Lightweight Design of New Energy Vehicle Wheel Hub Based on ANSYS Workbench

Wangyu Zhou *

Department of mechanical engineering, The Hong Kong Polytechnic University, Hong Kong, China

* Corresponding Author Email: zadmiy999@gmail.com

Abstract. Lightweighting of new energy vehicles is a core approach to improve cruising range and reduce battery costs. As a key load-bearing component, the structural optimization of wheel hubs is crucial to the overall vehicle performance. This paper takes the wheel hub of a new energy car as the research object, combining the streamlined structure of bird wings and the characteristics of lightweight bones to design a five-spoke rotating lightweight wheel hub. 6061-T6 aluminum alloy is used as a lightweight material. A 3D model is established through Creo. Static, modal, bending fatigue and radial fatigue analyses are carried out using ANSYS Workbench. The results show that the mass of the lightweight wheel hub is reduced from 13.2kg to 9.38kg, with a weight reduction rate of 28%. Moreover, the simulation results show that the maximum stress, deformation and fatigue life all meet the design standards, achieving a balance between strength and lightweighting. This provides theoretical and technical references for the research on lightweighting of new energy vehicle wheel hubs.

Keywords: New energy vehicles, Wheel hub, Lightweighting, Finite element analysis.

1. Introduction

With the increasingly prominent global energy shortage and environmental pollution problems, new energy vehicles, as the core carrier of green travel, have become the mainstream direction of the transformation of the automotive industry. According to the data from the China Association of Automobile Manufacturers, in 2023, the production and sales of new energy vehicles in China both exceeded 9 million, with a market share of over 30%, showing a strong development trend [1]. However, due to the integration of the three-electric system, the overall mass of new energy vehicles is 20% - 40% higher than that of traditional fuel vehicles, which directly leads to problems such as shortened cruising range and rising battery costs. Studies have shown that a 10% reduction in the mass of new energy vehicles can increase the cruising range by 5% - 10% and save 15% - 20% of battery costs. Therefore, lightweight design has become a key means to break through its performance bottlenecks [2].

As a key load-bearing component connecting the vehicle body and tires, the mass proportion of the automobile wheel hub is not as large as that of the body and chassis, but it directly affects the unsprung mass and driving stability of the vehicle, so the demand for lightweighting is particularly urgent [3]. Traditional wheel hub design focuses more on strength and safety but insufficiently considers the balance between lightweighting and performance. At present, wheel hub lightweighting is mainly achieved through structural optimization, application of lightweight materials and advanced forming processes. Among them, structural optimization has the advantages of low cost, easy implementation and complete research theory. For example, Fang Baotao and Xu Dan achieved an effective weight reduction of about 8.6% under the condition of meeting static load requirements only by adjusting the number and thickness of wheel hub spokes [4]. Wu Jiantao, Sun Li and others established a wheel hub model with reference to the shape of alternate phyllotaxy plants, combined the changes of shape characteristics with parameterization technology by means of computer technology, and constructed a parametric bionic model of the wheel hub through a generative algorithm, providing a large number of references for wheel hub shape design [5].

This paper takes the aluminum alloy wheel hub of a family car as the research object, aiming at its lightweight demand, optimizes the wheel hub structure combined with the bionic design concept.

Verifies the static, modal and fatigue performance of the lightweight wheel hub through finite element analysis technology. Aiming to explore a wheel hub design scheme that takes into account both strength and lightweighting, and provides references for the lightweight upgrading of key components of new energy vehicles.

2. Wheel Hub Structure

The wheel hub is mainly composed of a rim and spokes, as shown in Figure 1. The rim is the part in direct contact with the tire, and its structural features include the barrel, outer lip, valve stem, etc., which directly affect the safety, operability and energy efficiency of the vehicle. However, the national standard GB/T 3487-2005 restricts the specification and size of the rim in many aspects, especially for the J-type contour wheel hub commonly used in family cars, whose size, angle, thickness and other aspects have relatively strict numerical specifications, so the structural design of the rim is usually not considered.

