ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

Numerical Analysis of Fluid Solid Coupling in the Opening Process of A Hydraulic Control Check Valve Based on Fluent

Linfeng Zhang[1], Dongsheng He[1], Qianlong He[1], Guangheng Zhou[1] and Tao Lv[1]

[1]Southwest Petroleum University, Chengdu 610500, China.

Abstract

Based on the dynamic grid theory and the newly developed UDF program, a three-dimensional fluid structure coupling analysis model is established for the transient opening process of a micro hydraulic control check valve in the hydraulic signal decoder of the intelligent completion. The coupling dynamic response of the valve core under the action of high pressure oil and fluid is solved by using the fluent 6DOF method, and the valve is simulated under the action of micro hydraulic cylinder and high pressure fluid During the opening process, the detailed visualization flow field cloud chart of the check valve was obtained, including the small scale flow characteristics around the valve core. At the same time, the real-time monitoring and analysis of the movement process of the valve core were carried out, and the movement characteristic curve of the displacement and speed of the valve core was obtained. The results showed that the valve completed the opening process in 1ms, but the transient hydraulic power fluctuated greatly during the opening process. It was pointed out that the old UDF program can not study the vibration of the valve core. The reason why the speed is 0 after the valve core reaches the maximum opening is that the hydraulic cylinder thrust is greater than the fluid force. The UDF program can not be used to force the valve core to reach the speed 0 after the limit displacement. At the same time, the feasibility and reliability of using micro hydraulic cylinder check valve system as the basic control unit of decoder are verified. The newly developed UDF is more suitable for studying the vibration of valve core.

Keywords

Completion decoder; check valve; dynamic mesh;CFD;Fluent UDF.

1. Introduction

In the intelligent well system, the downhole flow control valve (ICV) has become the preferred equipment for achieving selective mining, flow cut-off, layered production mixing, and production throttling. [1].The downhole flow control system is an important part of the intelligent regulation of the entire downhole production. At present, the mainstream solution for remote control ICV at home and abroad is hydraulic control. The completion system studied in this paper uses 3 pipelines to control 6 production layers. The application of different pressure sequence signals can achieve the purpose of production adjustment for downhole reservoirs. The hydraulic decoder first analyzes the pressure sequence signal that controls the production layer and is pressed by the ground, and then generates an action response that controls the opening degree of the ICV throttle. It does not require workover operations and is the core component of the entire hydraulic control system.

The decoder is mainly composed of a level unlocking module,a hydraulic oil return line self-locking module,a hydraulic line interlocking module, and a pressure value signal decoding module. The design must meet the characteristics of simple, stable and reliable. As shown in Figure 1, each module integrates multiple micro-cylinders and hydraulic control check valve (hereinafter referred to as check valve) components. A micro-cylinder and a micro check valve

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

form the basic hydraulic control unit. They are closely related to each other and are arranged inside the decoder to directly respond to the decoding action of the hydraulic signal. In order to ensure the anti-leakage ability, the spool movement direction is opposite to the fluid flow direction. Due to the constraints of the special working conditions in the well, there are special requirements for the performance of the check valve in the decoder: small pressure loss during forward conduction; fast response and high reliability; no vibration, noise and shock, of which pressure loss Small and fast response is one of the basic guarantees for implementing remote control commands on the ground, so it is necessary to conduct an in-depth analysis of the transient opening process of the check valve.

The volume of the check valve studied in this paper is small (the spool mass is only about 6g). During the working process, the vibration and noise caused by the instability of the flow field belong to the category of fluid-solid coupling. The experimental design of the mechanism and the installation of sensors It is more difficult to verify, and it is more appropriate to study such problems using computational fluid dynamics (CFD), fluid-structure interaction (FSI) dynamic analysis, and numerical calculation methods using moving grid technology. Domestic and foreign literatures use CFD and experimental methods to establish the analysis model of conventional check valves, and study their stability and non-linear motion characteristics, but less research on hydraulic check valves. In addition, the research in the literature The valve body movement direction of the check valve is consistent with the fluid flow direction, while the fluid flow direction of the hydraulic control check valve studied in this article is opposite to the valve core movement direction.

