Analysis and Optimal Design of Antenna System's Forced Performance

Baofu Tang\textsuperscript{1}, Jinwei Wang\textsuperscript{1}, Yihan Zhong\textsuperscript{2}, Qian Zhang\textsuperscript{2}, and Jianguo Cai\textsuperscript{2,*}

\textsuperscript{1}Nanjing Research Institute of Electronics Technology, Nanjing 210039, China
\textsuperscript{2}National Prestress Engineering Research Center, Southeast University, Nanjing 210096, China

Abstract. The application of antenna system is becoming more and more widely. With the increasing demand for active phased array radar antenna, a kind of antenna array and antenna structure system for active phased array radar antenna is designed. According to the engineering requirements, the structural scheme is specifically designed, and the static analysis under static load and dynamic under random load are carried out by using finite element analysis software. The mechanical analysis verifies the excellent mechanical properties of the structure scheme, the accuracy of engineering accuracy and the excellent ability to bear random load. This paper puts forward a kind of platform optimization scheme which applies prestress to antenna platform, and proves the rationality of the scheme through calculation.

1 Introduction

With the development of science and technology and modern information industry, the demand for reliable, stable and high-performance antenna systems in various fields such as communication and military affairs is becoming increasingly strong \cite{1,2}. After decades of upgrading, radar antenna system has gradually transformed from passive phased array to active phased array. Active phased array radar antenna system gradually occupies the mainstream position of modern radar antenna system with high resolution, excellent anti electronic interference ability, stable and reliable continuous working ability and large coverage ability \cite{4,5}. Through the active phased array technology, the antenna system, with its high adaptability, can be better combined with electronic information technology, electric scanning technology, numerical simulation technology, and microwave phase-shifting technology, so as to make the formation of digital beam more convenient and fast, and the system can obtain excellent and stable working performance \cite{6-8}. Although the performance of active phased array radar antenna is excellent, its design has high requirements for structural accuracy, complex structure composition and high overall cost. Mei et al. \cite{9} proposed a three-layer antenna structure composed of right composite panel and Nomex honeycomb structure. The microstrip patch antenna is integrated into the structure, which has high electrical loss ability, which can effectively improve the performance of the antenna and improve the continuous working ability of the antenna.

Based on the structural design requirements of active phased array antenna system, a new structure system including support, platform and antenna body is proposed in this paper. According to its load condition, static analysis and modal analysis of the structure
are carried out, and the flatness, shafting accuracy and stress meet the design requirements. At the same time, according to the random vibration load spectrum of the structure, the excellent mechanical properties of the proposed structure are verified, and the mechanical structure of the platform is optimized.

2 Design model and calculation conditions of antenna structure

2.1 Structural model

It should not be indented. Leave 28 mm of space above the title and 10 mm after the title. The structure of the antenna system is composed of four lower supporting legs, a lower circular support, an antenna platform, an antenna body, a central rotation axis and lateral triangular supports on both sides. The structure model is shown in figure 1.

![Figure 1. Overall structure model.](image)

The antenna structure system is supported on the ground by four supporting legs. The antenna body includes a sub-array frame, a reflective surface and an outer protective layer. The entire antenna body can be rotated around the central axis of rotation to give full play to its working performance. In the overall structure, the sub-front frame is made of 5A05, and the material of the rest of the structure is made of Q345.

2.2 Calculation conditions

In the static analysis of the structure, only the gravity load and wind load are considered. Wind speed $v=30\text{m/s}$, basic wind pressure $0.7875\text{KN/m}^2$. The wind load is applied to the protective layer connected with the reflecting surface according to the actual situation, and its direction is perpendicular to the reflecting surface. In order to accurately simulate the actual force, consider the angle of the antenna body rotating around the central axis of rotation, and divide it into four load conditions: $0^\circ$, $30^\circ$, $60^\circ$, and $90^\circ$.

Since the antenna structure needs to be transported, it may be subjected to various random loads, which will have a greater impact on the durability and stability of the structure. Therefore, it is necessary to perform dynamic analysis on the whole structure according to the random vibration load spectrum.

3 Static analysis

3.1 Finite element analysis model

In order to accurately simulate the actual force of the front antenna structure, the finite element software ABAQUS is used to analyze the structure, and the pre-processing software Hyper Mesh is used. According to the actual situation of the structure, shell elements and
solid elements are used to complete the simulation. The shell elements are S3 and S4R elements, and the solid elements are C3D4 and S8R elements. The element connection is simulated by node coupling, and the finite element model is shown in figure 2.

