Academic Journal of Science and Technology
ISSN: 2771-3032 | Vol. 1, No. 3, 2022

Research on Structural Optimization Design of Plate Gate

Yulu Wei[1, 2, 3, 4], Siqi Liu[1, 2, 3, 4]

[1]Institute of Land Engineering and Technology, Shaanxi Provincial Land Engineering Construction Group Co., Ltd., Xi'an 710075, China
[2]Shaanxi Engineering Research Center of Land Consolidation, Xi'an 710075, China
[3]Key Laboratory of Degraded and Unused Land Consolidation Engineering, Ministry of Natural Resources, Xi'an 710075, China
[4]Shaanxi Provincial Land Engineering Construction Group Co., Ltd., Xi'an 710075, China
[5]Land Engineering Technology Innovation Center, Ministry of Natural Resources, Xi'an 710075, China

Abstract: The bulkhead gate of simple structure, easy fabrication, easy maintenance, safe and reliable degree is high, widely used in water conservancy and hydropower engineering, have stronger effects of spatial structure, this paper adopts three-dimensional finite element stress analysis, modeling and research of the relationship between the structure parameters and the structure performance, and then through the ANSYS software to optimize design. The analysis results show that the optimized gate has lighter weight and saves more materials under the condition of satisfying the strength and stiffness constraints.

Keywords: Plate steel gate, Finite element, ANSYS modeling, Static analysis, The optimization design.

1. Introduction

Plate steel gate is the earliest and most widely used in water conservancy and hydropower projects around the world. Plate steel gate is a complex spatial structure system, and the traditional plane structure system cannot fully and accurately reflect the overall structural characteristics of the gate [1]. Therefore, an analysis method is needed that can fully and accurately reflect the coordination of the whole structure and each part, and can carry out realistic simulation of load and boundary constraints, so as to improve the credibility of the analysis results. On this issue, finite element method can be used for analysis and more accurate results can be obtained [2]. At the same time, in order to make the gate have a better stress condition, and considering the economic benefit, save materials, reduce the cost, it is necessary to optimize the design of the gate.

Many people have done researches on hydraulic gate, such as Wang Haoqiang [3] who studied the static and dynamic problems of flat gate, and Zhang Chunhua [4] who discussed high head gate. This paper is the use of finite element analysis method, the use of ANSYS software for static analysis of the gate plate and beam structure, and the gate beam section as a design variable, optimize the design.

2. Basic Principle of Finite Element Analysis of Flat Steel Gate

2.1. Finite Element Hypothesis of Plate Steel Gate

Plate steel gate is a complete spatial structure system, the external force and load are borne by all the gate components, need to use space finite element method to calculate. The plate steel gate can be regarded as a structural system composed of plate and beam. During the unit division and finite element analysis of the gate, the adjacent plate element and beam element have the same structural nodes [5], and some assumptions need to be made [6] (the XY plane is the middle plane of the plate, and the Z direction is perpendicular to the middle plane, as shown in fig. 1). (1) Assume that the gate material is continuous, uniform, elastic and isotropic; (2) Assuming that the deformation of the gate is not large, Kirchhoff postulated can be used for analysis. The analysis process accords with that the normal section of the structure has not changed before and after the deformation, which is $\gamma_{zx} = 0$, $\gamma_{zy} = 0$.(3)When the gate plate is bent by force, the middle surface keeps the original shape unchanged, $u = v = 0$.(4)The thickness of thin plate remains unchanged after deformation, $\varepsilon_z = \frac{\partial_w}{\partial_z} = 0$.(5)Each horizontal layer in the thin plate is not extruded, $\sigma_z = 0$.

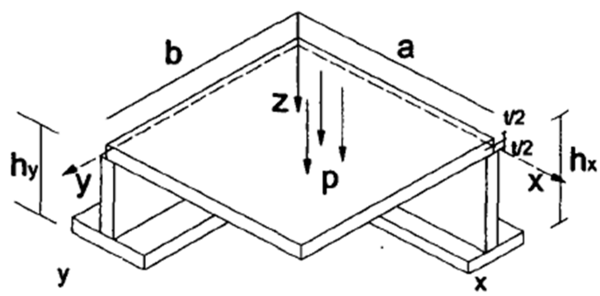

Figure 1. Composite structure of slab and beam

2.2. Finite Element Analysis and Optimization Design Based on ANSYS Software

Firstly, the finite element model is created and the mesh is divided, including two modeling methods of entity modeling, direct generation, and two meshing methods of free meshing and mapping meshing. Secondly, load is applied, constraint conditions are set and solved. When the calculation is finally completed, the processing results can be viewed through the post-processing of ANSYS [7].

