

Lightweight Design of New Energy Vehicle Wheels Using ANSYS Workbench

Minghui Wei *

School of Mechanical Engineering, Shenyang Jianzhu University

* Corresponding Author Email: 1243260842@qq.com

Abstract. Against the backdrop of an intensifying global energy crisis and the ongoing advancement of the “dual carbon” goals, vehicle lightweighting has emerged as a key technological direction for improving driving range and energy efficiency, with wheel lightweighting playing a particularly crucial role in this effort. This paper focuses on the wheel hub of a specific vehicle model. Adopting a lightweight structural design approach, the authors used Creo to perform 3D modeling and structural optimization of the wheel hub, resulting in a lightweight design solution; and perform finite element simulations of the component under various operating conditions, including radial and bending loads, using ANSYS. The test results show that the hub we designed has a significantly reduced weight, and its strength, hardness and fatigue resistance can all meet the requirements of actual use. This study provides technical guidance and practical insights for the lightweight design and performance validation of new energy vehicle wheel hubs.

Keywords: New Energy Vehicles; Wheels, Weight Reduction; Finite Element Analysis.

1. Introduction

Against the backdrop of an increasingly severe global energy crisis and the ongoing implementation of “dual carbon” policies, the automotive industry is rapidly evolving toward electrification, lightweighting, and intelligent technology. The problems of high fuel consumption and high emissions in traditional gasoline-powered vehicles are becoming increasingly prominent, while for new energy vehicles, limited battery technology means that driving range is not a key factor hindering their widespread adoption. Studies have shown that for every 10% reduction in a vehicle’s curb weight, energy consumption can be reduced by 6% to 8% and driving range can be increased by 5% to 10%; therefore, lightweight design has become a key strategy for improving the performance of new energy vehicles[1].

As a core load-bearing component of a vehicle’s drivetrain, the wheel hub directly bears radial loads, bending loads, and road impact loads; its structural strength, fatigue performance, and weight reduction directly impact vehicle safety, handling, and fuel efficiency [4]. Traditional steel wheels are dense and heavy, making it difficult to meet the weight-reduction requirements of new energy vehicles; aluminum alloys, on the other hand, offer advantages such as low density, high specific strength, corrosion resistance, ease of machining, and recyclability, and have become the material of choice for lightweight wheels. Among them, 6061-T6 aluminum alloy is the most widely used in the production of automotive wheels because this material has balanced overall performance [1] [4].

Currently, wheel hub lightweighting is primarily achieved through three approaches: material optimization, structural innovation, and advanced manufacturing processes. Using software such as Creo for 3D modeling and ANSYS for finite element analysis has become the industry standard for R&D. Through radial fatigue, bending fatigue, and modal analysis, it is possible to accurately assess the stress distribution, deformation behavior, and anti-resonance performance of the hub under complex operating conditions. Combined with topological optimization, this approach can further eliminate redundant material, thereby achieving a synergistic improvement in both weight reduction and safety [2, 3, 4].

Although significant progress has been made in the field of lightweight aluminum alloy wheels both domestically and internationally, conventional straight-spoke designs still suffer from issues such as significant stress concentration and limited potential for weight reduction. To this end, this

paper focuses on new energy vehicle wheels, employing a petal-inspired biomimetic design. Using ANSYS Workbench, we conduct multi-condition simulation analysis and topological optimization with the aim of designing a lightweight aluminum alloy wheel that meets strength, fatigue, and modal requirements, thereby providing a reference for the R&D of similar products.

2. Wheel Concepts and Model Design

2.1. Wheel Hub Construction

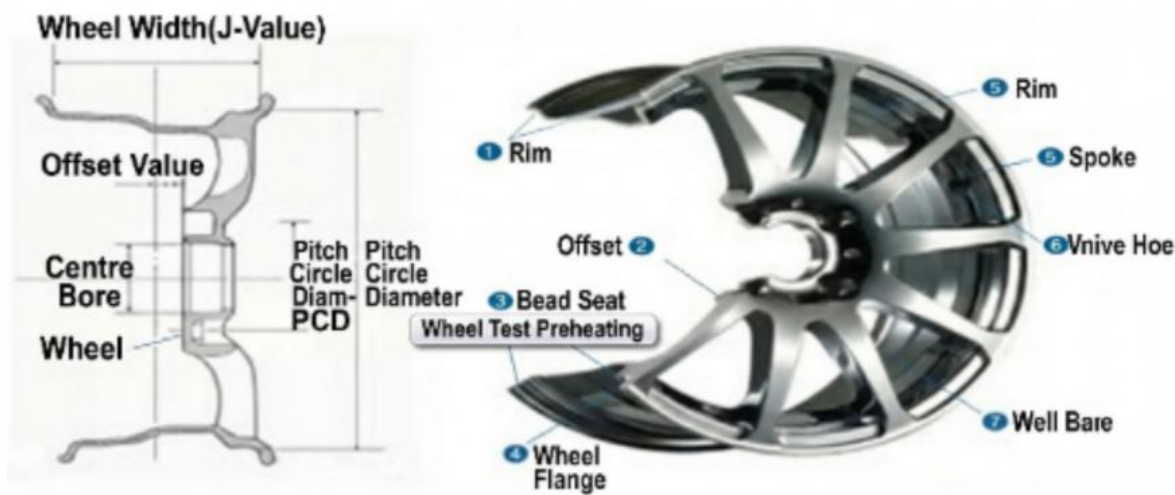


Fig 1. 3D schematic diagram of the hub.

