Numerical Analysis of Pipeline Vibration Characteristics in Large Pumping Stations

Xu Cundong¹,², Tian Junjiao¹,²*, Zhou Dongmeng³, Yao Zhipeng¹,², Gu Fengyou¹,² and Zhang Peng¹,²

¹ College of Water Conservancy, North China University of Water Resources and Electric Power, Zhengzhou, 450046, China
² Collaboration Innovation Center of Water Resources Efficient Utilization and Protection Engineering, Henan Province 450046, China
³ College of Energy and Power Engineering, Lanzhou University of Technology, Lanzhou 730050, China
* Corresponding author’s e-mail: tjunjiao@163.com

Abstract. This paper focuses on the high frequency vibration of the pipeline structure during operation of the large cascade pumping station. In order to explore its causes and inherent vibration characteristics, this paper takes #1 pressure pipeline of No.7 pump of Gansu Province Jingdian Project as the research object. The principle of additional mass method is used to establish a simplified fluid-solid coupling simulation model, and then the model is used to carry out numerical simulation test on the pipeline under different unit opening and closing conditions. After analyzing the numerical results of the simulation, the vibration mode and displacement deformation law are discovered and compared with the DASP field monitoring results. The results show that: under four representative working conditions, the vibration deformation of the pressure pipeline mainly occurs in the elbow section and the connecting section of the large and small tubes; the amplitude of the vibration mode increases with the increase of the modal order; furthermore, the water outlet pipe that is not easily vibrated in the high-order mode is gradually vibrated and deformed; in addition, the numerical simulation results are in good agreement with the field test results. This research can provide reference for the optimization design of pipeline vibration isolation and vibration reduction, and also provide a theoretical basis for the subsequent research on pipeline structural damage diagnosis and its safe operation.

1. Introduction

During the operation of a large high-lift pumping station, the pressure pipeline structure is affected by fluid force and causes a change in the flow field. At the same time, this change will also react on the structure, resulting in coupling phenomena such as inertia and damping of the elastic system. It even produces complex fluid-solid coupling dynamic vibration characteristics, thus forming a fluid-structure coupling system with feedback[1]. The long-term irregular vibration caused by the above phenomenon has a great influence on the mechanical characteristics of the pipeline structure, which will not only seriously affect the service life and safe operation of the pump station, but also endanger the safety of the project[2]. In practical engineering, the numerical methods commonly used in the analysis of pipeline vibration mainly include transfer matrix method, characteristic line method and
finite element[3]. Among them, the transfer matrix method[4] is mostly used to analyse the vibration characteristics of low-frequency bent pipe; the characteristic line method[5] can only be used for the calculation of simple pipes, because it will be distorted due to low precision when analysing complex pipes; the finite element method[6] has the advantages of high calculation accuracy and wide application range, so it can simulate various complicated pipeline structures well. Therefore, the finite element method is the most commonly method for analysing the vibration characteristics of conveying pipeline structure. At present, research on the vibration characteristics of pressure pipelines has yielded fruitful results[7]-[8]. However, most of the research is limited to the simple tubular structure of the non-twisted tube or the short straight type, and there are few studies on effective solid-liquid physics for dynamic characteristics of complex tubular structures such as manifolds or variable diameters. At the same time, it is also lack of systematic analysis of the causes and vibration characteristics of pipeline vibration under different unit opening and closing conditions. Therefore, this paper selects #1 pressure pipeline of No.7 pump of Gansu Province Jingdian Project as the research object. The additional water mass method is used to establish a simplified fluid-solid coupling model based on ANSYS, and then the finite element simulation calculation under different working conditions is carried out to simulate the interaction between fluid and structure. Finally, the vibration characteristics of the high-lift pumping station pressure pipeline are revealed. Furthermore, the numerical analysis results are compared with the modal parameters of the on-site DASP monitoring test results to obtain accurate design parameters. This provides technical support and theoretical basis for the vibration reduction optimization measures of the pump station pressure pipeline and the selection of safe operating conditions.

