Dubey et al. 2018. *Int. J. Vehicle Structures & Systems*, 10(3), 220-222
ISSN: 0975-3060 (Print), 0975-3540 (Online)
doi: 10.4273/ijvss.10.3.13

International Journal of
Vehicle Structures & Systems
Available online at www.maftree.org/eja

Finite Element Simulation of Wheel-Rail Interaction: Technical Note

Shiv Prakash Dubey[a], Satish C. Sharma and Suraj P. Harsha

Mech. & Industrial Dept., Indian Institute of Tech. Roorkee, Uttarakhand, India
[a]Corresponding Author, Email: dubeyshiviitr@gmail.com

ABSTRACT:

This paper deals with quasi-static analysis of wheel-rail interaction. The model has been developed for analysing the contact patches behaviour, pressure distribution, von mises stress and strain. A solid model has been developed using SOLIDWORKS on the basis of UIC-60 rail profile and S-1002 wheel profile. Finite element analysis of the solid model has been done using ANSYS software. It has been observed that wheel-rail interaction is nonlinear and exceeded the yield strength of wheel material. The analysis of the worn thread of wheel has enabled the identification of the contact patches and critical sections of the wheel-rail interface.

KEYWORDS:

Finite element analysis; Wheel-rail interaction; Contact patches; Contact stress; ANSYS

CITATION:

S.P. Dubey, S.C. Sharma and S.P. Harsha. 2018. Finite Element Simulation of Wheel-Rail Interaction: Technical Note, *Int. J. Vehicle Structures & Systems*, 10(3), 220-222. doi:10.4273/ijvss.10.3.13.

1. Introduction

For the study of the wheel-rail contact region, the stress, contact pressure and deformation in a quasi-static condition of the wheel-set are essential. Railway engineers and researchers in this field used Finite Element Analysis (FEA) and created contact formulations for the wheel-rail interaction and matched their results with tests or previous literature. Numerical analysis was performed before 1970 and study of contact patches was performed by mathematical modelling after 1970. British Rail has developed Medyna software. Modern multibody software packages e.g. GENSYS [1] are used as an essential part of the design process for new vehicles and for investigating the service problems with existing vehicles [2-3]. Iwnicki [4] developed several computation techniques, investigated vehicle dynamic behaviour and validated his approach through the experiments. The results from the simulation and experiments not found in exact agreement, however these analyses have become a benchmark for other researchers. Finding out the contact area required face split selection of wheel as well as rail. Wheel rolls over the rail to form contact patches of elliptical in nature. For the prediction of von mises stress, stain, deformation and displacement field, the wheel-rail contact surface modelling is required.

Smith [5] studied normal and tangential load on an elastic solid and found out the stress developed in the contact zone. Haines [6] studied the contact stress distribution in an elliptical shape which was subjected to radial and tangential force. Sackfield [7] applied Hertz contact theory for a number of applications. Sladkowski [8] developed FE software which was used for finding out the contact stress on wheel-rail interaction. Weist [9] computed the normal force distribution for the contact between wheel and rail crossing panel using FEA and his experiment validated the Hertz theory and the Kalker's model. Kalker's linear theory of creep is validated by other researchers [10]. Donzella [11] has given failure diagram which is subjected to rolling. Monfared [12] detailed the crack propagation theory in rotational system and validated the results using FEA. Zwierczyk and Váradi [13] analysed the pressure distribution in elliptical, rectangular, circular surface with help of ANSYS tool. Arslan [14] studied on the fundamental way of handling wheel-rail contact analyses from the FE standpoint and highlighted the required steps for realistic 3D solutions. 3D FE analysis results obtained show good agreement with real life problems experienced.

By applying a gradually increasing point load at the center of the wheel, the resulting damage on the wheel-rail can be assessed. The stress developed in wheel-rail profile is tensile and compressive in nature. Initially a small crack is generated when the stress developed in the contact zone exceeds the critical stress. This small crack will then propagate and lead to fracture and failure of the wheel-set material. The main objective of this paper is to find out the critical stress and contact patches at the point of wheel-rail interaction through FEA for a quasi-static loading condition.

