Universe Scientific Publishing

Application of Finite Element Optimization Design of ANSYS APDL Structure in Bridge Design

Le Zhou

Jiyang College of Zhejiang A&F University, Zhuji 311800, China.

Abstract: For the application of ANSYS APDL in the optimization design of bridge structure, three-span continuous beam bridge is selected, and ANSYS APDL is used for modeling, element selection, meshing, adding constraints and loading constraints. Furthermore, the finite element analysis is carried out. Subsequently, the results are obtained. It is worth noting that the deformation cloud map contributes to obtaining the force and deformation basis of the bridge design model, supports the optimal design of the bridge, and provides a theoretical basis.

Keywords: ANSYS APDL Structure; Finite Element Optimization Design; Bridge Design; Application.

1. Application and basis of bridge design and manufacture

The rapid development of China's economy urges the bridge, which plays an important role in transportation, to expand rapidly with the continuous extension of the transportation network [1]. In order to meet the needs of economic development and transportation, the load, span and deck width of the bridge are increasing, and the form of bridge structure is also evolving. Therefore, the calculation, mechanical analysis and construction methods of the bridge are becoming more and more complex. The safety and security of bridges, a kind of national infrastructure and an indispensable part of transportation, are essential to social development and national stability. Most importantly, the finite element analysis method has been more and more widely used in bridge engineering because of the rapid and effective development of computer technology. Among them, the ANSYS finite element software developed by American companies is trusted by designers because of its powerful function. After bridge modeling, the force can be directly simulated and the stress results under various working conditions and loads can be analyzed and calculated. The finite element principle enables static load mechanical analysis and safety assessment of various structural bridges [2].

2. Brief description of bridge structure

2.1 Superstructure

The superstructure of the bridge refers to the main load-bearing structure that crosses obstacles (such as rivers, valleys or other lines, etc.) when the line is interrupted, and it is also the general name of crossing the bridge hole above the bridge bearing (unhinged arch line or rigid frame main beam bottom line). When the span is larger, the structure of the superstructure is more complex, and the construction difficulty increases accordingly [3].

2.2 Substructure

The substructure includes pier, abutment and pier foundations. The pier acts as a structure that supports the span structure of a bridge in a river or on a bank. The abutment is located at both ends of the bridge, with one side connecting with the embankment to prevent the embankment from collapsing, and the other side supporting the end of the bridge span structure. The pier and abutment foundation is a structure that ensures the safety of the bridge pier and abutment and transfers the load to the foundation.

2.3 Support

The bridge support is an important structural component that connects the upper structure and the lower structure of the bridge. It is located between the bridge and the cushion stone. It can reliably transmit the load and deformation (displacement and corner) of the upper structure of the bridge to the lower structure of the bridge. An important force transmission device for bridges. There are two kinds of fixed bearing and movable bearing.

2.4 Ancillary facilities

The ancillary settings of the bridge include deck paving, drainage and waterproofing systems, anti-collision railings, expansion joints, and so on. The smoothness, abrasion resistance, non-warping, and non-permeability of the bridge deck pavement are the keys to ensure driving comfort. The drainage and waterproof system should be able to quickly drain the bridge area water and minimize the possibility of water seepage. The anti-collision railing is a structural measure to ensure safety, and it is also the best decoration for viewing. Expansion joint: the gap between the upper structure of the bridge span or between the upper structure of the bridge and the end wall of the bridge abutment to ensure the displacement of the structure under the action of various factors [3].

3. Finite element analysis of bridges based on ANSYS structure

3.1 Solid modeling through ANSYS

The SOILDWORKS modeling is saved as a .x _ t format file in ANSYS APDL or the bridge is modeled by ANSYS APDL itself, in which the modeling is roughly as follows: three-span continuous box girder, structural truss is composed of three-span truss, the length of each section of truss is 30m, the total length is 60m. The material is EX 2.06E11, with the Poisson's ratio of 0.31, the bulk density of 7.85E4, and the linear expansion coefficient of 1.2000E-5. The other parameters are shown in Figure 1. The loading includes concentrated force of side span, and the overall temperature rise is 20 ℃. Figure 2 shows the modeling model of ANSYS finite element model for three-span continuous beam bridge. In order to obtain more accurate results and simplify the calculation process, different structures in the bridge structure are simplified and replaced equivalently. The dimensions of the components on the bridge are modeled in proportion to the actual size. Each complex structure is simplified to a plane model.

