Recent Advances of Flow Rate Simulation in Gear Pumps

Ruihan Wu
Ulink College of Shanghai, Shanghai, China
15000440219@xs.hnit.edu.cn

Abstract. The gear pump is a rotating pump that moves and changes depending on the working volume that is created between the gear and the pump cylinder to transport or pressurize the liquid. It is one of the most widely used hydraulic pumping devices. It has been adopted in road equipment, conveyor systems, and other industrial machineries. Its performance is critical to the engineering equipment. With the development of fluid mechanics, the computational fluid dynamic (CFD) technique is widely used to predict the internal flow filed of gear pumps. CFD can provide researchers more intuitive observation and analysis to help the designers to enhance the effectiveness of gear pump. This paper summarizes the principle of flow simulation methods for gear pump and its applications. The research results and key findings are summarized. Defects of existing research are pointed out and suggestions for future research are provided. To sum up, this paper has a certain reference value for researchers who would like to know the hydraulic simulation process for gear pumps.

Keywords: Gear pump, Flow simulation, Hydraulic device.

1. Introduction

In order to pressurize or transfer liquid, gear pumps use the movement and between the pump cylinder and the meshing gear, a change in the operating volume. Pump body, front and back covers, two gears, and two closed spaces make up the pump. Two equally sized gears intertwine and spin in a closely matched shell. The interior of the hull is similar to the "8" shape. As the wheel rotates, the amount of space on the side of the wheel changes from small to large, creating a void and fluid to inhale. As the region on the other side of the gear shrinks from a large to a small volume, the liquid is forced into the pipeline. When a tooth touches the fluid space of another tooth, the incompressible liquid is mechanically forced out since the tooth and the liquid cannot both occupy the same place at the same time. This occurrence happens continually as a result of the teeth's ongoing interaction. As a result, the pump's output offers a constant outflow. The pump discharges constantly while the driving shaft continues to rotate. The speed of the pump is closely correlated with its flow.

A kind of computational fluid dynamics known as "flow simulation" digitally imitates the flow of surface water or collecting systems. This is usually referred to as Computational Fluid Dynamics (CFD). Flow simulations are performed to predict and analyze the effects of complex geometry on flow in an operating environment. It can be used to predict the impact of changes on the system. The AMESim opens a new way of simulation method for gear pumps. The AMESim is more effective and easier to operate, with its powerful analytical ability, it has the advantage of being suitable for direct connection simulation of multi-collar model [1]. The basic principles relevant to hydraulic engineers are summarized in [2, 3], including velocity decomposition theorem, Kelvin's theorem, Helmholtz's theorem, etc. It is found that engineers should develop the ability to analyze and solve practical problems by qualitative and semi-quantitative methods using the simplest and most basic principles. Yang et al. [4] investigated the application case of hydraulic transformer It is found that hydraulic transformer is a new flow control element, which can drive line and rotating load efficiently by combining with hydraulic constant voltage network system. Yang et al. [5] investigated the advantages of water hydraulic technology. It is found that corrosion, wear, cavitation, sealing, lubrication and other difficulties exist in the research of water hydraulic technology and some is solved or will continue to develop in future study. Liu et al. [6] investigated the detailed processes of fluid simulation using computer animation physics. It is found that with the help of Euler and Lagrange methods, the control of detail strategy, acceleration strategy and control strategy can make the whole simulation more realistic and flexible. Li et al. [7] investigated Fluent, the universal software used in fluid mechanics. It is found that Fluent can be used to analyze the Couette flow between concentric spheres and shells, and calculate the stable T Aylo R vortex family and their bifurcation process.

The objective of this paper is to summarize the methods capable of simulating the flow inside the gear pumps. Section 2 categorize the existing methods and present typical applications of various methods in the flow simulation of gear pumps. Conclusion and prospects are presented in Section 3.

2. Main body

The existing methods capable of simulating the internal flow of gear pumps generally can be categorized into four groups: 0D, 1D, 2D and 3D method. In this section, typical research using each method are discussed.

