Application of SolidWorks® & AMESim® - based Simulation Technique to Modeling, Cavitation, and Back-flow Analyses of Trochoid Hydraulic Pump for Multi-step Transmission

Myung Sik Kim[1], **Won Jee Chung**[1], **Jun Ho Jang**[1], **Chang Doo Jung**[1]
[1] School of Mechatronics, Changwon National University, South Korea

Abstract - *Flow rate control is the uppermost concern for trochoid hydraulic pump. Cavitation within the flow field of pump has the most influence on flow rate control reason at approximately 3500 ~ 4000 RPM high speed rotation of pump. In this paper, based on AMESim® and SolidWorks®, we will present how to simulate cavitation by analyzing the control factors of trochoid pump; hydraulic pressure change of outlet, flow rate according to rotation speed of inner rotor, leakage through gap between outer rotor and inner rotor, and discharging angle of outlet. The proposed methodology of cavitation simulation will enables field engineers to have access to the design of trochoid pump more easily and thereby to have more concrete control over the flow rate of pump by realizing its analysis model similar to its actual product model.*

Keywords: Trochoid Hydraulic Pump, Modeling, Cavitation, Back-flow, AMESim®, SolidWorks®.

1 Introduction

Recently the role of transmission has been focused on according to the emphasis of driving performance, and the competitiveness of the transmission takes the largest part in the competition of automobiles together with engine. It is essential to improve the lubrication system and its performance due to the increase in the role and performance level of transmission, and the interests on these issues have also been increased. A trochoid (hydraulic) pump has been largely used in the lubrication system. Because the trochoid pump shows a simple structure, easy control in the flow rate per one rotation and the flow rate, and an advantage in its miniaturizing, it is very adaptable for the engine and transmission of automobiles due to the low variance in its efficiency because of the relatively small movement between outer and inner rotors.

It is essential to improve the lubrication system and its performance due to the increase in the role and performance level of transmission, and the interests on these issues have also been increased. A trochoid hydraulic pump has been largely used in the lubrication system. Because the trochoid pump shows a simple structure, easy controls in the flow rate per one rotation and the flow rate, and an advantage in its miniaturizing, it is very adaptable for the engine and transmission of automobiles due to the low variance in its efficiency because of the relatively small movement between outer and inner rotors.

As shown in Fig. 1, the trochoid pump investigated in this paper represents an eccentricity in its rotation axis due to the structure of outer and inner rotors and that shows sliding contact (see Fig. 2 in detail). Also, it shows a difference in rotation speed as much as the difference in the number of teeth. While the rotors are rotated, the space generated by the contact between rotors is also rotated and generates an increase in volume. Then, it discharges a fluid to the outlet by absorbing a fluid from the inlet according to the increase or decrease in the volume. In addition, it prevents reverse flow (or back-flow) because the chamber between the inlet and the outlet is not connected.

Fig. 1 Trochoid Hydraulic Pump

Machining error and operating condition can be enumerated as the reason which can degrade the efficiency of trochoid pump. But cavitation within the flow field of pump can be placed as the most dominant reason during high speed rotation. Such cavitation can cause noise and vibration by increasing pulsation as well as the falloff of flow rate efficiency. Thus it is important to design the trochoid pump which can avoid the occurrence of cavitation in terms of the performance, endurance, noise and vibration of the pump. However it is well known that cavitation can inevitably occur at approximately 3500 ~ 4000 RPM (Revolution Per Minute) high-speed rotation speed of pump [1]. Therefore it is required to examine the phenomena of the cavitation factors which have influence on the degradation of flow rate efficiency through cavitation simulation.

In previous papers including Yang et al.[1] and Nam et al. [2], the simulation of trochoid pump has utilized a professional analysis program or language such as CFD® (specifically CFX ®) or C-code (e.g. C++) which is not an easy tool for a field engineer. This software tool can realize mesh and then make a cavitaion model. But the field engineer based on his experience has a hurdle in modifying the cavitation model according to the need of a customer. In this paper, we will present how to simulate cavitation by using the most popular 3-dimensional (or 3D) modeling tool SolidWorks® and a hydraulic analysis program AMESim®. This proposed methodology of cavitation simulation will be useful for fast modification of trochiod pump design.

