Analysis of the flow dynamics characteristics of an axial piston pump based on the computational fluid dynamics method

Bin Zhang a, Jien Ma b, Haocen Hong a, Huayong Yang a and Youtong Fang b

a The State Key Lab of Fluid Power Transmission and Control, The Zhejiang University, Hangzhou, China; b China Academy of West Region Development, The Zhejiang University, Hangzhou, China

ABSTRACT
To improve its working performance, the flow ripple characteristics of an axial piston pump were investigated with software which uses computational fluid dynamics (CFD) technology. The simulation accuracy was significantly optimized through the use of the improved compressible fluid model. Flow conditions of the pump were tested using a pump flow ripple test rig, and the simulation results of the CFD model showed good agreement with the experimental data. Additionally, the composition of the flow ripple was analyzed using the improved CFD model, and the results showed that the compression ripple makes up 88% of the flow ripple. The flow dynamics of the piston pump is mainly caused by the pressure difference between the intake and discharge ports of the valve plates and the fluid oil compressibility.

ARTICLE HISTORY
Received 10 March 2014
Accepted 3 August 2015

KEYWORDS
Computational fluid dynamics; flow ripple; piston pump; pressure pulsation

1. Introduction

An axial piston pump, the most important component of hydraulic systems, is widely used in the fluid power industry because of its robustness, controllability, wide operating range and compact size. Noise level, an important performance of the piston pump, becomes increasingly important with the ever-increasing demands of working performance and environmental concerns. The vast majority of the pump's noise is known as fluid-born noise, which is influenced mainly by the flow dynamics, especially when the flow ripple and cylinder pressure overshoot or undershoot. Flow dynamics characteristics can cause much damage to the pump, such as noise, lower volumetric efficiency, overturning of the cylinder block, cavitation, and short service life (Huang, 2004; Ma, Xu, Zhang, & Yang, 2010); therefore, the study of the pump's flow dynamics characteristics is essential.

In order to study the influence of the slipper model on hydraulic pumps’ flow dynamics characteristics, a time-dependent mathematical model of a hybrid trust pad bearing as a slipper of swash-plate-type axial piston pumps and motors was presented (Kazama, 2005). A hybrid trust bearing was developed as a slipper model of water hydraulic piston pumps and motors, and numerical solutions of the time-dependence problem were also built for a wide range of operations under water-lubricated conditions.

Different aspects of axial piston pump designs have also been studied and received attention with regard to improving its performance. Pressure distribution, temperature, and friction of the oil film between piston
and cylinder block have been intensively investigated (Bergada & Davies, 2012).

Advances in material technology have improved contamination tolerance and fatigue technology and a new test method is described for measuring the source flow ripple and source impedance of the positive displacement hydraulic pump, which was called the ‘secondary source’ method. The ‘secondary source’ method was developed for the evaluation of the pressure ripple characteristics of the hydraulic pump (Edge & Johnston, 1990a, 1990b).

A significant amount of work has been carried out with regard to piston slip behavior. Bergada and Kumar (2011) conducted research on the output flow ripples and leakage of axial piston pumps, building a test rig which was able to measure the dynamics as a function of turning speed, outlet pressure and swash-plate angle. In addition, flow ripple testing has been studied using a secondary source method and more recently the source admittance method (Ericson & Palmberg, 2007). In 2012, Pelosi and Ivantysynova (2012) created a geometric multi-grid scheme for the solution of the Reynolds equation in the piston-cylinder interface of an axial piston machine.

Due to complexities in geometry and physics, computational fluid dynamics (CFD) pump simulation has historically been very challenging and time consuming. It is wildly used for cases with cavitation (Ding, Visser, Jiang, & Furmanczyk, 2011). Nowadays, CFD simulations are widely used by many hydraulic research groups around the world. Ivantysynova and Huang (2005) studied the suction flow condition of a bent axis axial piston pump using CFD, and Rokala, Koskinen, Calonius, and Pietola (2008) analyzed the flow condition both in axial piston pumps and gear pumps using CFD software. Kumar, Bergada, and Watton (2009) conducted research on the static and dynamic characteristics of a piston pump slipper with a groove. In addition, they also built a test rig to compare experimental and CFD results. Casoli, Vacca, and Berta (2009) studied the flow field in the lateral clearance of external gear pumps using CFD. This provided useful information about the net flow entering or exiting a tooth space volume.

