Dynamic characteristics analysis of a hydraulic valve by FEM and CFD method

Xiaoli Huang 1, Qian Liu 1, Qiang Wei 1, Xiaohe Deng 2, *

1 Wuchang Shipbuilding Industry Group Co., Ltd, Wuhan, China
2 School of Automotive Engineering, Wuhan University of Technology, Wuhan, China

*Corresponding author e-mail: xiaohe_deng@whut.edu.cn

Abstract. The dynamic characteristic of the flood valve is an important index to ensure its safety and reliability. In this paper, CFD computational fluid dynamics (CFD) method is used to model and simulate the three-dimensional unsteady internal flow field of the valve during its working process, so as to obtain the dynamic distribution of the internal flow field parameters of the valve under specific working conditions. The modal analysis of the valve body is carried out to provide a calculation method and data support for the evaluation of the vibration and noise level during the operation of the valve.

1. Introduction

Virtual simulation technology is an economical and fast method to study the working process of the flood valve, and it is a very important research means in engineering. Compared with the physical prototype test method, virtual simulation eliminates the labor and material costs required by model making, and can better adapt to the uncertainties caused by the manufacturing error and the change of operating conditions in engineering.

At present, the virtual simulation analysis of valves mostly adopts the numerical analysis method, which mainly involves the CFD flow modeling or optimization problems. Gomez et al. used CFD method to conduct dynamic simulation on the analysis of cone valve characteristics [1], studied the possibility of improving dynamic characteristics by changing fluid viscosity, and considered the effect of cone Angle on pressure. Wang et al. studied the use of ANSYS-FLUENT [2] to simulate the cavitation flow of a control valve. In the paper [3], the interaction between liquid and channel wall was considered and the application of fluid-structure coupling technology is adopted.

In this paper, CFD computational fluid dynamics method is adopted, and based on the commercial software Fluent and the relevant pre and post processing system, the 3D unsteady internal flow field dynamic modeling and simulation of the working process of the side-flood valve is carried out, so as to obtain the dynamic distribution of the flow parameters in the valve under specific working conditions, such as: When conditions permit, the model is compared and modified through experiments, so that the virtual simulation results have the same accuracy as the physical prototype bench test, which provides a calculation method and data support for the evaluation of the vibration and noise level in the working process of the flood valve.
2. Theoretical basis of CFD simulation of internal flow field in flood valve

2.1. Control equation

Different from the Lagrange description method commonly used in solid mechanics, the motion law of fluid is generally described by Euler method: the control equation is used to mathematically describe the conservation of mass, momentum and energy in the flow process. The laws of conservation of mass, conservation of momentum and conservation of energy are the basic laws of fluid motion.

For flow problems in turbulent state, additional turbulence equations should be followed to describe the statistical laws of complex turbulent motions. According to the continuum hypothesis, the flow of the fluid must follow the law of conservation of mass: in unit time, the increase in the mass of the fluid microelement (in CFD, it is called "Control volume" or "Cell" after dispersion) is equal to the net mass flowing into the microelement during the same time interval. Namely, the mass conservation equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0$$

(1)

Similarly, any flow must also follow the law of conservation of momentum: the rate of change of the momentum of the fluid in the infinitesimal body with respect to time is equal to the sum of all external forces acting on the infinitesimal body. Based on this, the momentum conservation equation can be derived:

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = \rho f_i - \frac{\partial}{\partial x_j} (\tau_{ij}) + \frac{\partial}{\partial x_j} (k \frac{\partial T}{\partial x_j}) + S_e$$

(2)

For a system with heat exchange, the law of conservation of energy must also be followed: the rate of increase of energy in the element is equal to the net heat flow into the element plus the work done on the element by physical force and surface force. The essence of this law is the "first law of thermodynamics". Therefore, the energy conservation equation can be obtained:

$$\frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_j} (\rho E u_j) = \rho f_j u_j - \frac{\partial}{\partial x_j} (\tau_{ij} u_i) + \frac{\partial}{\partial x_j} (k \frac{\partial T}{\partial x_j}) + S_e$$

(3)

In the above three governing equations:

$$\tau_{ij} = (\mu + \mu_t) \frac{\partial u_i}{\partial x_j} + \frac{2}{3} (\mu + \mu_t) \frac{\partial u_i}{\partial x_j} - \frac{2}{3} (\mu + \mu_t) \frac{\partial u_i}{\partial x_j} \delta_{ij}$$

(4)

$$E = e + \frac{1}{2} u_i u_j$$

(5)

Where, $\delta_{ij}$ is the Kolonik operator, and $S_e$ is the source term.

The transient dynamics method of unsteady internal flow field is used to model and simulate the internal flow field of valve.