Optimization design is to continuously search the feasible design scheme through the computer, modify and optimize within the specified range, until the optimal solution is obtained. ANSYS optimization design has two ways, one is batch processing method, can use the command to enter the entire optimization file, save time, is conducive to large complex structures; The other is interactive through the GUI, where you can see the optimized results in real time and the interaction is more flexible.

3. Three-dimensional Modeling and Static Analysis of Plate Steel Gate

3.1. 3d Modeling of Gate

The gate is a submerged flat steel gate with a thickness of 0.025m and a height of 5.4m. The load span is 5.5m. The calculated span is 4.94+0.2×2=5.34m. According to the structural arrangement of the gate, the X-axis is set along the transverse axis of the main beam and the secondary beam; Y axis along the side beam, vertical direction of the secondary beam; Z axis is the direction of water flow; Zero is set at the bottom of the gate. Only hydrostatic pressure and gate dead weight are considered for load. Constraint at the bottom of the gate with a vertical connecting rod constraint, that is, Y constraint; There is Z direction constraint along the flow direction at the sliding blocks on the two sides of the gate. There is an X-direction constraint in the uppermost center of the gate to prevent the gate from moving in the X-direction.

On this basis, ANSYS software is used to establish the gate model, as shown in Figure 2. Shell element SHELL181 and beam element BEAM188 are mainly used.

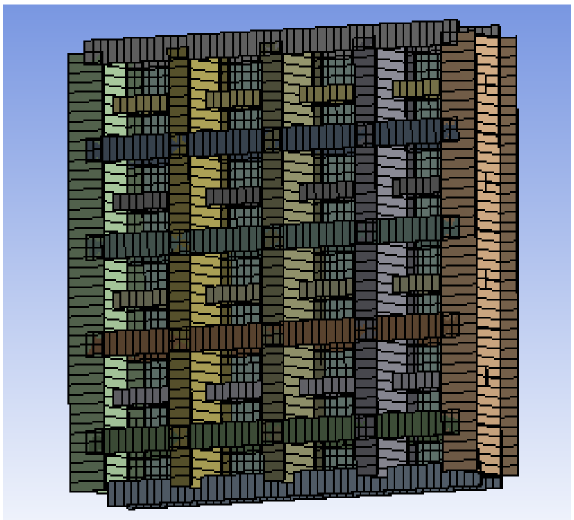

Figure 2. Finite element model diagram of gate

3.2. Static Analysis

3.2.1. Stress Analysis

As can be seen from the cloud diagram of the maximum principal stress of the panel (figure 3), the stress of the panel mainly occurs at the contact surface of each beam, while the maximum principal stress occurs at the contact surface of the fifth main beam. Due to hydrostatic pressure and stress concentration, the maximum principal stress occurs near the lower slider constraint. The maximum principal stress is 57.22MPa, far less than the allowable stress 160MPa.

