Research Article

Lightweight Design of Solar UAV Wing Structures Based on Sandwich Equivalent Theory

Hongjun Liu,1 Dong Zhou,2 Bing Shen,2 and You Ding2

1Science and Technology on UAV Laboratory, Northwestern Polytechnical University, Xi’an 710065, China
2School of Aeronautics, Northwestern Polytechnical University, Xi’an 710072, China

Correspondence should be addressed to Hongjun Liu; hjliu@nwpu.edu.cn

Received 19 July 2021; Accepted 22 September 2021; Published 18 October 2021

Academic Editor: Jinyang Xu

Copyright © 2021 Hong Jun Liu et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

In this paper, an equivalent method based on sandwich plate is deduced, and the equivalent parameters of the honeycomb plate are obtained. With these equivalent parameters, the honeycomb plate equivalent FEM simulation model and actual model are established, and three-point bending simulations of the equivalent model and the actual model three-point are completed. Then, a three-point bending test of a real honeycomb sandwich panel was performed for comparison with the simulation result, which agrees well with the test result and shows the effectiveness of the equivalent model. The equivalent model of honeycomb sandwich plate win ribs is established for structural topology optimization and wing static simulation analysis, and a prototype of the solar UAV is made for flight testing according to the topology optimization results. The simulation and prototype test results indicated that the sandwich equivalent theory is suitable for the lightweight design of solar UAV wing structures with honeycomb sandwich plate materials, and this method can provide a reference for the same type of wing structure design.

1. Introduction

Solar UAVs are unmanned aerial vehicles that operate by converting solar energy into electric energy. To reduce energy consumption and prolong battery life, a large number of composite materials with high specific strength, high specific stiffness, and light weight are widely used in the bodies of UAVs, and the structural optimization method is often used to reduce the structural weight while ensuring the structural strength design requirements are met. A honeycomb sandwich panel is a common material, that is generally composed of upper and lower symmetrical skin layers and a honeycomb core in the middle, for the wing structures of UAVs. The skin layers are mainly responsible for bearing tensile and bending stress within the sandwich panel, while the honeycomb core has the function bearing transverse shear stress. Due to the characteristics of composite materials and honeycomb sandwich structures, the honeycomb core has a much more complex modeling process requiring a larger number of calculations, which increases the difficulty of wing structure design. Therefore, the optimization design of composite wing structures with honeycomb sandwich structures is a very important topic.

Honeycomb sandwich panel structure composite materials are widely used in modern national defense industries due to their high specific strength, high specific stiffness, and light weight. These materials are generally composed of upper and lower skin layers that mainly bear the tensile and bending stresses within the sandwich panel, and a middle honeycomb core that bears the transverse shear stresses [1, 2]. As to the characteristics of the materials, use of a honeycomb sandwich structure increases the number of design variables and the analysis complexity. Therefore, analyzing and calculating the mechanical properties of the honeycomb sandwich panel structure is an important topic.

Because honeycomb sandwich structures are different from continuous solids, they are not easy to directly model and analyze through finite element software, and scholars have carried out many studies on them. Habip et al. studied the stability of honeycomb sandwich structure [3–6]. Guj et al. established the core equivalent continuum sandwich
plate model for finite element frequency response analysis [7–9]. Soliman et al. used different equivalent theories in the finite element static analysis of sandwich plate [10, 11]. In previous studies, the analytical or numerical analysis approach was often used for a single honeycomb core [12, 13]. In regard to complex honeycomb sandwich structure analysis, these methods are often not appropriate. To analyze the honeycomb structure more intuitively and accurately, Aktay et al. simulated the skin panel and established the microhoneycomb core model by shell elements [14]. Zou Weijie et al. also built a micromodel to analyze the in-plane modulus of the Nomex-honeycomb core [15, 16]. For the current finite element simulation analysis software NASTRAN, ABAQUS and others have no cell library or material setting interfaces, so the actual model can only be built manually, and the modeling process is complex and inefficient. Therefore, in the process of calculation, some special methods are often adopted such as the equivalent method [4, 5], which can solve this problem well.

The existing equivalent methods include equivalent plate theory, honeycomb plate theory, and sandwich plate theory, and the equivalent model changes with different equivalent theories [10–14]. The whole honeycomb sandwich structure can be equivalent to an anisotropic thin plate with uniform thickness by the first two methods, while the panel is not changed, and only the honeycomb core is equivalent by the last method. In practical applications, some simulation data can be obtained by the equivalent model, which can help improve the work efficiency and ensure a high precision without the complex modeling process. Sandwich theory is a commonly used method to deduce equivalent theory [17–20].

