[1],* Xinqiao Ding
[1] Xia Liu

Modal Analysis of Automobile Wheels based on Lightweight Technology

Abstract: - Compared with traditional wheels, magnesium alloy wheels have the advantages of higher strength, toughness, lighter weight, beautiful appearance and so on. This paper takes magnesium alloy wheel as the research object, and uses Pro/e and ANSYS to help complete a lot of work such as 3D model construction and finite element analysis and calculation, to ensure that the magnesium alloy wheel can meet its strength requirements while being lightweight. The strength analysis and modal analysis of the designed wheel model are carried out under various working conditions. According to the analysis results, the first design model is further simplified, and then the calculation and analysis are repeated many times. It is found that the new model meets the mechanical properties requirements under multi-working conditions, ensures that the requirements of lightweight design are met, and provides relevant theories and methods for wheel design and structural optimization of other materials.

Keywords: Wheel; Modal analysis; Magnesium alloy; Natural frequency; Light weight.

I. INTRODUCTION

As one of the important parts of the car, the car wheel plays a very important role in the operation of the car, it not only needs to support the weight of the vehicle and transfer the impact from the road, but also high-speed rotation,which may produce violent vibration. These vibrations may cause resonance in the vehicle, resulting in excessive noise and vibration of the vehicle. Wheel weight has an important impact on vehicle performance and fuel consumption, in the context of environmental protection and energy saving, lightweight has become one of the main development trends of the current automotive industry.

Liu Yihua selected the wheel hub of a certain car as the object of analysis, used SolidWorks software to build a model according to the structural parameters and appearance of the wheel hub, and used the finite element software to conduct a topological analysis of the car hub with the constructed model . Zhang Honggang, Wang Shasha, et al. used HyperWorks finite element analysis software to carry out strength analysis on the spokes before and after optimization. Under the premise of meeting the strength requirements, the optimized spokes have lighter weight, to achieve the goal of lightweight, and achieve the purpose of reducing the vehicle assembly mass and vehicle fuel consumption . Zhang Shengchao from Qingdao University took the automobile wheel as the research object, carried out the lightweight design of the wheel, designed the wheel with carbon fiber and epoxy resin as the composite material layer, based on the finite element analysis model, and used COMSOL to carry out structural topology optimization design of the constructed parametric model, and finally obtained a wheel model with a reasonable structure . Lv Wenhua, Pang MAO, and others from Zhejiang University of Science and Technology took the hub of a small electric vehicle as the research object, established the three-dimensional model of the hub, and used the finite element software ANSYS Workbench to carry out modal analysis and static mechanics analysis, put forward the improved design scheme of the hub structure, and analyzed the dynamic characteristics of the hub . Sui Jingyu et al. used the finite element method to calculate and analyze the lightweight design of the heavy truck hub, and the results obtained were consistent with the load-bearing analysis results of the original structure hub, and the strength of the optimized design hub was reliable and met the design requirements . Gong Liqiang of Jilin University used ANSYS workbench software to conduct finite element analysis of steel wheels to achieve the purpose of lightweight . Song Xiaoyan of Shenyang University of Urban Construction conducted finite element analysis and structural optimization design of automobile hubs to solve the problems related to the development of wheel series products such as structure, strength, life, and optimization design .

H Karim and P X Ku did a study on car wheels. They used the software SolidWorks to build the geometric model of the wheel, and used the finite element analysis software to conduct simulation analysis. In the case of the use of different lightweight materials, including but not limited to aluminum and magnesium alloys, the stress

[1] School of Automotive Engineering, Wuhan Vocational College of Software and Engineering (Wuhan Open University), Wuhan, 430205, Hubei, China
*Corresponding author: Xinqiao Ding

and deformation distribution of the two-spoke and multi-spoke wheel design is analyzed, and the basic design of the wheel is optimized to achieve the optimal level . Ryo KIYOTAKI and Kohei ONO measured the noise value generated around the wheel and the vibration acceleration of the wheel surface when a commercial vehicle was running at low speed and the braking force was slightly working, and conducted modal analysis and noise transfer path analysis in the computer to study the mechanism of noise generation . R.Vijayakumar and C.Ramesh did a study on car rims. The rims were modeled using SOLIDWORKS 2016. Then the rim model was imported into ANSYS WORKBENCH 16 for modal analysis [10]. Based on mechanics principles, P. F. Xu, S. Y. Duan and F. Wang proposed a method combining reverse modeling technology and topology optimization method to obtain wheel hub lightweight [11].

Magnesium alloy material was selected for the wheel designed in this paper, and this lightweight material can effectively reduce the weight of the wheel. Three-dimensional modeling and structure optimization were carried out by using the three-dimensional software PROE, and finite element analysis software ANSYS Workbench was used for analysis [12]. The force and modal analysis of the wheel under design conditions were obtained, and it was found that the original design might be too redundant. After modifying the design parameters and finite element simulation analysis, the final data and index are determined. The design scheme verified by finite element analysis software provides relevant theories and methods for wheel design and structure optimization of other materials. Future research can be carried out from in-depth exploration of the properties of lightweight materials, continuous improvement of manufacturing processes, and further exploration of the potential of wheel design.

II. Materials and Methods

A. *Three-dimensional model construction of the wheel*

The rim profile used in this design is a 5° deep slot rim J profile, the specification is 6J×15. The design rim has a nominal diameter of 380.2 mm, or about 14.96 inches. The setting of the design parameters of the wheel model is shown in Tables 1 and 2.

Table 1. Design parameters of the wheel mode

Product standard	Design load(F_v)	Static load radius (R)	Offset distance(d)	Strengthening factor (S)
6J×15	326.25kg	253.5mm	38mm	2

Table 2. Parameters of the wheel

Product standard	Bending moment(N·m)	Set speed(rpm)	Thread torque(N·m)	Required life(h)
6J×15	1378	1700	110	200000

Pro/e software was used to complete the design of the rim and spoke, and parts such as the outlet valve were partially modified to build a complete three-dimensional model of the wheel, as shown in Figure 1.