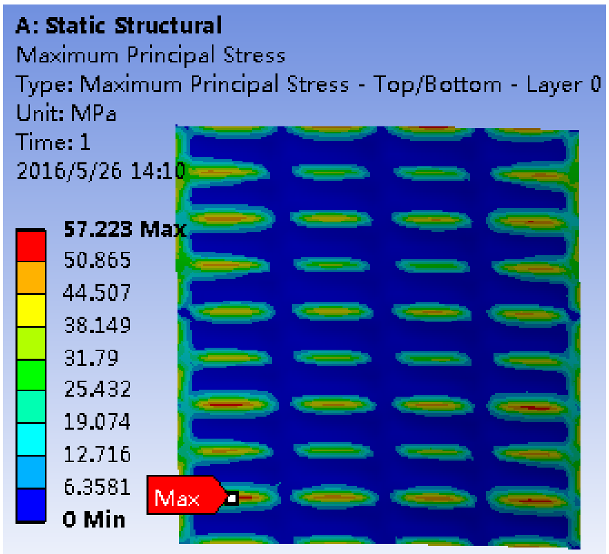

Figure 3. Cloud diagram of maximum principal stress of panel

It can be seen from Table 1 that the maximum normal stress of the main beam and secondary beam increases from top to bottom, and the maximum normal stress occurs in the middle of the fifth main beam and the fourth secondary beam, which are 143.49mpa and 63.45mpa respectively. This is mainly caused by the gradual increase of hydrostatic pressure with the increase of water depth. In addition, the maximum normal stress of the second main beam is abrupt compared with the rest of the main beam, which is due to the stress concentration caused by the sliding block constraint. The stress of the secondary beam is symmetrically distributed, and the maximum stress of each beam occurs on both sides of the middle position, which is caused by the connection between the middle position and the vertical secondary beam.

Table 1. Maximum normal stress of main and secondary beams (Unit: MPa)

Stress	Main beams					Secondary beams			
	First main beams	Second main beams	Third main beams	Fourth main beams	Fifth main beams	First secondary beams	Second secondary beams	Third secondary beams	Fourth secondary beams
Maximum normal stress	90.66	124.13	125.5	139.91	143.49	34.20	39.48	42.44	63.45

3.2.2. Deformation Analysis

It can be seen from the panel maximum displacement cloud diagram (4) that the maximum displacement of the panel is 4.04mm, less than the maximum allowable displacement in the specification: 7.12mm for the main beam and 5.34mm for the secondary beam, which meet the stiffness constraints. The maximum displacement occurs in the middle of the contact surface with the fifth main beam and the bottom beam.

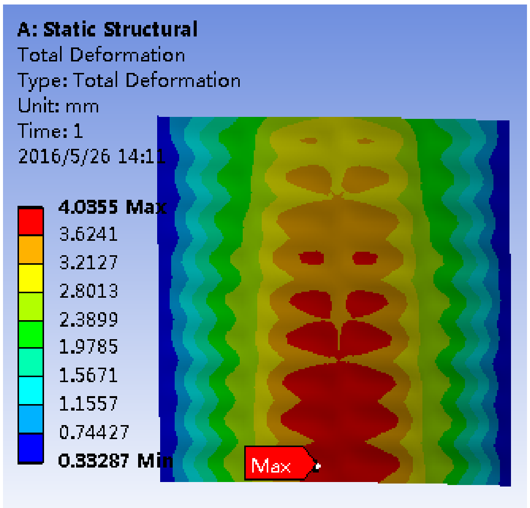

Figure 4. Panel maximum displacement cloud image

The maximum displacements of the main beam and the secondary beam in the downstream direction (Z direction) are shown in Table 2. From top to bottom, the maximum displacements increase gradually and occur in the middle position of the fifth main beam and the middle position of the fourth beam, which are 3.7mm and 3.74mm respectively, meeting the stiffness requirements.

4. Structural Optimization Design of Flat Steel Gate

4.1. Optimization Parameters and Constraints

According to the statics analysis, the main force and deformation of the gate occur on the beam, and the design variables are the geometric dimensions of main beam I1, secondary beam I5, and bottom beam I4, as shown in Fig.5. The maximum stress and deflection of each beam structure are selected as state variables. In the optimization project, the constraint conditions of strength, stiffness and geometry are satisfied, and finally the objective function of the minimum gate weight and the minimum gate volume is achieved.

Table 2. The maximum displacement of main beam and secondary beam in the Z direction (Unit: mm)

Stress	Main beams					Secondary beams			
	First main beams	Second main beams	First main beams	Second main beams	First main beams	Second main beams	First main beams	Second main beams	First main beams
Maximum displacement	2.83	3.22	3.47	3.62	3.73	3.10	3.34	3.61	3.74

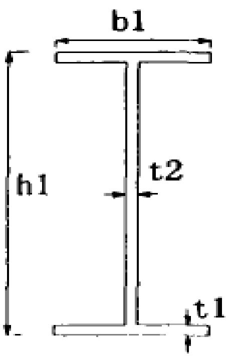

Figure 5. Section drawing of main beam

4.2. Optimization Process and Result Analysis

4.2.1. Optimize the Basic Process

The basic optimization process is as follows: ① Set input parameters and response parameters and parameterize them; (2) Define the value range of design variables and generate sampling points; (3) Calculate the sampling points and fit the response surface through the sampling points; (4) Set the objective function, that is, related constraints; (5) 2000 design points were calculated on the response surface, and the optimal three groups of parameters were selected according to the constraints. (6) Select one of the three optimal parameters for analysis.

