Simulation Environment for Fatigue Analysis of Car Wheel Rim under Radial Fatigue Test

Mihail Grovu[1], Cristi Irimia[1], Ioan Calin Rosca[2], Calin Husar[1], and Maria Violeta Guiman[2]

[1] Siemens Industry Software, Brasov, Romania
{mihail.grovu.ext,cristi.irimia,calin.husar}@siemens.com
[2] Transilvania University, Brasov, Romania
{icrosca,violeta.guiman}@unitbv.ro

Abstract. The importance of the wheels for the car is clear. The wheels with mounted tyres have to carry the vehicle weight and to handle with the steering system, providing the cushioning effect. The automobile wheel must be strong enough to perform the above functions supporting static and dynamic loads. Durability design and analysis is an essential element in achieving the mentioned requirements and involves a multi-disciplinary approach. If it is designed a new wheel rim, the prototype should be passing the radial and cornering fatigue test. This paper presents the simulation environment for car wheel rim under radial fatigue test with the purpose to evaluate the wheel rim strength. This paper reviews, also, some modern durability philosophies and the computer based tools available to meet the needs of design, test and development engineers.

Keywords: steel wheel rim · fatigue analysis · strain life · static analysis

1 Introduction

The wheel rim is a transfer element between the tire and the vehicle. The main requirements of an automobile wheel rim are the following: a) it should be as light as possible so that unsprung weight is least; b) it should be strong enough to perform the carriage function of the vehicle weight supporting static and dynamic loads; c) it should be possible to remove or mount the wheel rim easily. These wheel rims are the most important for cars to run. They must be very safe because they are related to life. Due to the continued rotational motions, the wheel rims of automobiles receive repetitive stresses. If this period becomes long-term, a fatigue fracture can take place [1,2].

Steel and light alloy are the foremost materials used in a wheel rim, however some composite materials together with glass-fiber are being used for special wheel rims. Stamped rims are made of steel, a light constructional material, which is pressed and then welded. They are also prepared because steel has poor corrosion resistance. Finally, steel wheel rims are usually coated with enamel or varnish. The main advantages of steel wheel rims are: a lower price than alloy wheels; flexibility, especially when

International Congress of Automotive and Transport Engineering, CONAT 2016
Transilvania University of Brasov, 2016
ISSN 2069-0401

running into a hole in the road, or when running over objects lying on the road; they absorb shocks and protect the car's body from deformations, which is very important for driving safety. The main disadvantage of steel wheel rims is their weight.

The main objective of this article is to evaluate the durability of a steel wheel rim of the car under radial fatigue test using finite element method [3,4]. Fatigue analysis is preceded by finite element model validation using theoretical and experimental modal analysis. This changeable load is applied on the wheel rim structure considering the initial stress field produced by the tire pressure and the inertial forces generated by the rotation of the wheel.

2 Theoretical and Experimental Modal Analysis

In the automotive industry, experimental modal analysis is used in a new vehicle development to realize different components and assemblies for road dynamic loads. Testing is important to cover some aspects in simulation process using finite element method. Theoretical and experimental modal analysis of the wheel rim implies the following steps: construction a finite element model of the rim to deduce modal parameters, experimentally determination of the modal parameters of the wheel rim under test, correlation of experimental measurement with simulation results [5,6]. The simulation and experimental studies are carried out with free-free boundary conditions.

The 3D model of the wheel rim was created using CATIA and the file was exported in STEP format into ANSYS program. The mesh is made with 10-node tetrahedral structural solid elements using an element edge length of 5 mm. The total number of nodes and elements is 196894 and 197083 respectively. The wheel rim material is Dual Phase Steel DP600-HR with elastic modulus E=2e11 Pa, density ρ=7850 kg/m3, Poisson's ratio υ=0.3, tensile yield strength YS=4.38e8 Pa and tensile ultimate strength UTS=6.16e8 Pa.