2. Basic theory of model calculation

2.1. Equation of fluid-structure interaction

Under the working condition, the dynamic problem of the pressure pipeline is essentially the fluid-solid coupling problem of the interaction between the water body and the pipeline structure, and its influence on the vibration characteristics of the structure cannot be ignored. According to the characteristics of common fluid-solid coupling problems, this paper uses finite element theory to establish a finite element model of pressure pipeline.

The fluid-solid coupling should satisfy the most basic conservation principle, that is, the fluid and solid at the interface of fluid-solid coupling should satisfy the equality or conservation of variables such as stress ($\tau$), displacement ($d$), temperature ($T$), and heat flux ($q$)[9].

The continuous conditions of the flow-solid interface are:

$$
\begin{align*}
\tau_f \cdot n_f &= \tau_s \cdot n_s \\
d_f &= d_s \\
q_f &= q_s \\
T_f &= T_s
\end{align*}
$$

(1)

Where, $f$ is a fluid; $s$ is a solid.

2.2. Equation of control for solid structure

The conservation equation of the solid structure is derived according to Newton's second law:

$$
\rho_s d_s \frac{\partial v_s}{\partial t} = \nabla \sigma_s + f_s
$$

(2)

Thermal deformation caused by temperature difference, $f_T$:

$$
f_T = \alpha_t \nabla T
$$

(3)
2019 International Conference on Civil and Hydraulic Engineering

IOP Conf. Series: Earth and Environmental Science 304 (2019) 042072
doi:10.1088/1755-1315/304/4/042072

Where, \( \rho_0 \) is the solid density; \( d_0 \) is the local acceleration vector for the solid domain; \( \sigma_0 \) is the Cauchy stress tensor; \( f_0 \) is the volumetric force vector; \( \alpha_T \) is the coefficient of thermal expansion; \( \nabla T \) is the temperature difference.

2.3. Additional water mass method

The additional water mass method is to treat the water body pressure as part of the rigid body mass of the pressure pipe, and apply the pressure to the finite element mesh node on the inner wall of the pipe to form an integral mass. It is a simplified dynamic analysis method that considers the effect of water on the structure of the pipeline[10]. In this paper, the simplified fluid-solid coupling model is used to attach the quality of the water to the grid nodes for calculation, which approximates the vibration effect of the water body on the pipe wall structure.

For the application of the additional mass method in the pipeline structure[11], the differential equation of motion is:

\[
M_0 \ddot{u} + C_0 \dot{u} + K_0 u = F_0
\]  

(4)

Where, \( M_0 \), \( C_0 \), \( K_0 \) are the overall mass matrix, damping matrix, and stiffness matrix of the pipe structure, respectively; \( u, \dot{u}, \ddot{u} \) are the displacement, velocity, and acceleration vectors of the pipe structure, respectively; \( F_0 \) is the external load matrix of the pipe as a whole.

The finite element vibration equation considering the additional mass of the pipe water body is:

\[
(M_0 + M_a) \ddot{u} + (C_0 + C_a) \dot{u} + (K_0 + K_a) u = F_0
\]  

(5)

Where, \( M_a \), \( C_a \), \( K_a \) are additional mass matrices, additional damping and additional stiffness matrices, all of which are produced by the action of water bodies.

If the water body is regarded as an ideal fluid, the damping force and the effect of the water body on the structural rigidity of the pipe can be neglected, and the formula (5) can be simplified as:

\[
(M_0 + M_a) \ddot{u} + C_0 \dot{u} + K_0 u = F_0
\]  

(6)

3. Project case

3.1. Project Overview

The Jingtaichuan electric power irrigation project in Gansu Province (hereinafter referred to as "Jingdian Project") is one of the large-scale high-lift steps-level irrigation projects in China. There are 46 large and small pumping stations, and the highest total lifting head of the project is 713 m. The pressure pipelines of most pumping stations are arranged in the form of multi-machine single tubes. Therefore, in actual operation, there are serious water flow excitation vibration problems in the pressure pipelines of many pumping stations. The pressure pipe of #1 pressure pipeline of No.7 pump of the Jingdian Phase II is composed of three inlet pipes and one outlet pipe, and the schematic diagram of the plane structure is shown in Figure 1.