As shown in Figure 1, an automotive wheel hub primarily consists of three parts: the rim, the spokes, and the mounting flange. The mounting flange is located at the center and features bolt holes and a center alignment hole for connecting to the steering knuckle or half-shaft. The wheel rim consists of an outer annular structure that directly supports and seals the tire; it can be divided into the flange, the bead seat, and the main body of the rim; The rim of the wheel adopts a double-shouldered and angled design, which can prevent the tire from axial movement, while the flange plays a limiting role to avoid tire detachment.

The spokes connect the wheel rim to the mounting flange in a radial pattern and transmit various loads encountered during driving. Offset (ET value) is a critical parameter for wheel rims; it refers to the vertical distance from the mounting flange to the centerline of the rim, and it affects the relative position of the wheel to the vehicle body and suspension.

2.2. Wheel model

The design of this wheel hub draws inspiration from the biomimetic concept of flower petals, featuring a multi-spoke configuration to optimize stress distribution and enhance lightweight performance. Petal-like structures are widely found in nature; their expansive, radial structure (Fig. 2) and uniform stress distribution provide natural inspiration for lightweight engineering design. This biomimetic configuration creates multiple continuous load-transfer paths in the circumferential direction, effectively dispersing stress concentrations caused by complex loads such as braking, steering, and road impacts, thereby enhancing the hub's fatigue resistance.

Compared to traditional straight-spoke designs, petal-shaped spokes significantly enhance the wheel's bending and torsional stiffness through their gradually varying cross-sections and curved profiles; At the same time, the naturally formed open spaces between the petals help significantly reduce weight. Even if some spokes are damaged, the adjacent petal-shaped spokes can still share the load, maintain overall structural stability and thereby enhance the hub's structural redundancy and safety margin.



Fig 2. Abstract Petal Design.

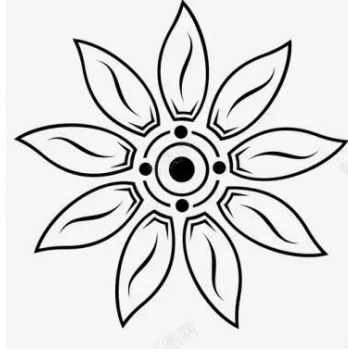


Fig 3. 3D model of the hub.

Building on the conventional spoke structure, this design incorporates a petal-inspired shape, further enhancing structural load-bearing capacity while improving aesthetic appeal. In addition, the wheel hub has been hollowed out to reduce material usage and achieve weight reduction. The specific parameters of the hub are shown in Table 1, and the corresponding 3D model is shown in Figure 3.

Table 1. Hub Specifications.

Wheel diameter	Wheel width	Wheel rim type	Eccentricity	PCD
18inches	8inches	J-type	30mm	5X112mm

3. Finite element model

3.1. Finite Element Method

The finite element method, which is based on the principle of variational calculus and the concept of discretization, is the mainstream numerical method for the mechanical analysis of engineering structures. This method breaks down continuous, complex real-world structures into a finite number of interconnected simple elements, transmits forces and displacements through the element nodes, establishes mechanical control equations for each element, and integrates them into a set of system equations. The solution ultimately yields the distribution of mechanical parameters—such as stress, strain, and displacement—under specified boundary conditions and loads, enabling accurate simulation of the mechanical behavior of complex structures under various operating conditions and providing a reliable basis for structural performance evaluation and lightweight optimization [5].

3.2. Introduction to ANSYS

ANSYS is a widely used general-purpose finite element analysis software in the engineering field, featuring comprehensive analysis modules for structural mechanics, fluid dynamics, thermal analysis, and multiphysics coupling. It offers extensive functionality and highly stable solutions. In the lightweight design of new energy vehicle wheels, ANSYS can handle the entire analysis process, from model import, meshing, and the application of loads and boundary conditions to solution calculation and result visualization. Its powerful post-processing capabilities clearly display key results such as stress contour plots, strain distributions, and fatigue life, enabling designers to quickly assess whether structural strength and stiffness meet requirements, and efficiently supporting iterative design of wheel hub structures and validation of lightweighting solutions^[5].

3.3. Grid division

After importing the 3D model of the wheel hub into ANSYS Workbench, set the global element size to 10 mm and complete the meshing. The final finite element model consisted of a total of 236,575 elements and 131,379 nodes, striking a good balance between computational accuracy and solution efficiency. The finite element mesh model that was created is shown in Figure 4.



Fig 4. Finite element model of the hub.

3.4. Material Properties

As the primary material currently used for automotive wheels, aluminum alloy, compared to traditional steel wheels, it offers outstanding advantages in terms of weight reduction, heat dissipation, machining precision, and aesthetic design, it effectively reduces the vehicle's overall energy consumption, improves driving stability, and extends tire life. Aluminum alloy wheels have a relatively low density, which can greatly reduce unsprung mass, thereby reducing vehicle energy consumption and environmental pollution, and at the same time, they have good thermal conductivity, which can quickly dissipate the heat generated during braking, reduce the risk of tire blowout caused by high temperature, and make vehicle driving safer. This material offers high machining precision; when used in conjunction with radial tires, effectively reduces road vibrations and shocks, improving ride comfort; In addition, aluminum alloys offer high ductility, after polishing and anti-corrosion treatment, it exhibits excellent structural and aesthetic properties and is suitable for a wide range of engineering applications.

