Journal of Applied Science and Engineering Innovation, Vol.4 No.1, 2017, pp.25-30
ISSN (Print): 2331-9062
ISSN (Online): 2331-9070

A Novel Lightweight Design Method for Automobile Hub

Jianxin Zhang, Guoying Yang
Key Lab of Advanced Design and Intelligent Computing,
Dalian University, Jinzhou New District, Dalian, *116622, China*

Abstract*:* In order to achieve the purpose of lightweight of automobile wheel hub, the paper carries on the structure optimization of the wheel hub from the concept phase. We establish the finite element model and mathematical model of the hub, then use the compromise programming method to solve the problem of multi-working condition and take it combines with the techniques of variable density topology optimization. Finally, we obtain three kinds of innovative structure of the wheel hub. We take a two times design for the three kinds of wheel hub and carry out the finite element analysis to verify. The result shows that the seven vice wheel hub is more meet the requirements than eight vice wheel, nine vice wheel and the original wheel. The quality of seven vice wheel is reduced 12.2% than the original hub under the premise of meeting the material's allowable stress.

Keywords: concept design; wheel hub; topology optimization; lightweight

INTRODUCTION

Energy saving and emission reduction has become the main direction of the development of automobile industry, automobile lightweight is the best way to realize the energy saving and emission reduction, reasonable structure design is an effective means of lightweight of the automobile. Automobile hub is an important part of the automobile in the running process, force and torque between the car and the ground are bear and passed by the hub, it directly affects the vehicle's running stability, safety, reliability, comfort, traction and appearance shape, it has great influence on vehicle's overall energy consumption and the life of tire. The development of lightweight structure technology of the automobile is very fast in our country, many domestic researchers carry out optimization of the automobile is based on existing structure of the car by finite element simulation technology, Although it has achieved good results, but it ignores the structure's conceptual design phase.

The conceptual design is an important stage of mechanical products, it is largely determines the product's function of the customer's requirements [Dieu 1988]. Compared with structure optimization in the later, the earlier design is less cost and has more design freedom. Through scientific analysis of the conceptual design stage, we can establish the ideal design model that it can reduce a lot of repeated modification when needs improvement in the late. It not only could allow us to accelerate product-design cycles but also reduce the cost [Gao et al., 2012; Geun et al., 2014; Sleesongson and Bureerat 2013] .

Topology optimization is the main design methods of structural's conceptual design, its basic idea is to convert the problem of seeking the optimal topology structure to the problem of solving optimal layout of materials within the design region, so as to achieve the optimum target [Eschenauer and Olhoff 1982; Feng 2013; Li et al., 2010; Sleesongsom et al., 2013]. Topology optimization is an innovative optimization design method and it is widely used in CAE, in this paper, we work out the finite element analysis for the automobile wheel hub by the HyperWorks software. We get the new material distribution of the hub's structure by the method of conceptual design under the premise of satisfying the strength, which not only reduces the weight but also obtains the innovation type structure of wheel hub, it has certain guiding significance for the wheel hub's lightweight design and Innovation design.

THE ESTABLISHMENT OF CONCEPTUAL DESIGN MODEL OF AUTOMOBILE WHEEL HUB

The process of product's conceptual design is a series of ordered, organized and targeted design activities [Wen 2011]. Conceptual design is the early stage of the product's engineering design process. This stage largely determines the performance of the product. Although this phase's actual investment accounted for only 5% of its costs, it determines the product's 70% of the total cost. In this paper, we use topological optimization as the conceptual design stage's method. Topology optimization is a kind of innovative design method, which has more design freedom and greater design space respect to size optimization and shape optimization [Saitou et al., 2005].