The compressible fluid and dynamic grid were employed in their model, so the output simulation pressures matched well with experimental testing. Wang (2010) ran a cavitation simulation of an axial piston pump by using the dynamic CFD method. Compressible fluid and a pump model with nine pistons were used, and the results were highly precise. Wieczorek and Ivantysynova (2014) created a piece of software called CASPAR by using CFD, which can analyze the pressure, temperature and friction of the oil film inside the pump. Ma, Xu, et al. (2010) added the compressible sub-model using user-defined functions in the CFD model to predict the flow ripple, as well as building a test rig of flow ripple to study the validity of the simulation. The comparisons with the experimental results show that the validity of the CFD model with compressible hydraulic oil is acceptable in analyzing the flow ripple characteristics.

This paper is concerned with the flow dynamics characteristics of a swash-plate piston pump. The mathematic model of an axial piston pump working system was built, and the cross angle and pre-compression angle of the valve plate were optimized based on the model. Moreover, the CFD model of the pump was improved, and the influence of its working parameters was investigated through the CFD simulations, and compared with experimental measurements. In summary, the flow dynamics characteristics model of the pressure and flow in a multi-cylinder pump was improved. The CFD pump model was also improved by introducing the compressibility fluid model to obtain acceptable results. The flow dynamics characteristics of the piston pump were analyzed using the improved model. Experimental test results were then employed to prove the validity of simulations.

2. Description of the CFD model

2.1. Models of compressible fluid oil

The traditional model in the CFD simulation using incompressible fluid oil is a natural choice, as it means that the oil density is constant. However, it also means that the pressure pulsation wave translates at an infinite speed, and is thus inconsistent with reality. Hence, it is necessary to model the compressibility of the pressure medium when pump flows are compressed by hydraulic oil. The compressibility of fluid oil is primarily described by the bulk modulus.

The reference density of the mineral oil is about 871 kg/m³ at an ambient pressure of $p_0 = 1.01$ bars. The relation between the bulk modulus $K$ and density $\rho$ is described as

$$\rho = \frac{p_0}{(1 - (p - p_0)/K)}.$$  \hspace{1cm} (1)

The sonic speed also has a close relationship with the density. The relation between density, bulk modulus and the sonic speed is given as

$$c_0 = \sqrt{\frac{K}{\rho}} = \sqrt{\frac{K - (p - p_0)}{p_0}}.$$  \hspace{1cm} (2)

The oil bulk modulus $K$ is the main parameter describing the oil compressibility, and it is also dependent on pressure and temperature. To obtain a high degree of precision in the model, the bulk modulus was tested by...
using the cross-correlation function method with pressure signals.

In order to precisely calculate the mathematical relationships between the pressure, viscosity and temperature, we introduce the Roelands formula,

\[ \eta = \eta_0 \exp \left\{ (ln \eta_0 + 9.67)[1 + 5.1 \times 10^{-9} \rho]^{2.3} \right\} \times \left( \frac{T - 138}{T_0 - 138} \right)^{-S_0} - 1 \right\}, \tag{3} \]

where \( \eta_0 \) is the viscosity in the atmosphere and \( T_0 \) is the temperature, \( Z \) is the Roelands parameter, and \( S_0 = \lambda(T_0 - 138)/(ln \eta + 9.67) \).

We chose anti-wear hydraulic oil as our test oil. From the measurement data in Table 1, we know that \( \eta = 0.188e^{-0.04717T} \), so \( \lambda = 0.0471 \) and finally we get \( S_0 = 1.16 \).

Then we can write the Roelands formula as

\[ \eta = 0.0457 \exp \left\{ 6.58 \times \left( 1 + 5.1 \times 10^{-9} \rho \right)^{2.3} \times 10^{-8} \right\} \times \left( \frac{T - 138}{303 - 138} \right)^{-1.16} \right\}. \tag{4} \]

The flow field of the whole model can be divided into three parts: the plunger cavity, the damping groove and the flow of import and export. The Reynolds number of the flow field in the plunger cavity is calculated as about 548, so the flow in the plunger cavity is a laminar flow. The Reynolds number of the flow field in the damping groove is calculated as about 2974, so the flow in the damping groove is turbulence. The Reynolds number of the flow field in the flow of import and export is calculated as about 1505, so the flow in the flow of import and export is a laminar flow.

