Article

Comparison of Manual Setting Weight Reduction and Topology Optimization of the Wing Tips of Electric Vertical Take-Off and Landing Aircraft

Yangyang Zeng 1, Jiayi Li 1, Shiyun Lin 1,2,3, Xiaolong He 1, Bing Li 4 and Tao Deng 1,2,3,*

1 School of Aeronautics, Chongqing Jiaotong University, Chongqing 400074, China; 631962010301@mails.cqjtu.edu.cn (Y.Z.); 631862010212@mails.cqjtu.edu.cn (J.L.); lsy718manu@cqjtu.edu.cn (S.L.); 631962010309@mails.cqjtu.edu.cn (X.H.)
2 Chongqing Key Laboratory of Green Aviation Energy and Power, Chongqing 401120, China
3 The Green Aerotechnics Research Institute of Chongqing Jiaotong University, Chongqing 401120, China
4 School of Mechatronics and Vehicle Engineering, Chongqing Jiaotong University, Chongqing 400074, China; 631904090106@mails.cqjtu.edu.cn
* Correspondence: d821722@cqjtu.edu.cn

Abstract: Urban air mobility aircraft are considered an effective solution to traffic jams. These particular aircraft come with different designs and are very different from traditional aircraft. An effective weight reduction method is required to enable ongoing design validation and verification. This paper presents the design of an aircraft with a high space utilization rate of the take-off and landing stage, which allows vertical take-off and landing and has long battery life. For the components fixed at the wing tips and used for installing ducted fans and based on the fused deposition modeling (FDM) process, this paper puts forward two methods to carry out the lightweight design according to the stress conditions of the aircraft, namely, the manual setting the internal structure and the topology optimization. The results show that when the force on the component is small, manual setting is very effective for the purpose of weight reduction, which can also improve the manufacturability of the aircraft.

Keywords: eVTOL; lightweight; topology optimization; additive manufacturing

1. Introduction

With the development of the world economy, traffic pressure has led to time being wasted and economic loss, as well as serious environmental problems. The development of three-dimensional urban transportation is becoming an important solution to these problems, which are a concern in all countries. NASA and other institutions have put forward the concept of electric vertical take-off and landing aircraft (eVTOL) [1]. eVTOL combines the advantages of both rotorcraft and fixed-wing aircraft, in which the arms of the rotor are set to a certain airfoil shape, and at the same time the power unit is placed at the wing tip. Meanwhile, it also has the vertical take-off and landing function of rotorcraft and uses the wings to generate lift in the horizontal flying stage [2,3]. Design optimization of the lightweight wing structure of aircraft and wing structure strength involving structure performance and material distribution are the key to improving the vertical take-off thrust-to-weight ratio. Not only can the shape requirements of key components be met and the weight reduced, but it is also more consistent with structural stress characteristics and stress distribution uniformity [4]. Currently, there are two main weight optimization methods: manual setting (based on intuition and experience) and topology optimization.

A series of classic and breakthrough designs have emerged based on past experience. Data and experience-based manual optimization is quite useful to obtain quick results on some common components with low stress. However, the limitation is that the experience cannot be easily inherited by subsequent researchers, and it does not perform as well in
Among all flight states of eVTOL aircraft, the stress condition of the vertical take-off and landing stage is the most complex. The duct is connected to the wing, and it is necessary to design a subsumption component so that a stable and effective connection between the ducted fan and the wing can be established. The design should meet the strength and stiffness requirements [7], as under working conditions, the component mainly undertakes the thrust and torque transmitted from the ducted fan, and then undertakes the gas load, acoustic load, and bending torque between itself and the wings. This paper is focused on the weight reduction of the wingtip component in the design and verification process of a small electric vertical take-off and landing aircraft (eVTOL), and makes a comparison between manual setting and topology optimization calculation of the internal structure. In terms of the manual setting structure, based on the strength characteristics of different internal filling structures of mechanical samples by additive manufacturing and according to the stress characteristics obtained from simulation, the internal structure of the wing tip was changed using a Boolean operation in 3D software so that the balance between the strength and weight of the component could be achieved. In terms of topology optimization, by continuously refining the mesh size, iterative calculation was conducted at different mesh sizes, and finally a lightweight model with the maximum stiffness was obtained. Focusing on this special stage of product design and modification, a method with higher efficiency, simpler calculation, and easier implementation will better meet the needs of the process to accelerate the manufacturing and respond rapidly to the market.

2. Design of eVTOL

The aircraft independently designed by our team is an eVTOL aircraft with a tandem wing layout and folding function, which has the advantages of a small floor area requirement and high flight efficiency. The structure is shown in Figures 1 and 2.

1. T-tail
2. Fuselage
3. Thrust power unit
4. Lifting power unit
5. Wing
6. Wing rod head

Figure 1. Structure diagram of the aircraft.
For low-speed aircraft, the main tail layouts are ‘+-shaped’ and ‘T-shaped’ layouts. Compared to the +-shaped tail, the T-shaped tail has higher control efficiency and better heading stability when the wing stalls [8]. After comprehensively considering the interference between the wing and tail when the aircraft is folded and the maneuverability at low speed, the T-shaped tail is used in this paper.