2.1. 0D method

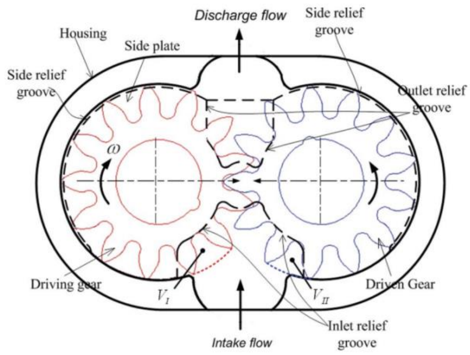

Figure 1. Typical gear pump section diagram

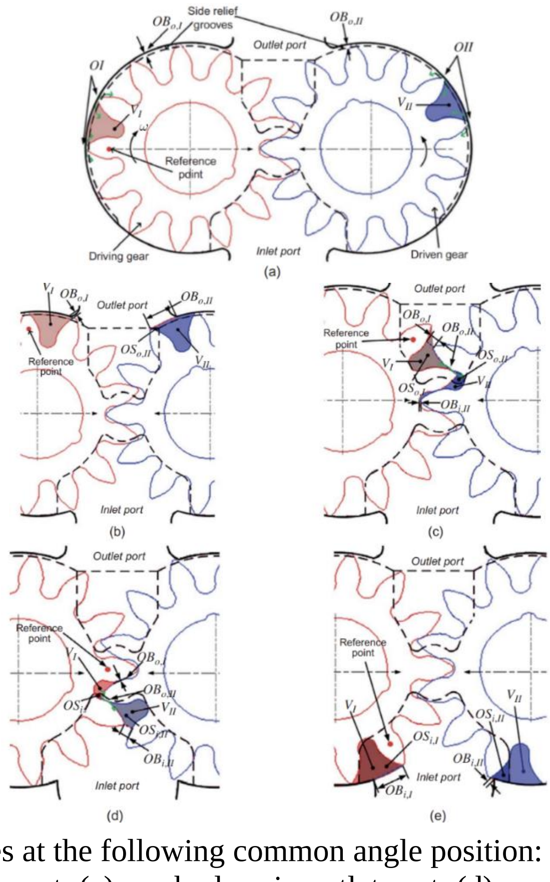

Figure 2. Check volumes at the following common angle position: (a) initial place (= 0); (b) opening place on the output port; (c) mesh place in outlet port; (d) mesh place in inlet port; and (e) aperture place toward outlet port

Wang et al. [8] developed the numerical modeling method based on the concept of centralized parameters and control volume is confirmed by the sample simulation and experimentation outcomes of a gear machine tool.

Wang et al. used the Generalized Control volume approach which is an abstraction from mathematics that describes the bodily functions of hydrodynamic systems and helps to model them. The control volume is surrounded by actual or imaginary surfaces that can be defined in terms of operating conditions and limits. In Figures 1 and 2, check volumes is a term for pockets.

The generalized control volume presented in Figure 3 is used to build a numerical model for a gear pump external with the same drive and drive gears. According to the angular positions, the CAD model in Figure 4 verifies the volumetric and geometric characteristics. Figure 5 shows the area of the rear multi-part gap.

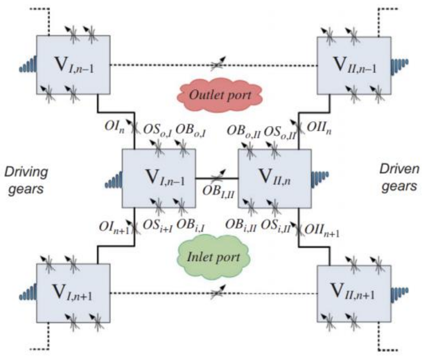

Figure 3. Block diagram of the general check volume of the gear pump

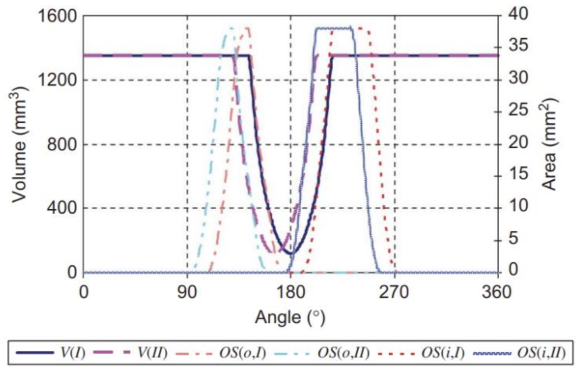

Figure 4. Side plates produce pocket volume and hole area

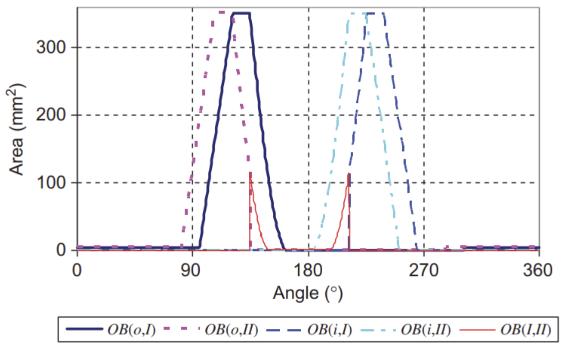

Figure 5. Area of intertooth clearance holes

When the pump is operating at constant speed, the pressure can be calculated as shown in Figure 6. Because of the overpressure groove between the lateral plate and the pump, compared to low pressure, high pressure remains in the pump chamber for a longer angular distance. Attention should be paid to the excitation of the peak pressure and the lowest pressure in the gear pump meshing area, because the key characteristics of the meshing area have a significant impact on the design of the gear pump. For succinct illustration, the turbulence through the hole is concentrated, as shown in Figure 7.

Figure 8 illustrates laminar flow through the clearance shows that the leakage along the tooth tip largely occurs where there is no pressure relief groove between the side plate and the housing. Figure 9 show the pump inlet and outlet flow with rotation Angle and frequency, respectively. The various dumping flow rates and volume change rates in the gear pocket are what generate the amplitude ripples of the two neighboring peaks of the flow.

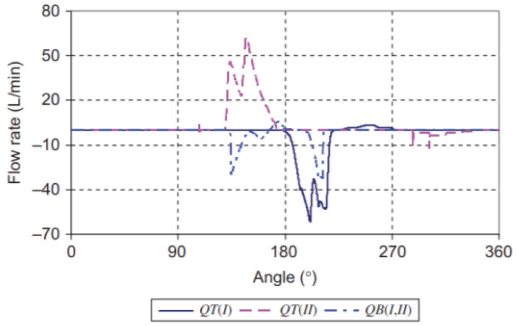

Figure 6. Shows how clumped turbulence flows through side plates and pockets with gaps