The experimental modal tests and analyses are carried out using vibration measurement and modal analysis software PULSE 12 (Brüel & Kjær). The test procedure comprises data acquisition from impulse hammer, curve fitting of measured frequency response functions for modal parameter estimation. The test bench is shown in Fig. 1 and the accelerometer positions are presented in Fig. 2.

Fig. 1. PULS 12 (B& K) test bench

Fig. 2. Accelerometer positions on wheel

Four theoretical modes (1, 3, 7 and 16) are shown in Fig. 3, Fig. 4, Fig. 5 and Fig. 6. Also, the natural frequencies obtained by test are specified for every displayed mode.

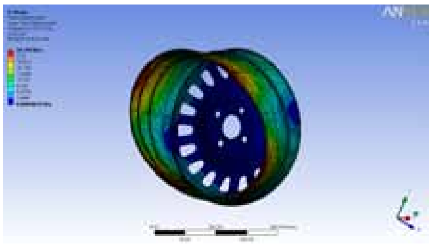

Fig. 3. Mode 1 - 159 Hz (test: 178 Hz)

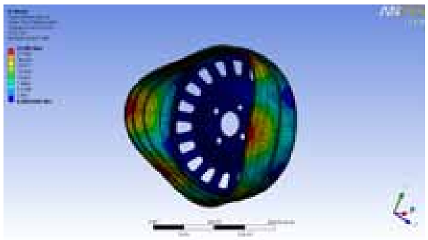

Fig. 4. Mode 3 - 421 Hz (test: 422 Hz)

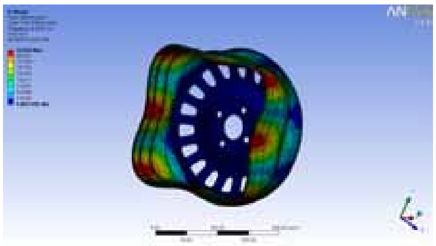

Fig. 5. Mode 7 - 814 Hz (test: 814 Hz)

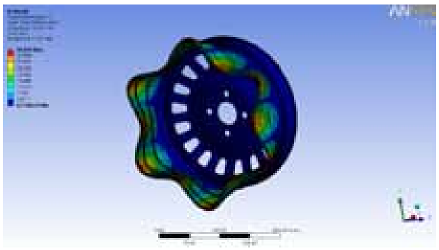

Fig. 6. Mode 16 - 1619 Hz (test: 1650 Hz)

The four accelerometers (type 4507 B) are mounted in four points located at 90° to each other. With the modal hammer (type 8206-003) kicks are applied in points located between accelerometers. The force generated by the hammer was considered as input quantity and as response was selected the speed vibration so the recorded accelerations had to be integrated. Some of the obtained results, natural frequencies and viscous damping coefficient, using information in measurement point, are shown in Table 1.

Table 1. Some natural frequencies and viscous damping coefficients obtained by test

Quantity	0°	90°	180°	270°
f [Hz]	178	178	178	178
ζ [%]	0.0132	0.0131	0.0131	0.0131
f [Hz]	422	426	422	426
ζ [%]	0,00448	0,00511	0,00591	0,00402
f [Hz]	814	814	814	814
ζ [%]	0,00061	0,00067	0,00066	0,00076

Performing experimental modal analysis helped in certifying the subsequently simulation results. The validated FE model will be used now to evaluate the fatigue life of steel wheel rim under radial fatigue test.

3 Analysis of the Steel Wheel Rim under Radial Fatigue Test

3.1 Fatigue Analysis Using FEA Platform

The finite element is a mathematical numerical method for solving ordinary and partial differential equations having the capability to solve complex problems represented in differential equation form. Classical methods alone usually cannot provide adequate information to determine the safe working limits. At the moment the finite element method is used almost to solve structural problems in engineering [7].

Fatigue is an important problem for components subjected to repeated loadings and it is a difficult design challenge to resolve. Practice has shown that large percentage of structural failure is attributed to fatigue and it continues to be the focus of fundamental and applied research. Variable loadings applied on a component or structure at stresses allowable for static loadings may cause the appearance of one crack. Under cyclic loading these cracks may continue to grow and can generate a failure when the remaining structure can no longer carry the loads.

