Research Article

Research on the Optimization of Automobile Plastic Front Frame Structure

Lu Wang, Lingfeng Tang, Peng Wu, and Jiaqi CHEN

School of Mechanical Engineering, Anhui Polytechnic University, Wuhu, Anhui 241000, China

Correspondence should be addressed to Lu Wang; 2210120116@stu.ahpu.edu.cn

Received 30 April 2022; Revised 7 July 2022; Accepted 12 July 2022; Published 16 August 2022

Copyright © 2022 Lu Wang 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.

Plastic instead of steel technology is one of the important means of automotive lightweighting, which can greatly reduce the weight of the car while ensuring the same performance of the product. For the lightweighting of the automotive all-plastic front-end frame structure, the initial structural design of the front-end frame was carried out by HyperMesh; based on the compromise planning method and entropy weight method, the compromise planning method with weight coefficients was used to unify the planning of each subobjective to obtain the integrated response function, and the minimization of the integrated response function was the objective. The main considerations were the lock limit crash load, the cushion area load, and the pedestrian protection impact load under typical operating conditions, the shape optimization, and topology optimization, respectively. The mass of the sheet metal front-end frame was reduced from the original 10.5 kg to 6.8 kg—a weight reduction of 35.24%. The front-end frame was extracted and the mid-plane was repaired by HyperMesh, the complete mid-plane was meshed, loads and properties were added to the mesh, the front-end frame was subjected to static and modal analysis in the OptiStruct module, and the stiffness and intrinsic frequency were verified. The results show that the smallest first-order mode in the modal analysis is 57.38 Hz, which is much larger than the 30 Hz required before design. The fatigue analysis of the front-end frame was then performed using N-code to derive the damage results; it is concluded that its life is 6122 s, much larger than the specified 1200 s. Through the verification of the relevant resonance test, it is judged that the performance of the test part meets the actual requirements comprehensively by checking the status after the test.

1. Introduction

As a very important means of transportation for human beings, the automobile industry has become an industry in many countries. At the same time, the automobile industry also generates a large amount of natural resource consumption, leading to increasingly serious environmental pollution. In response to the negative impact caused by automobiles, all countries have made energy saving and emission reduction an important technical indicator for industrial production. Studies have shown that for every 10% reduction in vehicle mass, up to 13% fuel savings can be achieved over the same distance. By reducing the mass of the car by 300 kg, the exhaust emissions can be reduced by 20% for the same driving distance. The significance of lightweight design on the dynamic performance, whole life cycle cost, energy saving, and environmental protection of the whole vehicle are becoming more obvious. Lightweighting is undoubtedly the future direction of transportation equipment development [1]. There are three main approaches to the application of automotive lightweighting technology: optimization of part structures; use of lightweight materials; and application of advanced forming technologies and manufacturing processes [2]. Topology optimization is a method to obtain the best material distribution in a given design region [3] so as to meet the mechanical performance requirements of a part with the least amount of material, which is widely used in structural concept design, and this method is of great importance for automotive lightweighting.

Lan et al. [4] combined with the multi-body dynamic analysis of the whole vehicle in multiple working conditions, used the compromise planning method to define a comprehensive optimization objective function for maximizing the static stiffness and modalities of the body under the
actual driving conditions of the whole vehicle, and determined the weight coefficients of each subobjective using the hierarchical analysis method. The results show that the stiffness and the first sixth-order frequency of the body have been improved to different degrees and the structure is more reasonable. Wang et al. [5] proposed a topology optimization method that considers the local stiffness of the structure, which can improve the overall stiffness of the support and give consideration to the local stiffness so that the stiffness distribution of the support is more reasonable. Zhang et al. [6] proposed a generalized averaging distance optimization objective function for the multi-objective study the construction method of multi-objective topology optimization design of structural materials. Wang et al. [7] proposed a finite element modeling method for abrasive wheels and long fiber-reinforced ceramic matrix woven composite (LFRCWC) specimens. The method was used to analyze the grinding process of 2.5D woven quartz fiber-reinforced silica ceramic matrix composites (SiO$_2$). The accuracy of the finite element method was verified by relevant grinding experiments. Soheil et al. [8] developed an analytical solution for the electromechanical bending response of a smart-layer piezoelectric composite rectangular plate, which included two flexible spring boundary conditions at opposite edges. Flexible spring boundary structures are introduced into the system by including rotating springs of adjustable stiffness which can vary according to the rotational fixation factor of the spring. The results of the analysis were verified against those obtained with the Abaqus finite element package. The comparison of the results shows good agreement. Qu et al. [9] developed a mathematical model of the optimization problem based on the level set topology optimization method with the maximization of structural stiffness as the objective function and the structural volume as the constraint. Kim et al. [10] proposed a nonlinear structural topology shape optimization scheme based on the artificial bee colony algorithm (ABCA). Xiao et al. [11] proposed a multi-scale topology optimization design method for gradient lattice sandwich structures with excellent structural performance. Jaeyub and Kim [12] developed a level set topology optimization method to efficiently optimize the overall response time of a heat transfer system using thermal eigenvalues. Chen et al. [13] showed lower SME values than steel when lightweight materials such as Al or Cu were used as the body of a thin-walled beam under bending and torsion conditions. Lightweight materials can greatly reduce the weight of the vehicle. However, it also reduces the bending and torsional stiffness of the vehicle body. Ding et al. [14] proposed a parallel optimization model for fiber-reinforced composite structures to obtain lighter fiber-reinforced composites and made a combination of structural topology optimization and fiber angle design, which is an effective method to suppress structural vibration and obtain lighter structures.