2. Materials and methods

For defining material properties, a solid model of wheel and rail with a specific geometry is developed using SOLIDWORKS. For the geometry, UIC-60 rail profile and S-1002 wheel profile are used. For analysing the contact stress, the face of wheel-rail is divided before meshing using SOLIDWORKS. Tetrahedral mesh is used for wheel-rail structure as shown in Fig. 1. There are 176698 nodes and 112617 elements in the meshed wheel-rail model. A 200 kN force is applied on the center face of the wheel gradually as shown in Fig. 2. The bottom face of the rail is fixed for simulation. The faces of the wheel are clamped in X and Z directions but free in Y direction (i.e. normal to rail). In this paper, the rotational effect on the lateral displacement is ignored. The split faces of the rail & wheel are shown in Fig. 3.

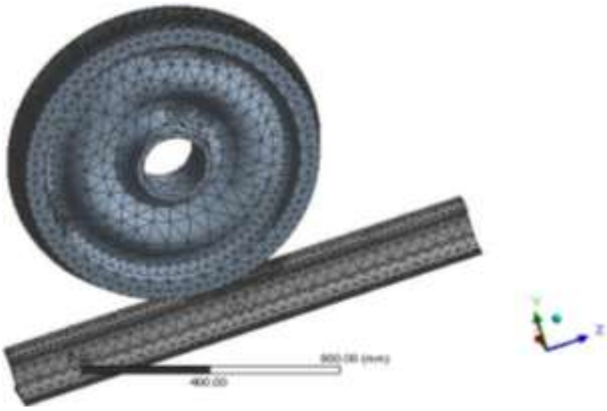

Fig. 1: Finite element model of wheel-rail

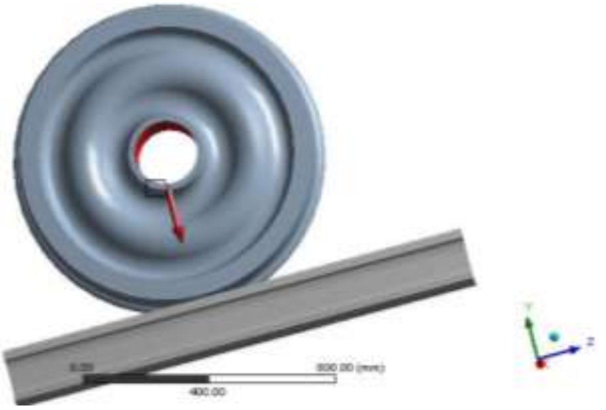

Fig. 2: Application of static load at the wheel center

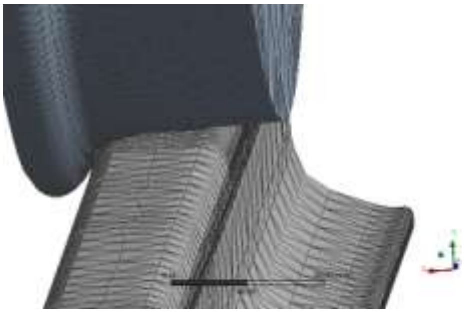

Fig. 3: Split face between wheel-rail interaction

3. Results and discussion

Once the model of wheel rail has been prepared, it is then imported in to ANSYS 17.2 workbench interface and solved. The wheel-rail contact is defined as sliding with a coefficient of friction as 0.15. Contact stress between the wheel-rail interactions is shown in Fig. 4. As shown in Fig. 5, the maximum von mises stress in the wheel is 851.61 MPa which is above its yield stress. As per material used in railway wheel The values of yield and ultimate tensile strength of wheel material are 520 MPa and 760 MPa respectively. As shown in Fig. 6, the maximum stress of 45.05MPa is observed near the thread of wheel and face of the rail. Inner thread of wheel worn out more rapidly than outer thread.