A methodology to predict crack initiation in the fatigue damage assessment of metallic structures typically used in ground vehicle industry is presented in this paper. The finite element model is integrated with a notch stress–strain analysis method. Local loads are modeled with linear elastic FE analyses considering the radial load as being zero-based fatigue loads. The computed stress–strain response is used to predict the fatigue crack initiation life using effective strain range parameters.

3.2 Wheel Rim Model

Fig. 7 and Fig. 8 show the CAD model of the wheel rim and FE model respectively.

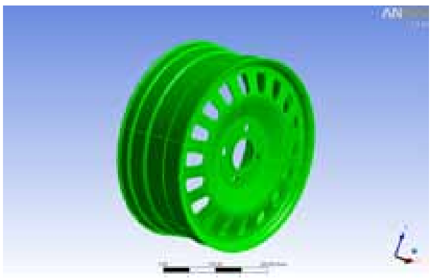

Fig. 7. CAD model of the wheel rim

Fig. 8. FE model of the wheel rim

The meshing was performed using the mesh generate options in the ANSYS Workbench.

3.3 Boundary Conditions and Loadings

The FE wheel rim model is fixed around the holes through which the screws pass to clamp the wheel rim on the disk spindle. The tire characteristics are 195/65R15. The rolling diameter is 634.5 mm. It is assumed that the car speed is 80 km/h which corresponds to an angular speed of the wheel of 75.26 rad/sec. The tire pressure is 30 psi (0.207 N / mm^2) and it is applied on the outer surface of the wheel rim. The total weight of the car is 1600 kg and the radial reaction force on the wheel rim is 3924 N, applied on the surfaces corresponding to the tire bead seats. Fig. 9 shows the following boundary conditions and loads: fixed surface around the clamped holes, inertial load and tire pressure. Fig 10 presents the way considered to apply the radial load.

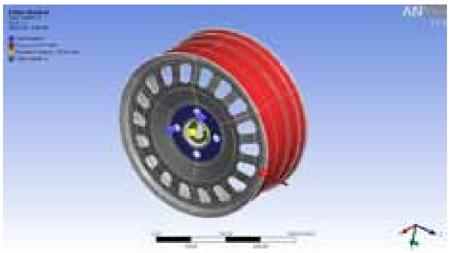

Fig. 9. Boundary conditions

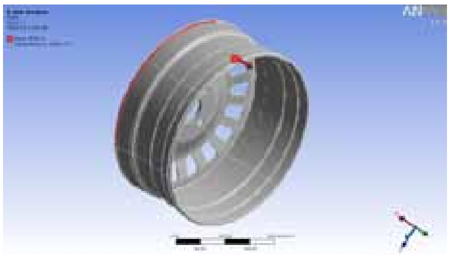

Fig. 10. Application of the radial force

3.4 Fatigue Analysis Process and Results

The total weight is distributed on all four wheels and it is balanced with vertical reaction forces from the road through the tires. These loads constantly compress the wheel rims in radial direction. The radial load becomes a variable load with the rotation of the wheel while the car is running. The evaluation of wheel rim fatigue strength under cyclic radial load is an important performance characteristic for structural integrity.

Fatigue life prediction using the stress approach is mostly based on local stress, because it is not possible to determine nominal stress for the individual critical areas. Fatigue analysis is done in two stages. In the first step, the static elastic tensions are evaluated. These are generated by the simultaneous action of inertial forces generated by the rotation of the wheel and the tire pressure. Due to the action mode of the mentioned loads, the maximum von Mises stress is lower when the two loads simultaneously act. In the next step, a cyclic radial load is applied, generated by the rotation of the wheel.

