Parametric Design and Optimization of Alloy Wheel Based on Dynamic Cornering Fatigue Test

Worawat Puangchaum, Supakit Rooppakhun*, Veena Phunpeng

School of Mechanical Engineering, Institute of Engineering, Suranaree University,
111 University Avenue, Muang, Nakhon Ratchasima 30000, Thailand

*Corresponding Author: supakit@sut.ac.th

Abstract

This study proposes a parametric design and optimization technique for alloy wheels based on a dynamic cornering fatigue test. A case study of 13-inch diameter alloy wheels was considered based on the standard dynamic cornering test. The effect of the width and thickness of the wheel rim on the principal stress and fatigue life based on minimizing the weight is described for each scenario. A total of twenty-seven design of experiment are created and expanded in response surface and sensitivity analysis using Finite Element Analysis (FEA) software. The geometric parameter of alloy wheel reveal the surface response to principal stress and fatigue life. The rim width significantly displayed higher sensitivity than the thickness of alloy wheel. The conclusion can be draw that the magnitude of maximum principal stress should not exceed as 145 MPa for passed the standard fatigue life of 100,000 cycles. The advantage of this parametric optimization technique is that it allows for an improved shape and size design process for alloy wheels.

Keywords: Parametric Design, Alloy Wheel, Dynamic Cornering Test, Fatigue life

1. Introduction

Automobile parts are continuously designed and developed to satisfy customer requirements in quality, comfort, reliability, safety and life cycle. To meet standards, improvements have to undergo experimental design and product analysis. Currently, computer simulation is widely used for the improvement and development of automotive components, as well as to save time⁽¹⁾.

According manufacturing process, they start from styling design department to design topology of wheel, the process developing safe, fuel efficient and lightweight to meet governmental regulations and industry standards. To meet these regulations and conditions, determination of the mechanical behavior of the wheel is important. However, testing and inspection of the wheel during the production process is time consuming and costly. For these reasons, it is important to reduce the development time and testing phase of a new model wheel. As stress analysis of the wheel involves complicated geometry, Finite Element Analysis (FEA) is generally used in the design stage of production development to inspect the mechanical performance of wheel. FEA simulation of the wheel tests can significantly reduce the time and cost required to finish the wheel design.

Wheels are one of the main components of a car which are adjacent to tires and support vehicle loads. Lightweight alloy wheels significantly differ from normal steel wheels because in that they are specifically designed for the improvement of steering, speed and fuel consumption. Generally, a passenger car wheel must undergo three standard tests before going to production: (1) the dynamic cornering fatigue test; (2) the dynamic radial fatigue test; and (3) the impact test. Various kinds of alloy wheel take a long time for fatigue destructive testing and the cost is high. Doing engineering analysis before testing therefore becomes important to reduce cost and get a fatigue life cycle estimate before testing ⁽²⁾

There has been some research paying special attention to fatigue prediction and weight reduction in aluminum alloy wheels. Liangmo Wang et al.,⁽³⁾ studied the effect of fatigue strength under static load conditions for various shapes of alloy wheel using a commercial finite element analysis

package. The results of this research suggested that a modified shape lead to a stress reduction in the alloy wheel and an increase in the fatigue life. The use of a computer successfully aided to help in the analysis in order to reduce the cost and shorten development time. B. anusha et al.,⁽⁴⁾ presented results of a strength analysis for a four wheeler using four different wheel shapes consisting of straight, inclined, y-shape and honey comp-shape. The study analyzed the model with variables including different materials and different load. Yeh Liang Hsu and Ming Sho Hsu⁽⁵⁾ emphasized weight reduction in a disc type wheel. This study used a sequential neural network approximation method to evaluate the discrete-variable engineering optimization. The algorithm searched for the optimal point in the feasible domain fatigue constraints. From these studies, the parametric variable was necessary to find the optimal geometric design of the alloy wheel.

In this study, parametric design and optimization was performed under dynamic cornering fatigue constraints included the objective of minimize weight. FE models of aluminum alloy wheels were constructed to analyze the wheel behavior based on the standard dynamic cornering fatigue test. Based on Goodman's fatigue criterion, the analysis was used to predict the fatigue life of the wheel, as well as the response surface sensitivity.