The structural complexity of honeycomb sandwich panels greatly increases the difficulty of topology optimization design. To carry out accurate and efficient optimization design work, it is necessary to adopt a suitable equivalent method to equate the honeycomb sandwich panel to a single-layer continuous solid material with similar macromechanical properties. Three-point bending tests and finite element simulations are used to verify the accuracy of the equivalent method. In this way, in actual engineering applications, the complex structure of the honeycomb sandwich panel is effectively avoided, and the accuracy of the optimized design is ensured at the same time.

In this paper, we develop a special graphic modeling interface for hexagonal honeycomb structures by using Python to redevelop the user graphical user interface (GUI) of ABAQUS, through which the actual hexagonal honeycomb structure model is created. Then, an equivalent model is built based on equivalent theory. Finally, a three-point bending experiment of real materials was carried out and compared with finite element simulation three-point bending analysis of the two models to verify the sandwich theory. The topological optimization design of the rib is carried out based on the equivalent model of the honeycomb sandwich composite, which is established by finite element modeling software. Then, after the finite element analysis of a complete wing structure, a prototype is made for the flight test, according to the optimization results, to prove the feasibility of this method.

2. Theoretical Deduction and Calculation

2.1. Equivalent Theory of Sandwich Panel. In early research, to simplify the analysis theory, some in-plane stiffness of the honeycomb core was often ignored, which led to a gradual increase in equivalent error. In fact, although the stiffness of the honeycomb core layer is small, it has a relatively large value compared with skin, so the in-plane stiffness should be considered as a necessary factor during equivalent calculation. The main idea of sandwich panel theory is that assuming the upper-lower skin obeys Kirchhoff based on the separation created by the middle honeycomb core, the ability to resist transverse shear stress was ignored. Meanwhile, the honeycomb core can resist the transverse shear stress considering its in-plane stiffness. Based on this theory, the sandwich structure will be equivalent to a homogeneous orthotropic layer with constant thickness. The honeycomb structure is shown in Figure 1.

Formula (1) is the equivalent elastic parameter of the hexagonal honeycomb core based on sandwich equivalent theory:

\[
\begin{align*}
E_x &= E_y = \frac{4}{\sqrt{3}} \left( \frac{\delta}{a} \right)^3 E, \\
G_{xy} &= \frac{\sqrt{3} \lambda \delta}{2 \sqrt{3} a} G, \\
G_{xz} &= \frac{\lambda}{\sqrt{3} a} G, \\
G_{yz} &= \frac{\sqrt{3} \lambda \delta}{2 \sqrt{3} a} G, \\
\nu &= \frac{1}{3},
\end{align*}
\]

where \(E\) and \(G\) are the engineering constants of sandwich materials; \(a\) and \(\delta\) are the length and thickness of the honeycomb wall, \(\nu\) is Poisson’s ratio, and \(\lambda\) is the correction coefficient, which varies between 0.4 and 0.6.

2.2. Theoretical Analysis of the In-Plane Elastic Constants of the Honeycomb Core. The elastic constants of a honeycomb core [13], such as flat compression elastic modulus \(E_{cz}\), shear modulus \(G_{cz}\), and \(G_{cy}\), are basic parameters for product design of honeycomb sandwich structures, and they are often used. However, the in-plane elastic modulus of honeycomb sandwiches is generally ignored or uses the data of lateral pressure and bending sandwich structures, which are tested experimentally during the process of product calculation (in fact, these data include the elastic performance of the honeycomb core). To obtain the in-plane elastic properties of a honeycomb core, this work used the static and deformation methods to deduce the theory and to perform theoretical derivation. The value ranges of other in-plane elastic constants of the hexagonal honeycomb core are given.
as follows:

\[
\begin{align*}
0.35 \left( \frac{t}{c} \right) E_s & \leq E_{cx} \leq 0.96 \left( \frac{t}{c} \right) E_s, \\
0.35 \left( \frac{t}{c} \right) E_s & \leq E_{cy} \leq 0.58 \left( \frac{t}{c} \right) E_s,
\end{align*}
\]

where \( E_{cx} \) and \( E_{cy} \) are the upper and lower limits of the elastic modulus of the honeycomb core, respectively.