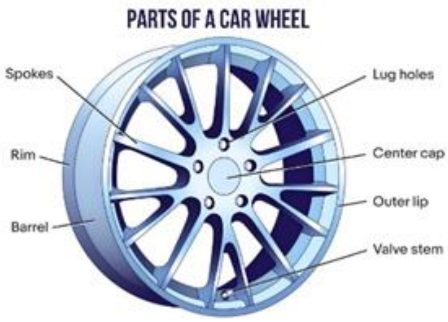


Fig 1. Wheel Hub Structure

The spoke is the part connecting the rim and the center of the hub, mainly playing the role of sharing the vehicle load and transmitting power. The differences between different automobile wheel hubs are mainly reflected by the spokes, but they generally conform to the following characteristics. Based on a centrally symmetric structure to ensure that the hub can evenly transmit the vehicle weight and the load while driving to the tire. Adopting large rounded corners and arc transitions to make each component connect smoothly and reduce stress concentration. Realizing the balanced design of hub strength and lightweighting through the reasonable proportion of hollow area. The shape design improves users' perceptual cognition of the vehicle style through the combination of shape elements [6].

3. Wheel Hub Model Design

Takes the aluminum alloy wheel hub of a family car as an example. The tire model is 245/45 R19 8J, with a wheel hub diameter of 482.6 mm, a wheel hub width of 203.2 mm, and a wheel hub mass of 13.2 kg. The wheel hub is a positive-offset wheel, with the centerline offset from the hub mounting surface by 30 mm. The mounting surface is biased to the outside of the wheel hub centerline. The wheel hub has a bolt pitch of 5×112 mm and a center hole diameter of 60.1 mm.

The wheel adopts the current mainstream five-spoke rotating structure. The shape design refers to the bird wing, as shown in Figure 2.

Fig 2. Schematic Diagram of Wing Structure

The wing, as the core organ for birds' flight, is highly adapted to the movement function. It has a streamlined wing shape structure. The feathers covering the surface form a smooth plane, which reduces air resistance while guiding airflow along the wing, creating conditions for generating power. In addition, it has a skeletal structure that balances lightweight and high strength. The hollow bones significantly reduce the total body mass, while the mesh-like bone walls ensure supporting strength. The 3D solid model of the designed wheel hub is shown in Figure 3.

Fig 3. 3D Solid Model of the Wheel Hub

After integrating the above elements, the wheel has streamlined flat spokes, and the hollow areas are reasonably distributed, with stepped grooves on the spoke surface. The design parameters of the wheel are shown in Table 1.

Table 1. Wheel Hub Parameters

Diameter/mm	Width/mm	offset/mm	Center Hole/mm	Pitch Circle Diameter/mm	Wheel mass/kg
482.6	203.2	25	60.1	112	9.38

4. Finite Element Analysis of the Wheel Hub

ANSYS is a large-scale finite element analysis software that integrates structural, fluid, electric field, magnetic field, and acoustic field analysis. It provides more than 100 types of elements, which can simulate various structures and materials in engineering. It can interface with most CAD software such as Creo and AutoCAD to realize data sharing and exchange functions, and is widely used in mechanical manufacturing, energy, automotive transportation, and other fields. It is a widely used computer-aided engineering software worldwide. Among them, Workbench is an integrated work platform in the ANSYS software, which can manage the ANSYS software in one place, realize synchronous operations such as modeling, meshing, and physical simulation, facilitate users to integrate data between different simulation functions, and thus effectively conduct data analysis.

4.1. Material Properties

Lightweight materials refer to materials with low density characteristics while meeting the core performance requirements such as structural strength and stiffness. Their core value lies in achieving energy efficiency improvement and performance optimization through weight reduction, which is the most direct and effective way in automotive lightweight. The designed wheel hubs are made of 6061-

T6 aluminum alloy, which has a higher yield strength than the currently mainstream A356 aluminum alloy. The physical parameters of the two materials are shown in Table 2.

Table 2. Wheel Hub Material Properties

Material	Elastic Modulus/GPa	Poisson's Ratio	Density/（g/cm3）	Yield Strength /MPa
6061-T6 aluminum alloy	69.04	0.33	2.7	259.2
A356 aluminum alloy	72.4	0.33	2.68	207.0

Since elastic deformation has no significant impact on wheel function, differences in elastic modulus are not necessary. The higher yield strength of 6061-T6 aluminum alloy indicates that it is more capable of coping with working conditions such as bending, allowing for greater flexibility in shape and size adjustment. Although its density is slightly higher than that of A356, its comprehensive performance is more suitable as a design material for lightweight wheels.