### 2.2. Geometric model of the simulation

In contrast to incompressible fluid, compressibility modeling must take account of the interaction between the pressure pulsations and the numerical boundaries, since the pressure reflections occur and distort the pump pulsation signal. In accordance with the experimental setup, a long pressure pipe with a throttle valve connects with the pump output orifice. The pressure level inside the pipe is determined by the flow rate and the area of the throttle (Manring, 2000; Manring & Zhang, 2001).

The area of the throttle according to contraction section terminal reflection model is described as

\[ A_\nu = \frac{q}{C} \sqrt{\frac{\rho}{2\Delta P}}, \tag{5} \]

where \( C \) is the throttling coefficient (and the simulation results show that \( C = 0.737 \) will obtain higher computational accuracy), \( q \) is the instantaneous flow rate of the pump, \( \rho \) is the density of the oil, and \( \Delta P \) is the pressure difference of the reflection terminal.

We can calculate the pressure pulsation as

\[ \frac{dQ_p}{dt} = \frac{\left( ((p_f - p_c)/\rho) - (Q_p^2/2C_g^2A_g^2) \right)}{\int_{x_1}^{x_2} (1/A_\nu) \, dx}, \tag{6} \]

where \( p_p \) is the pressure of the pump inlet, \( p_f \) is the pressure of the pump inlet, \( p_c \) is the pressure of the cylinder blocks and \( C_g \) is the discharge coefficient.

According to the test rig, the maximum length of a pressure pipe was 2 m. An outspread pipe of about 0.5 m in length was set after the throttle to obtain a constant pressure at the outlet of the end pipe. On the suction side, where smaller pressure pulsations occur due to a smaller bulk modulus and the resulting lower sonic speed, a suction pipe with a maximum length 0.5 m was modeled. Figure 1 shows the complete model, which consists of the suction with pressure pump, the pump, the pressure pipe, and the throttle valve with an outspread pipe.

For the physical geometry of the computational domain, the internal geometry of the pump was used. Figure 2 shows the outline of the investigated model with the shape of the pump, suction pipe and pressure pipe with throttle valve. Some of the pump key parts are displayed in the figure. The pump physical model comprised the inlet manifold, outlet manifold, cylinder volume in front of the nine pistons, and small oil film between the valve plate and the cylinder clock, which is all the oil-filled space inside the pump except for the pump housing. The nine pistons’ chamber was built in the simulation, so

| Temperature (°C) | 20 | 30 | 40 | 48 | 57 | 76 | 85 | 95 |
|-----------------|----|----|----|----|----|----|----|----|
| Viscosity (×10⁻² Pa·s) | 7.322 | 4.570 | 2.792 | 1.900 | 1.304 | 0.839 | 0.742 | 0.572 |

---

**Figure 1.** The complete CFD model.
the pistons’ flow transaction can be analyzed. The pistons’ chamber was set along a taper distribution according to the A4 V piston block structure. The oil film between the valve plate and the cylinder clock was a spherical surface, which clearly showed the load-carrying capability of a valve plate. The throttle groove in the valve plate was used to decrease the flow ripple and pressure shock, and also analyzed for different structures.

### 2.3. Model grid

There are two major movements in the computational domain for the pump geometry model. One is the rotating cylinder block against the static valve plate. The rotation of the cylinder block will introduce a sliding grid into the analysis and the sliding grid solver option for the CFD software FLUENT. The sliding grid option provides the grid interface for the sliding surfaces between the static inlet/outlet part of grid and the part of the rotating cylinder block. An oil film of about 10 μm was used to separate these two grid parts. The oil film was filled with at least six cells in its height to improve the computational accuracy. The object of creating the grid is to be as cubic as possible and to have as small a difference as possible in the cell size between neighboring block cells, as well as avoiding the stability problem and the subsequent convergence difficulties.

The other major movement is the up and down movement of the pistons, which causes the oil to pass through the pump. The piston movement was modeled by a dynamic grid with a moving and deforming mesh for the lower parts of the cylinders. A change in deformation is made between each time step, and the grid deformation inside the cylinder is governed by the angular position and speed of the cylinder according to the studied pump (Ma, Fang, Xu, & Yang, 2010). When the compressible fluid is used in the simulation, the size of the grid cells must be smaller than the wavelength of the pressure waves.

