Proceedings of the 2nd World Congress on Industrial Engineering and Operations Management
Windsor, Canada, October 14-16, 2025

Proceedings of the International Conference on Industrial Engineering and Operations Management

Publisher: IEOM Society International, USA DOI: 10.46254/WC02.20250015
Published: October 14, 2025

Modal analysis of wheel hub

Jiaying Zhang
Fairmont preparatory academy
Anaheim, USA
karenzhang1206@163.com

Abstract

Modal analysis is a method used to study the specific vibration characteristics of an object under natural modes. For wheel hubs, modal analysis effectively identifies their vibration characteristics under different limitations, thereby further optimizing the design. This paper establishes a wheel hub model using UG software and performs a modal analysis of a circular aluminum alloy wheel hub using ANSYS. The calculated results of the first ten modal frequencies demonstrate that the overall structure of the wheel hub is relatively safe. This study provides a valuable reference for the simulation analysis of similar types of wheel hubs.

Keywords
Hub, Modal analysis, and wheel.

1. Introduction
As a critical component of a vehicle, the wheel hub plays a vital role in its operation—it bears and transmits the forces and torques between the vehicle and the ground, directly influencing the overall stability, safety, smoothness, and traction performance of the vehicle. Conducting a modal analysis of the wheel hub allows for the study of its natural frequencies, helping to prevent resonance that may occur when these frequencies align with road-induced excitations or tire imbalance frequencies. This paper utilizes ANSYS software to perform a finite element analysis of the wheel hub, calculating its natural frequencies and implementing improvements to achieve a relatively safer wheel hub design model.

Current research on wheel hubs primarily focuses on material innovation, structural optimization, advanced manufacturing, and smart technologies. The wheel hub is a key component of passenger cars, and its weight directly affects overall vehicle performance and fuel efficiency. In China, current research mainly focuses on energy use and emissions of power and long-distance vehicles, while studies specifically on wheel hub lightweighting remain limited(Zhang et al., 2023). During the wheel hub forming process, defects such as folding and edge cracking are common. Current research focuses on using finite element simulation to optimize process parameters (e.g., temperature, deformation speed, and friction coefficient) to improve forming quality and structural strength. To reduce the risk of defects, studies are exploring how controlling deformation temperature and speed can enhance metal flow uniformity, thereby improving the mechanical properties and overall forming precision of the wheel hub (Li, 2023). Sun (2016) conducted mechanical and modal analyses of a vehicle wheel hub using finite element methods. Various lightweight materials were applied to the hub through Hypermesh software, and structural dimensions were optimized. Finally, performance metrics for the wheel hub under different materials were obtained. Driven by energy conservation and environmental protection demands, wheel hub lightweighting has become a key focus in automotive structural optimization. Research commonly adopts finite element analysis combined with lightweight materials and structural optimization to achieve weight reduction goals (Liu, 2022). As a key structural component bearing vehicle weight and road excitation, the wheel hub features complex design requirements with high demands on mechanical performance. Current research focuses not only on structural strength and stiffness, but also on modal and dynamic stiffness analysis to optimize its dynamic response characteristics. By using finite element modal analysis to reveal the vibration behavior of the hub, accurate response models can be established, thereby improving NVH (Noise, Vibration, and

Harshness) performance and enhancing overall vehicle comfort and safety (Jing et al., 2023). Currently, weight reduction design of wheel hub bearing flanges has become a common approach to lower manufacturing and operational costs. However, studies on the performance of flanges after weight reduction remain limited. For third-generation wheel hub bearing inner flanges, recent research has begun to apply modal analysis to investigate how different flange geometries affect performance under lightweight conditions, providing valuable insights for design optimization (Liang et al., 2022).

The purpose of my research is to study the natural frequency and modal shape of the wheel hub and master its vibration frequency in a certain frequency band, so as to effectively avoid the occurrence of resonance and ensure the vehicle to operate in a safe and reliable environment.

2. Modal

2.1 Analytic model building

The automobile wheel hub model was established by using UG modeling software, as shown in Figure 1.

