Research Article

Dynamic Simulation of a Warship Control Valve Based on a Mechanical-Electric-Fluid Cosimulation Model

Jiazhen Han,1,2 Yudong Xie,1,2 Yong Wang,1,2 and Qixian Wang3

1School of Mechanical Engineering, Shandong University, Jinan 250061, China
2The Key Laboratory of High-efficiency and Clean Mechanical Manufacture, Ministry of Education, Jinan 250061, China
3Ocean College, Zhejiang University, Hangzhou 310058, China

Correspondence should be addressed to Yudong Xie; ydxie@sdu.edu.cn and Yong Wang; meywang@sdu.edu.cn

Received 10 November 2020; Revised 1 March 2021; Accepted 9 April 2021; Published 19 April 2021

Academic Editor: Han Zhang

Copyright © 2021 Jiazhen Han et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

Control valves have an important function in the warship power system. In engineering practice, the fluid oscillation inside the control valve causes the additional load to the valve actuator. When the additional load is added to the original load of the valve, it is possible that the required driving force (or driving moment) of the valve is greater than the maximum force (or moment) output by the actuator, which may cause the abnormal stop of the actuator. Conventionally, the interaction effect of the valve mechanical and electric components on the valve chamber’s flow field cannot be considered in computational fluid dynamics (CFD) simulations, so the oscillating fluid loads cannot be accurately obtained. In order to solve this problem, the mechanical-electric-fluid integrated valve model, using the FLUENT and AMESim cosimulation method, was developed to embody the interaction effect between the components of each part of the control valve and exhibit the fluid oscillation during the operating process of the control valve. Compared with the pure software simulations, the unsteady flow characteristics and dynamic response of the actuator were synchronously obtained in this study, which accurately captured the sudden fluid loads required for further compensation. At the same time, the differences in performance of different valve plugs were compared. The stability time of the valve plug and oscillation amplitude of the unstable fluid loads were distinct for control valves with different flow characteristics. The results can aid in understanding the instability mechanism of the fluid load in the control valve better, which provides the calculation basis for compensating the additional load on the valve plug and improve the reliability of the control valve.

1. Introduction

Warship control valves are valves that regulate the fluid transportation of the nuclear islands of nuclear power plants, conventional power plants (marine steam turbines), and auxiliary control systems in commercial and military ships. When warships perform complex actions, with the sudden influx of a large-flow working medium, the control valve is in the dynamic process of starting, closing, and adjusting. An unsteady fluid-structure interaction (FSI) effect is formed among the pressure, flow, and structures (such as the valve plug) in the control valve. The interaction between the flow characteristics of the fluid and the movement of the valve internals leads to unstable and oscillating fluid loads on the valve plug, which causes the additional load on the actuator. When the additional load is superimposed on the original load of the control valve, the driving force or driving torque required by the valve may be greater than the maximum force or torque output by the actuator (to avoid damage to the valve sealing surface, valve stem, or valve shaft due to the output force or torque of the actuator greatly exceeding the valve load, the maximum output force or torque is 1.8 times of the normal load), which would cause the actuator to fail to act or perform abnormally. In order to eliminate the influence of the fluid oscillation and improve the control accuracy and reliability of the control valve, the output torque of the actuator of the control valve can be compensated on demand. For example, when the valve plug is
subject to a large fluid load, the actuator will output a large torque; when the valve plug is subject to a small fluid load, the actuator will output a small torque. Therefore, it is necessary to accurately obtain the oscillating fluid loads on the valve plug, so as to achieve the purpose of compensating for the additional coupling load.

The unsteady fluid load and fluid oscillation of the control valve have previously been extensively explored. Owing to the dynamic movement of the valve, pressure and flow fluctuations can occur within the valve systems in rocket propulsion systems, and it may be accompanied by cavitation, resonance, and even failure. To predict the transient behavior of the valve motion and provide guidance for engine operations, simulations of valve motion using a mesh library strategy were presented by Ahuja et al.[1], Cavallo et al.[2], and Shipman et al.[3]. The results captured the valve’s flow characteristics and unsteady phenomena. Saha et al.[4] investigated the dynamic flow process of a pressure valve using FLUENT computational fluid dynamics (CFD) software. They applied a force balance method in combination with dynamic mesh technology to predict the spool position and calculate the transient variation of the flow forces. The simulation results also demonstrated the effect of the friction coefficient between the spool and the valve body on the spool movement. To investigate the effect of high-pressure safety valve dynamics on steady flow performance, Beune et al.[5] conducted CFD analysis with FSI using a multiple grid approach. The results indicated the opening characteristics and possible instabilities of the valve for liquid and gas flows. Additionally, they proposed targeted analysis methods for sensitivity to valve dynamics and opening characteristics. Investigating the vibration phenomenon of the steam valve in the piping system of the power plant under partial opening conditions, Morita et al.[6] focused on the flow fluctuations around the steam control valve to understand the valve instability using experiments and CFD calculations. The results indicated that the complex flow attached to the valve body can cause fluctuation in the rotating pressure and an asymmetric load under the middle-opening condition, which are the main cause of valve vibrations. Misra et al.[7] studied the mechanism that causes the self-excited vibration of a pipeline system based on a fluid-structure dynamic model of the control valve circuit. The numerical results indicated that the self-excited vibration mechanism of the control valve was complex and was affected by the valve itself and the pipeline system, such as water hammer effect, negative hydraulic stiffness, or acoustic feedback. To further understand the mechanisms of flow-induced vibrations in the Venturi valve, Yonezawa et al.[8] conducted numerical simulations and experiments using rigid and flexible valve head supports. They explained the inherent relationship between the unsteady flow and valve head vibrations for various operating conditions. They also verified that the high-pressure region on the surface of the valve plug was the main reason for the lateral oscillating fluid force, which was a negative damping force. Many other studies of control valves have also conducted [9–14].

