The International Journal Of Engineering And Science (IJES)
|| Volume || 5 || Issue || 10 || Pages || PP 11-23 || 2016 ||
ISSN (e): 2319 – 1813 ISSN (p): 2319 – 1805

Effects of Surface Roughness and Fluid on Amplifier of Jet Pipe Servo Valve

Xuan Hong Son PHAM
Department of Mechanical Engineering, HoChiMinh City University of Technology, Vietnam

ABSTRACT
In manufacturing process of jet pipe electro-hydraulic servo valve, it must be having a lot of errors in the inner wall of servo valve components, which fluid flows will have a certain roughness. Jet pipe electro-hydraulic servo valve, in fact, can use variety of fluid during its working. With CFX software, this article studies effect of surface roughness of parts and working fluid on pressure characteristics of amplifier of pilot stage in jet pipe servo valve. Through mathematical models and simulation, it is shown that effect of wall roughness of parts on flow characteristics of the pilot stage amplifier is much. Analysis of many different roughness, it is found out that the greater surface roughness is, the smaller velocity of jet flow is, and the recovery pressure decreases but the magnitude of change is not much. In addition, the relationship between the viscosity of fluid and the pressure characteristics of the pilot stage is close.

Key words*: Rough surface; Smooth surface; Fluid; Amplifier.*

Date of Submission: 17 May 2016 Date of Accepted: 29 October 2016

I. INTRODUCTION

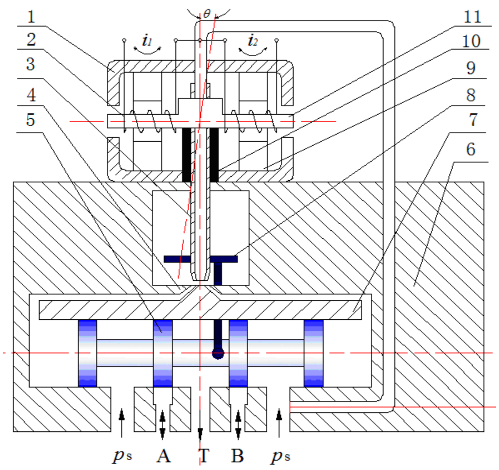
Jet pipe electro-hydraulic servo valve has high reliability and ability in pollution resistance, which can develop in the large civil aircraft. Comparing jet pipe servo valve with nozzle flapper servo valve, it can been seen that pressure and volumetric efficiency of jet pipe servo valve reach to 70 percent while the nozzle flapper servo valve is about 50 percent, as a result, the jet pipe servo valve can produce bigger pressure and control flow, so more and more authors study jet pipe electro-hydraulic servo valve.

Using CFD simulation software, jet pipe servo valve is studied under different working media, structure size effects on flow field characteristics by Li, et al., [1]. Mala,et al, [2] use model of surface roughness and viscosity (RVM) to simulate influence of wall roughness on laminar flow in micro-channel. Normally, the existence of roughness will increase transfer of momentum from boundary layer near wall, the additional momentum transfer can be led into the model of eddy viscosity turbulence which is similar to the viscosity of roughness μR. If the value of μR near wall is bigger, the channel center gradually disappears. The results of numerical simulation of RVM model and their experiment results are very well. Transition in advance caused by surface roughness can be well explained by RVM model (Mala, et al) [3]. Nie simulated ports of valve spool of circular seam, flapper nozzle valve with water and other hydraulic components in the form of a typical orifice under high pressure characteristics [4]. YueHua Yang, etc associated methods of mathematical modeling, simulation and experimental research for flow field of jet pipe electro-hydraulic servo valve pre-stage, carry out theoretical analysis and experimental research, the integrated use of computational fluid dynamics, theory of cavitation of multiphase flow, and influence of various parameters on flow field were analyzed [5]. Influence of symmetric distribution between two parallel plates of three-dimension roughness on pressure-driven flow and velocity field through micro-channels, 0.001< Re <10, is researched by Hu, et al. [6], the conclusion is the existence of roughness will increase pressure drop about 0.1 percent, simultaneously, geometric shape, size, gap and plate spacing are involved. Through the modified model of viscosity coefficient of surface roughness for gas flow within micro-tube, Chen, et al., [7] researched the influence of wall surface roughness of micro-tube on gas flow within micro-tubes. The numerical simulation agreed well with the experimental data. Using the Fluent and Gambit software, Ji, et al., [8] analyzed effect of jet pipe amplifier structure on its static characteristics, and presented flow characteristics under different deflections of jet pipe. Stokes flow near wall of sphere under the condition of rough surface was studied by Lecoq, et al., [9], they transformed rough surface into an effective smooth surface, simultaneously, in numerical analysis, slip wall condition was used instead of no slip wall condition to study sufferable resistance of sphere. Yang, et al., [10] analyzed theoretical and experimental research on flow field characteristics of orifice of circular tube, cone valve, and cavitation characteristics at valve ports also launched a discussion on the research results achieved in this paper. Using matching expansion method, Michael, et al., [11], obtained slip coefficient of flowing through rough surface. Zhou, et al., [12] established jet control equations for abrasive two-phase of liquid-solid, using the standard *k-e* turbulence model in Fluent software to carry out the numerical simulation, and the effect of surface roughness on stream-flow is concluded. Jansons proposed micro-scale slip boundary condition under rough surface [13]. Yao, et al., studied the reasonability and applicability of the flow conductance theory for jet pipe valve according to jet flow dynamics, the inside radius of jet pipe was replaced by a flow radius in the theory of flow conductance [14]. Tuck and Kouzoubov [15] researched a modified slip boundary condition to represent the effects of small roughness-like perturbations to an otherwise-plane fixed wall which acted as a boundary to steady laminar flow of a viscous fluid. This boundary condition involves a constant apparent backflow at the mean surface. Nie, et al., [16] studied the pressure distribution between two throttles of two-step throttle, and theoretical analysis or experimental research for flow field characteristics of throttle was carried out. Through the above close analyses, research on the influence of surface roughness and fluid on hydraulic amplifier of jet pipe is also rare, so this paper is proposed.