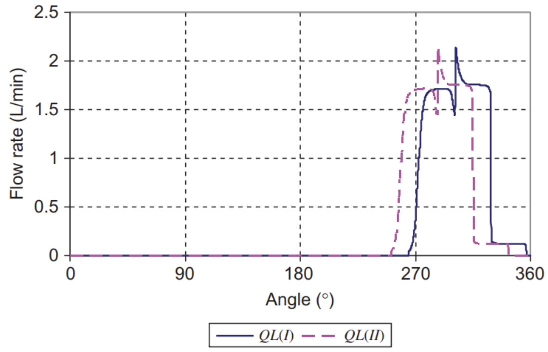

Figure 7. OI and OII experience laminar flow for pockets.V_I and V_{II}

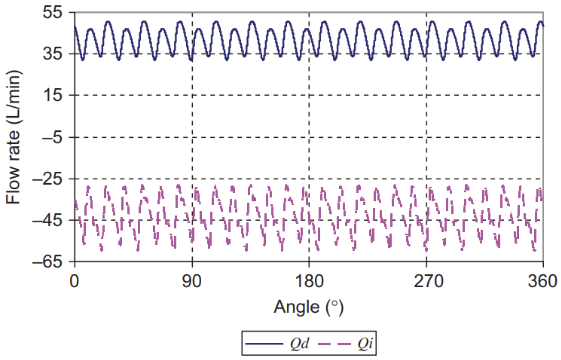

Figure 8. The gear pump's overall exit and input flow rates

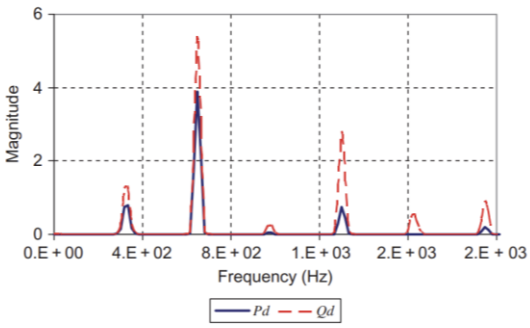

Figure 9. The pump's exit and the frequency response of the inlet channel

The two harmonies, which respectively occur at 325 Hz and 650 HZS in the frequency domain., are determined by the pump's gear ratio and shaft speed. In this instance, a second harmonic of 1300 Hertz indicates that the design must be altered in order to decrease wear and fluid propagation noise at the high frequency. Three distinct design side plates were employed, and the simulation and actual experiment were conducted in the gear pump to ensure that the simulation model was accurate. To address the issues with volume efficiency and noise, experiment with different side plate designs. Figure 10 displays the information that the three plates produced.

To lower the peak and noise levels, S1and S2 and S3 are attempting to increase the instantaneous flow area of the mesh area. However, increased internal traffic may result in a reduction in the pump's volumetric efficiency. The results of the pump volume efficiency test and simulation are shown in Figure 11. Under various operating pressures, there is a solid link between the efficiency trend curve. The side plates' noise test results show that the sound pressure level of S3 is decreased from the sound pressure level of S1 by a maximum of 4dB. Efficiency and noise are two issues that must be balanced while designing the side plate.

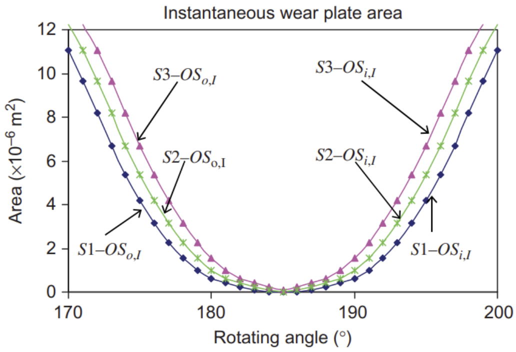

Figure 10. Spaces created by three side plates for orifices

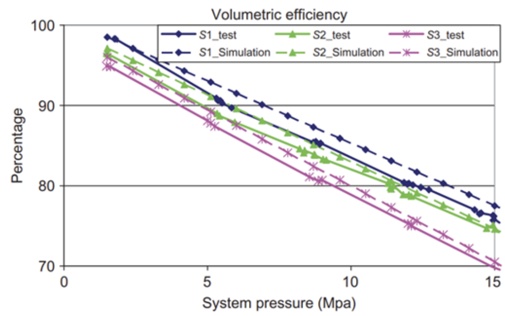

Figure 11. Efficiency in volume using three separate side plates

The hydraulic performance and dynamics of the generalized control volume external mesh gear pump have been numerically simulated, according to Wang et al. The idea of fluid control volume and throttling is expanded, and under plausible assumptions, the entire rotating process of gear machining is applied. The main response of the model can provide a large number of dynamic characteristics and correlation, and provide Suggestions for design decision and modification of gear pump.

2.2. 1D method

Harrison et al. [9] demonstrated the simplicity of modeling the rotor in 1D by automating the extraction of model inputs from the pump CAD file, which are crucial. Performance of boom pumps is anticipated and compared to experimental findings in terms of flow rate, loss of all power, efficiency in terms of volume, the relationship between total efficiency and pump speed, the rise in pressure, the tolerance for clearing, and the temperature. The rotor pump's friction prediction model is provided, and the overall torque necessary to drive the rotor pump is directly when compared to the outcomes of the experiments. In addition, the volumetric efficiency is predicted using an internal leakage model that takes into account the physical clearance and temperature of the inner and outer gears.