Figure 1. Wheel hub model made with UG software

The experimental wheel hub material studied in this paper is aluminum, and the material properties are (Table 1):

Table 1. Material properties

Density	7850 kg*m^-3
Tensile Yield Strength	2.5E+08 Pa
Poisson’s Ratio	0.3
Shear Modulus	7.6923E+10 Pa
Young’s Modulus	2E+11 Pa

2.2 Formula

The wheel hub can be approximated as a spring-mass system, with its natural frequency f given by:

$$f = \frac{1}{2}\sqrt{\frac{k}{m}}$$

Where k is the equivalent stiffness of the wheel hub, and m is the equivalent mass of the wheel hub.
The equivalent stiffness of the wheel hub, based on the mode of action, can be divided into radial stiffness, axial stiffness, and torsional stiffness.

The radial stiffness kr is:

$$kr = \frac{nEA}{L\cos^{2}\theta}$$

where:
n is the number of spokes
E is the Young's modulus of the material,

A is the cross-sectional area of a single spoke,

L is the length of a spoke,

θ is the angle between the spoke and the radial direction.

The axial stiffness ka is:

$$ka = \frac{3EI}{L^{3}}$$

where:
I is the moment of inertia of the wheel rim cross-section.

The torsional stiffness kt is:

$$kt = \frac{nGJ}{L}$$

where:

G is the shear modulus of the material,

J is the polar moment of inertia of a single spoke.

3. Simulation

First, a free modal analysis was conducted without applying any boundary conditions to the structure. The first six natural frequencies and their corresponding mode shapes were extracted. The simulation results showed that the first six natural frequencies were approximately zero, indicating the simulation conditions are reasonably set. Subsequently, fixed constraints were applied at the mounting plate to simulate the working condition of the hub. A second modal analysis was then performed to evaluate the changes in the dynamic characteristics of the structure under operational constraints.

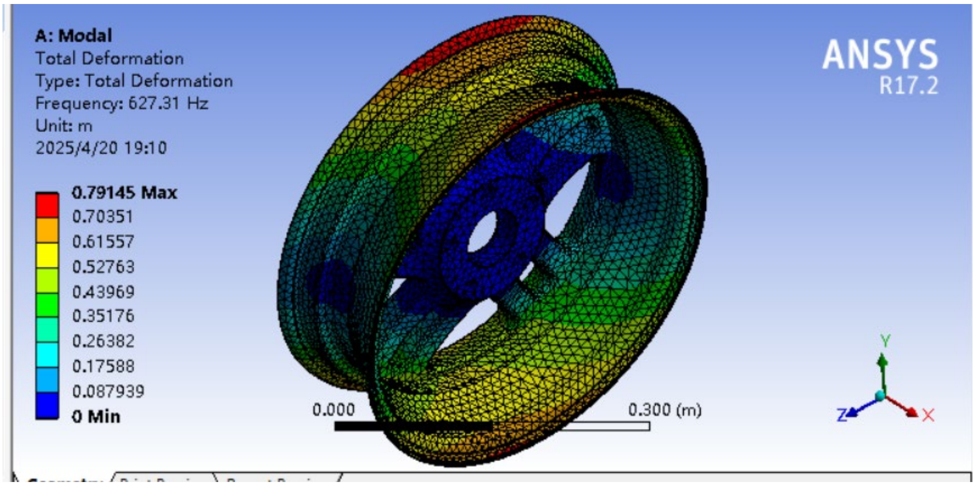
3.1 Meshing

The 3D wheel hub model obtained in UG is exported to ANSYS for finite element analysis. Due to the complex geometry of the hub, tetrahedral elements were employed for discretization. Local mesh refinement was applied in critical areas, such as fillets and spoke connections, to enhance the accuracy of the modal analysis. A final element size of 10 mm was adopted, balancing computational efficiency with sufficient accuracy