A lightweight structure was used in the fuselage to ensure smaller take-off and landing space. The folding and unfolding device is driven by servos. The dynamic layout is a tandem wing layout, the lift-drag characteristics of which change with the relative position and shape of the front and posterior wings [9]. Different layout schemes have different advantages and disadvantages, as shown in Table 1.

| Layout Scheme                      | Advantages                        | Disadvantages                                      |
|-----------------------------------|-----------------------------------|----------------------------------------------------|
| Both wings up                     | Large lift–drag ratio and good stability | The drag difference between the front and posterior wings is large and the efficiency is low |
| Both wings down                   | Low lift–drag ratio fluctuation    | Small lift–drag ratio                              |
| One wing up and one wing down     | Maximum lift–drag ratio           | Relatively poor stability                           |

The thrust power plant and lift power plant are electric power systems that combine batteries, motors, propellers or ducted fans, and control systems [10]. The thrust power unit uses the forward-pulling layout of the fuselage so that the lift coefficient is larger at a high angle of attack, which is beneficial in providing larger lift at low speed [11]. Vertical lift devices mainly include multi-rotor and ducted fans to reduce the noise and size of the aircraft and improve its efficiency. To make the wing structure more compact, ducted fans were placed at the wing tips to achieve vertical take-off and landing.

In this paper, the components used to install ducted fans at the wing tips were studied. The study aims to reduce the weight of this component with minimal impact to its original shape and strength. Meanwhile, it was also expected that the weight loss process would be simple, fast, and easy to operate. Two commonly used lightweight schemes were formulated: manual setting honeycomb structure weight reduction and topology optimization calculation.
3. Weight Reduction Method of Manual Setting

The wing rod head is a structural component connecting the wing and the lift power unit. Under a vertical take-off and landing condition, the component bears the concentrated load caused by the lift, which causes serious stress concentration at the uneven transition site of the component. Meanwhile, the cyclic change in lift with time induces fatigue failure of the component. Therefore, the design of the component should not only guarantee its structural strength, but also meet the requirements of lightweight design.

In this paper, the detailed process of manually setting the filling structure to reduce weight refers to the stress characteristics obtained in the simulation calculation and the strength characteristics reflected by the mechanical test samples with different manual setting filling. The solid part is extracted into a hollow thin-walled part in the three-dimensional software, and then a structure with more balanced weight and strength in the experiment is used to carry out a Boolean operation to reconstruct the internal structure of the component to achieve the purpose of weight loss. The process is based on certain test data and experience, which needs to be verified by static simulation again. This process relies on previous simulation and mechanical testing, so the result must be verified by simulation again to ensure that the optimized part can still meet the strength requirements. The process is shown in Figure 3.

Figure 3. Manual lightweight method.

3.1. Finite Element Analysis

On the premise of achieving calculation accuracy, the model was simplified, and the following assumptions were made: (1) The influence of the surface quality of the model in FDM manufacturing process was not taken into consideration [12], (2) the model was isotropic, and (3) the small-hole and small-size structure in unimportant regions were neglected.

The rod head of the wing tip of the aircraft is a model in the shape of a wing tip, on which two connecting holes are made for installing the shafts connected to the wing, and the middle hole is for the installation of ducted fans. PLA material parameters in the ANSYS material library were used. The model uses Solid 187 elements. After various trials,
starting from the default mesh size, the global mesh was gradually reduced by 1 mm each time, and finally adopted a global 3 mm mesh size. Tetrahedrons meshing method. To guarantee the convergence of the calculation results, local mesh refinement was conducted at the edge of the front hole after verification and analysis. Under this mesh configuration, there was a total 50,555 elements and 77,501 nodes, and the results are shown in Figure 4.

![Mesh of the model.](image)

**Figure 4.** Mesh of the model.

The two mounting holes on the left side of the component are used to connect to the wing, and remote displacement is set on each hole. The translational and rotational degrees of freedom along the X, Y, and Z axes were all set to 0. The behavior was set to deformable, as shown in Figure 5a.

![Boundary conditions of the model.](image)

**Figure 5.** Boundary conditions of the model. (a) Freedom constraint; (b) force setting.

Gravity was set according to the force on the component in operation. The maximum thrust parameter of the purchased electric duct is 9.8 N, so the tension was set to 9.8 N on the duct mounting hole. On the inner wall of the mounting hole, the moment brought by the rotation of the duct should be added. The true value of this moment is complicated to measure or calculate and is not the focus of this paper. To be on the safe side, a much larger moment was used than the actual torque of 6000 N·mm [13], as shown in Figure 5b.