\[
\begin{align*}
\frac{\sin \theta \cos \theta}{\cos \theta} & \leq \mu_{xy} \leq \frac{\sin \theta (\sin \theta + (t/c))}{\cos \theta (\cos \theta + (t/c))}, \\
\frac{\sin \theta \cos \theta}{(d/c) + \cos^2 \theta} & \leq \mu_{yx} \leq \frac{\cos \theta (\cos \theta + (d/c))}{\sin \theta (\sin \theta + (t/c))},
\end{align*}
\]

where \( \mu_{xy} \) and \( \mu_{yx} \) are the upper and lower limits of Poisson’s ratio of the honeycomb core.

\[
0.14 \left( \frac{t}{c} \right) E_s \leq G_{xy} \leq 0.65 \left( \frac{t}{c} \right) E_s,
\]

where \( G_{xy} \) is the upper and lower limits of the in-plane shear modulus of the honeycomb, \( t \) is the thickness of the honeycomb wall, \( c \) is the side length of the honeycomb grid, \( d \) is the side length of the honeycomb grid adhesive strip, \( \theta = 60^\circ \) in the positive hexagon honeycomb, \( d = c \), \( E_s \) is the elastic modulus of the honeycomb wall, and the result takes the average value of the upper and lower limits during calculation.

### 3. Finite Element Modeling of Honeycomb Sandwich Panel

#### 3.1. Material Selection and Calculation of Equivalent Parameters

In this paper, the honeycomb sandwich panel used in the wing rib of a UAV is selected to make samples. The sandwich core is made of Nomex (aramid paper) NRH-2-48 (0.05), and the upper and lower panels are made of 0.22 mm-thick DAN1208 carbon fiber woven fabric. The size and specification of the sample are shown in Table 1.

The performance parameters of the materials are shown in Table 2.
In Table 2, $E_1$ and $E_2$ represent the elastic modulus of materials in the $X$ and $Y$ directions, respectively, $G_{12}$, $G_{13}$, and $G_{23}$ are the shear modulus of materials in the XOY, XOZ, and YOZ planes, respectively. Equivalent parameters for honeycomb cores are shown in Table 3.

3.2. Model Establishment. The honeycomb sandwich panel is an anisotropic material, and its size in the length and width directions is much greater than that in the thickness direction, so the in-plane bending moment and stiffness are ignored in the calculation process. In practical applications, the sandwich panel mainly bears the shear load vertical to the skin. The load of the finite element model and the three-point bending test are also along the shear direction. The loading position was at the centerline of the sample piece, and the loading rate was set as 0.2 mm/min. To ensure consistency between the simulation and experiment, the finite element model is built to the same size as the standard experimental sample piece, whose length and width are 140 mm × 15 mm, and the span is 70 mm. The stress diagram is shown in Figure 2.

Because there is no model library of honeycomb structures in ABAQUS software, a special honeycomb structure modeling interface is developed using Python. During the modeling process, only relevant parameters are input according to user needs, and the honeycomb model will be generated automatically. The whole model is built by three-dimensional shells, and its operation interface is shown in Figure 3. Now, using ABAQUS software, the equivalent model and the actual sandwich core model are established as follows.

The equivalent model of the honeycomb sandwich panel is a single shell, which is divided into three layers. In addition, each layer is given different materials. The upper and lower layers are carbon fiber woven fabric of DAN1208 with a thickness of 0.22 mm; the middle layer is the equivalent sandwich core with a thickness of 2 mm. The final model is shown in Figure 4.
Different from the equivalent model, the actual model needs to model the upper and lower skins and sandwich cores. The upper and lower skins are thin shells, and the middle core is a honeycomb structure. The model interaction adopts merge fusion. The final model is shown in Figure 5.

3.3. Finite Element Simulation Analysis of Two Models. The load is applied to the finite element model as the predesigned method with a rate of 0.2 mm/min. After carrying out the mechanical analysis and calculation of the models by the ABAQUS finite element software mechanical analysis module, the calculation results are read by the ABAQUS postprocessing visualization module, and the stress $s$, displacement $u$, and strain $e$ of the model are displayed in a cloud chart. To better compare the results of the cloud chart after analysis, the magnification factor of the cloud chart is set to 2.

Figures 6, 7, and 8 show the analysis results of the two models. Figure 6 shows the stress, strain, and displacement nephogram of the sandwich equivalent model. Figures 7 and 8 show the nephograms of the stress, strain, and displacement of the actual honeycomb core substructure model from different perspectives. The purpose of this paper is to analyze the stress and deformation of the two models after loading, so there is no failure comparison analysis.