In this article, we use HyperMesh to design the initial structure of the front-end frame based on the compromise planning method and the entropy weight method, using the automotive all-plastic front-end frame as the research object; a compromise planning method with weight coefficients is used to obtain the integrated response function by unifying each subobjective and minimizing the integrated response function as the objective. Considering mainly the loads under typical operating conditions of lock limit crash loads, cushion area loads, and pedestrian protection impact loads, the topology of the vehicle front-end frame was optimized, and the final structure of the vehicle front-end frame was derived from actual production conditions. The HyperMesh all-plastic front-end frame is used to extract and repair the mid-plane, the mid-plane is meshed, and the resulting mesh is then set for material and thickness. Six static analyses and the results of the first six orders of modal analysis were carried out on the resulting mesh model, and fatigue analysis of the front-end frame was carried out using N-code software to derive its life and damage results. The final optimization results show that the mass of the front-end frame has been reduced from the original 10.5 kg to 6.8 kg—a weight reduction of 35.24%—which is a significant weight reduction, and the results show that the smallest first-order mode in the modal analysis is 57.38 Hz, a value much greater than the 30 Hz required before design, and its lifetime of 6122 s is much greater than the specified 1200 s. Based on the above optimization results, a test piece is made and a resonance test is carried out on the test piece to verify that the performance of the test piece meets the actual requirements based on the state of the test piece after the resonance check test.

2. Multiobjective Topology Optimization for Automotive All-Plastic Front-End Frames

The multiobjective topology optimization preprocessing includes the selection of the processing multiobjective optimization method, full plastic front-end frame modeling, front-end frame morphology optimization, and front-end frame module topology optimization.

2.1. Multiobjective Topology Optimization Theory. The front-end frame of a vehicle close to the engine will be subjected to various loads and needs to ensure that the stiffness and intrinsic frequency meet the operating requirements. As the different properties of stiffness and vibration frequency differ significantly in terms of the magnitude of the values, a method is required to process the subtargets to eliminate the effects of the different properties and the large difference in values.

2.1.1. Compromise Programming Approach. The compromise programming approach is a mathematical method proposed to solve multiobjective decision problems. The compromise planning method with weight coefficients is used to process multiple objectives so that the multiobjective problem is transformed into a single-objective problem with the integrated function model [15] as
\[
C(\rho) = \min \left( \sum_{k=1}^{m} w_k \left( \frac{C_k(\rho) - C_k^{\text{min}}}{C_k^{\text{max}} - C_k^{\text{min}}} \right)^p \right)^{1/p},
\]

where \(C(\rho)\) is the combined objective function; \(m\) is the total number of subobjectives; \(w_k\) is the weight coefficient of the \(k\)th subobjective; \(C_k(\rho)\) is the objective function of the \(k\)th subobjective; \(C_k^{\text{min}}\) is the minimum value corresponding to the objective function of the \(k\)th subobjective; \(C_k^{\text{max}}\) is the maximum value corresponding to the objective function of the \(k\)th subobjective; and \(p\) is the penalty factor, generally taken as 2.

2.1.2. Entropy Method. Different sub-objectives in a multijective optimization problem have different impacts on product performance, so it is important to determine the degree of impact of subobjectives on product performance and select reasonable weight coefficients for each sub-objective. The traditional methods for determining the weight coefficients of subobjectives consist of the empirical method, orthogonal method, and hierarchical analysis method. Using the entropy weight method as the subgoal weight determination method, the entropy weight method is an objective assignment method, and in the process of using it, the entropy value and entropy weight of each subgoal are calculated by using the information entropy according to the degree of variation of each subgoal, and the calculated entropy weight is used as the weight coefficient of the subgoal [16].

With \(m\) items to be evaluated and \(n\) evaluation indicators, the original data matrix \(R = (r_{ij})_{mn}\):

\[
R = \begin{bmatrix}
  r_{11} & \cdots & r_{1n} \\
  \vdots & \ddots & \vdots \\
  r_{m1} & \cdots & r_{mn}
\end{bmatrix},
\]

where \(r_{ij}\) is the \(i\)th evaluation value scale under the \(j\)th indicator.

The weight of the indicator of the \(i\)th item under the \(j\)th indicator to the sum of all items of the \(j\)th indicator \(p_{ij}\):

\[
p_{ij} = \frac{r_{ij}}{\sum_{i=1}^{m} r_{ij}}.
\]

Entropy value of the \(j\)th indicator:

\[
e_j = -k \sum_{i=1}^{m} p_{ij} \times \ln p_{ij},
\]

where \(k = 1/\ln m\).

The entropy weight of the \(j\)th indicator \(w_j\):

\[
w_j = \frac{1 - e_j}{\sum_{j=1}^{n} (1 - e_j)}.
\]

When each alternative item has exactly the same value on indicator \(j\), the entropy of the indicator reaches the maximum value of 1 and its entropy weight is 0, which indicates that the indicator fails to provide useful information to decision-makers and can be considered to remove the indicator. Therefore, the entropy weight itself is the differentiation degree of the evaluation object under the indicator, rather than the importance coefficient of the indicator, and the entropy weight reflects the stability of the influence of the subobjectives on the integrated objective function.

2.2. Modeling of the Automotive All-Plastic Front-End Framework. The automotive full plastic front-end frame model is modeled in CATIA with 3D data to get the initial 3D model, and then the initial 3D model is imported into HyperMesh for meshing to get the mesh model; the purpose of this step is to transform the physical model into a mathematical model. Load and constraints are added to the mesh for shape optimization, and the full plastic front-end frame model of the car is obtained according to the optimization results and combined with the actual situation.