This paper uses aluminum alloy as the material for the wheel hub, as this material possesses good casting properties, Flow properties and mechanical properties, suitable for manufacturing complex structures, Wheel hub components with multiple curved surfaces can meet the strength, stiffness, and safety requirements for vehicle operation (see Table 2).

Table 2. Properties of Aluminum Alloys.

Material Name	Modulus of elasticity	Poisson's ratio	Density	Yield strength
A356 aluminum alloy	$6.9 \times 10^4 \text{ Mpa}$	0.33	2660 kg/m^3	205Mpa

4. Simulation and Analysis

4.1. Vehicle Characteristics

During the simulation analysis, the radial and bending loads acting on the wheel hub are calculated using a specific vehicle model as a reference. The detailed specifications of the reference vehicle are listed in Table 3:

Table 3. Reference Vehicle Specifications.

Curb weight	Maximum gross weight	Tire Size
1730kg	2180kg	R18

4.2. Radial Load Analysis

4.2.1. Loads and Boundary Conditions

When driving in a straight line, the wheels are subjected to repeated vertical loads, such as the vehicle's weight and road impacts. Therefore, a radial load fatigue analysis must be performed, to assess whether the hub is likely to fail within a specified number of load cycles. The test method for radial load is formulated in accordance with the national standard GB/T 5334-2021 "Requirements and Test Methods for Bending and Radial Fatigue Performance of Passenger Car Wheels", and we have made corresponding adjustments according to actual use conditions.

According to GB/T 5334-2021 "Requirements and Test Methods for Bending and Radial Fatigue" According to the requirements for determining wheel radial load specified in Passenger Car Wheel Performance, the formula for calculating the hub radial load is as follows:

$$F_r = F_v \cdot K \quad (1)$$

Specifically:

F_v ——the maximum vertical static load of the wheel is clearly specified by the vehicle or wheel manufacturer, which is the rated load of the wheel.

K ——Weighting factor, set to 2 (see Table 4)

Table 4. Correlation Coefficient of Radial Load.

Material	Strengthening Factor K	Minimum Number of Cycles
Steel	2.25	600000
Aluminum Alloy	2	1150000

The maximum load acting on the hub [6] can be expressed as follows:

$$F_{\max} = \frac{W_1 \times n_s}{3} + \frac{G}{6} \quad (2)$$

Among these,

W_1 —— Curb Weight.

n_s —— Load Influence Factor, generally taken as 1.2.

G ——Fully Loaded Vehicle Weight.

Based on the parameters of the reference vehicle, the curb weight of the vehicle is 2,030 kg, and the load under fully loaded conditions is $G = (2180 - 1730) \times 9.8 =$

4410N. Therefore, the radial load acting on the wheel hub was calculated to be 4325 N.

In addition, the wheel hub is subjected to tire inflation pressure. For reference tire, the recommended pressure ranges from 2.5 to 2.7 bar. According to Table 5, the test inflation pressure was set at 3.5 bar.

Table 5. Test Inflation Pressure.

Tire Inflation Pressure under Service Load (kPa)	Test Inflation Pressure (kPa)
≤ 160	280
161-280	450
281-450	550
≥ 450	≥ 1.2 times the operating pressure

Since the wheel hub does not come into direct contact with the ground, gravitational force is applied, the gravitational acceleration of 9.8 m/s^2 imposes a fixed support constraint on the inner surfaces of the five bolt holes on the hub, All loads were applied as shown in Figure 5.

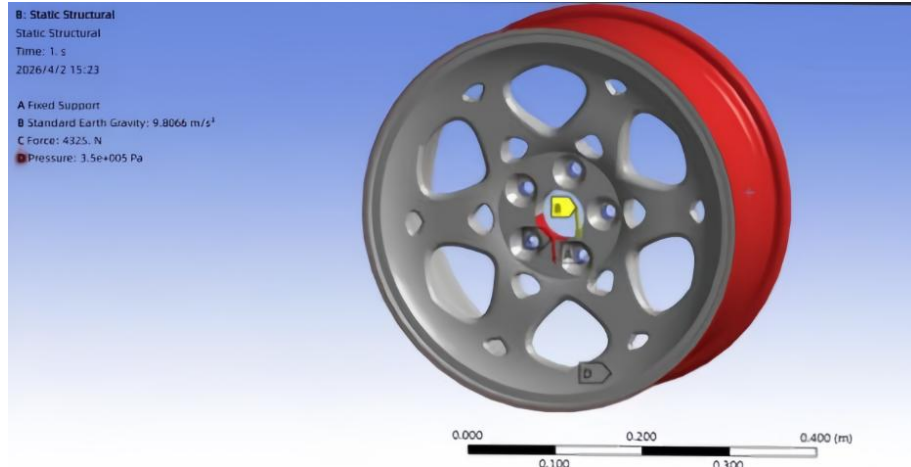
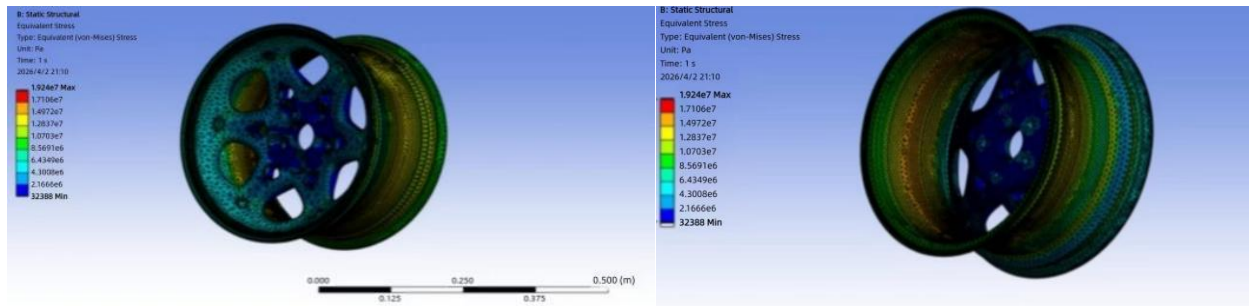