In this paper it is analyzed the car wheel rim fatigue under cyclic radial load using the local strain-life method. Unlike the stress-based fatigue life prediction, in which only elastic stresses and strains are presented, the strain-life method takes into consideration the local cyclic plastic deformations. Most components may appear to have nominally cyclic elastic stresses, but notches, welds or other stress concentrations may present local plastic deformations. The wheel rims are made of steel plate which is in the first time stamped (pressed) and then the resulted components are welded.

The relation of the total strain amplitude ε_a, total elastic amplitude e_a^e, total plastic amplitude e_a^p and the fatigue life in reversals to failure ($2N_f$) is expressed in the subsequent form:

$$\varepsilon_a = \varepsilon_a^e + \varepsilon_a^p = \frac{\sigma_f^{'}}{E}(2N_f)^b + \varepsilon_f^{'}(2N_f)^c \qquad (1)$$

The strain-life data are the following: $\sigma_f^{'}$ - fatigue strength coefficient, b - fatigue strength exponent, $\varepsilon_f^{'}$ - fatigue ductility coefficient, c - fatigue ductility exponent. Schematic of a total strain-life curve is shown in Fig. 11.

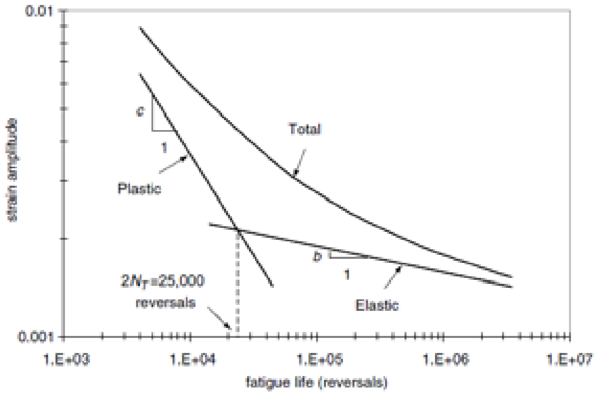

Fig. 11. Schematic of a total strain-life curve

The strain-life parameters for the car wheel rim material, Dual Phase Steel DP600-HR are shown in Table 2.

Table 2. Strain-life parameters for Dual Phase Steel DP600-HR

	A	B	C	D	E	F
1	Strength Coefficient (Pa)	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient (Pa)	Cyclic Strain Hardening Exponent
2	1.007E+09	-0.087	2.441	-0.757	9.04E+08	0.114

Fatigue analysis is done using Fatigue Tool of the Ansys Workbench 14.0 platform. Over the static loading represented by the inertial forces and the tire pressure it overlaps the pulsating radial force. This operation is achieved using Solution Combination facility.

Fig. 12 presents the equivalent von Mises stresses, maximum and minimum principal stresses and maximum shear stresses in the wheel rim. The extreme stress values are in the region of one ventilation holes.