II. SIMULATION OF THE MIDDLE OF FLOW FIELD FOR JET PIPE AMPLIFIER

2.1 Schematic diagram and working principle of jet pipe electro-hydraulic servo valve

A two-stage, four-way, closed ports electro-hydraulic flow control servo valve is assumed to be the power control device for the hydraulic system which was analyzed. It converts electric signal into a hydraulic signal, with fast dynamic response, high precision, small hysteresis, good linear degree, the requirements of control signal are small. It is also a kind of high performance and high precision control component. The basic elements of a jet pipe electro-hydraulic servo valve are shown in Fig. 1, in which there are two magnetic poles of torque motor. A jet pipe electro-hydraulic servo valve can be separated into two-stages: The pilot-stage includes the torque motor, jet pipe, flexure tube and receiver holes. The second stage body includes the spool and sleeve assembly. The movement of the spool opens orifices are connected to a pressure supply and sink and to both sides of the main actuator piston.



1-Upper permanent magnet; 2-Control coil; 3-Jet pipe; 4-Receiver holes; 5-Valve spool; 6-Valve body; 7-Valve sleeve; 8-Feedback spring; 9-Lower permanent magnet; 10-Flexure tube; 11-Armature of jet pipe

Fig. 1 Schematic diagram of jet pipe servo valve

Hydraulic fluid at system pressure travels through the pilot-stage wire mesh filter into a feed-tube and out of the projector jet. The projector jet directs this hydraulic fluid stream to two receivers, each of which is connected to the second stage spool's end chambers. The pilot-stage torque motor receives an electrical signal (current to the coils) and converts it into a mechanical torque on the armature and jet pipe assembly. The torque output is directly proportional to the input current. As more current is applied to the valve, the greater forces are exerted to rotate the armature assembly around its pivot point, resulting in more fluid impinging on one receiver than the other. The result of differential pressure between the spool's end chamber triggers spool movement and, in turn, uncovers stage porting causing fluid to flow to and from, depending on spool direction, the two valve control ports. The direction of spool displacement is opposite to the jet pipe rotation. As the spool moves, the feedback spring generates a force at the jet pipe which opposes the torque motor's force. The spool continues to move until the force generated by the feedback spring equals the force produced by the torque motor. Then the jet pipe position is returned to be centred over the two receivers. A small differential pressure usually remains across the ends of the spool to overcome Bernoulli flow forces that tend to close the valve and feedback spring force. The spool displacement is proportional to the control current in the torque motor. As the spool moves, fluid is metered proportionally to and from the second stage control ports. When the input signals to the torque motor vary in amplitude and polarity, the second stage spool accurately follows the signals and meters fluid accordingly [17~19].

2.2 Simulation of the middle of flow field for jet pipe amplifier

Equation of Navier-Stokes is the principle of conservation of momentum in the form of fluid movement, the momentum equation of the direction *i* is given as follows

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\upsilon\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + F_i \qquad (1)$$

where, P is pressure; F_i is external force;
v is dynamic viscosity; ρ is density.

For laminar flow and all the relevant length, they can include turbulence in the lattice, directly solve the N-S equation is possible (by direct numerical simulation). In general, range of suitable criterion problems is even greater than range of possible model of today's massive parallel computer. In these cases, the turbulence simulation needs to introduce turbulence model. Large maelstrom simulation, and RANS expression (Navier-Stokes equation of Reynolds average), or model of *k-ε* model and Reynolds stress together, dealing with two kinds of technology of these criteria. In many instances, N-S equation or other equations are solved simultaneously.

k-ε model is for improving the mixing length model and algebraic expression of turbulence length dimension of wall complex flow. Equation of *k* is conveyance of turbulent kinetic energy, and equation of *ε* is turbulent kinetic energy dissipation rate. Standard model of *k-ε* is introduced by variable turbulent kinetic energy and control equation of turbulent kinetic energy dissipation rate. The continuity equation or momentum equation becomes

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \qquad (2)$$

$$\frac{\partial(\rho \vec{u})}{\partial t} + \nabla \cdot (\vec{u} \otimes \vec{u}) - \nabla \cdot (\mu_2 \nabla \vec{u}) = -\nabla p' + \nabla \cdot (\mu_2 \nabla \vec{u})^T + \vec{T}_1$$

$$\mu_2 = \mu + \mu_i$$

$$\mu_i = T_2 \rho \frac{k^2}{\varepsilon}$$

where $\vec{T}_1$ is physical vector, $\vec{T}_2$ is empirical constant,
μ_2 is calculation viscosity of effective turbulence,
μ_i is turbulent viscosity, p' is amending pressure.

The use of the simplified three-dimensional models is shown in Fig.2. According to the symmetry model, it can be cut from the symmetry plane. In preprocessing of CFX, fluid and flow condition are set as follows:

Fluid to be used is aviation hydraulic oil of 10, density of 850 [kg/m^3], dynamic viscosity of 0.02125 [kg/ms], activated turbulence model, and using the standard model of *k-ε*. Assuming that there is no heat exchange, and heat transfer equations closed-form, the inlet pressure is p_s = 10[MPa], the outlet pressure of 0[MPa], the atmospheric pressure of 1atm, the gap between nozzle and receiver holes is 0.6[mm], the jet pipe diameter of r = 0.3[mm], the receiver holes diameter of R=0.3[mm], and the cross-section is symmetrical , the maximum number of iterations of solving is 100 times, the maximum residual error is 0.0001, other surfaces are no-slip walls.

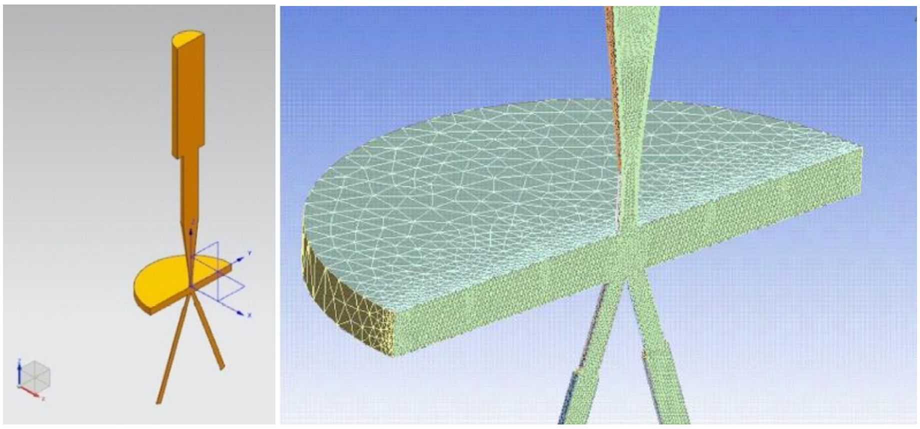


Fig.2 Simplified 3D model for jet pipe of pre-stage in the middle and CFX mesh generation

III. EFFECT OF SURFACE ROUGHNESS ON PRESSURE CHARACTERISTICS

During flowing from the inlet to the outlet, fluid will inevitably have pressure loss, which includes local pressure loss caused by change of pipeline section or energy loss caused by vortex flow nearby jet pipe exit, and local pressure loss caused by flowing of fluid in receiver holes. Further, during flowing in pipeline of wall roughness, fluid will have pressure loss and energy loss caused by its viscosity.