The topology optimization includes variable density, evolutionary structural optimization (ESO), homogenization and level set. The variable density method can reflect the essential character of topology

Corresponding Author: Jianxin Zhang, Key Lab of Advanced Design and Intelligent Computing, Dalian University, Dalian 116622, China.

optimization respect to other method, its concept is simple and the number of design variables is less, therefore, the variable density method is more suitable for practical engineering application's structure optimization design. Variable density method is to introduce a intermediate variable of density between 0~1, each unit's "cell density" from the design space of finite element model as the design variables. There have a certain relationship between the "cell density" and the elastic modulus E of structure of the material, which is Continuous values between 0~1. If the unit density is 1 after optimization, it indicates that this unit's material is very important and it needs to retain. If the unit density is 0 after optimization, it indicates that this unit's material is not important and it can be removed. Accordingly, it achieves the efficient use of materials and realizes the lightweight design [Jiao et al., 2013; Yuan et al., 2013].

The design variables, constraint conditions and design goals are the three elements of optimization design. The main idea of automobile wheel hub's conceptual design model is also to determine the design variables, constraint conditions and design goals. In this paper, we make the material's unit density of wheel hub as design variable, the maximum volume fraction as constraint conditions, the minimum compliance (maximum stiffness) as target, and so we establish a mathematical model of the wheel's conceptual design [Liu and Wang 2005]. As shown below:

$$\begin{cases} Min: \ C(x) = U^T KU = \sum_{e=1}^{N} (x_e)^p u_e{}^T k_0 u_e \\ KU = F \\ \dfrac{V(X)}{V_0} \le f \\ 0 \le X_{\min} \le x^e \le X_{\max} \end{cases} \quad (1)$$

Where, x is the design variables (relative density); x_e means unit design variables; $C(x)$ is structural flexibility; F is load matrix; U is displacement matrix; K is the whole stiffness matrix; u_e is unit displacement matrix; k_0 is element stiffness matrix; N is the number of design variables; $V(X)$ means the effective volume in the design variables; f is Volume coefficient; $X_{\max}$ is the design variable upper limit of unit; $X_{\min}$ is the design variable upper lower of unit; p is the penalty factor and V_0 is the effect volume as the design variable is 1.

THE ESTABLISHMENT OF CONCEPTUAL DESIGN MODEL OF AUTOMOBILE WHEEL HUB

A wheel has two main components: rim and spoke, due to the rim has the standards prescribed by the state, so we only research on the lightweight design for the spoke part in this paper. According to the actual size of a small car wheel hub, we establish the three-dimensional geometric model. We get the hub's concept geometric model based on the established three-dimensional geometric model by filled with the design space of spoke as shown in figure 1.

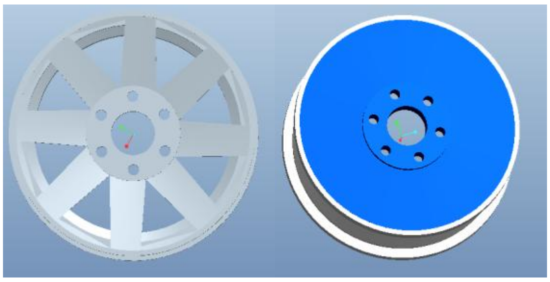


Figure 1 3D geometric model of wheel hub (left: 3D model of a vehicle wheel; right: the conceptual model)

The model's material is A356 (ZALSi7Mg) from the small car hub and its properties can be got from the mechanical manual [Wen 2010], its properties as follows are shown in Table 1.

Table 1 Material property of A356

Material type	Elastic modulus (MPa)	Poisson's ratio	Density (kg/m^3)	Nature	Stress (MPa)
A356	71000	0.3	2730	Isotropic	240

THE ESTABLISHMENT OF THE FINITE ELEMENT CONCEPT MODEL OF THE WHEEL HUB

Mesh

The 3D model is stored in IGS format and then imported in finite element analysis software. The number of units divided into hexahedral mesh is usually less than the number of units divided into tetrahedral mesh. It not only reduces the solution time but also obtains higher precision. We divide the model into hexahedron and get 185203 elements and 220900 nodes. The warping degree, length width ratio, twist and Jacobian number are all reached the satisfied parameter index. We define the design area and the non design area at this stage. Figure 2 shows the finite element concept model of the wheel hub and its design area and non design area.

