A Novel Method Combining Reverse Modeling and Topological Optimization for Lightweight Design of Automobile Wheel Hubs with Hollow Ribs

P.F.Xu[1], †S.Y.Duan, S. Liu[1]

[1] College of Mechanical Engineering, HeBei University of Technology, Tianjin City, 300401 China

†Corresponding author: dsy605106@163.com

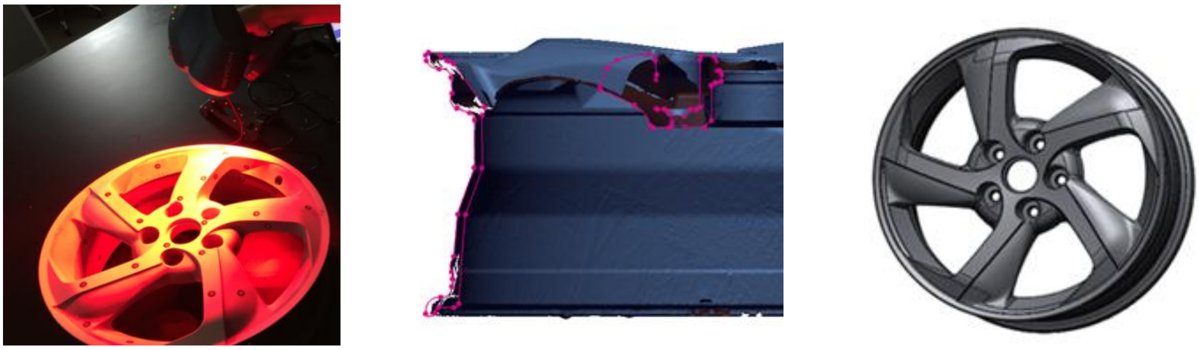
Abstract

The lightweight of wheel hubs is very important for reducing the unsprung mass and critical to improve the vehicle dynamic and braking performance of vehicles, and hence the control stability and comfortability. However, many current experience-based lightweight designs of wheel hubs have shortcomings resulting in uneven distribution of materials in terms of mechanics principles. This work develops a new method that combines the reverse modeling with topological optimization techniques to obtain lightweight wheel hubs, based on the principles of mechanics. A reverse modeling technique is firstly used to scan and create the initial 3D geometry of the wheel hub with solid ribs. The finite element (FE) method is then used to perform stress analysis to find the maximum stress and its location of wheel hub under multiple conditions. The FE model is then divided into optimization region and unoptimized region: the former the interior portion, and the latter is the outer surface of the wheel hub. A topology optimization is next the conducted, leading to a wheel hub with hollow ribs: the interior material of the ribs of the wheel hub is automatically removed. Finally, the hollow wheel hub is reconstructed with wall thickness about 5mm, via again a reverse modeling technique. Our results show that the reconstructed model can reduce the mass of 12.7% compared to the pre-optimized model (from 11.26 kg to 9.83 kg). The present method of combined with reverse modeling and topology optimization can guarantee the optimal distribution of wheel hub material based on mechanics principle. It can be performed automatically and hence shorten the time for optimal lightweight designs to improve the performance of the many other existing structures and structure components.

Keywords: Reverse modeling; hollow wheel hub; Finite element; Topology optimization; Lightweight structures.

The 3D geometry model of the wheel hub

Reverse modeling technique is used to create the 3D geometry of the five-spoke aluminum alloy wheel hub of a selling SUV. The weight and size of wheel hub are 10.5 kg and 17×6.5 inches, respectively. Current mainstream 3D handheld scanner was selected to scan the wheel hub due to that the size of the wheel hub is not very large and the appearance accuracy requirement is not high [5]. Creating the wheel hub model includes the following three main steps, shown as in Fig. 1. In the first step, the date acquisition is to scan the wheel hub with a hand-held laser scanner with an accuracy of 0.5 mm, thereby obtaining a reverse mesh as shown in Fig. 1 (a). In the second step, the reverse mesh editing is the optimization of the surface contour of the reverse mesh in the reverse software as shown in Fig. (b). In the last step, the 3D geometry model of the wheel hub available for topology optimization with in the solid modeling step is obtained based on the reverse mesh.



