Research on the Influence of Turbulence Model on the Flow Field Characteristics of Rotating Valve

Yuning Song*, Liang Han

Department of Mechanical and Power Engineering, Yingkou Institute of Technology, Yingkou, China

*Corresponding author e-mail: songyuning1230@163.com

Abstract. In order to study the fluid flow characteristics of the key components in hydraulic excited rotary valve, and then determine the area where the vortex and negative pressure to find the weak link in the structure and analyze the performance of the valve. Simulate the flow field in the rotary valve by Fluent fluid analysis software, and setting different opening degrees under the same boundary conditions. Studies showed that the velocity at the exit junction of the fluid in the rotary valve, and the vortex degree of pressure is large at the oil groove. When the opening degree is small, the negative pressure is large, the vortex is obvious. With the opening becomes large, and the negative pressure and the vortex are improved. The research provides a theoretical basis for the optimization and improvement of the rotary valve. Studies have shown: The velocity of the flow channel calculated under the turbulence RNG $k-\varepsilon$ model is the largest, Realizable. The velocity of the flow channel calculated under the model is the smallest. For different models, the values of the low speed region are different, where Realizable $k-\varepsilon$ Minimum model speed, The RNG $k-\varepsilon$ model has the highest speed and standard. The speed of the $k-\varepsilon$ model is in the middle; The high speed area is mainly concentrated in the corner area of the oil tank and valve mouth, The area of high speed area of the RNG $k-\varepsilon$ model is smaller than that of the other two models, and the standard of the model is smaller than that of the other two models. The $k-\varepsilon$ Model and the Realizable $k-\varepsilon$ is no significant difference in the area of high speed area calculated by the model. The comparison draws the following conclusions: The results of the turbulence RNG $k-\varepsilon$ model and the other two groups of models are more accurate when calculating the low velocity flow field. The Realizable $k-\varepsilon$ Model and the RNG $k-\varepsilon$ has the calculation results of the model are similar.

1. Introduction

The rotary valve is the key part of the vibrator to realize vibration. When the rotary valve rotates, the flow area is always changing, from small to large to large. When the flow area is small, when the high pressure oil comes over, it will form a high pressure effect because the valve mouth area of the rotary valve is very small, which will lead to a higher flow rate of the oil at this time, resulting in a sharp decrease in the pressure here, prone to cavitation, noise and other problems, so it is very necessary to analyze the flow field of the rotary valve [1-3].
Using Flunet simulation software to compare three kinds of rotary valve flow field under the turbulence model, study the internal flow field characteristics, explore the rotary valve internal structure change and speed of internal flow field of the pressure change law of such improvement and optimization of valve to provide certain reference and provide theoretical basis for the further study of rotary valve.

2. Rotary valve flow model

Counter rotating valve flow field analysis, the first step to establish the particular moment in the process of valve rotating fluid model inside the rotary valve. As quickly and accurately to the creation of the flow field model of the rotary valve, using CATIA 3 d modeling for rotary valve, to reflect the actual properties, and save computer computing resources in the shortest possible time calculated in accordance with the actual results, to counter rotating valve in unnecessary simplified geometric feature [4-6]. And then set up less than rotary valve diameter geometry, through the Boolean subtraction Boolean operations tools tools, solid model geometry created minus the rotary valve, in order to obtain the flow model of the rotary valve is shown in figure 1.

Rotary valve, inlet and the oil tank in the process of rotation, to repeat the tank working condition is the same, you just need to analyze the oil inlet and the oil tank [7], don't have to analyze all the groove on the rotor, so the overall flow model can be cut, choose a counter rotating valve shaft oil a mouth with the corresponding flow model of oil exports to cooperate.

![Figure 1. Flow passage model of rotary valve](image1)

![Figure 2. Liquid flow direction Diagram](image2)

![Figure 3. Runner motion Diagram](image3)

According to the structure and working principle of the rotary valve, the overall flow model is composed of the tao and sump oil import oil, in order to facilitate analysis, we approximate as a way to sump oil as stationary, inlet oil way along a certain direction, to form a new whole oil [8-10]. Figure 2 for the direction of liquid flow diagram, figure 3 sketches for flow movement.

According to the changing rule of the oil duct, select the oil duct half tong state and all-pass based oil inlet cross section for the circular flow model of two work time as shown in figure 4. And store model to step through the geometry format, the model imported into preliminarily set up the model in Ansys.
3. Meshing flow model
Port model after after processing to create good input and output side also need to model for grid partition, three-dimensional import to flow into the mesh, after tetrahedral mesh structure, after dividing the file to. MSH format, so that the simulation can be directly into Fluent. Meshing after completion of this step, import it on to the next step in the fluent flow field simulation analysis [11]. Model after import, the first step is to set up the solver, sets the solver to don't calculate coupling analysis of the steady state. Again to liquid density, liquid viscosity properties such as setting. According to actual working condition of oil in the oil outlet end and the end of the essential boundary conditions for the whole entity region for fluid region [12].