2 Trochoid Modeling using SolidWorks®

Figure 2 shows the rotor shape of pump based on trochoid profile. The trochoid pump is a type of gear pump, which is considered as a positive displacement pump in the upper level. The positive displacement pump has constant dispensing flow rate according to the single rotation of shaft regardless of loading pressure.

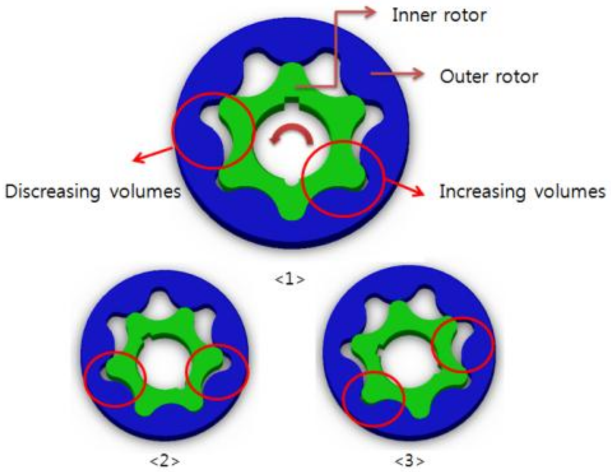


Fig. 2 Rotor shape of trochoid pump

Drawing of trochoid curve was suggested as various methods in many theses. The mathematical model composed of Eqs. (1) and (2), which has been mentioned representatively in previous papers including Jang et al.[3], forms basically-known trochoid curve condition:

$$K = \frac{-R_r \sin\theta + e \sin(N+1)\theta}{R_r \cos\theta - e \cos(N+1)\theta} \quad (1)$$

$$X = x + \frac{R_c}{\sqrt{1+K^2}}$$

$$Y = y - \frac{R_c K}{\sqrt{1+K^2}}$$

$$X = x - \frac{R_c}{\sqrt{1+K^2}}$$

$$Y = y + \frac{R_c K}{\sqrt{1+K^2}} \quad (2)$$

where Rr is the radius of rolling circle, N is the number of teeth of outer rotor, and Rc is the radius of circle of trace of wheel, θ is rotation angle of basic circle, e is eccentricity as shown in Fig. 3 (the drawing of trochoid curve SolidWorks®).

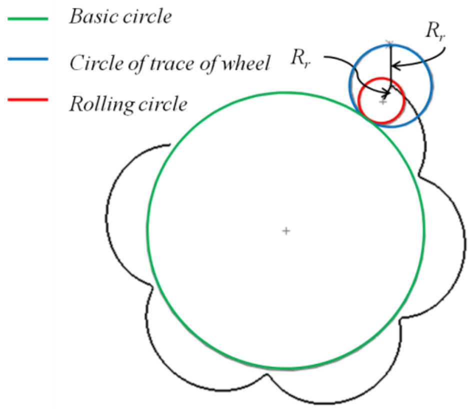


Fig. 3 Trochoid Curve Profile

Figure 3 means that filed engineers who are familiar with SolidWorks® can draw easily the trochoid model by using SolidWorks® based on Eqs. (1) and (2) not by virtue of C-code which is difficult for the engineers. Specifically, using the motion analysis module of SolidWorks®, the rolling circle can be rotated based on the basic circle in order to generate trochoid curve as shown in Fig. 3. Only the basic condition, *Rr* > *e*, should be satisfied. Next, when the circle of trace of wheel is made to be rotated according to the trochoid curve through the motion analysis module of SolidWorks®, the shape of inner rotor can be designed as shown in Fig. 4.