2.2. Turbulence model

k-epsilon turbulence model is proposed. Among them, the transport equations of turbulent kinetic energy $K$ and turbulent kinetic energy dissipation rate $\epsilon$ are shown as follows.

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \epsilon$$

(6)

$$\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_j} (\rho \epsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_{1\epsilon} \frac{\epsilon}{k} \frac{\partial k}{\partial x_j} - \rho C_{2\epsilon} \frac{k}{k + \sqrt{\epsilon}} \frac{\epsilon^2}{k + \sqrt{\epsilon}}$$

(7)

In which,

$\rho$: fluid density;
$K$: turbulent kinetic energy;
$E$: turbulence kinetic energy dissipation rate;
$\epsilon$: turbulence kinetic energy dissipation rate;
U: momentum in the y direction;  
Gk: the term generated by the turbulent kinetic energy k caused by the average velocity gradient. 

$$G_k = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j}$$  \hspace{1cm} (8)$$

Where, \( \mu \) is dynamic viscosity; \( \mu_t \) is turbulent viscosity. In this paper, the Standard k-epsilon model is used to describe the turbulence in the gas-liquid two-phase flow field in the high-pressure blowout system.

2.3. Near-wall treatment

The Standard k-epsilon model is mainly suitable for fully developed turbulence. These models are effective for the flow in the core turbulent region but have limitations for the insufficiently developed turbulence on the wall surface. The Standard k-epsilon turbulence model is supplemented by introducing a "wall function" to describe the near-wall region semi-empirically. In other words, the wall function method and the Standard K-Epsilon turbulence model are used to solve the problem of high-speed turbulent flow in the whole fluid solution domain. Using the wall function method, it is not necessary to encrypt the wall area when dividing the mesh.

The wall function method sets the first node P on the near wall in the region where the logarithm law holds. Point P is within the region of fully developed turbulence. The dimensionless velocity \( u^+ \) and the dimensionless distance \( y^+ \) are introduced to describe the flow in the logarithmic law layer and viscous bottom layer:

$$u^+ = \frac{u}{u_\tau} = \frac{u}{\sqrt{\tau_w / \rho}}$$  \hspace{1cm} (9)$$

$$y^+ = \frac{y \rho u_\tau}{\mu} = \frac{y}{\nu} \sqrt{\tau_w / \rho}$$  \hspace{1cm} (10)$$

Where, \( u \) represents the time-average velocity of the fluid; \( u_\tau \) is the wall friction velocity; \( \tau_w \) is wall shear stress; \( y \) is the distance to the wall.

2.4. PISO algorithm

PISO (Pressure Implicit with Split of Operators) is an Implicit operator segmentation algorithm based on Pressure. At first, it is a pressure velocity calculation program for non-iterative calculation of unsteady compressible flow.

The control equation of fluid mechanics is formally expressed as the coupling of the continuity equation and the motion equation, and is reflected in the coupling of the velocity and pressure in the variables. The velocity and pressure need to be solved at the same time. In numerical calculation, the method of solving velocity and pressure simultaneously coupled is called "simultaneous algorithm". Generally, the algorithm is cumbersome and the solution efficiency is not high. The "split algorithm" is another solution strategy. Its idea is to establish the iterative relation between the velocity and the pressure, and to seek the stable solution in the iteration. Some of the most popular algorithms in CFD, such as SIMPLE, SIMPLEC, and PISO, are "split algorithms". Both SIMPLE and SIMPLEC algorithms are two-step algorithms, one prediction and one correction. The PISO algorithm adds a correction step, which is one-step prediction and two-step correction. The advantage of doing this is that both the momentum equation and the continuity equation are satisfied. The specific steps are as follows:

1) Firstly, a velocity field is assumed to calculate the coefficients and constant terms of the momentum discrete equation;

2) Then assume a pressure field \( P^* \);

3) Using the pressure field \( P^* \), the discrete momentum equation (2) is solved to obtain the velocity field;
4) Solve the pressure correction equation derived from the discrete continuity equation to obtain the first pressure correction \( P' \);

5) The velocity field after the first improvement is obtained by modifying the velocity according to \( P' \);

6) Solve the second pressure correction equation from the improved velocity field of the first time, and get \( P'' \);

7) According to the second correction speed of \( P'' \) and the correction pressure at the same time, the sum is obtained;

8) Recalculation of the momentum discrete equation by using the modified U, V and W is taken as a new repeating step 2, and the next iteration is carried out until a convergent solution is obtained.

With the development of CFD algorithm, PISO algorithm has been widely used in the iterative calculation of steady and unsteady flow. In this paper, PISO algorithm is used to solve the 3D unsteady internal flow field of flood valve.

