17th International Conference
ENGINEERING MECHANICS 2011
Svratka, Czech Republic, 9 – 12 May 2011

IMPACT BEHAVIOUR OF A NEWLY DESIGNED RAILROAD WHEEL

P. Navrátil*, P. Janíček*, L. Brabenec*, P. Marcián*, M. Matug*, A. Civín*

Abstract: *The FEM study presented in this paper deals with a continuum mechanics approach to determination of the stress-strain behaviour in a newly designed shape of railroad wheel for different ways of ride. The reliability of the railroad wheel is related to achieved failures of material. This paper emphasises on the failures of material that are related to the first plastic deformation of the wheel material. This study focuses on designing the new shape of disc with a usage of special algorithm written for the ANSYS software and to determine an influence of stress waves in the material of the railroad wheel .This analysis gives answer to a rail-railroad wheel interaction on impact behaviour of a railroad wheel and a newly designed disc shape in different ways of ride. It shows the weak spots of the designed wheel.*

Keywords: ***Railroad wheel, stress-waves, finite element method, transient analysis.***

1. Introduction

Computational simulations of stress waves belong to the most often solved problems in the applied mechanics. These transient simulations are very difficult to solve, because they need a very fine mesh which means that the time of transient simulation is very long. Our task was to solve a newly designed railroad wheel (further referred to as: RW) in terms of stress-strain states. The design and the shape of the RW (Navrátil et al, 2009) based on mechanical compliance analysis. For this case new algorithm was written for usage in the APDL. For simulation of stress waves we can use 2D or full 3D geometry. Using 2D geometry is very inaccurate; therefore, to get the right results it is necessary to use full 3D geometry of the RW. The cross-section shaped RW is irregular for usage of 2D simulation; thus, it is not suitable to use 2D simulation for the real wheel. The stress results obtained from the analysis are compared with the possible value of yield strength (Dozela et. al, 2010) and the critical spot on the wheel will be discussed further.

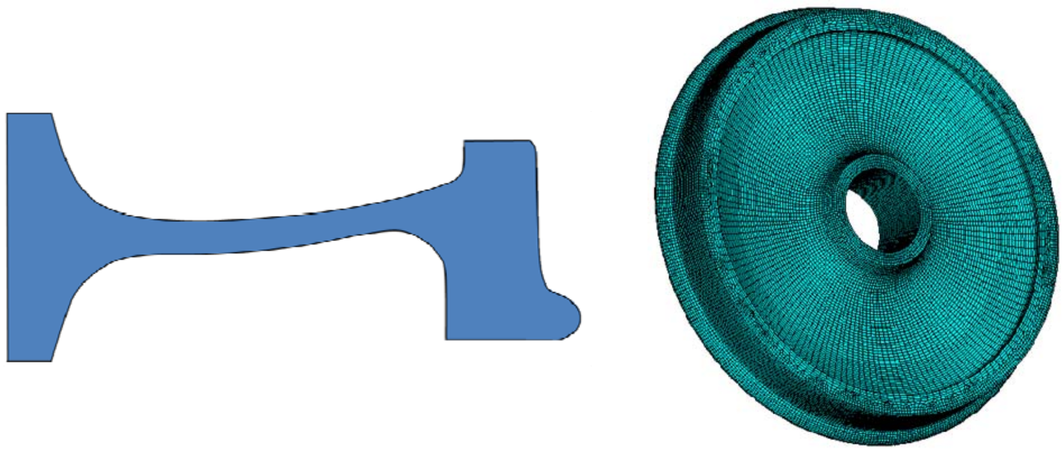
2. Methods

The finite element method was used for the computational simulation as it is the most common numeric method used. For the analysis we used the ANSYS 11.0 software (Ansys Inc., Canonsburg, PA, USA). The model is compared from different points of view: the model of geometry, model of materials, model of boundary conditions and loads.

2.1. Model of geometry

The model of the wagon RW was gained from the shape optimization routine by using the FEM simulations. We assumed that the value of the mechanical compliance in normal direction would be high and the value of mechanical compliance in axial direction would be low. Both presumptions come out from the last analysis (Navrátil et. al, 2009). For obtaining the new shape the cosine function and displacement analyses were used. A large set of displacement analyses give values to axial and normal mechanical compliance. From these results the maximum ratio of axial to normal mechanical compliance was obtained. We can observe the final shape of wheel and mesh in Fig. 1. The rail was assumed to have 1:40 cant.

