Proceedings of the 6th JFPS International
Symposium on Fluid Power, TSUKUBA 2005
November 7-10, 2005

THE CFD SIMULATION RESEARCH ON HYDRAULIC TRANSFORMER

Ma Jien, Xu Bing, Ouyang Xiaoping, Yang Huayong

The Key State Lab of Fluid Power Transmission and Control
Zhejiang University, Hang Zhou 310027, PR China
(E-mail: jienma@hotmail.com)
(Tell: +86 571 87952500 224)

ABSTRACT

A new hydraulic transformer (named NHT) prototype has been designed and tested by Zhejiang University in China. The results show that the HT can realize the function of transforming pressure, but its parameters such as efficiency and noise should be improved further. To help optimizing it, CFD-simulation using commercial software FLUENT is introduced. And the fluid field inside this hydraulic component can be theoretically analyzed. The velocity field and pressure distributing at different places such as the oil gap between the cylinder block and the valve plate, and between the valve plate and the end cap, or inside the cylinder chamber are founded. And the flow fields were analyzed in different rotate speeds and control angles. Besides the flow field was discussed under different grid cell numbers. The conclusions contribute to the understanding of the flow conditions inside the NHT and thereby supply dependences for optimization on the NHT.

KEY WORDS

CFD, Hydraulic transformer (HT), Simulation

INTRODUCTION

During the development of the secondary control, the idea of a hydraulic transformer enabling the extraction of controlled oil flow from the pressure net without incurring large valve losses was pursued [1]. Therefore, the new hydraulic transformer (named NHT) with three rather than two control kidneys was studied widely by company INNAS, Linköping University, Caterpillar Inc, Rexroth, Zhejiang University and so on. Today, an improved NHT prototype has been designed and tested by Zhejiang University in China. And this prototype broadened the control angle of the NHT. The characters such as flow, pressure, efficiency and noise were tested on a Common Pressure Rail (CPR) system, which could provide a steady pressure at the supplying rail.

Currently CFD (computational fluid dynamic) have become popular in application of hydraulic components analysis [2]. This technique has made very good progress in recent years. That together with the increasing speed of computers makes CFD very interesting to use in this line of work [3]. Therefore, this paper introduced an application of CFD commercial software on study of the flow configuration and pressure distribution inside the NHT. The overall aim of this investigation is to contribute to the understanding of the flow conditions inside the NHT and thereby supply the dependences for further optimization on the NHT.

THEORETIC ANALYSIS

The structure of the new hydraulic transformer is similar to a conventional seven-cylinder, bent-axis axial piston pump, except that the NHT has three control kidneys rather than two. Therefore, the problems of bent-axis axial piston pump such as cavitations, volumetric loss, and noise are all faced while investigating the NHT. The NHT's control angle can be altered by turning around the valve plate, and the flow conditions inside the NHT are changed at different control angles.

Cavitations are the main reason to place a premium on noise of hydraulic components; therefore it goes by the name of cancer in liquid flow. Cavitation or the formation of gas filled babbles occurs when the static pressure somewhere inside the NHT drops below a certain critical level [4]. These bubbles are transported to high pressure region of the NHT where they are compressed violently. These not only lead to increased volumetric loss and vibration levels but also could bring damage of the NHT if a gas bubble is located near a wall when it is compressed [2]. Hence, it is important to understand the phenomena involved inside the NHT in order to minimize the pressure drop and thereby utilize the available inside pressure in the most optimum way. A scope of our investigations is therefore to find where it might cavitate inside the NHT by resorting to CFD simulation tool.

EXPERIMENTS

A prototype of new hydraulic transformer is shown in Figure 1, which was designed by Zhejiang University. A diagram of the test setup is shown in Figure 2 and this is a CPR system, which means that all of the appliances in these are connected to the same rail and are offered the same "common" pressure. This system mainly consisted of a NHT, two fluid power pumps, two accumulators, three pressure transducers and three flow meters mounted at the inlet and outlet of the NHT, a cooling system, safe precaution units and a 200 l tank. Besides, a noise Fourier analyzer was added to register the noise level and the rotation speed was measured with an optical sensor.

