Research on Optimization Design of Shipborne Carbon Fiber Reinforced Polymer Frame Structure

H Wang¹², RC Yu¹², YZ Chen¹²* and L Hua¹²*

1. Hubei Key Laboratory of Advanced Technology for Automotive Components, Wuhan University of Technology, Wuhan 430070, China
2. Hubei Collaborative Innovation Center for Automotive Components Technology, Wuhan 430070, China
E-mail: yzchen@whut.edu.cn; hualin@whut.edu.cn

Abstract. Display consoles are the core frame structure of the command and control system on the military combat unit, which can be called the “brain” of the equipment. With the great development of new research on materials and applications, using Carbon Fiber Reinforced Polymer (CFRP) instead of aluminum alloys to fabricate the display console frame structure has become an international hot point. However, whether the components obtained by CFRP can meet the requirements of complex working conditions, and whether the structural design is reasonable, are still existing problems. This study analyzed the stress distribution of the bottom frame structure of CFRP display console under impact and vibration conditions. The frame structure was optimized based on the stress distribution and topology. The results show that the maximum stress value of the frame structure under complex conditions was 16.97MPa, the maximum stress after optimization was 22.94MPa. The reduction of weight of this complex structure was 13%. The prototypes have passed the actual working condition test. This study shows that the route method based on impact stress to optimize the wall thickness of CFRP structures is feasible and effective.

1 Introduction
The display consoles carry the command and control system of combat unit. These components are highly integrated with the detection, monitoring and controlling of signals, so as to realize the functions of alert detection, precise guidance and electronic countermeasures. Therefore, it is essential to ensure the reliability of these frame structures in complex service conditions such as explosion or strong vibration.

The display console products used in China is the second-generation fabricated by aluminum alloys⁴. However, the third-generation display console using Carbon Fiber Reinforced Polymer (CFRP) has been widely applied in western counties. In order to achieve military products with lightweight and high-strength, it is very urgent to develop shipboard display consoles using CFRP. The CAE analysis method has been widely used for the sake of improving forming efficiency and strength, reducing costs. With the help of ANSYS software, some modal analysis on an airborne CFRP display console was performed⁵⁶, the structure was optimized according to the obtained modes and natural frequencies. Yu Yang⁶ used ANSYS software to establish a finite element model of a shipborne display console, performed modal analysis on the display console. The overall structural strength of

* Corresponding author: YZ Chen(yzchen@whut.edu.cn) and L Hua(hualin@whut.edu.cn)
the display console according to the loading condition was evaluated. YAN Hong\(^5\) combined the fault-tolerant hierarchical sequence method to optimize the platform under an airborne display console to achieve a weight loss of 1.3 kg.

However, there is more attention should be paid to the reliability of shipborne equipment when it was subjected to explosions such as mines and torpedoes. At the same time, in order to meet relevant requirements, it is necessary to optimize the structure of CFRP components to achieve weight reduction. In this paper, the stress analysis of the CFRP frame structure under impact condition was analyzed. The wall thickness was optimized based on the impact stress distribution. Then, vibration simulation was carried out basing on optimization results. The final specimen manufactured according to the optimization results passed the test successfully.

2. Research methods and solutions

2.1 Research route and optimization method

This study was carried out according to the following method. Firstly, the finite element model of the CFRP frame structure was established and the original analysis was carried out. Then, the structure optimization was performed basing on the stress distribution under impact condition. Thirdly, whether the structural strength of the optimization result satisfies the requirements of complex working conditions such as vibration and impact was analyzed.

The analysis of the CFRP frame structure finite element model was carried out by OptiStruct\(^6\). After determining the constraints of the optimization model, according to the optimization purpose of this paper, the wall thickness of the frame structure was taken as the design variable, the impact stress was used as the constraint, and the minimum mass was used as the objective function to optimize.