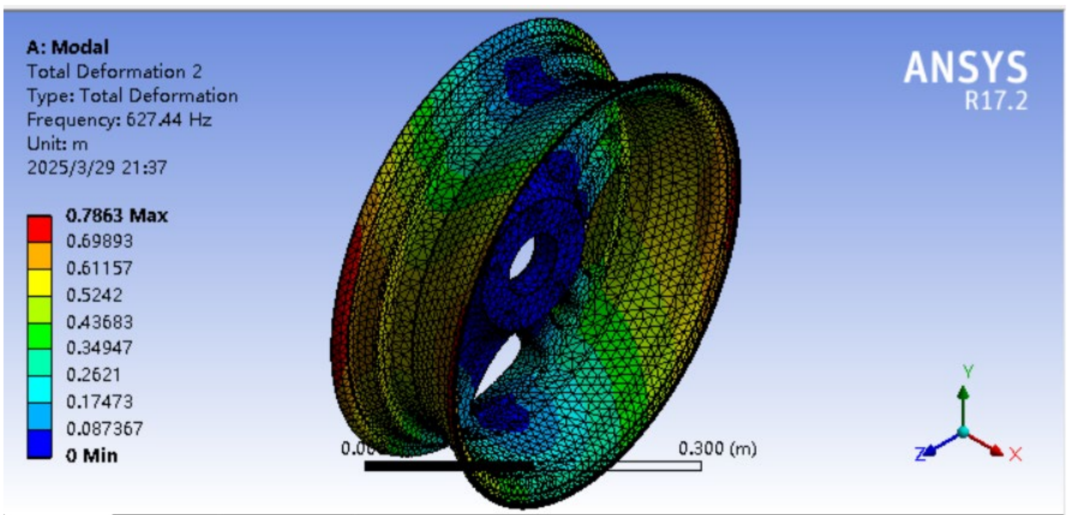
3.2 Results

The sixth-order mode in the ROTX direction has a relatively large modal mass participation ratio of 0.965882, indicating that this mode is the primary vibration mode of the wheel hub (Figure 2). The corresponding natural frequency is 1001.01 Hz.