The schematic diagram of SHW rotor pump that will be studied in the paper that written by Harrison et al. The inner gear is a shaft-driven drive gear, whereas the outer gear is a driven gear. The inner gear of the thermal actuator rotates more slowly than the outer gear because one tooth is missing from the inner gear compared to the outer gear. As seen in Figure5 depicts depiction of a rotor pump's typical 1D flow network. The number of pump chambers is equal to the number of teeth on the outer gear since the entire rudder pumping cycle needs one turn of the outer gear. The volume of these chambers fluctuates as the pump turns, and they also communicate with the input volume and outlet volume via the distribution of the port areas. For simplicity's sake, Figure 12 only shows one volume on each inlet and output side. Thermal pumps normally have intake and exit ports on both sides of the gear.

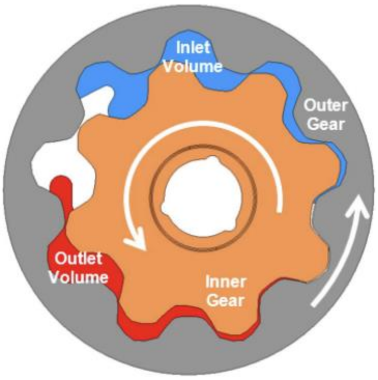

Figure 12. Schematic of an SHW gerotor pump

The system of pump flow is discretized into various volumes comprised of pipes then shunts in the one-dimensional flow model of the GT-Suite simulation program, and the pipeline channels are further discretized into sub-volumes. At each time step for each subvolume, the one-dimensional the conservation of mass, momentum, and energy in the Navier-Stokes equations is resolved. The issue of species conservation is taken into consideration while trying to find a solution to the systemic issue of species movement. It is crucial that aeration content or cavitation can be considerably impacted by the rotor pump's performance. According to Figure 13, the solver uses a one-dimensional staggered grid approach to solve scalars, with measurements of pressure, internal energy, temperature, and density at the center of each subvolume, and measurements of mass flow, velocity, and species flow at the edge. Instead of taking into account the transitory species content in a particular subbody, the solver determines the transient mass fraction of each species in each subbody. Because the fluid mixture is considered to be uniformly mixed inside a given subvolume, the flow from one subvolume to another will be dictated by the species mass fraction in the upstream subvolume.

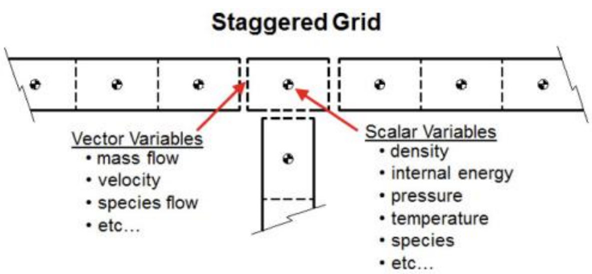

Figure 13. GT - SUITE with staggered grid solution to solve the Navier Stokes equation and a one-dimensional material transport

The user is required to supply the flow system's boundary conditions, such as the beginning pressure, fluid characteristics, and air content in the working medium, after the one-dimensional model has been built. It is also necessary to specify the pump's speed. Depending on the kind of study, the pump speed can either be set to the steady-state analysis's constant speed or to a transient profile.

The procedure for importing the pump's CAD model into GEM3D to discretize the flow rate and create the 1D flow model is shown in Figure 14. Figure 15 illustrates how the process model contains automatically produced volume and zone profiles.

Figure 14. One-dimensional flow models are created by loading CAD models into the GEM3D preprocessor

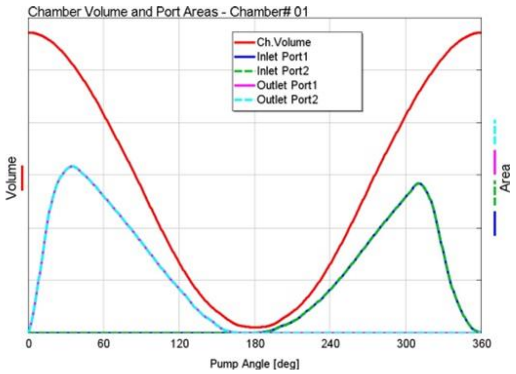

Figure 15. GEM3D was used to create a distribution map of SHW pump volume and port area

Figure 16 displays the model schematic for the SHW transmission oil pump. The SHW pump has nine external teeth, nine cavity volumes, and one turn of the external gear corresponds to one cycle of the pump. Blue highlights the flow line from the oil pan to the chamber, while red highlights the flow path from the chamber to the output. The green highlights indicate the leakage channels between the chambers.

