the trapped volume is equalized with that of the outlet or inlet, which is not possible with the classical 2D model. Emphasis is placed on the contact of the teeth with the actual grooves, from where pressure equalization occurs.

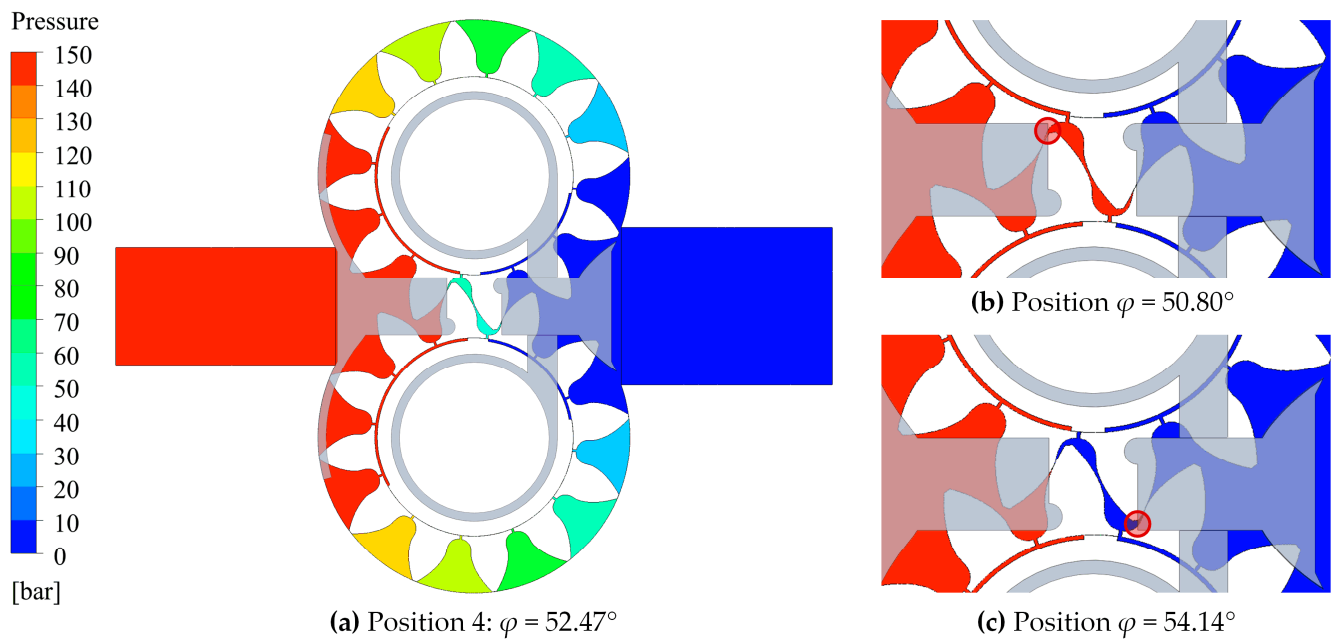


Figure 10. Pressure distribution and relief grooves in positions with minimum instantaneous flow rate. The actual contact zones between the trapped volume and the relief grooves are marked with red circles: (a) Position 4; (b) The inter-teeth zone connected to the outlet zone; (c) The inter-teeth zone connected to the inlet zone.

Figure 11a shows the influence of the rotation frequency n , min^{-1} , and Figure 11b presents the influence of the output pressure Δp on the current and average values of the pump flow rate q . The curves were obtained by simulations with the advanced 2D model. The average values of the flow rate q for all investigated operating modes are shown in the figure, as they are further used to validate the obtained results. It can be seen that with a decrease in n and with an increase in Δp , the average value of the flow rate q decreases. The form and the period of the characteristic $q(\varphi)$ are preserved and practically do not depend on the rotation frequency and the load. The variation of the average flow rate q when changing the rotation frequency n from 1450 to 950 min^{-1} is high (-35.5%) and is proportional to n , while when changing the load from 5 to 150 bar, it is low (-2.5%) and has a small influence from the working pressure. Practically the same reductions in the average flow rate are obtained with the classic 2D model [3], and the differences between the classical and the advanced 2D model here are less than 1%.