However, the electric control valve is a coupled closed-loop system consisting of mechanical components called the regulating mechanism and actuator, an electric component called the control system, and a fluid component called the flow field of the valve chamber. As the valve works, there exists coupling among the fluid flow, the mechanical component, and the electric component. In the numerical simulation of the valve flow field, CFD software such as FLUENT or CFX, cannot model the actuator and control system. As a result, the effect of these two components on the flow field cannot be considered, inevitably resulting in a large simulation deviation of the fluid load on the valve plug. In addition, most of the displacement curves of the valve plug in the current studies were known beforehand. However, in most automatic valves, such as the electric control valve, the valve plug displacement is the result of automatic feedback in the system. And the opening of this control valve depends on the pipeline pressure fluctuations, the electric driving force of the actuator, and the unbalanced force of the mechanical structure, which are in turn affected by the valve opening. Certainly, the opening curve is difficult to determine in advance. Software such as AMESim or Simulink can calculate the opening curve of the valve plug. But during the dynamic adjustment of the control valve, a coupling between the flow field and the regulating mechanism occurs because of the effect of FSI: the fluid affects the movement of the valve plug by changing its loads, whereas the variation in the valve plug position directly changes the flow field and continues to affect the movement of the valve plug. Here, the valve housing model of the control valve is simplified to a one-dimensional model. Considering the unsteady coupling effect in this limited area is difficult, and the effect of the flow field on the regulating mechanism, actuator, and control system is not adequately considered; thus, the calculation accuracy of the fluid load on the valve plug is also affected. With the increasing complexity of such concerns, the relevant literature shows that some scholars in the fields of machinery, electric power, and architecture [15, 16] have adopted the coupled cosimulation method using multidisciplinary software to effectively solve some specific problems. In general, the basic function of the control valve is to change the medium flow, pressure, temperature, or other parameters by receiving the control signal output from the regulating control unit. In this study, we selected the flow rate as the control target. The actuator of the control valve adjusts the opening and closing of the valve plug according to the output signal of the controller so that the outlet flow rate attains the desired value. This is a complex feedback control process. Therefore, the focus of this study was to conduct a holistic study on the mechanical-electric-fluid system of the control valve using the coupling method of both FLUENT and AMESim and consider their mutual effect between the various valve components that are difficult to observe in a single software simulation. Ultimately, the fluid load on the valve plug is captured, which provides the calculation basis for further load compensation.

By means of the cosimulation, an integrated model of the control-valve regulating mechanism, actuator, control system, and flow field of the valve chamber was established.
verification of this coupled scheme was performed. The velocity and displacement of the valve plug were computed in AMESim to obtain the opening curve of the valve plug. We applied a user-defined function (UDF) and dynamic mesh technology in FLUENT to control the movement of the valve plug, and the transient flow field of the valve plug at various movement times was obtained. Thus, the likely unsteadiness of the flow, dynamic response of the actuator, and different behavior of the control valve with various flow characteristics are presented in this paper.

2. Mathematical Models

2.1. Flow Equation. The fluid in this study was in an incompressible and turbulent flow. Hence, the flow equations satisfied the conservation laws of mass, momentum, and energy [17, 18], which were numerically solved in the commercial software package FLUENT.

The governing equations are described as follows:

Continuity equation:

\[ \nabla \cdot (\vec{U}) = 0. \] (1)

Momentum equation:

\[ \frac{\partial (\rho \vec{U})}{\partial t} + \nabla \cdot (\rho \vec{U} \vec{U}) = -\nabla p + \nabla \cdot (\nabla \cdot \vec{U}) + \rho \vec{f}, \] (2)

where \( \vec{U} \) is the velocity vector, \( t \) is the time, \( \rho \) is the density, \( p \) is the pressure, \( \nabla \cdot \) is the gradient, and \( \vec{f} \) is the body force vector.

As heat exchange is not involved, the energy equation is not considered.

2.2. Fluid-Structure Interaction Dynamics Equation. Figure 1 shows the sketch map of the simulated control valve. The valve was mainly made up of a valve body, a valve plug, a valve stem, an electric actuator, a controller, a valve displacement sensor, and a flowmeter. In this study, the control target was the valve outlet flow rate. The operation principle of the control valve was as follows: when the warship’s working conditions or operating conditions changed, the outlet flow rate of the control valve deviated from the desired value. Through the detection and transmission of the flowmeter and transmitter, the outlet flow rate was sent to the controller. The controller compared this flow rate signal with the target signal and calculated an output using the corresponding control rule. Based on this output, the motor calculated speed and torque. This torque would be output by the actuator. Then, according to the feedback information of the valve positioner, the flow area between the valve plug and the valve seat was changed to make the valve outlet flow rate consistent with the desired value, so as to realize the flow control behind the valve.