(a) (b) (c)

Figure 1. Reverse modeling, (a) data acquisition, (b) reverse mesh editing, (c) solid modeling

Numerical analysis of the wheel hub

Aluminum alloy A356 material is selected for casting wheel hubs. The material parameters of the aluminum alloy A356 material are shown in Table 1. [2]. The finite element analysis model is assembled from a wheel hub model and a torque shaft as shown in Fig. 2 [6]. For the one part, the mass of wheel hub model can be calculated in Abaqus and it is 11.26 kg. Due to the modeling error in reverse modeling, the mass of the FE wheel hub model is not equal to that of the real wheel hub. For the other part, axle was modeled using the material parameters of Q235-B material, and according to the formula (1) the length of axle is 500mm. Load: The static F_i (i= 1,2,3,……24) was applied every 15° within 360° of the axle end, the static load F_i applied at the axle end is 5265N, the role of F_i points and directions as shown in Fig. 2. In order to simulate the actual operation of the wheel hub in a cycle when the load suffered. The bending moment of a single wheel in this cycle was provided by the automobile manufacturer for 2632Nm. In 错误!未找到引用源。, the axle length of the loading moment and it is 500mm, and according to the formula (1):

$$M = FL \tag{1}$$

Boundary conditions: The six degrees of freedom of the rim based plane and the rim edge surface were constrained. as shown in the red area of 错误!未找到引用源。 [7].

Table 1. Material properties of A356

Elasticity Modulus	Poisson's ratio	Density	Yield stress	Ultimate stress
72.00GPa	0.33	2.67×10^3 kg/m^3	218.0MPa	283.0MPa

The 10-node modified quadratic tetrahedron was applied to assembly model. This element type makes it easy to mesh complex contact-containing models. The elements number of the wheel hub FE model was169173. The node number of the wheel hub was 281034. The maximum stress position within wheel hub ratation could be obtained, and the maximum stress was 128.29MPa as shown in Fig. 4(a). Topology optimization will be based on this.

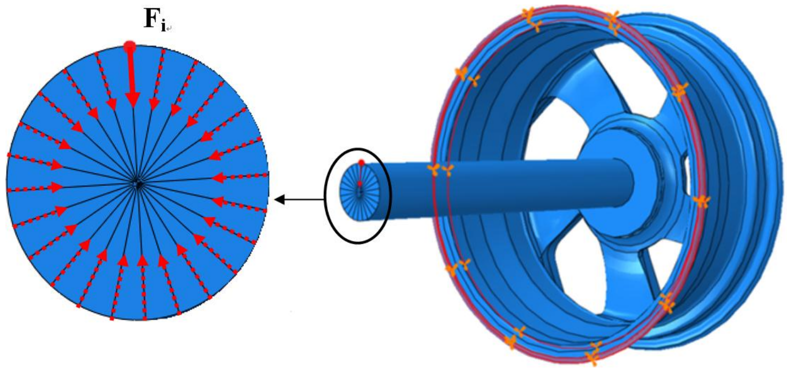


Figure 2. Finite element model of wheel hub

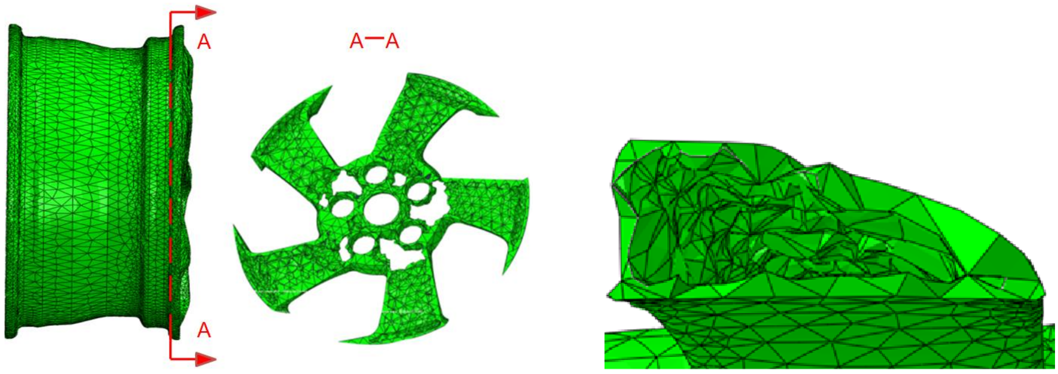
Structure optimization and Model reconstruction

In this section, a model of hollow wheel hub was obtained by topology optimization technology. According to material mechanics, it could be known that the normal stress of a beam was proportional to the distance from the neutral layer as Eq. (2).