3.1Mathematical description of flow near wall region

For fully developed turbulent flow on fixed wall, in different distances along wall normal, flow can be divided into the wall area and the core region. It can be considered fully developed turbulence in the core region. The wall area is affected obviously by wall surface, and can be divided into three layers: viscous sub-layer, transition layer, logarithmic layer. Viscous sub-layer is a very thin layer that clings to solid wall, the viscous force plays leading role in exchange of kinetic energy, heat and mass, and all flow is laminar, velocity component parallel to wall surface along direction normal is linear distribution. Transition layer is outside of viscous sub-layer, in which effect of viscous force and turbulent shear stress is about equal, the flow condition is more complex. Because of the tiny thickness of transition layer, it usually doesn't draw obviously in project, and merge into logarithmic layer. Logarithmic layer is in the outermost layer, then the impact of viscous force is not obviously, the turbulent shear stress is mainly dominant, flow is in fully developed turbulence status, velocity distribution is near the logarithmic layer.

To describe viscous layer, two dimensionless parameters are introduced, and formulae of velocity and distance are given as follows

$$u^{+} = \frac{u}{u_{\tau}} \quad \text{and} \quad y^{+} = \frac{y\rho u_{\tau}}{\mu} = \frac{y}{\nu}\sqrt{\frac{\tau_w}{\rho}} \tag{3}$$

where, u_τ is the shear velocity, $u_{\tau} = (\tau_w / \rho)^{0.5}$

u is time-average velocity, τ_w is the wall shear stress,

y is the distance from the wall.

For the flow near the wall, and at low *Re* number, turbulence is not fully developed, effect of the turbulence impulse is not as big as fluid viscosity. Normally, using of the wall function method and the model of *k*-*ε* in low *Re* number, and coordinate the standard model of *k*-*ε* can successfully solve problems of flow calculating for the near wall and low *Re* number. The wall function for the physical quantity on the wall is directly linked to the unknown volume of turbulence core area. It must be used cooperatively with model of *k*-*ε* in high *Re* number. The basic idea of the wall function is solution model of *k*-*ε* by using flow of turbulent core area, and it isn't solved in the wall surface. Use directly semi-empirical formula, not only avoid linking between physical quantity and solving variable quantities but also avoid using calculation of viscosity turbulence model. Through low *Re* number method on the wall normal direction by using the refined mesh (very thin expanded layer) it can solve description of boundary layer.

$$\begin{aligned} u^{*} &= \sqrt[4]{C_{\mu} k^{2}} \\ y^{*} &= \frac{y\rho u^{*}}{\mu} \\ u^{+} &= \frac{1}{\kappa}\ln\left(\frac{y^{+}}{1+0.3k^{+}}\right) + C_1 \\ k^{+} &= \frac{y_R \rho u^{*}}{\mu} \end{aligned} \tag{4}$$

where, C_μ is constant of experience, turbulent viscosity is the same turbulent kinetic energy k,

C_1 is constant related to surface roughness,

κ is the Karman constant,

y_R is equivalent sand roughness.

Using ANSYS CFX method for a gradual change to the wall function and low *Re* number method, and without loss of precision. Viscous sub-layer considers $u^+ = y^+$, and in the log-law region, the near wall tangential velocity is related to the wall shear stress.

3.2 Analysis of friction coefficient for different fluid flows

It must be some errors in the manufacturing process of jet pipe servo valve, the inner wall of jet pipe module is surface that fluid runs through, and it also has certain roughness. In addition, the flowing process of fluid from the inlet to the outlet is turbulence. These factors will inevitably produce pressure loss along the way.

Two fluctuating velocity components produced in turbulent flow are the difference between laminar and turbulence, including u' for the velocity in the flow direction and v' for the velocity in the perpendicular to the flow direction. Viscous sub-layer is a very thin layer close to solid wall, the viscous force plays leading role in exchange of kinetic energy, heat and mass, and almost all flow is laminar. The velocity component parallel to the wall in the normal direction along the surface is linear distribution.Considering the *Re* number, flowing in thin tube is turbulence if $Re > 2320$, thus there is a relation between pressure loss along the length of the tube Δp and other parameters such as velocity, density, diameter, length and friction coefficient, and given as following:

$$\Delta p = \lambda\rho \frac{V_0^2 \chi}{2D} \qquad (5)$$

where V_0 is the mean fluid velocity,
χ is the length of tube,
λ is the friction damping ,
D is the diameter of tube,
ρ is the fluid density.

From (5), it can be seen that the along way pressure loss in thin tube Δp is proportional to the friction damping λ. The greater the roughness is, the bigger the pressure loss is. Therefore, in jet pipe electro-hydraulic servo valve, the roughness of thin tube surface of hydraulic amplifier must be fitting. Conversely, if there is no roughness in the inner surface of jet pipe, the flow in thin tube is stuck. Coefficient of frictional resistance is a dimensionless factor, which mainly depends on the tube diameter or velocity, viscosity and density of fluid, and it is also a function of the wall roughness.