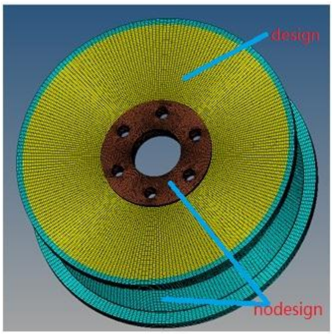


Figure 2 Design and non-design areas in the finite element concept model

Loading and boundary conditions

Condition one: the moment condition of wheel hub [Zhang 2009]. The hub mainly bears three aspect of load in the moment conditions, the first load is pre tightening force which generated by fastening bolts that between the wheel and the drive, the second is the centrifugal force which generated by the wheel hub's high-speed rotation, the third is the moment which generated by transmission shaft. We use pretension element to simulate pre tightening force by optistruct software; the influence of centrifugal force on the wheel hub is very small and we do not do in-depth research, ignore it; The third moment is more complex, in order to simplify the finite element computation, we transform the wheel's dynamic load (moment) into static load, we divide the hub circumference into 12 parts and each part's interval is 30 deg. We use the RBE3 element to simulate the transmission shaft and apply the same size load sequentially along the vertical direction of the drive shaft at the end of the propeller shaft. Under this condition, we fix the wheel's rim edge. Figure 3 is the moment simulation:

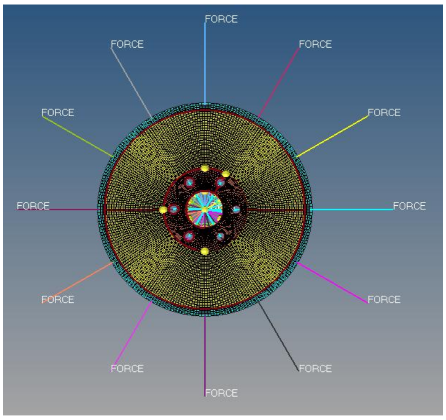


Figure 3 Simulation of the hub's moment condition

Condition two: the radial condition of the wheel hub [Zhang 2009]. The hub's loads mainly include the centrifugal force, the tire pressure and the opposite reaction from ground. We ignore the centrifugal force just like at the moment condition; the tire pressure is 0.25 MPa according to the experience; the hub's opposite reaction from ground (radial load) is $F_r = F \cdot k$. In this formula, F_r is the radial load, F is the wheel's maximum rated load, k is the enhancement coefficient of simulation. Because the hub is subjected to radial load repeatedly, so the loads we given must be sequentially applied in the equal angle interval by the RBE3 unit in the rim of the circle, the angle is 60 degrees, it divides the circle into six equal parts and it requires to apply six force in one cycle. At this point, we fix the bolt's freedom. Figure 4 is the radial simulation:

THE HUB'S LIGHTWEIGHT DESIGN BASED ON TOPOLOGY OPTIMIZATION

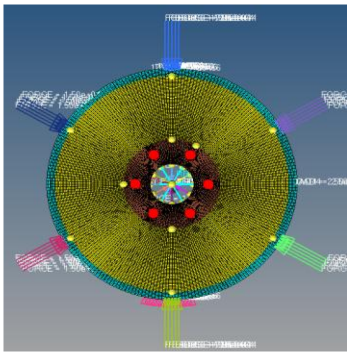


Figure 4 Simulation of the hub's radial condition

We mainly study the hub's radial condition and the hub's moment condition in this paper, each condition has different constraints and different stress, so the topology optimization will get two different topological structures, and thus the stiffness optimization under multiple loads has become a multi objective optimization problem. In order to achieve the optimal topology structure, we introduces the compromise programming method to solve the optimization problem of multi working condition, the compromise programming method can transform the multi-objective optimization problem into a single objective optimization problem through split the different effect that the design variables impact on multiple targets [Rao and Freiheit et al., 1991; Xiang et al., 2012]. The following is a compromise programming mathematical model of the minimum compliance under various conditions:

$$\begin{cases} Min: \ C_h(x) = \sqrt{\omega_1^2\left(\dfrac{c_1(x) - c_1^{\min}}{c_1^{\max} - c_1^{\min}}\right)^2 + \omega_2^2\left(\dfrac{c_2(x) - c_2^{\min}}{c_2^{\max} - c_2^{\min}}\right)^2} \\ KU = F \\ \dfrac{V(X)}{V_0} \le f \\ 0 \le X_{\min} \le x^e \le X_{\max} \end{cases} \quad (1)$$