* Ing. Petr Navrátil, Prof. Ing. Přemysl Janíček, DrSc., Bc. Ladislav Brabenec, Ing. Petr Marcián, Ing. Michal Matug and Ing. Adam Civín: Institute of Solid Mechanics, Mechatronics and Biomechanics, Faculty of Mechanical Engineering, Technická 2896/2; 616 69, Brno; Czech Republic, e-mails: ynavra26@stud.fme.vutbr.cz, janicek@fme.vutbr.cz, ybrabe04@stud.fme.vutbr.cz, ymarci00@stud.fme.vutbr.cz, ymatug01@stud.fme.vutbr.cz, ycivin01@stud.fme.vutbr.cz



a) A cross-section of the new RW *b) A meshed the RW*

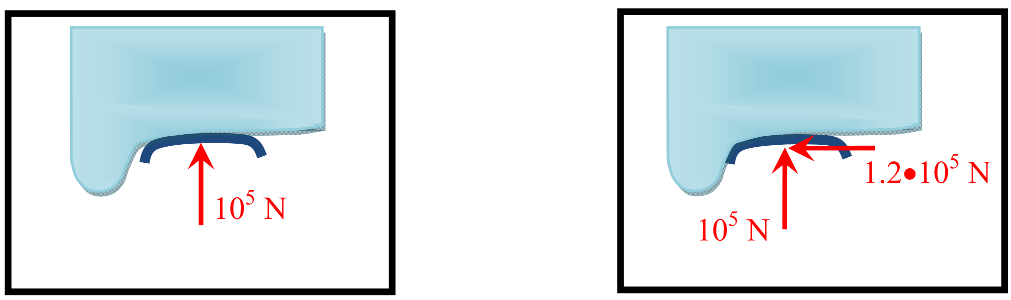
Fig. 1: A geometry and mesh of the RW.

2.2. Model of material

The material of the wheel was assumed to be homogeneous isotropic and linear elastic. We used a material characteristics Young's modulus E = 210 000 MPa Poisson's ratio ν = 0.3 and yield strength Re = 420 MPa corresponding with R7T steel (Dozela et. al, 2010).

2.3. Model of boundary conditions and loads

In case of a stress waves, the load was realized as you can see in Fig. 2 – see below. The RW was loaded in a two ways. For the case of straight ride we assumed 10 tons load to one railroad wheel, i.e. 10^5 N value of normal force loading. In case of a ride in an arc the normal load is the same as it is in the case of the normal ride. The second load in the case of the ride in the arc is loaded in axial direction of RW, with force value $1.2 \bullet 10^5$ N. The boundary conditions were set only in the RW hub and all domains of freedom were set to zero value.



a) A straight ride *b) Ride to arc*

Fig. 2: A load modes of the RW.

3. Results

Analyses in FEM simulation program ANSYS were created. The first step of simulations was to design the new RW. The ratios of mechanical compliances were compared. You can observe the best results in figure 1a), which has a ratio of mechanical compliances of 57.944 (the maximum ratio) in normal and axial direction. Secondly, we used computational simulation for solving stress waves of the RW. In the first step the case of straight ride was solved. An achieved Von Mises results can be seen in Fig. 3, which shows one of the critical state.

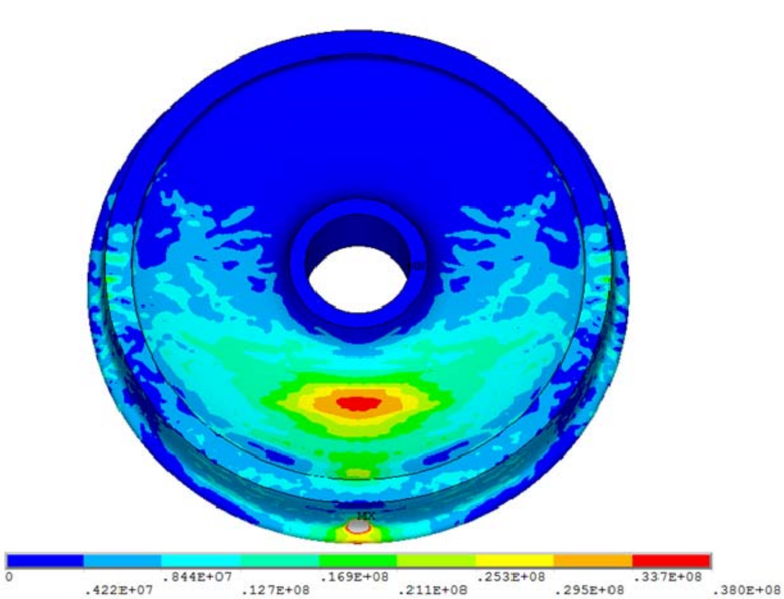


Fig. 3: The stress waves for the case of straight ride – Von Mises.