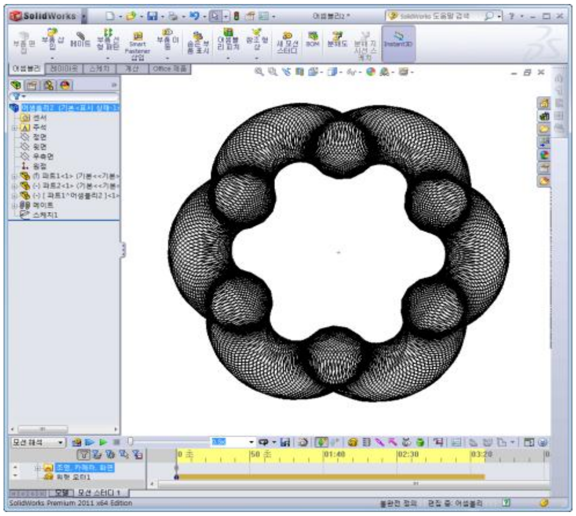


Fig. 4 Generation of Inner Rotor Shape

3 Flow Field Area Modeling Using SolidWorks®

As shown in Eq. (3), the dispensing flow rate Q_{pump} of the positive displacement pump can be obtained by using the change of flow volume *displ* per one rotation and the rotational angular speed ω_{pump}. Especially *displ* can be expressed by Eq. (4) because the thickness *H* of trochoid pump is constant. In Eq. (4), *Area* denotes the crossectional area of flow rate while θ indicates rotation angle. Now we need the data of area change according to rotation angle, *i.e*. $\frac{dArea}{d\theta}$ by using SolidWorks® as a CAD (Computer Aided Design) tool.

$$Q_{pump} = displ \cdot \omega_{pump} \tag{3}$$

$$displ = \frac{dV}{d\theta} = H \cdot \frac{dArea}{d\theta} \left[\frac{m^3}{rad}\right] \tag{4}$$

The data of area change ($dArea$) can be obtained by using angular velocity ratio of trochoid pump. It can be performed by changing the area through the rotation of each rotor according to the rotation ratio of inner and outer rotors. The speed ratio can be determined by Eq. 5) according to the number of outer rotor teeth (*N*):

$$\frac{\omega_{out}}{\omega_{in}} = \frac{N-1}{N} \tag{5}$$

where ω_{out} (ω_{in}) denotes the angular velocity of outer (inner) rotor.

When the drawings of the inner and outer rotors are completed, flow field area modeling can be done through the element conversion technique of SolidWorks® as shown in Fig. 5 where inlet and outlet are simplified. In specific, the sketch of 3 parts, *i.e*., inner rotor, outer rotor and inlet/outlet, are first designated as 'BLOCKs' by using BLOCK technique of SolidWorks®. Then the flow field area (to be explained later) can be modeled by performing the PROTRUSION BASE (similar to PAD technique of Solidworks®) technique of SolidWorks®. Consequently 7 models of flow field area are designated as shown in Fig. 5.

Now, the area change of one flow field should be investigated according to Eq. (5) (in other words, according to the rotation of inner rotor BLOCK). Specifically, while making inlet and outlet BLOCK fixed, the outer rotor BLOCK is rotated based on Eq. (5) as the inner rotor BLOCK is rotated, by using FORMULA EDIT technique of SolidWorks®. In every rotation, the area of one flow field can be changed in shape according to rotation angle of inner rotor as shown in Fig. 6. Thus the area change of one flow field can be depicted as a graph of Fig. 7 which will be used later for the simulation of trochoid pump using AMESim®.

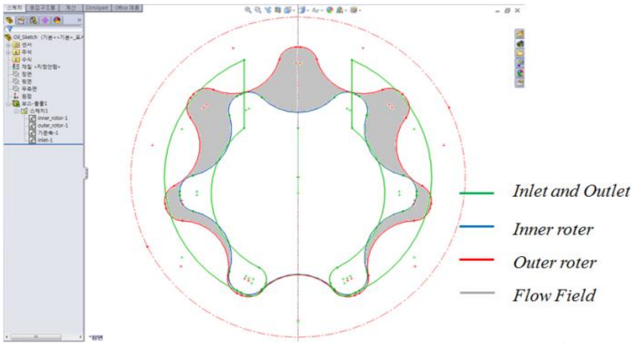


Fig. 5 Flow field modeling using SolidWorks®

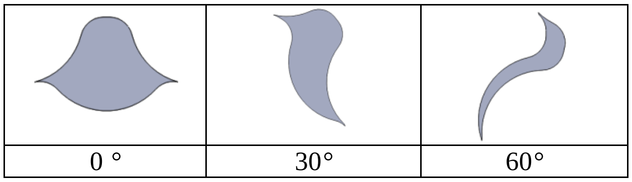


Fig. 6 Area change of one flow field according to rotation angle of inner rotor (shape)