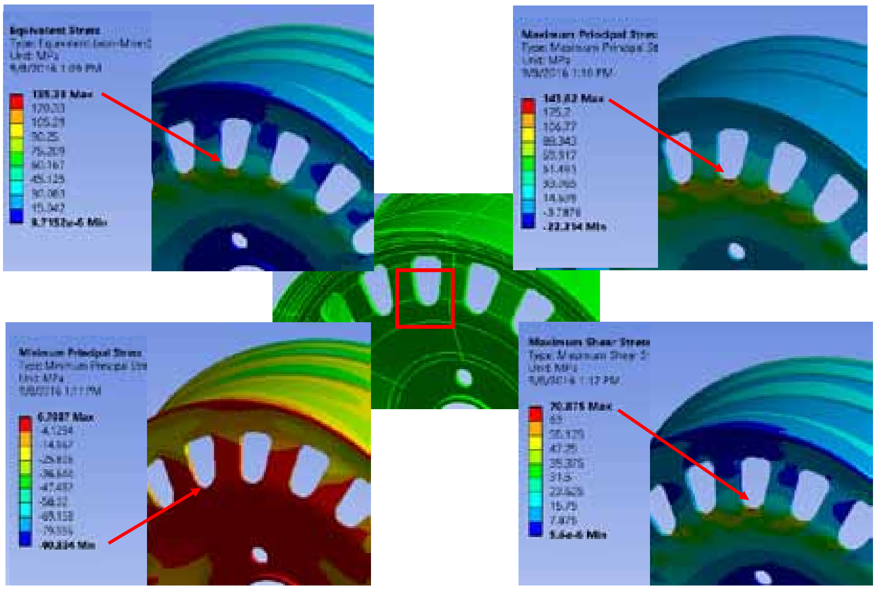

Fig. 12. Equivalent, Maximum and Minimum Principal and Maximum Shear Stresses

Details of Fatigue Tool are the following: Analyze Type → Strain Life, Mean Stress Theory → Morrow, Stress Component → Max Principal, Infinite Life → **1e9** cycles. It is considered that 1 cycle corresponds to 1 rotation of the wheel. The fatigue analysis result is shown in Fig. 13, the minimum life being **8.83e8** cycles in the region of one ventilation holes.

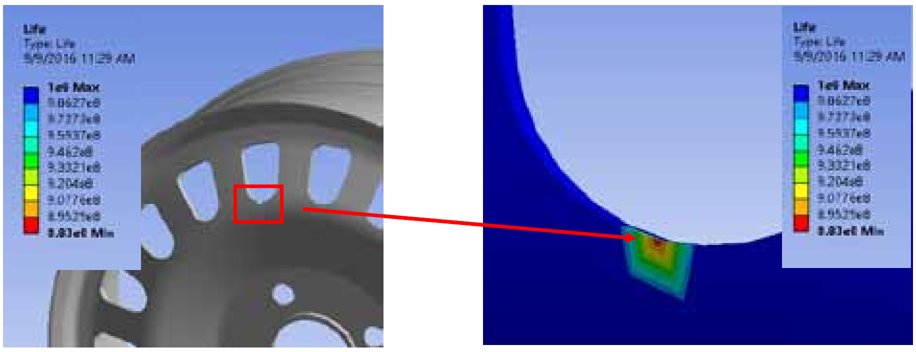

Fig. 13. Fatigue analysis results

During the static analysis of the steel wheel rim, it was discovered that the maximum stress concentration occurred in the region of one ventilation hole. It is observed that when the wheel rotates at a speed of 718.7 rpm with tire pressure of 30 psi (0.207 N / mm2) and under the radial load of 3924 N, the fatigue crack begin to propagate at the point of maximum stress concentration.

The fatigue material properties are based on uniaxial stresses but in the reality stress fields are usually multiaxial. Biaxiality indication is defined as the ratio between the principal stress smaller in magnitude and the larger principal stress. The principal stress nearest zero is ignored. A biaxiality of zero corresponds to uniaxial stress, a value of –1 corresponds to pure shear, and a value of 1 corresponds to a pure biaxial state. Fig. 14 shows biaxiality indication under mentioned loads.

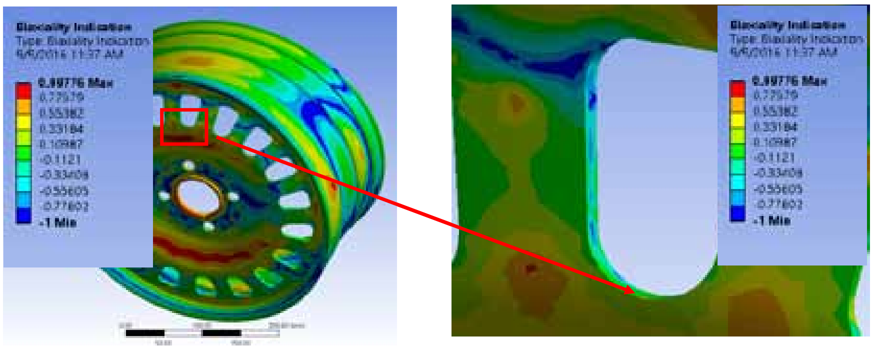

Fig. 14. Biaxiality indication

As it can be seen, the preponderance of this model is under a pure uniaxial stress. The most damaged point occurs at a point of predominantly uniaxial stress. It results as fair to use the steel properties obtained by uniaxial fatigue testing of a DP600-HR specimen.