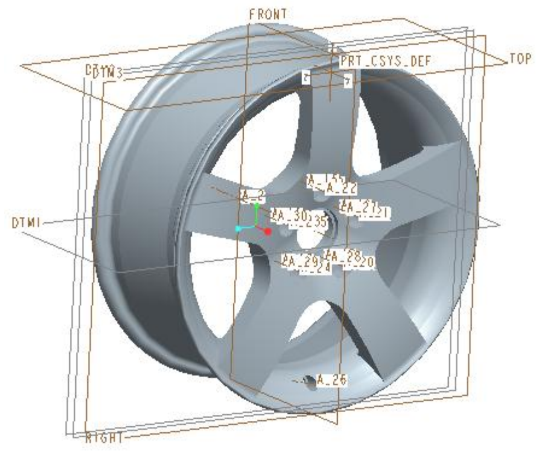


Fig.1 Constructed 3D model of the wheel

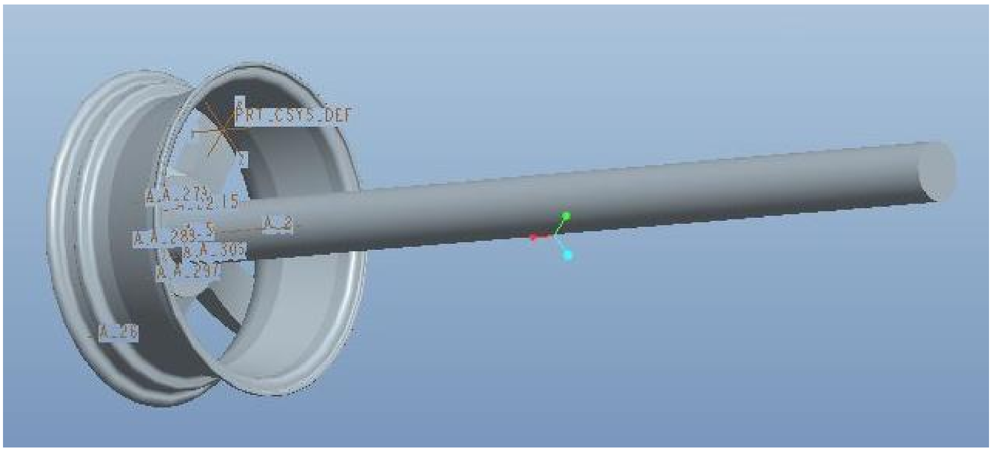


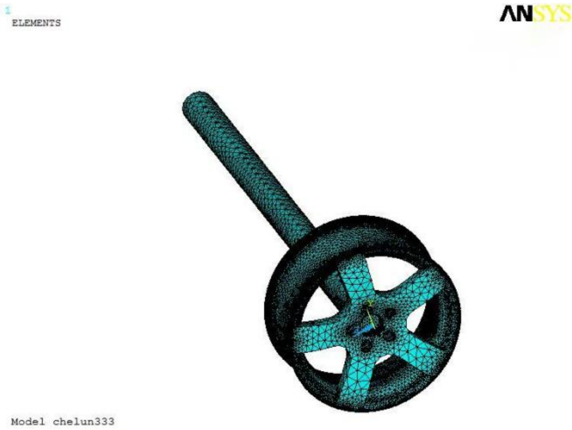
Fig.2 Preliminary set wheel entity analysis model

B. Static analysis of wheels

1) Model Import

A loading shaft was introduced into the magnesium alloy wheel model to analyze the model, as shown in Figure 2.

After the entity analysis model is added, its element types and material properties are defined, which can be entered or selected in the program dialog box as required. After defining the attributes of the model, the wheel model is meshed. There are two types of partition: free grid and mapped grid. The free grid can more easily control the accuracy of model grid division, so this paper chooses the free grid division type. Figure 3 shows the gridded model. Figure 4 shows the finite element model after constraints are applied.



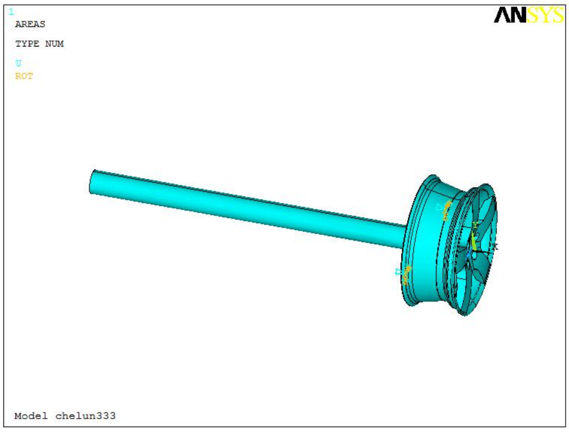


Fig.3 Gridded wheel model **Fig.4** Wheel model after constraints

2) Determine the material properties of the wheel

The material chosen is AM60B, often referred to as magnesium manganese alloy, also known as deformation alloy. Magnesium alloy AM60B has high strength and stiffness. In addition, the magnesium alloy also has good corrosion resistance and heat resistance, and can maintain good mechanical properties under harsh environments such as high temperature and high humidity.

The materials selected for the design (AM60B) are mainly the following three:

Elastic modulus E: 6.9E10

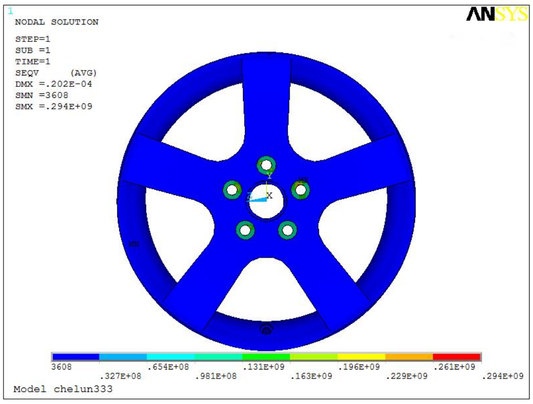
Density ρ: 1800Kg/m^3

Poisson's ratio: 0.35.

3) Results of static analysis

According to the conditions of use, there are generally three different loads acting on the wheel, respectively, bolt preload, rotating centrifugal force, and bending moment. The following is a calculation and analysis of these three different loads to explore the impact on magnesium alloy wheels.