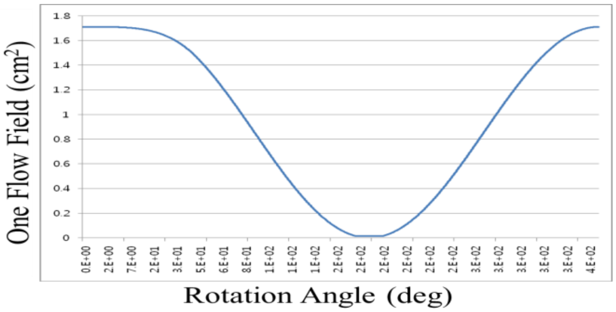


Fig. 7 Area change of one flow field according to rotation angle of inner rotor (graph)

In a similar manner to the method mentioned above, the area change of one flow field in inlet and outlet of trochoid pump (see Fig. 8) can be easily obtained in the graph of Fig. 9, which will be also used for cavitation simulation in AMESim®.

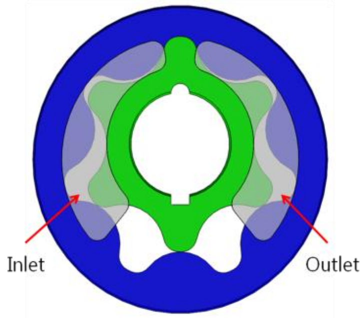


Fig. 8 Inlet and outlet flow field modeling using SolidWorks®

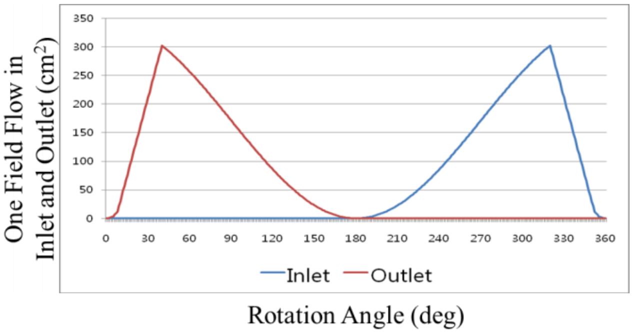


Fig. 9 Area change for inlet and outlet flow fields

4 Hydraulic Circuit For Trochoid Pump And Cavitation Simulation Using AMESim®

The objective of hydraulic circuit modeling for trochoid pump using AMESim® is to realize flow and simulate cavitation so as to control the flow rate control of trochoid pump. In case of hydraulic circuit modeling of only one flow field for trochoid pump, the area change data of Fig. 7 and the thickness of rotor can be made equivalent with the piston model as shown in Fig. 10.

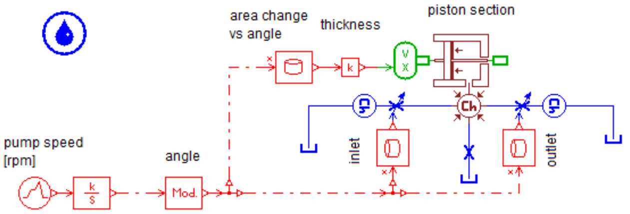


Fig. 10 Hydraulic circuit modeling of only one flow field using AMESim®

Hydraulic circuit modeling of trochoid pump is required to show the volume generated at the location of each particular angle when the inner rotor comes in contact with the outer rotor. This can be carried out at the location of each phase change, *i.e.*, 360°/N. Finally *N* models are generated similarly as Fig.10 and connected each other as shown in Fig. 11.