In this formula, $c_1(x)$ is the flexibility value of the moment condition, $c_2(x)$ is the flexibility value of the radial condition, $c_1^{\max}$ and $c_1^{\min}$ are the maximum flexibility value and the minimum flexibility value under the moment condition, $c_2^{\max}$ and $c_2^{\min}$ are the maximum flexibility value and the minimum flexibility value under the radial condition, ω_1 is the weight coefficient under the moment condition, ω_2 is the weight coefficient under the radial condition.

After the weight coefficients of those two kind conditions are defined, we can establish the optimization process of the hub concept model, and the following is the basic step of the optimization problem:

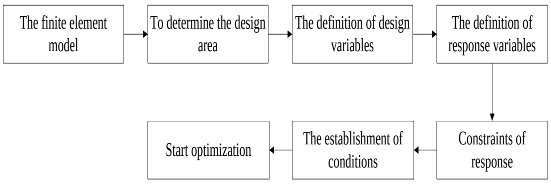
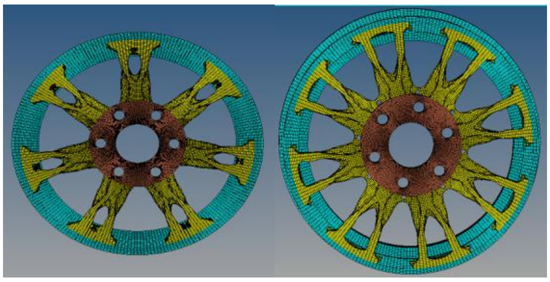


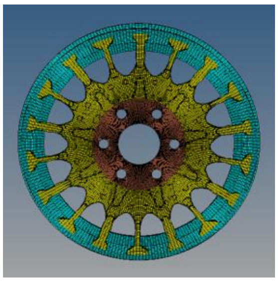
Figure 5 The optimization process

We should define the design area of the hub's finite element model first, and then define the design variable, the response variable and its application of constraint. In this paper, the relative density is defined as its design variables, the volume and the flexibility are defined as its response variables. We find the optimal distribution of materials by using the volume as the constraint and the minimum compliance as the target. The mathematical model of it such as Eq. (1) displayed, and then establish all the working conditions of the hub. When all the steps are checked without any error, we start the optimization.

After several times of iterative calculation, we obtain three material distribution models of the hub, which are shown as follows:



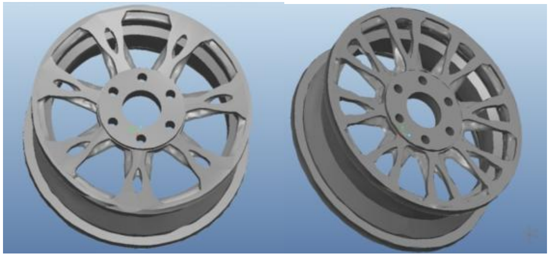
(a) Seven-vice wheel (b) Eight-vice wheel



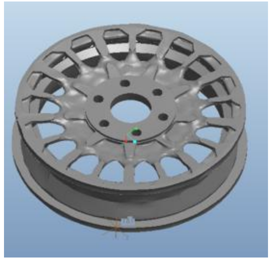
(c) Nine-vice wheel

Figure 6 The three structure types after optimization

The optimized model is irregular and it is difficult to achieve by using the existing manufacturing processes, we need to redesign in CAD software if we want to apply it to practical engineering. OSSmooth is a semi-automated tool, it can explain to the CAD model according to the deformation of certain grid for the results of the topology optimization, and then put the optimized results into CAD software for design reference. In this paper, we use OSSmooth module from Hyperworks to generate iGS file and import the file to the CAD software. The diagrams are shown below after introducing:



(a) Seven-vice wheel (b) Eight-vice wheel



(c) Nine-vice wheel

Figure 7 3D geometric model of wheel hub after introducing

It needs to two times design for the optimized model after importing to the CAD software. We establish 7 vice wheel, 8 vice wheel and 9 vice wheel respectively base on the reference of the optimized model. The geometric model of redesign not only can be realized in the production process but also can facilitate the finite element analysis and validation in the next step. Figure 8 is the three-dimensional model after the redesign.