The valve plug and stem frequently moved along the vertical orientation because of the constraint of the valve body structure. When the control valve was operating, the valve plug and stem moved under the action of four forces: the fluid load \( F_1(t) \), electric driving force \( F_2(t) \), gravity force \( G \), and friction force between the valve stem and the packing \( F_f \). As shown in Figure 1. According to Newton’s law, the acceleration \( \dot{Y}(t) \) of the valve plug and stem relates to their equivalent mass \( m \) as follows:

\[
\begin{align*}
\{ m\dot{Y}(t) &= F_1(t) + F_2(t) + G + F_f \} = c\dot{Y}(t),
\end{align*}
\] (3)

where \( \dot{Y}(t) \) is the velocity of the valve plug and stem, \( c \) is the equivalent viscous damping coefficient, and the friction force is equivalent to a viscous damping force, which is proportional to the speed.

3. Cosimulation Models

3.1. Schematic Design for the Cosimulation. The schematic of the cosimulation platform is shown in Figure 2. Only one executable application was available in the cosimulation. The FLUENT model was identified as the master and pilots of the AMESim model, which was identified as a slave.

For the two models to communicate with each other, an interface referred to as “UserCosim” was created on the AMESim sketch. With this interface, a dynamic link library (DLL) was generated from the AMESim model. During the cosimulation, FLUENT exchanged the input and output of the data and implemented the call of DLL files through a secondary development function called the UDF. In this process, the medium software Microsoft Visual Studio was used to achieve the bridging role. All data were synchronously transmitted and exchanged, and communication between the two models occurred only at fixed time steps (see Figure 3).

Furthermore, the movement of the valve plug was completed by the dynamic mesh technology. In FLUENT, the dynamic mesh model can be used to simulate flows where the shape of the flow domain was changing with time because of motion on the flow domain boundaries. The motion can be a prescribed motion (for example, you can
define the linear and angular velocities about the center of gravity of a solid body with time) or an unprescribed motion where the subsequent motion was determined based on the solution at the current time (for example, the linear and angular velocities were computed from the force balance on a solid body). In order to apply the dynamic mesh model, a starting volume mesh and the description of the motion of moving zones should be provided. FLUENT provided three dynamic mesh motion methods to update the mesh in the moving zone, which were smoothing, layering, and remeshing methods.

3.2. AMESim Model for the Cosimulation. The cosimulation model was built on the AMESim system (see Figure 4). The model is divided into two parts: the data exchange module and integrated model of the control valve. The data exchange module, called the UserCosim interface, was used to set the exchanged variables. The left side of the UserCosim interface was the variables calculated by AMESim and provided to FLUENT, and the right side was the variables calculated by FLUENT and sent to AMESim. Every variable appeared in pairs and was correlated with each other using the receiver and transmitter submodels. And submodels with the same color belong to the same variable. A receiver submodel can be used to receive a variable from a transmitter submodel without any visible connection on the sketch. In other words, the receiver output was actually a duplicate of the transmitter input. All the exchanged variables on the boundary of the two software are presented in Figure 5. The integrated model was composed of the control-valve flow field, regulating mechanism, actuator, and control system. In AMESim, the flow field was represented by inlet and outlet pressure sources, which were built using fluid models from the hydraulic library for flow boundary conditions. The working fluid was liquid water with a density of 998.2 kg/m³ and dynamic viscosity of 0.001003 kg/(m·s). The regulating mechanism and the actuator were represented by a mass, rack, pinion, reducer, gear, and motor, which were built from the mechanical library. The function of the electrical motor was to convert the output signal of the controller into the thrust or torque of the valve. The function of the gear, reducer, rack, and pinion was to convert the thrust or torque into the displacement. The regulating mechanism was composed of the valve plug and stem, which were represented by a mass block. And a displacement sensor was connected next to the mass block to obtain the valve plug displacement. The function of the regulating mechanism was to change the flow area of the valve according to the displacement signal. The control system was represented by a proportional-integral-differential (PID) controller and a series of sensors, which were built using electric signal models to link the control units and physical sensors. Here, the sensor used a flowmeter, which was installed on the downstream pipeline of the control valve to monitor the valve outlet flow and connected with the PID controller to form a flow feedback system. The comparison element compared the feedback signal output by the flowmeter with the target value signal to obtain the deviation signal. Then, the controller calculated the deviation signal and transmitted the results to the actuator. The initial parameters of each part of the model are listed in Table 1.

In the AMESim to FLUENT direction, when the cosimulation was launched, AMESim provided the pressure source values of the valve inlet and outlet as boundary conditions for FLUENT. The PID controlled the motor
operation according to the flowmeter measurement provided by FLUENT to adjust the valve opening and closing to maintain the actual flow rate close to the desired flow rate. Meanwhile, AMESim calculated the force balance on the valve plug during the opening and closing of the control valve based on Equation (3). Subsequently, the displacement of the valve plug, resulting from the force balance calculation in AMESim, was fed back to FLUENT to enable it to update the valve plug position and the dynamic mesh around the valve.

3.3. FLUENT Model for the Cosimulation

3.3.1. Parameter Settings of the Numerical Approach in FLUENT. For the CFD simulation model of the control valve, the profile curve of the valve plug was designed according to the formula of linear inherent flow characteristics [20, 21]. The front and rear pipelines of the valve were lengthened twice and six times, respectively, compared with the pipe diameter to ensure sufficient fluid flow. The grid was generated using ANSYS ICEM. An unstructured grid was used to mesh the entire flow field, and the grid around the valve seat and plug was refined. Before selecting the final cell number, a grid independence study was performed with several types of computational cells. Figure 6 shows that the flow rate at the valve outlet varied with different grid numbers. When the grid number was 120000, the flow rate did not change significantly. To save computation time, we adopted this grid scheme. Similarly, the time step was selected to be $10^{-5}$ s and the number of internal maximum iterations for a single time step was 300 after several levels of tests. Here, the dynamic parameters of the control valve no longer produced a further change with the variation in the time step. Figure 7 shows the grid model of the control valve with a 10% opening. The grid size was approximately 120000 cells.