The factors that can be simply controlled in real time through the formation of the AMESim® hydraulic circuit of a trochoid pump include: the rotation speed of pump, the shape angle of inlet and outlet, gap between outer rotor and pump casing, and gap between inner rotor and outer rotor. In cavitation simulation, these control factors are important for the flow rate control of trochoid pump. Especially, from the viewpoint of a field engineer, AMESim® is more uselful for this cavitation simulation, compared with a traditional analysis software of fluid mechanics, *i.e.*, CFD®, because CFD® needs the renewal of mesh modeling every time each control factor has a different value while AMESim® needs only the input values of control factors without any change of hydraulic circuit modeling.

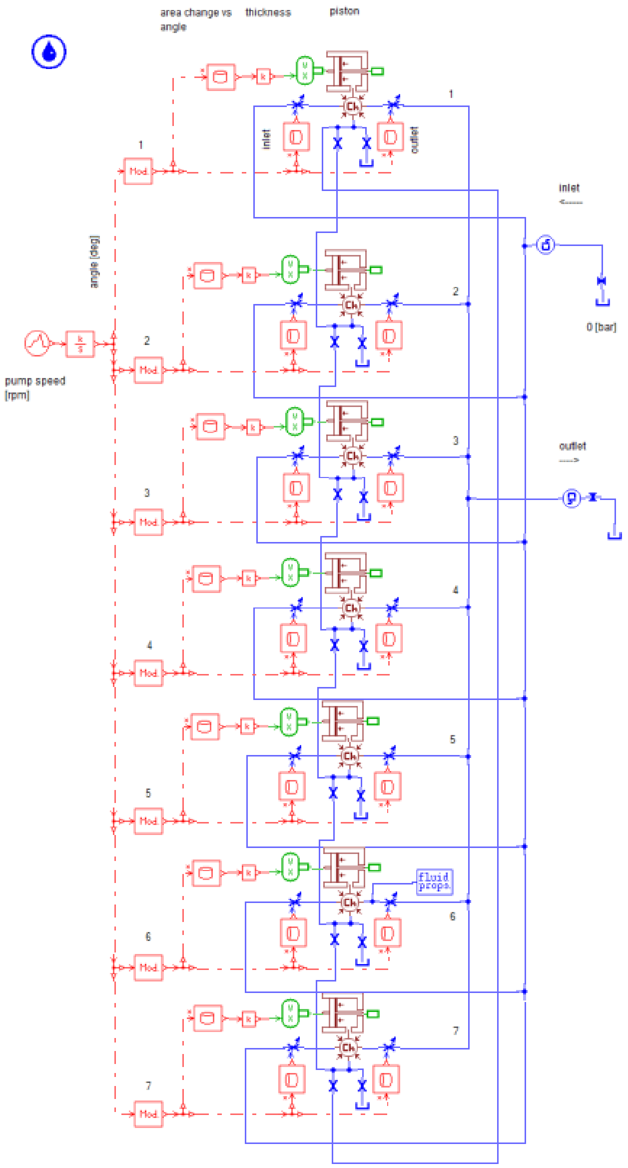


Fig.11 Connected *N* hydraulic circuit modeling of trochoid pump using AMESim®

As mentioned before, cavitation can inevitably occur at approximately 3500 ~ 4000 RPM high-speed rotation speed of pump. The cavitation usually results in increasing pulsation and thereby degrades flow rate efficiency. This is why cavitation simulation is required for the control of flow rate. In this paper, the control factors of trochoid pump including the rotation speed of pump, the pressure difference at inlet and outlet, the shape angle of outlet, gap between outer rotor and pump casing, and gap between inner rotor and outer rotor are analyzed in the cavitation simulation through the connected *N* hydraulic circuit modeling of trochoid pump using AMESim® (see Fig. 11).