In thin tube of surface roughness, assuming that e is the absolute roughness and δ is the thickness of laminar boundary layer. Because coefficient of frictional resistance is a function of the inner wall roughness, so it depends on the size and the shape of roughness factor. From the specific roughness factor e, setting up the ratio of e/D and it is called relative roughness. Through relationship between e and δ, fluid motion is divided by three cases, including hydraulically smooth flow ($e < \delta$), hydraulically rough flow ($e > \delta$) and turbulent transition ($e = \delta$). Hydraulically smooth is steady turbulence: when the laminar boundary layer thickness is greater than the absolute roughness ($e < \delta$), the energy loss depends on fluid viscosity. Wall roughness almost isn't affected by changing of flow velocity in laminar boundary layer region, and fluid flow itself generates turbulence at this moment, i.e. $\lambda = f(Re)$. Hydraulically rough is rough wall turbulence: when the absolute roughness is greater than the thickness of laminar boundary layer ($e > \delta$), the roughness mainly causes turbulence to fluid flow, hence, flowing will encounter obstacle of roughness, then generate resistance. The relative roughness e/D is formed from different surface roughness elements, thus it influences to flow characteristics of hydraulic amplifier strongly, i.e. $\lambda = f(e/D)$. If coefficient of frictional resistance increases, energy loss also increases, therefore, recovery pressure will be small. Turbulent transition: when the absolute roughness and the thickness of laminar boundary layer are the same ($e = \delta$), there are two reasons of yielding turbulence, including turbulent motion characteristics due to roughness and smooth motion characteristics, they exert together their influence, i.e. $\lambda = f(e/D, Re)$. Pressure loss is the internal roughness of tube wall and energy loss is fluid viscosity will together occur, thus recovery pressure will be small, too.

The roughness of the inside tube wall of jet pipe pre-amplifier is shown in Fig.4

If fluid in tube is laminar flow, coefficient of frictional resistance is: $\lambda = 64/\mathrm{Re}$ .According to Prandtl's law, total resistance in turbulence equals to the sum of viscous resistance and additional resistance

$$\tau = \mu \frac{du}{dy} + \rho\chi^2 \left(\frac{du}{dy}\right)^2 \qquad (6)$$

where χ is Prandtl's mixing length, $\chi = \kappa.y$;
μ is dynamic viscosity;
v is kinetic viscosity.

When $y > \delta$, total resistance in turbulence is

$$\tau = \rho\chi^2 \left(\frac{du}{dy}\right)^2$$

$$\Rightarrow \sqrt{\frac{\tau}{\rho}} = \kappa.y\frac{du}{dy} = u_\tau$$

$$\Rightarrow \frac{du}{u_\tau} = \frac{1}{\kappa}\frac{dy}{y}$$

So
$$\frac{u}{u_\tau} = \frac{1}{\kappa}\ln(y) + C_1 \qquad (7)$$

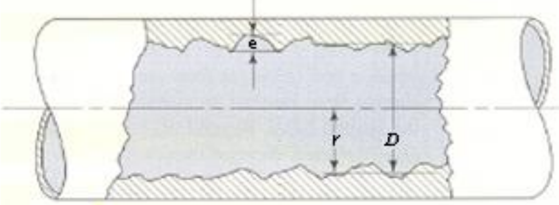


Fig.4 Local wall roughness of jet pipe pre-amplifier

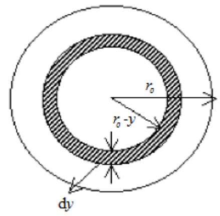


Fig.5 Differential of ring area

(**1**) Friction resistance coefficient analysis of hydraulic smooth (semi-empirical formula)

If the laminar boundary layer thickness is greater than the absolute roughness $\delta = \frac{5\nu}{u_\tau} > e$, it is called hydraulic smooth. Considering flow rate of the laminar boundary layer u_w and the boundary layer thickness δ. Let: $N = \frac{u_w}{u_\tau} = \frac{u_\tau \cdot \delta}{\nu}$, then it can obtain:

$$\delta = \frac{N\nu}{u_\tau} \tag{8}$$

In addition, from (7) it can be seen that: $\frac{u_w}{u_\tau} = \frac{1}{\kappa}\ln(y) + C_1$, then $N = \frac{1}{\kappa}\ln(\delta) + C_1$

$$\Rightarrow N = \frac{1}{\kappa}\ln\left(\frac{N\nu}{u_\tau}\right) + C_1$$

$$\Rightarrow C_1 = N - \frac{1}{\kappa}\ln\left(\frac{N\nu}{u_\tau}\right) = N - \frac{1}{\kappa}\ln\left(\frac{\nu}{u_\tau}\right) - \frac{1}{\kappa}\ln(N) \tag{9}$$

Combining Eq. (9) to Eq. (7), it can be obtained as follows: $\frac{u}{u_\tau} = \frac{1}{\kappa}\ln\left(\frac{yu_\tau}{\nu}\right) + N - \frac{1}{\kappa}\ln(N)$

$$u^+ = \frac{1}{\kappa}\ln(y^+) + G_1 \tag{10}$$

where $G_1 = \frac{u_\tau y}{\nu} - \frac{1}{\kappa}\ln\left(\frac{u_\tau y}{\nu}\right)$, $u^+ = \frac{u}{u_\tau}$, and $y^+ = \frac{u_\tau y}{\nu}$

According to conditions of the Nikuradse's hydraulic smooth experiment, it is κ of 0.4 and G_1 of 5.5.

On the wet area of circle, considering the distance from the circle centre to the differential of the ring area is (r_0 - y), and determination of the ring area is shown in Fig.5.

Let $\mathrm{d}\omega = 2\pi(r_0 - y)\mathrm{d}y \quad \Rightarrow \frac{\mathrm{d}Q}{u_\tau} = \frac{u}{u_\tau}\mathrm{d}\omega = \left[\frac{1}{\kappa}\ln\left(\frac{yu_\tau}{\nu}\right) + \frac{yu_\tau}{\nu} - \frac{1}{\kappa}\ln\left(\frac{yu_\tau}{\nu}\right)\right]2\pi(r_0 - y)\mathrm{d}y$

$$\Rightarrow \frac{Q}{u_\tau} = \iint_\omega \frac{\mathrm{d}Q}{u_\tau} = 2\pi\int_0^{r_0}\left[Ar_0 - Ay + \frac{r_0}{\kappa}\ln\left(\frac{yu_\tau}{\nu}\right) - \frac{y}{\kappa}\ln\left(\frac{yu_\tau}{\nu}\right)\right]\mathrm{d}y$$

$$\frac{Q}{u_\tau} = 2\pi\left[Ar_0 y - \frac{A}{2}y^2 + \frac{r_0}{\kappa}y\ln\left(\frac{yu_\tau}{\nu}\right) - \frac{r_0}{\kappa}y - \frac{1}{2\kappa}y^2\ln\left(\frac{yu_\tau}{\nu}\right) + \frac{1}{4\kappa}y^2\right]_0^{r_0}$$

where dω is the ring area,

Q is the volume flow rate through thin tube.

