Optimization Analysis and NC Simulation Processing of the Wheel-Mold

WANG Guo-qiang1 2, Sun Hongyan3* and Zhai Xujun2

1Key Laboratory of Modern Agricultural Equipment and Technology, Ministry of Education, Jiangsu University, Zhenjiang, Jiangsu, 212013, China
2Department of Agricultural Engineering, Jiangsu Agriculture & Animal Husbandry Science and Technology Vocational College, Taizhou, Jiangsu, 225300, China
3Taizhou College of Nanjing Normal University, Taizhou, Jiangsu, 225300, China

*Corresponding author’s e-mail: 601848884@qq.com

Abstract: The structure of the wheel hub was analyzed by the topology optimization module in the Workbench. According to the analysis results, the structure of the wheel hub was redesigned. Comparing the optimization of the deformation displacement, stress and quality of the wheel to before, it can be found, the deformation of the wheel displacement is ignored, but the quality is reduced from 9.5693kg to 7.3619kg, which achieves the purpose of car lightweight. Finally, the CNC programming module of CATIA software is used to simulate the machining of the automobile wheels, and the G code is executed by the CNC machine tool. The processability and processing efficiency of the products are improved accordingly, which shortens the products from design to manufacturing, and greatly enhances the competitiveness of enterprises.

1. Introduction

In recent years, with the further aggravation of energy shortage issues, energy conservation and emission reduction have become the core issues in the development of the automotive industry. In order to achieve the effect of vehicle energy conservation and emission reduction, vehicle lightweighting has become a major development trend in the automotive industry [1]. The lightweight of a car is defined as the use of the best and effective method to reduce the overall weight of the car and improve the dynamic performance of the car while ensuring the safety and strength of the car. There are two options for achieving lightweight vehicle: the one changes the structural size to meet the requirements of safety performance requirements; the other uses alternative materials under the premise of meeting the strength performance requirements. Nowadays, light alloys such as aluminum alloys and magnesium alloys have been widely used in the automotive industry to achieve the effect of reducing their own weight [2-3]. The wheel hub is an important safety component of the car, the choice of material is not only the development direction of the car lightweight, but also affects the safety and steering stability of the car, the aluminum alloy material is the best choice. At the same time, in order to achieve high strength, light weight, beautiful appearance and other requirements, the optimization design of structure and layout is also needed to obtain the optimal design scheme in shape and size as well as materials [4].

At present, the lightweight of automobile wheels mainly focuses on material replacement [5-6],
structural design and optimization \[^{7-9}\], fatigue resistance and service life \[^{10-11}\], etc., and its design factors include rim thickness, spoke surface radius and center angle, thickness of hub mounting boss, wheel roll thickness, wheel profile size, wheel process parameters versus mass, rim structure and hub structural unit displacement variation, etc. However, a few studies are concerned with the analysis of the influence of the size and position of the mounting surface of the hub on strength properties and vehicle weight reduction.

Since the wheel is mainly composed of rims and spokes, the rim has been standardized, so the width and diameter of the rim have been determined when the tire model is determined. However, the shape, size and appearance of the spokes are different according to different design concepts \[^{12}\]. Therefore, in the process of the design, the size and position of the mounting surface of the aluminum alloy wheel hub are designed by CATIA software according to the tire and rim requirements. Then the ANSYS software is used to check the strength of the wheel hub, and the topology optimization function of ANSYS software is applied to the spoke. Optimized design ensure that it meets the requirements of aesthetics, economy and light quality while meeting the strength requirements. Finally, the CATIA software is used to program and virtualize the product, and generates G code that is directly executed by the machine. Through the inspection of virtual machining, the machinability and processing efficiency of the product are improved accordingly, and the product is shortened from R&D to manufacturing, the cycle has greatly enhanced the competitiveness of enterprises.

2. Basic structure of automobile wheels

The structure of the car hub is shown in Figure 1. The hub, also called the rim, is a metal component that supports the inside of the tire and is connected to the central shaft. Generally, the hub is mainly composed of two parts: a rim and a spoke. In addition, there are components such as a rim, a deviation, a valve hole, and a groove bottom.

The rim refers to the outer side of the hub, mainly to support the effect of the tire; the spokes mainly refer to a part of the center of the hub to the edge of the rim. The offset is the vertical distance between the spoke and the rim. The offset is also different depending on the hinge structure; the rim is the rim portion of the tire that supports the tire. The valve hole is a hole that protrudes from the tire valve. The bottom of the groove is the groove on the inside of the rim, which is mainly used to remove the tire.