2.2 Analysis conditions of impact and vibration

According to the GJB-150.16-1986/GJB-150.18-86 environmental test method for military equipment, the impact and vibration conditions of the frame structure are shown in the table1. The table shows that the vibration test is mainly divided into two frequency ranges including 1~16Hz and 16~60Hz. Among them, the displacement value of the component is required to reach 1.0mm in the range of 1~16Hz. The acceleration of the component is required to reach 10m/s\(^2\) in the range of 16~60Hz. The acceleration of the component is required to reach 10m/s\(^2\) in the range of 16~60Hz.

| Partition     | Test parameters | Acceleration(m/s\(^2\)) |
|---------------|----------------|------------------------|
| Frequency(Hz) | Displacement(mm) |                        |
| 1~16          | 1.0             | ----                   |
| 16~60         | ----            | 10                     |

Impact tests are generally tested using a hammer method. Before the test, the hammer is lifted to a certain height and then falls freely. The peak acceleration requirement is 7g, and there are certain requirements for the duration and speed variation, as shown in table 2.

| Waveform      | Test parameters |
|---------------|-----------------|
|               | Peak acceleration(g) | Duration(mm) | Speed change(m/s) |
| Half-sine wave| 7               | 11            | (2Ad/π)\(^2\), 2.10 |

2.3 Finite element model

Accurate modeling of the CFRP frame structure before finite element analysis is the basis for obtaining acceptable simulation results. In this paper, HyperMesh was used to simplify the CAD model of the CFRP frame structure and extracted the middle surface. After the necessary repaired, the mesh was accurately divided. In view of the fact that the material of the frame structure was carbon fiber composite material, the layer thickness was set using HyperLaminate including proper layup
angle and layup order of carbon fiber. The frame structure of this paper used Japan Toray T700 carbon fiber, adapting orthogonal layering method in the direction of 0° and 90°. The material performance parameters are as follows in table 3:

| Material  | Density (g/cm³) | Elastic Modulus(Gpa) | Poisson's ratio | Tensile strength (Mpa) | Bending strength(Mpa) | Elongation |
|-----------|----------------|---------------------|----------------|------------------------|----------------------|------------|
| T700      | 1.35           | 26.935              | 0.3            | 555                    | 530                  | 1%         |
| Resin     | 1.0            | 0.7                 | 0.45           | 18                     | ---                  | 70%        |

In order to obtain more accurate simulation results and analyze the optimization characteristics of composite materials, this study selected the typical framework structure for analysis. Its three-dimensional picture is shown in the figure 1. Its general size was about 700*450*600mm and consisted of 7 parts with weighs about 20.1kg.

There were seven parts in the CFRP frame structure, which were connected by bolts. In order to improve the accuracy of CAE analysis and obtain more accurate simulation results, the method of adding rigid connection bolt was used to simulate the connection state of each component of the frame structure. 56 simulation bolts were added to the key parts. The bolt position is shown in the white dot of the figure 2 below.

In the service process of frame structure, there were extra assembly parts in the components. Therefore, the exact load was added in finite element model according to the true weight of the extra assembly parts. The main weight is shown in the figure 3 and figure 4, which mainly includes the chassis and the wiring harness, and the total weight is 77 kg. According to the connection form of the chassis and the wiring harness board with the frame structure, weightless rigid connecting parts RBE2 was created at each connection position in the frame structure and intersected at a point as shown in figure 5. F1 and F2 were set as the chassis and wiring harness using the concentrated force loading mode.

3. Results and discussion

3.1 Stress distribution of CFRP frame structure
The stress distribution of the frame structure under the impact condition is shown in the figure 6 below. It can be seen from the figure that the maximum stress of the frame structure was 16.97Mpa, which located at a bolt hole position of the top plate. This means that the structure meet the requirements for the use of CFRP without any damage. At the same time, the overall stress value of the bottom plate is about 16.8Mpa. From the simulation results, it can be found that the stress of the plate 1, the plate 2 and the back plate were actually very small, indicating a larger optimization space for the following analysis.
3.2 Stress analysis after optimization
Using the optimization method mentioned, the thickness of the frame structure was decreased. The optimized stress distribution is shown in figure 7. It can be seen from the figure that the maximum stress of the frame structure after optimization was 22.94Mpa, which is located at a bolt hole position of the bottom plate and meets the requirements for the use of carbon fiber composite materials without damage. At the same time, the bottom plate is more stressed, and the overall stress value is about 22Mpa. In general, the optimization results meet the strength requirements of vibration and impact conditions.