Meanwhile, $\int \ln\left(\frac{yu_\tau}{\nu}\right)dy = y\ln\left(\frac{yu_\tau}{\nu}\right) - y$

and $\int y\ln\left(\frac{yu_\tau}{\nu}\right)dy = \frac{y^2}{2}\ln\left(\frac{yu_\tau}{\nu}\right) - \frac{y^2}{4}$

Moreover, $\lim_{y\to 0}\left[y\ln\left(\frac{yu_\tau}{\nu}\right)\right] = 0$,

and $\lim_{y\to 0}\left[y^2\ln\left(\frac{yu_\tau}{\nu}\right)\right] = 0$.

This indicates that:

$$\frac{V_0}{u_\tau} = \frac{yu_\tau}{\nu} - \frac{1}{\kappa}\ln\left(\frac{yu_\tau}{\nu}\right) - \frac{3}{2\kappa} + \frac{1}{\kappa}\ln\left(\frac{r_0 u_\tau}{\nu}\right) \tag{11}$$

Then, $\tau_0 = \frac{\lambda}{8}\rho V_0^2$

$$\Rightarrow u_\tau = \sqrt{\frac{\tau_0}{\rho}} = V_0 \times \sqrt{\frac{\lambda}{8}}$$

Combining $u_\tau = V_0 \times \sqrt{\frac{\lambda}{8}}$ to Eq. (11), it is rewritten as follows:

$$\frac{1}{\sqrt{\lambda}} = \frac{1}{\sqrt{8}}\left[\frac{yu^*}{\nu} - \frac{1}{\kappa}\ln\left(\frac{yu^*}{\nu}\right) - \frac{3}{2\kappa} + \frac{1}{\kappa}\ln\left(\frac{1}{2\sqrt{8}}\right)\right] + \frac{1}{\kappa\sqrt{8}}\ln\left(\mathrm{Re}\sqrt{\lambda}\right)$$

$$\Rightarrow \frac{1}{\sqrt{\lambda}} = 0.88\ln\left(\mathrm{Re}\sqrt{\lambda}\right) - 0.91 \quad \text{or}$$

$$\frac{1}{\sqrt{\lambda}} = 2.0262\log\left(\mathrm{Re}\sqrt{\lambda}\right) - 0.91 \qquad (12)$$

where V_0 is the mean velocity on the cross-section. Range of Re number is: $3\times10^3 < Re < 10^6$.

(**2**) Friction resistance coefficient analysis of hydraulic rough (semi-empirical formula)

If the laminar boundary layer thickness is smaller than the absolute roughness $\delta = \frac{5\nu}{u_\tau} > e$, it is called hydraulic rough. Then the expression of velocity given as follows: $\frac{u_w}{u^*} = \frac{\ln(me)}{\kappa} + const$

Thus, $const = \frac{u_w}{u^*} - \frac{\ln(m)}{\kappa} - \frac{\ln(e)}{\kappa}$ (13)

where u_w is wall velocity of laminar boundary layer under the condition of $y = me$.

Combining Eq. (13) to Eq. (7), the velocity distribution is rewritten as follows:

$$\frac{u}{u^*} = \frac{1}{\kappa}\ln\left(\frac{y}{e}\right) + \frac{u_w}{u^*} - \frac{\ln(m)}{\kappa}$$

Let $G_2 = \frac{u_w}{u^*} - \frac{\ln(m)}{\kappa}$,

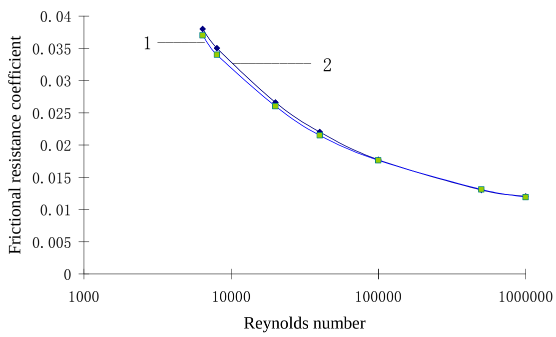
then $\frac{u_w}{u^*} = \frac{\ln(me)}{\kappa} + G_2$

$$\Rightarrow \frac{1}{\sqrt{\lambda}} = \frac{1}{\sqrt{8}}\left(B - \frac{3}{2\kappa} - \frac{1}{\kappa}\ln 2\right) + \frac{1}{\kappa\sqrt{8}}\ln\left(\frac{D}{e}\right)$$

According to conditions of the Nikuradse's hydraulic rough experiment, it is κ of 0.4 and G_2 of 8.48. Thus,

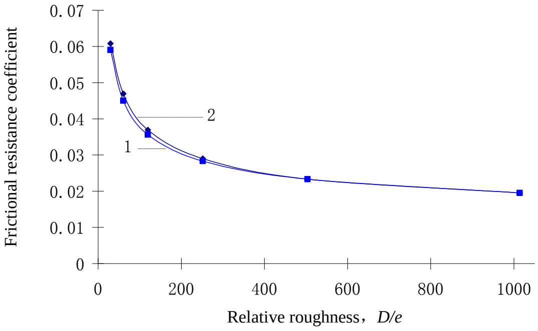
$$\frac{1}{\sqrt{\lambda}} = 1.06 + 2.03\log\left(\frac{D}{e}\right) \qquad (14)$$

From Eq.(14), it can be seen that resistance coefficient λ is proportional to the absolute roughness e. If resistance coefficient λ and e increase together, level of turbulence will increase, then energy dissipation will also increase and strongly affect to the pressure magnitude in jet pipe. Associating Eq. (5) and Eq. (14), it can be seen that when value of roughness is bigger, energy loss of fluid in tube is also bigger, as a result of this, the pressure loss along the length of the tube Δp is proportional to the absolute roughness e. Calculation of friction coefficient should give in Fig. 6*a* and Fig. 6*b*.



1. Coefficient of calculating friction resistance
2. Nikuradse's frictional resistance coefficient

Fig.6*a* Coefficient of λ for hydraulic smooth region.



