Research on Lightweight of Docking Structure Based on ANSYS

Shaolei Chai¹, Ming Chen¹, Jigui Mao², Guangjun Long¹ and Nianpeng Wu³,*

¹ State Grid Ningxia Electric Power Co., Ltd, Ningxia, CHN;
² State Grid Ningxia Electric Power Co., Ltd Wuzhong Power Supply Company, Ningxia, CHN;
³ China Electric Power Research Institute, Beijing, CHN.
*Corresponding author email: wnp_67@163.com

Abstract. The docking structure has broad application prospects in the construction of tower assembly. However, the docking structure has a relatively large quality, which has a greater impact on the operations of high-altitude workers. This paper studies the impact force on docking structure during the docking process. The finite element analysis (FEA) model of the docking structure and the tower section is established and the impact force of the docking structure under various wording conditions is calculated. Based on the calculation results of the impact force the lightweight research of the docking structure is carried out and the optimization constraint conditions are proposed. The simulation model is established by ANSYS and the optimization design of guiding tools and vertical limit tools are completed by adjusting the structural parameters. After the optimization design, the docking structure is trial-produced and tested. Through the method of FEA and experiment, a butting device that meets the requirements of strength and rigidity and is lighter in weight is obtained. Compared with the existing docking structure, the weight of the optimized docking structure is reduced by about 39%. The research results can provide a reference for the design of the docking structure.

1. Introduction
With the large-scale use of mobile cranes to tower constructions, the dangers of safety have become increasingly prominent[1]. Multiple construction staff need to perform installation operations under the suspended tower[2-3]. In this case, the safety of the construction staff is at a high risk if the mobile crane failure occurs. In order to reduce the number of workers or working time for the tower construction, the construction technology of mobile crane for tower erection by using docking device is being promoted and applied.

The existing docking device has a relatively high quality, which has a greater impact on the construction of high-altitude workers. In order to avoid the above problems, it is necessary to reduce the weight of the docking device and study the impact force of the docking device. In order to shorten the research period and reduce the research cost, the finite element model for the existing docking device was established with ANSYS software for design.

2. Calculation of Dynamic Impact Force Based on ANSYS

2.1. Boundary Conditions and Load Cases
The docking structure includes a guide tool, a vertical limit tool, and a horizontal limit tool[4], as shown in Figure 1.
As shown in Figure 1, the guide tool and vertical limit tool are installed on the in-place tower section, so we fixed the lower end surface of the in-place tower section in ANSYS[5].

During the docking process, the tools may be impacted in the following three ways.
- Working condition A: The suspended tower section drops vertically at a constant speed and hits the guide rail of the guide tool.
- Working condition B: The suspended tower section moves horizontally at a constant speed and hits the guide rail of the guide tool;
- Working condition C: The suspended tower section drops vertically at a constant speed and hits the vertical limit tool.

2.2. Finite Element Model Establishment
In ANSYS simulation, the use of beam elements or solid elements to establish a three-dimensional model of the tower section will lead to extremely long simulation time, so the suspended tower section is simplified as follows:
- Build a trapezoidal tower model according to its top and bottom dimensions, as shown in Figure 2(a).
- Keep the model quality and center of mass position consistent with the actual tower section [6-7].
- The suspended tower section is established using shell elements and it is set as a rigid body [8].

The guiding tool and the vertical limit tool are established using solid elements, as shown in Figure 2(b).

2.3. Impact Simulation Results
The impact force of each working conditions is shown in Figure 3:
It can be seen from the figure that the peak value of vertical impact force of the guiding tool is 1040N, the peak value of horizontal impact force of the guiding tool is 1712N, and the peak value of vertical impact force of the vertical limit tool is 7783N.

3. Weight Reduction Design of Docking Structure

3.1. Constraints

The vertical limit tool is used for positioning [9], referring to the deviation of the tower hole, it is determined that the stiffness condition of the vertical limit tool is that the deformation should be less than 1.5mm.

In condition of static simulation conditions, the safety factor of the docking structure should not be less than 1.5, so the allowable stress of each tool should not be greater than two-thirds of the yield strength of the material [10].

In condition of dynamic impact simulation, in order to reduce the weight of each tool, it is stipulated that each tool should not undergo overall plastic deformed.

In summary, the constraints for weight reduction of the docking structure are shown in Table 1:

| Tools name            | Allowable deformation (mm) | Allowable stress (MPa) |
|-----------------------|----------------------------|------------------------|
|                       |                            | Static simulation      | Impact simulation    |
| Guiding tool          | ≤3                         | ≤236                   | Individual element ≤550 |
| Vertical limit tool   | ≤1.5                       |                         | Majority elements ≤355 |

3.2. Weight Reduction Goals

According to the impact simulation results of each tool before reduction, combined with reduction constraints, the reduction objectives of each tool are determined in Table 2.

| Tools name            | Maximum deformation | Maximum stress               | Weight reduction goals                                      |
|-----------------------|---------------------|------------------------------|------------------------------------------------------------|
| Guiding tool          | 0.63mm              | Individual element 226.7 MPa | Ensure that the guiding tool meet the constraints and      |
|                       |                     | Majority elements 158.4 MPa  | processing performance, and reduce the quality.             |
|                       |                     | Individual element 560.1 MPa | Reduce the maximum stress in the impact area and reduce     |
|                       |                     | Majority elements 345.2 MPa  | weight as much as possible.                                 |
| Vertical limit tool   | 0.56mm              |                              |                                                            |
3.3. Weight Reduction Process and Results

**Weight reduction method.** During the weight reduction process, all parameters were adjusted many times. Research and calculation based on the impact simulation model will lead to a longer time-consuming. In order to speed up the research progress, the optimization method is determined as follows:

- According to the impact simulation results, the peak impact force of each working condition is regarded as the static load of each tool;
- Adjust the structural parameters of each tool. Then obtain the structural parameters that meet the static constraint conditions through static simulation calculation;
- Finally, the structural parameters that meet the static constraints are checked by dynamic impact simulation calculation.