The simulation results are shown in Figure 6a. The simulation results show that the maximum stress was 3.18 MPa at the part connecting to the wing. The stress condition at the mounting location of the duct was more complicated under actual working conditions. Therefore, special attention should be paid to the stress condition in this location, as shown.
in Figure 6b. The maximum stress occurred at a location of 25 degrees outward along the axis, as shown in Figure 6c.

Figure 6. Simulation results. (a) Stress simulation results; (b) stress of the duct installation hole; (c) the relative positions of maximum stress.

3.2. Additive Tensile Test

The main concept of additive manufacturing (AM) is to change the 3D manufacturing into the ordered arrangement of 2D planar structures with thickness. The model of the part is cut into slices, and the part is manufactured by stacking the slices layer by layer. This process mainly depends on computer-aided design and control technology [14]. Additive manufacturing technology has several advantages, including a wide range of materials, low cost, fast molding speed, and the capacity to manufacture complex structural parts more easily. In particular, a very simple way to manufacture the internal structure of the closed cavity is suitable for exploratory research at the initial stage of product design, which can reduce waste of expensive materials [15].
The principle of FDM technology is to extrude consumables into the heated region, which are extruded from the nozzle after being heated and melted. With the horizontal movement of the nozzle on the machining platform, the preset cross-sectional structure of each layer is printed. With the vertical movement of the machining platform, the cross-sectional structure grows continuously, and finally, the required parts are manufactured. The principle is shown in Figure 7.

In the above-mentioned processing, the printing quality is significantly influenced by the interlayer thickness, the diameter of the material extruded by the nozzle, the temperature of the heating platform and heating part for the nozzle, the speed of the feeding roller shaft, and the movement speed and accuracy of the nozzle and platform [16]. The printer has a simple structure and is easy to operate, and can use many types of materials that can be recycled later on. Given these advantages, this additive process has gained a lot of attention and is widely applied [17]. Meanwhile, with the continuous expansion of application fields, the micro FDM manufacturing process and large-size FDM manufacturing process are constantly developing and being put into application.

The total length and overall thickness of the test samples were 170 mm and 4 mm, respectively, and the length and width of the parallel part were 55 mm and 10 mm, respectively, as shown in Figure 8a. To reduce the influence of the clamping part on the mechanical properties of the whole sample component, only different fillings were set in the middle parallel part during printing. Two filling regions of 55 mm in length, 8 mm in width, and 2 mm in height were symmetrically dug on both sides of the parallel region, as shown in the blue region in Figure 8b. Five types of fillings were used, namely, solid, hollow, slant cross, cross, and slant, as shown in Figure 8.
Figure 8. Geometric structure of the test sample. (a) Size of solid sample; (b) hollow; (c) slant cross; (d) cross; (e) slant.

CURA was used to slice the model of the sample component to generate the printing code, and a 3D printer was used to print the model. The printing parameters set in the software are shown in Table 2.

Table 2. Printing parameters.

| Variable                        | Parameter          |
|---------------------------------|--------------------|
| Material for printing           | PLA                |
| Layer thickness                 | 0.1 mm             |
| Extrusion temperature           | 205 °C             |
| Hot bed temperature             | 55 °C              |
| Printing speed                  | 60 mm/s            |
| Number of layers of outline     | 2                  |

A tensile test was carried out after the drawing and additive manufacturing, as shown in Figure 9.

Figure 9. Tensile test of the test sample. (a) Tensile test; (b) unused sample; (c) used sample.
After the test was completed, the invalid test data were removed, and the tensile strength and other characteristics of each sample component were obtained by polynomial fitting. At the same time, the proportion of different filling types was calculated according to its shape. Here, the filling rate was defined as the ratio of the volume of the intermediate filling region to the solid volume of the sample component:

\[
D = \frac{V_{\text{feature}}}{V_{\text{solid}}}
\]  

(1)

As can be seen from Figures 10 and 11, when the sample component was hollow, its tensile strength was only 22.89 MPa, which was only 34.09% of when it was solid. The tensile strength was 38.44 MPa when the cross structure was used as the filling, which was 57.25% of when it was solid. The tensile strengths of slant cross filling and slant filling were close, at 69.03% and 67.46%, respectively. It can be seen from the test data that the strengths of slant and slant cross filling were close to solid filling, and the filling rates of the sample component were 11.28% and 21.62%, respectively.

It was found that slat filling method achieved higher strength in comparison to the hollow structure by 97.86%. Therefore, the slant honeycomb filling structure should be

![Figure 10. Tensile stress–strain curves of samples.](image1)

![Figure 11. Tensile strength of sample components with different fillings.](image2)
used for the wingtip rod head at an oblique 45° along the orthogonal direction, where the stress is large to achieve a better filling effect.