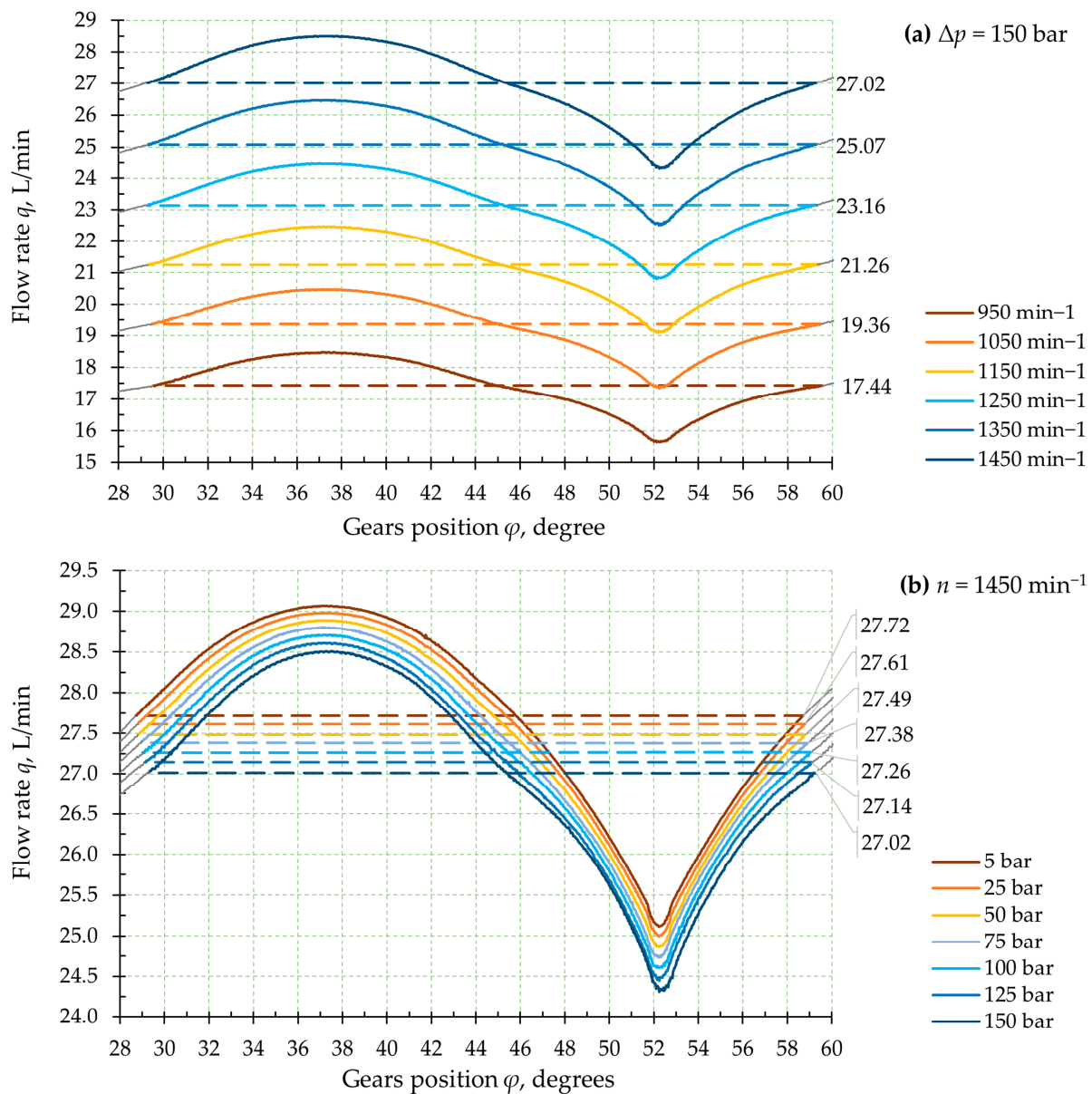


Figure 11. Simulated flow rate: (a) $\Delta p = 150$ bar and $n = 950\text{--}1450$ min^{-1} ; (b) $n = 1450$ min^{-1} and $\Delta p = 5\text{--}150$ bar.

