The CFD Analysis of Twin Flapper -nozzle Valve in Pure Water Hydraulic

Li Zhang, Jing Luo*, Rui-Bo Yuan, Min He

Institute of Fluid Power Control Engineering, Faculty of Mechanical and Electrical Engineering, Kunming University of Science and Technology, Kunming 650093, China

Abstract

Based on aqueous medium with low viscosity, poor lubrication, high vapor pressure and corrosion, and other special physical, chemical properties, this paper analyzed the pure water flow through the valve port, the cavitation corrosion, spiral vortex, energy loss, temperature rise and other issues may occur near the entrance. Flow field model has been established by the SolidWorks software. And the internal flow field in visual simulation analysis has been done through the FLUENT software. The pressure, speed, temperature, turbulent kinetic energy cloud pictures have been obtained, the analytical results has been validated. Meanwhile it prepared for structure optimization of pilot valve.

© 2011 Published by Elsevier Ltd. Selection and/or peer-review under responsibility of Kunming University of Science and Technology

Keywords : water hydraulic; FLUENT; twin flapper -nozzle valve; CFD

1. Introduction

Pure water hydraulic transmission technology is a fluid transmission and control in one of the new direction of development. It has broad application prospects. Compared with conventional hydraulic oil, pure water has advantages of clean, safe, inexpensive, energy saving, wide range of sources. But since its low viscosity, poor lubrication, high vapor pressure, corrosion and other shortcomings, and the water flows through the valve port, it often appears spiral vortex in the local area and the speed of the redistribution, fluid irregular rotation, collision, reflux, the movement of energy consumption, resulting in stress, energy loss, and even produces cavitations, noise, damages to the body and shortens the service

* Corresponding author. +86-13708413265
Email address:luojing9539@126.com
life of hydraulic valves. Therefore for water pressure twin flapper -nozzle valve and its internal flow field of study is very necessary [1].

This paper according to the valve port characteristics of the water pressure twin flapper -nozzle electric-hydraulic servo valve pilot valve, applies software FLUENT to valve port flow field of CFD analysis, and then qualitative determines the degree and the location of the cavitations erosion, whirlpool problems. The results of pure water hydraulic components design and structure optimization have the certain reference value.

2. Water pressure twin flapper -nozzle electric-hydraulic servo valve structure and principle

Structure of the water pressure twin flapper -nozzle electric-hydraulic servo valve is shown in Figure 1. Since water has a strong corrosive, the materials used in conventional hydraulic control valves can not be directly used for the production of the water hydraulic control valves. Now commonly engineering materials used in water hydraulic control valve are stainless steel, corrosion resistant alloys, engineering ceramics, and engineering plastics and so on. This paper use stainless steel to make the body of double nozzle flapper valve, use corrosion resistant alloy to make baffle and other key components of the water pressure control valves, use engineering ceramics to make valve core, valve cover and other components. As can be seen from Figure 1, no control current, the armature of the main valve located in the middle position of the upper and lower guide magnet which supported by the spring tube, baffle in the middle of the two nozzles, the main valve spool lever ball in the constraints in the median, then the main valve without pressure output. When there is differential control current, in the main valve armature to generate the counterclockwise electromagnetic torque, so that the armature plate assembly counter-clockwise around the spring rotation center deflection, push the main valve spool moves. When the water pressure at both ends of the spool rod deformation on the spool and the feedback generated by the reaction and the fluid momentum balance slide valve, the spool to stop movement. When the load pressure is constant, the output flow which the main valve of water pressure twin flapper -nozzle electric-hydraulic servo and the control current in proportion [2,3].

3. Water pressure twin flapper -nozzle valve mathematical model

FLUENT has a wealth of physical models, advanced numerical analysis methods and powerful post-
processing functions can be used to simulate from incompressible to highly compressible complex flow.

3.1. Control equation

Conservation of mass and momentum conservation law describes hydraulic machinery internal fluid flow of the basic law. In the Cartesian coordinate system \((x, y, z)\), taking into account the incompressibility of fluid, steady flow and the continuity equation and momentum equation of mathematical expression [4]:

\[
\begin{align*}
\frac{D\rho}{Dt} + \rho \vec{v} \cdot \nabla \vec{v} &= 0 \\
\rho \frac{D\vec{v}}{Dt} &= -\nabla p + \mu \nabla^2 \vec{v}
\end{align*}
\]

(1)

(2)

Engineering applications, the basic method of calculating turbulent for instantaneous equations to homogenization solution, instantaneous value=average value +pulsating value, and according to Renault average law, can get the following equation:

\[
\frac{\partial}{\partial x_i} (\rho \bar{u}_i) = 0
\]