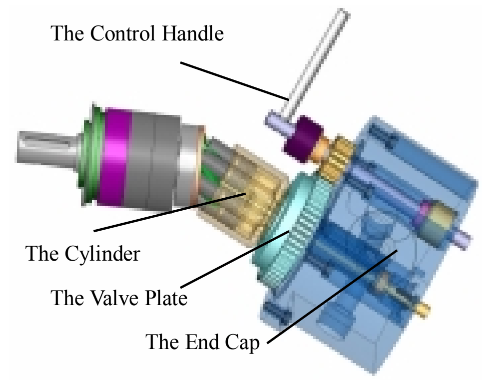


Figure 1 A prototype of new hydraulic transformer

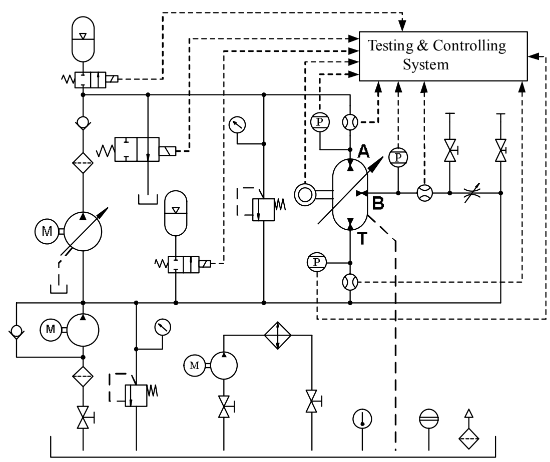


Figure 2 The test setup of a NHT

During a test procedure, the control angle was changed by the control handle of the NHT, which lead to the variety of delivery pressure. And the supply pressure for port A in Figure 2 would also be alternated to different levels by the variable pump. The noise level and the volumetric loss both increased while the rotation speed was largening. Such an increase could be studied in Figure 3 in the next page which shows the noise level and the volumetric loss as a function of the NHT rotation speed from test runs with an ordinary hydraulic oil of viscosity class ISO VG 46.

In order to investigate this further, some other approach was chosen. A way of using CFD and numerically determine the flow field inside the NHT was chosen in this investigation.