![Figure 1. The #1 pressure pipeline structure of No.7 pump of pump station.](image)
this paper establishes the ANSYS finite element model with "1" pressure pipeline of this pump station as the research object, revealing the vibration characteristics of the pressure pipeline.

3.2. Model establishment

The size parameters of each part of the "1" pressure pipeline of No.7 pump of the Jingdian Phase II is shown in Table 1 below. The head and flow parameters of each water inlet branch pipe are shown in Table 2.

| Pipeline component | Size | 
|--------------------|------| 
| ①                 | inside diameter 1.70 m, thickness 14 mm | 
| ② (gradient segment 1) | left inside diameter 1.20 m, thickness 12 mm, right inside diameter 1.70 m, thickness 14 mm | 
| ③                 | inside diameter 1.20 m, thickness 12 mm | 
| ④                 | inside diameter 0.80 m, thickness 12 mm | 
| ⑤                 | turning site inside external diameter 2.40 m; inside diameter 0.80 m, shaft radius 1.6 m; thickness 12 mm | 
| ⑥ (gradient segment 2) | bottom inside diameter 0.50 m; upper inside diameter 0.80 m; thickness 12 mm | 
| ⑦                 | inside diameter 0.50 m; thickness 12 mm | 

Table 2. Summary of pump unit parameters

| Pump unit number | Model of pump | Head of delivery/ m | Design flow rate / m³/s | 
|-----------------|--------------|---------------------|------------------------|
| #1              | 24SH-19      | 32                  | 32680                  | 
| #2              | 1200S-32     | 32                  | 10800                  | 
| #3              | 1200S-32     | 32                  | 10800                  | 

The material of the pipe is steel, which is simplified into a homogeneous material. The density is 7.85 g/cm³, the modulus of elasticity is 2.06×10⁵ MPa, and the Poisson's ratio is 0.25. The pipe structure of the additional mass model is established by the SOLID45 unit, and the coupling of the water body and the pipe is realized by setting the mass21 unit at the pipe wall mesh node. The model is segmented using a hexahedral mesh. Under the requirement of satisfying the grid independence test error, 8520 simulation calculation grids are finally obtained through trial calculation. The model meshing results are shown in Figure 2.

3.3. Boundary constraint

The selection of the boundary constraints determines the reliability of the simulation results of the pipeline structure. Therefore, it is necessary to set the various displacement constraints, fluid pressures, etc. applied to the components in the modal analysis reasonably[9].

In the pipeline simulation, according to the pipeline operating conditions, the boundary constraints imposed are mainly as follows: First, a full constraint is imposed on the outer surface node in the finite element model of the water inlet pipe; second, the full constraint is imposed on the outer surface node of the main outlet pipe buried in the wall segment; third, applying a downstream displacement constraint at the disconnection of the manifold; fourth, applying radial displacement constraint on the
First, #1, #2, #3, #4 piers; fifth, the applied gravity acceleration is 9800 mm/s². The model after defining the boundary constraints is shown in Figure 3.