Fig 5. Schematic Diagram of Constraints and Loads.

4.2.2. Results of the radial load analysis

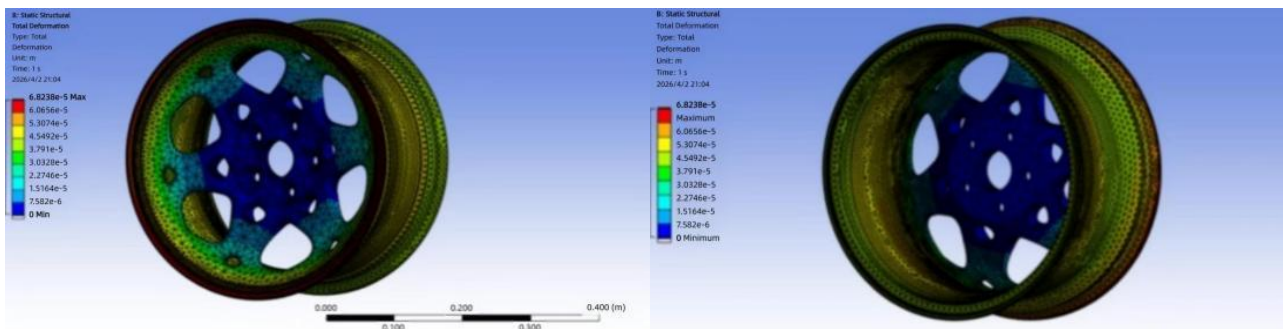
Finite element simulations were conducted to analyze the radial fatigue testing of the hub. The simulation results yielded stress distribution maps and total strain maps of the hub, Fatigue life probability plots and safety factor probability plots, as shown in Figure 6-8.



Equivalent Stress Contour Map (Front and rear views)

Fig 6. Stress contour plot for radial load.

Figure 6 shows the equivalent stress contour plots for the front and rear surfaces of the hub under radial fatigue analysis. ANSYS simulation results indicate that, The maximum stress in the hub under radial loading is approximately 19MPa, significantly lower than the yield strength of aluminum alloy (205 MPa), meets design requirements. The place with the maximum stress appears at the rear of the hub, that is, the contact area between the rim and the spoke, which is consistent with the stress distribution observed in actual use.



Total deformation cloud map (Front and rear views)

Fig 7. Contour plot of total deformation under radial load.

Figure 7 shows the total deformation contour plot of the front and rear surfaces of the hub under radial fatigue analysis. ANSYS simulation results indicate that the maximum deformation under radial loading is approximately 0.06 mm, The results are within acceptable limits. The maximum deformation occurs at the rim, and the results meet the design requirements.

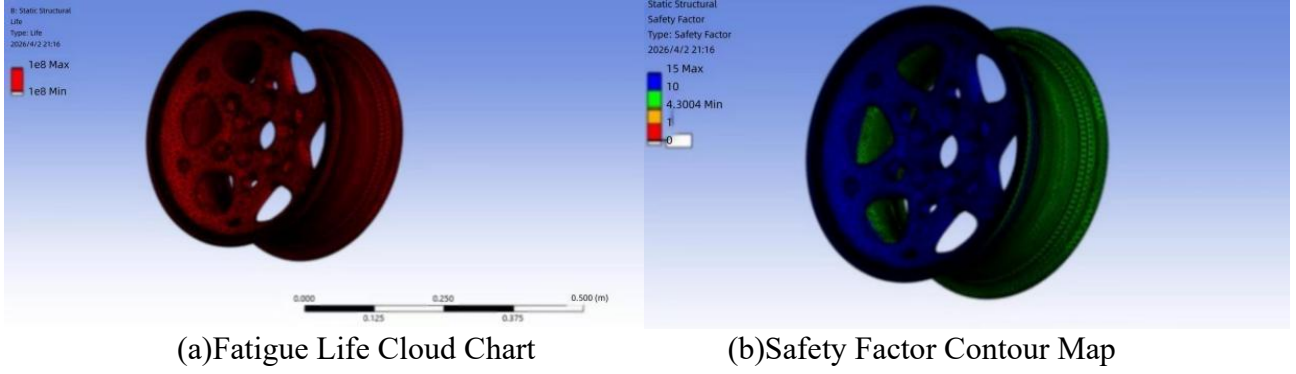


Fig 8. Radial Fatigue Analysis Contour Plot.