4.2. Static Analysis

Before conducting finite element analysis, it is necessary to discretize the model and decompose the model into an appropriate number of element regions. Due to the uneven distribution of the overall structure of the wheel, the thickness of different parts varies, and most parts are curved surfaces. To better mesh the curved surfaces and edges of the wheel, it is necessary to make the meshing more refined. Therefore, tetrahedral elements are used for meshing, the element size is set to 10 mm, and the mesh size of the overall structure is also 10 mm. The number of nodes is 184,825, and the number of elements is 102,029. The meshing result is shown in Figure 4 [7].

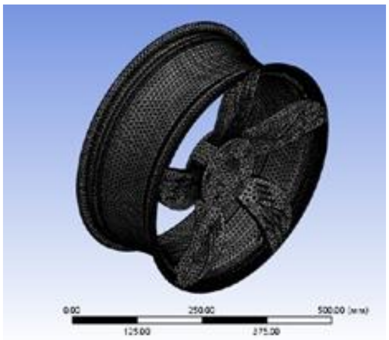


Fig 4. Wheel Hub Meshing

Static analysis simulates the state of the wheel under its own weight, vehicle body weight, and tire pressure when it is stationary. According to the installation and stress conditions of the car wheel, the wheel is fixed on the wheel bearing through the wheel bolt holes. When conducting static simulation of the wheel, fixed constraints need to be applied to the 5 bolt holes of the wheel, as shown in Figure 5.

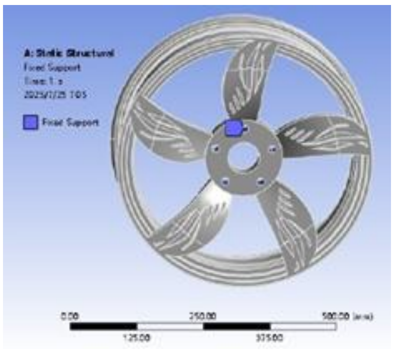


Fig 5. Static Stress Constrains

The main parameters of the reference car are shown in Table 3.

Table 3. Vehicle Parameters

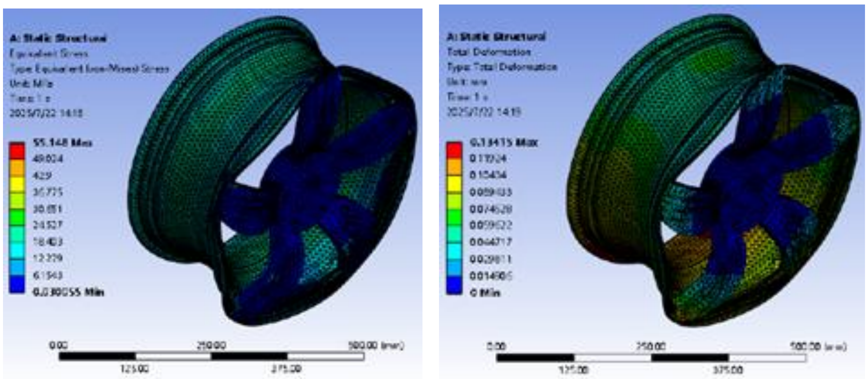
Curb Weight /kg	Full Load Mass /kg	Maximum Power /kW	Maximum Torque /Nm	Wheel Mass /kg	Tire Pressure /MPa
2060	2435	196	390	13.2	250

Then, the maximum load borne by a single wheel is:

$$F_{max} = \frac{wn_i}{3} + \frac{G}{6} \tag{1}$$

In the formula:
W is the curb weight of the vehicle.
n_i is the load influence coefficient, generally taken as 1.21.
G is the maximum load of the vehicle which is 3675 N.
Therefore, $F_{max} = \frac{wn_i}{3} + \frac{G}{6} = 8755\ N$.

The tire pressure borne by the wheel is 250 MPa, and the corresponding test inflation pressure is 450 MPa according to the national standard GB/T 5334-2021. The results obtained after applying the above loads are shown in Figure 6. It can be seen from Figure 6 (a) that the maximum stress is 55.15 MPa, which occurs at the connection between the rim and the spokes, and the maximum value is much less than the material yield stress. It can be seen from Figure 6 (b) that the maximum deformation is 0.13 mm, and the maximum displacement occurs at the rim.