The simulation was carried out in software ANSYS FLUENT 17.0; the RNG turbulence $k$-$\epsilon$ model was adopted [22, 23]. The flowing fluid inside the valve was liquid water as the same as AMESim. All the walls were assumed to have no slip. Pressure boundary conditions defined by the UDF were applied to the inlet and outlet of the control valve. During transient valve flow, the valve plug and stem moved inside the valve body. The boundary surface of the valve plug and stem was defined as the dynamic mesh zones in the dynamic mesh setting. The motion type of the dynamic zones was rigid body motion, which was an option available in FLUENT. Mesh around the dynamic regions needs to be defined to accommodate valve plug and stem motion. Smoothing and remeshing were employed for this mesh definition. The displacement of the valve plug, received from AMESim, was defined in the UDF. And it can be read by the macro "DEFINE_CG_MOTION()" from FLUENT to regenerate the mesh. A segregated solver was used and the PISO algorithm was applied to the coupling of pressure and velocity. And second-order upwind scheme was used for all discrete terms. In the transmit simulation, under relaxation factors have an important influence on the convergence of the simulation. Here, the pressure, the density, the body forces, $k$, $\epsilon$, and the turbulent viscosity were set as 0.3, 1, 0.7, 0.8, and 1, respectively. When the residuals were less

Table 1: Nominal parameters of the AMESim model.

| Part            | Parameter            | Unit | Value |
|-----------------|----------------------|------|-------|
| Control valve   | Inlet pressure resource | MPa  | 10    |
| Control valve   | Outlet pressure resource | MPa  | 8     |
| Rack            | Pinion diameter      | mm   | 30    |
| Gear            | Large diameter       | mm   | 100   |
| Gear            | Small diameter       | mm   | 50    |
| PID controller  | Proportional gain    | —    | 80    |
| PID controller  | Integral gain        | —    | 1     |
| PID controller  | Derivative gain       | —    | 0     |
| Constant signal | Desired flow rate    | L/min| 80000 |
| Mass            | Quality              | kg   | 10    |

Figure 4: Modified the AMESim model for cosimulation.

Table 4: Modified the AMESim model for cosimulation.

Figure 5: Exchanged variables between two solvers.
than $10^{-7}$ for continuity and $10^{-6}$ for other parameters, the
simulated flow field was considered to have attained the
convergence criterion.

In the FLUENT to AMESim direction, when the cosi-
mulation was launched, FLUENT calculated the flow around
the valve plug based on Equations (1) and (2), and then the
value of the fluid load on the valve plug due to this flow, as
well as the value of the inlet and outlet flow rate, was sent to
AMESim. The movement area of the valve plug was con-
trolled using dynamic mesh technology. Ultimately, data
exchange and coupling occurred between the codes of
AMESim and FLUENT.

3.3.2. FLUENT Validation. In order to prove the reliability
of the parameter settings and the solution scheme in the
FLUENT simulation, a comparison validation was per-
formed with the results of Qian et al.[24]. The model was
based on a pilot-control globe valve and simulated in ANSYS
FLUENT 17.0, maintaining all the parameter settings the
same as those of Qian et al. [24]. Figure 8 shows the steady
valve plug displacement for our computational results and
experimental results of Qian et al.[24]. The trends of the
simulation results and experimental results were within an
acceptable range. This demonstrated the feasibility of the
pure FLUENT simulation and validated the numerical set-
tings used for the simulation of the valve in this study.

3.4. Verification for the Coupled Scheme. Due to the limi-
tation of the experimental conditions, we conducted the
verification of the coupled scheme by comparing the results
between the pure FLUENT simulation and the FLUENT/
AMESim cosimulation with the same parameters. The model was based on a simplified spring-loaded check valve. The governing equation for this motion of the valve is expressed as follows:

\[ F'(t) = kX(t) + F_f + m' \ddot{X}(t), \]

where \( F'(t) \) is the fluid load on the valve plug, \( X(t) \) and \( \ddot{X}(t) \) are the displacement, velocity, and acceleration of the valve plug, respectively, \( k \) is the stiffness of the spring, \( m' \) is the mass of the valve plug, \( c \) is the equivalent viscous damping coefficient, and the friction force of the valve plug \( F_f \) is also equivalent to a viscous damping force, which is proportional to the speed of the valve plug.

In the pure FLUENT simulation, the valve plug motion was solved using an iterative scheme based on the classical Runge-Kutta method. When the number of iteration time steps was zero, the valve was in a static state. The acceleration of the valve plug \( \ddot{X}^n(t) \) at \( n \) time steps was calculated by

\[ \ddot{X}^n(t) = \frac{F^n(t) - kX^n(t) - c\dot{X}^n(t)}{m}, \]

where \( F^n(t) \) is the fluid load on the valve plug, \( X^n(t) \) is the displacement of the valve plug, and \( \dot{X}^n(t) \) is the velocity of the valve plug at \( n \) number of time steps.