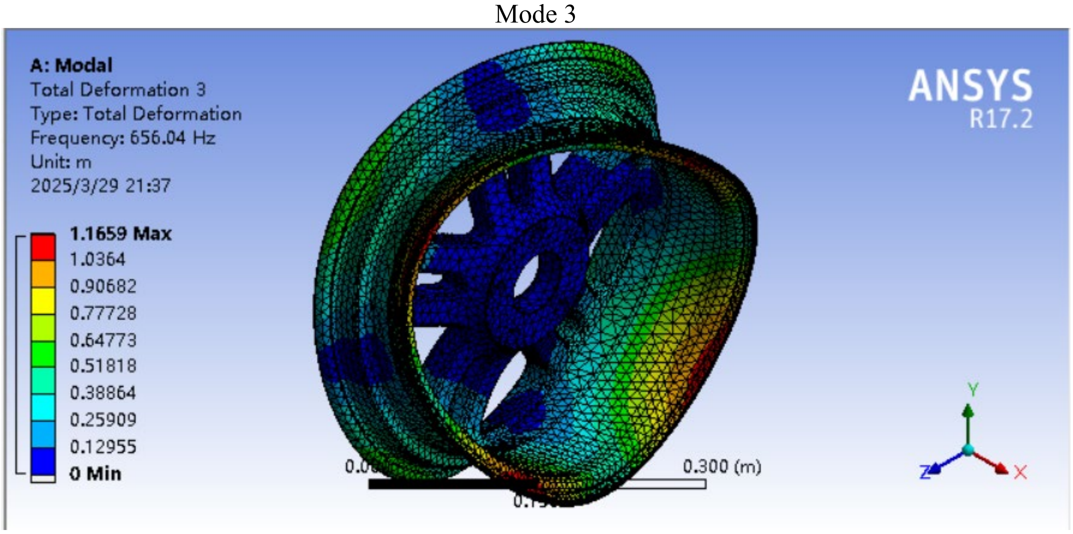
Mode 1



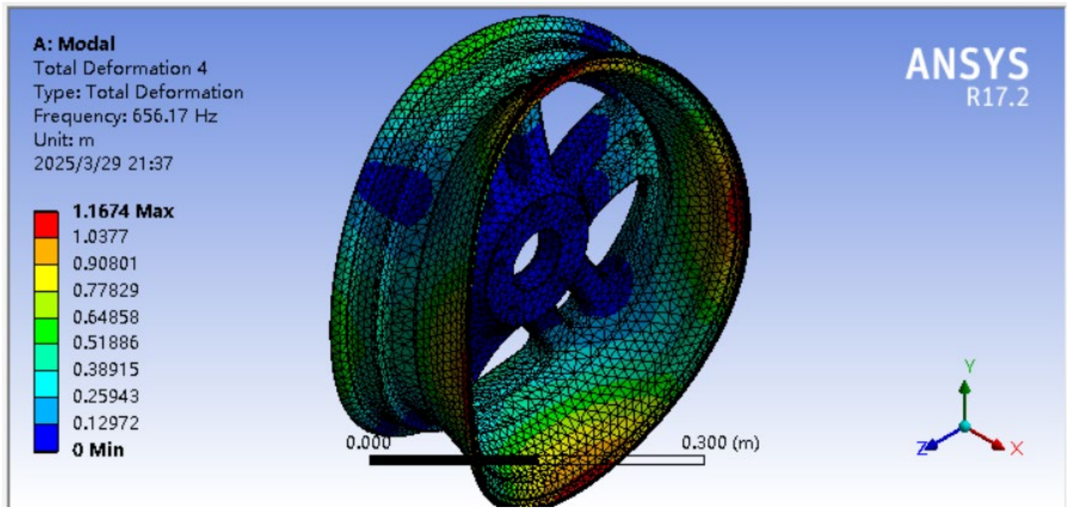
Mode 2



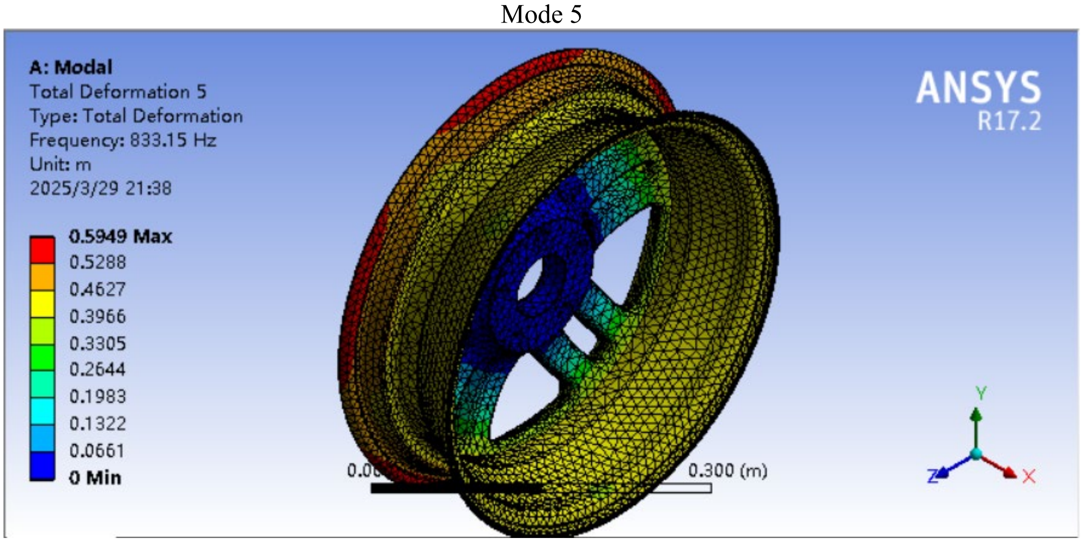
Mode 3



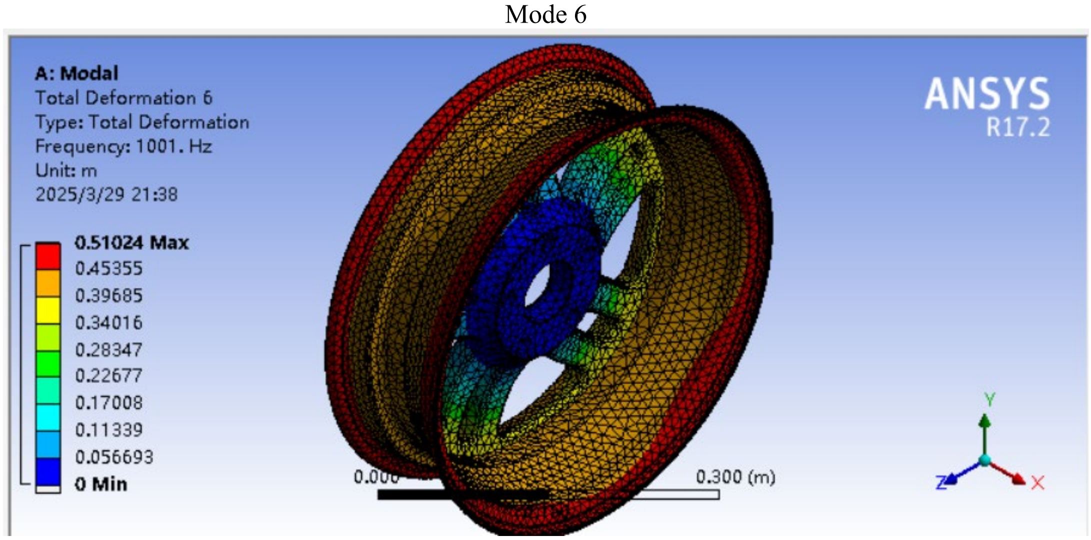
Mode 4



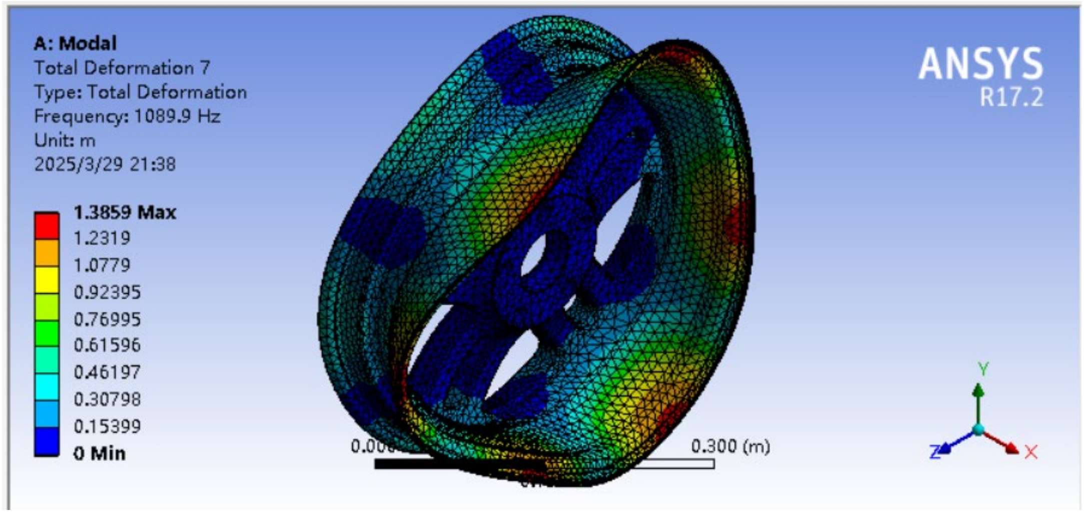
Mode 5



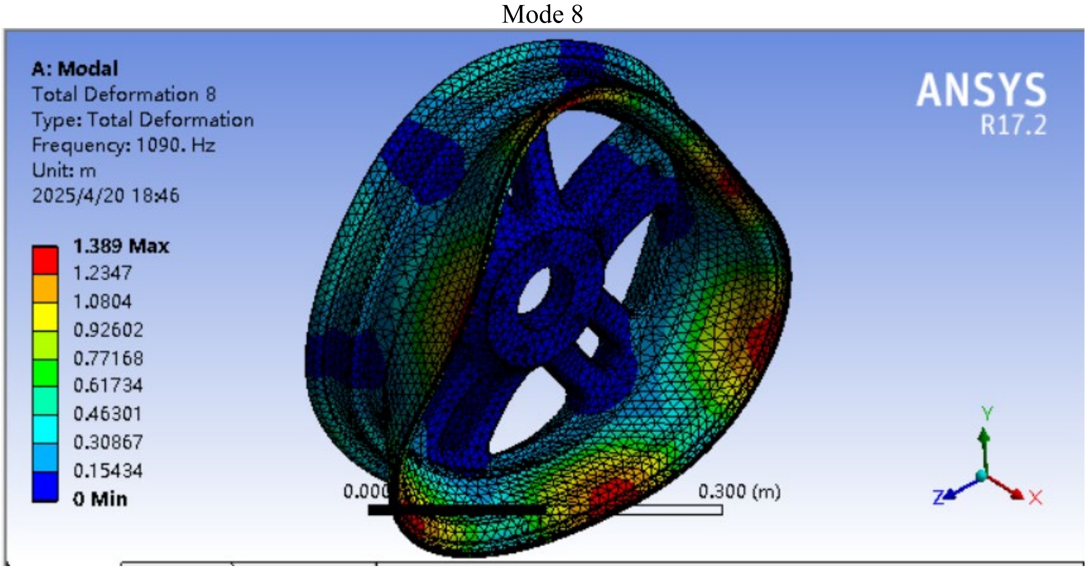
Mode 6



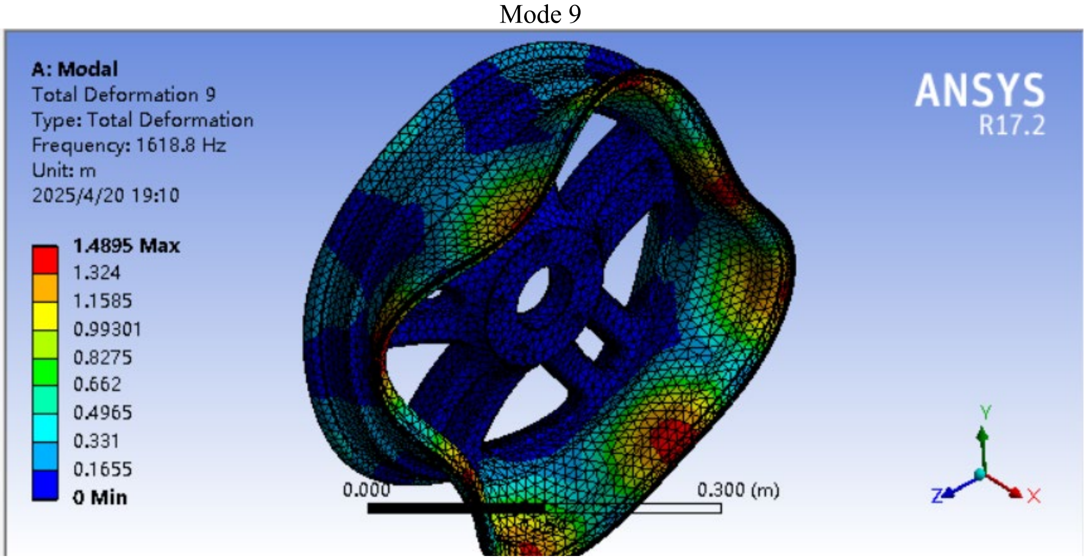
Mode 7



Mode 8



Mode 9



Mode 10

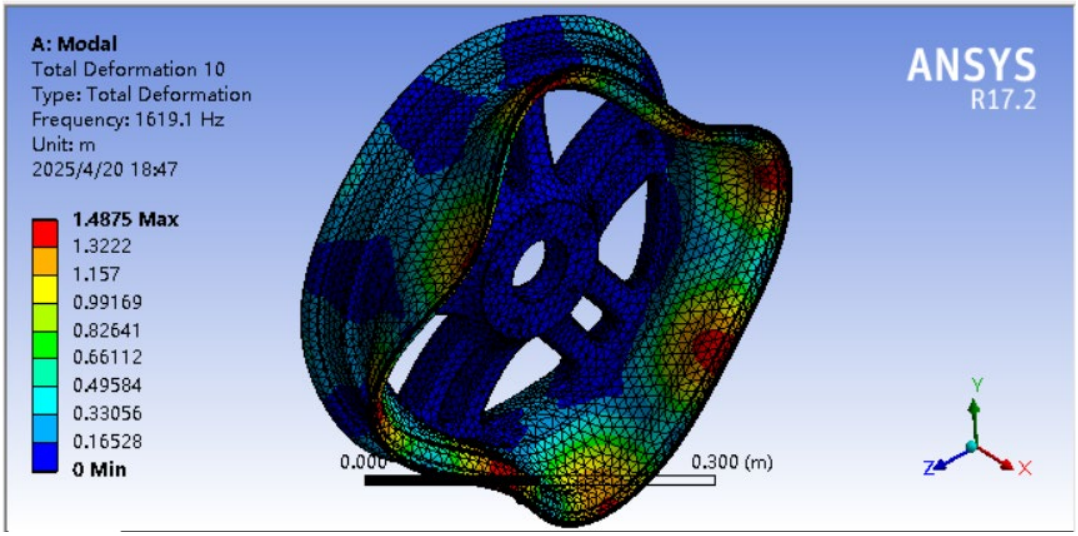


Figure 2. (Mode 1-10) Primary vibration mode of the wheel hub

3.3 The proportion of quality for each mode

Table 2 shows the proportion of quality for each mode.

Table 2**.** Proportion of quality for each mode

Mode frequency	To total mass
1	0.412689E-02
2	0.595537E-04
3	0.321157E-08
4	0.714690E-08
5	0.402500E-09