Figure 8 shows a scatter plot of fatigue life and safety factors under radial loads. According to Table 3, the number of iterations is set to 10,000, 000, and the engineering data source also incorporates an appropriate S-N curve for A356 aluminum alloy. ANSYS analysis results indicate that the hub structure fully meets the required fatigue life specifications under radial loads, the minimum safety factor is approximately 4.3, the gap between the rim and the spokes at the connection point is significantly greater than 1, this indicates that the structure is safe under radial loading conditions.

4.3. Bending Load Analysis

4.3.1. Loads and Boundary Conditions

According to GB/T 5334-2021 "Requirements and Test Methods for Bending and Radial Fatigue Performance of Passenger Car Wheels", Figure 9 shows the principle of the wheel bending load test, and in this test, the bending load applied to the hub is essentially the bending moment acting on the center hole. To simulate the effect of bending moment in the test, a loading arm of the appropriate length must be used, the effect of bending moment is simulated by applying an eccentric force at the end of the loading arm.

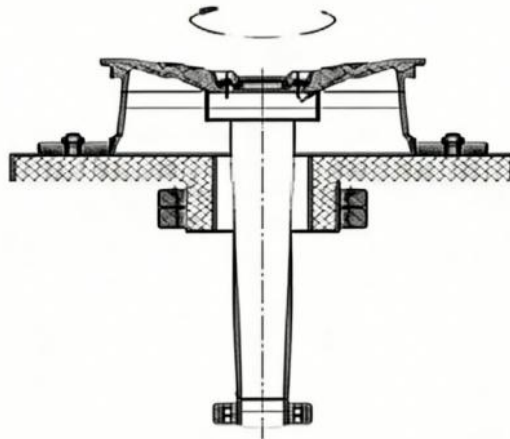


Fig 9. Principle of bending fatigue testing.

According to the GB/T 5334-2021 standard, the magnitude of the bending moment can be expressed as:

$$M=(R\times\mu+d)\cdot F_v\cdot S \quad (3)$$

Among these,

R—The static load radius of a tire refers to the maximum static radius of the wheel under load, as specified by the vehicle or wheel manufacturer, measured in meters (m),

μ — The tire-road friction coefficient is set to 0.7 (see Table 5).

d—Wheel offset can be positive or negative, with a value of 30 mm.

F_v — The maximum vertical static load or rated load for the wheel, as specified by the vehicle or wheel manufacturer, expressed in newtons (N).

S—The amplification factor is set to 1.6, as shown in Table 6.

Table 6. Bending Load-Related Coefficient.

Material	Coefficient of Friction μ	Strengthening FactorS	Minimum Number of Cycles
Steel	0.7	1.60	30000
Aluminum	0.7	1.60	120000

Based on the above parameters, the bending moment M acting on the hub is determined to be 2136 N·m. The length of the loading arm is set to 400 mm. The resulting eccentric force $f = \frac{M}{L}$ is 5341 N. The force acts at the end of the loading arm. We set a fixed support constraint on the inner surface of the rim, which can limit the rotation of the hub and simulate the reaction force of the tire on the hub.

The complete load and boundary conditions are shown in Figure10:

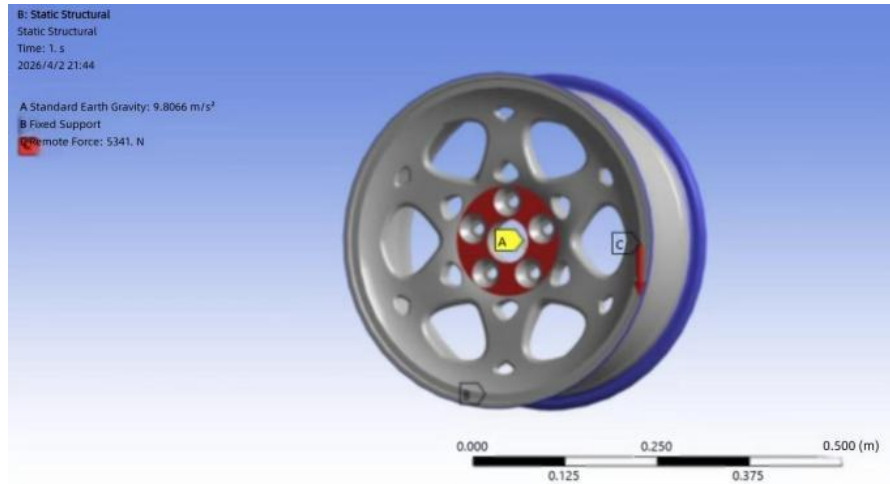
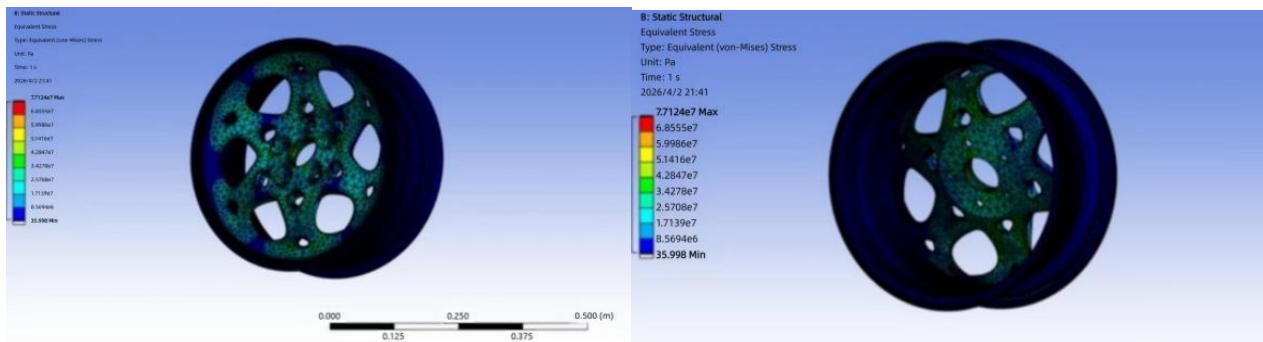