5. Experimental Study for Validation

A laboratory experimental setup has been developed and used for testing an external gear pump at different operating modes. Figure 12 presents the hydraulic circuit diagram of the experimental system and its realization. A detailed description of the shown system is given in [3].

The developed experimental setup makes it possible for a real-time measurement of the gear pump basic working parameters: flow rate, output pressure and rotation frequency. The system consists of the studied pump, driven by an electric motor with a power of 7.5 kW and a nominal rotation frequency of 1450 min^{-1} . The electric motor is equipped with a frequency inverter enabling the pump rotation frequency to be changed, and hence the flow rate. The pump outlet pressure is varied by a loading system consisting of a direct-operated pressure relief valve and a parallel-connected hydraulic throttle check valve. The maximum pressure value is set with the pressure-relief valve and the pressure load value is set with the precision throttle valve for a certain mode of operation.

Flow rate and pressure are measured using a gear flow meter equipped with a pressure sensor. The signals from the two sensors are processed by a developed data acquisition (DAQ) system which is based on NI USB 6211. For signals recording and processing, a user interface has been developed in LABView® ver. 2010 environment. The recorded in a table form experimental data are processed automatically via a specially developed program procedure in MATLAB® ver. 2009.

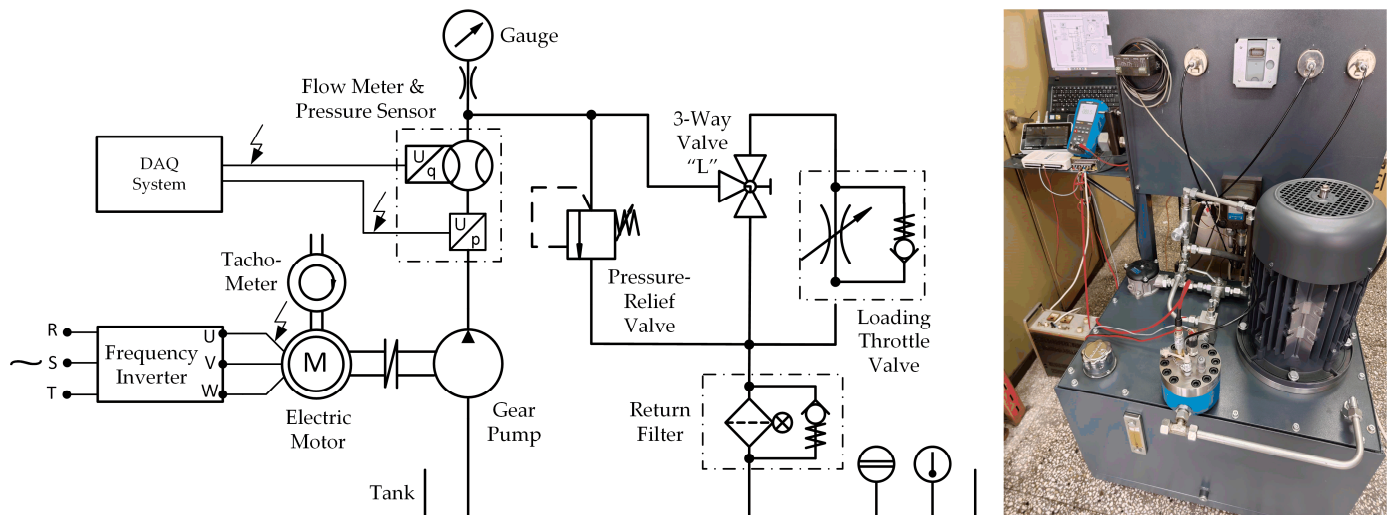


Figure 12. Experimental setup—hydraulic circuit diagram and implementation.

Experimental studies were carried out with the described setup in order to validate the results of the numerical studies based on the advanced 2D CFD model. Experimental and numerical studies were performed in two groups of operation modes. In the first group, a maximum output pressure load of 150 bar is maintained ($\Delta p = \text{const}$), and the rotation frequency is a variable parameter in the range of 950–1450 min^{-1} . The experiment design and results are shown in Table 1. A results comparison via the characteristic of the average pump flow rate q_{AVG} as a function of the rotation frequency n is shown in Figure 13a. In order to distinguish the designation of the flow rate obtained by the experiment from that obtained numerically, the indices *EXP* and *CFD* have been added to the q_{AVG} in Tables 1 and 2.