3.3. Optimization Results

Based on analysis and calculations, there was a stress concentration on the wingtip mounting hole that connects to the wing. The installation region of the ducted fan was complicated. Apart from the reaction of the thrust and additional torque of the ducted fan in the simulation, there were also forces from vibration and aerodynamic interference. Thus, special attention should be paid to the region with complex forces. After comprehensively considering installation and manufacture, the honeycomb filling structure was used with the end of two holes connecting the component with the wing as the interface, along the direction of maximum stress and the direction of 45°, as shown in Figure 12. As a result of the optimization, the weight of the component was changed from 90.53 g in solid state to 34.42 g. Under this optimization scheme and compared with the solid condition, the weight loss was 61.98%. Compared with the hollow condition with interface, the extra weight of the honeycomb filled structure was 7.46 g.

Three kinds of meshes were established—4 mm, 3 mm and 2 mm—and the maximum variation of the equivalent stress was less than 1.5% under these conditions. Among them, at the 3 mm mesh size, there were 34,959 mesh elements and the number of mesh nodes was 62,934. The stress characteristics of the model at this time are shown in Figure 13. The strength conditions in the test still met the requirements. In an actual situation, however, the stress of the component will be more complex, and the maximum von Mises stress calculated by simulation was obviously lower than the strength capacity of PLA material in the FDM process. Therefore, the objective of weight reduction and structural optimization of the parts was achieved.
where $x$ within the preset safety region through several iterations according to the preset conditions. (Stiffness) was set to 3.

The mathematical model of the problem of material arrangement in engineering [20]. The mathematical model of the design region into finite elements to form a new continuum by removing the regions within the preset safety region through several iterations according to the preset conditions in order to achieve weight reduction [19]. Topology optimization studies the fundamental problem of material arrangement in engineering [20]. The mathematical model of the topological optimization consists of the design variables, the optimized objective function, and the constraint function.

$$\begin{align*}
\{ f = \min f(x) \quad ( x = \{x_1, x_2, \cdots, x_n\} ) \\
\text{s.t.} & \quad g_i(x) \leq 0 \quad i = 1, 2, \cdots, I \\
& \quad h_j(x) \leq 0 \quad j = 1, 2, \cdots, J \\
& \quad x_{KL} \leq x_k \leq x_{LU} \quad k = 1, 2, \cdots, n
\end{align*}$$

(2)

where $x = \{x_1, x_2, \cdots, x_n\}$, representing that there are $n$ design variables; $f(x)$ is the optimized objective function; $g_i(x)$ and $h_j(x)$ are the constraint functions of the variables; and $x_{KL} \leq x_k \leq x_{LU}$ is the value range of the variables [21].

4.1. Simulation of Topology Optimization

In the ANSYS WORKBENCH analysis platform, the solutions in Static Structural could be passed to the settings of the Topology Optimization module. Meanwhile, the optimization region was set in Mechanical, and the entire model was selected as the design region. Meanwhile, all the outer surfaces of the model were selected as the exclusion region. Weight was set as the response constraint, a material removal rate similar to the manual setting results in previous tests, and the range of the material retention rate was set from 38% to 43%. By setting the range to a small value, a smaller weight of final material based on the optimization results could be more easily obtained. The maximum number of iterations was set to 500. The convergence accuracy was set to 0.1%. The penalty factor (Stiffness) was set to 3.

The results of topology optimization were very significantly influenced by the mesh size. In this paper, the mesh units were gradually refined from the default value. By adjusting the number of meshes, the material retention rate was continuously reduced to achieve better results in weight reduction.

According to the ANSYS help document, maximum stiffness is the goal of topology optimization calculation. Additive manufacturing is the manufacturing method, which
is not sensitive to complex internal shapes. For the topology optimization results, simulation verification is not represented in this paper. However, for some beam structures or thin plates with equal cross-sections, these types of parts are limited by the shape and processing method. The results must be derived for model reconstruction to meet the original requirements of equal thickness. The reconstructed model is verified by simulation calculation again [22].

4.2. Optimization Results

This study used a total of 12 different global mesh sizes, and the results are shown in Figure 14. The optimized mesh size and weight are shown in Table 3.

---

**Table 3. Optimized weight percentages.**

| Mesh Size | Optimized Weight Percentage (%) |
|----------|---------------------------------|
| Default  | 92.43                            |
| 5 mm     | 87.08                            |
| 4 mm     | 81.51                            |
| 3 mm     | 75.12                            |
| 2 mm     | 66.80                            |
| 1 mm     | 50.26                            |

**Figure 14.** Topology optimization results with different global mesh sizes.
Table 3. Optimized weight percentages.

| Mesh Size | Optimized Weight Percentage (%) | Mesh Size | Optimized Weight Percentage (%) |
|-----------|---------------------------------|-----------|---------------------------------|
| Default   | 92.43                           | 0.9       | 64.72                           |
| 5         | 87.08                           | 0.8       | 62.37                           |
| 4         | 81.51                           | 0.7       | 60.26                           |
| 3         | 75.12                           | 0.6       | 57.54                           |
| 2         | 66.80                           | 0.4       | 55.27                           |
| 1         | 66.28                           | 0.3       | 50.26                           |

The optimization results obtained from the software show that with the increase in the number of meshes, the proportion of material that could be removed by the optimization increased. In the 0.3 mm mesh size, the optimized weight percentage was 50.26%; that is to say, 49.74% of the material was removed compared to the original weight of the component. With the refinement of the mesh during the topology optimization of the simulation, it could be seen that the region removed first was the red region shown in Figure 15. Then the blue region occurred and gradually expanded and merged to the red region. In further refinement, some of the material in the green region also started to be removed. It can be seen that the strength of the component with the safest region was the red region.