Fig 10. Schematic diagram of bending load and constraints.

4.3.2. Results of bending fatigue analysis

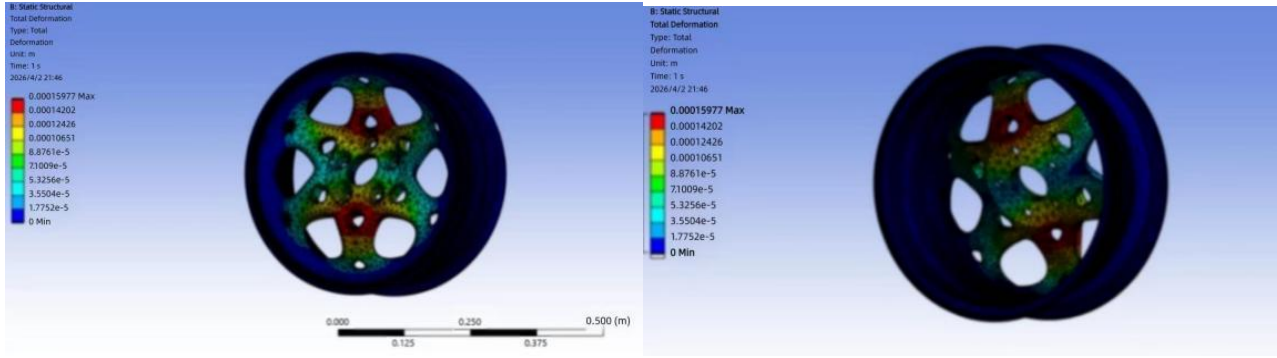
Finite element simulations were conducted to analyze the bending fatigue tests of the hub. The results include stress distribution maps, total strain maps, fatigue life maps, and safety factor maps for the hub, as shown in Figures 11–13.



Equivalent Stress Contour Map (Front and rear views)

Fig 11. Equivalent stress contour plot under bending load.

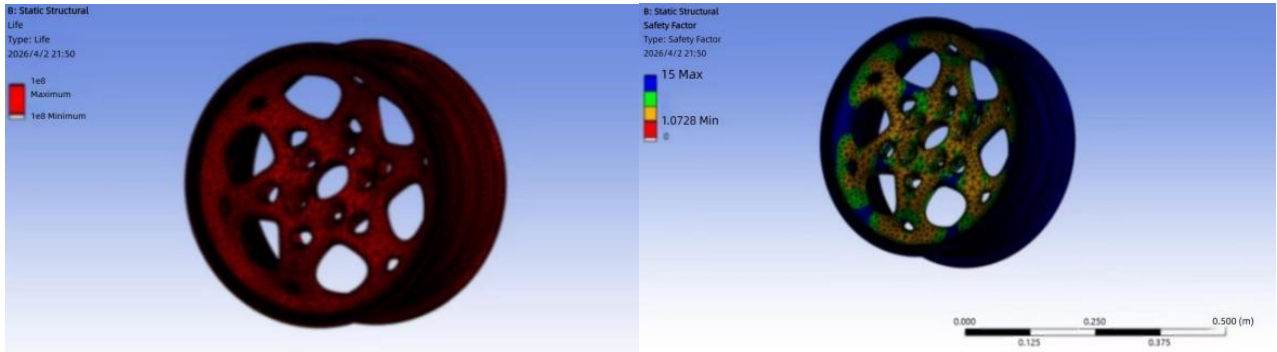
Figure 11 shows the equivalent stress contour plot from the bending fatigue analysis. ANSYS simulation results show that the maximum equivalent stress under bending load is about 77.1 MPa, which is lower than the yield strength of aluminum alloy, so it meets the design requirements. The peak stress occurs on the rear surface where the rear wheel rim meets the spokes, stress concentrations are relatively high in this area, making it a potential hazard zone.



Overall deformation map (Front and rear views)

Fig 12. Contour plot of total deformation from bending load analysis.