(1) The role of bolt pre-tightening force



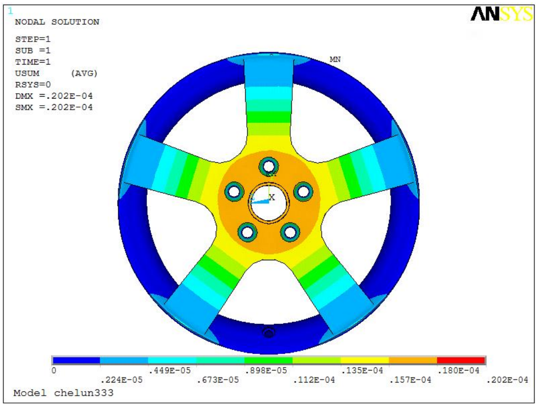
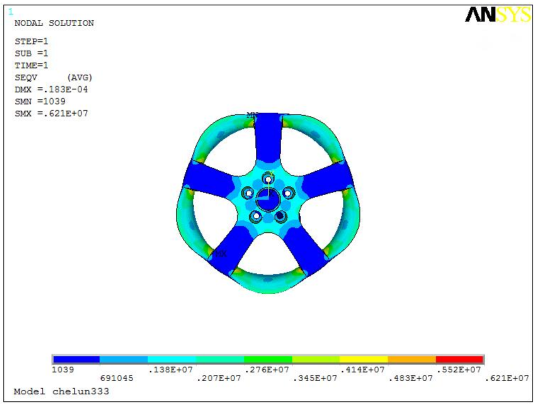


Fig.5 Force distribution diagram **Fig. 6** Displacement diagram

The pre-tightening force of the bolt was 120NM, and the analysis results showed that the maximum stress value was 294Mpa, which was located on the contact cylinder of the bolt hole between the fastening nut and the wheel, as shown in Figure 5. The maximum deformation of the wheel is 0.020mm, which is located in the area near the wheel hub in contact with the bolt, as shown in Figure 6. It can be seen that under the action of conventional bolt preload, the stress and deformation of the wheel meet the requirements.

(2) The role of rotating centrifugal force

The wheel rotates at a high speed, resulting in rotational centrifugal force in the structure. Assuming that the rotational speed of the wheel is uniform and constant, the rotational angular speed is input into ANSYS as the load boundary condition for calculation, and the final result is shown in Figure 7.



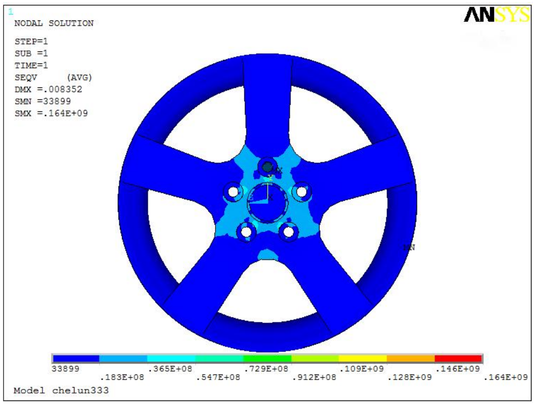


Fig.7 Centrifugal force distribution diagram **Fig.8** Stress cloud diagram of the wheel

The results show that under the action of centrifugal force, the force distribution of the wheel is relatively uniform, and the maximum stress is only 6.21Mpa, which is located at the joint of the spoke and the rim. In addition, the stress in the middle of the two spokes is also relatively prominent, reaching between 1.38 and 2.07Mpa, and the stress on the spoke is the smallest. The analysis shows that the centrifugal force has a certain influence on the overall structure of the wheel.

(3) Bending moment effect

The influence of the bending moment on the overall structural strength of the analysis object is very important. The bending moment is input into the preset finite element model as the boundary condition to analyze and obtain the final data. As shown in Figure 8, the maximum stress value of the model reaches 164Mpa around the bolt hole, and the stress value in the middle of the two spokes is within the range of 36.5~54.7Mpa. Both are lower than the yield strength of the selected material.

(4) The combined force of the bolt pre-tightening force, rotating centrifugal force, and bending moment

The above three force values of bolt preload, rotary centrifugal force， and bending moment were input at the same time and analyzed as boundary conditions, and the wheel stress distribution was obtained as shown in FIG. 9.

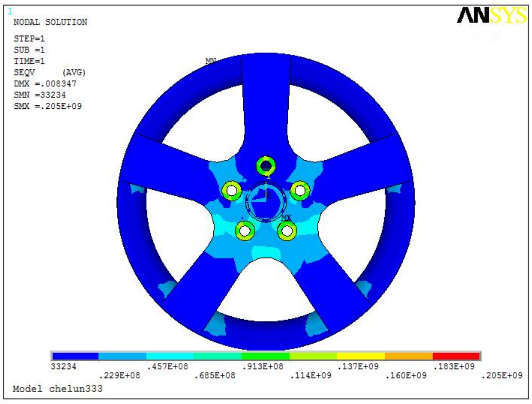


Fig.9 Distribution of wheel stress under load

The maximum stress value of the model is 205Mpa, which is located at the nut seat, and is mainly caused by the pre-tightening force of the bolt. The stress of the rim will be affected by both the bending moment and the speed, and the stress value and the change value in the analysis are very small, and it is safe. The position between two adjacent spokes and the joint of the spokes on the mounting plate are stressed greatly, which are affected by the bending moment and speed. The force on the spoke is not affected by the bending moment and speed, which is relatively stable.

C. Modal analysis of wheels

1) Free vibration of the model without velocity influence

The following results were obtained through calculation (see Figure 10-16) and statistical analysis was made (see Table 3). The first 6 frequency values are almost 0, and the values and changes are small, and the modal analysis of rigid body can be avoided. The results show that the frequencies of order 7, 8, 9, 10 and 12, 13 are close to each other and the vibration direction is orthogonal in the free vibration without velocity influence.