![Figure 2. Geometric model of the #1 pressure pipeline](image1)

![Figure 3. Constraints of model boundaries](image2)

### 3.4. Calculation project status

In order to accurately and effectively simulate the vibration characteristics of #1 pressure pipeline, this paper combines the four representative working conditions listed in Table 3 with the actual operation conditions and actual operating conditions of the pumping station. Then the paper simulates the working mode of the pressure pipeline under different working conditions and analyses the deformation, and compares and analyses the real-time DASP monitoring results.

| Working condition | #1 Pump machine | #2 Pump machine | #3 Pump machine |
|-------------------|-----------------|-----------------|-----------------|
| 1                 | Closing         | Opening         | Closing         |
| 2                 | Closing         | Opening         | Opening         |
| 3                 | Opening         | Opening         | Opening         |
| 4                 | Closing         | Opening         | Opening         |

4. Modal calculation and DASP on-site monitoring verification

When the modal simulation calculation is carried out on the #1 pressure pipeline, the initial force generated by the water pressure in the pipeline and the influence of the water pressure on the stiffness matrix are first calculated by static force. And then the modal analysis of the pressure pipeline is carried out to obtain the former 6-order modal shape of the pressure pipeline under the four schemes of units opening and closing conditions. While the modal vibration shape pattern is a figure that is magnified 50 times by ANSYS, as shown in Figure 4.
4.1. Modal analysis
By analysing the calculation results of the former 6-order modal vibration of working conditions 1~4, the following conclusions can be obtained. Firstly, for the working condition 1: After the #2 pump is turned on, the #2 inlet pipe and the #2 outlet pipe are more susceptible to vibration in the low-order mode, and the vibration deformation is large; conversely, the #2 outlet pipe and the #2, #3, #4 pier are more susceptible to vibration in the higher-order mode, and the vibration deformation is larger, while the inlet pipe is less susceptible to vibration; the vibrations of the #1 and #3 inlet pipes are always small regardless of the low-order mode or the high-order mode. It can be seen that the weakest part of the pressure pipe that is most susceptible to vibration and easy to generate vibration deformation is the #2 outlet pipe and the #2, #4 pier. Secondly, for the working condition 2: after the #3 pump is turned on, the overall vibration mode of the pressure pipe changes greatly; the most prone to vibration and vibration deformation of the pressure pipe is at the #3 inlet and outlet pipes. It can be seen that the weakest parts of the pressure pipe that are most susceptible to vibration are the #2, #3 inlet and outlet pipes. Thirdly, for the working condition 3: after the three units are simultaneously opened, the overall vibration mode of the pressure pipe changes; the weakest position where the pressure pipe is most susceptible to vibration deformation is the intersection of the #3 pump inlet and outlet pipe as well as the main branch pipe (Figure 4). Fourthly, for the working condition 4: the vibration characteristics of working condition 4 are similar to those of working condition 1, except that after the #3 pump is turned on, the overall vibration mode of the pressure pipe changes greatly. It can be seen that the opening or not of the #3 pump has a great influence on the vibration of the overall pipe. The weak position where the pressure pipe is most prone to vibration deformation is the intersection of the main branch pipe, the #3 inlet and outlet pipe and the #3 pier.

4.2. Displacement analysis
Table 4 shows the results of the modal displacement deformation analysis of the former 6-order mode under working conditions 1~4.

The conclusion is as follows. Firstly, for the working condition 1: the #2 inlet pipe and outlet pipe are more susceptible to vibration in low-order mode, and vibration deformation is larger; conversely, the #2 outlet pipe and the #1, #2 pier are more susceptible to vibration in the higher-order mode, and the vibration deformation is larger, while the inlet pipe is less susceptible to vibration; the vibrations of the #1 and #3 inlet pipes are always small regardless of the low-order mode or the high-order mode. It can be seen that the weakest part of the pressure pipe that is most susceptible to vibration and easy to generate vibration deformation is the #2 outlet pipe and the #1, #2 pier. Secondly, for the working condition 2: the #2, #3 inlet pipe, the elbow pipe and the main branch pipe intersection are susceptible to vibration in the low-order mode; while the #1 pipe (bent pipe), #4 pier and the main branch pipe intersection are susceptible to vibration in the high-order mode. Therefore, the weakest part of the pressure pipe that is most susceptible to vibration and vibration deformation is the #2, #3 inlets and outlet pipe, the main branch pipe intersection and the elbow. Thirdly, for the working condition 3: the inlet pipe and the elbow of each unit are easy to vibrate in the low-order mode; the main branch pipe junction and the #3 outlet pipe are easy to vibrate in the high-order mode; the #3 outlet pipe amplitude is significantly larger than the inlet pipe. Fourthly, for the working condition 4: the junction of the inlet pipe and the main branch pipe is easy to vibrate in the low-order mode, and the amplitude of the #3 inlet pipe is greater than the #1, #2 inlet pipe; the #3 outlet pipe and the #1, #2 pier are susceptible to vibration in the high-order mode; the vibrations of the #1 inlet pipes is always small regardless of the
low-order mode or the high-order mode. It can be seen that the weakest part of the pressure pipe that is most susceptible to vibration and vibration deformation is the main branch pipe intersection and the #3 inlet and outlet pipe.