2.2.1. Front-End Frame Load Analysis. The front-end frame is subjected to complex forces during driving. From the data given by the host manufacturer, the loads under three typical working conditions, namely the latch limit crash load, the cushion area load, and the pedestrian protection impact load, were calculated. The load cases of the original metal front-end frame under the above three working conditions were obtained through the dynamic analysis of the whole vehicle model as shown in Table 1.

2.2.2. Initial Finite Element Modeling of the Front-End Frame. The initial model of the front end of the car was built based on the original sheet metal model. As the modulus of plastic is less than that of metal, the initial model had to occupy as much space as possible to ensure stiffness, but to avoid collisions with surrounding parts when driving, the initial model had to have a distance of 10 mm or more from the surrounding parts. The initial model was preprocessed in HyperMesh and all parts were divided using a hexahedral mesh with 4 mm sides, generating a total of 504,514 cells and 553,916 nodes, each with 6 degrees of freedom. Reb2 cells were created at each mounting point to facilitate the application of loads and constraints. The front-end frame material is PP-LGF30. The finite element model is shown in Figure 1.

To reduce the overall deformation of the front-end frame under one operating condition, the minimization of the overall structural flexibility is used as an objective in the static subobjective optimization. Structural flexibility reflects the deformation capacity of a structure under load and can be understood as the elastic potential energy stored in the structure or as the work done by an external force. Under the conditions of external forces, the structural flexibility is equivalent to the inverse of the stiffness and is given by the following equation [15].

\[
C = \frac{1}{2} u^T f,
\]

where \(C\) is the structural flexibility (N·mm); \(u\) is the displacement matrix of the nodes (mm); and \(f\) is the load (N).
\[ f = Ku, \]

(7)

where \( K \) is the stiffness matrix.

\[ C = \frac{1}{2}u^T Ku = \frac{1}{2} \int \varepsilon^T \sigma \, dV, \]

(8)

\[ C = \frac{1}{2}u^T f = \frac{\sigma^T f}{2K}, \]

where \( \varepsilon \) is the strain under load (mm); \( V \) is the structural volume of the design section (mm\(^3\)); and \( \sigma \) is the stress (MPa).

The topology optimization is carried out in HyperMesh with the flexibility and volume fraction as the response, with the minimum flexibility as the optimization objective and the upper limit of volume fraction not greater than 0.4 as the constraint, and the results are shown in Figure 2.

According to the optimization results in the above figure, the red part with high material density is considered to be retained and the blue part is removed. Considering the three working conditions and frequency optimization results vary greatly, the initial structure of the front-end frame topology optimization is obtained by considering the four topology optimization results comprehensively as shown in Figure 3.

Since the maximum load on the front-end frame is the engine lock limit pull force, which is \( z \)-direction (in the figure, the direction from bottom to top is \( z \)-axis positive direction), that is, perpendicular to the ground upward, and the most solid place of the front-end frame is the longitudinal beam mounting part, so combined with the shape optimization, this shape optimization will design the middle part of the front-end frame as this structure, which can transfer the engine lock limit pull force to the longitudinal beam mounting part through this structure.

2.3. Automotive Front-End Module Topology Optimization.

The static single-objective and dynamic single-objective optimization are carried out separately, and the subobjective weights are obtained according to the optimization results by applying the entropy weight method, and then the multiobjective optimization problem is transformed into a single-objective optimization problem by the compromise planning method. The final front-end framework of the car is obtained by combining the solution results with the actual conditions.

2.3.1. Static Single-Objective Optimization of Automotive Front-End Frameworks.

Static single-objective optimization is, as the name suggests, the topological optimization of static loads on the front-end frame [17], that is topology optimization is performed for three operating conditions: latch limit tension, pedestrian protection impact force, and cushion area force. The topology optimization was carried out for each of the above three static working conditions, and the objective function was defined as the minimum structural flexibility in the optimization, with the upper limit of volume fraction set to 0.5. The mesh type is a hexahedral mesh with 4 mm sides, generating 356221 cells and 395465 nodes, each with 6 degrees of freedom. The cloud diagram obtained after iteration is shown in Figure 4, and the iterative process curve is shown in Figure 5.

From 5, it can be seen that the optimization objective function converges, so the optimization results are credible. From the optimization results in Figure 4, it can be seen that the cells on the top and bottom of the beam on the front-end frame are red, that is, the cell density is high, while the density of the middle cell is small, so the material on the top and bottom sides can be retained and the middle material can be removed in the subsequent optimization. The structural flexibility of each iteration step of the three working conditions is obtained by static single-objective topology optimization, which can provide a reference for subsequent optimization.

2.3.2. Dynamic Multiobjective Optimization of Automotive Front-End Frameworks.

Dynamic multiobjective optimization of front-end frames simply means optimizing the intrinsic frequency characteristics of the front-end frame [18]. Low-order inherent frequency characteristics generally refer to the first, second, and third-order modes, which are collectively referred to as low-order modes. When the
Figure 2: Topology optimization results. (a) Optimization results in the minimum flexibility of the latch limit tension condition; (b) optimization results in minimum flexibility of pedestrian protection impact force conditions; (c) optimization results in the minimum flexibility of the force conditions in the cushion area; and (d) optimization results in the first three sections of intrinsic frequency normalized weighted topology.

Figure 3: Front-end framework shape optimization results.