4.1 Hydraulic Pressure Change of Outlet

As the first result of cavitation simulation, Fig. 12 shows the hydraulic pressure change of outlet at the 3000RPM rotation speed of inner rotor in a stabilized zone except a transient status of outlet pressure. The pressure change shows a cyclic (or periodic) characteristic at every 360°/*N* depending on the number of teeth for the outer rotor. Since *N* is 7 in this paper, in a stabilized zone, the pressure change cycle is repeated seven times for one rotation of inner rotor as shown in Fig. 12. This confirms the periodicity shown in the analyses using CFD® (Won *et al*. [3] and Yang *et al*.[1].)

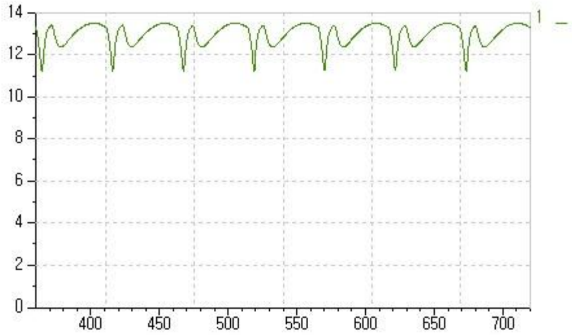


Fig. 12 Hydraulic pressure change of outlet

4.2 Flow rate according to Rotation Speed of Inner Rotor

Figure.13 shows the result of flow rate according to the rotation speed of inner rotor. As shown in this figure, a theoretical flow rate ([2]) delineates linear profile according to the rotation speed of the rotor. This means that the fluid of pump is filled 100% in the flow field without cavitation because the gap between the inner rotor and the outer rotor is not taken into consideration for theoretical study. However, since cavitation has been considered in the simulation, it can be noticed that the flow rate decreases at more than 4000RPM. As stated in refs. [1] and [2], it is confirmed that as the rotation speed of pump increases, cavitation is generated in the flow field, which makes the flow rate decreased due to the leakage of fluid into the outlet.

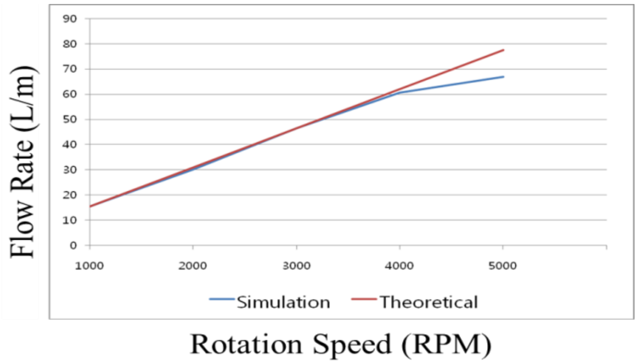


Fig. 13 Flow rate according to rotation speed of inner rotor

4.3 Discharging Angle of Outlet

In cavitation, rise of pulsation is inevitable. Moreover, in the high speed operation of trochoid pump, inlet flow resistance is enlarged so that cavitation phenomena itself can be increased. This means that the quantity of fluid to be transported in an isolated flow field can be decreased due to the increase of cavitation which has been induced by the decrease of the hydraulic pressure for the flow field. Thus back-flow into the isolated flow field can be resulted in at the position shown in Fig. 14. When back-flow to the isolated flow field is discharged to the outlet again by the rotation of rotor, it can result in higher pulsation by increasing the discharge pressure of outlet.

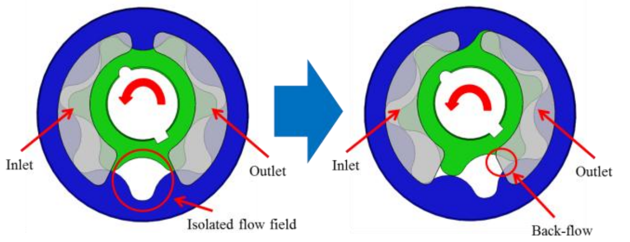
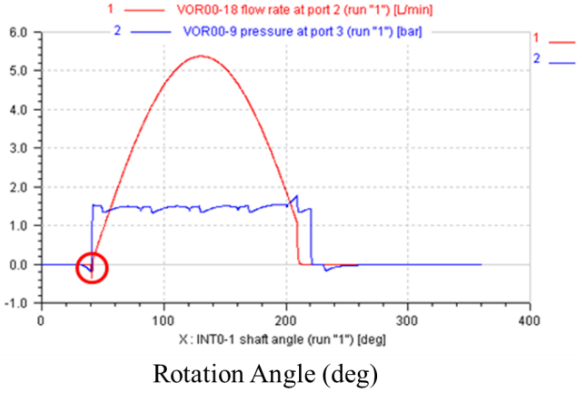