### 2.4. Initialization and boundary conditions

Since the flow is non-stationary, an unsteady solver was used in the simulation. The length of the time step is an important parameter. It is essential to keep the information pressure and velocity changes from travel further than one cell between each time step, otherwise there can be instabilities in the numerical simulation. As there are nine pistons in one pump, when the velocity of the pump is about 2000 r/min, the theoretical simulation time is 0.0033 s. In order to ensure the accuracy of the simulation, the setting simulation time will be greater than 0.0033 s. The standard SIMPLE algorithm was employed for the pressure-velocity coupling problem with second order discretization. The simulation time step was set as 0.00001 s and the iterative precision as 0.0001 to ensure the accuracy of the result.

The pistons move periodically with a cycle of $60/(Zn)$, where $Z$ is the piston number and $n$ is the cylinder rotation speed. To solve the unsteady problem, the flow field

---

**Figure 2.** The geometric model used as the computational domain.

**Figure 3.** The periodic position of the pistons.
is first initialized and then calculated at the specified time step length. As can be seen from the periodic position of the pistons in Figure 3, α is the angle between two pistons. When the cylinder rotates through the angle of α, the length condition of the pistons is exactly the same as their original positions. The shaded area represents the oil discharge inlet and the other represents the oil suction outlet. The velocity of each plungers is given as

\[
\begin{align*}
V_{ix} &= \omega K_1 (1 + K_2) \times \sin\left(\frac{(20 + 40)i \pi}{180}\right) \times \sin 5^\circ \\
&\times \left[1 + K_2 \times \cos(157t + (20 + 40)i \pi / 180)\right]^2 \\
V_{iy} &= \omega K_1 (1 + K_2) \times \cos 5^\circ \\
&\times \left[1 + K_2 \times \cos(157t + (20 + 40)i \pi / 180)\right]^2 \\
V_{iz} &= \omega K_1 (1 + K_2) \times \cos\left(\frac{(20 + 40)i \pi}{180}\right) \times \sin 5^\circ \\
&\times \left[1 + K_2 \times \cos(157t + (20 + 40)i \pi / 180)\right]^2
\end{align*}
\]

(7)

where \( \omega \) is the revolution of the main shaft of the plunger pump, and \( K_1, K_2 \) are Plunger pump structure parameters, which can be calculated as

\[
\begin{align*}
K_1 &= \frac{R \tan \beta \tan \phi}{\sin \phi (1 + \tan \phi \tan \beta)}; \\
K_2 &= \tan \beta \tan \phi.
\end{align*}
\]

(8) (9)

Therefore, we just need to investigate the flow condition during one basic angle of α to reduce the computation time. In this way, the theoretical computation time will be 1/9 of the whole computation time based on Figure 3. In order to ensure the accuracy, the actual computation time should be a little more than the theoretical computation time (Kazama, 2005).

Both the inlet and outlet were treated as pressure boundary conditions with different static pressure levels, depending on the chosen simulation case. The calculations were made with the pressure-based and segregated solver, respectively. The CFD flow calculations were performed on a Dell workstation using state-of-the-art computer hardware for several days.

3. Results and discussion

It should be emphasized that the character of the fluid oil is most meaningful with respect to the flow ripple of the piston pump. Therefore, the primary objective of this study is to probe the fluid oil character, especially the oil compressibility and viscosity.

3.1. Influence of the oil viscosity on the model

The viscosity of the fluid oil is mainly influenced by temperature and pressure. Therefore, the viscosity-temperature and viscosity-pressure models were added to the simulation model by using the user-defined function separately in the same simulation model. The simulation results for the pump flow ripple and the pressure in the piston chamber are shown in Figures 4 and 5. It was found that the differences in the flow ripple and pressure obtained from these different viscosity models are small. It can be estimated that the dynamic flow of the piston pump is barely affected by the viscosity characteristics. As the oil viscosity is more sensitive to temperature than pressure, the temperature still slightly influences the pump dynamic flow and enlarges the flow ripple. The viscosity of fluid oil decreases as the temperature increases; thereby, the leakage flow is enhanced. The flow ripple of
leakage flow, which is one pump's part of the total flow ripple, also increases. Thus, the simulation results show that the pump flow ripple using a viscosity-temperature characteristics model is larger than the original model. It should be emphasized here that the impact of leakage is a relatively weak pulse in the total flow ripple compared with the compressible flow ripple, which is discussed in detail below. Therefore, it can be concluded that the viscosity-temperature characteristics model will only be needed in the case of a high working temperature.