(a) Stress Diagram (b) Deformation Diagram

Fig 6. Static Stress Condition Diagram

4.3. Modal Analysis

The wheel is affected by various vibration frequencies during operation. Modal analysis aims to avoid resonance of the wheel under different working conditions. The main excitations that the vehicle is subjected to during driving are as follows [8]:

(1) Road excitation

The excitation frequency input by road unevenness is:

$$f_{road} = un \tag{2}$$

In the formula:
n is the spatial frequency, ranging from 0.011$m^{-1}$to 2.83m^{-1}.
u is the vehicle speed, with a vehicle speed of 80 - 120 km/h.
Therefore, the maximum value of road excitation is $f_{road} = 94.33$Hz.

(2) Unbalance caused by tires

The excitation frequency caused by tire unbalance is expressed as:

$$f_{tire} = \frac{u}{2\pi r} \tag{3}$$

In the formula r is the tire rolling radius which is 351.55 mm. Therefore, the maximum excitation frequency caused by tire unbalance can be calculated as $f_{tire} = 15.1$ Hz.

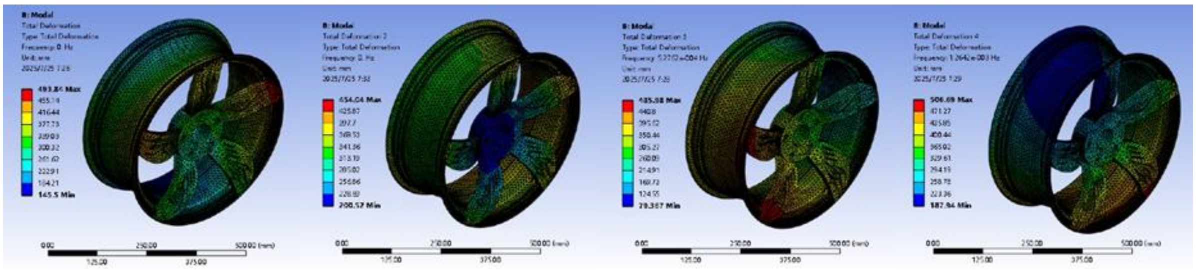
(3) Unbalance caused by the transmission system

The excitation frequency caused by the unbalance of the transmission shaft is:

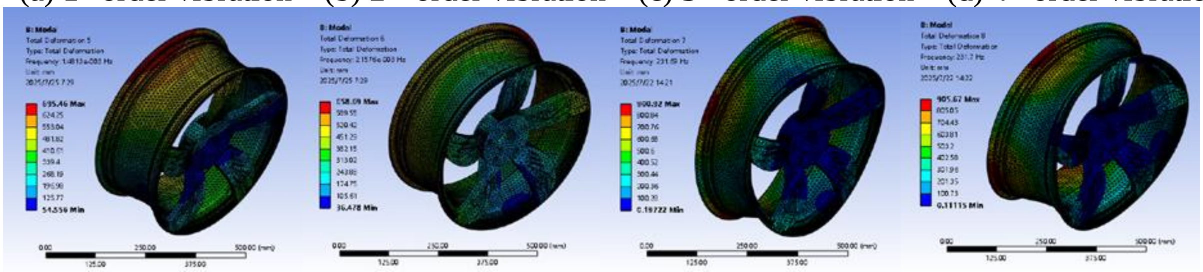
$$f_{Trans} = \frac{ui_0}{2\pi r} \tag{4}$$

In the formula, i_0 is the main reduction ratio, the maximum value is taken as 5 here. Taking the same vehicle speed as above, the maximum excitation frequency caused by the unbalance of the transmission shaft is $f_{Trans} = 75.4$Hz.