4. Match flow valve flow field simulation results analysis
Liquid in the process of flow, when external conditions change, the liquid flow rate will also change, so the internal flow field will therefore produce turbulence, turbulence will cause the energy exchange between the liquid, and thus affects the momentum, energy and the concentration of liquid. In general there are many types of fluid turbulent flow calculation method, but when solving Reynolds averaged navier-stokes equation and turbulence model is best [13]. So this section respectively, using the standard $k-\varepsilon$ model, the RNG $k-\varepsilon$ model and Realizable $k-\varepsilon$ model of rotating valve port model for the simulation and study the influence of different flow channel model flow field analysis results.

4.1. Turbulence model contrast
The standard $k-\varepsilon$ model with its stability, economic and the advantages of high calculation accuracy in the most widely used in many types of flow field model. The standard $k-\varepsilon$ model, which make use of turbulent kinetic energy equation and the dissipation rate equation [14].

Figure 4. Catia valve port state model

Figure 5. Grid effect diagram of runner model of rotary valve
\[
\frac{\partial(p\kappa)}{\partial t} + \frac{\partial(\rho ku_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu + \mu_t}{\sigma_k} \right) \frac{\partial \kappa}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_\kappa
\]  
(1)

\[
\frac{\partial(p\kappa)}{\partial t} + \frac{\partial(\rho ku_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu + \mu_t}{\sigma_k} \right) \frac{\partial \kappa}{\partial x_j} \right] - C_{1\varepsilon} \frac{\varepsilon}{\kappa} (G_k + G_3 \lambda b) - C_{2\varepsilon} \rho \varepsilon^2 \kappa + S_\varepsilon
\]  
(2)

Type: \( G_\kappa \) : Velocity gradient is the turbulent kinetic energy; 
\( S_\varepsilon \) : is to define the turbulent dissipation; 
\( u_r \) : is the viscous coefficient of turbulence model.

RNG \( k-\varepsilon \) model is similar with the standard \( k-\varepsilon \) model, but the RNG \( k-\varepsilon \) model is added in the turbulent dissipation rate equation of the auxiliary calculation [15], makes the larger changes in velocity of the flow field calculation more accurate, and the model when calculating the strong spin fluid calculation accuracy is higher, when calculating the effects of low Reynolds RNG \( k-\varepsilon \) model also has certain advantages.

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho ku_i) = \frac{\partial}{\partial x_j} \left[ \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k
\]  
(3)

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho ku_i) = \frac{\partial}{\partial x_j} \left[ \frac{\alpha_k \mu_{eff}}{\kappa} \frac{\partial k}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{\kappa} (G_k + G_3 \lambda b) - C_{2\varepsilon} \rho \varepsilon^2 \kappa - R_\varepsilon + S_\varepsilon
\]  
(4)

Realizable \( k-\varepsilon \) model compared with the above two models, the calculation of jet diffusion velocity is more accurate [16].

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho ku_i) = \frac{\partial}{\partial x_j} \left[ \frac{\alpha_k \mu_{eff}}{\kappa} \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k
\]  
(5)

\[
\frac{\partial}{\partial x_j} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu + \mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S_\varepsilon + \rho C_2 \frac{\varepsilon^2}{\kappa + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{\kappa}
\]  
(6)

4.2. Parameter Settings

To study the influence of different turbulent flow field model, need to choose the standard \( k-\varepsilon \) model respectively, the RNG \( k-\varepsilon \) turbulent model selection interface simulation model and the Realizable \( k-\varepsilon \) model, as shown in figure 6. This group of simulation in addition to the turbulence model to choose the other parameters are consistent, set the inlet pressure to 11 MPa, outlet 5MPa pressure, fluid density.

![Turbulent model selection interface.](image)
4.3. Turbulence model convergence speed

Respectively to simulate three kinds of flow field, the different turbulence model there are many differences between the iteration speed. In the case of other parameters are the same, the three kinds of turbulence model, comparing the computing time, such as table 1.