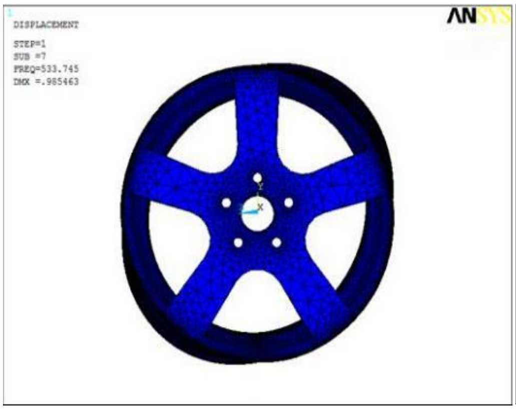


Fig.10 The 7th-order modal analysis diagram

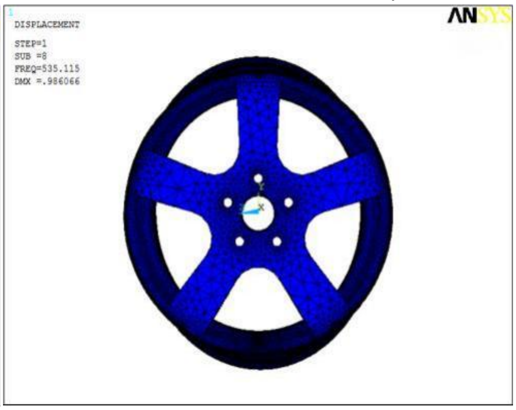


Fig.11 The 8th-order modal analysis diagram

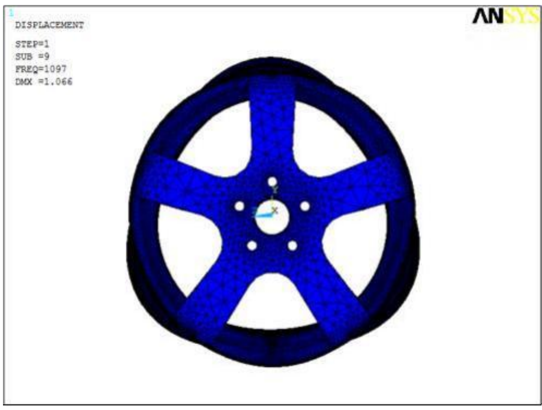


Fig.12 The 9th-order modal analysis diagram

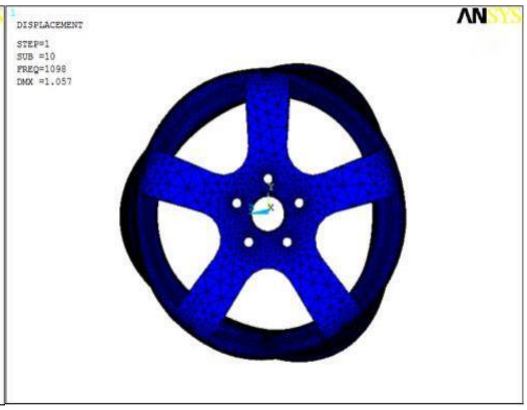
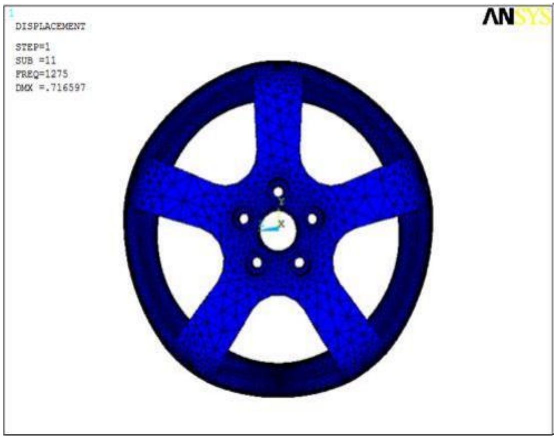


Fig.13 The 10th-order modal analysis diagram



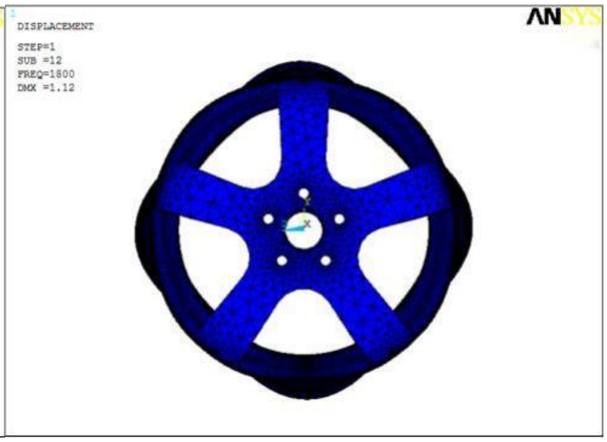


Fig.14 The 11th-order modal analysis diagram **Fig.15** The 12th-order modal analysis diagram

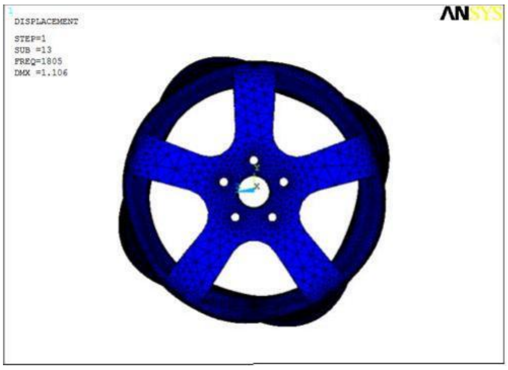


Fig.16 The 13th-order modal analysis diagram

Table 3. Frequency of wheels

Order	Frequency value (Hz)
1	0.0000
2	0.0000
3	0.19123
4	0.32874
5	0.44426
6	0.52073
7	533.7
8	535 1
9	1096.9
10	1098.5
11	1275 5
12	1799 6
13	1805.4

2) Analysis of constrained vibration results without velocity influence of the model

The calculation results of its constrained vibration (see Table 4). The respective frequency values and vibration modes between the neighbors are very close, and the vibration direction is orthogonal. The vibration modes and displacement distributions of each order are obtained, as shown in FIG.17-23.

Table 4 Frequency of wheels

Order	Frequency value (Hz)
1	629.8
2	632.8
3	665.5
4	666.7
5	839.6
6	1060.7
7	1111.9