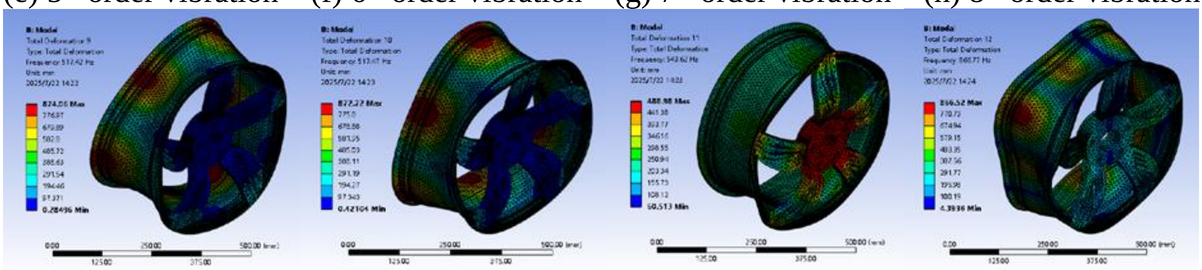
The frequencies and deformations of each order of vibration modes of the wheel hub under free mode, obtained using ANSYS, are shown in Figure 7.



(a) 1st order vibration (b) 2nd order vibration (c) 3rd order vibration (d) 4th order vibration



(e) 5th order vibration (f) 6th order vibration (g) 7th order vibration (h) 8th order vibration



(i) 9th order vibration (j) 10th order vibration (k) 11th order vibration (l) 12th order vibration

Fig 7. Vibration Mode Diagrams

Table 4. Modal Frequencies of Orders 7–12

Modal order	7	8	9	10	11
Frequency/Hz	231.69	231.70	517.42	517.47	543.67

It can be found from Figure 7 that the natural frequencies of the first 6 orders are close to zero. This is because in the rigid body mode, the wheel hub only undergoes translation or rotation without deformation. The elastic body mode is the key mode for evaluating the dynamic characteristics of the structure. In the free mode analysis, since the elastic body mode starts from the 7th order, only the simulation results of the last 6 orders of vibration modes are analyzed.

As shown in Table 4, the frequency range of the wheel under free mode is 231.69–866.77 Hz. During the vehicle's operation, the maximum road excitation frequency is 94.33 Hz. The excitation frequency caused by tire unbalance is 15.1 Hz and the excitation frequency caused by the unbalance of the transmission shaft in the transmission system is 75.4 Hz. Through comparison, it is found that all these excitation frequencies are much lower than the frequencies of the wheel hub under free mode, which can effectively avoid resonance risks and meet the design requirements.

4.4. Bending Fatigue Analysis

Turning is a common working condition during vehicle operation, which aims to simulate the bending moment load borne by the wheel when the vehicle turns, brakes, or tilts laterally due to uneven road surfaces. In accordance with the relevant requirements of the national standard GB/T 5334-2021, a loading arm is required when conducting the bending test of automobile wheels. According to the wheel design, the length of the loading arm is 1000 mm, and the shaft diameter is 60.1 mm. The shaft of the loading arm is connected to the center hole, with the radius being the same as the hole diameter. A 3D model of the automobile aluminum alloy wheel hub was established in Creo, as shown in Figure 8.

Fig 8. Geometric Model of Wheel hub and Loading Arm

T The bending moment M is calculated as follows:

$$M = (R \times \mu + d) \times F \times S \tag{5}$$

In the formula:

R is the static load radius.

μ is the friction coefficient, generally taken as 0.7.

d is the wheel offset, with a value of 25mm.

F is the maximum rated load of the wheel, which is 8755 N.

S is the strengthening coefficient, taken as 1.6.

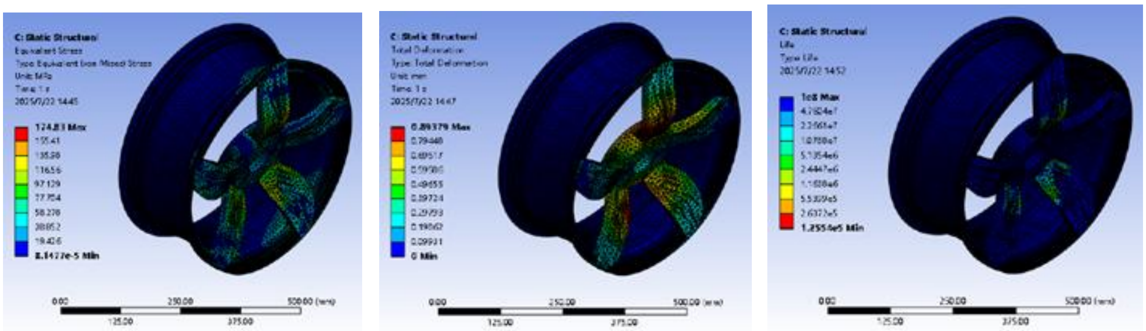
Then, $M = 3797.4\ Nm$.