![Figure 15. Material removal process in topology optimization.](image)

5. Comparative Analysis between the Two Weight Optimization Methods

The design of eVTOL aircraft is still facing many problems, such as too many modifications and new structures. In the process of continuous design, modification, verification, and iteration, a design scheme with quick response and weight reduction is needed to improve the economy and timeliness of the design process to meet the needs of users [23]. A comparative analysis on the manual setting structure and topology optimization for weight reduction from the perspectives of computing time of different meshes, the topology optimization effect brought on by mesh refinement, the required computer hardware, etc., will be significant for the optimization of such a continuum structure.

The calculation results of the components studied by the finite element method change with the refinement of the mesh. When the mesh is continuously refined and the change of the maximum stress value is no longer obvious, it can be judged that the calculation result converges and mesh independence is achieved. As shown in Figure 16, for the calculation of structural statics, starting from the default mesh size, the result of each mesh refinement is less than 5% compared with the previous one. Compared with a 4 mm mesh size, the change of the maximum equivalent stress of the 3 mm mesh size part was only 0.38%. It can be considered that the equivalent stress of the part converged at the 3 mm mesh size, and there was no need to further refine the mesh for more accurate results.

![Figure 16. Maximum equivalent stress of all results from static analysis.](image)
12.24% lower than the manual setting method.

5. Comparative Analysis between the Two Weight Optimization Methods

5.1. Comparison of Elapsed Time with the Same Mesh Density

Based on this component, to carry out a comparison of subsequent mesh refinement, the model used the mesh division method to divide the global mesh and further refined the constraint position. Solid 187 element was used. In the mesh generation step of finite element analysis, the integer or an integer-like method is usually used to refine the mesh size. In this paper, when the mesh was bigger than 1 mm, the global mesh was gradually refined in a 1 mm step value from the default mesh size. For mesh sizes smaller than 1 mm, the global mesh size was gradually refined in a step value of 0.1 mm. A total of 12 types of mesh densities were established using an iterative solver. For the topology optimization calculation, there were 725,215 meshes and 1,232,924 nodes at most. Under this mesh condition, the final optimized material removal rate was 49.74%, which was still 12.24% lower than the manual setting method.

Taking the global mesh size as an independent variable, as shown in Figure 17, and the number of meshes as the independent variable, as show in Figure 18, when it was bounded by a mesh size of 0.7 mm, the total number of meshes was 136,639. Once the mesh size was less than 0.7 mm, the calculation time of the topology optimization simulation increased significantly. Figures 17 and 18 also indicate the difference between static stress calculation and topology optimization calculation in time consumption.

![Figure 16. Maximum equivalent stress of all results from static analysis.](image1)

![Figure 17. Elapsed time of simulation in different global mesh size.](image2)
When the mesh size was greater than 2 mm, the average single iterative calculation time of analysis under 3 mm mesh.

c
der the same mesh size; (c) comparison of topology optimization under all mesh sizes and static analysis under 3 mm mesh.

Figure 18. Elapsed time of simulation in increased mesh elements.

To better present the difference in time consumption between the two simulation calculations under the same mesh size, the time consumption of the two simulations were compared to obtain the time consumption ratio, as indicated in Figure 19a. As shown in Figure 19, compared with static calculation, the time of the topology optimization calculation increased exponentially, and this increasing trend further expanded with the refinement of the meshes.

Figure 19b shows the single iterative calculation efficiency of topology optimization. When the mesh size was greater than 2 mm, the average single iterative calculation time of topology optimization was faster than static calculation. However, when the mesh size was

Figure 19. Time ratio. (a) Time ratio of two optimization methods under the same mesh size; (b) time ratio of average single iterative calculation time of topology optimization to static analysis under the same mesh size; (c) comparison of topology optimization under all mesh sizes and static analysis under 3 mm mesh.

Figure 19b shows the single iterative calculation efficiency of topology optimization. When the mesh size was greater than 2 mm, the average single iterative calculation time of topology optimization was faster than static calculation. However, when the mesh size was
smaller 0.9 mm, the single average iteration time was noticeably longer than the calculation time of the static module. It was also observed from the curve that the average iteration time of topology optimization under the same mesh size was longer than static calculation with the refinement of the meshes, and this trend also continuously expanded.