Figure 4: Static load topology optimization results. (a) Topology optimization results of latch limit tension load; (b) topology optimization results of pedestrian protection impact load; and (c) topology optimization results of cushion area load.
external vibration frequency reaches or is close to the frequency corresponding to the low-order mode, the parts will resonate, producing harmful vibration and noise and shortening the service life of the parts. The engine is the source of vibration in a vehicle, and the front-end frame is close to the engine; so to ensure that the front-end frame is used properly, its modalities need to be increased. Generally speaking, the source of vibration in a car comes only from the engine and the external environment, which has a low vibration frequency and does not cause vibration in the car parts. As the modal order is larger, the corresponding frequency is larger, and the engine excitation frequency is about 30 Hz; so to avoid the resonance of the front-end frame, its low-order modal needs to be improved.

Topological optimisation of each of the first three orders of modes of the front-end frame, where the objective function is defined as the frequency maximum and the upper limit of the volume fraction is set to 0.5. The cloud diagram obtained after iteration is shown in Figure 6, and the iterative process curve is shown in Figure 7.

Figure 6 shows that the first three iterations of the modal curve fluctuate but still converge. Figure 6 shows that the higher cell density in red is considered to be retained, while the lower cell density in blue can be removed in the subsequent optimization. The frequency of each iteration step of the first three modal steps is obtained through dynamic single-objective topology optimization, which can be used for subsequent optimization.

2.3.3. Subobjective Weight Distribution based on Entropy Weighting Method. The degree of influence of each sub-objective on the structure varies, so it is necessary to set the corresponding weight for each sub-objective, and this article proposes to use the entropy weight method to calculate the weight value of each sub-objective. The entropy weight method needs to consider the finite element analysis results obtained for each sub-objective after individual optimization. The optimization results from the previous section are derived from a three-dimensional model and modified appropriately to obtain a specific model, which is then used to carry out the analysis; the results of which are shown in Table 2.

Table 2 shows the results of the analysis after optimizing the subgoals individually, which is equivalent to the raw data of equation (2). The entropy value and entropy weight of each subgoal are calculated in turn using equations (3) to (5), using the entropy weight calculation process for the first subgoal as an example.

The weight of the first indicator to the sum of all indicators is calculated from the following equation:

\[ p_{i1} = \frac{r_{ij}}{\sum_{j=1}^{6} r_{ij}} = \frac{2125.69}{2125.69 + 2001.23 + 1635.6 + 1736.18 + 1864.69 + 1863.84} = 0.1899. \]  

Similarly, the remaining indicators as a proportion of the sum of all indicators are shown in Table 3.

There are six indicators and \( m = 6 \), so the coefficient \( k \) is:

\[ k = \frac{1}{\ln 6} = 0.588. \]  

The entropy value \( e_1 \) of the first indicator is calculated from the following equation as:

\[ e_1 = 0.9979. \]  

The entropy weight \( w_1 \) of the first indicator is calculated from the following equation as:

\[ w_1 = \frac{1 - 0.9979}{1 - 0.9979 + 1 - 0.9992 + 1 - 0.9984 + 1 - 0.9949 + 1 - 0.9975 + 1 - 0.9959} = 0.1335. \]
Figure 6: Results of dynamic load topology optimization. (a) First-order modal topology optimization results; (b) second-order topology optimization results; and (c) third-order modal topology optimization results.

Figure 7: Dynamic load topology optimization iteration curve.

Table 2: Analysis results after optimizing subobjectives individually.

| Individual optimization of subtargets | Structural flexibility (N * mm) | First-order mode (Hz) | Second-order mode (Hz) | Third-order mode (Hz) |
|--------------------------------------|---------------------------------|-----------------------|------------------------|------------------------|
| Locking limit pull                   | 2125.69                         | 78.64                 | 71.35                  | 78.66                  |
| Pedestrian protection against impact forces | 2001.23                        | 81.23                 | 76.95                  | 85.32                  |
| Force in the cushion area            | 1635.6                          | 94.63                 | 68.62                  | 79.65                  |
| First-order mode (Hz)               | 1736.18                         | 75.32                 | 88.6                   | 95.46                  |
| Second-order mode (Hz)              | 1864.69                         | 77.62                 | 98.94                  | 106.58                 |
| Third-order mode (Hz)               | 1836.84                         | 69.54                 | 76.37                  | 120.75                 |
The entropy values and entropy weights of each indicator are given in Table 4.

2.3.4. Topology Optimization of Automotive Front-End Frames based on a Compromise Planning Approach. The maximum value of the first three orders of intrinsic frequency is used as the objective function and the volume fraction as the constraint function in modal topology optimization. However, during topology optimization, it may happen that the frequency value of one of the orders reaches a maximum and the modal values of the other orders fall to a lower level and the frequencies of the different orders may be switched in order with each other. The natural frequency iteration curve of the objective function oscillates. To overcome this phenomenon, this article adopts the average frequency method [19], adds corresponding weight coefficients to different modes of order, obtains the new function

\[ C_p = \min \left\{ \frac{0.1335}{6118.25 - 2542.64} \left( a_1 - 2542.64 \right)^2 + \frac{0.129}{5833.27 - 2954.16} \left( a_2 - 2954.16 \right)^2 + \frac{0.6304}{386572 - 1911.95} \left( a_3 - 1911.95 \right)^2 \right\}^{1/2} \]  

(13)

According to the figure above, the front-end frame structure can be clearly obtained. The element density in the red part is larger, which can be considered to be retained, and the material in the blue part is smaller, which can be removed. The optimization results can provide a reference for subsequent optimization. Considering that the front-end frame is an injection molded part, the front-end frame is designed in the form of a plastic board to ensure the precise size of the parts after injection and reduce the volume shrinkage rate and warpage deformation. According to Figure 11, slabs are laid in the red part, materials are removed in the blue part, and the thickness of each plate should be kept the same as far as possible. Finally, considering the actual manufacturing situation, the automobile's all-plastic front frame is obtained as shown in Figure 12. The weight of the original sheet metal front frame is 10.5 kg and the weight of the plastic front frame is 6.8 kg, reducing 35.24%; the weight reduction effect is obvious.