At present, the specifications of tires and rims have been standardized. Combined with the actual research objects of this project, the used tire model 7.5R15 is used, and the rim width and diameter is determined according to GB/T3487-1996 rim national standard, as shown in Table 1 \[^{13}\].

| Rim diameter (inches) | Nominal diameter (mm) | Rim width (inches) | Nominal width (mm) |
|-----------------------|-----------------------|-------------------|-------------------|
| 10                    | 253.2                 | 4.5               | 114.3             |
| 12                    | 304                   | 5                 | 127               |
| 13                    | 328.4                 | 5.5               | 139.7             |
| 14                    | 354.8                 | 6                 | 152.4             |
| 15                    | 380.2                 | 6.5               | 165.1             |
| 16                    | 405.6                 | 7                 | 178               |
| 17                    | 436.6                 | 7.5               | 190.5             |

After obtaining the basic dimensions of the aluminum alloy rim, the CATIA software is used to model the aluminum alloy rim to obtain the initial three-dimensional model of the aluminum alloy rim.
3. Strength Analysis and Topology Optimization Design of Automobile Hub Based on ANSYS Workbench

After obtaining the initial design of the spokes, it is generally necessary to optimize the design, that is, to optimize the quality and volume of the spokes under the premise of ensuring the design strength, to meet the development trend of lightweight vehicles, and also to reduce energy consumption.

3.1 Introduction of ANSYS Workbench

In 1970, John Swanson officially founded ANSYS, and is still the world's largest CAE company. In the finite element analysis process of ANSYS 13.0, it is generally divided into three modules: pre-processing, solution and post-processing modules. The pre-processing module is mainly the process of product modeling and meshing; the solution module is the process of applying load and boundary conditions to the pre-processed model; the post-processing module calculates and analyzes the structure of the solution module, and graphs the result. And the way of the curve is expressed [14].

3.2 Wheel finite element model establishment

Select the analysis type "Static Structural" static analysis, and import the wheel model into the Workbench in neutral file (*.igs) format. The part material is aluminum alloy. The mechanical properties of related materials are shown in Table 2.

| parameter           | unit   | data  |
|---------------------|--------|-------|
| Density             | Kg/m3  | 2690  |
| Young's modulus     | Pa     | 6.9e10|
| Poisson's ratio     | μ      | 0.31  |
| Yield Strength      | Mpa    | 245   |
| tensile strength    | Mpa    | 390   |

3.3 Meshing

The meshing method selects the "automatic division" mode, and the mesh size is "Media" medium precision; the divided mesh is shown in Fig 2.
Fig 2. Schematic diagram of hub grid division

It can be seen that the rim portion of the hub has a larger mesh division, because the structure of the rim portion is relatively regular and continuous, and the analysis is more accurate. The slightly larger mesh division does not have much influence on the finite element result and can save analysis time, and the meshing of the spokes and the central threaded hole is relatively small, because the spokes are the main force-receiving position, and the cross-section of the spokes and the spokes and the spokes of the spokes and the center of the spokes vary greatly, and the stress distribution at these locations more complex, which need to refine the grid.

3.4 Add boundary conditions
The force of the automobile hub model is more complicated. The force of the wheel hub is simplified according to the actual spoke force. The specific loads and constraints are as follows:

Select the center threaded hole of the hub as the fixed restraint position. When loading, consider the actual main force of the car, the weight of the car, the driving force of the car, and the pressure of the car tire. The loading diagram is shown in Figure 3.

Fig 3. Schematic diagram of hub constraint

3.5 Topology optimization analysis
Using topology optimization, the topology of the aluminum alloy wheel material that can be optimally manufactured and optimally processed is obtained until the wheel is satisfied. The analysis type is “Shape Optimization” topology optimization module. Compared with the static analysis, the meshing and applied boundary conditions are the same, and the spoke volume optimization target is set to 40%. The analysis result is shown in Figure 4.
Fig 4. Analysis results of hub topology optimization

As can be seen from the analysis results, the red area (Remove) is a removable area, and the pink area (Marginal) is a removable and reservable area boundary.

Referring to the results of topology optimization analysis, considering the aesthetics and processing technology of the spokes, the spokes are redesigned, and the newly designed hub is shown in Figure 5.

Fig 5. Optimization Design effect of Hub

Tab3. Comparison of hub optimization

| Parameter                  | Before optimization | Optimized |
|----------------------------|---------------------|-----------|
| Deformation displacement   | 0.397mm             | 0.495 mm  |
| Maximum stress value       | 176.19Mpa           | 241.18Mpa |
| Quality                    | 9.5693kg            | 7.3619 kg |

It can be seen from the data in Table 3 that the optimized wheel hub has little change in deformation displacement (0.397mm before optimization and 0.495mm after optimization), indicating that the strength is not much different before and after optimization; but the quality is 9.5693kg. The reduction is 7.3619kg, which greatly reduces the quality, and the spoke design is also more beautiful, achieving the purpose of lightweighting the car.

4. CNC simulation processing

4.1 Introduction to CATIA Software Programming Module

In the current field of computer-aided manufacturing, CATIA software is one of the most widely used CNC software. With the CATIA software, tool paths are available for all machining tasks. The machining methods supported are contour milling, hole machining, multi-axis machining, wire cutting machining and turning machining. In addition, the generated machining program can meet the needs of various types of CNC machine tools after being processed by the processing module[15].