2. Dynamic Cornering Fatigue Analysis

The dynamic cornering fatigue test simulates the load condition of a wheel during a turn while driving. Fig.1 shows a typical set-up loading method for evaluating cornering fatigue, according to JWL⁽⁶⁾. In Fig. 1, the downside flange of the rim is clamped for the test, and a load arm shaft is attached to the mounting surface of the wheel. A test load applies a constant cyclical rotation bending on the arm shaft end. The test will be passed if at least 1.0×10^5 rotations occur without a crack failure in the wheel. The bending moment of testing can be calculated from eq.1.

Table 1. Chemical ingredients of Al Alloy A356.

Si	6.5-7.5%
Fe	<0.15%
Mg	0.27-0.29%
Ti	0.1-0.15%
Sr	0.005-0.0155%

Fig. 1 Contour setup of dynamic cornering fatigue testing machine.

$$M_{bmax} = S \cdot F_v (\mu \cdot r_{dyn} + d) \quad (1)$$

Where S is the safety factor, F_v is the maximum load that wheel can support (N), μ is the friction coefficient between the tire and road is equal to 0.7, r_{dyn} is the maximum rotation radius that tire act on wheel (m), d is the offset distance from the center alignment of wheel (m).

3. Material properties and FE modeling

3.1 Material Properties

In alloy wheel industries, Alloy A356 is usually used, which has different chemical ingredients depending on each producer. The particular ingredients used in this research are shown in Table 1

Material properties of the wheel were tested using a tensile test on a standard test piece by using a universal testing machine and collecting data from the test piece. The calculated average values for these properties are shown in Table 2.

Table 2. Material Mechanical Properties

Property	Value
Yield strength (MPa)	220
Ultimate Tensile (MPa)	265
Elongation (%)	3
Modulus of Elasticity (GPa)	70

Fig. 2 Stress-life curve of Alloy A356

The standard test piece was further tested under a fatigue resistance test, and the testing data collected in order to find a relationship between the stress and life cycle (S-N curve) for input to dynamic cornering fatigue model. The results of this test are shown in Figure 2.

3.2 Finite element modeling

The finite element modeling and analysis was carried out using ANSYS. The meshing focussed on the spoke and Mounting surface, having a smaller element size in these regions than on the rim, with an element size of 5 mm. Both regions affect the stress, which in turn affects the fatigue life time. A tetrahedron grid with refinements in the curved area was used, as shown in Fig. 3. A spacer with a thickness of 50 mm was used to replace the shaft in order to reduce the number of elements and save time in the analysis. The analysis used boundary conditions and bending moments similar to the cornering fatigue test. The static results used to compute the fatigue analysis and display the results are presented in the next section.

3.3 Boundary condition

In the dynamic cornering fatigue test simulation, the downside outboard flange of rim of the wheel was clamped securely to the test device, and a rigid load arm shaft was attached to the mounting surface of the wheel, as shown in Figure 1. A test load was applied to the arm shaft to provide a constant cyclical rotation bending moment.

Finite element modeling requires us to specify the boundary conditions and load, which were set similar to Dynamic Cornering Fatigue test, with a fixed support at the flange rim. The dynamic cyclical load was represented by 24 discrete loads, 15° apart like the cycle load shown in fig.4. The simulation perform load at 0 degree, by the remote force to apply from back surface of spacer to point B. For each node, the maximum and minimum principal stress during the load cycle were extracted to determine the mean stress σ_m and amplitude stress σ_a of node, as defined below :

(a) bolt hole.

(b) spoke and rim.