3. Finite Element Analysis of the Automotive All-Plastic Front-End Frame

The front-end frame was extracted and the center face was repaired using HyperMesh, and the complete center face was meshed, with material and thickness settings. Static and modal analyses were carried out on the mesh model, and by frequency-weighted reciprocal of the first three modes, and obtains the maximum value of the mode by finding the minimum value of the function. The average frequency method was used to obtain the function iteration curve as shown in Figure 8, and the iteration results are shown in Figure 9.

Since the response function corresponding to the three working conditions is the structural flexibility and the response function corresponding to the mode is the frequency, the units of the two response functions are inconsistent and the values differ greatly; in addition, the structural flexibility requires the smaller the better and the mode requires the larger the better, so the compromise planning method with weight coefficients is used to construct the integrated objective function, and the single-objective optimization iteration curve can obtain the individual eigenvalues. The integrated objective function is obtained by substituting each eigenvalue into equation (1) as equation (13). The iterative curve of the integrated objective function response is shown in Figure 10, and the topology optimization results are shown in Figure 11.

Fatigue analysis was carried out on the front-end frame using N-code to derive its life and damage results.

3.1. Automotive All-Plastic Front-End Frame Pretreatment. The software used for the preprocessing of the front-end frame is HyperMesh. Considering that the structure of the front-end frame is thin-walled and large, the use of a 3D solid mesh would slow down the calculation, so this article uses a mid-plane sheet instead of a 3D solid for the simulation analysis.

3.1.1. Front-End Framework Middle Surface Extraction and Repair. The mid-plane, as the name implies, is the middle surface of the thinner solid, and HyperMesh software is used to automatically extract the mid-plane. The HyperMesh software is very powerful in extracting the neutral surface, but there are some defects in the automatic extraction of the neutral surface, such as incomplete surface, distorted surface, and the neutral surface are far from the middle of the solid, so it is necessary to repair the neutral surface manually after the automatic extraction of the neutral surface to make it meet the requirements of FEA. The 3D data model of the front-end frame of the car is imported into HyperMesh to automatically extract the mid-plane and repair the mid-plane, and the results are shown in Figure 13.
3.1.2. Front-End Framework Middle Surface Mesh Division. After completing the mid-plane extraction, the mid-plane finite elementization, that is meshing, is required. HV_he meshing is extremely important, and the quality of the mesh will directly affect the accuracy of the analysis results. HV_he shell mesh is used in the stress analysis, the size is set to 5 mm, and the style is set to a mixture of triangular and quadrilateral meshes, and the results are shown in Figure 14, and a partial enlargement of the longitudinal beam mounting point location is shown in Figure 15. To reduce the effect of low mesh quality, the low-quality meshes need to be modified until all meshes meet the analysis conditions. HV_he mesh refinement is used to refine the stresses in the concentrated parts of the mesh. HV_he stresses obtained after the mesh refinement are larger than when the mesh is not refined, and the stresses obtained after the mesh refinement are generally more accurate. The refinement generates a total of 60,341 cells with 57,312 nodes, each with 6 degrees of freedom. HV_he mesh refinement results are shown in Figure 16.

3.1.3. Mesh Irrelevance Analysis. The accuracy of the calculation is proportional to the number of meshes for the same product with different number of meshes, but too large a number of meshes can also cause problems such as excessive computational effort, long analysis time, and high hardware requirements for the simulation equipment. Therefore, this article considers that a smaller number of meshes is better, provided that the analytical calculation results do not vary much, that is the number of meshes divided is irrelevant to the results.

3.1.4. Front-End Frame Thickness, Load, and Material Settings. After meshing is completed, the thickness of each mesh needs to be set, using an automatic thickness setting, and the results are shown in Figure 17. Determining the coordinates of the load application points in the CAD model, as well as the magnitude and direction of the load, the load application point and the latch mounting point with a rigid unit are connected, the load is added to the load application point using the constraint card, and the size and direction of the load are defined. The front-end frame engine lock rigid connection unit and load settings are shown in Figure 18. To reduce the impact of errors in the mechanical parameters of the material, it was necessary to find reasonable material data. The material was defined as PP-LGF30 in the material card, and the material was determined to have a modulus of elasticity of 4900 MPa, a Poisson’s ratio of 0.34, and a density of 1.13 g/cm³.

3.1.5. Boundary Conditions. There is also a certain amount of error in setting the boundary conditions, so loads need to be applied and constraints added according to the actual
working conditions of the part. Rigid units are created in the mounting holes on the left and right sides of the model, and then the 6 degrees of freedom are constrained at the center of the rigid unit as shown in the red triangle in Figure 19.

3.2. Static Analysis of the Automotive All-Plastic Front-End Frame. The static analysis of the car’s all-plastic front-end frame mainly has six loads: latch limit tension, cushion impact, pedestrian impact, condenser support strength, radiator support strength, and oil cooler mounting strength. The displacement cloud of the six loads is shown in Figure 20, the stress cloud is shown in Figure 21, and the summary table of static analysis information is shown in Table 5.