Fatigue Sensitivity shows how the fatigue results change as a function of the loading at the critical location on the model, Fig. 15.

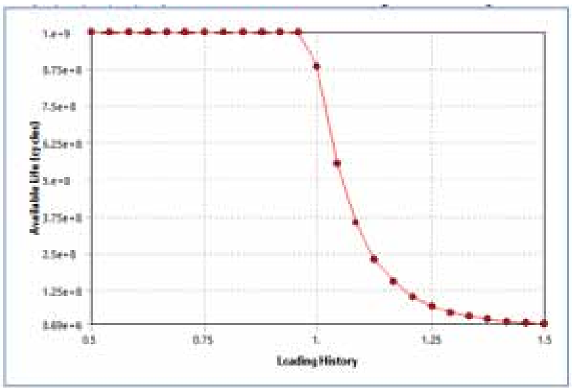

Fig. 15. Fatigue sensitivity

In this paper the sensitivity is determined for the life if the FE load was 50% of the current load up to 150% of the current load. A value of 100% corresponds to the life at the current loading of the model. The fatigue life for current load is **8.83e8** cycles, and decreases to **8.69e6** cycles (10161%), if the load increases to 150%.

4 Conclusions

The radial fatigue test is intended to find the structural performance of a wheel rim for normal use on passenger cars, light trucks and multipurpose vehicles. The wheel rim has to withstand repeated radial loading for a defined number of cycles in order to pass the test.

This paper presents a computational methodology for fatigue estimation of the wheel rim subjected to the repeated radial loading. In the first step it is made an experimental modal analysis to validate FE model and to confirm the subsequently simulation results. The validated FE model will be used now to evaluate the fatigue life.

The fatigue life of a wheel rim is calculated from the stress values obtained from static analysis, based on the local strain approach in conjunction with linear elastic FE stress analyses. The stress–strain response at a material point is computed with a cyclic plasticity model coupled with a notch stress–strain approximation scheme.

The predicted failure locations are in the same sections to the real crack initiation regions and are reliable with other reported analysis.

References

1. Janardhan, J., Ravi Kumar, V., Lalitha Narayana, R.: Radial Fatigue Analysis of An Alloy Wheel, Int. Journal of Engineering Research and Applications, Vol. 4, Issue 12(Part 6), December 2014, pp.253-258.
2. Kim, K. S., Choi, D. S., and Cho, J. U.: A Study on Structural Fatigue Durability of Automotive Wheel at Driving, Indian Journal of Science and Technology, Vol 8(15), IPL031, July 2015.
3. Burande, D.H., Kazi, T.N.: Fatigue Analysis of Alloy Wheel for Passenger Car under Radial Load, International Journal of Engineering Research and General Science Volume 4, Issue 2, March-April, 2016.
4. Venkateswara, R. K., Dharmaraju, T.: Analysis of Wheel Rim Using Finite Element Method, International Journal of Engineering Research & Technology (IJERT) Vol. 3, Issue 1, January, 2014.
5. Brincker, R., Kirkegaard, P. H., Editors, Special Issue: Operational Modal Analysis, Mechanical Systems and Signal Processing, 24, 1209-1323, 2010.
6. Hermans, L., Van der Auweraer, H.: Modal testing and analysis of structures under operational conditions: industrial applications, Mechanical Systems and Signal Processing, 13, 193-216, 1999.
7. Jozsa, I.R., Preda, I., Grovu, M.: Studiu de proiectare pentru un mecanism de direcţie cu pinion şi cremalieră (Design study of a car rack-and-pinion steering mechanism), International Conference COMEC 2009, Brasov, Romania, 2009.