In the second group of operation modes, a constant nominal value of the rotation frequency is maintained ($n = \text{const}$), and the pressure load is a variable parameter in the range of 5–150 bar. The experiment design and results for this configuration are shown in Table 2. A results comparison via the characteristic of the average pump flow rate q_{AVG} as a function of the pressure Δp is shown in Figure 13b.

Estimation of the percentage match between the model and experiment results is performed by calculating the quantitative index *FIT*. It is defined as

$$FIT = \left(1 - \frac{|q_{AVG,EXP} - q_{AVG,CFD}|}{q_{AVG,EXP}} \right) 100, \% \quad (12)$$

where $q_{AVG,EXP}$ is the experimentally obtained average flow rate of the pump, and $q_{AVG,CFD}$ is the simulation average flow rate obtained by the advanced 2D CFD model.

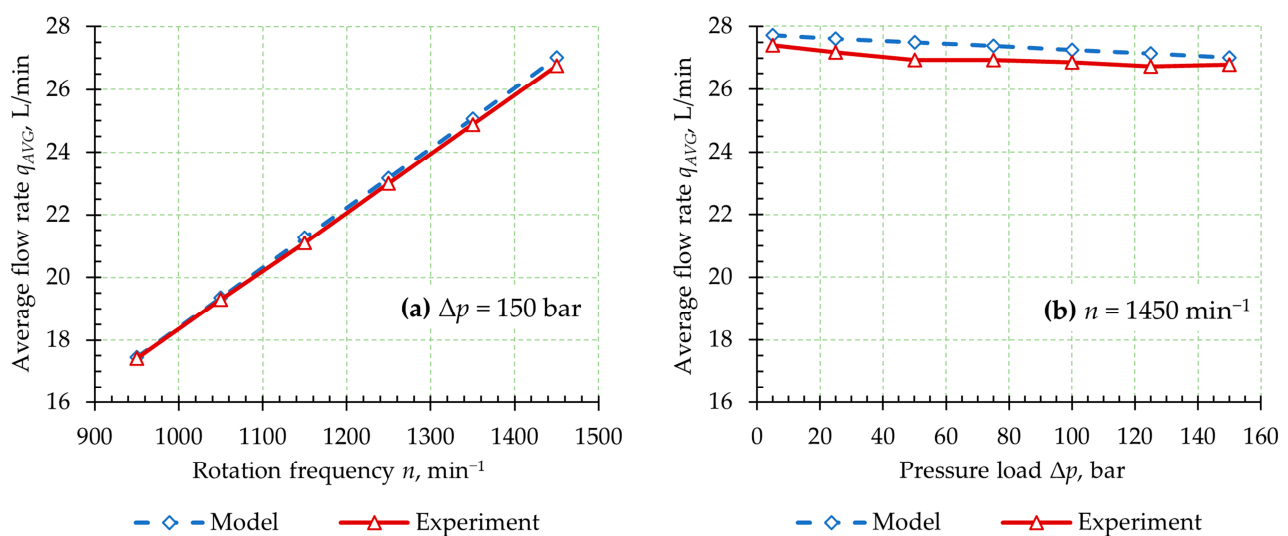
The graphical comparison between the numerical and real experiment results (Figure 13) shows an excellent match for all operating modes. This is once again confirmed by the calculated quantitative indicator *FIT*, of which the value varies in the range of 97.93–99.82% (Tables 1 and 2). It is possible to achieve an even better match by fine adjustment to the imaginary meshing gap value described in Section 2, but the currently achieved match proves that the developed advanced 2D CFD model successfully considers the design features of this pump type, and especially the influence of the relief grooves on the flow process.