3.3. Modal Analysis of the Automotive All-Plastic Front-End Frames. The results of the first six orders of restrained modal vibrations of the all-plastic front-end frame are
shown in Figure 22, and the values of each mode are shown in Table 6. Figure 22(a) shows that the first-order mode shape displacement is mainly at the engine lock. Nevertheless, it can be seen from Table 6 that the minimum first-order mode is 57.38 Hz, which is much higher than the required 30 Hz before the design, and resonance can be effectively avoided in the process of vehicle driving [20].

### 3.4. Automotive All-Plastic Front-End Frame Fatigue Analysis

Since the speed of the car is changing during the driving process, the force acting on the engine lock is also changing all the time. To prevent the front-end frame from fatigue damage that leads to structural fracture at the time of the forward collision, it is necessary to perform fatigue analysis on the front-end frame before manufacturing [21]. The fatigue analysis is preprocessed in HyperMesh, and the solver uses N-code, an ANSYS company.

Fatigue analysis for the engine lock is subjected to alternating stresses due to airflow in the engine cover; according to the data provided by the host manufacturer, the stress amplitude for fatigue analysis is 1000N. Before fatigue analysis, a static analysis is required. The static analysis is performed in HyperMesh and the results are exported to OP2 format, and the results of the static analysis are shown in Figure 23.

The purpose of the fatigue analysis is to check the strength of the engine lock and the upper beam. N-code was used as the solver for the fatigue analysis, the static analysis result file OP2 was imported into N-code, and the load spectrum and material properties were defined in the software. The total time of the load spectrum is set to 200 s, the sampling frequency is 1000, the form of the load action is white noise, and the solution engine is SN CAE Fatigue. The material is PP-LGF30, which is not available in the material library, so the custom material is chosen, the material modulus of elasticity is 4900 MPa, the ultimate tensile strength is 300 MPa, and the standard error is chosen to be 1N. The SN curve of PP-LGF30 material is shown in Figure 24, and the solution result is shown in Figure 25.

Observing the two cloud diagrams, it can be seen that the damaging cloud and the lifetime cloud are almost the same, that is, the parts that are easily damaged have a shorter lifetime. From the damage cloud diagram, it can be seen that the most easily damaged parts of the front-end frame are the longitudinal beam mounting holes and engine latch mounting holes. The shortest life is 6122 s, which is much longer than the required 1200 s, so the front-end frame meets the design requirements.

### 4. Experimental Validation

The resonance of the front-end frame was tested and verified according to the requirements specified in JIS D1601-1995
Figure 20: Front-end framework displacement cloud. (a) Latch limit tension displacement cloud chart; (b) cloud diagram of cushion impact displacement; (c) cloud image of pedestrian impact displacement; (d) displacement cloud diagram of condenser support strength; (e) displacement cloud diagram of radiator support strength; and (f) cloud map of oil cooler installation strength displacement.

Figure 21: Front-end framework stress cloud. (a) Cloud diagram of ultimate tensile stress of lock; (b) impact stress cloud diagram of cushion; (c) cloud map of pedestrian impact stress; (d) stress cloud diagram of condenser support strength; (e) stress cloud diagram of radiator support strength; and (f) oil cooler installation strength stress cloud diagram.

Table 5: Static analysis information sheet.

| Load                      | Size (N) | Mounting point displacement (mm) | Maximum stress (MPa) | Design requirements (mm) | Does it meet the design requirements? |
|---------------------------|----------|----------------------------------|----------------------|--------------------------|---------------------------------------|
| Locking limit pull        | 5300     | 14.69                            | 138                  | 15                       | Yes                                   |
| Cushion impact            | 1000     | 0.773                            | 19.61                | 4                        | Yes                                   |
| Pedestrian impact         | 5000     | 15.35                            | 207.8                | 20                       | Yes                                   |
| Condenser support strength| 400      | 0.619                            | 6.811                | 3                        | Yes                                   |
| Radiator support strength | 350      | 0.62                             | 5.607                | 3                        | Yes                                   |
| Oil cooler mounting strength | 170     | 0.309                            | 3.592                | 2                        | Yes                                   |
Figure 22: Modal analysis of the first six orders of the front-end framework. (a) First-order modal finite element analysis results; (b) second-order modal finite element analysis results; (c) the third-order modal finite element analysis results; (d) fourth-order modal finite element analysis results; (e) fifth-order modal finite element analysis results; and (f) sixth-order modal finite element analysis results.

Table 6: Modal values of each order.

| Modal | First-order | Second-order | Third-order | Fourth-order | Fifth-order | Sixth-order |
|-------|-------------|--------------|-------------|--------------|-------------|-------------|
| Value (Hz) | 57.38 | 78.77 | 86.69 | 90.19 | 106.11 | 116.85 |

Figure 23: Results of the static analysis of the front-end frame. (a) Static analysis of displacement cloud map; and (b) statics analysis of stress cloud map.

Figure 24: SN curve for PP-LGF30 material. (a) Static analysis of displacement cloud map; and (b) statics analysis of stress cloud map.
Figure 25: Fatigue analysis solution results. (a) Cloud image of fatigue damage of front frame; and (b) cloud diagram of front-end frame fatigue life.