The 3 mm mesh size was a special size, because the judgment of mesh independence could be concluded at this situation. In practical application, this means that the static calculation was terminated. The static calculation time at this mesh size was compared with all the cases of topology optimization calculation, as shown in Figure 19c. This comparison aimed at analyzing the time of topology optimization and whether it was necessary to further carry out the static calculation. However, the topology simulation still needs further calculation to compare the time consumed. It could be seen that after the 0.7 mm mesh size, the time consumed reached as many as hundreds of times.

Meanwhile, the weight of the members was reduced after the analysis results of the static stress of the structure were obtained. It was only necessary to carry out a Boolean operation with the reinforced filling structure along the direction of the maximum stress after the shelling of the model. Nevertheless, the results of topology optimization were only in STL format, which had a large number of surface triangles, and multiple triangular faces shared vertices. Each vertex needed to be marked in the file, and it was prone to broken surfaces. After using the smooth finite element method and other methods, the number of triangles further increased, which was not conducive to mesh defect detection and repair, subsequent analysis of transformed substances, and mesh division [24]. The model with a triangular surface could not be directly analyzed and calculated in the next step. In view of this, more processing time will be needed for surface reconstruction. To solve the subsequent processing problems of the model of topology optimization, self-adaptive bubble topology optimization with fixed mesh and a topological derivative can be used to avoid operations such as mesh updating and redivision so as to improve the stability of numerical calculation [25].

5.2. The Variation Trend of Topology Optimization Efficiency with Mesh Size

To study the computational efficiency of topology optimization with the changing trend of mesh refinement, the mesh growth rate is defined as \( G_1 \), which means the ratio of the number of elements between two mesh sizes, calculated as:

\[
G_1 = \frac{\text{Mesh}_{i+1} - \text{Mesh}_i}{\text{Mesh}_i}
\]  

(3)

The growth rate of material removal as a result of topology optimization is defined as \( G_2 \), which means the ratio of the volume removed by the material of the topology optimization result between two mesh sizes, calculated as:

\[
G_2 = \frac{\text{Removal}_{i+1} - \text{Removal}_i}{\text{Removal}_i}
\]  

(4)

For the simulated mesh, it is hoped that the growth rate of material removal \( G_2 \) could be as close as possible to the mesh growth rate \( G_1 \), and \( G_2 \) could even be greater than \( G_1 \). This means that the computer resources have been used efficiently. The mesh efficiency \( \eta_M \), which means the ratio of \( G_2 \) to \( G_1 \), can be calculated as:

\[
\eta_M = \frac{G_2}{G_1}
\]  

(5)

By re-arranging the equations of \( G_1 \), \( G_2 \), and \( \eta_M \), Figure 20 was obtained. In general, the process of mesh refinement could improve the material removal rate of topology optimization. However, as mesh is further refined, the increasing speed of removal rate decreases. The 2 mm mesh size had about 20,000 more elements than that of the 1 mm mesh size. In terms of the optimization results, the material removal rate of 1 mm mesh
size was slightly higher than that of the 2 mm mesh size. Figure 20 clearly indicates that the growth rate of material removal from the default mesh size to the 5 mm mesh size and from the 4 mm mesh size to the 3 mm mesh size was higher than the growth rate of the mesh size, that is, $\eta_M > 1$. After the mesh was refined with a size smaller than 1 mm, the material could continue to be removed. However, the optimization gains during this period brought on by the mesh refinement were very limited.

![Figure 20. Utilization efficiency of mesh growth.](image)

### 5.3. Computer Resource Mobilization

The growth of computer computing power means that the meshes can be further refined, which ensures the accuracy of the simulation calculation. Finite element analysis can be divided into three basic processes: pre-processing, solution, and post-processing. An important part of pre-processing is to discretize the model by dividing the meshes. In the starting process of mesh refinement and calculation, not too many computer resources will be occupied, but the calculation accuracy can obviously be improved. However, although the meshes are refined to a certain degree, further refinement will result in significantly higher requirements for computer memory and hard disk space. Moreover, the improvement in the accuracy will not be significant [26]. As shown in Figure 21, this study determined that the maximum memory used in structural statics simulation and topology optimization calculation was basically the same at each mesh size. At a 0.6 mm mesh size, the number of meshes was about 200,000. For a global mesh size greater than 0.6 mm, the memory occupied was basically maintained below 32 GB. However, in the subsequent calculation, the memory occupied surged to more than 128 G.
weight reduction of topology optimization components was about 50%, which was 12% less than the other method (62%). However, the calculation of topology optimization was based on the maximum stiffness, the strength consistency of which was better. Nevertheless, the time consumed with this method was a dozen times higher than static calculation, which requires computing resources that mean that the simulation cannot be done on personal computers. In addition, the use of servers, or even server clusters, often involves time waiting in line.