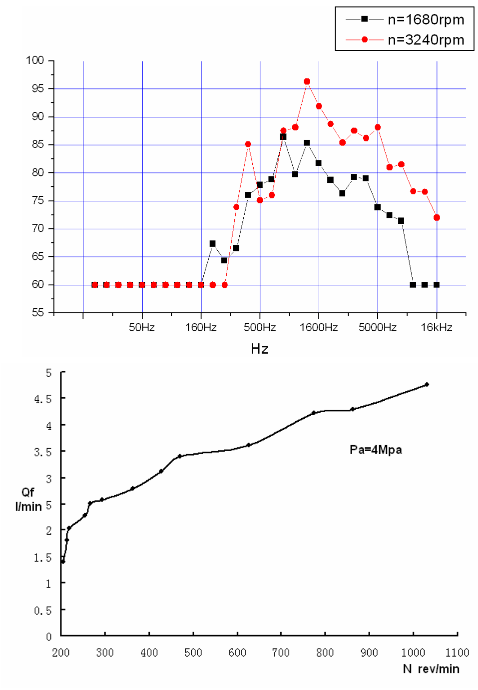


Figure 3 The noise level and volumetric loss characters with the rotation speed

CFD-SIMULATIONS

A commercial CFD software package called Fluent was used for all of the simulations in this research. This package consists of a pre-processor, Gambit for geometry and mesh generation; a combined solver; and a post-processor for analyzing and presentation of simulation results. The solver is based on the finite volume method (FVM) to discretize the governing equations for mass, momentum and energy transfer [5].
The internal flow geometry of the NHT comprises of three inlet/outlet manifolds, the cylinders, the valve plate, two thin gaps, one between the valve plate and the cylinder and another between the valve plate and the end cap. This geometry would introduce difficulties for analysis as the computational domain, and there is a need for simplification, otherwise the CFD-model will become very large and time-consuming to solve even on very powerful computers.
The final solution was to separate this geometry into two parts, the upper one and the lower one, which are represented in Figure 4 below with a control angle of 60°. Each part includes three geometries at different control angles with different rotation speeds. Specially, a two-dimensional CFD computations focus on the cylinders was analyzed to see the detailed flow situations at this part, because the flow field is very complex here and cavitation is most likely to occur.
The properties of the fluid were chosen according to the same hydraulic oil with a fluid density of 910 kg/m^3 and a viscosity of 0.0048 kg/m-s that was used in the experiments and such a fluid makes it possible to use incompressible flow in the simulations.
Under normal operating conditions, the flow inside the NHT is probably laminar with some local disturbance of turbulent flow. Since the aim was to keep the simulation as simple as possible, laminar flow was used in the three-dimensional simulations. While as for the two-dimensional simulation turbulence model was chosen to reach the accurate results for the increase in computing effort is acceptable even at this flow situation.
Three different viewing sections of the NHT are used to present the results in this paper:

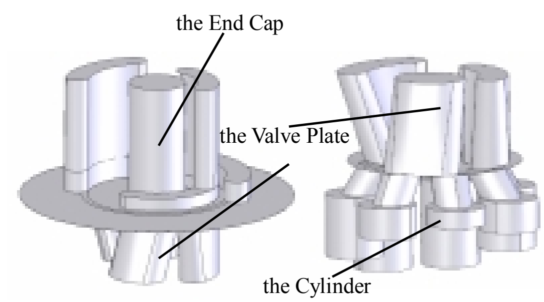
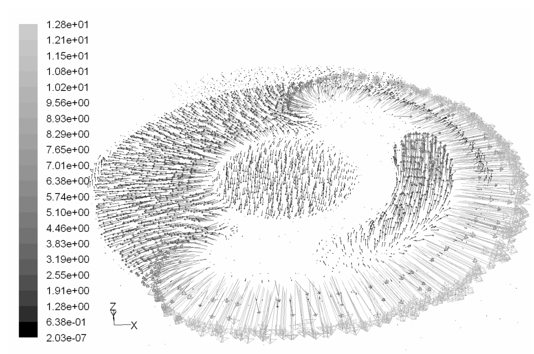


Figure 4 The 3D simulation geometries

Results of the upper part 3D simulation

The geometry of the upper part includes three inlet/outlet manifolds, part of the valve plate, and the thin gap between the valve plate and the end cap, which are shown on the left of Figure 4. The pictures in Figure 5 and Figure 6 show the surface of a cut-out at the middle section of the thin gap, including velocity vectors on the upper part and static pressure contours on the lower. The pictures in Figure 5 and Figure 6 are the results of the simulation at the same situations except for the different grid cell numbers.



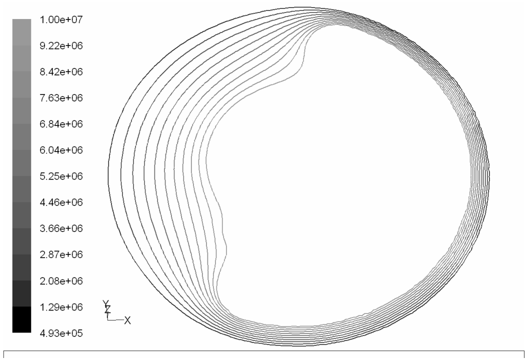


Figure 5 Velocity vectors and pressure contours of the upper part with a mixed grid cells number of 466 000