Table 1. Model and experiment results, $\Delta p = \text{const.}$

Model			Experiment			FIT
n	Δp	$q_{AVG,CFD}$	n	Δp	$q_{AVG,EXP}$	
min^{-1}	bar	L/min	min^{-1}	bar	L/min	%
950	150	17.44	950	147.86	17.41	99.82
1050	150	19.36	1051	147.67	19.29	99.70
1150	150	21.26	1148	148.23	21.11	99.27
1250	150	23.17	1249	148.00	23.00	99.29
1350	150	25.07	1348	147.81	24.87	99.17
1450	150	27.02	1451	146.96	26.75	98.94

Table 2. Model and experiment results, $n = \text{const.}$

Model			Experiment			FIT
n	Δp	$q_{AVG,CFD}$	n	Δp	$q_{AVG,EXP}$	
min^{-1}	bar	L/min	min^{-1}	bar	L/min	%
1450	5	27.72	1450	6.43	27.41	98.88
1450	25	27.61	1451	23.46	27.19	98.43
1450	50	27.49	1447	49.78	26.94	97.93
1450	75	27.38	1452	74.68	26.93	98.34
1450	100	27.26	1452	101.74	26.85	98.49
1450	125	27.14	1448	124.50	26.74	98.51
1450	150	27.02	1452	148.63	26.79	99.16

**Figure 13.** CFD and experimental results comparison: (a) $q_{AVG}(n)$; (b) $q_{AVG}(\Delta p)$.

6. Conclusions

The main contribution of this article is the original design of an advanced 2D CFD model of the external gear pump. Beyond the possibilities of real construction, this model successfully considers the influence of relief grooves, which is traditionally possible with 3D CFD models only. Radial and tangential channels are added to the classical 2D model geometry, which successfully represents the connections between inlet, inter-teeth, and outlet zones. The values of the numerically obtained average flow rate match the experimentally measured ones very well without applying any additional geometric or computational modifications. Modeling in 2D is more economical in terms of computational resources, allowing multiple results to be obtained quickly in different operating modes. In addition, the presented model can become part of more complex hydraulic system models in order

to study the effects of pump operation on the velocity and pressure in discharge pipelines, which is our next goal.

Author Contributions: Conceptualization, N.N., A.M. and I.K.; methodology, A.M. and N.N.; CFD model, N.N.; validation, A.M.; formal analysis, N.N. and A.M.; investigation, A.M. and N.N.; resources, A.M. and N.N.; data curation, N.N. and A.M.; writing—original draft preparation, A.M. and N.N.; writing—review and editing, A.M., N.N. and I.K.; visualization, N.N.; supervision, A.M.; project administration, I.K.; funding acquisition, I.K. All authors have read and agreed to the published version of the manuscript.

Funding: This work has been accomplished with financial support by the European Regional Development Fund within the Operational Programme “Bulgarian national recovery and resilience plan”, procedure for direct provision of grants “Establishing of a network of research higher education institutions in Bulgaria”, and under Project BG-RRP-2.004-0005 “Improving the research capacity and quality to achieve international recognition and resilience of TU-Sofia (IDEAS)”.

Data Availability Statement: The raw data supporting the conclusions of this article will be made available by the authors on request.