Figures 6, 7, and 8 show that the stress and strain nephograms of the equivalent model and the actual model including the honeycomb core structure show the same trend and location distribution characteristics under the same external load. From the cloud image of both, it can be seen that the maximum stress is concentrated in the area of load application. Due to the different modeling approaches, the equivalent model is a plate and shell model, which directly shows the stress concentration position, while for the actual model,
the value and position of stress on the upper and lower panels and honeycomb core are different. Oiwa et al. and Sun et al. studied the physical properties of honeycomb sandwich panels after bending loads [21–24]. Experiments have shown that honeycomb sandwich panels may undergo panel buckling, honeycomb shear, and adhesive debonding under bending loads. However, as the thickness of the honeycomb layer decreases, the characteristics of the actual honeycomb sandwich panel and the equivalent material model are closer. The simulation results in this paper also proved this finding.

Figures 9, 10, and 11 show the stress distribution nephograms of the upper and lower panels and honeycomb core of the actual model. The stress is mainly concentrated in the lower plate of the sandwich structure, and the maximum stress of the middle honeycomb core is relatively small, which is related to the force transmission structure and energy transmission mode of the honeycomb core. This shows that the honeycomb structure has the effect of releasing energy, which reflects the superiority of the honeycomb structure.

It can be seen from the results shown in all the cloud charts that the maximum stresses of the two models are 127.3 MPa and 122.4 MPa, and the maximum displacements are 3.71 mm and 3.58 mm, respectively. The maximum stress and the maximum displacement share the same order of magnitude in the data, and the values are relatively close. The stress concentration area of the actual model is in the same position as in the equivalent model, and the displacement deformation cloud chart also has high consistency. This shows that the sandwich equivalent model can be used to simulate the performance of honeycomb sandwich structures.

4. Three-Point Bending Test

A three-point bending experiment of honeycomb sandwich structure samples made according to the relevant...
requirements of China national standards was designed, and the loading diagram is shown in Figure 12. There were 5 groups with 3 samples in each group, and the test span was 70 mm. The material’s fracture limit data can be tested by performing a three-point bending experiment on samples with a microforce material testing machine named INSTRON-5848.

During the test, the sample is restrained by two-point support, and the loading mode is set as displacement loading. In this work, the loading rate is 0.2 mm/min. When placing the samples, it is necessary to mark the position of the samples in advance to prevent the samples from generating an eccentric bending moment due to a deviation in position on the support seat. The tested sample is shown in Figure 13.

The stress, strain, and displacement of each group of samples were recorded in time during the experiment, and the average value of each set of test data was calculated. All the data are shown in Table 4.

### 5. Comparative Analysis of Simulation Data and Experimental Data

The simulation analysis results of the two finite element models in Section 3.3 are compared with the three-point bending test data, and the error is calculated. The maximum stress and maximum displacement comparison data are shown in Table 5.

<table>
<thead>
<tr>
<th>Sample number</th>
<th>Maximum stress/MPa</th>
<th>Maximum strain</th>
<th>Displacement/mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>A1</td>
<td>125.97</td>
<td>1.14</td>
<td>3.38</td>
</tr>
<tr>
<td>A2</td>
<td>114.94</td>
<td>1.22</td>
<td>3.42</td>
</tr>
<tr>
<td>A3</td>
<td>122.84</td>
<td>1.22</td>
<td>3.52</td>
</tr>
<tr>
<td>B1</td>
<td>118.23</td>
<td>1.1</td>
<td>3.37</td>
</tr>
<tr>
<td>B2</td>
<td>114</td>
<td>1.1</td>
<td>3.46</td>
</tr>
<tr>
<td>B3</td>
<td>127.36</td>
<td>1.18</td>
<td>3.66</td>
</tr>
<tr>
<td>C1</td>
<td>112.83</td>
<td>1.21</td>
<td>3.41</td>
</tr>
<tr>
<td>C2</td>
<td>115.64</td>
<td>1.05</td>
<td>3.48</td>
</tr>
<tr>
<td>C3</td>
<td>118.33</td>
<td>1.14</td>
<td>3.42</td>
</tr>
<tr>
<td>D1</td>
<td>115.37</td>
<td>1.15</td>
<td>3.31</td>
</tr>
<tr>
<td>D2</td>
<td>122.72</td>
<td>1.01</td>
<td>3.52</td>
</tr>
<tr>
<td>D3</td>
<td>115.23</td>
<td>1.19</td>
<td>3.56</td>
</tr>
<tr>
<td>E1</td>
<td>115.83</td>
<td>1.1</td>
<td>3.47</td>
</tr>
<tr>
<td>E2</td>
<td>104.74</td>
<td>1.07</td>
<td>3.39</td>
</tr>
<tr>
<td>E3</td>
<td>122</td>
<td>1.24</td>
<td>3.42</td>
</tr>
<tr>
<td>Average</td>
<td>117.4</td>
<td>1.14</td>
<td>3.43</td>
</tr>
</tbody>
</table>