![Structural finite element analysis model (0 degree wind direction).](image)

Figure 2. Structural finite element analysis model (0 degree wind direction).

In the static analysis of the structure, the four supporting legs Ux, Uy, Uz in the model are restrained from three degrees of freedom. The direction of the gravity load is vertical downwards, opposite to the Y axis. When the antenna body rotates by 0°, the antenna body is perpendicular to the antenna platform. The analysis models of other working conditions are shown in figure 3.

![Analysis model of each working condition.](image)

Figure 3. Analysis model of each working condition.

### 3.2 Displacement

Through simulation calculation, the displacement nephogram of array under static load can be obtained under different working conditions. The overall displacement of the structure under 60° is shown in the figure 4. The displacement of the structure under various working conditions are shown in table 1.

![Overall displacement of structure under 60° working condition.](image)

Figure 4. Overall displacement of structure under 60° working condition.
### 4 Random vibration analysis

The finite element analysis software ABAQUS is used to analyze the random vibration of the front antenna under transportation. The power spectrum input curve under random vibration is shown in figure 5.

**Figure 5.** Power spectrum input curve under random vibration.

#### 4.1 Finite element model

According to the actual situation of transportation, the boundary condition of the antenna structure in the transportation state is that all the degrees of freedom of the corresponding positions of the two longitudinal beams in the middle of the antenna platform are constrained. The specific constraints are shown in figure 6.

**Figure 6.** Front antenna model under transportation (the box is the constraint condition).

In the analysis process of the finite element software, there are two load steps, load step 1 Frequency and load step 2 Random Response. The first load step is to find the modal properties of the structure, and the second load step is on the basis of load step 1, the modal superposition method is used to calculate the response of the structure under random vibration. Considering that the low-order modes control the response of random vibrations, the first 30 modes of the front antenna are calculated in load step 1, and the first 30 modes of the front antenna are calculated in load step 2. Random vibration response in the range of 5-500Hz.
4.2 Modal analysis

In the load step 1, the first 30 modes of the structure are calculated. Since the low-order modes are the main factor for the random vibration response of the structure, the first five modes of the antenna structure in the transportation state are extracted, as shown in table 2.

Table 2. The first ten modes of the antenna structure in transportation.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency /Hz</th>
<th>Mode</th>
<th>Frequency /Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>14.113</td>
<td>6</td>
<td>24.141</td>
</tr>
<tr>
<td>2</td>
<td>15.904</td>
<td>7</td>
<td>29.826</td>
</tr>
<tr>
<td>3</td>
<td>18.367</td>
<td>8</td>
<td>32.009</td>
</tr>
<tr>
<td>4</td>
<td>18.842</td>
<td>9</td>
<td>32.152</td>
</tr>
<tr>
<td>5</td>
<td>20.423</td>
<td>10</td>
<td>37.448</td>
</tr>
</tbody>
</table>

4.3 Displacement and acceleration response

According to the constraints, select the response of 4 points, the middle point of the sub-array frame, the middle point and end point of the antenna platform, and the cylindrical center point of the lower support leg. The random vibration response of each point is shown in table 3.

Table 3. Random vibration response of the intermediate point of the sub-front frame.

<table>
<thead>
<tr>
<th>Point position</th>
<th>U1 response extreme frequency /Hz</th>
<th>U2 response extreme frequency /Hz</th>
<th>U3 response extreme frequency /Hz</th>
<th>A1 response extreme frequency /Hz</th>
<th>A2 response extreme frequency /Hz</th>
<th>A2 response extreme frequency /Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Middle point of subarray skeleton</td>
<td>15.95, 18.82, 20.3</td>
<td>14.11</td>
<td>15.95, 18.82</td>
<td>14.11</td>
<td>14.11</td>
<td></td>
</tr>
<tr>
<td>Middle point of antenna platform</td>
<td>14.11, 15.95, 20.39</td>
<td>18.38, 32.15</td>
<td>14.11, 15.95, 32.10, 68.88</td>
<td>14.11, 18.38</td>
<td>18.38, 32.15</td>
<td></td>
</tr>
<tr>
<td>Antenna platform endpoint</td>
<td>14.11</td>
<td>14.11, 32.15</td>
<td>14.11</td>
<td>14.11, 49.91</td>
<td>68.88</td>
<td></td>
</tr>
<tr>
<td>Center point of lower support leg cylinder</td>
<td>15.95, 20.39, 32.10</td>
<td>18.38</td>
<td>20.39, 32.10, 52.14</td>
<td>14.11, 18.38</td>
<td>20.39, 32.10</td>
<td>18.38</td>
</tr>
</tbody>
</table>