$$\sigma = E\left(y/\rho\right) \tag{2}$$

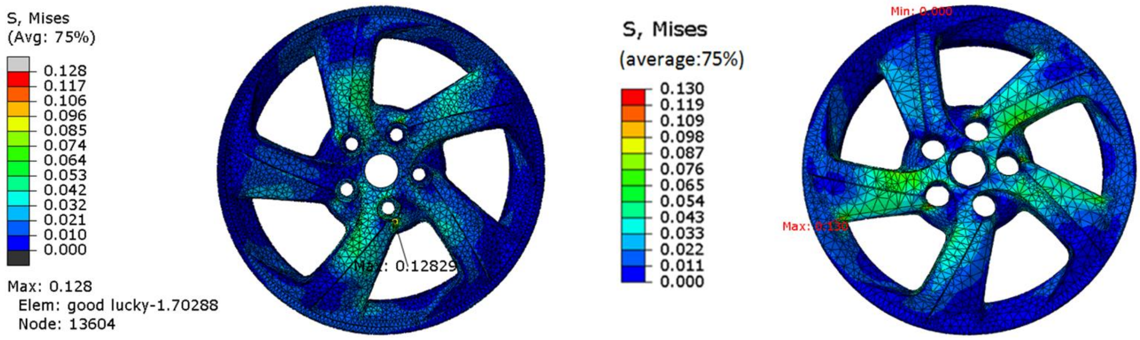
The surface material of the wheel hubs withstands most of the load on wheel hubs. The material of wheel hubs with less efficient can be removed. Variable density topology optimization was used to reduce the wheel hub material in Abaqus. The FE model is then divided into optimization region and unoptimized region: the former the interior portion, and the latter is the outer surface of the wheel hub. The design area allows to reduce the material during optimization and it was spokes of wheel as shown in Fig. 2. Cell density is design variable. The optimization objective function is set to minimize strain energy values. The purpose is to maximize the overall stiffness of the hub. The optimization Constraint is set to optimized volume ≤ 75% of the original volume. The method of freezing the surface load area can be chosen to preserve the material of the wheel hub surface. The load is a pressure of 1 Pa applied to the surface of the design area during optimization (This stress generated by this pressure was 1×10^{-6} % of the maximum stress on the FE model), so it could be neglected. The topology optimization would remove the material inside the wheel hub mesh model. The hollow structure could be obtained as shown in Fig. 3.

The optimized mesh model was derived and reconstructed using the same method with the reverse modeling. A new weight of the hollow wheel hub can be obtained as 9.83 kg. The weight of optimized wheel hub is 12.7% (1.43 kg) less the original wheel hub model. The same finite element analysis was applied to the optimized model. The larger element can speed up computing, and the result as shown in Fig. 4(b), the maximum stress position had changed. It located in the area where spoke and rim connecting position, with a maximum stress of 130MPa. From the FE results, it can be seen that the wheel hub mass is significantly reduced while the maximum stress of the original wheel hub and the optimized wheel hub is almost the same. This result is satisfactory [8].



(a) (b)

Figure 3. The optimized wheel hub model, (a) the optimized model is cut along the A-A section the, (b) radial sectional view of a hollow single-spoke.



(a) (b)

Figure 4. (a) Original wheel hub stress cloud. (b) Optimized wheel hub stress cloud

References

[1] Shuyong Duan, Yourui Tao, Xu Han, Xujing Yang, Shujuan Hou, Zhangping Hu. (2014) Investigation on structure optimization of crashworthiness of fiber reinforced polymers materials. Part B 60 471-478.

[2] Ch. P. V. Ravi Kumar, Prof. R. Satya Meher. (2013) Topology Optimization of Aluminium Alloy Wheel. International Journal of Modern Engineering Research. pp-1548-1553.

[3] G. Chiandussi, I. Gaviglio, A. Ibba. (2004) Topology optimization of an automotive component without final volume constraint specification. Advances in Engineering Software 35 609-617.

[4] Denghong Xiao, Hai Zhang, Xiandong Liu, Tian He and Yingchun Shan. (2014) Novel steel wheel design based on multi-objective topology optimization. Journal of Mechanical Science and Technology 28 (3) 1007-1016.

[5] Li Z, Xiang HY, Li ZQ, Han BA , Huang JJ. (2013) The Research Of Reverse Engineering Based On Geomagic Studio. Applied Mechanics and Materials, ISSN: 1662-7482, Vols, 365-366, pp 133-136.

[6] ISO 3006:2005-Road vehicles--Passenger car wheels for road use--Test methods.

[7] Liu GR, Quek SS. (2013). *The finite element method: A practical course, 2nd edition.*

[8] Barbieri SG (Barbieri Saverio Giulio) ; Giacopini M (Giacopini Matteo); Mangeruga V (Mangeruga Valerio); Mantovani S (Mantovani Sara). (2017) A design strategy based on topology optimization techniques for an additive manufactured high performance engine piston. Procedia Manufacturing, Volume 11,Pages 641-649.