Acknowledgments: This work has been accomplished with financial support by the European Regional Development Fund within the Operational Programme “Bulgarian national recovery and resilience plan”, procedure for direct provision of grants “Establishing of a network of research higher education institutions in Bulgaria”, and under Project BG-RRP-2.004-0005 “Improving the research capacity and quality to achieve international recognition and resilience of TU-Sofia (IDEAS)”.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Findeisen, D.; Helduser, S. *Ölhydraulik*; Springer: Berlin/Heidelberg, Germany, 2015.
2. Ivantysyn, J.; Ivantysynova, M. *Hydrostatic Pumps and Motors: Principles, Design, Performance, Modelling, Analysis, Control and Testing*; Academia Books International: New Delhi, India, 2001.
3. Mitov, A.; Nikolov, N.; Nedelchev, K.; Kralov, I. CFD Modeling and Experimental Validation of the Flow Processes of an External Gear Pump. *Processes* **2024**, *12*, 261. [\[CrossRef\]](#)
4. Stan, L.C.; Gordes, A.N.; Agape, A.G. Numerical simulation of a gear pump. *Int. J. Mod. Manuf. Technol.* **2023**, *15*, 107–114.
5. Patil, S. Numerical Simulation of Multi-Dimensional Flows in a Gear Pump. Master’s Thesis, Youngstown State University, Youngstown, OH, USA, 2006.
6. Maccioni, L.; Concli, F. Computational fluid dynamics applied to lubricated mechanical components: Review of the approaches to simulate gears, bearings, and pumps. *Appl. Sci.* **2020**, *10*, 8810. [\[CrossRef\]](#)
7. Mali, P.S.; Joshi, G.S.; Patil, I.A. Performance improvement of external gear pump through CFD analysis. *Int. Res. J. Eng. Technol.* **2018**, *5*, 430–433.
8. Del Campo, D. Analysis of the Suction Chamber of External Gear Pumps and Their Influence on Cavitation and Volumetric Efficiency. Ph.D. Thesis, Universitat Politècnica de Catalunya, Barcelona, Spain, 2012.
9. Munih, J.; Hočevár, M.; Petrič, K.; Dular, M. Development of CFD-based procedure for 3D gear pump analysis. *Eng. Appl. Comput. Fluid Mech.* **2020**, *14*, 1023–1034. [\[CrossRef\]](#)
10. Manring, N.D.; Kasaragadda, S.B. The theoretical flow ripple of an external gear pump. *J. Dyn. Syst. Meas. Control Trans. ASME* **2003**, *125*, 396–404. [\[CrossRef\]](#)
11. Choudhuri, K.; Biswas, N.; Mandal, S.K.; Mitra, C.; Biswas, S. A numerical study of an external gear pump operating under different conditions. *Mater. Today Proc.* **2022**; *in press*.
12. Zhao, X.; Vacca, A.; Dhar, S. Numerical Modeling of a Helical External Gear Pump with Continuous-Contact Gear Profile: A Comparison between a Lumped-Parameter and a 3D CFD Approach of Simulation. In Proceedings of the BATH/ASME 2018 Symposium on Fluid Power and Motion Control FPMC2018, Bath, UK, 12–14 September 2018.
13. Orlandi, F.; Muzzioli, G.; Milani, M.; Paltrinieri, F.; Montorsi, L. Development of a numerical approach for the CFD simulation of a gear pump under actual operating conditions. *Fluids* **2023**, *8*, 244. [\[CrossRef\]](#)
14. Corvaglia, A.; Rundo, M.; Bonati, S.; Rigosi, M. Simulation and experimental activity for the evaluation of the filling capability in external gear pumps. *Fluids* **2023**, *8*, 251. [\[CrossRef\]](#)
15. Gafurov, S.; Rodionov, L.; Makaryants, G. Simulation of Gear Pump Noise Generation. In Proceedings of the 9th FPNI Ph.D. Symposium on Fluid Power FPNI 2016, Florianópolis, Brazil, 26–28 October 2016.
16. Tang, C.; Wang, Y.S.; Gao, J.H.; Guo, H. Fluid-sound coupling simulation and experimental validation for noise characteristics of a variable displacement external gear pump. *Noise Control Eng. J.* **2014**, *62*, 123–131. [\[CrossRef\]](#)