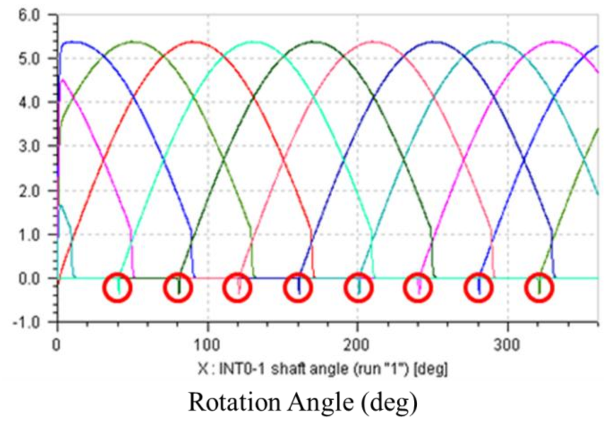
Fig. 14 Back-flow due to cavitation

The flow rate of back-flow is observed through Fig. 15 (a). In one flow field, the out-flow shows under 0 due to momentarily deterioration of pressure when the isolated flow field (see Fig. 14) meet the outlet position. Thus this showed the back-flow phenomenon. As shown in Fig.15 (b), when it is applied to all flow fields (*i.e.*, chambers), it can be seen that this back-flow has occurred so as to have a significant effects on flow rate.

To cope with this back-flow problem, the method to delay the discharging instant of outlet (in other words, the method to decrease the discharging angle of outlet) has been utilized in ref. [6,7]. In this paper, this method has been also adopted in order to prevent the back-flow phenomenon induced by the isolated flow field.



(a) Back-flow of one flow field



(b) Back of all flow fields

Fig. 15 Flow rate of back-flow

As shown in Fig. 16, the discharge instant of the outlet position has been delayed so that the compression time of isolated flow field has been increased. As a result, the pressure difference between the isolated flow field and the outlet position has been decreased so that the back-flow problem has been suppressed. This can be verified in Fig. 17 as the discharging angle of outlet is delayed at intervals of 0°, 1°, 2°, 3°, 4°, and 5°. As shown in this figure, the effect of back-flow has not occurred for 4° above even though it has occurred foe 4° below. Moreover the flow rate has been increased to small extent for 4° above. Unfortunately, Fig. 18 shows that the pressure peak at the discharging instant has been observed so that the leakage through gap between outer rotor and inner rotor should be inevitable.

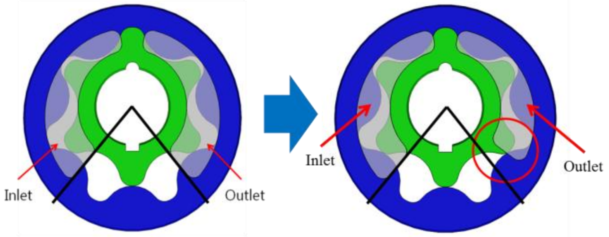


Fig. 16 Outlet configuration when discharging angle is decreased by 5°

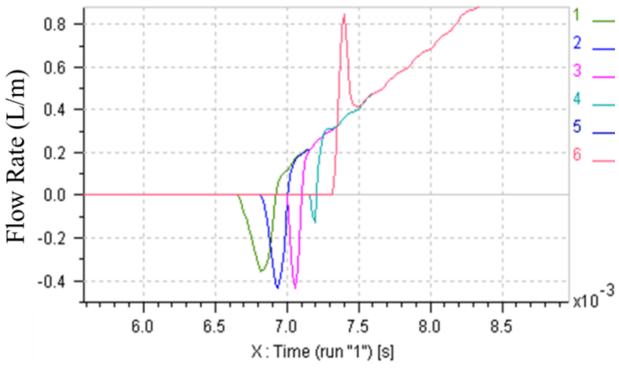


Fig. 17 Flow rate according to discharging angle of outlet

As mentioned in Nam *et al*. [2], it can be stated that the reduction of back-flow can decrease the loss of flow rate more effectively than the leakage between the gap. Consequently to decrease the discharging angle to some extent is to prevent the degradation of flow rate. Further study on the decreasing extent of discharging angle will be left for the flow rate control in detail.