The results obtained in the topology optimization system are shown in Figure 14. There is still a certain gap between the final weight reduction effects of the two methods. Based on the stress characteristics of the solid component, the weight reduction method of setting the filling combined with the test data was used to eventually obtain components with a lesser weight. Further simulation also proved that the components designed by this weight reduction method can still meet the strength requirements. Using this method, the simulation calculation can use a 3 mm global mesh size without re-refinement, and the computational resources required are small and easy to implement in this case. The final weight reduction of topology optimization components was about 50%, which was 12% less than the other method (62%). However, the calculation of topology optimization was based on the maximum stiffness, the strength consistency of which was better. Nevertheless, the time consumed with this method was a dozen times higher than static calculation, which requires computing resources that mean that the simulation cannot be done on personal computers. In addition, the use of servers, or even server clusters, often involves time waiting in line.

Through ANSYS finite element analysis software, the stress characteristics of the components containing ducts at the wing tip were obtained. Meanwhile, by referring to the specific strength characteristics of the mechanical sample component with different filling structures, the internal structure of the model was reconstructed, as shown in Figure 12. The results obtained in the topology optimization system are shown in Figure 14. There was still a certain gap between the final weight reduction effects of the two methods. Based on the stress characteristics of the solid component, the weight reduction method of setting the filling combined with the test data was used to eventually obtain components with a lesser weight. Further simulation also proved that the components designed by this weight reduction method can still meet the strength requirements. Using this method, the simulation calculation can use a 3 mm global mesh size without re-refinement, and the computational resources required are small and easy to implement in this case. The final weight reduction of topology optimization components was about 50%, which was 12% less than the other method (62%). However, the calculation of topology optimization was based on the maximum stiffness, the strength consistency of which was better. Nevertheless, the time consumed with this method was a dozen times higher than static calculation, which requires computing resources that mean that the simulation cannot be done on personal computers. In addition, the use of servers, or even server clusters, often involves time waiting in line.
Similarly, this happens in various industrial products, and is not limited to the eVTOL mentioned in the paper. In the design stage, there may be not a final design shape. The shape design will constantly change. Designers often put forward brand-new designs different from previous products, which often need sample tests to be verified. Apart from some special positions with greater stress, the capacity of materials in most cases is much higher than the actual stress, the difference of which can even reach one or two orders of magnitude. For these components with a large strength margin whose weight needed to be reduced, it was very efficient to shell the model after static calculation and fill the structure according to stress characteristics. Meanwhile, it had lower hardware requirements. For the final product with a determined shape and by using the topology optimization method with a high mesh density for the calculation, the strength and stiffness of the components were less weakened, indicating that the quality of the product would be better.

7. Conclusions

The paper used ANSYS to carry out the mechanical analysis of the component with a ducted fan installed on the wing tip of the aircraft, and the weight reduction design of the component was carried out using two methods: manual setting and topology optimization. The method of manual setting the filling structure had a better weight reduction effect. The weight-reduction component plays an important role in improving the battery life and loading capacity of the eVTOL.

The FDM additive manufacturing process was used to prepare mechanical test sample components with different filling structures. The results of the tensile tests show that the filling mode with an inclination of 45° along the stress direction could obtain relatively high strength characteristics under the condition of lighter filling.

In terms of time and hardware requirements, both manual setting and topology optimization were compared against each other. The results show that manual setting is more effective and significant in the lightweight design of a continuum structure with small stress.

Author Contributions: Conceptualization, Y.Z. and J.L.; methodology, S.L.; software, J.L. and Y.Z.; validation, Y.Z., J.L. and X.H.; formal analysis, Y.Z.; investigation, J.L.; resources, Y.Z. and T.D.; data curation, J.L.; writing—original draft preparation, Y.Z. and J.L.; writing—review and editing, T.D. and S.L.; visualization, J.L. and B.L.; supervision, T.D.; project administration, Y.Z. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by Key Projects of Technological Innovation and Application Development of Chongqing, grant No. cstc2019jscx-fxydX0028; the Science and Technology Research Key Program of Chongqing Municipal Education Commission, grant No. KJZD-K2020000701; and the Open Project Found of Chongqing Key Laboratory of Green Aviation Energy and Power, grant No. GATRI2021F01001A.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Not applicable.