Fig. 3 finite element modeling of the alloy

$$\sigma_m = \frac{1}{2}(\sigma_{\max} + \sigma_{\min}) \quad (2)$$

$$\sigma_a = \frac{1}{2}(\sigma_{\max} - \sigma_{\min}) \quad (3)$$

Several different criteria are commonly used in predicting fatigue failure. In this study, Goodman's criteria was used, which can be written as :

$$\left(\frac{\sigma_a}{S_e / n} \right) + \left(\frac{\sigma_m}{S_u / n} \right) = 1 \quad (4)$$

Where S_e is the endurance limit, S_u is ultimate strength of the material, and n is the design factor

Fig. 4 boundary condition set-up 0° loading

Fig.5 An example of a 3D response surface⁽⁸⁾

3.4 Parametric optimization

Design of experiment (DOE) is a technique originally developed for model fitting with experimental data. In design optimization, DOE can be used to fit the simulated response data to mathematical equations. These equations, also referred to as response surface equations, serve as models to predict the response for any combination of design variable values⁽⁸⁾. An example is shown in Fig. 5.

The design experiment to find the optimal parameter to improve the geometry used face-central composite designs (CCDs), also known as Box-Wilson^(9,10) designs, These are appropriate for calibrating full quadratic models.

For wheels that are predicted to pass the cornering fatigue test, the parameters that need to be considered are thickness (t) and width (W) of the spoke alloy wheel. It then becomes a structural optimization problem: to minimize the weight of wheel, subject to geometry and fatigue constraints of more than 1×10^5 cycles, and a constrained maximum principal stress less than or equal to 150 MPa. This problem can be formulated as follows:

$$\text{Minimum weight } (x), \text{ such that } g_e(x), \quad (5)$$

where x is the design variable, such as the thickness and width of spoke wheel; and $g_e(x)$ are constraints of fatigue and principal stress. The model will be 3 prototypes at the fillet at the end of the rod spoke wheel of radius(R) 10, 12.5 and 15 respectively. In each prototype has 9 scenario, shown in fig. 6.

Fig. 6 parameter design of optimization

4. Results and discussion

In this study, the results of static structural and fatigue analysis of alloy wheels with different thickness and width of spoke alloy wheel were investigated for a total of 3 prototypes. In the analysis, maximum principal stress theory was used to compute for support usability all prototypes of alloy wheel. Fig. 7 graphically displays the relationship between maximum principal stress in each scenario. The maximum principal stress in each case occurs at a different value, but the area that the maximum stress occurs at is in the same location, as shown in fig 10. In design point no. 4 for each prototype (R10, R12.5, R15), it was found that the value of the maximum principal stress is 149.29, 147.5 and 145.29 MPa in prototype 1 (R10), 2(R12.5) and 3(R15) respectively. In design points five to nine, the parameter modified was the thickness, which was decreased with increasing design number. It was found that the maximum principal stress increased with decreasing thickness.

The effect of the parameter changes in the scenarios on the fatigue life and weight, shown in fig. 8 and 9. In the fatigue analysis, only the design point 4 of prototype 3(R15) passed the cornering fatigue test, with the life of wheel being 1.02×10^5 cycles at all spokes area of alloy wheel, as shown in fig. 11.

Fig. 7 Maximum Principal Stress each design point

Fig. 8 Fatigue life each design point

Fig. 9 Weight at each design point

The results can be used in the Design of Experiments by fitting them to a response surface, which can quickly provide the approximated values of the output parameters. Fig. 12 displays response surfaces for the principal stress, fatigue life and weight reduction of design point 4 of the prototype 3(R15). Fig. 12a shows the relationship between the width, thickness and principal stress, and is a parabolic surface. The y axis of graph is the parameter of the width, and the x axis is the parameter of the thickness. If the parameter of x axis is increased the value affect to thickness of the spokes wheel will be reduced. Finally, the z axis is the principal stress. It was found that when the thickness increases affect to principal stress have high value of any parameter width, After that principal stress reduce when parameter thickness decrease. The thickness and width parameter over which the optimal value of the principal stress does not exceed 150 MPA is approximately 60.5-61 mm and 18mm respectively.

(a) Front view

(b) back view

Fig. 10 Maximum Principal stress of design point no.4

Fig. 11 fatigue life of design point no.4

Fig. 12b shows the relationship between the width, thickness and fatigue life. The thickness and width parameters giving the optimal value for fatigue life are over a similar range with the principal stress. Out of this range the fatigue life is reduced significantly.