The viscosity will only have an impact on performance in a super-high pressure situation, which is much higher than the normal operating pressure for the pump. Thus, the viscosity-pressure model is negligible in the simulation when analyzing the dynamic characteristics of a pump.

3.2. Influence of the oil compressibility on the model

To further study the impact of compressible fluid oil, the CFD simulation was used to analyze the same pump with nine pistons at the same working conditions, but one model with compressible oil and the other with incompressible oil. The flow ripples and chamber pressure results are respectively shown in Figures 6 and 7. The flow ripple rate of the model with compressible oil is about 17.8%, which is close to the testing flow rate of the piston pump, while the pulsation rate of the flow ripple is just 5.8% for the model with incompressible oil. Therefore, the simulation accuracy has been significantly improved by using the improved compressible fluid model in the CFD simulation.

According to the simulation results, both the pressure in the rise and fall duration time and the pressure changes in the high and low piston chamber are much greater in the model with compressible oil, when considering the elasticity, than in the model with incompressible oil. Pressure oscillation occurs during the high-low pressure transformation process in the piston chamber in model with compressible oil because the fluid elasticity produces fluid oil reciprocal transformation between kinetic energy and elastic potential energy, while in the 'rigid' fluid, the rise and fall time of the pressure is relatively short and the effect of the intrusion process disappears more quickly. Thereby, it is hard to catch the peak pressure in the simulation. The simulation result is more precise in predicting the pressure characteristics in a piston chamber by considering the influence of fluid compressibility.

3.3. Precision of the model

To study the validity of the CFD model, a nine-cylinder pump running at 2000 r/min delivering a mean low of 80 L/min was investigated and compared with the results of the simulation. The results of the comparison are shown in Figure 8.

The maximum flow ripple amplitude of the CFD model simulation is about 14.2 L/min, while the experimental result is about 14.9 L/min. Compared with the total pump flow rate of 80 L/min, the flow ripple rate of the experimental result is about 18.6%. The flow ripple rate of the CFD result is about 17.8%. Therefore, the error of the multi-cylinder pump CFD model in the flow ripple prediction is only about 4.6%, meaning that the accuracy of the CFD simulation is acceptable.

The ripple cycle is about 3.33 ms based on a rotation speed of 2000 r/min, and was used for both the experiment and simulation. The curve of the simulation results changes with the trend of the experimental results; they both have a flow peak at the beginning and then gradually go down. Therefore, these results show that the flow
ripples from the CFD simulation are well in agreement with the experimental measurements, and thus the CFD model with compressible fluid oil is shown to be highly accurate and acceptable for use in analyzing the flow ripple characteristics of the piston pump.

3.4. Composition of the flow ripple

It was shown in the simulation that the flow ripple rate of the piston pump increases from 5.8% to 17.8% with the same boundary conditions by using the compressible fluid model, and a comparison with the experimental result of 18.6%. Thus the simulation accuracy has been significantly improved by using the improved compressible fluid model. To further investigate the compressible influence, the inflection factors of the flow ripple – including the compression ripple, leakage ripple and kinematic flow ripple – were analyzed in a simulation using the compressible fluid model.

The kinematic flow ripple can be evaluated by using the traditional flow ripple model. The instantaneous flow rate is evaluated by using the classical flow ripple equation given by

\[ q = \sum_{i=1}^{N} A_p v_i = k A_p \rho \omega \sum_{i=1}^{N} \sin[\omega t + (i + 1) 2\pi / Z], \] (10)

where \( N \) is the piston number in a high pressure zone, \( A_p \) is the piston cross-sectional area, and \( v_i \) is the axial velocity of the \( i \)th piston. Then, the flow ripple rate of a piston pump is evaluated as

\[ \delta = \frac{q_{\text{max}} - q_{\text{min}}}{((q_{\text{max}} + q_{\text{min}})/2).} \] (11)

It is easy to see that the flow ripple rate of the nine-piston pump is \( \delta = 1.5 \% \) when only considering the influence of the kinematic flow ripple. The geometrical flow ripple is the dynamic flow character of a piston pump at the piston number changing in a high-pressure zone, which has a high error compared to the real pump flow ripple.