3.3 Comparison of the results
The stress and weight of the CFRP frame structure before and after optimization are shown in the table 4. It can be seen from the table that before optimization, the plate 1, 2 and the back plate had less stress and had a larger optimization space. After optimization, the side plate 1, the side plate 2 and the back plate are respectively reduced by about 0.53 kg, 0.32 kg, and 0.09 kg, while the maximum stress of other large stress components, were increased compared to before optimization, but they are still much smaller than the allowable stress of T700 carbon fiber composites, and the optimization results meet the requirements.

| Optimization | Bottom plate | plate 1 | plate 2 | plate 3 | plate 4 | Back plate | Top plate | Total |
|--------------|--------------|---------|---------|---------|---------|------------|-----------|-------|
| before       | Max stress/Mpa | 16.8    | 13.0    | 7.5     | 15.1    | 14.2       | 0.36      | 16.97  | 16.97 |
|              | weight/kg    | 4.59    | 3.72    | 3.71    | 2.99    | 0.90       | 0.67      | 3.54   | 20.12 |
| after        | Max stress/Mpa | 22.94   | 16.52   | 9.9     | 16.98   | 20.87      | 0.51      | 22.63  | 22.94 |
|              | weight/kg    | 3.95    | 3.19    | 3.39    | 2.57    | 0.78       | 0.58      | 3.04   | 17.5  |

3.4 Specimen and test results
According to the results of CAE analysis, we used RTM technology to manufacture various parts of CFRP frame structure, and carried out the assembly work such as punching, connecting and repairing, and obtained the finished products as figure 8 and figure 9. According to the relevant national test standards, the vibration (figure 10) and impact (figure 11) tests of the frame structure were carried out. The results showed that the optimized frame structure has successfully passed the impact and vibration tests, and met the requirements of use. It showed that the method provided in this paper can significantly improve design efficiency, reduce costs and achieve products lightweight.
4. Conclusion

1) This study used Hypermesh and Optistruct to analyze the stress distribution of carbon fiber composite frame structure under impact and vibration conditions, and the topology of the structural wall thickness was optimized based on the impact stress rather than the modal analysis.

2) In original analysis, the maximum stress of the CFRP frame structure is 16.97Mpa. After optimizing the frame structure, the structure weight loss was 13%, and the maximum stress was 22.94MPa, which meets the requirements of parts. The prototype components passed the actual working condition test.

3) This study shows that the route method based on impact stress to optimize the wall thickness of composite structures is feasible and effective.

5. Acknowledgments

This study is supported by National Key R&D program of China No. 2018YFB1106700, National Natural Science Foundation of China (No. 51775398 and 51805392), the Fundamental research Funds for the Central Universities (WUT: 2018lll067GX, 2018lll074GX and 2018IVA06).

References:

[1] Chen C and Yang N 2009 Development of modeling and structural design of radar display console Selected Works of Mechanical and Electronic Science Conference of Electronic Mechanical Engineering Branch of China Electronics Society

[2] Sun F 2015 Structural dynamics simulation analysis and optimization of a carbon fiber display console J. Electronic Mechanical Engineering 31 pp 58-64

[3] Hang W, Sun F and Shen W 2015 Design and experiment of helicopter loaded display console J. Electromechanical Engineering 31 pp 39-41,45

[4] Yu Y and Yang W 2011 Structural dynamics simulation analysis of a shipboard display console J. Electronic Mechanical Engineering 27 pp 58-61

[5] Yan H 2016 Lightweight design of an airborne display console based on topology optimization J. Machinery and Electronics 34 pp 22-25

[6] Hong Q, Zhao K and Zhang P 2012 OptiStruct & HyperStudy theoretical basis and engineering application (Beijing: Mechanical Industry Press)