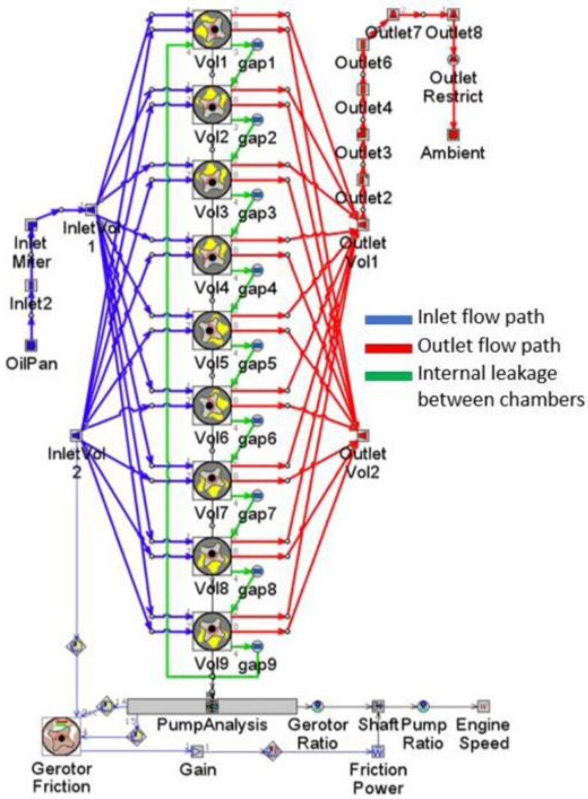

Figure 16. Diagram of the 1D flow model illustrating each flow route

At lowest and maximum clearances, 4 sets of tests were conducted at 125°C and 30°C oil temperatures, respectively. Two outlet pressures are displayed on 2- and 10-gauge bars in each of the figures below. To achieve the intended average outlet pressure, the model replicates the actual orifice in the experiment using an orifice diameter with calibration. The discharge pressure inaccuracy of the model is less than 1% in comparison to the experimental data. Figure 17 displays the association between these four scenarios and traffic. The volumetric efficiency forecast in Figure 18 is also consistent from a flow perspective.

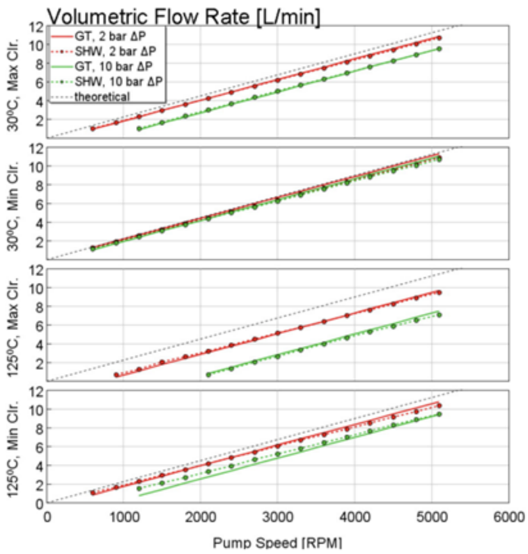

Figure 17. Model flow versus experimental flow comparison. In all directions: 125°C -Max clearance, 125° c-min clearance, 30°C - Max clearance

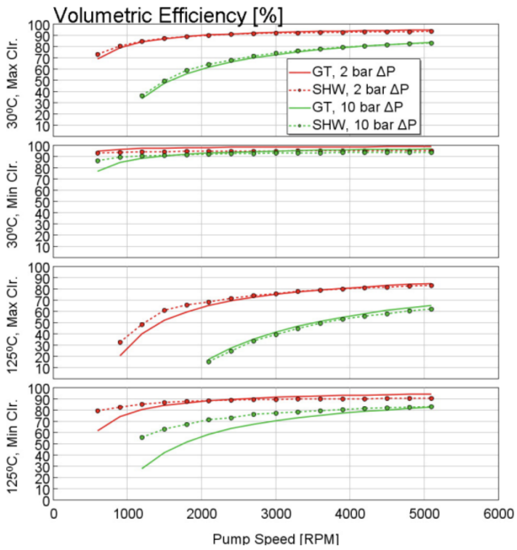

Figure 18. The model and the experiment are compared for volumetric efficiency. In all conditions: 125°C - Max clearance, 125° c-min clearance, 30°C - Max clearance

The total pump power loss and correlation for these four situations are shown in Figure 19. The best relationship may be seen between maximal clearance and cold temperature. Because fluid recirculation does not cause a large amount of fluid to enter or depart the film, the cold oil is sheared by spinning gears in the film, causing the fluid to heat up, which is one explanation for the overpredicted the loss of power at low temperatures and minimum gaps. As a result, compared to the model's assumption of the same oil pan temperature, the oil temperature in these oil films will be greater in reality. This impact is most noticeable in the cool oil, low clearance, and large speed conditions. This concept can be continued to study by establishing the pump internal heat model, so as to predict the oil temperature in the oil film more accurately.

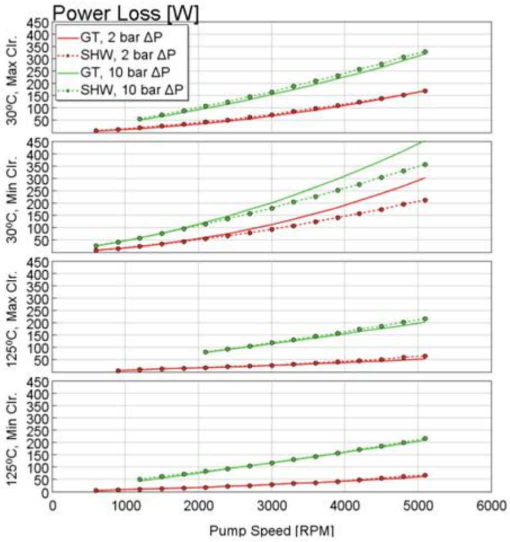

Figure 19. Comparison of power loss between model and experiment. Bottom to top Maximum clearance at 30°C, minimum clearance at 30°C, and maximum clearance at 125°C, respectively

