Gate well block 3D finite element structure calculation based on ansys workbench

Du Wu¹, Yu Li¹ and Mengjie Zhang¹
¹Power China Huadong Engineering Corporation, Hangzhou 310014, Zhejiang, China
Corresponding author’s e-mail: wudu024@163.com

Abstract. The article is based on finite element calculation software. First, the author establishes a 3d finite element model of gate well block. Second, the author meshes reasonable grid and sets up contact between assemblies. Third, the author applies stress and constraints. Last, the author analyzes stress distribution and displacement in different directions. According to the result, we can find weak areas of the structure and provide theoretical basis for structural reinforcement.

1. Introduction
Gate well block is a structure in a hydraulic building. When the power station is running, water is in the gate well block. But when the gate falls during maintenance, one side of the gate obstructs water. So we can know from the above analysis that the gate well block is a structure which is forced in three directions along the X, Y and Z axes[1]. If we use plane finite element for calculation, then it is difficult to analyze the true force of the structure. So it is necessary to use 3D finite element for structural force analysis.

Ansys workbench belongs to CAE software. Comparing with traditional CAE software, it has a more user-friendly interface and its modeling function is also significantly improved over ansys apdl. Its meshing grid function is realized based on Divide & Conquer, and different meshing grid methods can be used for each part of the geometry. All grid data is uniformly written to a common data center. Self-contained solver can be used for post-processing. At the same time, the data in data center also can be imported into ansys apdl for post-processing. In summary, ansys workbench has easier to grasp features than the previous ansys apdl[2].

2. Establishment of finite element model
This model is for the analysis of gate well block in pumped storage power stations. First, the model is created according to the actual elevation and size of the gate well block by catia 3D modeling software. In the model, the length of the surrounding rock is 5 times the height of the gate well block. Then we need save the file output as an igs file after modelling[3]. Last, we need import the igs file into ansys workbench. The gate well block model is shown in Figure 1 and the surrounding rock model of the gate well block is shown in Figure 2. (The coordinate system direction of the coordinate system direction is not shown in the subsequent figures is the same as the coordinate system of Figure 1.)
3. Meshing grid and applying constraint

Gate well block and surrounding rock grid meshing uses ansys workbench's own meshing divider[4]. For concrete model, the meshing method adopts the Hex Dominant method which can provide a standard hexahedral meshing, and grid size is set as 0.3 meter. For the surrounding rock, the meshing method also adopts the Hex Dominant method which also can provide a standard hexahedral meshing, and grid size is set as 5 meter. Contact surface between surrounding rock and concrete is need to set the contact size, and the size is the same with the small element size on the contact surface, so the size is set 0.3 meter. The grid size of the part where the surrounding rock meets the concrete can be made as much as possible, which can make calculations is easier to converge in the post-processing and the calculations is more accurate[5].

In normal operating conditions, the contact between the surrounding rock and concrete is set as “bonded”, which is applied to all contact regions. If contact regions are boned, then sliding or separation between faces or edges is not allowed. The constraint of surrounding rock bottom is set “fixed support”, which makes bottom can’t move along the X Y Z axis. The four sides of front, back, left and right cannot move in the radial direction. The setting constraint is closest to the actual situation[6]–[7].
4. Load and correlation coefficient selection

The calculation takes ultimate limit state of normal operating conditions as an example, so we need choose the right combination of loads.

Basic load combination: Hydrostatic pressure(Corresponding normal water level) + Water hammer pressure + Structural weight. This calculation does not consider external water pressure, which can make calculations safer.

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

| 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 (1) According to the formula (1), the calculated head of this model is 118.619 m. Specific load application is shown in Figure 5 and Figure 6.

5. Post-processing and result analysis

Post-processing adopts the software’s own processor. The final calculation result of the model is normal convergence. By analyzing the model stress cloud map, the 1st principal stress about gate well block stress distribution range is -1.54 Mpa ~ 16.0 Mpa, and the maximum point of the 1st principal stress is at the floor concrete groove. Y-direction stress distribution range is -3.23 Mpa to 8.78 Mpa, and the maximum point of stress in the Y direction is the same as the position of the maximum point of the first principal stress. Z-direction stress distribution range is -2.57 Mpa ~ 7.69 Mpa, and the maximum point of stress in the Z direction is also the same as the position of the maximum point of the first principal stress. Model displacement distribution range is 0~0.63cm. The specific stress cloud map distribution is shown in Figure 7 ~ Figure 10.
Compare the calculation results of ansys workbench with the calculation results of ansys apdl, the stress distribution range in each direction is shown in Table 2.

|                      | Workbench |          | Ansys     |          |
|----------------------|-----------|----------|-----------|----------|
| 1st principal stress (Mpa) | -1.54~16.0 | -3.23~8.78 | -2.57~7.69 |
| Y-direction (Mpa)    | -3.01~8.97 | -2.12~7.42 |

6. Conclusion
The three-dimensional finite element analysis is based on ansys workbench. The maximum point of different direction stress cloud map and maximum displacement point position are the same with actual engineering monitoring results. The distribution of stress cloud map is basically consistent with the distribution of ansys apdl stress cloud map. It can be concluded that ansys workbench is suitable for the calculation of hydraulic structures. At the same time, the software is simple to operate, and the human-machine interface is more friendly and easy to grasp. So ansys workbench can be widely used in hydraulic building calculations.
References

[1] You Jun Zhang, Nan Zhao, Jie Lu. Finite Element Modal Analysis of Reciprocating Compressor Crankshaft Based on ANSYS Workbench[J]. Applied Mechanics and Materials, 2014, 2916(488).

[2] Fu Yang Chen, Yan Wu, Yu An He. Modal Analysis for the Fuselage of the New Tube Automatically Benchmarked Equipment Based on ANSYS Workbench[J]. Applied Mechanics and Materials, 2014, 3411(623).

[3] Ying Peng. Research of Thermal Analysis Collaboratively Using ANSYS Workbench and SolidWorks Simulation[J]. Applied Mechanics and Materials, 2012, 1499(127).

[4] Bao Jun Wang, Yi Tao, Jiang Wei Li, Rui Chao Chang, Lei Qiang Zong. Optimization Design of Lifting Beam for Heavy Engine Compartment Based on Ansys Workbench [J]. Applied Mechanics and Materials, 2014, 3365 (602).

[5] Xue Mei Qi, Jing Dong Zhang. ANSYS Workbench for Static Analysis of Excavator Arm [J]. Advanced Materials Research, 2014, 3181(926).

[6] Wei Feng Liu, Yan Min Zhang, Ke Xing Song, Pei Feng Zhao, Li Zhang. Finite Element Analysis and Optimization for the High Voltage Disconnector Self-Elastic Contact Base on ANSYS Workbench [J]. Materials Science Forum, 2012, 1534 (704).

[7] Kun Cheng. Finite Element Analysis and Structural Optimization of the Box on the ANSYS Workbench [J]. Advanced Materials Research, 2011, 1220 (211).