The leakage flow results of a piston pump and comparison with the mathematical model are presented in Figure 9 (for a detailed discussion, see [18]). The leakage flow ripples go up and down periodically, and the results match well from the curves changing rule. The average leakage flow is about 2.5 L/min, and the volumetric efficiency of the pump is around 97%, which is consistent with the real piston pump. The amplitude of the leakage flow is 0.58 L/min, and the leakage flow ripple of the piston pump is 0.7%. It can be seen from the simulation results that the compression ripple is the main part of the flow ripple and accounts for 88%. The remainder leakage flow ripple has the lowest proportion of 4%, and the geometrical flow ripple makes up the remaining 8%.
The maximum flow ripple amplitude of the CFD model simulation result is about 14.2 L/min, compared with the 4.6 L/min result without using the compressible fluid model. The amplitude of the geometric flow ripple is 1.2 L/min and the amplitude of the leakage flow ripple is 0.58 L/min. Figure 10 shows the different flow ripple comparisons. The flow ripple comparisons show that the compressibility effects of the fluid oil are mainly attributed to the pump flow ripple; thus, the compression ripple of the pump should be controlled in order to decrease the pump flow ripple.

3.5. Compressibility effects

In order to control the pump flow ripple, it is necessary to understand the principle of the compressibility effect operation. The pump compression ripple is mainly generated when the piston bore passes over the transition regions of the intake port and discharge port of the valve plate. As shown in Figure 11, the pressure difference firstly causes fluid oil to back up into the piston bore when the piston bore transitions from the intake to the discharge ports, and secondly tends to create a significant flow oscillatory motion due to the fluid oil operating at the conservation of mass and the definition of the fluid bulk modulus.

The cause of the flow ripple consists of three parts: geometric pulsation, elastic ripple and leakage pulse. In this simulation, all of these aspects which cause a flow ripple in piston pump are considered. As the simulation results show, it was found that there is a backward flow in the chamber. The backward flow is caused by the oil flowing back into the chamber from the outlet due to the pressure difference. As shown in Figure 12, a change in the velocity vector of the flow field can easily be figured out. At the initial position, there was a big pressure difference, which caused all the flow in the damping groove to flow back to the chambers with a high flow velocity. Along with the movement of the piston, the pressure in the chamber raised. Then the pressure difference decreased,
so that the backward flow was divided into two parts, one of which still flowed into the chamber under the effect of inertia and the other of which was pushed into the relief groove. Finally, with the movement of the piston, the pressure difference disappeared.

The reason for this peculiarity is strictly a result of the volumetric compression and expansion within the chamber because of the compressibility of the fluid oil. After this, the flow gradually becomes smooth. The flow dynamic motion within the transition regions of the valve plates directly causes the pump flow ripple outflow. Because of the pressure difference between the intake and discharge ports of the valve plates and the fluid oil compressibility, the flow dynamics of the piston pump is inevitable. In the first case, the relief groove can be used in the valve plate to release the jump in pressure, and the flow dynamics can be controlled by using optimized structural parameters in the relief groove. In the second case, the fluid bulk modulus, which describes the compressibility of the fluid, can be heightened to control the flow dynamics.

4. Conclusions

In this paper, from the comparisons between the numerical simulation and the experimental results, the following conclusions are drawn. First, 3D dynamic calculations were employed to obtain the flow field within the piston pump body using CFD software. The improved CFD model with compressible fluid oil increases the flow ripple rate, and the accuracy reaches to about one-magnitude-order; thus, the validity of this model is acceptable for analyzing the flow ripple characteristics based on the comparisons with the experiment. Second, the flow ripple is greatly influenced by the compressibility of the fluid oil, based on the testing flow rate of the piston pump. Therefore, the simulation accuracy has been significantly improved by using the improved compressible fluid model in the mathematical model and the CFD simulations. Third, the compression ripple is the main part of the flow ripple, an accounts for 88% of it. Of the remainder, the leakage flow ripple has the lowest proportion of 4%, and the geometrical flow ripple accounts for the remaining 8%. Finally, the flow dynamics of the piston pump is mainly caused by the pressure difference between the intake and discharge ports of the valve plates and the fluid oil compressibility. In order to control the flow ripple of the piston pump, the structural parameters of the relief grooves in valve plates can be optimized and the fluid bulk modulus can be heightened.