3. Establishment and simulation of CFD model for flood valve

3.1. Establishment of 3D model

As shown in Fig. 1, the 3D geometric models of each flow parts of the flood valve were first established in CATIA and assembled into a whole. On this basis, the geometric model of the flow field in the valve body was worked out through Boolean operation (in order to ensure the accuracy of CFD solution, the outlet pipeline was extended by 200mm), as shown in Fig. 2.

![Fig. 1 3D Geometric Model of the flood valve](image1)

![Fig. 2 3D Geometric Model of the internal Flow Field of the flood valve](image2)

3.2. Discrete Grid

Mesh division is the process of discretization of fluid domain, which is the discretization of governing equations in space mathematically, and belongs to the preprocessing of CFD model. In this paper, the professional pre-processing software HyperMesh is used to divide the structured mesh of the blowout system.

The quality of mesh directly affects the precision of solution and the cost of calculation. In general, the generation of unstructured mesh is relatively simple, but its disadvantage is that the solution accuracy is not high, especially it can not track the free liquid level well. For the same solution precision, many times the number of grids is needed. Both theoretical and engineering studies show that structured grids have more obvious advantages when solving complex flow field problems. Less grids can obtain more reliable calculation results and consume much less computer time. Based on the above considerations, structured grids were used in this paper to discretization of the geometric space of the sea gate valve.
The number of hexahedral structured grids obtained was 2.4 million, among which the grids were properly encrypted where the wall surface and section changed. The results of grid dispersion are shown in Figure 3 and Figure 4.

![3D CFD grid model of the internal flow field of flood valve](image1)

![Local grid of internal flow field of flood valve](image2)

**Fig. 3** 3D CFD grid model of the internal flow field of flood valve  
**Fig. 4** Local grid of internal flow field of flood valve

### 3.3. Boundary conditions
According to the actual working conditions, the boundary conditions are determined as follows:
1) It is assumed that the pipeline is also filled with seawater in the initial state;
2) The entry is velocity boundary condition, and \( v = 3.0 \text{m/s} \);
3) The export is continuous;
4) Other boundary conditions include: fixed wall surface.

### 3.4. Simulation results
According to the simulation calculation, the cloud diagram of velocity of internal flow field of flood valve can be obtained, as shown in Fig. 5. The pressure cloud diagram of the flow field inside the flood valve is shown in Fig. 6. The flow field diagram of the valve is shown in Fig. 7.

### 4. Modal analysis of valve body based on finite element method
The structural mode of flood valve is one of the important contents to characterize its dynamic performance. In order to reduce the vibration and noise of the valve in the process of operation, the mode of the flood valve are calculated. For the original structure scheme, the finite element model of the valve body is shown in Fig. 8. The first six modes of the valve body are shown in Fig. 9.

![Cloud diagram of velocity of internal flow field of flood valve](image3)

**Fig. 5** Cloud diagram of velocity of internal flow field of flood valve
Fig. 6 Pressure cloud diagram of internal flow field of flood valve

Fig. 7 Diagram of internal flow field of the flood valve

Fig. 8 Finite element model of valve body
In order to avoid possible resonant frequencies, the original valve body structure was optimized and modal analysis was performed. The optimized finite element model is shown in Fig. 10, and the optimized mode shapes are shown in Fig. 11.

According to the internal flow field diagram, it can be seen that the dynamic exciting force is mainly caused by the internal flow field vortex. Through modal analysis, the main vibration shapes and weak parts of the structure vibration can be obtained. The strength and stiffness of the valve can be improved through the optimization of the local structure, so as to reduce the vibration of the valve during operation and achieve the purpose of reducing the noise.
Fig. 10 Finite element model of valve body after optimization

Fig. 11 First six modes of valve body after optimization
5. Conclusion
By CFD and FEM method to carry out the flood valve in the internal flow field in the three dimensional transient working process of the modeling and simulation, the internal flow field inside the dynamic distribution of flow parameters is obtained, and optimized the flood valve before and after the modal is analyzed. This method was verified in the flood valve products dynamic performance simulation analysis of the feasibility.

References
[1] Gomez, I.; Gonzalez-Mancera, A.; Newell, B.; Garcia-Bravo, J. Analysis of the Design of a Poppet Valve by Transitory Simulation. Energies 2019, 12, 889.
[2] Wang, H.; Zhu, Z.; Zhang, M.; Li, J.; Huo, W. Investigation on cavitating flow and parameter effects in a control valve with a perforated cage. Nucl. Eng. Technol. 2021.
[3] 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. Fluids Struct. 2019, 90, 432–449.