17. Fiebig, W. Influence of the inter teeth volumes on the noise generation in external gear pumps. *Arch. Acoust.* **2014**, *39*, 261–266. [[CrossRef](#)]
18. Mucchi, E.; Rivola, A.; Dalpiaz, G. Modelling dynamic behaviour and noise generation in gear pumps: Procedure and validation. *Appl. Acoust.* **2014**, *77*, 99–111. [[CrossRef](#)]
19. Woo, S.; Oppenwall, T.; Vacca, A.; Rigosi, M. Modeling noise sources and propagation in external gear pumps. *Energies* **2017**, *10*, 1068. [[CrossRef](#)]
20. Fiebig, W. Noise Control of Fluid Power Units. In Proceedings of the 23rd International Congress on Sound and Vibration: From Ancient to Modern Acoustics, Athens, Greece, 10–14 July 2016.
21. Carletti, E.; Pedrielli, F. Sound Power Levels of Hydraulic Pumps using Sound Intensity Techniques: Towards More Accurate Values? In Proceedings of the 12th International Congress on Sound and Vibration, Lisbon, Portugal, 11–14 July 2005.
22. Osiński, P.; Deptuła, A.; Deptuła, A.M. Analysis of the Gear Pump's Acoustic Properties Taking into Account the Classification of Induction Trees. *Energies* **2023**, *16*, 4460. [[CrossRef](#)]
23. ISO 16902-1:2003; Hydraulic Fluid Power—Test Code for the Determination of Sound Power Levels of Pumps using Sound Intensity Techniques: Engineering Method—Part 1: Pumps. ISO: Geneva, Switzerland, 2003.
24. Mitov, A.; Nedelchev, K.; Kralov, I. Experimental Study of Sound Pressure Level in Hydraulic Power Unit with External Gear Pump. *Processes* **2023**, *11*, 2399. [[CrossRef](#)]
25. Yoon, Y.; Park, B.-H.; Shim, J.; Han, Y.-O.; Hong, B.-J.; Yun, S.-H. Numerical simulation of three-dimensional external gear pump using immersed solid method. *Appl. Therm. Eng.* **2017**, *118*, 539–550. [[CrossRef](#)]
26. Corvaglia, A.; Ferrari, A.; Rundo, M.; Vento, O. Three-dimensional model of an external gear pump with an experimental evaluation of the flow ripple. *Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci.* **2021**, *235*, 1097–1105. [[CrossRef](#)]
27. Frosina, E.; Senatore, A.; Rigosi, M. Study of a high-pressure external gear pump with a computational fluid dynamic modeling approach. *Energies* **2017**, *10*, 1113. [[CrossRef](#)]
28. Močilan, M.; Husár, Š.; Labaj, J.; Žmindák, M. Non-Stationary CFD Simulation of a Gear Pump. In Proceedings of the 21st International Polish-Slovak Conference “Machine Modeling and Simulations 2016”, Hucisko, Poland, 8–11 September 2017.
29. Jędraszczyk, P.; Fiebig, W. CFD model of an external gear pump. In *RESRB 2016: Proceedings of the 13th International Scientific Conference, Wrocław, Poland, 22–24 June 2016*; Springer: Berlin/Heidelberg, Germany, 2017; pp. 221–231.
30. Castilla, R.; Gamez-Montero, P.J.; Del Campo, D.; Raush, G.; Garcia-Vilchez, M.; Codina, E. Three-dimensional numerical simulation of an external gear pump with decompression slot and meshing contact point. *J. Fluids Eng. Trans. ASME* **2015**, *137*, 041105. [[CrossRef](#)]
31. Corvaglia, A.; Rundo, M.; Casoli, P.; Lettini, A. Evaluation of tooth space pressure and incomplete filling in external gear pumps by means of three-dimensional CFD simulations. *Energies* **2021**, *14*, 342. [[CrossRef](#)]
32. Ghionea, I.G. Applied Methodology for Designing and Calculating a Family of Spur Gear Pumps. *Energies* **2022**, *15*, 4266. [[CrossRef](#)]
33. Torrent, M.; Gamez-Montero, P.J.; Codina, E. Model of the Floating Bearing Bushing Movement in an External Gear Pump and the Relation to Its Parameterization. *Energies* **2021**, *14*, 8553. [[CrossRef](#)]
34. Strasser, W. CFD Investigation of Gear Pump Mixing. In Proceedings of the ASME International Mechanical Engineering Congress and Exposition, Orlando, FL, USA, 5–11 November 2005.
35. Strasser, W. CFD investigation of gear pump mixing using deforming/agglomerating mesh. *J. Fluids Eng. Trans. ASME* **2007**, *129*, 476–484. [[CrossRef](#)]
36. Ghazanfarian, J.; Ghanbari, D. Computational fluid dynamics investigation of turbulent flow inside a rotary double external gear pump. *J. Fluids Eng. Trans. ASME* **2015**, *137*, 021101. [[CrossRef](#)]
37. Qi, F.; Dhar, S.; Nichani, V.H.; Srinivasan, C.; Wang, D.M.; Yang, L.; Bing, Z.; Yang, J.J. A CFD study of an Electronic Hydraulic Power Steering Helical External Gear Pump: Model Development, Validation and Application. *SAE Int. J. Passeng. Cars—Mech. Syst.* **2016**, *9*, 346–352. [[CrossRef](#)]
38. Li, X.; Zhang, L.; Zhang, Y. Numerical Study on the Influence of Different Operating Conditions on Mixing Uniformity of the Helical Gear Pump Mixer. *Processes* **2023**, *11*, 3223. [[CrossRef](#)]
39. Ouyang, T.; Mo, X.; Lu, Y.; Wang, J. CFD-vibration coupled model for predicting cavitation in gear transmissions. *Int. J. Mech. Sci.* **2022**, *225*, 107377. [[CrossRef](#)]
40. Killedar, J.S. CFD Analysis of Gear Pump. Master's Thesis, Youngstown State University, Youngstown, OH, USA, 2005.
41. Heisler, A.S.; Moskwa, J.J.; Fronczak, F.J. Simulated Helical Gear Pump Analysis using a New CFD Approach. In Proceedings of the ASME Fluids Engineering Division Summer Conference FEDSM2009, Vail, CO, USA, 2–6 August 2009.
42. Labaj, J.; Husar, S. Analysis of Gear Pump Designed for Manufacturing Processes. *Appl. Mech. Mater.* **2015**, *803*, 163–172. [[CrossRef](#)]
43. Lee, J.-H.; Lee, S.-W. Numerical Simulations of Cavitation Flow in Volumetric Gear Pump. *J. Korean Soc. Vis.* **2011**, *9*, 28–34.
44. Del Campo, D.; Castilla, R.; Raush, G.A.; Gamez Montero, P.J.; Codina, E. Numerical analysis of external gear pumps including cavitation. *J. Fluids Eng. Trans. ASME* **2012**, *134*, 081105. [[CrossRef](#)]
45. Zhou, W.; Yu, D.; Wang, Y.; Shi, J.; Gan, B. Research on the Fluid-Induced Excitation Characteristics of the Centrifugal Pump Considering the Compound Whirl Effect. *Facta Univ. Ser. Mech. Eng.* **2023**, *21*, 223. [[CrossRef](#)]