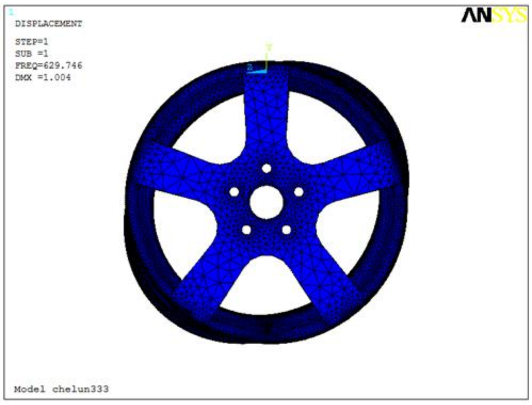


Fig.17 The 1st-order modal analysis diagram

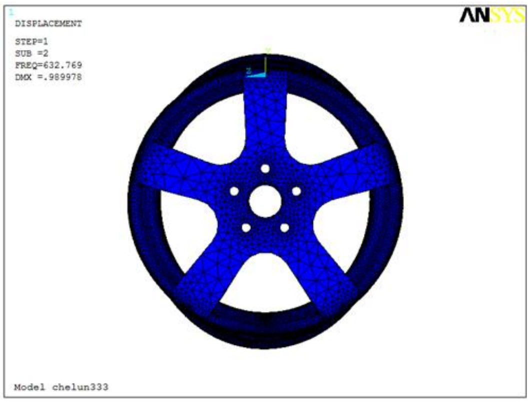


Fig.18 The 2nd-order modal analysis diagram

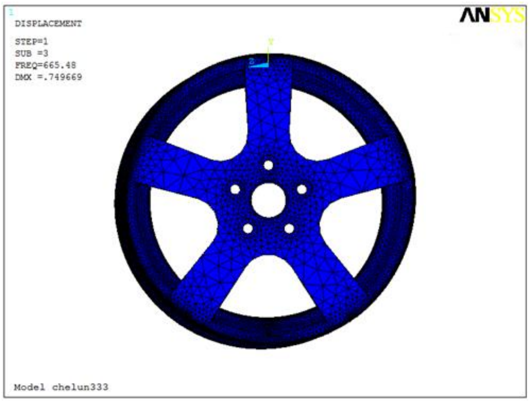


Fig.19 The 3rd-order modal analysis diagram

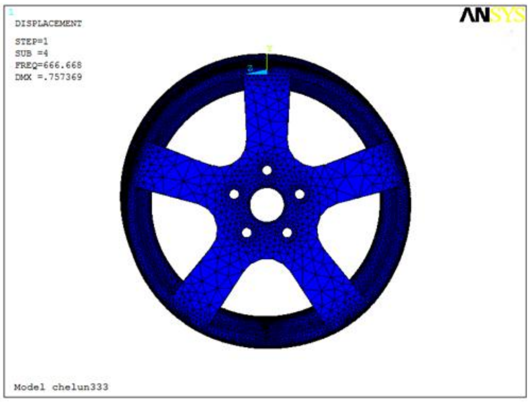


Fig.20 The 4th-order modal analysis diagram

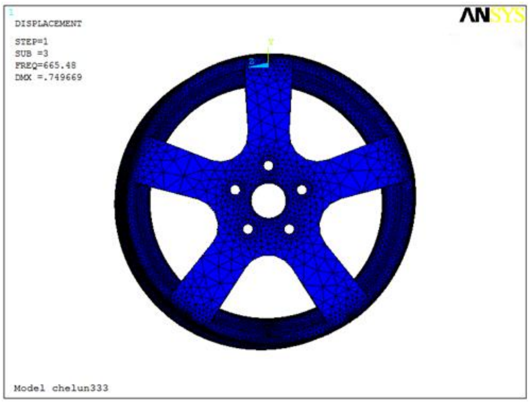


Fig.21 The 5th-order modal analysis diagram

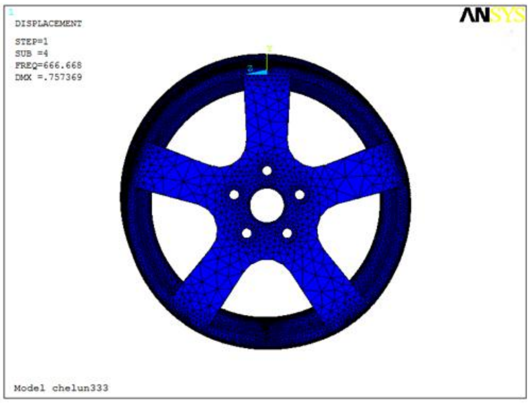


Fig.22 The 6th-order modal analysis diagram

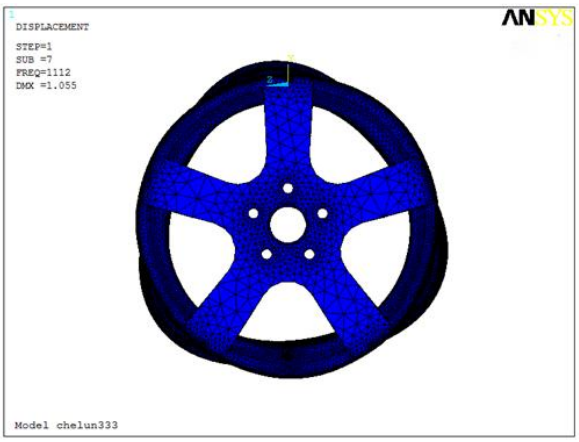


Fig.23 The 7th-order modal analysis diagram

In the analysis, the first 13 orders of free vibration and the first 7 orders of constrained vibration are selected in the preset range. When the model does not influence speed, the wheel will undergo constrained vibration, and the natural frequency of the first stage is 629.8Hz, while the speed range of the L-shaped four-cylinder engine of the vehicle selected for design is 0-5600rpm, and the vibration frequency of these engines is generally within the range of 0-200Hz, and there is no resonance between the two and a certain difference. In this case, the amount of material can be reduced by adjusting the wheel model parameters, and the purpose of lightweight can be achieved.

III. Discussion

A. *Wheel optimization adjustment*

1) Structural adjustment

Through the previous research, it is not difficult to find that the force value of the analysis object is far less than the predetermined range of the design, which indicates that there is still redundancy in the use of materials. To achieve the light weight of the vehicle, save materials and control costs, the simplification of the model is necessary. Figure 24 shows the new model after the adjustment of the model.

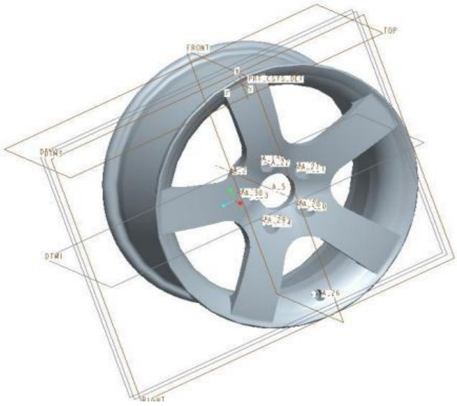


Fig.24 Improved wheel model

The volume of the original design model is 0.003431502m³ and the weight is 6.01Kg. The volume of the new model after adjustment is 0.00287261m³ and the weight is 4.99Kg. Compared with the previous model, the weight of the new model is 17% less than that of the previous model.