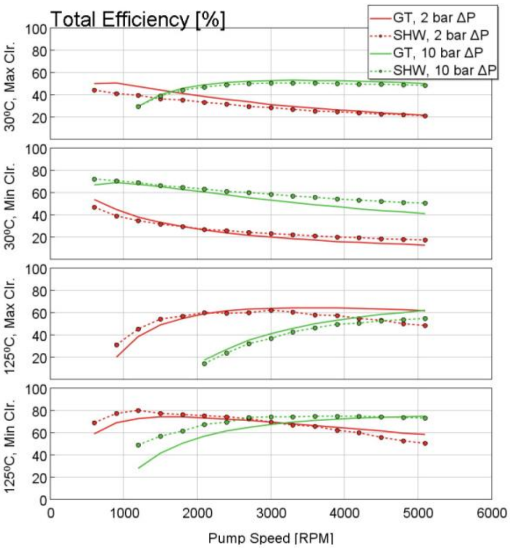

Figure 20. Comparison between the model's and the experiment's overall efficiency. In all conditions: 125°C - Max clearance, 125° c-min clearance, 30°C - Max clearance

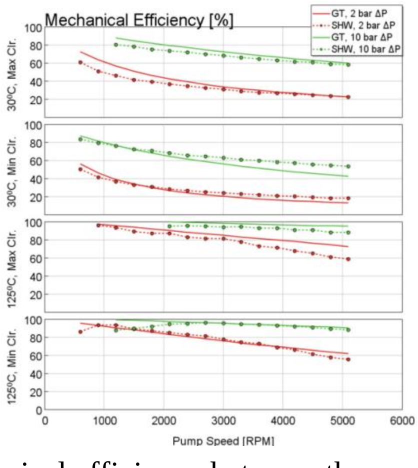

Figure 21. Comparison of mechanical efficiency between the experiment and the model Bottom to top Maximum clearance at 30°C, minimum clearance at 30°C, and maximum clearance at 125°C, respectively

The correlation between overall efficiency in these four scenarios is shown in Figure 20. With two exceptions, there is generally good consensus regarding overall effectiveness. First off, when the minimal clearance is present and the temperature is cold, the friction power is overestimated, which results in an underestimation of the total efficiency. Second, at high temperatures, clearance is at its lowest, therefore low flow rate estimates will also result in poor total efficiency estimates. The correlation between these four factors and mechanical efficiency is seen in Figure 21. Total efficiency divided by volume efficiency is known as mechanical efficiency.

By modeling the pump in 1D, engineers can immediately examine the trade-offs between cost and efficacy in terms of clearance tolerances. Where a smaller clearance under normal cold start circumstances might result in a greater overall efficiency, but a lower clearance can result in higher manufacturing costs. Engineers may use 1D modeling to forecast performance trends and make early tolerance selections in the design process, achieving the highest levels of efficiency while retaining affordable production prices, as shown in Figure 22.

Figure 22. The total efficiency is distributed between the minimum and maximum gap

The flow rate, volume efficiency, power loss, and total efficiency of an experimental rotor pump have all been included into a one-dimensional model. The models are often connected.

2.3. 2D method

Borghi et al. [10] used the lumped parameter approach to derive the axial equilibrium of the machine tool by contrasting the numerical results with the experimental results through numerical calculation and quasi-steady-state experimental representation. Measurement of lateral clearance height for different external gear pumps and different operating conditions.

2D internal code to determine the pressure distribution in the lateral clearance, the internal force on the bearing seat and balance force comparison (the radial pressure distribution on the intertooth volume as the initial condition for calculation. Through analysis, it is found that at least one coefficient should be increased to keep the bush balanced. The height and tilt of the clearance must also be changed as operating circumstances vary.

To sum up, the transverse clearance height and bearing block tilt, as expected, have a great impact on volutional efficiency, the difference between the predicted and measured results can depend on the interior surface's lack of flatness, localized deformation and the gear's wear process on the inner surface of the bearing block can determine the increased transverse clearance in the actual pump.

2.4. 3D method

Axial-flow pumps were used as an illustration by Dinget et al. [11] to discuss the newest CFD technology for pump cavitation simulation as well as to show and prove that using CFD as a tool for production design and diagnosis can be significantly increased by using good CFD software and implementing cutting-edge numerical algorithms and physical models.

Fluid mechanics plays a crucial part in the operation and performance of pumps. Therefore, Computational Fluid Dynamics (CFD) is a great technology for examining the construction and functioning of pumps.

The program delivers precise predictions of flow, pressure, and cavitation in the pump over a brief period of time by solving first-principle conservation equations in conjunction with sophisticated physical models. The technical information about the code is described in the next part, with an emphasis on the crucial elements that directly affect output and accuracy.

In past simulation, people are used to treating liquids as liquid, constant, incompressible fluid. In reality, Due to the aeration, liquid compressibility and cavitation, the liquid's average density varies. Cavitation is the process by which the primary liquid creates a bubble as the pressure is less than that of steam, and the bubble bursts when the pressure is bigger than the steam pressure. Cavitation can potentially produce loud noises and/or significant local damage when the vapor bag expands before collapsing. A mixed-flow pump blade with cavitation-induced damage is seen in Figure 23. (Brennen, 1995).

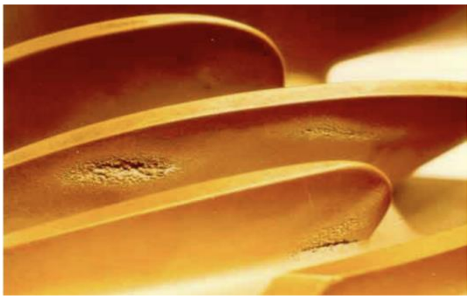

Figure 23. Cavitation damage on a pump blade for mixed flow

The currently used code employs a sophisticated implicit technique without interpolation. This method starts by using highly powerful proprietary algorithms to identify regions of overlap between interfaces that aren't compatible. Then, the units on each side of the interface are connected by using this region as a common surface. The internal contact between two adjacent cells within the same grid domain and the surface are not addressed differently throughout the simulation process. As a result, the solution will be fully implicit and conservative, and it will also automatically implement all higher-order schemes. Figure 24 depicts a case of this type matching in action. The implicit and conservative coupling of the two volumes improves the solution's efficiency, dependability, and accuracy in terms of mass, momentum, and energy.