46. Bilalov, R.A.; Smetannikov, O.Y. Numerical Study of the Hydrodynamics of an External Gear Pump. *J. Appl. Mech. Tech. Phys.* **2022**, *63*, 1284–1293. [[CrossRef](#)]
47. Martínez, J. Mesh Handling for the CFD Simulation of External Gear Pumps. In *Positive Displacement Machines: Modern Design Innovations and Tools*, 1st ed.; Sultan, I., Phung, T., Eds.; Elsevier: Amsterdam, The Netherlands, 2019; pp. 345–368.
48. Ull, A.R. *Study of Mesh Deformation Features of an Open Source CFD Package and Application to a Gear Pump Simulation*; Enginyeria Aeronàutica, Universitat Politècnica De Catalunya: Barcelona, Spain, 2012.
49. Mucchi, E.; Dalpiaz, G.; Rivola, A. Dynamic behavior of gear pumps: Effect of variations in operational and design parameters. *Meccanica* **2011**, *46*, 1191–1212. [[CrossRef](#)]
50. Marinaro, G.; Frosina, E.; Senatore, A. A Numerical Analysis of an Innovative Flow Ripple Reduction Method for External Gear Pumps. *Energies* **2021**, *14*, 471. [[CrossRef](#)]
51. Adake, D.G.; Dhote, N.D.; Khond, M.P. Experimentation and 2D Fluid Flow Simulation over an External Gear Pump. *J. Phys. Conf. Ser.* **2023**, *2601*, 012029. [[CrossRef](#)]
52. Szwemin, P.; Fiebig, W. The Influence of Radial and Axial Gaps on Volumetric Efficiency of External Gear Pumps. *Energies* **2021**, *14*, 4468. [[CrossRef](#)]
53. Cengel, Y.A.; Cimbala, J.M. *Fluid Mechanics: Fundamental and Applications*; McGraw-Hill: New York, NY, USA, 2013.
54. Ansys Fluent. *12.0 Theory Guide*; Ansys Inc.: Canonsburg, PA, USA, 2009.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.