2) Comparison of intensity before and after

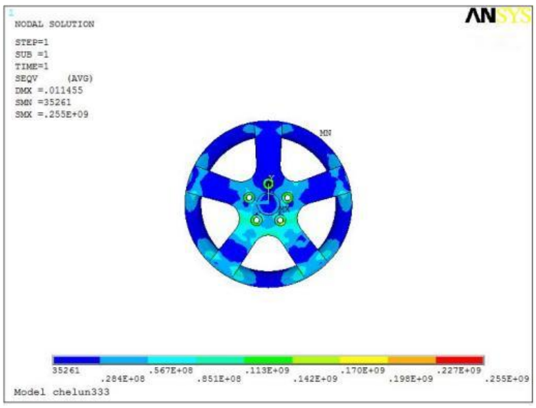


Fig.25 Stress distribution of the model after adjustment

In the static analysis of the model, the maximum force value is 255Mpa, located in the nut seat, which is the same as before and also caused by the pre-tightening force of the bolt, as shown in Figure 25. At the same time, under the influence of both bending moment and speed, the stress of the rim is only between 0.36MPa and 57.2MPa, and its stress value and variation value are very small, so it is a safe area. The stress value of the blank position between the two spokes and the joint of the spoke on the mounting plate is between 56.7MPa and 142MPa. The force value on the spoke is not affected by the bending moment and speed, and is relatively stable.

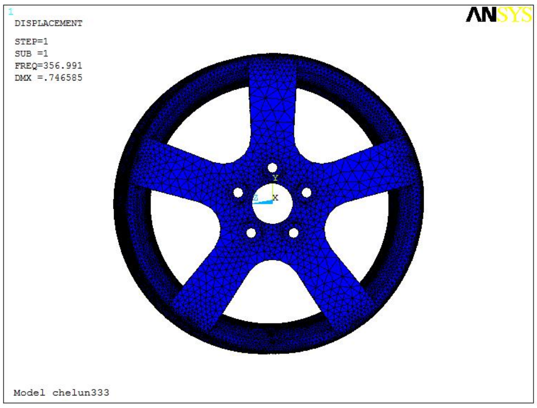
According to the analysis, the strength of the new model can still meet the requirements and standards as the original model under the condition of thinning thickness, decreasing volume, and reducing weight.

B. Conduct modal analysis of the new model after adjustment

The results of the calculation of vibration in its confinement (see Table 5). It is not difficult to find that when the velocity influence is 0, the calculated results are still close to the vibration patterns, and the vibration directions are still different, and the directions are still orthogonal. Finally, the vibration patterns and displacement distributions of each order are obtained, as shown in FIG. 26 to FIG. 33.

Table 5. Frequency of wheels

Order	Frequency value (Hz)
1	357.0
2	359.9
3	461.2
4	462.9
5	491.5
6	789.9
7	1045.9



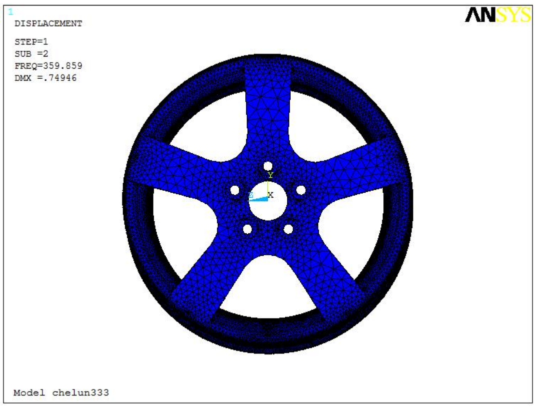
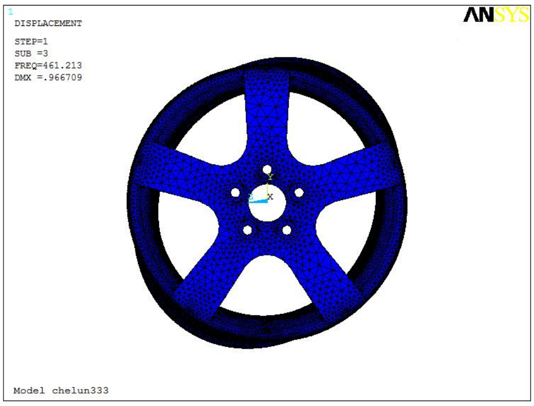


Fig.26 The 1st-order modal analysis diagram **Fig.27** The 2nd-order modal analysis diagram



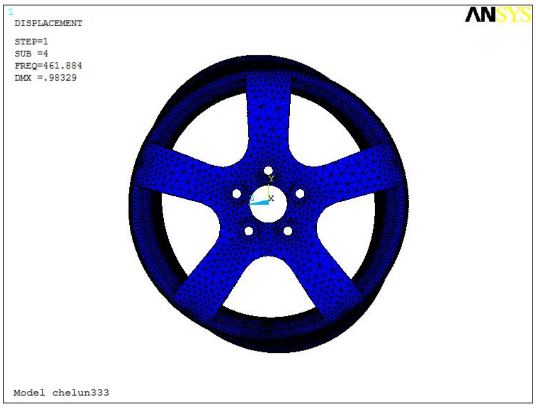
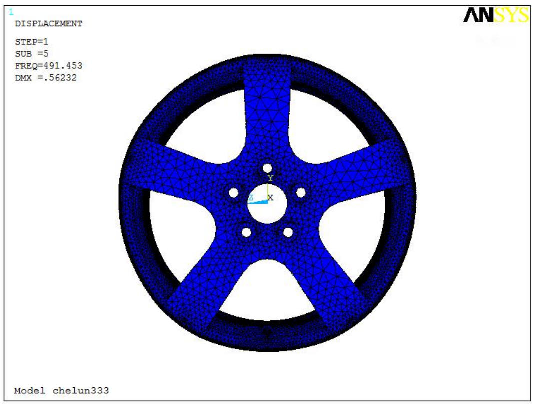


Fig.28 The 3rd-order modal analysis diagram **Fig.29** The 4th-order modal analysis diagram



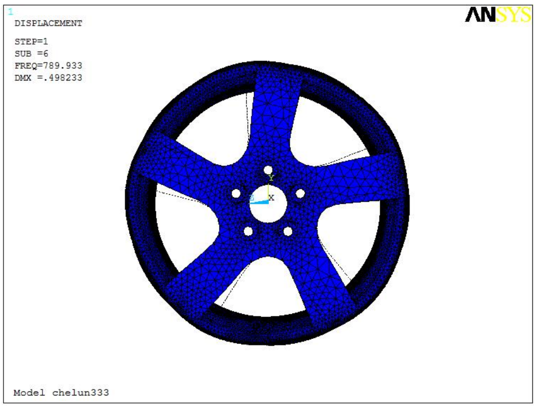
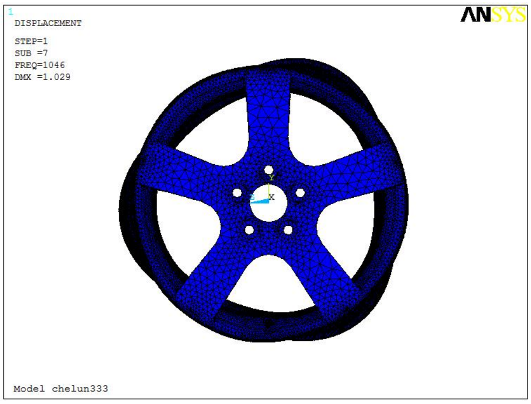