Fig. 4: Contact stress between rail wheel interactions

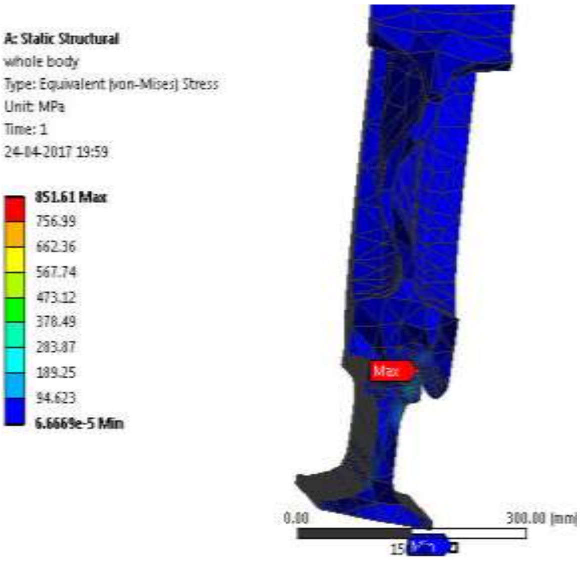

Fig. 5: Maximum stress between flange of wheel and face of rail

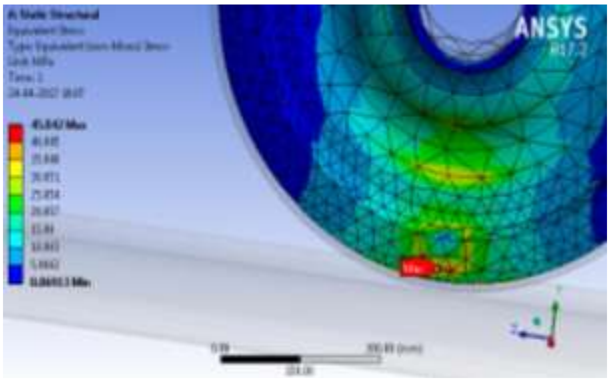

Fig. 6: Maximum stress on outer thread of wheel

Contact patches play important role for transferring load from wagon to rail so it is complex in nature which influence under various static and dynamic conditions. From Fig. 7, it can be seen that there are five different zones on the contact patches i.e. sticking zone, sliding zone, near zone, far zone and over constrained zone. It is observed that the shape of contact patches is elliptical in nature. The maximum pressure is at the centre of ellipse. The minimum pressure is at peripheral. There are two critical sections on wheel-rail interface i.e. wheel flange and rail face & wheel thread and rail face as shown in

Fig. 8. The maximum von mises strain is 3494 micro-strains as shown in Fig. 9. About 0.3% of maximum von mises strain is worn out at the thread of wheel.