Table 1. Comparison of iterative steps for different turbulence models

| turbulence model types       | iteration steps |
|-----------------------------|----------------|
| standard $k-\varepsilon$ model | 203            |
| RNG $k-\varepsilon$ turbulent model | 251            |
| Realizable $k-\varepsilon$ model | 212            |

Compared in table 1 using the iterative steps under different turbulence models, when the solver Settings at the same time, the convergence criterion is used in counter rotating valve port model simulation when different turbulence models for simulation speed there is a certain gap. The table 1 shows that the standard $k-\varepsilon$ model of iterative fastest, iterative step was only 203 steps, Realizable $k-\varepsilon$ model iteration step speed close to the standard $k-\varepsilon$ model for 212, iterative speed is the slowest RNG $k-\varepsilon$ model for 251 steps. Therefore, only focus on convergence speed, the standard $k-\varepsilon$ model compared with other two kinds of models, fastest convergence speed, can maximum limit save the simulation time.

4.4. Different turbulence model mean field analysis

Selection process of produced liquid transfer valve fully open the flow field in the center of surface for the analysis of rotary valve internal pressure, velocity and turbulent kinetic energy, in the past usually only select the default turbulence in the flow field analysis of model simulation [17,18], do not know to choose other turbulence models will have what kind of impact on the simulation results, so before you study the influence of other factors flow field using different turbulent flow field in the simulation model of rotating valve, analysis the different turbulence models of rotary valve pressure contours and velocity contours relation and distinction between [19, 20].

When the liquid process of rotary valve fully open, the standard $k-\varepsilon$ model RNG $k-\varepsilon$ model and Realizable $k-\varepsilon$ model simulation to get the rotary valve of neutral surface stress nephogram, as shown in figure 7.

![Figure 7. Stress cloud diagram of valve.](a) Standard $k-\varepsilon$ model (b) RNG $k-\varepsilon$ turbulent model (c) Realizable $k-\varepsilon$ model

By observing and comparing the turbulent flow field under the model of stress nephogram may know, rotary valve flow passage of the maximum pressure in appear within the inlet, when the liquid through the oil into the tank because the area is reduced, less stress, and in the liquid from the oil hole into the tank, the corner there will be a minimum of pressure in the runner. Then the liquid from the tank into the low pressure oil cavity, and further reduce the pressure on the tank and oil cavity between the corner area of low pressure area. Three kinds of turbulence model under the stress nephogram of all show the same general trend in the corresponding position of high and low pressure area, and gradient similar...
cloud image change. Highest in maximum and minimum pressure when the system pressure is the same, but three groups at the lowest pressure calculation model is different, the differential pressure after the minimum Realizable \( k-\varepsilon \) model calculation, the export under the most pressure, RNG \( k-\varepsilon \) model calculated maximum differential pressure, outlet pressure to a minimum, the standard model of differential pressure center.

![Velocity cloud of valve.](image)

(a) Standard \( k-\varepsilon \) model   
(b) RNG \( k-\varepsilon \) turbulent model   
(c) Realizable \( k-\varepsilon \) model

Figure 8. Velocity cloud of valve.

When the liquid process of rotary valve fully open, the standard \( k-\varepsilon \) model RNG \( k-\varepsilon \) model and Realizable \( k-\varepsilon \) model simulation speed of the rotary valve neutral cloud, as shown in figure 8.

By comparing observation under three kinds of turbulence model of flow field velocity cloud know, liquid after getting into the mouth, flow area with reduced, liquid flow rate increases, and in the valve port and groove formed high-speed flow; When the liquid into the oil cavity flow area due to increased liquid flow rate is reduced, the high-speed flow gradually disappear. On the maximum velocity and minimum speed calculation, the minimum speed are the same and all is zero, but the larger gap between the maximum velocity is calculated by the RNG \( k-\varepsilon \) turbulent model to calculate the flow velocity, Realizable \( k-\varepsilon \) model to calculate the minimum flow speed. To the low speed region of different model to calculate the value is different, some Realizable \( k-\varepsilon \) model minimum speed, RNG \( k-\varepsilon \) model maximum speed, speed of the standard \( k-\varepsilon \) model centered; High-speed area mainly concentrated in the tank and valve mouth corner area, high-speed RNG \( k-\varepsilon \) model district area is smaller than the other two models, the standard \( k-\varepsilon \) model and Realizable \( k-\varepsilon \) model to calculate the area of high-speed area.

Comparing the turbulence model to calculate the flow field of the rotary valve features, although the RNG \( k-\varepsilon \) model calculation is more accurate for low speed region, at the same time, the model calculated the pressure change is bigger, and long time than other two groups of RNG \( k-\varepsilon \) iteration; Although pressure changes the minimum Realizable \( k-\varepsilon \) model, a relatively large number but iteration steps.

5. Conclusion
Combination of Fluent simulation software of rotating valve flow field characteristics of rotating valve, comprehensive comparison speed rotary valve interior flow field of the pressure cloud: rotary valve under different turbulence model oil inlet oil pressure is the same, oil outlet oil pressure slightly difference, differential pressure, the largest is the RNG \( k-\varepsilon \) model and iterative longest; RNG \( k-\varepsilon \) turbulent model calculation with the other two groups when calculating the low velocity of flow field results are accurate and Realizable \( k-\varepsilon \) model is more similar with RNG \( k-\varepsilon \) model calculation results.