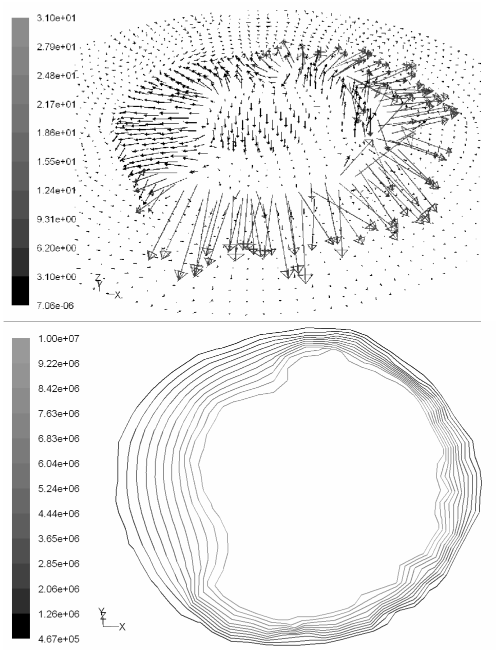


Figure 6 Velocity vectors and pressure contours of the upper part with a mixed grid cells number of 42 000

Results of the lower part 3D simulation

The geometry of the lower part includes the cylinders, part of the valve plate, and the thin gap between the valve plate and the cylinder, which are shown on the right of Figure 4. The pictures in Figure 7 and Figure 8 show the surface of a cut-out at the middle section of the thin gap, velocity vectors on the upper part and pressure contours on the lower. The pictures in Figure 7 and Figure 8 are the results of the simulation at the same situations except for the different grid cell numbers.

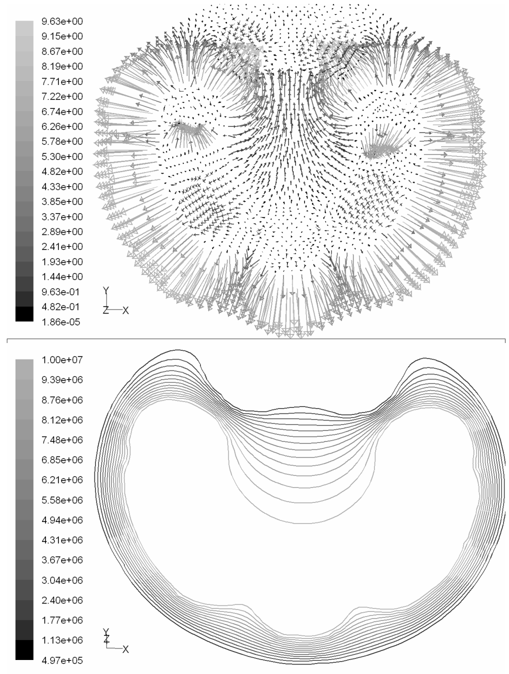


Figure 7 Velocity vectors and pressure contours of the lower part with a mixed grid cells number of 258 000