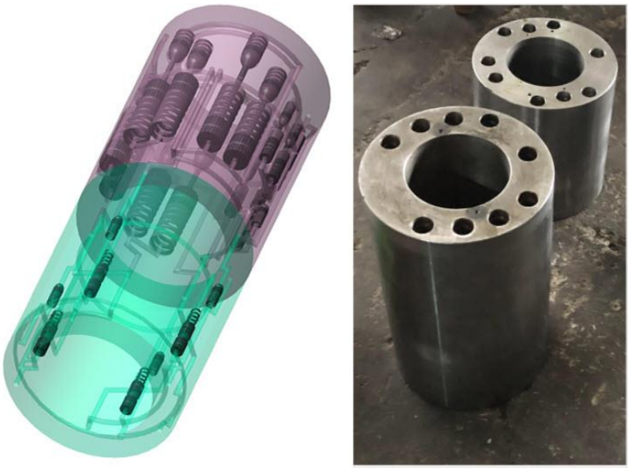


Fig 1. Decoder 3d perspective

Literature [2] used Fluent to establish a CFD model of gas-liquid one-way ball valve, and analyzed the influence of valve core material, spring stiffness, gas content and inlet flow rate on the transient opening stability of one-way ball valve. In [3], the dynamic grid method was used to analyze the effect of spring stiffness on the transient hydrodynamic force of the hydraulic poppet valve. References [4, 5] established a fluid-solid coupling numerical model of a one-way valve in a volumetric pump based on Fluent moving grid technology. The user-defined function (UDF) was used to control the displacement of the valve core. The transient response is obtained, and a loss coefficient similar to the test is obtained. The effects of simulation parameters such as time step, turbulence model, and solution model on the numerical results are compared. The literature [6] used the CFD method to study the speed, pressure distribution, and pressure loss of the valve body during the opening and closing of the shut-off valve. The vortex, water hammer, and backwater area generated when the fluid passed the valve passage

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

were studied. Point out that the opening and closing of the valve will cause large pressure fluctuations. Reference [7] studied the opening process of a check valve for aircraft fuel system based on CFdesign software and experiments. It was pointed out that the maximum opening position of the spool is closely related to the spring stiffness, the shape of the spool and the internal flow path. Reference [8] used the 6DOF module of the Fluent software to establish an unsteady flow field model during the opening of the cartridge valve. The valve core displacement, control cavity pressure, and steady-state hydrodynamic force calculated by numerical results are consistent with the theoretical formula. Compared with the conventional method, the result of the CFD method is closer to the actual movement of the spool. Literature [9] studied the vibration and noise phenomena of poppet valves through design experiments, and pointed out that the transient hydrodynamic force caused the regular periodic vibration phenomenon of poppet valves. References [10, 11] based on a three-dimensional fluid-structure coupled finite element dynamics simulation analysis and a direct coupling algorithm, established a numerical analysis model for the entire process of the conical throttle valve from closed to open and then closed. The factors that significantly affect the valve core vibration frequency are the quality of the valve core and the amount of oil volume. The contact and collision between the valve core and the valve seat will increase its vibration frequency.

The above-mentioned literature based on the Fluent moving grid to achieve fluid-structure coupling analysis, the UDF (hereinafter referred to as UDF1) used to achieve the analysis of valve core vibration, the core control command of UDF1 is: when the valve core reaches the preset maximum opening degree After that, the spool speed will no longer change with time after the instant of zero, and the spool vibration is not considered, which is inconsistent with the actual working conditions.

2. Structural Model

2.1. Working Principle

As shown in Figure 1, the working principle of the check valve is: In the initial state, the normally closed check valve is affected by the pressure difference between the inlet and outlet. The small ball in the check valve blocks the internal flow path, and the hydraulic oil cannot communicate. When the hydraulic oil connected to the micro-cylinder first pushes the internal piston downward, the piston push rod moves to contact the small ball inside the check valve. Because the thrust Fp of the piston on the small ball is greater than the fluid's resistance Fv With the sum of the spring elastic force Fs, the small ball gradually opens under the thrust of the piston at this time, and the movement direction of the valve core is opposite to the fluid flow direction. The movement of the valve core is affected by the piston thrust and the impact force of the hydraulic oil. The essence is to use a pair of displacement changes of the piston to control a check valve with a large pressure difference. Therefore, it has a simpler driving device and less driving energy consumption. With the increase of the valve opening, the throat flow velocity slows down, the internal pressure difference of the check valve decreases, the impact force of the hydraulic oil on the valve core ball is reduced, and the force on the valve core ball is more complicated. Therefore, the CFD simulation method is used. Able to study such questions intuitively.

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

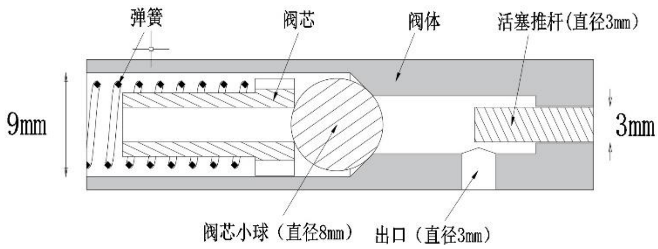
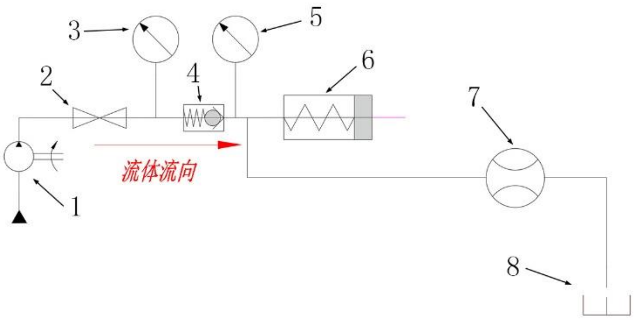


Fig 1. Schematic diagram of check valve structure

Because the direction of the fluid in the check valve is opposite to that of a conventional check valve, as shown in Figure 2, a simple test was designed to explore its ability to prevent leakage when closed. The results show that when the valve is closed, a When pressurizing, the oil pressure only acts on the left end of the valve core. The larger the pressure difference, the greater the force between the valve core and the valve body, and the better the anti-leak effect of the valve.