6	0.305604E-08
7	0.657994E-08
8	0.641553E-10
9	0.672166E-03
10	0.121242E-01
11	0.674013
12	0.482302E-05

13	0.949554E-08
14	0.128819E-07
15	0.104039E-06
16	0.819627E-08
17	0.102675E-02
18	0.285051E-03
19	0.186940E-08
20	0.231996E-08

3.4 Modal analysis
Through analysis, the calculated natural frequency of the wheel hub is approximately 1001.01Hz.
This result suggests that, in subsequent applications and design processes, the operating frequency of the wheel hub should be carefully avoided near 1001.01 Hz to prevent resonance phenomena, which could otherwise lead to excessive vibration, fatigue failure, or structural damage.

The determination of the natural frequency provides critical guidance for the future optimization of the wheel hub design, material selection, and system tuning, ensuring the overall reliability, safety, and performance of the final product.

References
Jing Juan, Wang Youhua. Finite Element Analysis of Modal and Stiffness Performance of Automobile Wheel Hub . Automobile Applied Technology, 112-115. 2023.

Li Fumeng. Isothermal Forming and Finite Element Analysis of AZ80 Magnesium Alloy Automobile Wheel Hub. Nanjing: Jinling Institute of Technology, 2023.

Liang Tian, Lei Kaiyin, Xia Zongyou, Li Shixing, Li Zhi. Modal Analysis of Different Lightweight Structures of Third-Generation Wheel Hub Bearings. Bearing, 2022.

Liu Yihua. Lightweight Design of Passenger Car Wheel Hub Based on Finite Element Analysis [J]. Special Purpose Vehicle, (9): 1-33. 2022

Sun Wenlong. Research on Lightweight Technology of Applying Lightweight Materials to Automotive Wheel Hubs. Beijing: Beijing Institute of Technology, 2016.

Zhang Bufeng, Zhao Yuxiang, & Niu Ruikun. Research on lightweight design of automobile wheel hubs, Modern Manufacturing Technology and Equipment, (10): 16-23. 2023.

Biography
Jiaying Zhang is a senior at Fairmont Preparatory Academy. She has a strong passion for physics and mathematics, and she enjoys exploring how these subjects connect to real-world applications. She is also an active member of my school's First Tech Challenge Robotics Team and Sustainable Engineers Club, where she collaborates on innovative projects such as the NASA TechRise competition.