### Table 5: Comparison of FEM simulation results and experimental results for three-point bending.

<table>
<thead>
<tr>
<th>Title</th>
<th>Maximum stress/MPa</th>
<th>Error</th>
<th>Maximum displacement/mm</th>
<th>Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sandwich equivalent model</td>
<td>127.3</td>
<td>8.43%</td>
<td>3.71</td>
<td>8.1%</td>
</tr>
<tr>
<td>Actual model</td>
<td>122.4</td>
<td>4.26%</td>
<td>3.58</td>
<td>4.3%</td>
</tr>
<tr>
<td>Three point bending experiment</td>
<td>117.4</td>
<td>/</td>
<td>3.43</td>
<td>/</td>
</tr>
</tbody>
</table>

From the data in Table 5, it can be seen that the maximum stress error of the two models is 8.43% and 4.26%, and the maximum displacement is 8.1% and 4.3%, after comparing the results of the equivalent model, actual model, and three-point bending experiment. All the errors are within 10%, which may be caused by the difference between the model and the real materials, the environment and other factors not considered. Table 5 also shows that the data of the equivalent model are higher than those of the actual model, which is related to the different structures, force transmission modes, and hexagonal side lengths of the two models in the simulation analysis. However, the data of the two models are very close, which shows the effectiveness of the finite element simulation model. Meanwhile, the results reflected by the equivalent model and the materials experiment are basically the same, which verifies the accuracy of the equivalent model. Overall, these results show that in later modeling, the equivalent model can be used instead of the real model for finite element analysis to reduce the modeling workload and improve efficiency, within a certain accuracy range.

### 6. Topology Optimization of Wing Ribs

Topology optimization is a type of structural optimization, which is an important means of lightweight design for wing structures. Its basic idea is to transform the topology
problem of seeking the optimal structure into the distribution problem of seeking the optimal material in the design space. The best distribution form of the material, that is, the optimal topology, can be found by, respectively, modifying and iterating the finite element model, which help the structure with a uniform material distribution to rearrange or delete elements. The variable density method is a common topology optimization algorithm that evolved from the homogenization method. Compared with the homogenization method, the variable density method has fewer design variables, simpler optimization procedures, and higher efficiency of optimization calculations, which lead to more suitability for solving practical engineering problems.

This work’s topology optimization of the wing rib will be carried out based on the variable density method. The main idea of the variable density method is to divide the structure into a finite number of small units to establish the density function relationship between the unit density and other material characteristics of the structure, such as modulus of elasticity, stress, and other parameters. This relationship is applicable assuming that the unit density changes between 0 and 1 and that the internal density of the unit is the same. Then, finding the optimal transmission path becomes a recombination problem of discrete variables. According to the influence of the element on the transmission path, the
Finally, the optimization algorithm is used to solve the problem. Finally, the elements with densities of 1 and 0 will be retained, and the topological structure including the "reduction hole" will be obtained.

According to the CATIA aerodynamic shape parameters of the provided wing, the finite element model of the wing rib is established as shown in Figure 14, which has a total of 13778 elements after meshing with 4-node elements. The final model is complete after applying the structural joint force load transformed from the aerodynamic joint force load to the model, setting the wing root fixed as the boundary condition, and taking the equivalent material parameter in Section 2.1 as the material attribute of the wing. During the topology optimization process, each cell is a microstructure representing the actual structure, and there is a one-to-one corresponding material and cell attribute for each cell. In each subregion, 15 elements will be deleted in each iteration step, and the remaining volume ratio is 0.4. The final rib topology is obtained after 37 cycles of optimization iteration, and the optimization process is shown in Figure 14.