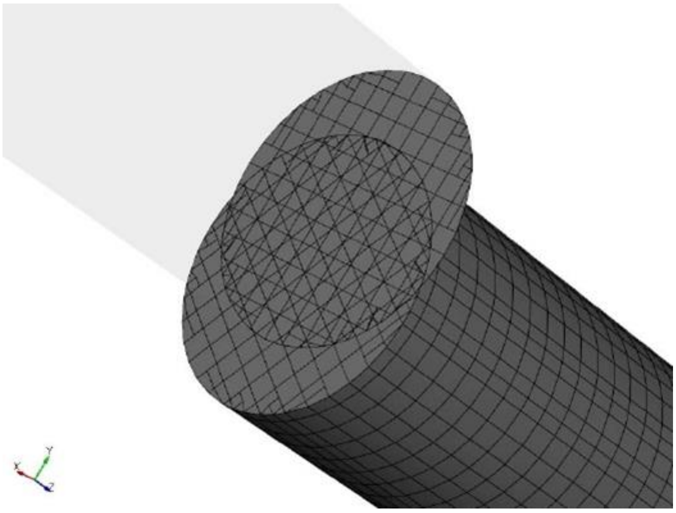

Figure 24. The grid interface between the two offset cylinders is out of alignment

Prior to the model test, CFD modeling of the new diffuser and impeller was performed throughout the design phase. Cavitation incidence, cavitation pattern, and pump performance were all monitored and documented during model testing. The outcomes of the CFD simulation are contrasted with the experimental outcomes. At the Dutch pump supplier's experimental testing facility, tests on the model pumps were conducted (Figure. 25 and 26).

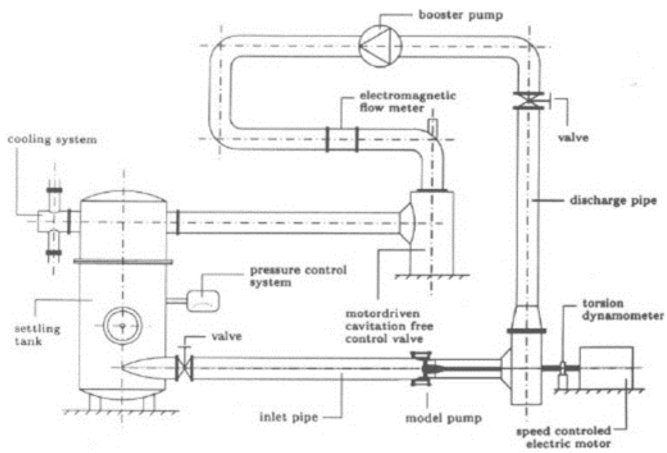

Figure 25. Model test loop schematic layout

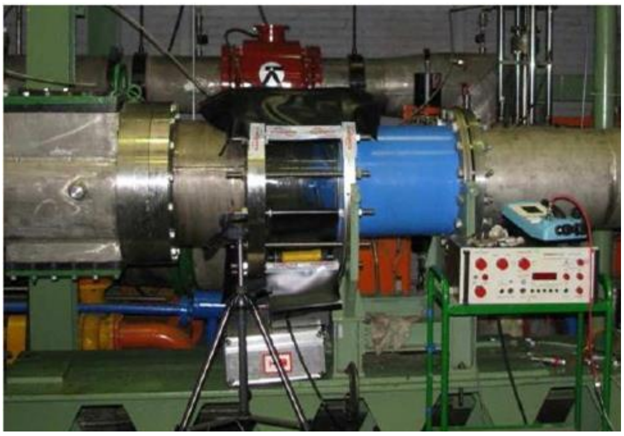

Figure 26. Diagram of the test device's test impeller installed in the test's flow display

The fluid volume's 3D solid model CAD shape is seen in Figure 27. Since the solid model, this geometry accurately represents the actual components utilized in the model testing setup. The CFD geometry definition process uses the same STP file that is used for component production. The intake, rotor, stator, and outlet are the four divisions of the fluid area. The connections are matched using an implicit interface, and each part is handled individually (Figure 28).