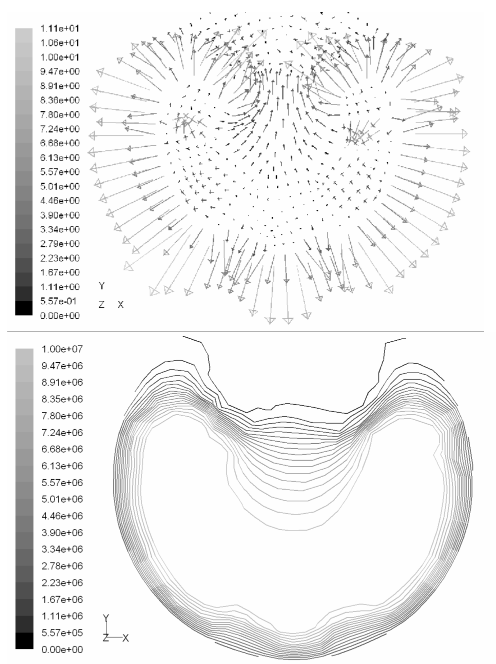
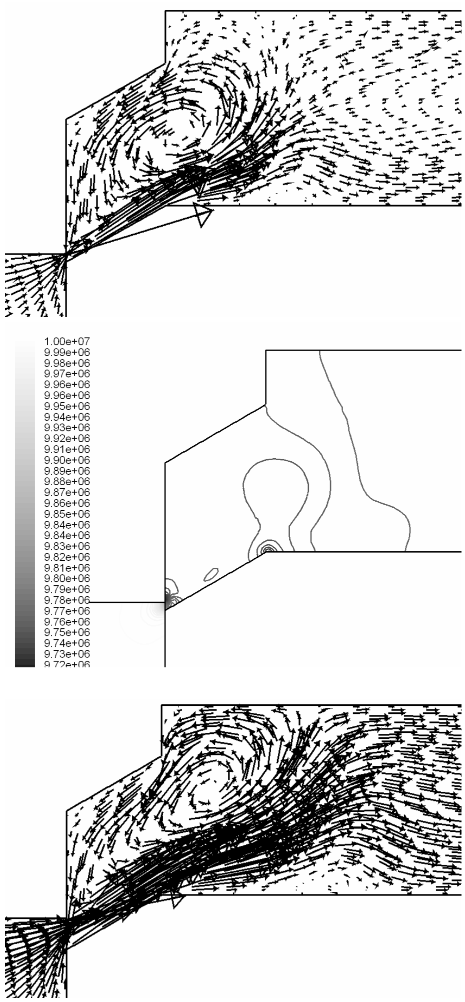


Figure 8 Velocity vectors and pressure contours of the lower part with a mixed grid cells number of 26 000

Results of the cylinders 2D simulation

The liquid flow in the cylinders is probably the most strong turbulent and likely to lead cavitation while a cylinder is just reaching or leaving a control kidney, which we named the opening area and the closing area. Because the valve plate of the NHT has three control kidneys and three typical control angles are investigated here, there were eighteen (3×3×2) groups of results totally. The pictures in Figure 9 show a group of results of cylinders' two-dimensional simulations at the control angle of 60° at the opening area of the high pressure kidney port A, velocity vectors on the upper and the corresponding pressure contours at the near following. The left part of these pictures represents the curvature of the valve plate port and the right represents a cylinder at different positions. The pressure levels of the pressure contours in these figures are the static pressure relative to the inlet pressure.