1- Hydraulic pump; 2- Manual valve; 3, 5- Pressure gauge; 4- Check valve; 6- Miniature cylinder; 7- Flowmeter; 8- tank

Fig 2. Pressure test schematic diagram of check valve

2.2. CFD Numerical Model

To facilitate the study of the problem, this article makes the following assumptions: the working fluid is an incompressible ideal fluid; the gravitational field is not considered and the vertical pressure difference is ignored; the sealing performance between the valve body, the valve core and the valve seat is good, and there is no leakage Phenomenon, the oil is incompressible, the valve parts are inelastic, and the fluid can completely fill the cylinder during the movement of the valve core.

The opening process of the check valve is actually the combined effect of the three forces of the piston thrust Fp, fluid resistance Fv and spring force Fs on the spool. If you want to simulate the opening process of a check valve, you must accurately analyze the force of the spool Therefore, this paper developed a numerical analysis model to simulate the force movement of the valve core, and established a CFD calculation model of the valve from fully closed to fully open. The flow type inside the valve must be determined before the simulation. Because the spool displacement in the study is only 2mm, and the internal turbulent flow field is complicated, the Fluent moving grid technology and UDF function are used to realize the numerical calculation of the transient process.

The numerical calculation uses the RNG k- turbulence model in Fluent software. The SIMPLE algorithm based on pressure-velocity coupling is used. This algorithm uses pressure correction under a staggered grid. The RNG k- model uses a finite volume method and a second-order upwind. It can solve the complex pressure distribution problem in the flow field of the valve

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

during the transient opening process, and its dual equation model takes into account the effects of turbulent vortices and low Reynolds number effects. The Reynolds stress tensor in its momentum equation uses the eddy viscosity model[4, 5, 12]:

$$\tau_{ij} = -\rho\overline{u_i^{'}u_j^{'}} = \mu\underbrace{(\frac{\partial\overline{u_i}}{\partial x_j} + \frac{\partial\overline{u_j}}{\partial x_i})}_{S_{ij}} - \frac{2}{3}\rho k\delta_{ij} \tag{1}$$

Where, $k = 0.5\rho\overline{u_k^{'}u_k^{'}}$ is turbulent energy, $\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$ is turbulent viscosity, C_μ=0.0848 , In addition, Turbulent kinetic energy and turbulent dissipation rates require additional transport equations, $\varepsilon=2v\overline{s_{ij}^{'}s_{ij}^{'}}$, where $s_{ij}^{'} = \frac{1}{2}(\frac{\partial u_i^{'}}{\partial x_j} + \frac{\partial u_j^{'}}{\partial x_i})$.

Therefore, using the RNG k-ε turbulence model with a standard wall function, this equation introduces a correction equation for the turbulent viscosity coefficient, which is closer to simulating problems such as flow separation, strong swirling and curved wall flow. Actual flow state. Turbulent kinetic energy k and its dissipation rate come from the following transmission equation[12, 13]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k\mu_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \qquad \frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + \rho C_1 S_\varepsilon - \rho C_2\frac{\varepsilon^2}{k+\sqrt{v\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b + S_\varepsilon \tag{2}$$

Where, $C_1 = \max\left[0.43, \frac{\eta}{\eta+5}\right]$, $\eta=S\frac{k}{\varepsilon}$, $S=\sqrt{2S_{ij}S_{ij}}$,In the equation, G_k represents the turbulent kinetic energy due to the average velocity gradient, which is the turbulent kinetic energy due to buoyancy, and G_b represents the effect of the compressive velocity turbulent pulsation expansion on the total dissipation rate. C and D are constants, and E and F are respectively Is the turbulent Prandtl number of G and H, the turbulent (or vortex) viscosity A can be calculated from A and A

G_k Represents the turbulent kinetic energy due to the average velocity gradient, G_b represents the Turbulent energy due to buoyancy, Y_m represents the effect of compressible turbulent pulsating expansion on the total dissipation rate, C_1 and $C_{1\varepsilon}$ is constant, S_k and S_ε are respectively the turbulent prandtl number of k and ε :

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{3}$$

Where C_μ is constant.

Model constant values within the equation: $C_{1\varepsilon} = 1.44$, $C_2 = 1.9$, $C_{3\varepsilon} = 1.0$, $C_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.2$.