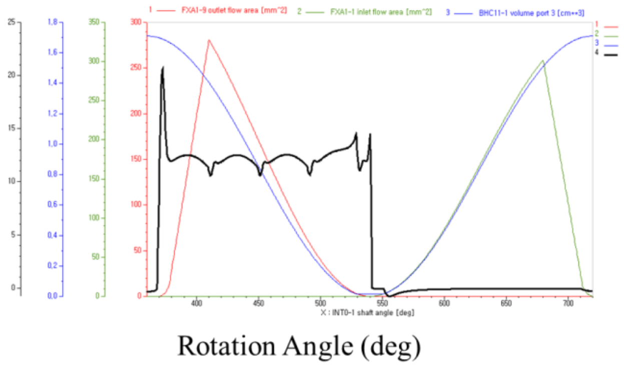


Fig. 18 Pressure distribution of outlet when the discharging time is delayed

5 Conclusions

The purpose of this paper aims at enabling field engineers to have access to the design of trochoid pump more easily and thereby to have more concrete control over the flow rate of pump by realizing its analysis model similar to its actual product model. For this purpose, first, we have used AMESim® which gives field engineers easy tool for analyzing the cavitation factors of trochoid pump including hydraulic pressure change of outlet, flow rate according to rotation speed of inner rotor, leakage through gap between outer rotor and inner rotor, and discharging angle of outlet, rather than a professional hydraulic analysis program or language such as CFD®.

In this paper, based on AMESim® with SolidWorks®, we have presented how to simulate cavitation by analyzing the control factors of trochoid pump which have influence on the degradation of flow rate efficiency. This proposed methodology of cavitation simulation will be useful for flow rate control through the fast modification of trochiod pump design. In further research, we expect that the flow rate over which a designer wants to have control would be optimized rapidly according to the change of environmental condition (e.g. the kind of hydraulic fluid, the application area of trochiod pump such as transmission or engine, *etc*.) by applying the proposed cavitation simulation methodology to the flow rate control using with MATLAB® or LabVIEW® at real time.

6 Acknowledgement

The authors of this paper were partly supported by the Second Stage of Brain Korea21 Projects

7 References

[1] S. Y. Yang, and S.J. Cha, “Simulation of Cavitating Flow in a Gerotor Oil Pump,” The Korean Society of Automotive Engineers, Autumn Con ference, 2006, pp. 152~158.

[2] K. W. Nam, S. H. Jo, and J. I. Park, “Numerical Simulation in the IC Engine Lubricating Gerotor Oil Pump,” The Korea Society of Mechanical Engineers, vol. 30, no.10, pp.1019~1025, 2006

[3] C. S. W, N. G. Kwon, and S. H. Kwon, “Flow Analysis of Automotive Oil Pump of Gerotor Type”

[4] J. S. Seo, H. S. Chung, and H. M. Jeong, “A Study on Gerotor Design with Optimum Tip Clearance for Low Speed High Torque Gerotor Hydraulic Motor”

[5] S. Neyrat, N. Orand, and D. Jonquet, “Modeling and Analysis of an Automatic Transmission Internal Gear Oil Pump with Cavitation”

[6] B. J. Kim, S. H. Seong, and S. H. Yoon, “Flow Analysis and Port Optimization of GeRotor Pump Using Commercial CFD Code”

[7] H. K. Moon, S, J. Jang, C. Kim, and H. Y. Cho, “Port Configurations on CFD Analysis in Gerotor Pump”