(3)

\[
\frac{\partial}{\partial x_i} (\rho \bar{u}_i \bar{u}_j) = - \frac{\partial}{\partial x_j} \left( \frac{\mu}{\rho} \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \frac{\mu}{\rho} \frac{\partial \bar{u}_i}{\partial x_j} \right) - \rho \bar{u}_i \bar{u}_j \right)
\]

(4)

\(k\) equation:

\[
\rho u_i \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \eta + \frac{k}{\sigma_s} \right) \frac{\partial k}{\partial x_j} \right] + \eta \frac{\partial \bar{u}_i}{\partial x_j} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_i}{\partial x_i} \right) - \rho \varepsilon
\]

(5)

\(\varepsilon\) equation:

\[
\rho u_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \eta + \frac{k}{\sigma_s} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + \frac{C_{\mu} \varepsilon}{k} \eta \frac{\partial \bar{u}_i}{\partial x_j} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_i}{\partial x_i} \right) - C_{\varepsilon} \frac{\varepsilon^2}{k}
\]

(6)

\[
\eta = C_{\mu} \rho k^2 / \varepsilon
\]

(7)

Where:

\(\rho\) — fluid density;

\(\mu\) — kinematic viscosity;

\(k\) — turbulent kinetic energy;

\(u\) — average velocity;

\(\varepsilon\) — dissipation rate;

\(\eta\) — turbulent viscosity.

In the equations has three coefficient \((C_1, C_2, C_\mu)\) and two constants \((\sigma_s, \sigma_s)\). In recent years, literature values has been basically the same with the table below, in the calculation carried out by FLUENT recommended values, as shown in table 1. Equation (3) ~ (7) constitutes the control equations of the \(k-\varepsilon\) two equation model.

| \(C_\mu\) | \(C_1\) | \(C_2\) | \(\sigma_s\) | \(\sigma_s\) |
|---|---|---|---|---|
| 0.09 | 1.44 | 1.92 | 1.0 | 1.3 |
3.2. Water pressure twin flapper -nozzle valve model establishment and the flow field calculation mesh division

According to the structure of the twin flapper -nozzle valve, application SolidWorks software to build pilot valve three dimensional flow model. Because the different displacement of will make baffle lead level of the flow field characteristics change, therefore, establish the paddle in different displacement fluid motion when model. Figure 2 shows the model diagram when the plate displacement is zero.

Water pressure twin flapper -nozzle valve flow field calculation mesh division is in GAMBIT grid generation program for completion, using its powerful grid generation capacity to automatic dividing the grid, then according to the structure characteristics of the flow field to refined the complex flow field [5], and finally reach the purpose of precise quickly solve. According to the established model of the CFD flow chart, in the process of use GAMBIT meshing, the initial set tetrahedral (Tgrid) grid, are static meshes, the interval is set to 0.8, so that the node with the minimum number to get as many number of grids, thus facilitating the solution. Finally obtained the grids as shown in figure 3 below.

3.3. Setting the flow field boundary conditions

Use FLUENT6 to define the boundary conditions of grid computing model, as shown in figure 2, define the fluid of import and export, the remaining boundary is defined as the WALL.

Use FLUENT6 to simulation calculation, set the plate displacement is 0 mm, import boundary
pressure of valve is 21 MPa and outlet boundary pressure of valve is the outside atmospheric.

3.4. Assumptions conditions

Because the actual three-dimensional movement of the fluid flow field is very complex. It is difficult to accurately simulated, and thus in the actual calculation of the computational model to make the following assumption [6].

(1) Take the working medium is water, and the corresponding parameters are: density: 998.2 kg/m³, kinematic viscosity: 0.001 Pa·s, coefficient of compressibility: 0.602 MPa, saturation vapor pressure: 2.34×10³ Pa, acoustic velocity: 1480 m/s.

(2) Since the compressibility of water is very small, pressure increases 101.3 KPa (each an atmospheric pressure), its less than a millionth of the volume change, so when calculation, the impact of compression can be ignored, according to the incompressible processing.

(3) The model of the Reynolds number Re = 7200, much larger than the critical Reynolds number, the water flow within the valve channel state is turbulent, so the turbulence equations were solved using the calculation.

(4) Wall relatively smooth, regardless of the influence of surface roughness for flow state.

(5) Adiabatic flow, the temperature not transmits to other substances in addition to fluid.

(6) No sliding surface.