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

3. CFD Analysis of Check Valve

3.1. Grid Parameters

This paper uses Fluent Meshing for meshing. In order to reduce the cost of calculation time and the need for dynamic mesh parameters, as shown in Figure 3, the computing domain is divided into three and connected by Interface, and the grid is updated only within the "valve" non-structural domain. The two domains of “top” and “down” use poly-Hexcore meshes. The whole model has a total of 535433 nodes and 1258055 elements. The mesh quality is above 0.38, which meets Fluent Requirements for solving calculations.

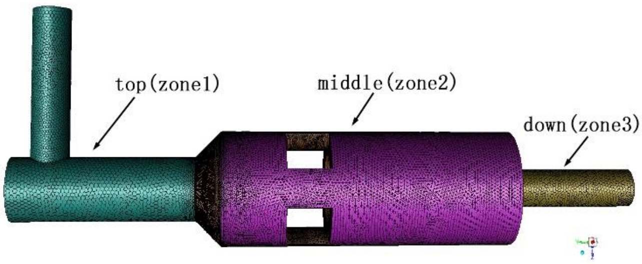


Fig 3. Grid model of check valve

3.2. Controlling Spool Movement with Fluent UDF

The movement of the spool is always determined by the resultant force. The simplified diagram of the force is shown in Figure 4 The second-order ordinary differential equation derived from Newton's second law can be used to simulate the movement of the spool. [14, 15] :

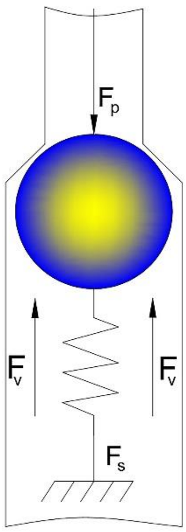


Fig 4. Ool force diagram

$$m\ddot{y} = F_{flow} + F_s - F_0 \tag{4}$$

Where the constant m is the total mass of all moving parts including pistons, spools, springs, etc., $\ddot{y}$ Is the acceleration in the moving direction (y direction) of the moving part, $F_s{}^y$ Is the spring force acting in the y direction, $F_{flow}{}^y$ Is the hydrodynamic force acting on the spool

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

element in the y direction. Obviously, equation (4) cannot be solved directly, and it needs to be discretized. The continuous differential equation is converted into the form of discrete difference to be suitable for numerical calculations. The expression on the left side of the equilibrium equation (4) is simplified to obtain moving parts (Spool) acceleration expression:

$$\ddot{y} = \frac{\dot{y}_{t+\Delta t} - y_t}{\Delta t} \tag{5}$$

Speed $\dot{y}$ can be further dispersed as:

$$\dot{y}_{t+\Delta t} = \frac{y_{t+\Delta t} - y_t}{\Delta t} \tag{6}$$

The displacement y_t introduced into the spool is an independent variable, The expression of the spring force is:

$$F_s^y = K \times y_{t+\Delta t} + F_0 \tag{7}$$

Where k is the stiffness coefficient of the spring, F_0 is preload of spring, Used to set the initial set pressure of the spring so that a new discrete equation:

$$y_{t+\Delta t} = \frac{F_v^y + F_0 - F_p + \left(\frac{m\dot{y}_t}{\Delta t} + \frac{m\dot{y}}{\Delta t^2} \right)_t}{K + \frac{m}{\Delta t^2}} \tag{8}$$

Because ANSYS Fluent cannot directly read the execution equation (8), the equation is processed programmatically using the UDF compilation module. The model parameters used in this article are shown in Table 1 below:

Tab 1. Main parameters of UDF program

Parameter	Value
body quality m	0.00608 kg
spring stiffness coefficient k	2600 N/m
maximum thrust of piston Fp	769 N
preload F0	5.2 N
incremental step Δt	0.00001 s

3.3. Dynamics Mesh Update Technology

This paper uses the spring approximate smoothing method and the local mesh remesh method, and combines the two methods to generate a dynamic mesh. When using a compiled UDF program to control the spool, since the calculation domain changes with the transient position of the spool, in order to avoid a negative volume mesh, the mesh needs to be updated before the next iterative calculation. The principle is:

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

(1) The edge between each two mesh nodes can be idealized as a spring linking the two nodes. Therefore, as the nodes move, a force proportional to the displacement will be generated. The force on the nodes can be derived from Hooke's law. composition [12, 15]:

$$F = \sum_{j}^{n_i} k_{ij}\left(\Delta y_j - \Delta y_i\right) \tag{9}$$

Where $k_{ij} = 1/\sqrt{|y_i - y_j|}$.

(2) Based on the principle of force balance where the resultant force at the nodes is zero, the iterative expression is:

$$\Delta y^{(t+1)} = \sum_{j}^{N_i} k_{ij}\Delta y_j^{(t)} / \sum_{j}^{N_i} k_{ij} \tag{10}$$

(3) The position of the internal nodes can be obtained by Jacobian iteration with the following expression:

$$y_i^{(t+1)} = y_i^{(t)} + \Delta y_i^{(t+1)} \tag{11}$$

(4) In Fluent Mesh technology, when the deformation rate of the mesh or the longest (0.15 mm) and shortest dimension (0.75 mm) of the mesh exceeds the set range, the mesh will be re-divided.