**Statics Calculation.** The simulation results of each model in the optimization process are shown in Table 3.

| Model No. | Guiding tool | Vertical limit tool |
|-----------|--------------|---------------------|
|           | Maximum deformation/mm | Maximum stress/MPa | Maximum deformation/mm | Maximum stress/MPa |
| 1         | 1.45         | 149.8              | 0.48               | 262.2              |
| 2         | 1.63         | 155.5              | 0.45               | 259.3              |
| 3         | 1.81         | 161.7              | 0.42               | 218.2              |
| 4         | 1.98         | 177.0              | 0.39               | 214.1              |

It can be seen Model 3 and Model 4 satisfy the static constraint conditions. However, Model 4 is lighter, so the impact calculation of Model 4 is carried out.

**Dynamic Impact Simulation Calculation.** Establish the dynamic impact simulation model to verify the structural parameters of Model 4, and the simulation results under 3 working conditions are shown in Table 4, and the result cloud diagram is shown in Figure 4.

| Tools name          | Horizontal impact guiding tool | Vertical impact guiding tool | Vertical impact vertical limit tool |
|---------------------|--------------------------------|-----------------------------|-----------------------------------|
|                     | Maximum stress/MPa | Maximum deformation/mm | Maximum stress/MPa | Maximum deformation/mm | Maximum stress/MPa | Maximum deformation/mm |
| Vertical limit tool  | 53.5               | 0.09                       | 31.2               | 0.07                       | 490.3              | 0.53                   |
| Guiding tool        | 91.3               | 1.37                       | 62.0               | 0.88                       | 265.8              | 0.45                   |
It can be seen from Table 4 and Figure 4:

- When the suspended tower impact the guiding tool in the horizontal direction, the mechanical property of each tool meet the constraint conditions.
- When the suspended tower impact the guiding tool in the vertical direction, the mechanical property of each tool meet the constraint conditions.
- When the suspended tower impact the vertical limit tool in the vertical direction, the deformation of each tool meet the constraint conditions. The maximum stress of the vertical limit tool is 490.3 MPa, which is a compressive stress and the area is extremely small. The maximum stress in the remaining areas is about 350 MPa. Therefore, the overall plastic deformation of the vertical limit tool will not occur.
- In summary, a design scheme that satisfies all constraints is obtained. After weight reduction, the total mass is reduced from 22.5 kg to 13.7 kg, which is a weight reduction of 39%.

4. The Prototype Test
Complete the layout of the guide tool test and the vertical limit tool test, as shown in Figure 5. Apply the rated load, 1.25 times the rated load, and 1.5 times the rated load in the corresponding direction, keep it for 5 minutes and then unload.
In summary, plastic deformation and overall damage of the docking structure after weight reduction didn’t occur during the test, and its strength and stiffness met the design requirements.

5. Conclusion
Through the research on the impact force of the docking structure, and finish the weight reduction design of the docking structure by using finite element method, the lightweight docking structure is produced and tested. The conclusions are as follows:

- The finite element model of the docking structure was established through ANSYS, and the accurate calculation of the impact force of the docking structure was completed.
- The finite element software was used to optimize the design of the docking structure, and the weight of the docking structure was reduced by 39%.
- Established a simulation program of docking structures, and proposed the constraint conditions for weight reduction of docking structures. And realized the determination of structural parameters through simulation calculation. Finally, a new optimization design method for butt joint structure is proposed.

6. Acknowledgement
Authors gratefully acknowledge the financial support of Science and Technology Project of State Grid Ningxia Electric Power Co., Ltd. (Research on mobile crane segmented tower construction technology based on docking device and crane arm magnetic flux leakage detection device, Grant No. SGNXWG00JJJS2100313).

7. References
[1] Wang C L, Jin Y K, Zhang J, et al 2019 QINGHAI ELECTRIC POWER vol 38 pp 44-47.
[2] Cheng J H, Wang L, Ding P J 2019 ZHEJIANG ELECTRIC POWER vol 38 pp 89-93.
[3] Qin J, Feng L, Wan J C, et al 2016 HOISTING AND CONVEYING MACHINERY pp 21-25.
[4] Miao Q, Huang K X, Xia Y J 2012 CONSTRUCTION TECHNOLOGY vol 41 pp 96-98.
[5] Zhu W 2018 Internal Combustion Engine & Parts vol 1 pp 116-117.
[6] Ma Z G, Huang D, Yao Q, et al 2021 Building Construction vol 43 PP 715-717.
[7] Fan J J, Chang J H, Ren W, et al. 2019 Journal of Appliance Science & Technology vol 5 PP 86-89.
[8] Chen Z, Tian H, Xu W Z, et al 2020 Machine Design and Manufacturing Engineering vol 49 pp 71-76.
[9] Liu W D, Duan W Y, Shen J, et al 2013. AIRCRAFT DESIGN vol 33 pp 64-67.
[10] Luo J K, Yin B 2010 Machinery Design & Manufacture vol 11 pp 185-186.