Figure 12 shows the total deformation contour plot from the bending fatigue analysis. ANSYS simulation results indicate that the maximum deformation under bending loads is approximately 0.15 mm. The position with the maximum displacement appears at the junction of the central claw-shaped structure and the petal-shaped structure of the spoke. The degree of deformation is moderate and meets the design requirements.



(a) Fatigue Life Cloud Chart

(b) Safety Factor Contour Map

Fig 13. Contour plot of bending fatigue analysis.

Figure 13 shows the fatigue life contour plot obtained from the bending fatigue analysis. The S-N curve settings are consistent with the radial fatigue analysis, set the minimum number of iterations to 1,000,000 (see Table 6). ANSYS analysis results show that the minimum fatigue life of the hub under bending loads is approximately 1,000,000 cycles, the minimum safety factor is approximately 1.07. These two values are very reasonable.

4.4. Modal Analysis

4.4.1. Theoretical Background

Modal analysis can be divided into free-mode and constrained-mode analysis. Free mode refers to the natural mode of the structure under unconstrained state, while constrained mode is the mode generated by the structure under specific boundary conditions; The essence of modal analysis lies in transforming and calculating coordinates. Free mode refers to the natural mode of the structure under unconstrained state [7], while constrained mode is the mode generated by the structure under specific boundary conditions. To investigate whether the hub will resonate with an external excitation, this study conducted a free modal frequency analysis of the hub.

Modal analysis involves studying the relationships between a system's physical parameter model, modal parameter model, and nonparametric model. The specific method is as follows: convert physical coordinates into modal coordinates, Decouple the differential equations describing the vibrations of a linear system into independent equations described by modal coordinates and modal parameters, Thus, the modal parameters are determined. According to D'Alembert's principle, for a linear vibrating system with N degrees of freedom, the differential equation of motion is:

$$[M]\{\ddot{x}(t)\}+[C]\{\dot{x}(t)\}+[K]\{x(t)\}=\{F(t)\} \quad (4)$$

In the equation, $[M]$ let this be the quality matrix, $[c]$ let the damping matrix be, $[K]$ let be the stiffness matrix, $F(t)$ let the force vector be, $x(t)$ let be the displacement vector, $\dot{x}(t)$ for the velocity vector, $\ddot{x}(t)$ for the acceleration vector, t for time.

4.4.2. Analysis Results

And then import the hub model into ANSYS software, with mesh division and material property settings consistent with those used in static load and fatigue load analysis. Since the first six frequencies in free modal analysis correspond to rigid-body modes that approach zero, has no practical engineering significance, Therefore, only the modal frequencies from the 7th to the 12th order are analyzed.

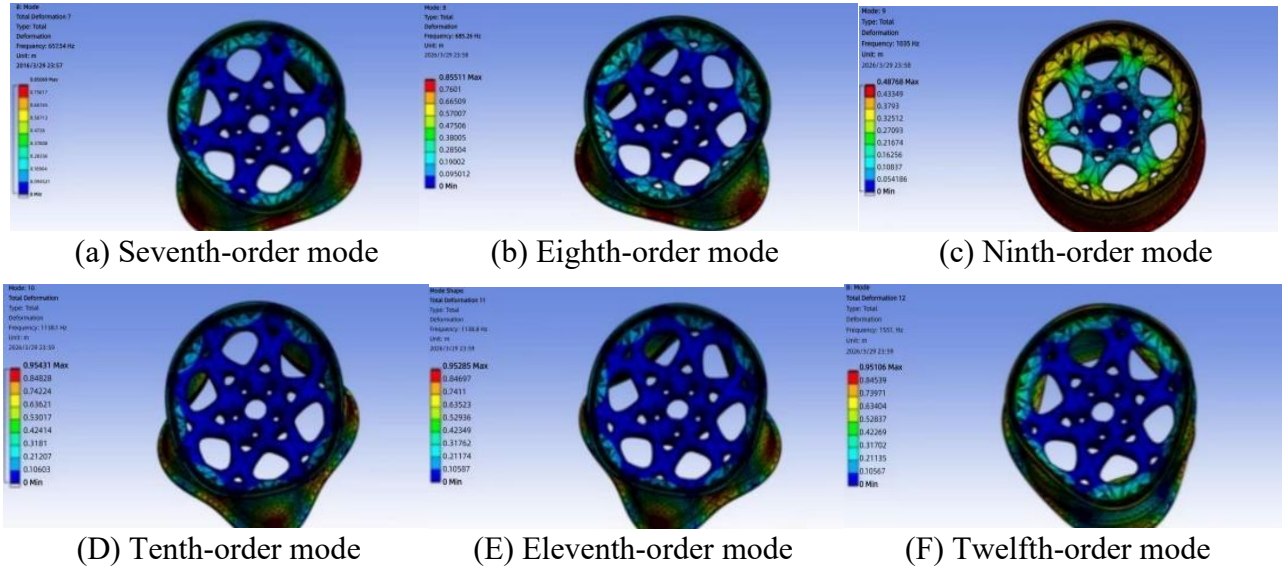


Fig 14. Modal shapes for each order.

Table 7. Modal Frequencies for Each Order.