1. Coefficient of calculating friction resistance
2. Nikuradse's frictional resistance coefficient

Fig.6*b* Coefficient of λ for hydraulic rough region

3.3 CFD simulation in cases of wall roughness

Aviation hydraulic fluid 10 is used, its density is 850 [kg/m^3], kinetic viscosity of 0.02125[kg/ms]. Turbulent model and standard model of k - ε are activated. Assuming there is no heat exchange, heat transfer is closed, the atmospheric pressure is 1[atm]. Taking symmetric plane of tube while other surface is not slippery, its wall roughness is e = 6.4 [μm]. Simulation parameters for pressure characteristics are shown in Table 1. Result of simulation CFD should look like Fig. 7. According to Fig.7, it can be seen that flow characteristics are not affected much in case of the wall roughness of 6.4[μm], and its flow is similar. Node velocities on a cross-section C which is perpendicular to symmetric plane are analysed, C is located the inner jet pipe, its distance from the nozzle end is 0.45[mm].

Velocity distribution on cross-section under different wall roughness is shown in Fig.8.

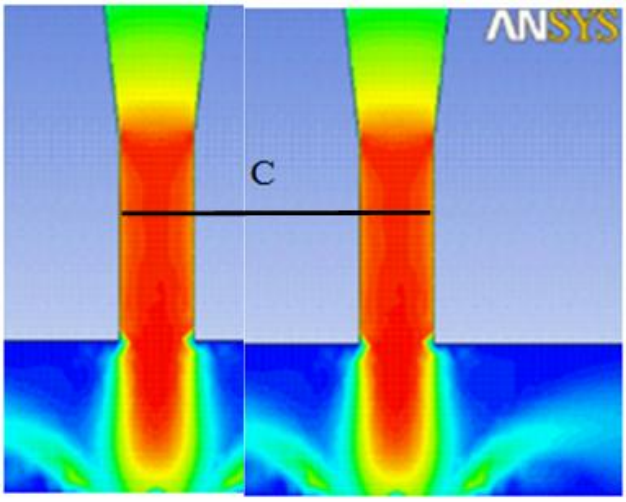


Fig.7 Comparison of cloud image of flow velocity for smooth wall with rough wall

Table 1. Simulation parameters for pressure characteristics

Parameter	Value
Boundary condition of the inlet p_s	10 [MPa]
Boundary condition of the outlet	0 [MPa]
Wall roughness e	6.4 [μm]
Density of aviation oil 10	850 [kg/m^3]

The number of grid nodes on cross-section C in nozzle is ten, i.e. velocity distribution on cross-section has ten data points while the wall surface doesn't include. Because of no slip wall, fluid velocity in wall is zero. In Fig.8, there is difference in velocity of nodes on cross-section C. At the same locations on cross-section, velocity of point in case of rough wall is slower than velocity of point in case of smooth wall. From curve 1 to curve 2, velocity is distributed with the roughness in turn as follows: e = 0; 0.8; 1.6; 3.2 and 6.4[μm]. It can be seen that value of roughness increases, the jet velocity will decrease, but the change of velocity is limited. The velocity at cross-section core in simulation CFD for pressure characteristics is shown in Table 2. In case of smooth wall, the velocity at cross-section core is 143[m/s] while the velocity is 142.2[m/s] in roughness wall, e = 6.4[μm].

Table 2. Fluid velocity at cross-section core under different surface roughness

e [μm]	0	0.8	1.6	3.2	6.4
V_0 [m/s]	143	142.62	142.55	142.48	142.2

Pressure drop affected by wall roughness is shown in Fig.9 .

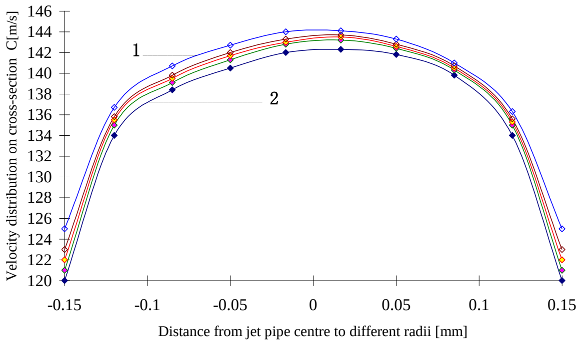


Fig.8 Velocity distribution under different wall roughness

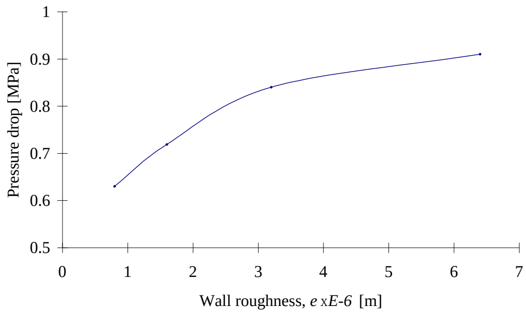


Fig. 9 Pressure drop affected by wall roughness

Therefore, the changes of wall roughness will impact on pressure loss Δp_a in thin tube. For the left receiver hole, the recovery pressure in the case of smooth wall minus the recovery pressure in the case of rough wall equals to the pressure loss Δp_a , and that is the pressure loss of fluid in the left receiver hole because of the rough wall.

In case of C_v = 0.91 and d = 0.3[mm], the approximate formula of the pressure drop can be given as follows

$$\Delta p_a = \frac{p_s - p_i}{\left(d / \lambda \chi C_v^2\right) + 1} = \frac{\chi (p_s - p_i)}{1.525 (\log e)^2 - 9.146 (\log e) + 13.713 + \chi} \quad (15)$$

where $\Delta p_a = p_a - p_a^+$,

p_a is the recovery pressure in smooth wall, and

p_a^+ is the recovery pressure in rough wall.

p_s is the supply pressure,

p_i is in-between pressure existing at the surface of receiver holes,

χ is the length of thin tube, e is the absolute roughness.