3.4. Boundary Conditions and Solver Settings

When Fluent performs numerical calculations, the pressure, medium parameters, calculation solver and fluid calculation domain must be defined. The hydraulic oil is regarded as incompressible and isothermal Newtonian fluid, and the fluid in the valve is 32 # hydraulic oil, density ρ=850Kg/m3, density μ=0.00118P·S, Inlet and outlet boundaries are set as pressure inlet and pressure outlet respectively, inlet pressure is 5 MPa, Flow rate 0.5m/s, Outlet pressure is 4.5 MPa, Because it is necessary to view the pressure, velocity distribution and various water flow phenomena over time during the valve opening and closing process when the valve is operating, a pressure-based unsteady solver is selected. A second-order format is used for the discrete pressure equation and The momentum equation, because the valve is in a highly turbulent state, the movement is more violent. In order to increase the convergence of momentum, the momentum under-relaxation factor is set to 0.4[16], The absolute standard of convergence for other numerical simulations is set as the default standard.

4. Numerical Analysis Of Non-Steady-State Dynamic Characteristics of Check Valves

4.1. Effect of Fluid-Structure Interaction on the Movement Characteristics of Spool

The check valve used in this article has a special structure and a small size (the spool radius is only 4mm). Therefore, the UDF program is used to extract and record the cooperative force of the fluid on the spool. The fluid force directly affects the movement of the spool and causes the valve to work One of the important factors of instability.

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

Figure 5 shows the change of the transient hydrodynamic force D over time of the spool F_f. Before the spool moves, the fluid force is maintained at about 30.6N, regardless of the magnitude of the thrust F_p, Moment of spool movement, F_f f has a tendency to increase slightly, and remains basically unchanged during the opening process. When the valve core is displaced to the maximum, it is affected by the thrust of the micro-cylinder, and the speed of the valve core is 0. At this time, the flow field around the valve core is extremely unstable. Transient hydrodynamics Transient hydrodynamic surge[17], Finally stabilized at about 30.6N and no longer fluctuates.

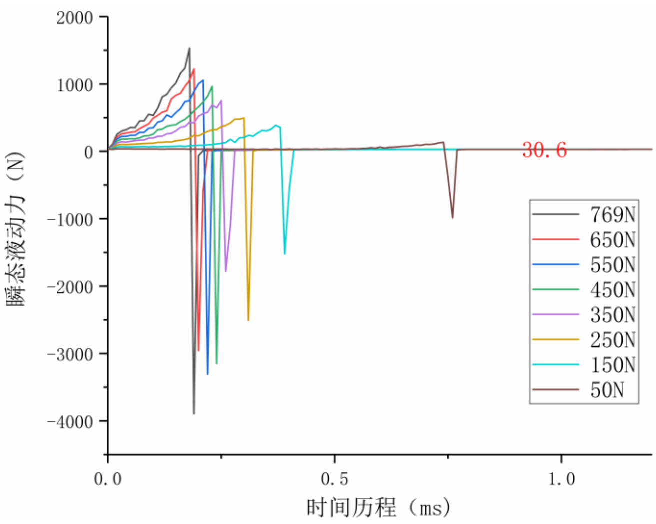


Fig 5. Transient hydrodynamic force curv

As shown in Figure 6, the maximum speed obtained by the valve core is directly proportional to the speed curve divided into two parts. After the valve is opened, the speed increases linearly under the combined action of the fluid force F_f, the spring force F_s and the thrust F_p, and is also affected by the micro-fluid. The effect of cylinder thrust is that after the valve core reaches the maximum displacement, the resultant force is in the negative direction of the y-axis. At this time, the valve core speed is 0, which again illustrates that using UDF1 to limit the valve core speed to 0 is inappropriate.

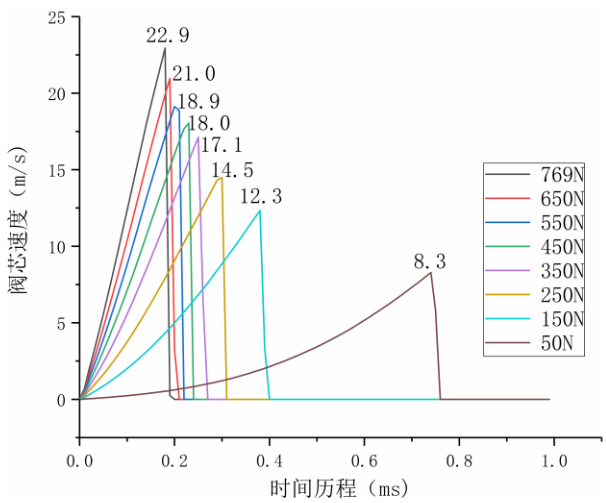


Fig 6. Spool velocity curve

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

Figure 7 shows the change law of the valve core displacement with time. After the valve is opened, under the combined force of the valve core, the displacement gradually decreases, and the larger the thrust F_p, the faster the displacement changes, and the valve core reaches the maximum in 1ms The opening degree and the quick response of the valve core after receiving the hydraulic thrust signal ensure the sensitivity of the decoder and achieve the goal of quickly controlling the opening degree of the ICV even when using hydraulic pressure.