4. Simulation results and analysis

This paper is based on the finite volume method of FLUENT software as assisted calculation. Figure 4 shows the contours of static pressure of the twin flapper -nozzle valve pilot valve, figure 5 shows the contours of velocity magnitude of the twin flapper -nozzle valve pilot valve, and figure 6 shows the contours of turbulent kinetic energy of twin flapper -nozzle valve pilot valve. The display for its internal state, observation on cut surface. In this paper use the axis of symmetry plane when \( y = 0 \), shown in figure 7.
From Figure 4, Figure 7a) two contours of static pressure can be seen, after entering the pilot valve pressure is gradually reduced, this is because the baffle in the initial position, high pressure water flow directly through the outlet of the servo valve, and can be seen from the figure, this time, the pressure on both sides of the pilot valve have the same trends. In the position of the panel, it can be seen that the pressure drop to small numerical, which may produce air bubbles.

From Figure 5, Figure 7 b), two contours of velocity magnitude can be seen, water flows through the nozzle to beat on the baffle, and the speed will produce greater change. Sudden change in speed may produce air bubbles and before entering the nozzle and the direction of fluid motion changes, will produce a small vortex, causing pressure changes, and thus may cause the valve vibration.

From Figure 6, Figure 7c), two contours of velocity magnitude can be seen, turbulent flow field of energy dissipation smaller, overall energy losses small. Maximum energy dissipation in the damper...
position is the position of maximum energy loss. Before the water by imports flows into the nozzle also can have the energy dissipation, but compared to the location of the baffle is smaller.

When the displacement of the spool is 0.002mm, the simulation results are shown in Figure 8. In order to visual representation the changes of the pressure, velocity and turbulent kinetic energy, observation on cut surface, the figure shows the axis symmetry plane when \( y = 0 \).

![Contours of static pressure](image1)

![Contours of velocity magnitude](image2)

![Contours of turbulent kinetic energy](image3)

**Figure 8  \( y = 0 \)  Contours of plane flow field state**

It can be seen from the figure, when the baffle moves a position, it makes one cavity pressure increases, another cavity pressure drops, and in the baffle position, the change of the speed is more severe in left cavity, and the two cavity in this position have vortex. Therefore, in here the energy loss is the biggest, the water temperature rises, the sharply changed of the speed and the vortex, which will produce harmful effects on the performance of the pilot valve.

5. Conclusions

Through the fluid simulation software FLUENT to simulate water pressure twin flapper-nozzle electric-hydraulic servo valve internal flow field, can be seen clearly the changes in flow field, temperature field, pressure field of the body cavity. Because the medium of pure water has special physical and chemical characteristics, from the internal flow field of the pilot valve can reach the following conclusion:

1. Simulated the flow field shows that the pressure drop in a small value near he baffle, which may produce air bubbles. Bubbles happen serious regional corresponding with the low pressure area of flow field, which degree has the directly relationship with the low pressure area range and pressure distribution. Therefore, in practical application, by the method of detecting the fluid pressure within the valve to predict the region that cavitations may occur.

2. Whirlpool occurred near the baffle and the corner of the valve chamber. The size of the energy
dissipation is concerned with the size of the vortex area. The larger the whirlpool area the greater the energy dissipation is. In the same way, the smaller the more the vortex, the greater the energy loss and the greater the flow noise are. The places at the baffle and the corner of cavity body are easy to generate energy loss and fluid noise. Because of the energy loss, leading to the temperature of the fluid have certain rise.

(3) The research method and conclusions on theory and application research of pure water hydraulic components has a very good reference value, and it has practical significance on flow structure optimization design of pure water hydraulic components.

Reference

[1] YANG Guolai, YE Qing, LIN Nan. The Research of the Flow Field Cavitation inside Water Hydraulic Poppet Valve by CFD [J]. MACHINE TOOL&HYDRAULICS, 2007(1): 148-150.

[2] CHEN Zhaoguo, DING Fan, CAI Yuehua. Research on Characteristics of Single-stage Twin Flapper -nozzle Electro hydraulic Servo Valve [J]. MACHINE TOOL&HYDRAULICS, 2007(4): 114-116.

[3] LONG Jingyu, YANG Yang, HUANG Hao, JIN Xiaohong, YAO Bi. Visual simulation of flow field inside main valve of dual—nozzle flapper electro hydraulic servo valve with Fluent [J]. Journal of Wuhan University of Science and Technology, 2010(3): 307-309.

[4] WANG Fujun etc. The Computational Fluid Dynamics Analysis. Beijing: TSINGHUA UNIVERSITY PRESS, 2004.

[5] HAN Zhanzhong etc. Fluent Fluid Engineering Simulation and Application Examples. Beijing: BEIJING INSTITUTE OF TECHNOLOGY PRESS, 2004.

[6] HE Min, YUAN Ruibo, ZHANG Zongcheng, LIU Sen, ZHANG Peng, BA Shaonan. The Particular Valve Cavity Water Hydraulic Slide Valve Flow Channel of the Modeling and Simulation Based on FLUENT [J]. Science Technology and Engineering, 2010 (1) 3081-3086.