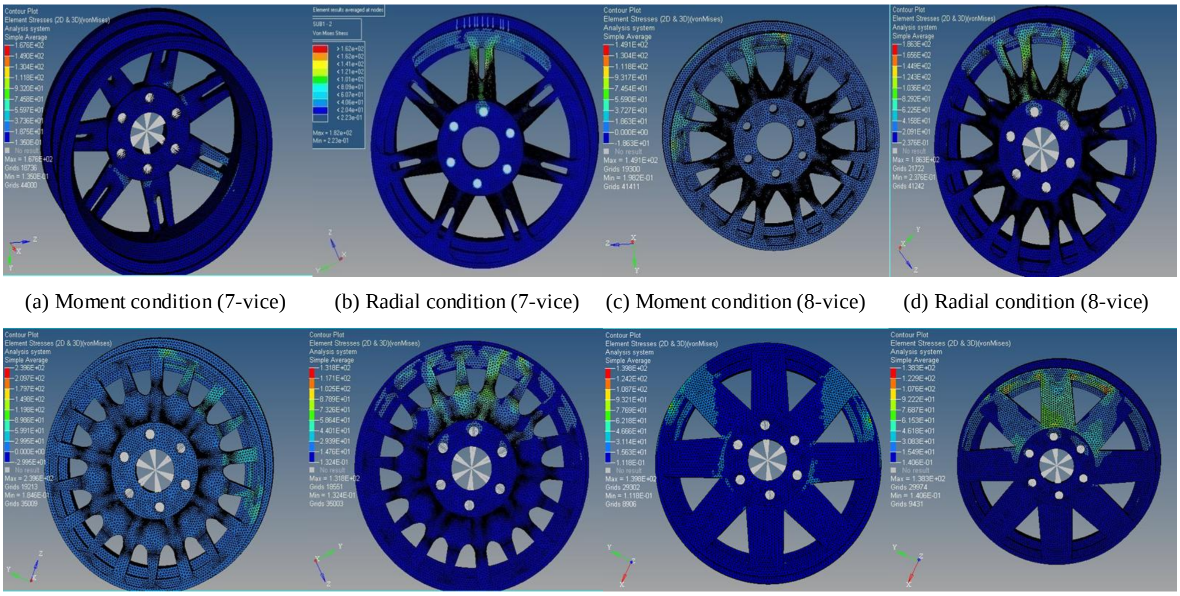
(a) Seven-vice wheel (b) Eight-vice wheel

Figure 8 (c) Nine-vice wheel

Figure 8 The 3D model after the redesign

THE FINITE ELEMENT ANALYSIS AND VERIFICATION OF THE REDESIGN WHEEL HUB

We import the redesign model (IGS format) into hypermesh software, and then establish finite element model, final we work out the finite element analysis for the model. The solution results are shown as Fig 9. From the analysis diagram we can know 1) stress of wheel hub is mainly concentrated in the junction of spoke and rim; 2) the maximum stress of the optimized wheel is bigger than that ofthe original wheel, but it is still in the range of the material strength; 3) The seven vice wheel is the lightest weight, and the maximum stress of condition one and condition two is in the range of Aluminum alloy material's allowable stress; 4) The eight vice wheel's maximum stress is also in the material's allowable range, but its mass and the maximum are greater than the seven vice wheel. We do not consider the nine vice wheel, because its maximum stress is equal to the material's maximum allowable stress. In summary, the seven vice wheel is more appropriate and its weight is reduced 12.2% than the original wheel.