Then, the displacement \( X^{n+1}(t) \) and velocity \( \dot{X}^{n+1}(t) \) of the valve plug motion at the next number of time steps \( n+1 \) become

\[ X^{n+1}(t) = X^n(t) + \dot{X}^n(t)\Delta t + \frac{1}{2}\ddot{X}^n(t)\Delta t^2, \]

\[ \dot{X}^{n+1}(t) = \dot{X}^n(t) + \ddot{X}^n(t)\Delta t, \]

where \( \Delta t \) is the time step.

The UDF was compiled to achieve the iteration scheme, and it was loaded into the FLUENT solver to control the movement of the valve plug. More detailed settings and information related to the grid independence study and the selection of the time step for FLUENT calculations are discussed in Section 3.3. The final grid model of the check valve is shown in Figure 9.

In the FLUENT/AMESim cosimulation, a cosimulation model for the data exchange module and the spring system was established in the AMESim solver, as shown in Figure 10. The valve plug was represented by a mass. For the exchanged variables between the two solvers, FLUENT calculated the fluid load on the valve plug based on Equations (1) and (2). This fluid load was input to AMESim to simulate the spring system based on Equation (4). The feedback on the velocity of the valve plug from AMESim was, in turn, delivered to FLUENT to update the valve plug position for the next iteration.

Figure 11 shows a comparison of the results of the displacement response and fluid load on the valve plug between the pure FLUENT simulation and FLUENT/AMESim cosimulation with the same parameters. A good agreement was observed, which demonstrated the feasibility of the coupled scheme.

4. Results

4.1. Dynamic Studies Related to Valve Timing. In the initial simulation status, the inlet and outlet pressures of the control valve were 10 and 8 MPa, respectively. Meanwhile, the valve opening was maintained at 10%. First, a steady simulation using FLUENT was conducted to obtain a stable flow field, and the flow rate at the valve outlet was 25,820 L/min. Then, the cosimulation was performed for transient calculation under the condition of setting the same time step (in AMESim and FLUENT solvers), residual accuracy (in FLUENT solver), and calculation tolerance (in AMESim solver). Additionally, the number of internal maximum iterations for a single time step was set in the FLUENT solver so that the dynamic simulation can converge at each time step. The outlet flow rate set by the PID controller was 80000 L/min. To attain the desired value of the outlet flow rate, the actuator of the control valve adjusted the opening and closing of the valve plug according to the output signal of the controller. When the flow rate at the valve outlet attained the target value, the valve opening no longer changed at a balanced position, and the cosimulation achieved convergence.

Figure 12 shows the dynamic curves of the valve plug displacement, flow rate at the valve outlet, fluid load, and driving force of the actuator on the valve plug and valve stem during the dynamic change of the valve opening. In the first 7 ms, the fluid load acting on the valve plug exhibited an oscillating and unstable form, whereas the driving force of the actuator originally increased and then decreased. And the direction of these two forces changed during this period of time. When the valve plug displacement increased to the first peak value of 40 mm, the valve opening was approximately 36%. In this process, the flow rate at the valve outlet transformed from a sharp acceleration with a large curve slope to a linear acceleration, which was related to the increase and decrease in the valve plug speed. Subsequently, the valve plug moved toward the closing direction, and the flow rate variation at the valve outlet slowed down. After exceeding 4% of the desired flow value, the outlet flow rate gradually decreased. Since the signal difference received by the PID controller was not zero, the valve plug continued to adjust continuously, and its displacement again exhibited several decaying fluctuations. Eventually, at time \( t = 0.35 \) s, the flow rate at the valve outlet was approximately 80000 L/min. The valve plug remained at a final opening of 30% with a very small fluctuation. Simultaneously, the resultant force of the valve plug was approximately zero, which indicated that the fluid load and driving force of the actuator acting on the plug tended to be stable, but there was a slight fluctuation.
greater or less than the actual adjustment force required by the valve plug, reducing the control precision of the control valve. Furthermore, oscillating fluid loads can cause vibration, noise, and other hazards on the valve plug. Therefore, the oscillating fluid loads need to be compensated on demand to reduce the valve plug oscillation and improve the stability of the control valve.

Figures 14 and 15 depict the pressure field and velocity distributions in the valve chamber during the valve plug movement. For the period of the simulation, the most representative time points were selected. The pressure distributions in Figure 14 show that the pressure of the flow channel showed a decreasing trend from the pipeline inlet to the pipeline outlet from $t = 0.001\text{ s}$ to $t = 0.6\text{ s}$. In the orifice region between the valve plug and valve seat, the flow area was limited and the pressure drop was large; the pressure drop primarily overcame the resistance of the flow channel in the control valve. As the valve plug moved from the first maximum displacement (at time $t = 0.007\text{ s}$) to the closing direction, a low-pressure region was generated on the side of the valve plug and seat. Along with the dynamic adjustment of the valve opening, this low-pressure region was continuously enlarged and moved away from the valve seat to the valve’s downstream chamber for $0.012\text{ s} < t < 0.090\text{ s}$. A part of the low-pressure region also appeared at the top of the valve housing. At $t = 0.6\text{ s}$, the valve plug has attained equilibrium, and the low-pressure region was gradually not apparent. Moreover, we observed that the upper end of the
Figure 12: Cosimulation results for the valve’s dynamic variation versus time. (a) Valve plug displacement. (b) Volume flow rate at the valve outlet. (c) Fluid load on the valve plug. (d) Driving force of the actuator.