IV. EFFECT OF FLUID VISCOSITY ON PRESSURE CHARACTERISTICS

4.1 Relation between the viscosity of fluid and the pressure loss

Viscosity is a property of fluid to prevent occurring itself shear deformation, it exists in inside fluid. To overcome its own internal friction is bound to work. Thus, viscosity of fluid is the root of forming the loss of mechanical energy. Consider a thin tube of circular cross-section, force balance diagram of flow volume molecular as shown in Fig.10

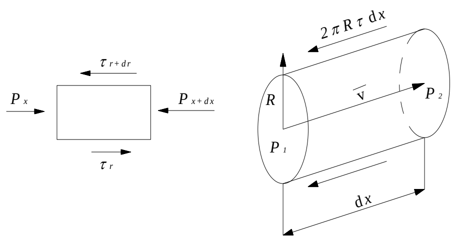


Fig. 10 Force balance diagram of flow volume molecular

From Fig.10, force balance equation for flow volume molecular is given by

$$(2r\pi P\,dr_x)_x - (2r\pi P\,dr_x)_{x+dx} + (2r\pi\tau\cdot dx)_r - (2r\pi\tau\cdot dx)_{r+dr} = 0 \Rightarrow \frac{\mu}{r_x}\frac{d}{dr_x}\left(r_x\frac{du}{dr_x}\right) = \frac{dP}{dx}$$

$$\Rightarrow\ u(r_x) = -\frac{R^2}{4\mu}\left(\frac{dP}{dx}\right)\left(1-\frac{r_x^2}{R^2}\right)$$

$$\Rightarrow\ \frac{dP}{dx} = \frac{P_2-P_1}{dx}$$

And the average velocity is determined from its definition, it gives

$$V_0 = \frac{2}{R^2}\int_0^R r_x\cdot u(r_x)dr_x = \frac{-R^2}{8\mu}\left(\frac{dP}{dx}\right) \qquad (16)$$

$$\Rightarrow\ \Delta P = P_1 - P_2 = \frac{32\,\mu\chi\,V_0}{D^2}$$

$$\Rightarrow\ \overline{Q} = \frac{\Delta P\,\pi D^4}{128\,\mu\chi} \qquad (17)$$

From Eq.(17), it can be seen that parameters of χ and D are constants in jet pipe electro-hydraulic servo valve, therefore, the pressure loss ΔP is not subordinate to χ and D. Furthermore, the gap between receiver holes and nozzle end, b, is a constant, thus the volume flow rate $\overline{Q}$ does not vary either. It has been found that the pressure loss ΔP is only subordinate to the dynamic viscosity, μ, and the greater the dynamic viscosity is, the more the pressure loss has. In addition, the pressure loss is proportional to viscosity of fluid, and if there is no friction coefficient of fluid flow, the pressure loss will be zero. Therefore, there is a close relation between the pressure loss and viscosity of fluid, and in jet pipe electro-hydraulic servo valve, viscosity of working fluid should be used as small as possible.

4.2 CFD simulation for pressure characteristics of jet pipe amplifier under different working fluid

Using simplified 3D model and the mesh of Ansys CFX software, flow field is simulated under different working fluids such as aviation hydraulic oil 10, jet fuel 3 and hydraulic oil HM 46. Jet fuel 3 can satisfy the oil requirement for flight of high altitude or cold zone and supersonic as it has thermal stability or resistance of oxidation. This is highly clean and has no mechanical impurities or moisture and no harmful substances, especially low sulfur. Hydraulic oil HM 46 is a high performance anti wear hydraulic oil developed for using in high pressure hydraulic systems operating under moderate to severe conditions in mobile service. It has excellent oxidation resistance, good anti-foam or thermal stability. Furthermore, its foam can quickly disappear under conditions of mechanical vigorous stir.

Parameters of working fluids are shown in Table 3.

Table 3 Parameters of working fluids

Parameters	Aviation oil 10	Jet fuel 3	Oil HM 46
Density [kg/m^3]	850	810	870
Kinematic viscosity at 20^0 C [mm^2/s]	25	1	156

Boundary condition and structural parameters of valve for simulation as follows

- Diameter of jet pipe nozzle is 0.3[mm], diameters of receiver hole is 0.3[mm], the gap between nozzle end and receiver holes is 0.5[mm], working liquid is hydraulic oil HM 46, its density is 870[kg/m^3], and its dynamic viscosity at 20^0 C is 0.1357[kg/ms].
- The inlet pressure is p_s =10[MPa], the outlet pressure is 0[MPa], the atmospheric pressure is 1[atm]. Using activated turbulence model and the standard k– ε model.
- Symmetric cross-section is surveyed, and the inner of jet pipe is smooth wall of non-slip surface.

Flow field characteristics of jet pipe in the middle are shown in Fig.11*a*, and Fig.11*b*.

To master effect of working fluid on the recovery pressure, it is necessary to carry out simulation of flow field under different working fluid and different positions of jet pipe, whereby the pressure characteristic is obtained. Simulation results are shown in Fig.12*a* and Fig.12*b*.