Figure 1 Plane model diagram

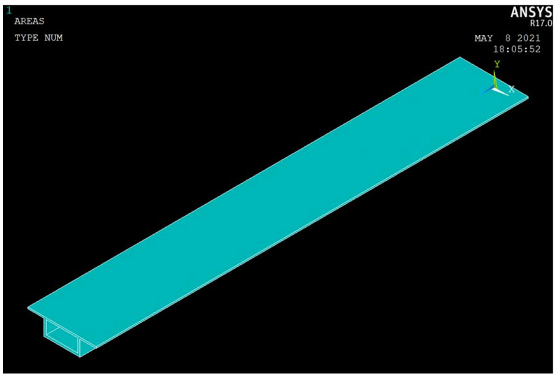

Figure 2 ANSYS model diagram

3.2 ANSYS bridge calculation model

3.2.1 Unit selection

The type of element is very important for finite element analysis, because it is the basis of finite element model [4]. In the ANSYS cell library, there are nearly 200 cell types, including two-dimensional, three-dimensional, beam, plate, shell and so on. Most of the engineering problems can be simulated and analyzed. The finite element model of bridge element is commonly used in beam element, so beam element is used to build three-span continuous bridge. The finite element model established according to the beam element not only simplifies the calculation, but also helps to determine the stress of the bridge structure, which is what we are most concerned about.

3.2.2 Simplified structure

Practice has proved that proper simplification is necessary for a complex structure without affecting the calculation accuracy. This greatly reduces the workload, while making the calculation and analysis results of the finite element software more accurate and making full use of computer resources. The actual structure of the bridge is more complicated. For the finite element calculation model of the bridge, part of the bridge structure can be further simplified and equivalently replaced [5].

3.3ANSYS's finite element model

3.3.1 Unit selection

In the finite element model of ANSYS, the beam elements in the library are selected by Add in Element Type according to the characteristics of 100 kinds of element library and bridge structure provided by ANSYS software. Each unit type has its own specific number and prefix that identifies the cell level, such as PIPE289, PLANE77, and so on. Since BEAM188 is suitable for analyzing slender to medium thick / thick beam structures and is widely used [6], BEAM188 is selected as the model of three-span beam bridge.

3.3.2 Meshing

Meshing refers to dividing the finite element model into many small elements, which is the top priority of the pre-processing of finite element analysis. The matching degree between the meshing and the calculation target and the quality of the mesh determine the quality of the finite element calculation in the later stage. Grid density scale plays a key role. If the mesh is too rough, it may lead to serious errors in the final calculation results. However, if the grid is too detailed, the computing process will increase, which will not only cost computing time, but also waste computer resources. Before dividing the grid, the grid size needs to be set. Specifically, click Preprocessor-Meshing-Mesh Tool on the right in ANSYS APDL, select Global Set under Size Controls and enter 0.15 to complete the goal of setting the cell size. Mesh: Lines in Mesh Tool can complete the meshing of the bridge model

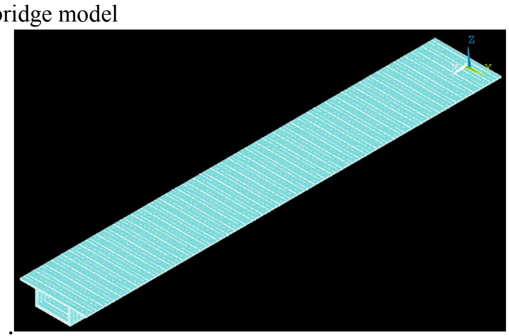

Figure 3 Meshing diagram

3.3.3 Loading and constraints

First of all, constraints are added to the three-span bridge model. That is, select the node Nodes in the Main Menu-Preprocessor-Modeling-Create in the main menu, and select In Active CS under Nodes. Then, it pops up a window for inputting coordinates, and inputting to add nodes in four places: X=0, 20, 40, 60, Y and Z are both 0. Then, the boundary conditions of degrees of freedom on X, Y and Z are added to the four nodes to simplify the mechanical model. Finally, the bridge model is saved and solved by adding loads and constraints to the model by picking key points in Solution-Define Loads-Apply-Structural-Force/Moment-On Key points, and the X-direction deformation cloud map, Z-direction deformation cloud map and stress cloud map are obtained by post-processing.

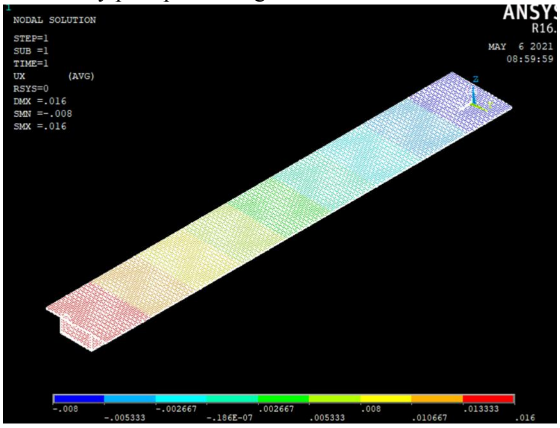

Figure 4 X-direction deformation cloud image

Through the deformation cloud map of the bridge in the X direction, it can be observed and analyzed that the overall deformation of the bridge is obvious, the deformation in the X direction is the smallest in the middle of the bridge, while the two sides of the bridge are deformed in the positive and negative direction of the Z axis respectively. In addition, the closer to both sides, the more obvious the deformation.