The dynamic change of the valve core can be clearly observed by using the method of numerical dynamic simulation. The change in the y-coordinate of the core of the valve core is monitored by Fluent. The curve is parabolic. At the moment the valve is opened, the valve core is subject to the combined effects of the hydraulic pressure difference, the impact of the fluid on the valve core, and the spring pretension force. The larger the F_p, the faster the valve core displacement and the The maximum displacement is reached first. At the initial stage of opening, the valve core has a large opening resistance and the valve core displacement changes slowly. As the piston thrust F_p continues to push the valve core, the valve core accelerates to the maximum displacement (2mm) and stops.

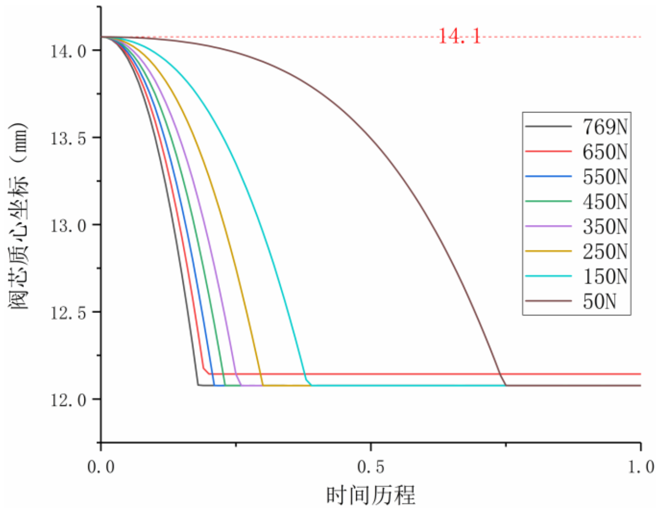
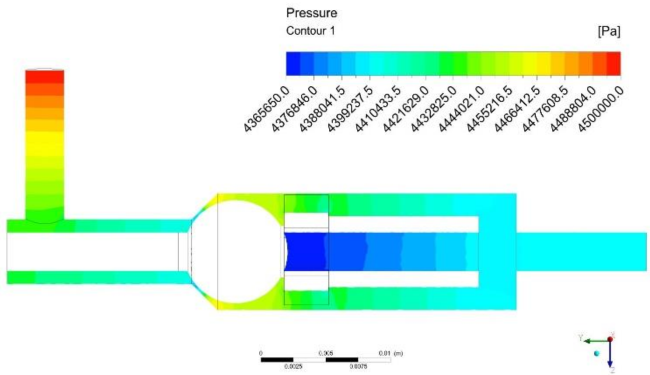


Fig 7. Curve of spool displacement

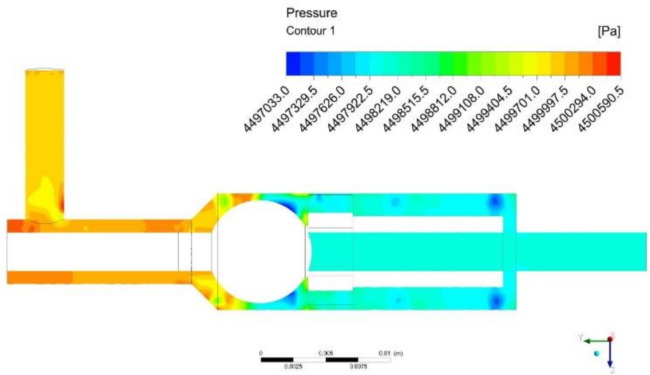
4.2. Pressure Field

As shown in Figure 8, it is a partial static pressure cloud diagram of each characteristic moment of the one-way valve transient opening process under the action of different piston thrusts. At the beginning of opening, the pressure in the flow field at the front and rear of the valve core changes significantly. The core is subjected to a large pressure, and the top of the valve core (spherical part) is subjected to a certain back pressure throughout the process, so the upper half of the valve core small ball has a higher pressure. When the valve has reached its maximum opening and the hydraulic oil begins to flow steadily, the maximum static pressure for the flow field around the spool is in the outlet area.