Fig.30 The 5th-order modal analysis diagram **Fig.31**The 6th-order modal analysis diagram



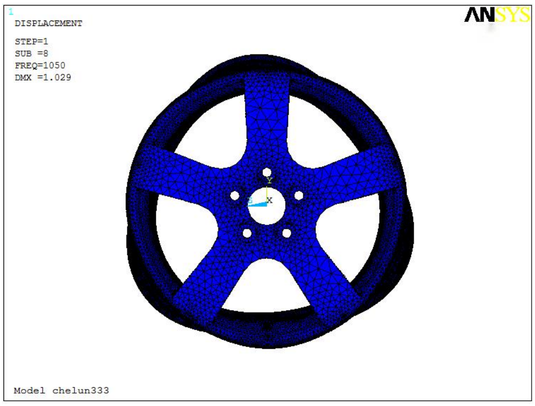
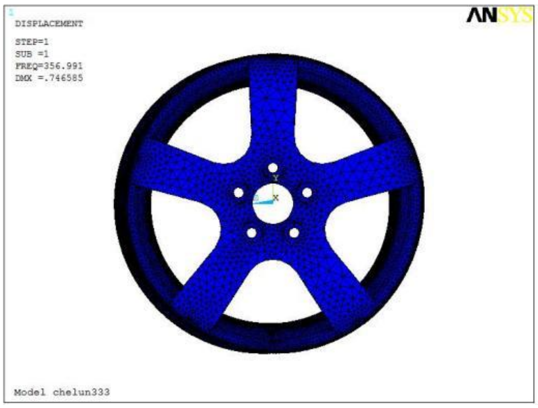


Fig.32 The 7th-order modal analysis diagram **Fig.33** The 8th-order modal analysis diagram

The results of the calculation of vibration in its confinement (see Table 6). On the basis that the velocity influence is 0, the calculated results are still pair-to-pair close to the vibration modes, and the vibration directions are still different, and the directions are still orthogonal. Finally, the vibration modes and displacement distributions of each order are obtained, as shown in FIG. 34 to FIG. 41.

Table 6. Frequency of wheels

Order	Frequency value (Hz)
1	357.0
2	359.9
3	461.2
4	462.9
5	491.5
6	789.9
7	1045.9



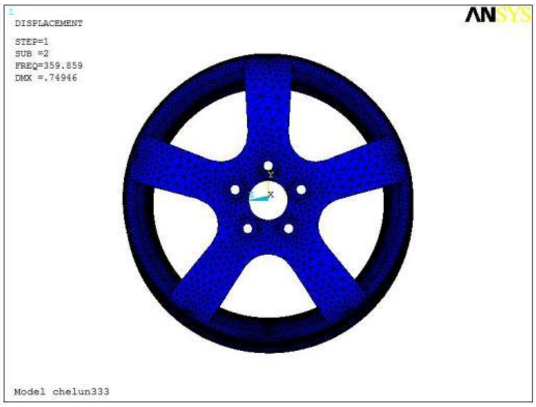


Fig.34 The 1st-order modal analysis diagram **Fig.35** The 2nd-order modal analysis diagram

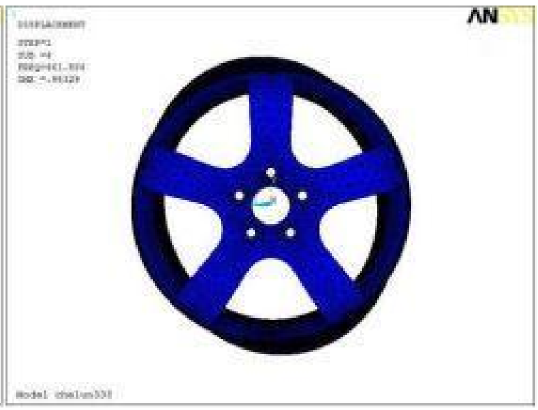


Fig.36 The 3rd-order modal analysis diagram **Fig.37** The 4th-order modal analysis diagram