Table 7: Resonance check vibration experimental conditions.

| Frequency range (Hz) | Acceleration (m/s^2) | Frequency sweep type | Sweep frequency | Number of frequency sweeps | Vibration direction | Test part number       |
|----------------------|----------------------|----------------------|-----------------|---------------------------|---------------------|------------------------|
| 5–100                | 5                    | Logarithmic sweep    | 1 Oct/min       | 1 cycle                   | Z                   | 1#(B-CROSS), 2#(A2)    |

Table 8: Test piece measurement points.

| Measurement point number | Acquisition channel | Location          | Model   | Number | Remarks         |
|--------------------------|----------------------|-------------------|---------|--------|-----------------|
| 1#                       | Input 1              | Workwear left     | 13100   | J5295  | Control points  |
| 2#                       | Input 2              | Workwear right    | 13100   | J5296  | Control points  |
| 3#                       | Input 3              | On the test piece | 23250   | J0445  | Measurement points |
| 4#                       | Input 4              | On the test piece | 23250   | J0447  | Measurement points |
| 5#                       | Input 5              | On the test piece | 23250   | J0448  | Measurement points |
| 6#                       | Input 6              | On the test piece | 23250   | J0449  | Measurement points |
| 7#                       | Input 7              | On the test piece | 23250   | J0450  | Measurement points |
| 8#                       | Input 8              | On the test piece | 23250   | J0456  | Measurement points |

Figure 26: Sensor mounting position. (a) B-cross sensor mounting position (1#–2#); (b) B-cross sensor mounting position (3#–8#); and (c) A2 sensor installation position (1#–8#).
Table 9: 1# test piece resonance point information.

| Measurement points | Resonance frequency (Hz) | Peak acceleration (m/s²) |
|--------------------|--------------------------|--------------------------|
| 3#                 | 62.9321                  | 13.8802                  |
| 4#                 | 63.3021                  | 15.4192                  |
| 5#                 | 61.1141                  | 11.2214                  |
| 6#                 | 63.3021                  | 8.9476                   |
| 7#                 | 61.1141                  | 14.6105                  |
| 8#                 | 63.3021                  | 12.1002                  |

Table 10: 2# test piece resonance point information.

| Measurement points | Resonance frequency (Hz) | Peak acceleration (m/s²) |
|--------------------|--------------------------|--------------------------|
| 3#                 | 58.3140                  | 13.5643                  |
| 4#                 | 57.9732                  | 8.6107                   |
| 5#                 | 58.6569                  | 36.9326                  |
| 6#                 | 59.0018                  | 43.3931                  |
| 7#                 | 59.2974                  | 7.3543                   |
| 8#                 | 58.6569                  | 46.0072                  |

![Figure 27: Continued.](image-url)
Figure 27: Continued.
Vibration Test Methods for Automotive Parts, using two test pieces respectively under specific test conditions to derive information on each resonance point and the vibration control response curve [22].

4.1. Detection of Test Conditions and Test State. The vibration test conditions are shown in Table 7. The test piece is fixed to the rigid support by screws, and the support is fixed to the shaking table surface by screws and pressure bars. In the vibration test, a total of 8 sensors are installed, and the sensor measurement point information is shown in Table 8. The sensor installation position is shown in Figure 26, and the sensor installation position is unchanged in each direction of the test, and the pick-up direction is the same as the vibration direction.

4.2. Experimental Results. The resonance point information of the resonance inspection test of the 1# swept frequency test piece is shown in Table 9, and the resonance point information of the 2# swept frequency test piece is shown in Table 10. The vibration control and response curves of the two test pieces are shown in Figure 27.

After the test, the appearance of the structural state is shown in Figure 28, and according to the experimental data, it can be seen that the mechanical integrity and other properties of the test parts meet the actual requirements.

Figure 27: Resonance check test control curve and response curve. (a) 1# sweep frequency test piece resonance check test control curve; (b) 1# sweep test piece resonance check test response curve; (c) 2# sweep frequency test piece resonance check test control curve; and (d) 2# sweep frequency test piece resonance check test response curve.

Figure 28: The state of the test piece after the resonance check test. (a) 1# (B-cross) test piece resonance check after test status; and (b) 2# (A2) test piece resonance check after the test state.
5. Conclusions

(1) The original 3D data model of the all-plastic front-end frame was drawn by CATIA based on the original sheet metal parts, and the model was imported into HyperMesh to divide the mesh to obtain the 3D solid mesh model. With the three working conditions of latch limit tension, pedestrian protection impact force, and cushion area force as well as the first three orders of modalities as sub-targets, static analysis of the mesh model using HyperMesh and topological optimization of the first three orders of frequency objectives using the mean frequency method to obtain limiting eigenvalues. Based on the single-objective optimization results, the entropy weighting method is used to calculate the weighting coefficients for the latch limit crash load, cushion area load, pedestrian protection impact load, and the first three orders of modal subobjectives for the fully plastic front-end frame topology optimization model of the vehicle. The compromise planning method with weights is used to construct a comprehensive objective function to obtain the final topology optimization results. The final result makes the mass of the original sheet metal front-end frame 10.5 kg and the mass of the plastic front-end frame 6.8 kg, with a weight reduction of 35.24% and an obvious weight reduction effect.

(2) HyperMesh was used to extract and repair the mid-plane of the 3D data model, and the mesh was divided on the mid-plane to set the thickness and material. The six loads and the first six orders of the modal analysis of the mesh model were analyzed statically and modally with an OptiStruct solver. The static analysis mainly analyzed the six loads and the first six orders of the modal analysis of the latch limit tension, cushion impact, pedestrian impact, condenser support strength, radiator support strength, and oil cooler installation strength, and the results were obtained to meet the design requirements; in the modal analysis, the smallest first-order mode is 57.38 Hz, which is much larger than the 30 Hz required before design, so it is known that resonance can be effectively avoided during the driving process of the car. The engine lock strength performance of the front-end frame was tested by N-code, and the fatigue analysis of the front-end frame showed that its lifetime was 6122 s, which was much longer than the specified 1200 s.