It can be seen from the above table that the random vibration response of the antenna system is obvious when the excitation frequency is 14.11 Hz., 15.95 Hz., 18.38 Hz., 20.39 Hz. and 32.15 Hz.. Displacement and acceleration response

According to the stress calculation results of the finite element software, the strength of the antenna array can be evaluated. The stress results are shown in figure 7.
It can be seen from figure 10 that the root mean square stress of antenna array under random vibration is generally small, but there is local stress concentration phenomenon, which is due to the inevitable stress concentration caused by component connection in the modeling process, which will not happen in the actual structure. Ignoring the stress concentration, RS11 and rs22 stress levels of the antenna array are low, which meet the requirements.

![Image](image.png)

(a) Root mean square stress of antenna array RS11

(b) Root mean square stress of antenna array RS22

**Figure 7.** Strength diagram of antenna array.

5 Prestress optimization scheme of antenna platform

Because the antenna structure will be continuously affected by various static and dynamic loads, the random load in the transportation process has a great impact on the structural stability. At the same time, in order to consider the economic and accuracy requirements of the structure, the structural scheme of the antenna platform is optimized. The prestressing tendons are arranged horizontally to apply prestressing force, and the effect of dead load on the platform is balanced by prestress.

According to the calculation results of the dead load of the upper platform, the number of prestressed reinforcement is calculated, and 1570 steel wire is used as the prestressed reinforcement, A0 = 38.5mm². The prestressed reinforcement is added to ABAQUS for calculation. The calculation model is shown in figure 8.

![Image](image.png)

**Figure 8.** Platform optimization model.
The corresponding prestressed members are made of Liuzhou OVM standard DM type pier head anchorage, including dm7a-13 DMA anchor and dm7b-13 DMB pier head anchor plate; the supporting jack is ydcw1000b. All related components are standard components. The loading scheme of prestressing force is as follows:

The tension end anchorage is arranged on the outer edge of the platform (no welding is required, opening at the end, and the prestressed construction is covered with a cover plate).

Put one end of the steel wire pier head, and the other end into the hole corresponding to the anchorage.

After 13 strands of steel wire on an anchor plate are in place, all the steel wires are put into the pier head at the fixed end (there is no need to open holes in the middle part for the construction of both sides of the prestressed reinforcement, and a certain range of holes should be opened at the corresponding positions for the construction of the middle two bundles of prestressed reinforcement, and the opening diameter is about 200 mm.

When the anchorage end of pier head is finished, the prestress at the tensioning end can be adjusted according to the actual situation. The adjustment effect of platform displacement is different due to the different tensioning amount and prestress in steel wire. The anchor cup of dm7a-13 at the tension end has thread in it. The anchor cup can move outward and the nut of the anchor can be tightened by the matching tensioning steel rod (with corresponding thread) after tightening. When the tension is in place, the nut contacts with the steel plate, and the nut is against the steel plate. The prestress is transmitted to the steel plate through the nut, and the steel plate transmits the force to the platform, thus establishing the prestress.

Through calculation, when the area of steel wire in the model is 741mm², the upward displacement of prestressed reinforcement is 4.72mm. At this time, the stress of prestressed steel wire is 250Mpa, which just offsets the displacement caused by dead load. At this time, the dead load is balanced. The stability and durability of the structure have been greatly improved.

6 Conclusion

In this paper, a new type of antenna structure system for active phased array radar antenna is designed. According to the static load and dynamic load, engineering accuracy requirements and performance requirements, the structure is specifically designed.

The static analysis and dynamic analysis of the structure under different loads and working conditions are carried out by using the finite element analysis software. The displacement of the structure and the structural response under the random load are obtained. It is verified that the force of the structure is reasonable, the structure performance meets the requirements of the actual project, and the rationality of the design scheme of the antenna structure system is proved.

According to the static load of the antenna platform, the optimal design of the platform structure scheme is carried out, the number of prestressed reinforcement and construction method are determined, and the rationality of the optimization scheme is verified.

References