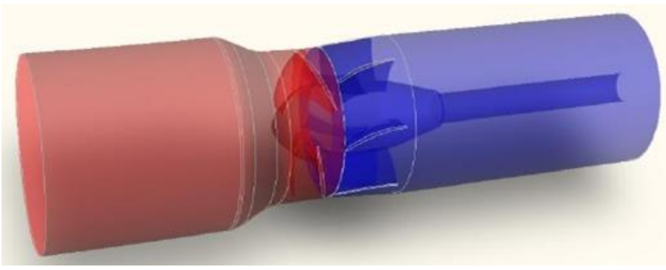

Figure 27. CAD surface of pump model

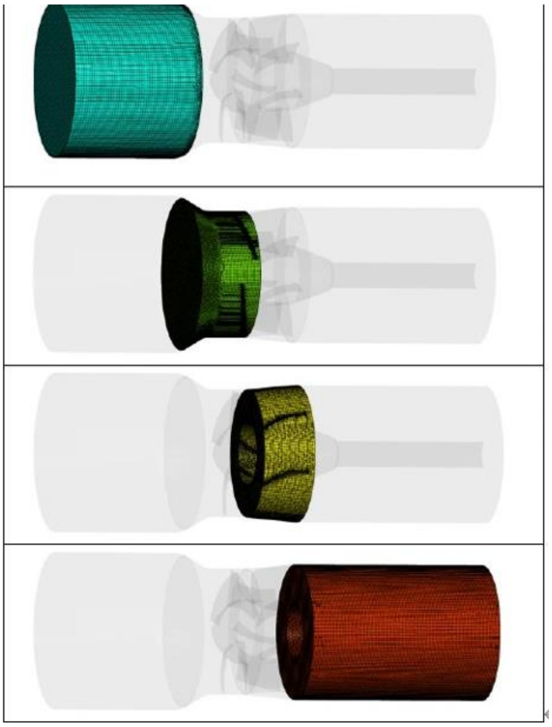

Figure 28. Grid of pump

The contrast between total pump head test results and CFD predicted results is shown in Figure 29. The graphic demonstrates that the head forecast is somewhat higher than the MRF simulation. The transient simulation findings are generally in close conformity with experimental data, with the exception of the lowest flow point, which is somewhat higher than the anticipated value.

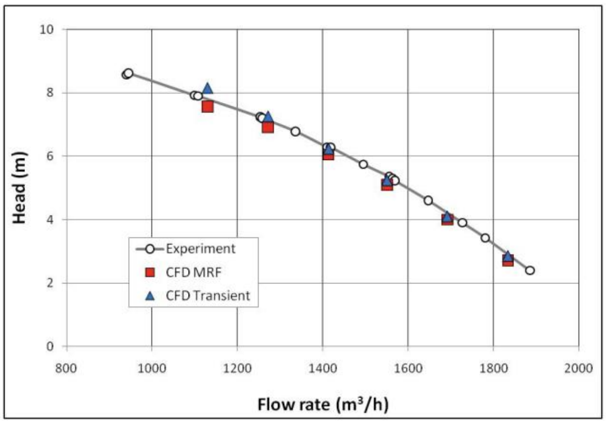

Figure 29. Comparison of test results and pump pressure head prediction; model pump, 1150 RPM

The critical concerns and major technologies of CFD modeling of pump performance and cavitation are explored, and a new generation of simulation tools for pumps as well as creative software design and development methodologies are presented. An axial-flow model pump was used to show the pump cavitation simulation's considerable increase in setup time, computing speed, and accuracy. Engineers may utilize a new generation of pump simulation tools to diagnose, enhance current designs, and create totally new pumps more quickly and efficiently.

3. Conclusion

This paper presents the recent advances of flow rate simulation in gear pumps. The main conclusions can be drawn as follows: (1) S. Wang et al. developed a numerical simulation method for hydraulic performance and dynamics of generalized controlled volumetric external gear pump.; (2) Harrison, J et al., developed an experimental model of a one-dimensional rotor pump, considering volumetric efficiency, flow rate, power loss, and overall efficiency; (3) Ding, H. et al. discuss pump performance and cavitation issues and outline innovative software design and development strategies for advanced CFD technology and a new generation of pump simulation tools. Based on the current work, future research should be devoted to the development of more convenient and fast simulation software for engineers to do further research on fluid mechanics.

References

[1] Yu You-guan; Guo-fang gong; Guo liang-hu. Simulation technology and its application in hydraulic system. HYDRAULICS PNEUMATICS & SEALS. 2005, (3).

[2] Wu Wang-yi. Fluid Mechanics (I, II) [M]. Peking University Press, 2004.

[3] Zhang Hai-ping. Hydraulic Speed Control Technology [M] China Machine Press, 2014.

[4] Yang Hua-yong; Ouyang Xiao-ping; Xu Bing. Development of hydraulic transformer. CHINESE JOURNAL OF MECHANICAL ENGINEERING. Volume 39, Number 5. May 2003.

[5] Yang Hua-yong; Zhou Hua; Lu Yong-xiang. Research Achievements and Developing Trends of Water Hydraulics. China Mechanical Engineering. Volume 11 Issue 12. December 2000.

[6] Liu You-quan; Liu Xue-hui; Zhu Hong-bin; Wu En-hua. Physically Based Fluid Simulation in Computer Animation. JOURNAL OF COMPU TER2AIDED DESIGN & COMPU TER GRAPHICS. Volume 17, Number 12. December 2005.

[7] Li Yong; Liu Zhi-you; An Yi-ran. A brief introduction to Fluent—a general purpose CFD code. JOURNAL OF HYDRODYNAMICS. Ser. A, Vol. 16, No.12. June, 2001.

[8] Wang, S.; Sakura, H.; Kasarekar, A. Numerical modelling and analysis of external gear pumps by applying generalized control volumes. Math. Comput. Modell. Dyn. Syst. 2011, 17, 501–513.

[9] Harrison, J.; Aihara, R.; Eisele, F. Modeling Gerotor Oil Pumps in 1D to Predict Performance with Known Operating Clearances. SAE Int. J. Engines 2016, 9, 1839–1846.

[10] Borghi, M.; Zardin, B.; Specchia, E. External Gear Pump Volumetric Efficiency: Numerical and Experimental Analysis; SAE Technical Paper; SAE International: Warrendale, PA, USA, 2009.

[11] Ding, H.; Visser, F.C.; Jiang, Y.; Furmanczyk, M. Demonstration and Validation of a 3D CFD Simulation Tool Predicting Pump Performance and Cavitation for Industrial Applications. J. Fluid. Eng. 2011, 133, 011101.