(3) According to the test requirements of JIS D1601-1995 Vibration Test Methods for Automotive Parts, a resonance test is carried out to verify the test parts. The resonance point information and vibration control and response curve of the test parts’ resonance check test is verified. According to the state of the test parts’ resonance check test, the performance of the test parts is judged to meet the actual requirements.

Data Availability

The data that 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 study belongs to the major projects of the “The University Synergy Innovation Program of Anhui Province (GXXY-2019-004)” and to the project of the “Teaching Research Project of Anhui Education Department (2019jyxm0229).”

References

[1] Q. Pan and C. Zhao, “Research on lightweight brake structure based on equal strength design,” Railway Locomotive & Vehicle, vol. 41, no. 03, pp. 65–69, 201.
[2] W. Zhenyang, “Research status of automotive lightweighting technology,” Automotive Engineer, no. 11, pp. 11-12, 2020.
[3] J. Shi, Research on Layout Optimization of Multiphase Materials and Dual-Modulus Materials, Northwest A&F University, China, Shaanxi, Xianyang, Yangling, 2016.
[4] F. Lan, F. Lai, and J. Chen, “Multi-case topology optimization of body structure considering dynamic characteristic,” Journal of Mechanical Engineering, vol. 50, no. 20, pp. 122–128, 2014.
[5] Y. Wang, C.-cai Zhou, and Y. Zhou, “Design of aero engine sliding tank bracket based on topology optimization,” Journal of Propulsion Technology, vol. 43, no. 02, pp. 207–217, 202.
[6] J.-xin Zhang, W. Liang, and Y. Xia, “Research on objective function construction method for multi-objective topology optimization of structures,” China Mechanical Engineering, vol. 27, no. 07, pp. 899–903, 2016.
[7] Y. Wang, H. Wang, J. Wei, B. Lin, J. Xu, and S Fang, “Finite element analysis of grinding process of long fiber reinforced ceramic matrix woven composites: modeling, experimental verification and material removal mechanism,” Ceramics International, vol. 45, no. 13, Article ID 15920, 2019.
[8] G. Soheil, F. Mozafari, N. Moslemi et al., “Analytical solution of the electro-mechanical flexural coupling between piezoelectric actuators and flexible-spring boundary structure in smart composite plates,” Archives of Civil and Mechanical Engineering, vol. 21, no. 1, pp. 1–25, 2021.
[9] D. Qu, H. Zhang, J. Xu, and Q Chen, “The theoretical research on solving accuracy based on level set topology optimization,” IOP Conference Series: Materials Science and Engineering, vol. 493, no. 1, Article ID 012030, 2019.
[10] Y.-Ho Kim and S.-Y. Han, “Topological shape optimization scheme based on the artificial bee colony algorithm,” International Journal of Precision Engineering and Manufacturing, vol. 18, no. 10, pp. 1393–1401, 2017.
[11] Mi Xiao, X. Liu, Y. Zhang, and G. Liang, “Design of graded lattice sandwich structures by multiscale topology optimization,” Computer Methods in Applied Mechanics and Engineering, vol. 384, 2021.
[12] H. Jae-yub and H. Kim, “Alicia. Level-set topology optimization for effective control of transient conductive heat response using eigenvalue,” International Journal of Heat and Mass Transfer, vol. 176, 2021.
X. Chen, Y. Zhang, and P. Jia, “Research on light weight of thin-walled beam for automobile body,” E3S Web of Conferences, vol. 268, 2021.

H. Ding, B. Xu, C. Huang, and D. Zunyi, “A Multi-Scale Discrete Material Optimization Model for Optimization of Structural Topology and Material Orientations to Minimize Dynamic compliance,” Structural and Multidisciplinary Optimization, vol. 64, no. 1, pp. 1–23, 2021.

Z. Zhang, C. Ren, and Z. Xu, “Multi-objective-oriented topology optimization study of automotive suspension control arm,” Journal of Mechanical Engineering, vol. 53, no. 04, pp. 114–121, 2017.

O. Wu, Q. Cheng, Y. Jin, and B. Wang, “Performance evaluation of water-lubricated tail bearing based on entropy weight fuzzy comprehensive evaluation method,” China Mechanical Engineering, vol. 31, no. 12, pp. 1407–1414, 2020.

A. Mortazavi, “A new fuzzy strategy for sizing and topological optimization of truss structures,” Applied Soft Computing, vol. 93, Article ID 106412, 2020.

X. Zang, Z. Gu, Mi Chengji, and W. Wu, “Static and dynamic multi-objective topology optimization of mining vehicle frame structure,” Automotive Engineering, vol. 37, no. 5, pp. 566–570, 2015.

S. Li, Research on Performance Calculation Analysis and Optimization Design of Vehicle Suspension Control Arm, South China University of Technology, China, 2018.

X. Cai, P. Chang, and X. Guo, “Fatigue analysis of wind turbine blade based on ansys and nCode Co-simulation,” Renewable Energy, vol. 39, no. 10, pp. 1342–1346, 201.

D. Yao, X. Gu, and S. Jia, “Structural optimization analysis of muffler bracket based on modal analysis,” Internal Combustion Engine and Accessories, vol. 20, pp. 74–76, 2021.

G. Tu, Research on Vibration Fatigue Test Method of Electric Vehicle Integrated Powertrain, Chongqing University of Technology, Chongqing, China, 2020.