7th	8th	9th	10th	11th	12th
385.2	421.7	478.3	532.6	589.1	643.8

Figure 14 shows the mode shapes of the hub from the 7th to the 12th order, and Table 7 summarizes the corresponding frequency values. A comprehensive comparison of these six frequency values with typical external excitation frequencies encountered during actual vehicle operation, this is used to determine whether the hub will experience resonance due to external excitation during operation, thereby ensuring the dynamic safety of the structure. Figure 14 shows the mode shapes of the hub from the 7th to the 12th order, and Table 7 summarizes the corresponding frequency values [8]:

$$f_{road} = un \quad (5)$$

In the equation: u represents the vehicle's maximum speed, taken as the standard maximum speed of 120 km/h; where n is the spatial frequency of the road surface, with a range of 0.11 to 2.83 m^{-1} ; Calculations show that the maximum frequency of road excitation is approximately 94 Hz, this value is well below the 7th natural frequency of the hub, so it will not cause resonance. As the primary source of vibration in new energy vehicles, the vibration frequency of the drive motor is calculated according to Equation (4.6) [9]:

$$f = \frac{n \times p}{60} \quad (6)$$

In the equation: n is the rated speed of the drive motor, taken as 6000 r/min; where the number of pole pairs of the motor is 2, and the calculation shows that the maximum excitation frequency of the motor is about 200 Hz. At the same time, the excitation frequency caused by the drive shaft imbalance is approximately 83 Hz, the excitation frequency caused by tire imbalance is approximately 14.7 Hz. A comprehensive comparison shows that the frequencies of the four typical external excitation types mentioned above are all significantly lower than the 7th natural frequency of the hub, 385.2 Hz, It can therefore be concluded that this petal-shaped biomimetic wheel hub will not resonate with external excitations under actual driving conditions, both the dynamic characteristics and vibration resistance meet design and safety requirements[8, 9].

Modal Summary: A free modal analysis of the petal-inspired hub reveals that the first six modes are rigid-body modes, with frequencies approaching zero, have no practical engineering significance; Modes 7 through 12 are effective plastic modes, with natural frequencies ranging from 385.2 to 643.8 Hz. Compare and verify it against four typical excitation frequencies: road irregularities, drive motor, drive shaft imbalance, and tire imbalance. The results show that the excitation frequencies for all types are significantly lower than the 7th natural frequency of the hub, effectively avoiding the resonance range. This indicates that this lightweight wheel has good dynamic stability under actual driving conditions, and its vibration resistance and dynamic characteristics can all meet the safety and design requirements.

5. Conclusion

This study uses petal-shaped structural elements to design a claw-petal hub, this spoke structure has a novel and unique design, which can provide some reference for actual production, and at the same time, the hub weight is controlled at 8.9 kg, successfully achieving the goal of weight reduction design, and also well balancing innovative design and lightweight performance.

Simulation analyses conducted using ANSYS Workbench have validated the hub's reliability: The maximum equivalent stress under radial loading is approximately 19 MPa, with a minimum safety factor of 4.3, the maximum equivalent stress under bending load is approximately 77.1 MPa, with a minimum safety factor of 1.07, The fatigue performance for both types of loads meets the design standards; modal analysis shows that, Under four typical external excitation conditions—such as uneven road surfaces and unbalanced tires—the wheel hub did not exhibit any resonance, both the overall mechanical properties and dynamic characteristics meet the design requirements.

References

- [1] Wang Kexue, Wang Dongbo, Huang Yong, et al. Current Status of Research on the Application of Aluminum Alloy Materials in Automotive Weight Reduction [J]. Aluminum Processing, 2022 (6): 3-5.
- [2] Lin Liao Feng. Research on the Current Status and Future Innovation Trends of Automotive Weight Reduction Technologies [J]. Automotive Parts, 2025 (6): 79-81.
- [3] Wang Qingzheng, Gao Shengping, Li Hui, et al. Lightweight Design and Material Innovation for New Energy Vehicles: A Study on Methods to Improve Energy Efficiency and Driving Range Auto Times, 2025: 122-124.
- [4] Zhang Shengchao. Lightweight Design of Automotive Wheels [D]. Qingdao: Qingdao University, 2020.
- [5] Guo Jiale, Fang Zichun. Research on the Application of Finite Element Analysis in the Lightweight Design of New Energy Vehicle Wheel Hubs [J]. AUTO TIME, 2025: 99-101.
- [6] Tian Jinyun. Numerical Simulation of the Automotive Wheel Hub Casting Process and Fatigue Performance Evaluation [D]. Yanshan University, 2014. (in Chinese)
- [7] Chen, Yiping, Wang Xu, Liu Yi, et al. Modal Analysis of a 10 kV Power Transformer Iron Core Based on ANSYS [J]. Equipment Maintenance Technology, 2024, (01):59-61. DOI: [1][1][1][1][1] 10.16648 /j.cnki.1005-2917.2024.01.014. (in Chinese)

- [8] Bai Yuan, Zhang Shuai, Liu Rong'e. Lightweight Design and Analysis of Automotive Wheel Hubs Based on Workbench [J]. Automotive Practical Technology, 2025 (3):61-65.
- [9] Chen Xiaoming, Yu Jianye, Liu Shun, et al. Lightweight Design of Aluminum Alloy Wheels for SUVs Based on Finite Element Simulation [J]. Journal of Northeastern University (Natural Science Edition), 2025, 46 (1):99-109.