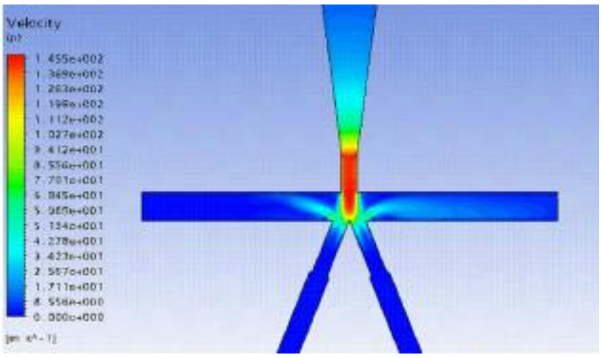


Fig.11*a* Velocity diagram of jet pipe in the middle

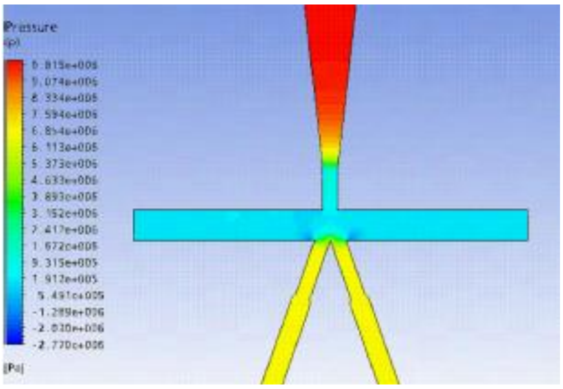


Fig.11*b* Pressure diagram of jet pipe in the middle

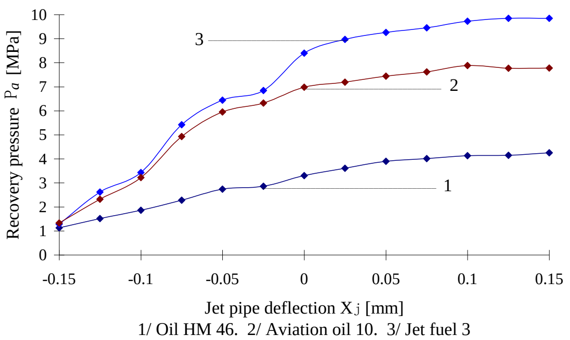


Fig.12*a*. Recovery pressure in the left receiving hole p_a^+

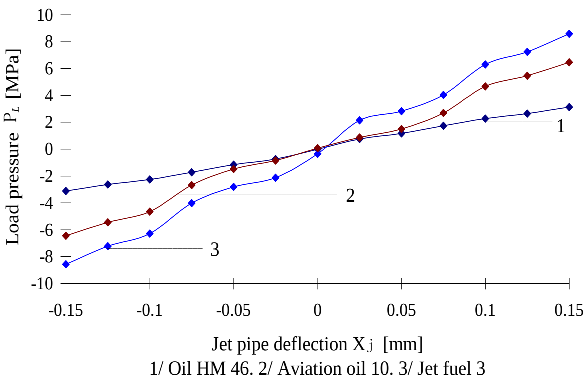


Fig.12*b*. Load pressure p_L

It is found out from Fig.12*a* that flow of oil HM46 at the jet pipe end is significantly weaker than flow of jet fuel 3, while the recovery pressure is smaller than the recovery pressure of jet fuel 3 at receiver holes. This happens because viscosity of oil HM 46 is heavy, and the effect of viscous resistance strengthens, fluidity weakens, therefore energy loss increases. Obviously, the recovery pressure in receiver holes increases under the raise of jet pipe deflection, but it is non-linear change. It is shown by Fig.12*b* that the load pressure is basically proportional to the jet pipe nozzle deflection. It can be seen that the greater viscosity is, the smaller load pressure is, but linearity of the load pressure curve is also better. The load pressure of jet fuel 3 is the biggest, on the contrary, the linearity of load pressure curve of oil HM 46 is the best but its magnitude is the smallest, however, variation range is limited.

V. CONCLUSION

Through mathematical models and simulation, it is shown that effect of wall roughness of the jet pipe servo valve parts on flow characteristics of pilot stage amplifier is ample.Through analysis of many different roughness, it is found out that the greater surface roughness is, the smaller velocity of jet flow is, and the recovery pressure decreases but the magnitude of change is not much. In addition, the relationship between the viscosity of fluid and the pressure characteristics of the pilot stage is close. In condition of the supply pressure is 10[MPa], diameters of receiver holes and jet pipe are 0.3[mm], wall roughness is 6.4[μm], aviation oil 10, jet fuel 3 and oil HM 46 are used, it can be seen that the greater viscosity of fluid is, the greater viscous stress in fluid is. A part of mechanical energy of fluid is lost because of the conversion of energy into heat, and then total of mechanical energy in fluid decreases. In process of change kinetic energy into pressure energy, the recovery pressure of the pilot stage amplifier will decrease in receiver holes.

REFERENCES

[1] Li R.P., Nie S.L., et al. Flow characteristics simulation of jet pipe servo valve working in different medium based on CFD. *Journal of Machine Tool and Hydraulics. Vol.39, No.3,* (2011).

[2] Mala G.M., Li D.Q. Flow characteristics of water in micro-tubes. *J. Heat Fluid Flow. Vol.20,* (1999), pp. 142-148.

[3] Qu W.L., Mala G.M. and Li D.Q. Pressure-driven water flows in trapezoidal silicon micro-channels. *Journal of Heat Mass Transfer,* 43, (2000), pp. 353-364.

[4] Nie S.L., Wu Z.J., et al. Experimental investigation on flow characteristics of annular gap damper in water hydraulics. *Proceeding of 6th ICFP*, Hangzhou, April, (2005), pp. 189-193.

[5] Yang Y.H. *Analysis and experimental research of pre-stage jet flow field in hydraulic servo valve*. Dissertation of Harbin Institute of Technology, (2006).

[6] Hu Y.D., Werner C., Li D.Q. Influence of three dimension roughness on pressure driven flow through micro-channels. *Journal of Fluid Engineering,* 125, (2003), pp. 871-879.

[7] Chen B.Y., Yao Z.H. Effect of surface roughness on gas flow in micro-channels. *Chinese Journal of Applied Mechanics. Vol.20, No.3,* (2003), pp: 1-5.

[8] Ji H., Wei L.J., et al. Investigation to the flow of jet pipe amplifier in a servo valve. *Journal of Machine Tool and Hydraulics. Vol.36, No.10*, (2008).

[9] Lecoq N.A., Cichocki R., et al. Drag force on a sphere moving towards a corrugated wall. *Journal of Fluid Mechanics*, 513, (2004), pp. 247-264.

[10] Yang Y.S., Zhang T.H., et al. Design of two-step conical throttle and its performance test. *Journal of Hydraulics and Pneumatics. Vol.11,* (2001), pp. 12-15.

[11] Michael J.M. and Stephen H.D. Slip over rough and coated -surface. *Journal of Fluid Mech*, 273, (1994), pp.801-817.

[12] Zhou Z.G. *Research on mechanism of affect from surface roughness to jet pipe performance*. Dissertation of Southwest University of Science and Technology, (2007).

[13] Jansons K.M. Determination of the macroscopic slip boundary condition for a viscous flow over a randomly rough surface with a perfect slip microscopic boundary condition. *Journal of Physical Fluid. Vol.31, No.1,* (1988), pp.15-17.

[14] Yao X.X., et al. Research on pressure characteristics of jet pipe servo valve. *Journal of Missiles and Guidance. Vol.4,* (2002), pp. 56-59.

[15] Tuck E.O., Kouzoubov A. Laminar roughness boundary condition. *Journal of Fluid Mech*., 300, (1995), pp. 59-70.

[16] Nie S.L., et al. Research on pressure distribution between two throttles of two-step throttle and its load rigidity characteristics. *Journal of Hydraulics and Pneumatics. Vol.8,* (2005), pp. 9-11.

[17] Yin Y.B., Meng W. Characteristics of electro-hydraulic servo valve with an asymmetric nozzle flapper. *Chinese Journal of Mechanical Engineering. Vol.22, No.8*, (2011), pp. 957-960.
[18] Yin Y.B., et al. Research on asymmetric high-speed pneumatic servo valve. *Journal of Fluid Power Transmission and Control. Vol.22, No.3*, (2007), pp. 4-8.
[19] Yin Y.B., et al. Analysis of torque motor stiffness of electro-hydraulic servo valve. *Journal of Machine Design and Research. Vol.25, No.5*, (2009), pp. 82-84.