The eccentric force borne by the wheel is:

$$f = \frac{M}{l} \tag{6}$$

In the formula, l is the length of the loading arm, taken as 1 m. The eccentric force borne by the wheel is calculated as f=3797.4N.

In accordance with the standards of the bending fatigue test, fixed constraints were applied to the rounded ends on both sides of the rim, and a concentrated force load was applied to the far end of the shaft to simulate the bending load test. The equivalent stress cloud diagram, equivalent displacement cloud diagram, etc., of the wheel were obtained, as shown in Figure 9 (a) and Figure 9 (b).



(a) Stress Diagram (b) Strain Diagram (c) Bending Fatigue Life Diagram

Fig 9. Bending Condition Diagram

Without considering the stress direction, only one case under bending load was analyzed, that is, the load was applied directly opposite the spokes. Under the static bending moment load, the maximum dangerous point of the aluminum alloy wheel structure is located at the connection between the spokes and the rim, with a maximum stress of 174.83 MPa. Since these two parts are directly connected, stress concentration at the connection is difficult to avoid even through methods such as filleting or thickening. In addition, the hollow parts on the spokes also show relatively large stress concentration, but it does not exceed the yield strength of the material.

The fatigue test is intended to verify the durability of the wheel under the long-term action of alternating loads. According to the test requirements, the bending fatigue cycle of the aluminum alloy wheel should be more than 120,000 times, and the radial fatigue cycle should be more than 900,000 times. The fatigue life distribution of the wheel can be obtained by using the ANSYS fatigue tool, as shown in Figure 9 (c). It can be seen from the figure that the minimum fatigue life of the wheel until failure under the action of bending load is 125,000 times, which meets the design requirements for wheel life.

4.5. Radial Fatigue Analysis

Without considering the stress direction, only one case under bending load was analyzed, that is, the load was applied directly opposite the spokes. Under the static bending moment load, the maximum dangerous point of the aluminum alloy wheel structure is located at the connection between the spokes and the rim, with a maximum stress of 174.83 MPa. Since these two parts are directly connected, stress concentration at the connection is difficult to avoid even through methods such as filleting or thickening. In addition, the hollow parts on the spokes also show relatively large stress concentration, but it does not exceed the yield strength of the material.

The radial load fatigue test of the wheel simulates the effects of the vehicle's own gravity and the vertical impact force encountered when the vehicle bumps up and down. In the radial simulation, the wheel is subjected to two types of forces: the maximum rated design load and the radial load. Since the bolt pre-tightening force only affects the stress in a small area around the bolt holes, it is an internal force and can be ignored in subsequent analyses. The radial load direction of the aluminum alloy wheel points to the wheel axis along the radius and acts on the tangential surface of the rim. According to the requirements for the wheel radial load, the calculation formula of the wheel radial load is:

$$F_r = F_v \cdot K_a \tag{7}$$

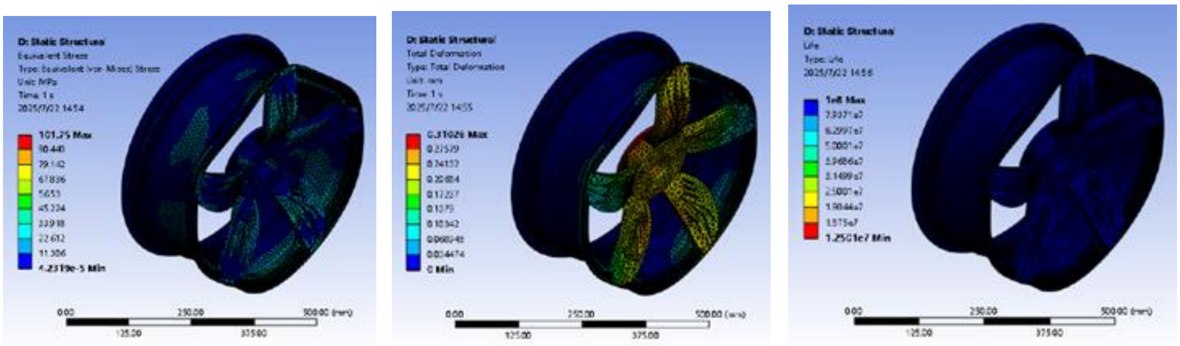
In the formula:

F_v is the maximum rated load of the wheel hub

K_a is strengthening test coefficient, taken as 2.5 according to the test requirements.

