Calculation of and Analysis on Inlet / Outlet Structure of Hydropower Station Based on Ansys

Du Wu¹, Zhao Tian¹, Mengjie Zhang¹, Yu Li¹, Zhiyu Li¹ and Sheng Zhang¹

¹Power China Huadong Engineering Corporation, Hangzhou, Zhejiang,310014, China
Corresponding author’s e-mail: wudu024@163.com

Abstract: Inlet / outlet of pumped storage power station is affected by bidirectional flow. Head is high and the force situation is more complicated. Simulation calculation based on finite element software ansys can simulate the force situation of inlet / outlet under different working conditions and analyze the distribution of stress in the stress cloud. So we can calculate structural reinforcement of inlet / outlet according to stress cloud. This calculation method can meet the use requirements of the structure.

1. Introduction
The stress condition of inlet / outlet of pumped storage power station is more complicated. The main performance is the following two points:

(1) Under the inflow and outflow conditions, it is affected by the incoming flow in different directions, and the hydraulic condition is more complicated. So it is easy to produce reverse flow rate under the outflow condition near the anti-vortex beam. This situation can cause the inlet and outlet to vibrate and can damage the structure of the inlet and outlet.

(2) At the normal water storage level, the head of the still water received by the inlet and outlet is high. Affected by external factors such as the flow rate of the grid, the distance between the inlet and outlet holes is large. Affected by the above two points, the local stress concentration phenomenon at the inlet and outlet is more obvious. Therefore, it is necessary to conduct a reasonable analysis of the force situation about the inlet and outlet.

Ansys is a large-scale general finite element analysis (FEA) software, which can share and exchange data with most computer aided designs. Using ansys for simulation calculation can reduce the design cost, shorten the design cycle and increase the reliability of the project. Using optimized design schemes can reduce material consumption and costs.

2. Establishment of finite element model
Creating the model is based on the actual size of the inlet and outlet. This calculation example uses planar finite element solution. In this calculation, we select two calculated sections for the anti-vortex beam section(1-1) and the adjustment section(2-2), whose model is according to the shape of the cross-section, as shown in Figure 1. The calculation model is drawn in CAD. The surrounding rock around the structural model needs to be established according to the actual situation. The range of surrounding rock in the model is selected to be 5 times the length of the longest side of the structure[1]. We can import the built calculation model into ansys, as shown in Figure 2 and Figure 3.
3. Meshing grid and applying constraint

For the inlet and outlet structure and surrounding rock, we need to adopt the same way to mesh. Meshing is done in ansys software. One unit node of concrete is according to 0.3 meter. The joint surface of the surrounding rock and concrete adopts the common node method, so the grid of the shared boundary between the surrounding rock and concrete is still 0.3 meter. The grid far away from the concrete area adopts the gradient method and spacing ratio is for 0.125/8. So the surrounding rock grid closed to the concrete area is dense and the surrounding rock grid far away from the concrete area is sparse. In this calculation model, the number of surrounding rock grids is reasonable. We can effectively reduce the calculation time and save the internal resources of the computer [2]. For the surrounding rock, the calculation model adopts the constraint method of fixed nodes, which can make the outermost node element of the surrounding rock neither move along the X direction nor move along the Y direction. The calculation model specifies the X direction as the horizontal direction and the Y direction as the vertical direction [3].

The unit division and constraints are shown in Figure 4 and Figure 5.
4. Load and correlation coefficient selection

This calculation model considers the structure subject to internal water pressure, surge pressure and concrete weight stress. But in this calculation model, we do not consider the gravity of surrounding rock and the pressure of surrounding rock[4].

Load partial coefficient under ultimate limit state is shown in table 1.

Table 1. Load combination and load partial coefficient

| Internal water pressure | Water hammer pressure | External water pressure | Structural weight |
|-------------------------|-----------------------|-------------------------|-------------------|
| Normal operating conditions | √ (1.0) | √ (1.1) | 0 | √ (1.1) |

Calculate water pressure=( Normal water level- Elevation of upper surface in the bottom plate) ×1.0+( Highest surge level- Normal water level)×1.1

Structural weight = structural body weight × 1.1

The force of the anti-vortex beam section is in Figure 6, the force of the adjustment section is in Figure 7.

5. Post processing and reinforcement calculation

Using the post-processing in ansys, we can analyze the X and Y direction stress of the structure. For anti-vortex beam section, X direction stress distribution range is -0.69 ~ 1.69Mpa and Y-direction stress distribution range is -1.74 ~ 0.24Mpa. For adjustment section, X-direction stress distribution range is -0.88~1.50Mpa and Y-direction stress distribution range is -1.82 ~ -0.01Mpa.

The stress cloud diagram is shown in Figure 8 ~ Figure 11.
Creating a path at the stress concentration point, we can read the stress distribution along the path, turn stress into internal force and calculate the axial force and bending moment at the path section. The formula method is used to calculate the reinforcement. The final reinforcement structure is shown in Table 2.

| Section position | Section height (mm) | Inner reinforcement | Outer reinforcement |
|------------------|---------------------|---------------------|---------------------|
| Bottom pier      | 1500                | 5Φ32                | 5Φ25                |
6. Conclusion
We can use ansys to establish the plane finite element analysis model and analyze the structure of the inlet and outlet of the hydropower station. The location about stress extreme point is completely consistent with similar projects. The distribution range of stress in the stress cloud is approximately the same as other projects and reinforcement results are also processed within a reasonable range. It can be seen that it is feasible to use ansys to analyze the structure and reinforcement about the inlet / outlet of hydropower stations. So it has a certain significance for promotion.

Acknowledgments
I would like to acknowledge my colleagues for their hard work in data collation and my unit for funding the page fee.

References:
[1] Zhang, Y. J. (2014) Finite Element Modal Analysis of Reciprocating Compressor Crankshaft Based on ANSYS Workbench. J. Applied Mechanics and Materials, 2916:48-52.
[2] Fu, Y. (2014) Modal Analysis for the Fuselage of the New Tube Automatically Benchmarked Equipment Based on ANSYS Workbench. J. Applied Mechanics and Materials, 3411(623):100-105.
[3] Ying, P. (2012) Research of Thermal Analysis Collaboratively Using ANSYS Workbench and SolidWorks Simulation. J. Applied Mechanics and Materials, 1499(127):127-132.
[4] Bao, J. (2014) Optimization Design of Lifting Beam for Heavy Engine Compartment Based on Ansys Workbench. J. Applied Mechanics and Materials, 3365(602):345-452.