(a) Moment condition (7-vice) (b) Radial condition (7-vice) (c) Moment condition (8-vice) (d) Radial condition (8-vice)

(e) Moment condition (9-vice) (f) Radial condition (9-vice) (g) Moment condition (original) (h) Radial condition (original)

Figure 9 The stress distributions of each wheel hub in moment and radial conditions

Table 2 Comparison results of each wheel

Wheel condition	Max. stress in moment condition（Mpa）	Max. stress in radial condition（Mpa）	Mass（kg）
Original wheel	140	138	7.4
Seven vice wheel	168	162	6.5
Eight vice wheel	149	186	6.8
Nine vice wheel	240	131	7.0

CONCLUSION

As an important means of conceptual design, topological optimization technology not only can get the reasonable distribution of material but also can realize the lightweight. We achieve the concept design of the automotive wheel hub by the topology optimization and obtain three innovation structure models. Through the verification and analysis, we conclude that the seven vice wheel's mass is reduced 12.2% than the original wheel under the premise of meeting the material's allowable stress, and there is a certain margin between the seven vice wheel's maximum stress and the material's allowable stress, which can be a further guidance for lightweight.

REFERENCES

Dieu F., 1988, "Structural Optimization of A Vehicle Using Finite Element Techniques", FISTA, 1988.

Eschenauer H. A., Olhoff N., 1982, "Optimization Methods in Structure Design. Euromech-Colloquium", Proceedings of the Euromech-Colloquium 164, Oct 1982.

Feng Y., 2013, "Optimization Analysis and Simulation of Topology Design of Automobile Engine Hood", Computer simulation, 30(12): 154-157

Gao Y., Zhou D. L., Cai H. G., 2012, "The Application of CAE Technology in Auto Concept Design", Chinese Journal of Automotive Engineering, 2(1): 63-65

Geun Y. K., Seung H. H., Kwon H. L., 2014, "Structural Optimization of A Knuckle with Consideration of Stiffness and Durability Requirements", The Scientific World Journal, 768692: 1-7

Jiao H. Y., Zhang Q. C., Li W. J., et al., 2013, "Periodic Topology Optimization Using Variable Density Method", Journal of Mechanical Engineering, 49(13): 132-138.

Li Y. W., Xu T., Zuo W. J., et al., 2010, "Topology Optimization Design of T- Joints in Concept Auto body", Journal of Jilin University (Engineering and Technology), 40(2): 352-356.

Liu Z. G., Wang M. G., 2005, "Conceptual Structure Design Based on the Topology Optimization Method", Mechanical and Electronics, 10: 27-29.

Rao S. S., Freiheit T. I., 1991, "A Modified Game Theory Approach to Multi-Objective Optimization", ASME Journal of Mechanical Design, 113(3): 286-291

Saitou K., Izui K., Nishiwaki S., et al., 2005, "Survey of Structural Optimization in Mechanical Product Development", Trans ASME, 21:214-226

Sleesongson S., Bureerat S., 2012, "New Conceptual Design of Aeroelast Wing Structures by Multiobjective Optimization. Eng". Optim, 45(1): 107-122

Sleesongsom S., Bureerat S., Tai K., 2013, "Aircraft Morphing Wing Design by Using Partial Topology Optimization. Struct Multidisc Optim", 48: 1109-1128

Wen B. C., 2010, "Mechanical Design Manual", China Machine Press, Jan 2010.

Wen B. C., 2011, "Design Methodology of Product", China Machine Press, Dec 2011.

Xiang X. L., Zuo K. T., Xiang Y., et al., 2012, "Multi-objective Topology Optimization Design on SUV frame", Computer Aided Engineering, 21(5): 20-24

Yuan L., Yu H. D., Zhao Y., et al., 2013, "Optimal Design of Supporting Structure in Large Antenna System Based on SIMP Method", Machine design and research, 29(1): 118-122

Zhang X. Z., 2009, "The Example Research for the Mechanical Structure by Topology Optimization Design", Northeastern University.