Therefore, $F_r = 21887.5N$.

In accordance with the radial load impact test standards, fixed constraints were applied to both ends of the aluminum alloy wheel, and the radial working condition diagram was obtained.



(a) Stress Diagram (b) Strain Diagram (c) Radial Fatigue Life Diagram

Fig 10. Radial Condition Diagram

It can be seen from Figure 10 that the maximum stress borne by the wheel after applying the load is 101.75 MPa, which is less than the yield strength of the aluminum alloy material. The maximum deformation occurs in the outer rim area of the wheel that bears the radial load; the rim part of the entire wheel structure is deformed, and the deformation of the outer rim is significantly greater than that of the inner rim. Overall, the deformation of the wheel structure is small, indicating that the stiffness performance of the wheel structure design is good. The minimum life of the wheel exceeds 1,200,000 times, which is much higher than the 900,000 times required by the test, meeting the performance test standards.

5. Conclusion

1) Structural Design Effectiveness: Inspired by the streamlined and lightweight features of bird wings, the designed wheel hub utilizes wing-shaped spokes with a well-balanced hollowing pattern, resulting in a 28% mass reduction. Workbench static analysis verified that the maximum stress was 55.15 MPa and the maximum deformation was 0.13 mm, meeting static load requirements.

2) Dynamic Performance Reliability: Modal analysis results show that the modal frequency range of the wheel hub's elastic body is 231.69–866.77 Hz, significantly higher than road excitation, tire unbalance excitation, and driveline excitation, effectively avoiding the risk of resonance.

3) Durability Compliance: Bending fatigue and radial fatigue analysis indicate that the minimum bending fatigue life of the wheel hub is 125,000 cycles, and the radial fatigue life exceeds 1.2 million cycles, meeting the requirements for long-term use under alternating loads.

In summary, this article uses ANSYS Workbench for multi-condition analysis, completing a complete lightweight wheel hub design and verification process. The designed wheel hub achieves the weight reduction goal while meeting national standards. The design ideas and verification methods have general reference significance for the lightweight design of other key components in the automotive industry, helping to improve the vehicle's range, reduce battery costs, and promote the performance optimization and industrial development of new energy vehicles.

References

[1] Z. Wang, Annual Report on the Big Data of New Energy Vehicle in China (2023), 1st ed. Singapore: Springer Nature, 2024. doi: 10.1007/978-981-97-4840-2.

[2] Men Lizhong, "Overview of Lightweighting for New Energy Vehicles," Automotive Engineer, no. 6, Brilliance Auto Engineering Research Institute, pp. 15–50, 2019. doi: 10.3969/j.issn.1674-6546.2019.06.004.

[3] CHEN KAI, CHEN XIAODI, XIONG DONGDONG, and WANG LUFEN, “Manufacturing method of light-weight rim, light-weight rim, wheel and automobile,” CN Patent, 2024.

[4] Fang Baotao and Xu Dan, "Lightweight Design of Automotive Aluminum Alloy Wheel Hub Based on ANSYS," Auto Time, no. 21, pp. 119–121, 2023, doi: 10.3969/j.issn.1672-9668.2023.21.040.

[5] Wu Jiantao, Sun Li, and Liu Qingying. "Research on the Construction of a Parametric Model for Bionic Automotive Wheel Hub Design." Journal of Machine Design, vol. 36, no. 8, pp. 115-119, Aug. 2019.

[6] Cheng Zhenbo, Ren Xinrong, Ke Shanjun, et al. "Parametric Design of Passenger Car Wheel Hubs Based on Perceptual Imagery." Journal of Machine Design, vol. 39, no. 4, pp. 135-141, Apr. 2022.

[7] Wei Zhou. Simulation and Optimization Analysis of Aluminum Alloy Wheel Hub Based on Finite Element [D]. Liaoning University of Technology, 2020.

[8] Xiao Chengmei. Structural Analysis and Lightweight Design of Automotive Wheel Hubs [D]. Shandong University of Science and Technology, 2020.