For this operating state can be said that it you should not overstep the value of yield strength of 420 MPa, because maximum value is 47.5 MPa. That means, in this case the plastic deformation will not occur. The second case was the ride in the arc. As we can see in Fig. 4, the Von Mises results were plotted, as in the previous case.

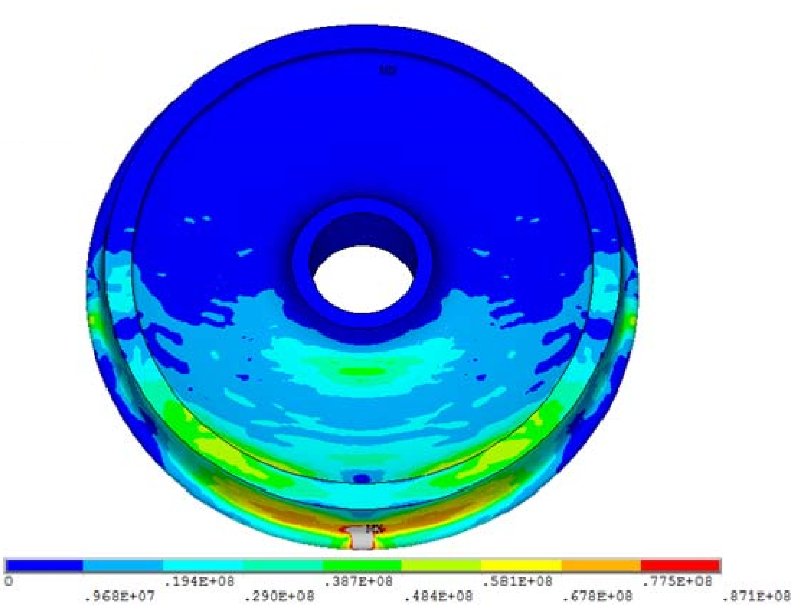


Fig. 4: The stress waves for the case of ride to arc – Von Mises.

In case of the ride in the arc a value of yield strength of 420 MPa for steel R7T cannot be overstepped, because the maximum value is 87 MPa. That means, in this case the plastic deformation will not occur. The stress wave analysis proved that the RW has some critical spots. The first critical spot is between the wheel disc and the wheel rim. Other critical spot is in the middle of the wheel disc. We can see both spots in Figs. 3 and 4. All figures are assessed in terms of stress-waves in the part of the wheel disc. The wheel hub doesn't contact load in the wheel rim. From the analyses ensue that stress in the wheel-rail contact spot exceed the value of yield strength. The average of the equivalent stress value in wheel-rail contact spot is near 880 MPa but in term of these analyses it is not essential.

4. Conclusions

The new shape of the railroad wheel is shown in this paper. The aim of this paper was to determine impact behaviour of the new shaped RW for different ways of the ride. The transient analyses of stress waves for two different cases of the ride were done. From these analyses the stress distributions were obtained. We can say that the stress fields stemmed by stress waves causes less value of stress than the value of the yield strength of R7T steel; however, as the analysis proved, the stress distribution on contact surface oversteps value that causes permanent plastic strain on rail-wheel contact spot. In conclusion, we can say that stress waves in the new designed type of the RW do not exceed a value of yield strength of R7T steel and maximum values of stress are much smaller than yield strength.

Acknowledgement

The authors are thankful for great support by the specific research FSI-S-11-11/1190 and specific research FSI-J-11-18.

References

Donzella, G., Faccoli, M., Mazzù, A., Perogalli, C., Roberti, R. (2010) Progressive damage assessment in the near-surface layer of railway wheel-rail couple under cyclic contact, Wear, 9 pages, in press.

Navrátil, P., Janíček, P., Brabenec, L., Matug, M., Marcián, P. (2009) A design influence of mechanical compliance railroad wheel to fracture behaviour, Proceed. of conference Computational Mechanics, Pilsen, 2 p.