Table 4. Displacement values of the former 6-order modal of the design project

| Order | Results of ANSYS | Results of DASP identification | Identification error |
|-------|------------------|-------------------------------|----------------------|
|       | Freq/Hz          | Damping ratio/%               | Freq/Hz              | Damping ratio/%               | Freq/Hz              | Damping ratio/%               |
| 1     | 9.42             | 2.83                          | 9.49                 | 2.82                          | 0.74                 | 0.35                          |
| 2     | 14.75            | 3.15                          | 15.41                | 2.92                          | 4.47                 | 7.30                          |
| 3     | 21.12            | 4.59                          | 21.77                | 4.96                          | 3.08                 | 8.28                          |
| 4     | 22.53            | 4.63                          | 22.25                | 4.55                          | 1.24                 | 1.94                          |
| 5     | 27.52            | 5.07                          | 28.74                | 5.24                          | 4.43                 | 3.35                          |
| 6     | 32.46            | 5.33                          | 32.81                | 5.48                          | 1.08                 | 2.81                          |

4.3. Displacement analysis
DASP software is a system with large-capacity data acquisition and signal processing for analysis and testing of noise, vibration, shock and information processing. Therefore, for working conditions 1~4, this paper selects the upgraded version of DASP V10 virtual vibration test system to monitor the target pipeline structure, obtain the vibration signal of each monitoring point under working condition, and then use DASP system software to analyse and process the signal, so as to extract structural modal parameters. The layout of measuring points 1~22 and the spacing of measuring points are shown in Figure 5. Among them, the sensors numbered with odd numbers are level sensors, and the sensors numbered with even numbers are vertical sensors.

Figure 5. Site diagram of the measuring point of the vibration pick-up

In this paper, the author selects the start-up combination of the pumping station under typical working conditions 3 for model verification. The former 6-order modal frequencies of the pipeline finite element model under this condition and the corresponding DASP field monitoring results are shown in Table 5. From Table 5, the highest frequency error is 4.47%, and the damping ratio error does not exceed 8.28%. In summary, the numerical calculation results agree with the field test results.

Table 5. Comparison between numerical simulation and DASP-testing under working condition 3

| Order | Results of ANSYS | Results of DASP identification | Identification error |
|-------|------------------|-------------------------------|----------------------|
|       | Freq/Hz          | Damping ratio/%               | Freq/Hz              | Damping ratio/%               | Freq/Hz              | Damping ratio/%               |
| 1     | 9.42             | 2.83                          | 9.49                 | 2.82                          | 0.74                 | 0.35                          |
| 2     | 14.75            | 3.15                          | 15.41                | 2.92                          | 4.47                 | 7.30                          |
| 3     | 21.12            | 4.59                          | 21.77                | 4.96                          | 3.08                 | 8.28                          |
| 4     | 22.53            | 4.63                          | 22.25                | 4.55                          | 1.24                 | 1.94                          |
| 5     | 27.52            | 5.07                          | 28.74                | 5.24                          | 4.43                 | 3.35                          |
| 6     | 32.46            | 5.33                          | 32.81                | 5.48                          | 1.08                 | 2.81                          |
5. Conclusion
By comparing and contrasting the previous case analysis and the model calculation results, it seems to reach a consensus of high frequency vibration of the pipeline structure during operation of the large cascade pumping station, and its causes and inherent vibration characteristics are explored. The conclusions are as follows:

a. On the basis of considering the fluid-solid coupling effect, the vibration modes and displacements of the #1 pressure pipeline structure of No.7 pump of Jingdian Pumping Station are analysed in this paper. It is found that #3 inlet pipe, #3 outlet pipe, main branch pipe intersection and bend pipe are more likely to cause vibration under normal working condition, while the #2 inlet pipe and outlet pipe are slightly vibrated, and #1 inlet pipe is not easy to vibrate; the maximum displacement deformation of the model is 20.847 mm, and the most prone to vibrational deformation is the intersection of the bent pipe, the branch inlet and the main branch.

b. The vibration mode frequency calculated by ANSYS is consistent with the extraction result of the on-site DASP monitoring test. Together, its fundamental frequency error is only 0.74%, the maximum frequency error is less than 5%, and the damping ratio maximum error is less than 10%, which also meets the engineering accuracy requirements. Therefore, the ANSYS finite element simulation model, which is established by using the principle of additional water mass method, to analyse the vibration characteristics of the pumping station water pipeline structure to obtain reasonable results. This research can provide reference for the optimization design of pipeline vibration isolation and vibration reduction, which are in the pumping station of the same type of multi-machine single-tube arrangement, and also provide a theoretical basis for the subsequent research on pipeline structural damage diagnosis and its safe operation.

References
[1] Li S.J. (2015) Dynamic Analysis of Fluid-Structure Interaction of Pipe Systems Conveying Fluid. Harbin Engineering University. Harbin.
[2] Li H.K., Yang M., Chen L. et al. (2014) Operational Safety Evaluation of Sluice Pier, Measurement and Diagnosis, 34(5):938-946.
[3] Bao J.H., Pan Z.W., Xu P.M. et al. (2010) Vibration analysis on a pipe system of pumps based on fluid-structure interaction. Journal of Drainage and Irrigation Machinery Engineering, 28(4):349-353.
[4] Zhang Z.Y., Shen R.Y. (2001) The modal analysis of the liquid-filled pipe system. Chinese Journal of Solid Mechanics, 22(2):143-149.
[5] Yang C, Fan S.J. (2011) Influence of pipe parameters on fluid-structure coupled vibration of a fluid-conveying pipe. Journal of Vibration and Shock, 30(7):210-213.
[6] Ye H.L., Shao P.Z. (2015) Dynamic Analysis and Parameters' Influences on Fluid-structure Interaction in a Fluid-filled Pipes System. Journal of Beijing University of Technology, (2):167-173.
[7] Keramat A, Tijsseling A S, Hou Q, et al.(2012) Fluid–structure interaction with pipe-wall viscoelasticity during water hammer. Journal of Fluids & Structures, 28(1):434-455.
[8] Zhou Z.J., He X. (2017) Simulation for fluid-solid interaction characteristics of hydraulic pipes with different curvatures. Journal of Vibration and Shock, 36(5):214-220.
[9] Liu Y.H. (2001) Fluid-solid dynamic coupling theory and its application in Hydraulic Engineering. Xi'an Jiaotong University, Xi'an.
[10] Ni B.Y., Sun S.L. (2012) Influence of additive mass variation of a missile during its entering into water. Journal of Vibration and Shock, 31(14):171-176.
[11] Zhang J.W., Wang T. (2017) Analysis of water pipeline vibration characteristics in cascade pumping station based on fluid-solid coupling interaction. Transactions of the Chinese Society for Agricultural Machinery, 48(3):134-140.