Figure 13: Transient variation of fluid loads on plug surfaces.
**Figure 14:** Pressure field distributions at different times.

**Figure 15:** Velocity magnitude distributions at different times.
valve plug was attached to a low-pressure region, which was prone to cavitation, resulting in potential damage to the valve plug. For the pressure field of $t = 0.007$ s, the fluid pressure in the valve cavity decreased from the bottom face to the upper face of the valve plug, and the maximum pressure in the vicinity of the bottom of the valve plug was approximately 9.8 MPa. In the subsequent 5 ms, the area occupied by the high-pressure region on the lower half surface of the valve plug gradually expanded; at $t = 0.012$ s, in particular, the pressure in the partial region was approximately 10 MPa. Referring to the fluid load curve obtained by AMESim shown in Figure 13, the fluid load acting on the valve plug continuously increased within this period (from $t = 0.007$ s to $t = 0.012$ s). The main reason was that as the valve plug moved toward the closing direction during this period, the fluid load at the orifice region became smaller and the pressure fluctuation in the valve cavity increased, which lead to a larger area of the high-pressure region on the lower surface of the plug. On the other hand, the fluid load on the valve plug was obtained by integrating the surface area of the valve plug with the pressure distribution on the upper and lower surfaces of the valve plug. Thus, the increase of the pressure difference between the lower and upper surfaces of the valve plug made the fluid load increase. A similar phenomenon occurred in the period from $t = 0.007$ to $t = 0.150$ s. Therefore, the calculation results of AMESim were observed to correspond with those of FLUENT.

Figure 15 shows the sequence of the velocity magnitude contours. In the initial opening process of the valve plug for $t < 0.005$ s, the flow area at the orifice was small. The flow channels were in a strong throttling state, which caused the fluid pressure to decrease and the velocity to increase in the orifice region. As the fluid accelerated through the narrow gap of the orifice, the valve seat functioned as a throat, resulting in a jet at $t = 0.007$ s that hit the valve plug at high velocities. After $t = 0.012$ s, the valve plug moved toward the closing direction, and an asymmetric jet occurred at both sides of the valve seat. The asymmetric accelerating fluid caused the hydraulic inductance owing to inertia. It acted on the surface of the valve plug and coupled with the moving valve plug as well as the uneven pressure gradient in this region, which were the main reasons for variable and oscillating fluid loads on the valve plug (see Figure 13).

From Figure 16, the velocity of the jet region was higher than that of the nearby fluid region when the fluid flowed through the orifice. In the first 10 ms, the high-speed and low-speed streams were mixed in the nearby region, resulting in a strong shear effect, which caused the formation of vortices in the low-speed region. Owing to the structural obstruction, vortices appeared on the left side and top wall of the valve body, where the fluid stagnated and its velocity decreased. At $t = 0.012$ s, vortex production at the lip of the valve seat resulted in unsteady shedding, and a large amount of energy was removed; the phenomenon of vortex shedding changed the pressure distribution around the valve seat (see Figure 14), which caused a sudden change in the fluid load on the valve plug. Furthermore, the jet spread out further to the valve's downstream chamber, which cannot avoid the fluid path's bend, resulting in the formation of a recirculation region as shown at $t = 0.028$ s. This recirculation region expanded to the bottom of the downstream pipeline and occupied the flow channel space. Consequently, the mainstream fluid was squeezed, and its flow direction was disturbed.

4.2. Dynamic Studies Related to Different Valve Plugs. To further explore the dynamic performance of the control valve plug, we changed the shape of the valve plug. The profile curves of two other valve plugs with the flow characteristics of quick-opening and equal percentage were designed according to the calculation formula of the inherent flow characteristic [20, 21], and the cosimulations for these two valves were conducted. During the simulation process, the initial opening percentage of the valve plug, desired flow rate at valve outlet, boundary conditions, and solution parameters were unchanged. The simulation results were as follows.

Figure 17 shows the variation in fluid loads on the valve plug under different plug shapes. By comparison, we observed that in the first 1 ms, the fluid load acting on the equal percentage plug exhibited high-frequency oscillations along the horizontal axis, which was far greater than that of the fluid load acting on the other two types of valve plugs. Therefore, more load compensation was needed for the equal percentage plug in this time period. The oscillation trend of the unstable fluid loads acting on the quick-opening plug was similar to that of the linear plug, which was characterized by irregular and pulse-like fluctuations, and its fluctuation amplitude was larger than that of the linear plug. As the valve opening increased, the amplitude attenuation of the fluid load acting on the linear plug was the fastest. The oscillating fluid loads experienced by three different types of valve plugs gradually decreased and then fluctuated within a small range. When the valve plug attained equilibrium, the fluid load acting on the quick-opening plug was the largest, whereas the equal percentage plug was the smallest, which differs by approximately 1.3 times, as shown in Table 2.

Figure 18 shows the dynamic change in the flow rate at the valve outlet. As the initial displacement of the valve plug increased, the outlet flow rate for the control valve with the flow characteristic of equal percentage fluctuated significantly within 1 ms. The flow behavior was consistent with the high-frequency oscillating fluid loads (Figure 17); the larger the flow fluctuation, the greater the oscillating amplitude of the fluid load on the valve plug. This process produced an impulsive force on the valve plug, which would cause structural failure. When the outlet flow rate for the three control valves with different flow characteristics tended to the desired value, all the valve plugs were balanced to achieve the goal of controlling the flow rate behind the valve.