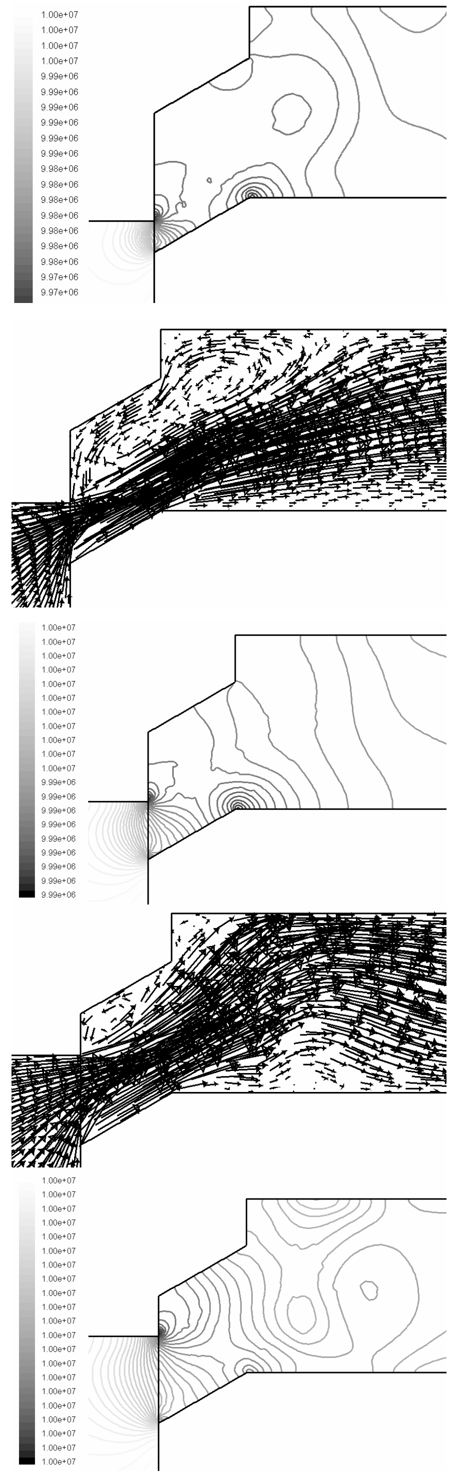


Figure 9 Results of the cylinders 2D simulation

DISCUSSION

According to Figure 5 to Figure 8, there are radial flow fields in the thin gaps because of the pressure difference. The volumetric loss can be calculated using this CFD software. Simulation results show that the volumetric loss is 7.8×10^{-5} l/min at the thin gap between the valve plate and the end cap and is 0.59l/min between the valve plate and the cylinder with a rotation speed of 1000 rpm. While the experiment result from the right picture in Figure 3 revealed the volumetric loss of about 4.5l/min at the rotation speed 1000 rpm. This can be concluded that the volumetric loss through the thin gaps is part of the total result under experiment and this is not the whole reason of volumetric loss. Another probable reason is the oil leaking at the cylinder. It can be studied from the figures that the volumetric loss is much lower at the thin gap between valve plate and end gap than that between the valve plate and cylinder. On the other hand, the liquid fields indicate that the total liquid loss includes not only the difference between measured inlet and outlet flow rate but also the liquid leaking inside the NHT from higher pressure kidney port to lower ones through the thin gaps. This phenomenon is difficult to observe and calculate through experiment.

The question is if the accuracy of the CFD-simulation is good enough to be used for the conclusions. Considering the complexity of the geometry, the cells number is probably a big source of error in the simulation. Therefore, results at different cells numbers were shown. And there is just a difference of smooth. Maybe to some extent the cell density only affects the accuracy of the value in each discrete point and not so much the flow field in general. So the results are relatively reliable.

The results in Figure 9 indicate the areas in the flow field with low pressure that could be important for cavitations. It is the area just behind the moving direction of cylinders, in which a vortex with low pressure generated according to these simulations. A smoother edge on the front of the cylinder will probably give a more favorable flow into the cylinder since it will reduce the size of the vortex. The results also show that the low pressure area at the high pressure kidney port A is much lower compare with the other two kidneys. The pressure of the low pressure areas is decrease with the rotating speed increasing, i.e. control angle increasing. Based on the theoretic analysis, this conclusion offers a possible explanation to the phenomenon that the noise level was heightened while the control angle increasing.

CONCLUSION

The simulation results correlate well with the experiments in this investigation. A possible explanation of the noise and the volumetric loss behavior is offered by the CFD-simulations. Furthermore, the use of CFD simulation tool can provide additional information on the direct acting the NHT flow configuration and pressure distribution; thereby supply the dependences for further optimization on the NHT. In general this investigation reveals the usefulness of the CFD-simulations in the NHT design.

Acknowledgement

The authors would like to express their gratitude towards the National Natural Science Foundation of China for providing necessary funding of this project (the grants No.50305032).

REFERENCES

1. Prof. Dr-Ing. Dr. h. c. mult. Wolfgang Backe, What are the Prospects Facing New Ideas in Fluid Power, Fluid Proceedings of the Sixth International Conference on Power Transmission and Control ICFP'2005, pp. 6-15.
2. Roger Yang, CFD Simulation inside Hydraulic Valves, SAE off-Highway Engineering, June 2002.
3. P-E Wiklund, G C Svedberg, Three-dimensional CFD-a Possibility to Analyse Piston Pump Flow Dynamics, Advances of CFD in Fluid Machinery Design, 2003, PP. 159-170.
4. Pär-Eric Wiklund, Suction Dynamic of Axial piston Pumps, [D], Royal Institute of Technology, KTH, Sweden, 1998.
5. FLUENT Inc., User's Guide 6.0[M], USA: Fluent Inc., 2001.