Fig. 12c shows the impact that changing the width and thickness has on the weight. The thickness and width parameter that provide an optimal value for weight is similar both response surface before. However, the optimum range is out of the parameter range for which the model will pass cornering fatigue test.

Fig. 13 shows the relationship between sensitivity and the results of the principal stress, fatigue life and weight analysis of prototype 3. It was found that the thickness parameter affects the results of principal stress and weight more greatly than the width parameter, but for the weight the sensitivity of both parameters does not significantly differ. In fatigue sensitivity, the parameter width affects the life of the fatigue greater than the parameter thickness, if the width parameter increasing to pass the cornering fatigue test will affect the weight is extremely high.

(a) Maximum principal stress

(b) fatigue life

(c) Weight reduction

Fig. 12 response surface of design point no.4, prototype 3

Fig. 13 result sensitivity of prototype 3

Conclusions

This study has presented a technique of parametric design and optimization of alloy wheels under the cornering fatigue test condition. The maximum principal stress and fatigue life were considered as the constrained parameters based on main objective of minimizing weight. The geometric parameters of width and thickness of the wheel rim were analyzed to determine the sensitivity of response. The design of experiment based on face-central composite criteria using finite element analysis software were analyzed. The results revealed that the alloy wheel rim was the critical area, displaying high damage stress distribution under the cornering fatigue test. For the response analysis, the rim width significantly displayed higher sensitivity than the thickness of the alloy wheel. The magnitude of principal stress should not exceed of 145 MPa based on the standard fatigue life of 100,000 cycles. The parametric design could improve the optimal shape and size of alloy wheel for meet the standard of dynamic cornering fatigue test.

Acknowledgment

The authors would like to acknowledge Suranaree University of Technology for funding and facility support among the research activities. The authors would also like to thank Dr James Varley for his helpful comments on this manuscript.

References

- (1) Mandage A.P., Sharma M.H., Rayate A.K., Kange P.N., Hirulkar N.S. : "Fatigue Life Estimation of An Aluminium Wheel Rim Using Finite Element Anlysis",

- International Journal for Science and Advance Research In Technology, vol. 2 issue 3, pp. 30-33. ,March 2016
- (2) U. Kocabicak, M. Firat : "Numerical analysis of wheel cornering fatigue test", Engineering Failure Analysis 8. 2001; pp. 339-345,2000
- (3) Liangmo Wang, Yufa Chen, Chenzhi Wang and Qingzheng Wang : "Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test", Journal of Mechanical Engineering, Vol 57(2011)1, pp.31-39, 2010
- (4) B. Anusha Srikanta and P veeraraju : "Geometrical and Material optimization of alloy wheel for four wheeler", International Journal of Research and Innovation(IJRI), Vol. 1, pp.1401-1402, 2014
- (5) Yeh Liang Hsu, Ming Sho Hsu : "Weight reduction of aluminum disc wheels under fatigue constraints using a sequential neural network approximation method", Computers in Industry, Vol. 46(2001), pp. 167-179, 2001
- (6) JWL Informal Document : "Testing Conditions for Light Alloy Road Wheels for Passenger Car", Regulation No. 1,2010
- (7) J.A. Bannantine, J.J. Comer, J.L. Handrock : "Fundamentals of Metal Fatigue Analysis", Prentice-Hall, Englewood cliffs, NJ, 1990
- (8) Xiaolin Chen, Yijun Liu : "Finite Element Modeling and Simulation with Ansys Workbench", CRC press, Taylor and Francis Group, NW, 2014
- (9) Douglas C. Montgomery : "Design and Analysis of Experiment", John wiley and sons, Inc., Arizoza,2013
- (10) M. Ahmadi, F. Vahabzadeh, B. Bonakdarpour, E. Mofarrah, M. Mehranian : "Application of the central composite design and response surface methodology to the advanced treatment of olive oil processing wastewater using Fenton's peroxidation", Journal of Hazardous Materials, B123, pp. 187-195, 2005