Figure 19 shows the flow curves when three control valves were in a stable state. We observed that the outlet flow rate for every control valve was approximately to the target value, but it had a certain deviation from the desired value, and the deviation degree was less than 0.01%, as shown in Table 3. In combination with the actual situation, the valve
could not completely maintain the desired value because of factors such as its own structure and external interference during the working process. Therefore, this result suggested that after three types of control valves with flow characteristics of equal percentage, quick-opening, and linear attained equilibrium, their outlet flow rate value with the desired value had slight differences.

On the other hand, although the valve plug was balanced, the flow rate at the valve outlet still exhibited slight fluctuations near the desired value. To compare the stability time of different valve plugs, we analyzed how fast the outlet flow rate stabilizes for each valve plug considering a deviation of 0.1% (within the engineering allowable range) as the stability criterion. In other words, when the outlet flow rate became and remained in the range of 79920–80080 L/min, it was considered to have attained stability. Figure 19 and Table 3 show that the outlet flow of the control valve with the flow characteristic of equal percentage was the first to attain the range of this zone, and its stability time was the shortest. In contrast, the control valve with the flow characteristic of quick-opening lagged behind the other two valves.

In summary, when a certain pressure is given at the inlet and outlet of the control valve (in this study, the inlet and outlet pressures were 10 and 8 MPa, respectively), the valve plug moves from a 10% opening to the balance position to control the flow behind the valve. The stability time of the control valve with the flow characteristic of equal percentage is fast, but this valve plug is subjected to sharply rapidly oscillating fluid loads in the movement process; the quick-opening valve plug takes a long time to attain stability and the fluid load acting on the plug is large in the balance position. The fluctuation amplitude of the fluid load applied to the linear valve plug is relatively low, and the damage to this valve plug is small. In practice, the selection and design
requirements for the control valve plug are different owing to different working environments. For example, when involving the high-security region on the warship, the control valve is required to respond rapidly in an emergency to achieve rapid regulation. Therefore, the valve plug’s stability time is the focus, and the valve plug with fast stability time should be selected. For control valves that work frequently, the valve plug is constantly in the process of dynamic adjustment, so we should choose the valve plug with a smaller fluctuation amplitude of the fluid load.

5. Conclusion

Based on the FSI, a cosimulation for the mechanical-electric-fluid system of the control valve was performed using the FLUENT and AMESim platforms. The interaction effect between the valve components and fluid flow was comprehensively considered. Through the coupling calculation and data exchange of the two software, the opening curve of the control valve in the dynamic adjustment process was obtained. The UDF and dynamic mesh technology were also used to solve the problem of instantaneous changes in the calculation region caused by the valve plug movement.

Compared with the pure simulations, this study performed a synchronous analysis of the unsteady flow field characteristics in the valve cavity and the dynamic response of the actuator. As the valve plug dynamically moved, a plurality of low-pressure zones of different sizes appeared in the valve chamber because of the FSI effect. The jet flow formed by the asymmetric high-speed fluid near the valve seat area was accompanied by the vortex shedding phenomenon. Moreover, we observed that the valve plug surface was subjected to unstable and oscillating fluid loads, which easily caused the valve plug to vibrate. These dynamic simulation results can give theoretical guidance for accurately compensating the additional fluid load on the valve plug.

The behavior of the control valve with different flow characteristics was compared by changing the valve plug shape. For inlet and outlet pressures of the control valve of 10 and 8 MPa, respectively, the valve plug beginning to move from the 10% opening position, the results indicated that the control valve with the flow characteristic of equal percentage was sensitive to the effect of the fluid flow. The fluctuation in the flow rate at this valve outlet and the oscillation of the fluid load acting on this valve plug were high for a small opening, exceeding those of the control valves with the flow characteristics of quick-opening and linear. And more load compensation was required for the equal percentage plug. In addition, the control valve with the flow characteristic of equal percentage was the first to attain equilibrium with the fastest stability time, whereas the control valve with the flow characteristic of quick-opening was the slowest. When the control valve was stable, the deviation between the outlet flow rate value and the desired value for every type of control valve with three different flow characteristics was negligible.
Owing to the various types of warship control valves, the operating characteristics of the valves should be completely mastered according to the actual working environments when selecting them. This study provides a new insight scheme on the dynamic analysis and performance evaluation in control valves. The results are significant to reduce the failure rate of the warship control valve, improve the control accuracy, and prolong the life of the valve.

**Data Availability**

The data used to support the findings of this study are included within the article.

**Conflicts of Interest**

The authors declare no conflicts of interest.

**Acknowledgments**

This research was funded by the National Natural Science Foundation of China (Grant nos. 51775310 and 51875316), the Key Research and Development Project of Shandong Province (Grant no. 2019GHY112052), the National Science Foundation of Shandong Province (Grant no. ZR2019MEE025), the National Key Research and Development Project of China (Grant no. 2016YFD0701104), and the Project for Scientific Research Innovation Team of Young Scholar in Colleges and Universities of Shandong Province (Grant no. 2020KJBJ001).

**References**

[1] V. Ahuja, A. Hosangadi, P. Cavallo, and R. Daines, “Analyses of transient events in complex valve and feed systems,” in *Proceedings Of the 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference And Exhibit*, pp. 1–13, Tucson, AZ, USA, July 2005.