Acknowledgments: The author of this work thanks Zhiyuan Ma, Liming Huang, and Huaixiang Xiang for their input in the plane manufacturing work. Without their valuable participation, it would not have been possible to complete the eVTOL and write the paper.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Wang, X. The Future of Urban Air Mobility Market. Aerosp. Power 2019, 2, 18–21.
2. Cohen, A.P.; Shaheen, S.A.; Farrar, E.M. Urban Air Mobility: History, Ecosystem, Market Potential, and Challenges. IEEE Trans. Intell. Transp. Syst. 2021, 22, 6074–6087. [CrossRef]
3. Wang, B. Research on Hybrid Vertical Takeoff and Landing Aircrafts. Ph.D. Thesis, National University of Defense Technology, Changsha, China, 2016.
4. Zhao, Y.; Chen, M.; Wang, Z. Additive manufacturing oriented topology optimization of nodes in cable-strut structures. J. Build. Struct. 2019, 40, 58–68. [CrossRef]
5. Li, R.; Zhang, Y.; Chen, H. Evolution and development of “man in-loop” in aerodynamic optimization design. Acta Aerodyn. Sin. 2017, 35, 529–543.
6. Aage, N.; Andreassen, E.; Lazarov, B.S.; Sigmund, O. Giga-Voxel Computational Morphogenesis for Structural Design. Nature 2017, 550, 84. [CrossRef]
7. Wen, D.; Chen, Y. Review and Prospect on The Research of Aero-Engine Casing Dynamics. J. Dyn. Control. 2013, 11, 12–19.
8. Li, P.; Chen, Y.; Wei, W.; Wan, J. Analysis of Effect of Empennage Layout of a Certain Civil Aircraft on Aircraft Stability. Trainer 2014, 45, 28–32.
9. Cao, M.; Chen, B.; Liu, K.; Qin, Z. Research on the Influence of the Installation Position of the Front and Rear Wings of the Tandem-Wing UAV on the Aerodynamic Characteristics. In Proceedings of the 3rd China Aerospace Safety Conference, Zhuhai, China, 28–30 September 2021.
10. Zhao, C.; Chen, L.; Lu, L.; Zhang, Q.; Sun, L. Advances in Electric Aircraft Technology. Sci. Technol. Rev. 2012, 30, 62–70.
11. Ma, J.; Zhang, C.; Li, X.; Chen, A.; Zhang, T. Study on Effect of Propeller Distribution on Aerodynamic Characteristics of Small Tandem-wing UAV. J. Ordnance Equip. Eng. 2020, 41, 54–59.
12. Chohan, J.S.; Singh, R.; Boparai, K.S.; Penna, R.; Fraternali, F. Dimensional Accuracy Analysis of Coupled Fused Deposition Modeling and Vapour Smoothing Operations for Biomedical Applications. Compos. Part B Eng. 2017, 117, 138–149. [CrossRef]
13. Zhao, H.; Li, J.; Cui, Z. Flight dynamics characteristics of miniature unmanned ducted vehicle. J. Aerosp. Power 2014, 29, 1721–1728. [CrossRef]
14. Zhong, W. Study on the Influence of 3D Printing Parameters on the Mechanical Properties and Molding Quality of PLA. Master’s Thesis, Jilin University, Changchun, China, 2019. [CrossRef]
15. Ngo, T.D.; Kashani, A.; Imbalzano, G.; Nguyen, K.T.Q.; Hui, D. Additive Manufacturing (3D Printing): A Review of Materials, Methods, Applications and Challenges. Compos. Part B Eng. 2018, 143, 172–196. [CrossRef]
16. Mohamed, O.A.; Masood, S.H.; Bhowmik, J.L. Optimization of Fused Deposition Modeling Process Parameters: A Review of Current Research and Future Prospects. Adv. Manuf. 2015, 3, 42–53. [CrossRef]
17. Liu, Y.; Xia, C.; Zhang, J.; Jiang, Z. Application Progress and Prospect of Fused Deposition Molding 3d-printing. Eng. Plast. Appl. 2017, 45, 130–133.
18. Gao, W.; Lv, X. Applications of Topology Optimization in Structural Engineering. Struct. Eng. 2020, 36, 232–241. [CrossRef]
19. Yin, Y.; Liu, Y.; Dou, L. Review of researches on structure topology optimization with material nonlinearity. J. Chongqing Univ. 2016, 39, 34–38.
20. Sigmund, O.; Maute, K. Topology Optimization Approaches A Comparative Review. Struct. Multidiscip. Optim. 2013, 48, 1031–1055. [CrossRef]
21. Liu, J.; Zhu, N.; Chen, L.; Liu, X. Structural Multi-Objective Topology Optimization in the Design and Additive Manufacturing of Spatial Structure Joints. Int. J. Steel Struct. 2022, 1–20. [CrossRef]
22. Torres-SanMiguel, C.R. Modeling and Simulation Process via Incremental Methods of a Production-Aimed Upper Limb Prosthesis. Appl. Sci. 2022, 12, 2788. [CrossRef]
23. Man, Y.; Han, J.; Jiang, L.; Chen, B.; Luo, M. Research and Implementation of Layout Rapid Design System for Aerospace Vehicle. Comput. Meas. Control 2020, 28, 182–187. [CrossRef]
24. Ren, N.; Wan, J.; Hu, R. Study on STL File Presentation Based on Topological Triangle. Trans. Chin. Soc. Agric. Mach. 2005, 49, 149–151.
25. Cai, S.; Zhang, W.; Gao, T.; Zhao, J. Adaptive Bubble Method Using Fixed Mesh and Topological Derivative for Structural Topology Optimization. Chin. J. Theor. Appl. Mech. 2019, 51, 1235–1244. [CrossRef]
26. Gu, C.; Wu, X. A review of FEM and trend of development. J. Front. Comput. Sci. Technol. 2008, 2, 248–259.