The CATIA software programming NC program flow is:

1. Create a design model and artifact planning;
2. Entering the processing environment;
3. Perform NC operations (including creating programs, creating geometry, defining tools, and defining machining methods, etc.);
(4) Performing simulation processing using the generated tool path file;
(5) According to different numerical control systems, select different post-processing programs to generate G codes that can be recognized by the numerical control system.

Using the CATIA software, the optimized hub is used as the processing object; taking rough machining as an example, the simulation results are shown in Figure 6. Through the inspection of virtual processing, the product's processability, processing efficiency and material effective use efficiency have been improved accordingly, shortening the cycle of product development from design to manufacturing, greatly enhancing the competitiveness of enterprises.

Fig 6. Schematic diagram for the numerical simulation results of hub

4.2 Post-processing generates G code
Select the CATIA software default post-processing program "IMS", select the CNC system as "FANUC", after executing the above program, the generated G code is shown in Figure 7.

Fig 7. G code for NC machining of hub

5. Conclusion
(1) For the automobile hub that completed the initial design (mainly the spoke structure design), the finite element software ANSYS Workbench was used to analyze the static force of the wheel hub; then according to the topology optimization module in Workbench, the spoke structure is optimized. According to the analysis results, the hub is redesigned to obtain the optimal spoke structure. It can be found that the deformation displacement of the rear hub is optimized. Not big, but the quality has been reduced from 9.5693kg to 7.3619kg, which greatly reduces the quality, and the spoke design is more beautiful, achieving the purpose of lightweighting the car.

(2) Using the NC programming module of CATIA software, the CNC wheel hub was numerically simulated and processed, and the NC simulation machining tool path and simulation machining results were obtained, and the G code that the CNC machine tool can execute was generated.

Acknowledgments
Authors gratefully acknowledge assistance or encouragement from colleagues, special work by technical staff or financial support through the National Natural Science Foundation of China (Grant No. 51605196) and the Jiangsu Agriculture & Animal Husbandry Science and Technology Vocational College (Grant No. NSF201707) should do so in an unnumbered Acknowledgments section immediately following the last numbered section of the paper.
References

[1] Luo Si-dong. (2004) Magnesium alloy development and application in automobile[J]. Automobile technology & material, 3(4): 38-39.
[2] Song Chun-qiang. (2006) Market Demand and Development Trend of Aluminum Wheels for Automobile [J]. Aluminium Fabrication, 171(5): 8.
[3] Xiao Jun. (2012) Extra wide tyre is a new measure for light weight development of heavy trucks in the future [J]. Rubber & Plastics Resources Utilization, pp: 16-21.
[4] Wang Ming-ming. (2011) Structure Design and Optimization of Aluminum Alloy Automobile Wheel D]. Changchun: Jilin University.
[5] Zheng Songlin, Zheng Zuanx, etc. (2011) An Analysis on the Dynamic Characteristics of Automotive Wheels of Magnesium Alloy [J]. Automotive Engineering, 2(1): 148-151.
[6] Lihong Wu, Siyuan Long, Hanxue Cao. (2004) Effect of Material Properties on Service Stress in Light Alloy Wheels. International Conference On Magnesium. Beijing, pp: 23-26.
[7] Zhang Zong. (2013) Structure optimization and test analysis of the aluminum alloy wheel [J]. Modern Manufacturing Engineering, 6(1): 116-120.
[8] Yan Shirong, He Longjun. (2014) Dynamic behavior analysis and structure optimization of magnesium alloy wheel of vehicle [J]. Journal of Fuzhou University (Natural Science Edition), 42(4): 584-590.
[9] Li Feng’e, Xu Shaoyong, Long Siyuan. (2007) Effects of Spoke Transition Radius on the Service Stress Concentration of Mg Wheel [J]. Special Casting & Nonferrous Alloys, 27(11), 841-842.
[10] Zhou Yuqing. (2008) Fatigue Life Estimation and Optimum Design of Magnesium Alloy Wheel [D]. Chongqing: University Chongqing.
[11] Wang Sumei, Bian Leilei, etc. (2015) Radial Fatigue Life Prediction of Large Size Aluminum Alloy Wheel Hub for Heavy Automotive Vehicles [J]. Agricultural Equipment & Vehicle Engineering, 53(7): 46-51.
[12] Li Jiaying. (2016) The Finite Element Analysis and Optimization Design of Heavy Duty Truck Wheel Hub [D]. Nanjing: Jiangsu University.
[13] Liu Jiaxing. (2010) Specification of Huatai aluminum alloy wheel product [S]. America TRA, pp: 12-18.
[14] Zhao Liang. (2013) Structure Optimization Design Based on Ansys Magnesium Alloy Wheels [D]. Qinghuangdao: Yanshan University.
[15] Qian Yanglin. (2010) Aluminum alloy car wheel mold UG study of CNC machining [J]. Manufacturing Automation, 4: 30-32.