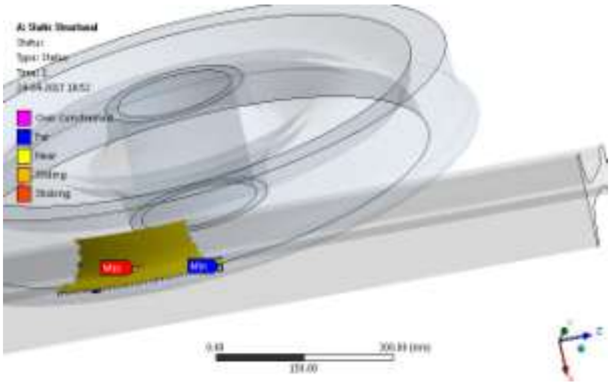

Fig. 7: Contact patches formation during wheel-rail interaction

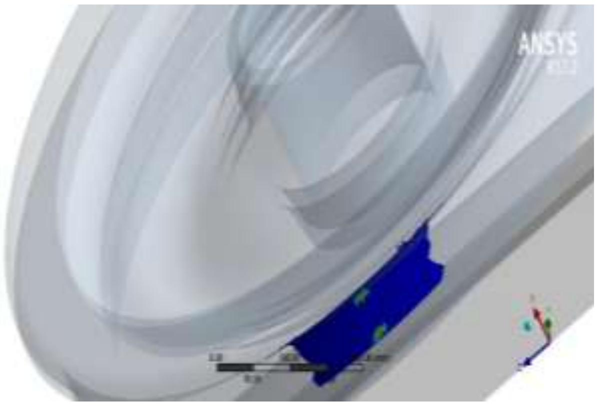

Fig. 8: Critical points of wheel-rail contact under static condition

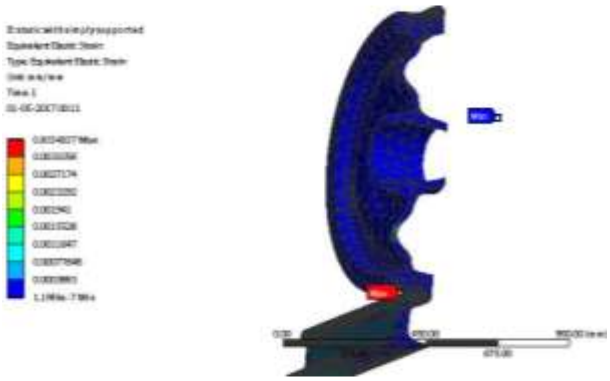

Fig. 9: Von-mises plastic strain

4. Conclusions

In this study, wheel-rail interaction in the context of Indian railway standard under static condition was assessed using FEA. A point load of 200 kN was applied at the centre of the wheel under suitable boundary conditions and wheel-rail contact formulation. It is noticed that wheel-rail interaction is nonlinear. Maximum von mises stress developed at the wheel interface was around 851.61 MPa which is beyond its yield strength 520 MPa. Therefore chances of crack initiation and crack propagation will arise. There were two critical sections of wheel - first one was the flange - rail face and second was the wheel thread - rail face for the observed elliptical in nature of contact patches.

REFERENCES:

[1] *GENSYS Users Manual*, Release 9910.

[2] S.K. Sharma, R.C. Sharma, A. Kumar and S. Palli. 2015. Challenges in rail vehicle-track modeling and simulation, *Int. J. Vehicle Structures & Systems*, 7(1), 1-9. https://doi.org/10.4273/ijvss.7.1.01.

[3] R.C. Sharma. 2012. Recent advances in railway vehicle dynamics, *Int. J. Vehicle Structures & Systems*, 4(2), 52-63. https://doi.org/10.4273/ijvss.4.2.04.

[4] S. Iwnicki. 1998. The Manchester benchmarks for rail vehicle simulation, *Int. J. Vehicle Mechanics & Mobility*, 30(3-4), 295-313.

[5] J.L. Smith. 1953. Stresses due to tangential and normal loads on an elastic solid with application to some contact stress problems, *J. Appl. Mech.*, 157-166.

[6] D.O. Haines. 1963. Contact stress distribution on elliptical contact surfaces subjected to radial and tangential forces, *Proc. IMechE*, 177(4), 45-54.

[7] A. Sackfield and D.A. Hills. 1983. Some useful results in the classical Hertz contact problem, *J. Strain Analysis for Engg. Design*, 18(2), 101-105.

[8] A.S. Sladkowski. 2005. Analysis of wheel-rail interaction using FE software, *Wear*, 258, 1217-1223. https://doi.org/10.1016/j.wear.2004.03.032.

[9] M. Wiest, E. Kassa, W. Daves, J.C.O. Nielsen and H. Ossberger. 2008. Assessment of methods for calculating contact pressure in wheel-rail/switch contact, *Wear*, 265, 1439-45. https://doi.org/10.1016/j.wear.2008.02.039.

[10] R.C. Sharma and S. Palli. 2016, Analysis of creep force and its sensitivity on stability and vertical-lateral ride for railway vehicle, *Int. J. Vehicle Noise & Vibration*, 12(1), 60-76. https://doi.org/10.1504/IJVNV.2016.077474.

[11] G.P. Donzella. 2010. A failure assessment diagram for components subjected to rolling contact loading, *Int. J. Fatigue*, 256-268. https://doi.org/10.1016/j.ijfatigue.2009.06.016.

[12] V. Monfared. 2011. A new analytical formulation for contact stress and prediction of the crack propagation path in rolling bodies and comparing with finite element model (FEM) results statically, *Int. J. Phys. Sci.*, 6(15), 3613-3618.

[13] P.T. Zwierczyk and K. Váradi. 2014. Frictional contact FE analysis in a railway wheel-rail contact, *Periodica Polytechnica*, 58(2), 93-99. https://doi.org/10.3311/PPme.7229.

[14] O.K.M.A. Arslan. 2012. 3-D Railway wheel contact using FEA, *Advance Engg. Software*, 325-331.