4.2.2. Result Analysis

After the above basic optimization process, a group of data is selected from the results as the optimal solution of this optimization, as shown in Table 3. The total volume of the gate decreased from 2.83m^3 to 2.31m^3, and the total steel consumption ratio was saved by 18.37%.

Table 3. The value range and optimal value of design variables (Unit: m)

Design variables	b1	t1	t2	h1
Value range	0.1-0.5	0.01-0.03	0.01-0.03	0.5-1.5
Optimal value	0.22	0.012	0.013	1.02
Design variables	b4	t7	t8	h4
Value range	0.1-0.5	0.01-0.03	0.01-0.03	0.3-0.8
Optimal value	0.20	0.025	0.013	0.61
Design variables	b5	t9	t10	h5
Value range	0.1-0.2	0.005-0.025	0.005-0.025	0.2-0.4
Optimal value	0.15	0.005	0.008	0.33

Based on the above parameters, the maximum principal stress of the optimized front panel is 57.22Mpa, and the maximum principal stress of the optimized front panel becomes 63.25Mpa, which meets the strength requirements. The maximum normal stress of each beam before and after optimization is as follows:

Table 4 Maximum normal stress of each beam before and after optimization (Unit: MPa)

Before and after optimization	Maximum normal stress of main beam	Maximum normal stress of secondary beam	Maximum normal stress of side beam	Maximum normal stress of bottom beam	Maximum normal stress of vertical secondary beam
Before optimization	143.49	63.45	25.84	63.45	38.
After optimization	160.00	78.62	34.59	85.31	42.47

It can be clearly seen from the table that the maximum normal stress of the main beam reaches the allowable strength after optimization. Due to the influence of water load, the lower pressure is large, and the maximum normal stress occurs in the middle of the fifth main beam. After optimization, the maximum use of the strength of the main beam, greatly increasing the actual utilization rate of steel. After optimization, the maximum normal stress of secondary beam, bottom beam, vertical secondary beam and side beam also increases to varying degrees, all of which meet the requirements of allowable strength.

5. Summary

Gate to the three dimensional space structure, the use of the space finite element method (fem) can be better for the gate, to simulate the boundary conditions and constraints, static optimization design using ANSYS software, the section size of beam as design variables, lightweight optimization, gate more make full use of steel strength, save nearly one 5 of the steel consumption, The optimization results are reasonable and have certain engineering application value. In this paper, only the static load is considered. If the dynamic load is considered, the natural vibration characteristics should be analyzed after considering the fluid-structure coupling effect.

References

[1] Cai Zhengkun, Qu Erren. Sluices and hoists[M].Beijing: Water&Power Press, 1988:10-100.

[2] Fan Chongren, Zhou Shizhi, Chengjizu. Hydraulic steel structure[M].Beijing: Water Resources and Electric Power Press，1986:35-58.

[3] Wang Haoqiang. Analysis and Optimal Design on Static Load and Dynamic Property of Plane Steel Gate[D]. Hohai University,2006

[4] Jia Zhen.Parametric Finite Element Analysis and Structure Lightweight Design on Plate Gate[D]. North China University of Technology,2015

[5] Li Beibei, Zhao Zhihong.Application of Ansys Software to Engineering Design[J] Zhejiang Hydrotechnics,2003(05):4-6.

[6] Zhang Wenjuan. Three Dimensional Finite Element Analysis of Radial Steel Gate [D]. Guangxi University,2002.

[7] Zhang Liaojun, Chen Wenlong, Ye Huashun. 3D-FEM analysis Hengren fixed wheel plane gate [J]. Water Conservancy & Electric Power Machinery,2003(01):12-15+46.