Disclosure statement

No potential conflict of interest was reported by the authors.

Funding

This work was supported by the National 04 Major Science and Technology Project [No. 2012ZX04004021]; The National Key Technology Support Program (2014BAF02B00) and the National Natural Science Foundation of China [51105331, 51475419].

References

Bergada, J. M., & Davies, D. L. (2012). The effect of oil pressure and temperature on barrel film thickness and barrel dynamics of an axial piston pump. *Meccanica, 47*(3), 639–654.
Bergada, J. M., & Kumar, S. (2011). A complete analysis of axial piston pump leakage and output flow ripples. *Applied Mathematical Modelling*, 36(4), 1731–1751.
Casoli, P., Vacca, A., & Berta, G. L. (2009). A CFD analysis of the flow field in the lateral clearance of external gear pumps. *Proceedings of the 11th Scandinavian International Conference on Fluid Power*. June 2–4 2009, Linköping University, Sweden, 786–794.
Ding, H., Visser, F. C., Jiang, Y., & Furmanczyk, M. (2011). Demonstration and validation of a 3D CFD simulation tool predicting pump performance and cavitation for industrial applications. *Journal of Fluids Engineering*, 133(1), 277–293.
Edge, K. A., & Johnston, D. N. (1990a). The ‘Secondary Source’ method for the measurement of pump pressure ripple characteristics part 1. Description of method. *Journal of Power and Process Engineering*, 204(11), 33–40.
Edge, K. A., & Johnston, D. N. (1990b). The ‘Secondary Source’ method for the measurement of pump pressure ripple characteristics part 2: Experimental results. *Journal of Power and Energy*, 204(11), 41–46.

Ericson, L., & Palmberg, J. O. (2007). The source admittance method for pumps with complex outlet channels. The Tenth Scandinavian International Conference on Fluid Power. 21–23 May 2007, Tampere, Finland, 279–293.

Huang, C. (2004). *CASPAR based slipper performance prediction in axial piston pumps*. Proceedings of 3rd FPNI-PhD symposium on fluid power. June/July 2004, Terassa, Spain, 229–238.

Ivantysynova, M., & Huang, C. (2005). *Thermal analysis in axial piston machines using CASPAR*. Proceeding of the 6th international conference on fluid power transmission and control. 20–22 May 2005, Hangzhou, China, 5–8.

Kazama, T. (2005). Numerical simulation of a slipper model for water hydraulic pumps/motors in mixed lubrication. Proceeding of the 6th IFPS International Symposium on Fluid Power. 7–10 Nov 2005, Tsukuba, Japan, 509–514.

Kumar, S., Bergada, J. M., & Watton, J. (2009). Axial piston pump grooved slipper analysis by CFD simulation of three-dimensional NVS equation in cylindrical coordinates. *Computers & Fluids*, 38, 648–663.

Ma, J., Xu, B., & Yang, H. (2010). Study on flow ripple of axial piston pump with CFD simulation using compressible fluid oil. *Chinese Journal of Mechanical Engineering*, 23(1), 45–52.

Manring, N. (2000). Tipping the cylinder block of an axial-piston swash-plate type hydrostatic machine. *Journal of Dynamic Systems Measurement and Control, Transaction of the ASME*, 122, 216–221.

Manring, N., & Zhang, Y. (2001). The improved volumetric-efficiency of an axial-piston pump utilizing a trapped-volume design. *Journal of Dynamic Systems Measurement and Control, Transaction of the ASME*, 123, 479–487.

Pelosi, M., & Ivantysynova, M. (2012). A geometric multi-grid solver for the piston–cylinder interface of axial piston machines. *Tribology Transactions*, 55, 163–174.

Rokala, M., Koskinen, K. T., Calonius, O., & Pietola, M. (2008). Tribological conditions between swash plate and slipper pad in variable displacement water hydraulic axial piston unit. The 6th International Fluid Power Conference Workshop, 1 Jan 2008, Dresden, German, 301–314.

Wang, S. (2010). The analysis of cavitation problems in the axial piston pump. *Journal of Fluids Engineering*, 132, 1–6.

Wieczorek, U., & Ivantysynova, M. (2014). Computer aided optimization of bearing and sealing gaps in hydrostatic machines—the simulation tool Caspar. *International Journal of Fluid Power*, 3(1), 7–20.