Figure 15 shows the change process of the strain energy of the wing rib, and Figure 16 shows the optimization histories. From the chart, we can see that in the first optimization cycle, the area where the rib is deleted is mainly concentrated in the upper part; at this time, the strain energy is the largest, and the bearing capacity of the rib is the lowest. Starting from the 15th cycle, the strain energy of the rib decreases in a jumping manner. During the process of finding the optimal distribution of the lightening holes, the elements with small loads are deleted, while the areas with large loads are retained. By the 30th optimization cycle step, the strain energy response curve has reached a stable value, and the volume of the lightening hole of the rib plate can be obtained. With the optimization of the reducing hole distribution, the stiffness of the rib tends to be stable, and the strain energy converges in the 37th optimization cycle. As a result, the rib has the greatest ability to resist deformation under aerodynamic.

Because the external load of the wing skin is transferred to the wing beam through the wing rib, and the force is transferred between the positioning hole of the wing rib and the beam, these areas must be preserved in the rib reduction design to bear the load. Elements with densities less than 1 are deleted because these areas have little influence on the bearing capacity of the rib and will not affect the ability to support and ensure the surface form of the skin. After topology optimization, the weight of the rib is effectively reduced while ensuring the stiffness of the rib to reduce the overall weight of the wing structure. From the result of calculation, the weight of a single topology rib is 33.3 g, which is to say that the total weight is reduced by 51.3% compared with the initial weight of 68.4 g, which greatly improves the utilization rate of materials.

7. Static Analysis of Wing Structure

The finite element model of the solar UAV wing with a double beam structure and ribs is established in ABAQUS, and the model includes 13 ribs, all of which are optimized topologies. The final finite element model is shown in Figure 17. The value of the load is set as 1.5 times the rated load, which is applied on the wing surface, and its boundary conditions are added at the wing root. Then, the analysis result can be obtained by analyzing the statics of the wing with ABAQUS. The stress and deformation cloud chart is as follows Figures 18 and 19.
From the simulation results, it can be seen that the structural load of the wing is mainly concentrated near the wing root, and the utilization rate of other parts of materials is small. The displacement and deformation diagram of the wing shows that the maximum deformation of the wing occurs at the wing tip, and its maximum value is 165.9 mm, which is far less than the maximum deformation required by the design of 350 mm. To summarize, the wing structure is not damaged because the stress and maximum displacement of the wing are within the design requirements.

8. Case Application

Because of the uneven distribution of the relief holes in the ribs after topology optimization, there are many serrated edges, which need to be smoothed, and the wing rib mainly bears the shear direction force, which is very small. In addition, the rib of the topology structure is not suitable for the installation of avionics systems because the area between the two sparholes is divided into three parts. Therefore, according to the practical characteristics of the UAV and the relief hole distribution of the wing rib structure obtained by topology optimization, the secondary design of the wing rib is carried out from the structural stress and process implementation, and the real composite wing rib of the honeycomb sandwich structure is made as shown in Figure 20. Figure 21 shows us a physical prototype of the wing, which includes the secondary design of the wing ribs. Then, the actual flight test of the wing is carried out on the prototype of the wing. The first flight time is 16 hours and 9 minutes with a stable flight condition, and the wing deformation is within the design requirements. The experimental results verify the feasibility of this scheme from the perspective of practical engineering applications. Figure 22 shows the actual flight test of the solar UAV.

9. Conclusion

In this work, the honeycomb core is equivalent to a homogeneous orthotropic layer with constant thickness by using sandwich equivalent theory, through which the elastic constant of the equivalent layer is calculated. It is found that the data of the equivalent model and the actual model have a high consistency according to the
comparison between the two simulation results, and the accuracy of the sandwich equivalent model can also be verified with the comparison result of the equivalent model and the real material experiment.

This shows that the sandwich equivalent model can be used as a simplified model of composite honeycomb sandwich structures for finite element simulation in a certain range of progress, and this method can also provide a reference for the subsequent modeling and simulation analysis of other honeycomb sandwich structures.

An equivalent model of a solar UAV wing rib structure is established based on the equivalent parameters, and, then, the rib topology, including lightening holes, is obtained after topology optimization by using the variable density method. After optimization, the weight of a single wing rib is 33.3 g, a reduction of 51.3% compared with the initial weight of 68.4 g.

A real test flight is carried out for the UAV, which includes the topology of the secondary designed ribs. The maximum deformation of the wing during flight is within the design requirements, which shows that the wing structure containing composite materials is optimized based on sandwich equivalent theory, and the feasibility of this method is verified again.

**Data Availability**

The data used to support the findings of this study are included within the article.

**Conflicts of Interest**

The authors declare that they have no conflicts of interest.

**Acknowledgments**

This research was supported by the Key R&D Projects in Shaanxi Province (S2021-YF-YBGY-1244).

**References**


