Structure design and mechanical analysis on a large spaceborne flat seam antenna

Wenjie Fan\textsuperscript{1,a}, Xiaopeng Li\textsuperscript{1,b}, Ning Li\textsuperscript{1,c}

\textsuperscript{1}Key laboratory of microwave remote sensing, Center for Space Science and Application, China

\texttt{fanwenjie@mirslab.cn} \texttt{lixiaopeng@mirslab.cn} \texttt{lining@mirslab.cn}

\textbf{Keywords:} mechanical analysis; flat seam antenna.

\textbf{Abstract.} Large spaceborne flat seam antennas boom recently. The structure design of the large flat seam antenna is introduced, and finite element model is built. The first frequency of antenna is obtained from modal analysis and it meet the requirement. The results of sine vibration and random vibration shows that strength satisfy the requirement and it has a certain margin.

\textbf{Introduction}

Following the rapid development of wireless communication technology, as a sender and receiver the antenna play an important role. For enhancing anti-jamming performance, the antenna must meets the requirements-narrow beam and low side lobe. So flat seam antenna would achieve the requirements. Spaceborne device must withstand rigorous vibration environment and it's necessary to research mechanical character of the large flat seam antenna.

\textbf{Large flat seam antenna structure}

The integrate structure comprise five parts: reflector, mounting plate, brace, brace cleat and waveguide. The size of reflector is 1200mm $\times$ 400mm, and about 1180 seam on the reflector, as shown in Figure 1. The reflector is made of 3A21 and mounting plate and brace is made of honeycomb sandwich. The mounting plate braces and support the reflector. Panel skin is made from CFRP (carbon fiber reinforced plastic) and aluminium core size is 4mm(side) $\times$ 0.05mm(thick). The material of brace cleat is 2A14 and waveguide is 3A21. Figure 2 shows the top view of antenna.

\textbf{Finite element model of plate seam antenna}

A finite element model (FEM) for analysis of the structure is an essential part of the design development process. Fig. 3 shows the finite element model of the large plate seam antenna. Reflector, mounting plate, brace, brace cleat and waveguide are divided into a large number of discrete shell elements. Material properties, shape, degree of freedom and connection are defined.

There are 1188 seams in the reflector and the size of each seam is 2mm $\times$ 10mm. The existence of seams bring much trouble to build model. So it should be do that assess the seams influence on antenna mechanical performance. Modal analysis results show that the seams influence on the first fundamental frequency is merely 3.1\%, and overall mechanical performances of antenna are concerned. So the seams are deleted in the FEM.
Modal analysis

The mounting hole of antenna are fixed in modal analysis. Table 1 shows the former five modal results and Figure 4 shows the former two modal shape. The first fundamental frequency is 104.7Hz and satisfies the requirement that it should be bigger that 100Hz. Vibration test photo of antenna is shown in Figure 5 and the 1st frequency is 107Hz, so the error is 2%.

<table>
<thead>
<tr>
<th>Natural frequency (Hz)</th>
<th>Modal effective mass</th>
<th>Modal shape</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X</td>
<td>Y</td>
</tr>
<tr>
<td>1</td>
<td>104.7</td>
<td>5.9%</td>
</tr>
<tr>
<td>2</td>
<td>114.4</td>
<td>23.8%</td>
</tr>
<tr>
<td>3</td>
<td>216.2</td>
<td>17%</td>
</tr>
</tbody>
</table>

Vibration analysis

Spaceborne antenna should stand the mechanical environment: acceleration, sine vibration and random vibration. The acceleration level is low, so it could be ignored. Sine vibration and random vibration are performed.

Sine vibration analysis. Low frequency sinusoidal vibrations occur as a result of the interaction between launch vehicle mode forms and loads occurring during combustion of the engines, during combustion of the engines sinusoidal vibrations occur, both in, and adjacent to, the launch direction.
The frequency range of sine vibration is from 5Hz to 100Hz. Y-direction is the lowest stiffness of antenna, so this article just shows the results of antenna in Y-direction.

Figure 6 shows acceleration response contour of antenna in Y-direction sine vibration. The maximum acceleration response is 101g and lies in two sides of antenna reflector.

![Figure 6 Acceleration response contour](image1)

![Figure 7 Acceleration frequency response curve](image2)

The base input acceleration curve and the maximum acceleration frequency response curve are shown in figure 7. It can be seen from the curve that the acceleration response increases from 70Hz and to the maximum in 100Hz.

The stress distribution contour of antenna is shown in figure 7 and the maximum stress is 73MPa and lies in one side of reflector. The minimum factor of safety is 1.9.

![Figure 8 Stress contour of antenna in sine vibration](image3)

![Figure 9 Acceleration contour in random vibration](image4)

**Random vibration analysis.** Acoustic loads and boundary layer turbulence are transformed into mechanical vibrations in the launch vehicle, which affect the antenna on spacecraft.

Figure 9 shows the acceleration distribution contour of antenna in Y-direction random vibration and the maximum acceleration RMS (root mean square) is 97.7grms and lies in the waveguide. Then the maximum acceleration RMS of reflector is 47grms.

![Figure 10 The acceleration response PSD](image5)

![Figure 11 The stress PSD contour in random vibration](image6)

Figure 10 shows the acceleration response PSD (power spectrum density) in Y-direction random vibration. The stress response PSD is shown in Figure 11 and the maximum value is 11.6MPa. The
maximum stress response PSD of brace cleat is 23.7MPa and the maximum value is 9.8 MPa in the waveguide.

Table 2 lists the stress value and factor of safety of each part of antenna.

<table>
<thead>
<tr>
<th>part</th>
<th>$3\sigma$ stress (MPa)</th>
<th>factor of safety</th>
</tr>
</thead>
<tbody>
<tr>
<td>reflector</td>
<td>34.8</td>
<td>1.3</td>
</tr>
<tr>
<td>brace 2A12</td>
<td>71.1</td>
<td>2.9</td>
</tr>
<tr>
<td>composite</td>
<td>144 (maximum principle stress)</td>
<td>1.93</td>
</tr>
<tr>
<td>waveguide</td>
<td>29.4</td>
<td>2.5</td>
</tr>
</tbody>
</table>

**Conclusion**

The structure design of the large flat seam antenna is introduced, and finite element model is built. The modal analysis conclusion is the 1st natural frequency is 104.7 and satisfies the requirement. The strength of antenna in sine vibration and random vibration also meet the requirement and the antenna has a certain safety margin.

**References**


DOI References

http://dx.doi.org/10.1109/TAP.1983.1143002