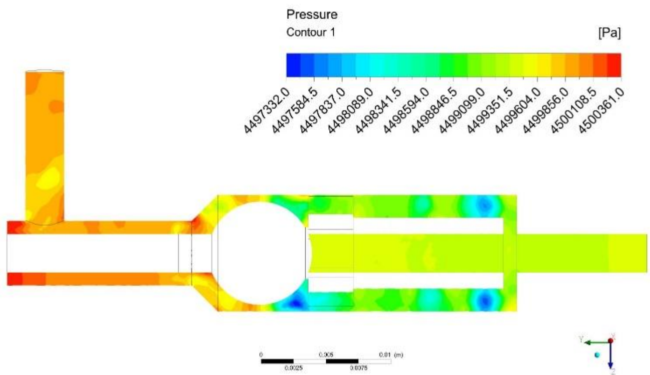
ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006



(a) Initial pressure contours at thrust 50N



(b) Pressure contours after stabilization at thrust 50N



(c)Pressure contours after stabilization at thrust 50N

Fig 8. Partial pressure contours of check valve

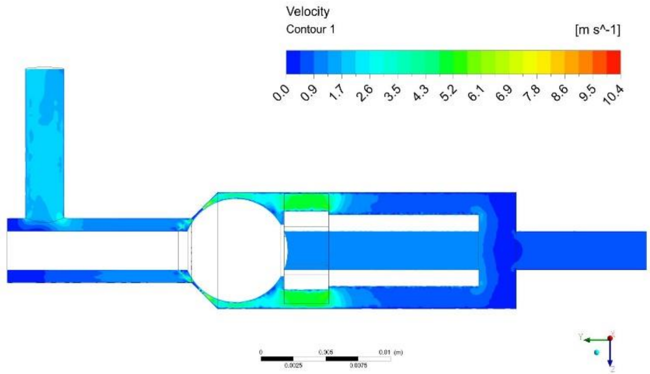
4.3. Velocity Field

Figure 9 shows the cloud distribution of the speed at each characteristic moment during the transient opening process. At the same time, the top of the valve core (spherical part) and the throat area of the valve cavity have the highest velocity, while the velocity of the top and bottom of the valve is relatively small; The maximum flow rate during opening is shown in Figure 19 (a). At this time, the maximum throat oil flow rate is as high as 10.4 m / s. When the valve core is fully opened and the internal flow field is stable, the flow rate is reduced to 2.7 m / s. Left and right, and the flow rate after the spool is stable has nothing to do with the thrust of the miniature cylinder.

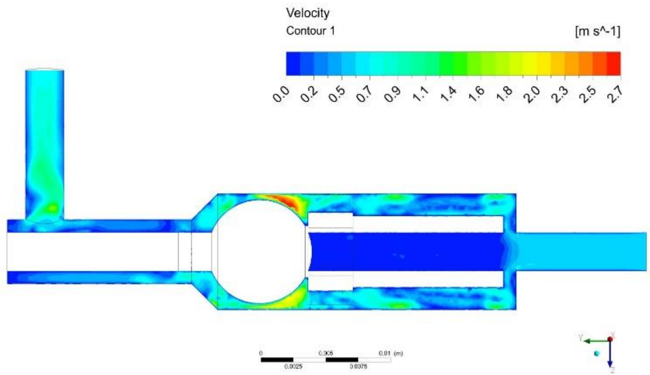
The smaller the annulus cross-sectional area formed by the valve core and the flow path, the faster the flow rate and the continuity equation are satisfied. As shown in Figure 10, the flow chart of the check valve flow line and the surface pressure of the valve core shows The force is

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

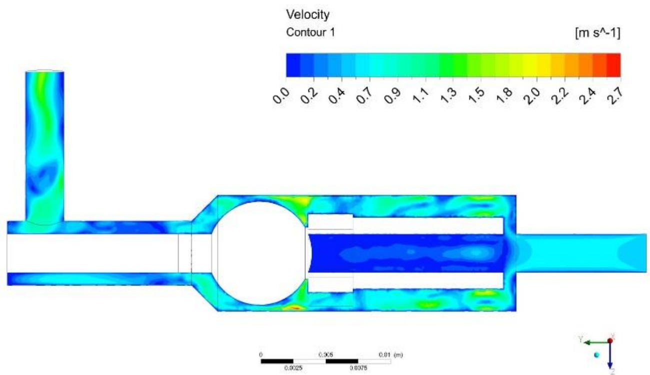
uneven, and the back pressure is concentrated on the top of the ball. The fluid medium is blocked by the valve core, and the flow direction changes. It is mainly divided into four flow directions. Therefore, a vortex is formed in the area behind the inlet and the bottom of the valve core. The component is composed of the lower part with a semicircular flow channel. The junction of the upper and lower parts causes a sudden change in the cross-sectional area of the annulus and changes the fluid flow direction. Therefore, a pair of vortex regions with opposite flow directions are formed here. Multiple energy losses.



(a) Initial velocity contours at thrust 50N



(b) Velocity contours after stabilization at thrust 50N



(c) Pressure contours after stabilization at thrust 50N

Fig 9. Partial velocity contours of check valve

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

5. Conclusion

Based on the professional CFD finite element software Fluent, a three-dimensional numerical model of a fine check valve used in a well completion hydraulic decoder is established in this paper. The 6DOF UDF program is compiled to control the movement characteristics of the valve core during the opening process. The dynamic grid update technology realizes the dynamic response of the high-speed flow from the closed state to the fully opened one-way valve, and obtains a good prediction result.