Acknowledgments
This work was financially supported by Liaoning province natural science fund guidance plan "Hydraulic support with impact resistant double security key techniques of large flow
valve”(20180550584) and Yingkou Institute of Technology Outstanding Science and Technology Support Project 2018.

References
[1] Guochao Zhao, Hui Wang, Yuning Song etc. Simulation and experimental verification of flow field dynamic characteristics of rotary shock valve [J/OL]. Mechanical Science and Technology: 1-7[2019-10-21]. https://doi.org/10.13433/j.cnki.1003-8728.20190185.
[2] Jing Wang. Study on a New Electro-hydraulic Exciter for Dynamic Battery Vibration Test-bed [J/OL]. Mechanical and Electrical Engineering Technology. 2019(09):94-99[2019-10-21]. http://kns.cnki.net/kcms/detail/44.1522.TH.20190919.1352.062.html.
[3] Wanrong Wu, Qian Huang, Weimei Wu, etc. Simulation Study of Two-DOF High Frequency Vibration Transfer Valve Based on AMESim [J]. World Science and Technology Research and Development. 2016, 38(06): 1228-1233.
[4] Yuanyuan Xie. Influence of Cycloidal Turn Valve Turner on Characteristics of Full Hydraulic Steering System [J]. Machine Tools and Hydraulic. 2017, 45(04): 122-128.
[5] Hui Wang, Yuning Song, Guochao Zhao, etc. Simulation and Analysis of Dynamic Performance of Double-stage Overload Protection Safety Valve [J]. Journal of Liaoning University of Engineering and Technology (Natural Science Edition). 2017, 36(07): 750-754.
[6] Hui Wang, Yuning Song, Yejiao Song. Influence of Structure Parameters on the Performance of Hydraulic Impulse [J]. Journal of Liaoning University of Engineering and Technology (Natural Science Edition). 2017, 36(10): 1075-1080.
[7] Dong Han. High Speed Switching Technology of Electro-hydraulic Vibration [D]. Zhejiang University. 2014.
[8] Xiao Wang. Design and Performance Analysis of Direct Dynamic Variable Area Gradient Turning Valve [D]. Dalian University of Technology, 2014.
[9] Weichuan Jing. Study on the Working Characteristics of Hydraulic Vibrator for Distribution Flow of Rotary Valve [D]. Liaoning Technology University, 2015.
[10] Lingrui Gu. Study on Control Strategy of Independent Core Rotary Valve under Large Inertia [D]. Zhejiang University. 2017.
[11] Wen Wang. Study on Dynamic Characteristics of Hydraulic Volatile Vibration System [D]. Taiyuan University of Technology, 2009.
[12] Hong Zhao. Study on Dynamic Characteristics of Hydraulic Shock Piling System [D]. North University China, 2018.
[13] Qi Sun. Experimental Study on Vibration and Working Characteristics of Hydraulic Hammer with Flow Distribution [D]. China Geosciences University (Beijing), 2012.
[14] Weirong Li. Study on Vibration Characteristics of High Frequency Uniaxial Electro-hydraulic Vibration Table [D]. Zhejiang Technology University, 2013.
[15] Hui Chen, Chunyue Lu, Hong Zhao,etc. Study on Dynamic Characteristics of Hydraulic Vibrator Piling System [J]. Machine Tools and Hydraulic, 2019,47(10):92-98+159.
[16] Hongcheng Chou, Hengsu Deng. Study on Dynamic Characteristics of Hydraulic Vibration System of Rotary Type Flow Distribution [J]. Machine Tools and Hydraulic, 1988(05): 30-35.
[17] Yuning Song. Study on the impact-resistant double-stage protection large-flow relief valve for hydraulic support [D]. Liaoning Technology University, 2017.
[18] Hui Wang, Jiyang Li. Flow-solid coupling of hydraulic support with large flow hydraulic controlled one-way valve [J]. Journal of Liaoning University of Engineering and Technology (Natural Science Edition), 2014, 33(05): 622-626.
[19] Hui Wang, Yuning Song, Yejiao Song. Simulation of the flow field of sliding boots of hydraulic axial piston pump [J]. Journal of System Simulation, 2015, 27(03): 634-642.
[20] Hui Wang, Xiaoman Zhang, Yuning Song. Flow characteristic curve of MATLAB solution electrohydraulic servo valve [J]. Journal of Electronic Measurement and Instrumentation, 2015,29(08):1236-1244.