Statics analysis and topology optimization of support base for airdrop based on ANSYS Workbench

Xiaqi Zhang¹, Jinlong Wang²*, Hui Chen¹ and Chengdong Yuan¹

¹Shandong University of Science and Technology, Qingdao, Shandong 266590, China
²Weifang University, Weifang, Shandong 261061, China

*Corresponding author’s workplace e-mail: web@wfu.edu.cn; wfxyjinlong@163.com

Abstract. This article takes the support seat used to connect a certain airdrop system and the aircraft as the research object. According to the force of the support seat, the finite element analysis software ANSYS Workbench is used to perform static analysis on the support seat, and the overall deformation and stress of the support seat are obtained. Based on the static analysis, the topological optimization of the support base is carried out, and the original support base structure is modified according to the optimized results. Finally, under the premise of ensuring the performance requirements of the support base, the mass and volume of the optimized support base are reduced by 12.11%, meeting the design requirements of lightweight structure, and also providing a reference for future design work in this field.

1. Introduction
With the development of world science and technology, the transportation of materials by air has become an indispensable mode of transportation for all countries in the world. Whether it is the deployment of domestic resources or international cooperation, air transportation tends to be used for the delivery of long-distance post-disaster materials, military materials and personnel. In addition, the aviation delivery capability also reflects the technological level of a country. Therefore, the parts used for the connection and support of the airdrop system and the aircraft play a pivotal role. The quality of the connection and support parts not only determines the reliable safety performance of the entire airdrop system, but also determines whether the materials after the airdrop can be used normally.

2. Establishment of finite element model
Using Solidworks 3D drawing software to draw the 3D model of the existing support base. The 3D model of the support base is shown in Figure 1. Its outline size is 173mm×145mm×100mm. The bottom surface of the support base has eight through holes with a diameter of 6mm. The function is to connect and fix, import the drawn support base model into ANSYS Workbench in x_t format for static analysis and topology optimization.
2.1. Definition of material properties
The material used in this article is 35CrMnSiA. 35CrMnSiA is a low-alloy ultra-high-strength steel with good mechanical properties. It is mostly used to manufacture medium-strength load-bearing components in the aviation field. The material properties are shown in Table 1.

Table 1. Material properties of 35CrMnSiA

| Material Name | Material Density (kg/m³) | Tensile Strength (MPa) | Yield Strength (MPa) | Young's Modulus (GPa) |
|---------------|--------------------------|------------------------|----------------------|-----------------------|
| 35CrMnSiA     | 7850                     | 1620                   | 1275                 | 196                   |

2.2. Meshing
Meshing is an important step in the pre-processing of finite element analysis. The quality of the mesh will directly determine the accuracy of the analysis results. At the same time, the difference in mesh size will also affect the length of the calculation time. Therefore, choose according to the calculation requirements. A reasonable grid size is particularly important. The mesh size used in this article is 2 mm, and the default automatic meshing mode of the software system is adopted. The support model after meshing is shown in Figure 2. The total number of elements obtained is 1781470, and the number of nodes is 1262055.

3. Static analysis
Static analysis is the most basic analysis in finite element analysis. Through static analysis, the deformation and stress of the model under force can be clearly obtained. According to the dynamic equation of the object in the mechanics theory:

\[
[M]\{\ddot{x}\} + [C]\{\dot{x}\} + [K]\{x\} = \{F(t)\}
\]
Where, $[M]$ represents the mass matrix, $[C]$ represents the damping matrix, $[K]$ represents stiffness coefficient, $\{x\}$ represents the displacement vector, $\{F\}$ represents the force vector.

In linear static structural analysis, time does not affect the results of the analysis, so formula (1) can be simplified as:

$$[K]\{x\} = \{F\}$$

(2)

The position of the double ear holes on the upper part of the support base is connected with the steel wire rope by bolts, and receives an upward static load of 50 kN. The eight through holes at the bottom of the support base are used for fixed connection. According to these constraints, the drawn support base model is imported into ANSYS Workbench in x_t format and restrains the support base, the schematic diagram of the restraint imposed by the support base is shown in Figure 3 and Figure 4.

According to the above constraints, the support base is statically analyzed, the results of the analysis are shown in Figure 5. Figure 5a is the total deformation cloud map of the support base, and Figure 5b is the support base stress distribution cloud map. It can be seen from Figure 5 that the support base after receiving a static load of 50 kN, the part of the tray near the ear holes deformed greatly, the maximum value was 0.20353 mm, and the stress at the position of the ear holes was more concentrated, and the maximum value was 306.97 MPa.
4. Topology optimization

Topology optimization is a kind of structural optimization, its basic principle is to optimize the material distribution in a specified area according to a given load situation, constraint conditions and performance indicators. This article optimizes the topology of the support base on the basis of static analysis. After optimization, the material removal of the support base model is shown in Figure 7, it can be seen from Figure 7 that the middle position of the bottom of the support base and the middle of the elevation there are more materials removed in the position, and relatively less material removed on both sides. According to the results of topology optimization and the processing technology of the parts, the original support model is structurally improved, and the shape of the optimized support is finally determined as shown in Figure 8.

Figure 7. Topology optimization result graph  
Figure 8. Structure improvement of support base

Import the optimized support model into ANSYS Workbench again for static analysis, the analysis results are shown in Figure 9 and Figure 10. According to the analysis data in Figure 5, Figure 6, Figure 9 and Figure 10, the data in Table 2 is obtained.

Figure 9. Deformation cloud diagram of support base after optimization  
Figure 10. Stress cloud diagram of support base after optimization

From the data in Table 2, it can be seen that the total deformation and stress distribution of the optimized support base are almost the same as before the optimization, and according to the material properties and performance requirements, the performance of the improved support base fully meets the requirements of use, but after optimization, the mass and volume of the support base are reduced by 12.11% compared to before optimization.
Table 2. Comparison of data before and after support base optimization

|                          | Before optimization | After optimization |
|--------------------------|---------------------|--------------------|
| Total deformation(mm)    | 0.20353             | 0.23759            |
| Stress distribution(MPa) | 306.97              | 305.06             |
| Volume(cm³)              | 1158.884            | 1018.596           |
| Mass(g)                  | 9097.236            | 7995.976           |

5. Conclusion

Through static analysis and topology optimization of the support base, the following conclusions are drawn:

(1) Through the static analysis of the support base, it is concluded that the tray at the upper part of the support base near the double ear holes deforms greatly and stress concentration is easily generated at the double ear holes;

(2) The mass and volume of the support base after the structural optimization are reduced by 12.11% compared with that before the optimization, and the performance of the optimized support base fully meets the requirements of use, and also conforms to the lightweight design concept of parts.

References

[1] Wang M H, Chen X H and Zhou J 2006 Finite element simulation and optimization of support base structure J. Computer Simulation. (01) 108-110+151.
[2] Sun W G and Qu X 2013 Support base process design J. Aeronautical Science & Technology. 01 57-58.
[3] Mu X H and Xu L 2019 Optimization of the casting process of the support base J. Foundry Engineering. 43(01) 5-7.
[4] Wang Z N, Wu J J, Xiang J M, Wang B S and Jin Y Q 2020 Lightweight design of maintenance tool support base based on topology optimization and 6σ robustness J. Journal of Mechanical Strength. 42(01) 94-101.
[5] Chen K, Cao J G, Huang S T and Sun Y 2020 Numerical simulation and process optimization of support base forming process J. Hot Working Technology. 49(05) 97-99.