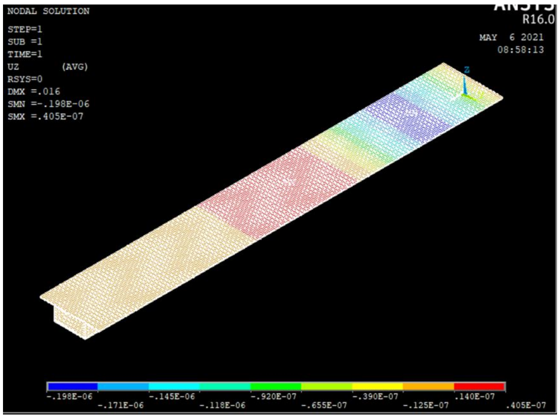

Figure 5 Z-direction deformation cloud image

According to the deformation cloud map of the bridge in Z direction, it can be observed and analyzed that the overall deformation of the bridge is obvious, the deformation in the middle of the bridge is the most obvious, and the deformation decreases gradually as it is closer to both sides, while the deformation in the middle of the bridge is negative to the X axis. The deformation occurs in the negative X direction from the far point of the coordinate axis to 1/5 of the bridge, and then slows down.

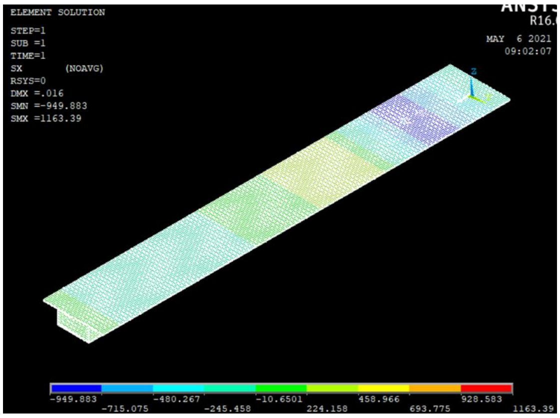

Figure 6 Stress cloud diagram

The stress cloud map of the bridge indicates that the overall deformation of the bridge is not obvious, where there are changes, there is negative deformation in the Z direction, while the negative X direction deformation occurs in the negative X direction. The deformation occurs in the negative X direction from the far point of the coordinate axis to 1/5 of the bridge, and then slows down.

4. Conclusion

ANSYS enables the establishment of large or complex models, and the program processing has the characteristics of small amount of data, simple and high efficiency. It not only allows geometric operation of nodes and elements, but also supports the use of surface elements, voxels and Boolean operations to build models, which is convenient to use the optimization design function of ANSYS program. It can be seen that ANSYS has a strong ability of modeling, dynamic analysis and topology optimization. The finite element model of the three-span continuous bridge is established by ANSYS, the mechanical model is simplified, and the structural statics method is used to analyze the force of the bridge. By loading the constraint, the deformation of the bridge under the action of concentrated force is simulated. However, in the specific design, it will involve theoretical knowledge of various aspects, especially the strong requirements for English proficiency, such as the skillful use of ANSYS APDL programs to solve practical engineering problems and optimize the design structure, as well as solid engineering knowledge and finite element theory [7]. Ultimately, through the simulation of the bridge, it provides a reference for the analysis of other types of bridges, and has a certain practical reference significance.

References

[1] Sun Yong, Cen Chengxian. Application of ANSYS in bridge engineering [J]. Journal of northeast forestry university,2005,33(2):104-105.

[2] Hong Hua, Yang Yongyong. Analysis of steel structure detail joints based on ANSYS [J]. Journal of Zhejiang tong vocational and technical college,2020,22(1); 12-16.

[3] Editorial Department of China Journal of Highway and Transport. China Railway Journal,2014,27(5); 1-96.

[4] Jin Shijia. Analysis on Optimization design of Turntable Structure of Aerial Working Vehicle Based on finite element Method [OL]. Science and Technology Forum,160.

[5] Chen Han. Stability Analysis of a Steel Structure Bridge Under Static Load Based on ANSYS [J]. Electrical Materials,2020:59-62.

[6] Liu Yuqing, Chen Xiaojian, Yang Shugeng, Wang Rui, Liu Xiaoyan. Oil field equipment,2012,41(4):39-43.

[7] Yan Chaochao, Shen Xiaoqin, Yu Fusheng, Zhu Baoxing, Liang Wei. Optimization Design of Frame Structure of Underwater Vehicle Based on ANSYs-Workbench [J]. Manufacturing automation,2020,42(10); 1-7.