[2] P. A. Cavallo, A. Hosangadi, and V. Ahuja, “Transient simulations of valve motion in cryogenic systems,” in *Proceedings Of the 35th AIAA Fluid Dynamics Conference And Exhibit*, pp. 1–11, Toronto, Ontario, Canada, June 2005.

[3] J. Shipman, A. Hosangadi, and V. Ahuja, “Unsteady analyses of valve systems in rocket engine testing environments,” in *Proceedings Of the 40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference And Exhibit*, pp. 1–10, Fort Lauderdale, Florida, USA, July 2004.

[4] B. K. Saha, H. Chattopadhyay, P. B. Mandal, and T. Gangopadhyay, “Dynamic simulation of a pressure regulating and shut-off valve,” *Computers & Fluids*, vol. 101, pp. 233–240, 2014.

[5] A. Beune, J. G. M. Kuerten, and M. P. C. van Heumen, “CFD analysis with fluid-structure interaction of opening high-pressure safety valves,” *Computers & Fluids*, vol. 64, no. 3, pp. 108–116, 2012.

[6] R. Morita, F. Inada, M. Morii, K. Tezuka, and Y. Tsujimoto, “CFD simulations and experiments of flow fluctuations around a steam control valve,” *Journal of Fluids Engineering*, vol. 129, no. 1, pp. 48–54, 2007.

[7] A. Misra, K. Behdian, and W. L. Cleghorn, “Self-excited vibration of a control valve due to fluid-structure interaction,” *Journal of Fluids and Structures*, vol. 16, no. 5, pp. 649–665, 2002.

[8] K. Yonezawa, R. Ogawa, K. Ogi et al., “Flow-induced vibration of a steam control valve,” *Journal of Fluids and Structures*, vol. 35, no. 776, pp. 76–88, 2012.

[9] C. Bolin and A. Engeda, “Analysis of flow-induced instability in a redesigned steam control valve,” *Applied Thermal Engineering*, vol. 83, pp. 40–47, 2015.

[10] G. Palau-Salvador, P. Gonzalez-Altozano, and J. Arrizabalaga, “Three-dimensional modeling and geometrical influence on the hydraulic performance of a control valve,” *Journal of Fluids Engineering*, vol. 130, no. 1, pp. 151–163, 2008.

[11] Y. Xie, Y. Wang, Y. Liu, F. Cao, and Y. Pan, “Unsteady analyses of a control valve due to fluid-structure coupling,” *Mathematical Problems in Engineering*, vol. 2013, Article ID 174731, 7 pages, 2013.

[12] J. Tecza, G. Chochua, and R. Moll, “Analysis of fluid-structure interaction in a steam turbine throttle valve,” in *Proceedings Of the ASME Turbo Expo Power for Land, Sea, and Air*, pp. 2329–2338, Glasgow, UK, June 2010.

[13] X. Sun, X. Li, Z. Zheng, and D. Huang, “Fluid-structure interaction analysis of a high-pressure regulating valve of a 600-MW ultra-supercritical steam turbine,” *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 229, no. 3, pp. 270–279, 2015.

[14] H. Wang, M.-j. Peng, Y.-k. Liu, S.-w. Liu, R.-y. Xu, and H. Saeed, “Remaining useful life prediction techniques of electric valves for nuclear power plants with convolution kernel and LSTM,” *Science and Technology of Nuclear Installations*, vol. 2020, pp. 1–13, 2020.

[15] R. Fadaeinedjad, M. MoaIlém, and G. Moschopoulos, “Simulation of a wind turbine with doubly fed induction generator by FAST and Simulink,” *IEEE Transactions on Energy Conversion*, vol. 23, no. 2, pp. 690–700, 2008.

[16] R. Zhang, K. P. Lam, S.-c. Yao, and Y. Zhang, “Coupled EnergyPlus and Computational Fluid Dynamics Simulation for Natural Ventilation,” *Building and Environment*, vol. 68, pp. 100–113, 2013.

[17] T. Kerh, J. J. Lee, and L. C. Welford, “Transient fluid-structure interaction in a control valve,” *Journal of Fluids Engineering*, vol. 119, no. 2, pp. 354–359, 1997.

[18] R. Messahel, C. Regan, M. H. Soulé, and C. Ruisi, “Numerical investigation of homogeneous equilibrium model and fluid-structure interaction for multiphase water flows in pipes,” *International Journal of Multiphase Flow*, vol. 98, pp. 56–66, 2018.

[19] LMS, LMS Imagine.Lab Amesim Generic Co-simulation Rev 13 User’s Guide, Roanne, France, 2013.

[20] Emerson Electric Co., *Control Valve Handbook*, Iowa, USA, 2005.

[21] B. Fitzgerald, *Control Valves for the Chemical Process Industries*, McGraw-Hill, New York, NY, USA, 1995.

[22] V. Yakhot and S. A. Orszag, “Renormalization group analysis of turbulence. I. Basic theory,” *Journal of Scientific Computing*, vol. 1, no. 1, pp. 3–51, 1986.

[23] L. Wang, J. Cui, and K. Yao, “Numerical simulation and analysis of gas flow field in serrated valve column,” *Computers & Fluids*, vol. 35, no.776, pp. 76–88, 2012.

[24] J.-Y. Qian, Z.-X. Gao, J.-K. Wang, and Z.-J. Jin, “Experimental and numerical analysis of spring stiffness on flow and valve core movement in pilot control globe valve,” *International Journal of Hydrogen Energy*, vol. 42, no. 27, pp. 17192–17201, 2017.