Based on the professional CFD finite element software Fluent, a three-dimensional numerical model of a fine check valve used in a well completion hydraulic decoder is established in this paper. The 6DOF UDF program is compiled to control the movement characteristics of the valve core during the opening process. The dynamic grid update technology realizes the dynamic response of the high-speed flow from the closed state to the fully opened one-way valve, and obtains a good prediction result.

(3) Using a new research method, the piston dynamic thrust F_p is reduced to a fixed force to analyze the dynamic response of the one-way valve during the opening process. The size of F_p directly affects the maximum speed value that can be obtained by the valve core. The influence of the final state flow field is small.

(4) When the check valve is opened, the bottom and top areas of the valve have the most violent fluid movement. The maximum flow velocity of the fluid in the valve is 10.4m / s, and the movement speed of the valve core reaches 22.9m / s. The size is small, to avoid the check valve inlet pressure is too high or the situation of sharp pressure fluctuations, the correct use can play the best performance and extend the service life.

References

[1] Hodges S, Olin G, Sides W. Hydraulically-Actuated Intelligent Completions: Development and Applications[M]. Houston, Texas: Offshore Technology Conference, 2000,3(01603663): 199-214.

[2] Shengchang Zhang, Lele Zhu, Yi Ma, et al. Numerical simulation on dynamic characteristics of Fluent ball valve under gas-liquid two phase flow[J]. Journal of Zhejiang University of Technology. 2016, 44(01): 1-4.

[3] Yi Jian, Qiang Wang, Xiao Hui, et al. Research on transient properties of hydraulic poppet valvewith dynamic mesh[J]. Chinese Hydraulics & Pneumatics. 2015, 2(7): 103-110.

[4] Alberto M B, Jesús Manuel F O, Andrés M. Numerical methodology for the CFD simulation of diaphragm volumetric pumps[J]. International Journal of Mechanical Sciences. 2019,150(0020-7403): 322-336.

[5] Menéndez-Blanco A, Fernández Oro J M, Meana-Fernández A. Unsteady three-dimensional modeling of the Fluid–Structure Interaction in the check valves of diaphragm volumetric pumps[J]. Journal of Fluids and Structures. 2019, 90(0889-9746): 432-449.

[6] Peng Ba,Zhangxing Zou,Weidan Chen. Dynamic numerical simulation of flow characteristics when globe valves are opened and closed[J]. Journal of Vibration and Shock. 2010, 29(10): 157-161.

[7] Zhiyong Ma,Qilin Jiang,Jianfeng Zhao,et al.Simulation analysis on the influence of the characteristics of a one-way valve on the flow resistance during the transition process[J]. Equipment Manufacturing Technology. 2014(2): 47-50.

[8] Xiaoxia Guo,Jiahai Huang,Quan Long,et al.Flow field analysis of hydraulic valve based on dynamic grid 6DOF method[J]. Journal of Taiyuan University of Technology. 2016.1(1007-9432): 11-15.

ISSN: 2637-6067 DOI: 10.6918/IJOSSER.202004_3(4).0006

[9] Guangfei Li,Huanlong Liu,Bin Deng,et al.Effect of flow force to the poppet valve vibration[J] Machine Tool & Hydraulics. 2014, 42(05): 28-30.

[10] Ming Li,Zhenhua Lu. FSI dynamic response analyses of a conical orifice valve during working process with several major influences[J]. Engineering Mechanics. 2017, 34(09): 239-247.

[11] Zhenhua Lu,Ming Li. Simulations of the unsteady fluid-structure coupling characteristices of a conical orifice valve[J]. Journal of Tsinghua University (Science and Technology). 2018, 58(01): 35-42.

[12] Qian J, Wei L, Jin Z, et al. CFD analysis on the dynamic flow characteristics of the pilot-control globe valve[J]. Energy Conversion and Management. 2014, 87(0196-8904): 220-226.

[13] Lisowski E, Czyżycki W, Rajda J. Three dimensional CFD analysis and experimental test of flow force acting on the spool of solenoid operated directional control valve[J]. Energy Conversion and Management. 2013, 70(0196-8904): 220-229.

[14] Hős C J, Champneys A R, Paul K, et al. Dynamic behaviour of direct spring loaded pressure relief valves: III valves in liquid service[J]. Journal of Loss Prevention in the Process Industries. 2016, 43(0950-4230): 1-9.

[15] Sun X, Qin B, Bo H, et al. Transient flow analysis of integrated valve opening process[J]. Nuclear Engineering and Design. 2017, 313(0029-5493): 296-305.

[16] Qinwei Yan. Dissertation Submitted to Zhejiang University of Technology for the Degree of Master [D]. Zhejiang University of Technology

[17] Mingxing Han,Yinshui Liu,Defa Wu,et al. Flow force reduction in large flow cartridge valve by optimizing geometric parameters of orifice[J]. Journal of Huazhong University of Scienceand Technology.2018,46(03):65-68.