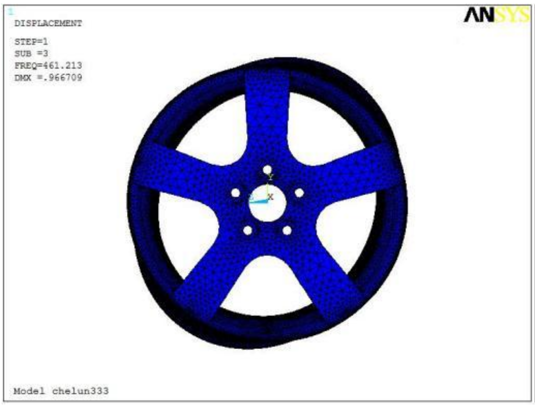


Fig.38 The 5th-order modal analysis diagram

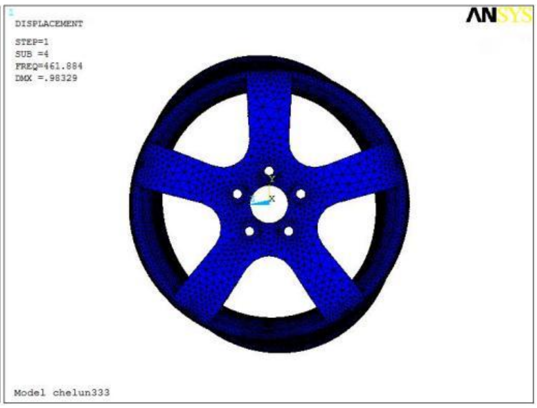


Fig.39 The 6th order modal analysis diagram

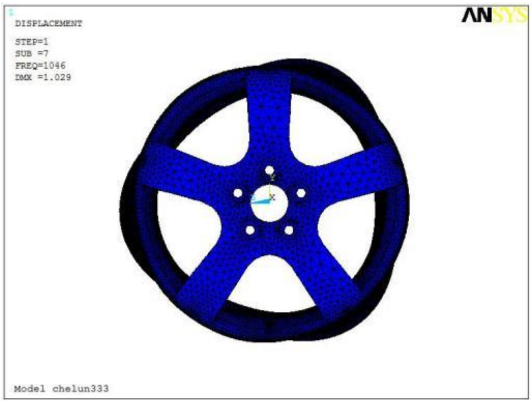


Fig.40 The 7th-order modal analysis diagram

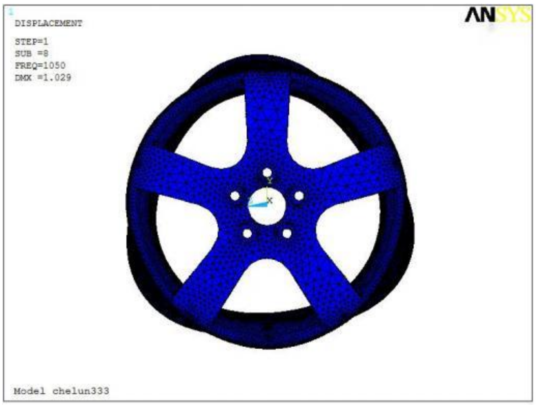


Fig.41 The 8th-order modal analysis diagram

The modal analysis of the new model is carried out. The first 13 orders of free vibration and the first 7 orders of constrained vibration are selected in the preset range. When the speed is 0, the wheel is constrained by vibration, and the natural frequency of the first stage is 357Hz, while the speed range of the L-shaped four-cylinder engine of the design vehicle is 0-5600rpm, and the vibration frequency of these engines is generally in the range of 0-200Hz, and there is no resonance between the two. It shows that the adjustment of the model parameters is successful, and the weight of the wheel can be reduced without reducing the mechanical properties of the wheel.

IV. Conclusions

In this paper, magnesium alloy wheels are mainly studied, and Pro/e and ANSYS are used to help complete a lot of work such as 3D model construction and finite element analysis and calculation, to ensure the mechanical properties of magnesium alloy wheels while they are lightweight. The strength analysis and modal analysis of various working conditions are emphasized in the designed model. The free and constrained vibration of the wheel is analyzed and the natural frequencies of each order of the wheel are obtained without the influence of the speed. Compared with the natural frequency of the wheel and the output frequency of the engine, the natural frequency of the first mode of the wheel is 629.8Hz (357.0Hz after improvement), while the general four-cylinder engine speed range is 0-5600mpa, and the vibration range is 0-200Hz, so it will not resonate with the engine.

Through several finite element analyses and calculations, the weight is reduced to 83% of the original setting without reducing each index, and the lightweight is realized, which provides a certain reference value for improving vehicle performance and reducing fuel consumption. Future research can be carried out by further optimizing material properties, improving manufacturing efficiency and quality, and tapping design potential.

Author Contributions:

Conceptualization, X.D. and X.L.; methodology, X.D.; software, X.D.; validation, X.D., and X.L.; formal analysis, X.D.; investigation, X.D.; resources, X.D.; data curation, X.L.; writing—original draft preparation, X.L.;

writing—review and editing, X.D.; visualization, X.D.; supervision, X.L.; project administration, X.L.; funding acquisition, X.D. All authors have read and agreed to the published version of the manuscript.

FUNDING:

This research received no external funding

INSTITUTIONAL REVIEW BOARD STATEMENT:

Not applicable

INFORMED CONSENT STATEMENT:

Not applicable.

DATA AVAILABILITY STATEMENT:

Hindawi Research Data Policies at https://www.hindawi.com/publish-research/authors/research-data/.

ACKNOWLEDGMENTS:

Thanks to all the staff involved in the research and the leaders of the School of Automotive Engineering, Wuhan Vocational College of Software and Engineering for their support.

CONFLICTS OF INTEREST:

The authors declare no conflicts of interest.

REFERENCES

[1] Liu Yihua. Based on the finite element analysis of car wheel hub lightweighting design [J]. special purpose vehicles, 2022 (9) : 30 to 33. DOI: 10.19999 / j.carol carroll nki. 1004-0226.2022.09.010.

[2] Zhang Honggang, Wang Shasha. Application of CAE Technology in Lightweight Design of Dump Truck Wheels [J]. Automotive Practical Technology, 2020(15):75-77. DOI: 10.16638/j.cnki.1671-7988.2020.15.025.

[3] Zhang Shengchao. Lightweight Design of Automobile Wheel [D]. Qingdao University, 2021. DOI: 10.27262/d.cnki.gqdau.2020.002060.

[4] Lü WenhuaPang Mao, Joshua Cobby. Dynamic characteristic analysis of small electric vehicle hub[J]. Modern Manufacturing Engineering, 2020(06):57-61. DOI: 10. 16731/ j. cnki. 1671- 3133.2020. 06. 010.

[5] Sui Jingyu, Liu Junwei, Wang Guoyuan, Shang Guorui. Lightweight Design and Strength Analysis of Heavy Commercial Vehicle Axle Hub[J]. China Auto, 2019(09):43-45.

[6] Gong Liqiang. Research on the Lightweight Steel Wheels of a Special Vehicle[D]. Jilin University, 2020. DOI: 10. 27162/ d. cnki. gjlin.2019. 000444.

[7] SONG Xiao-yan. Finite Element Analysis and Structural Optimization Design of Automobile Wheel Hub[J]. Value Engineering, 2018, 37(16):123-124. DOI: 10.14018/j.cnki.cn13-1085/n.2018.16.048.

[8] Karim, H.&Ku, P.X.(2022). Effect of design and material on structural rigidity of automobile wheel - an optimisation approach. Journal of Physics: Conference Series, 2222. doi:10.1088/1742-6596/2222/1/012007

[9] Ryo KIYOTAKI1 Kohei ONO1 , Zhe LI , Takumi NAITO and Osamu TERASHIMA. Study on the Generation of the Tonal Noise Emitted from an Automobile Wheel. Advanced Experimental Mechanics, Vol.7 (2022), 130-136

[10] Vijayakumar, R., Ramesh, C., Boobesh, R., Surya, R.R., & Rajesh, P.S. (2020). Investigation on automobile wheel rim aluminium 6061 and 6066 Alloys using ANSYS WORKBENCH. Materials Today: Proceedings.

[11] Xu, P.F., Duan, S.Y., & Wang, F. (2020). Reverse Modeling and Topological Optimization for Lightweight Design of Automobile Wheel Hubs with Hollow Ribs. International Journal of Computational Methods.

[12] Kim, J., Kwon, S., Ryu, S., Lee, S., Jeong, J., & Chung, J. (2022). Noise Identification for an Automotive Wheel Bearing. Applied Sciences.