Three Dimensional Finite Element Analysis on Railway Rail

M A Zulkifli1, K S Basaruddin1, Y Abdul Rahim1, M Afendi1, P Gurubaran1, I Ibrahim1

1School of Mechatronic Engineering, Universiti Malaysia Perlis, Pauh Putra Campus, 02600 Arau, Perlis, Malaysia.
2School of Manufacturing Engineering, Universiti Malaysia Perlis, Pauh Putra Campus, 02600 Arau, Perlis, Malaysia.

*Corresponding author: khsalleh@unimap.edu.my

Abstract: A prediction of fracture risk on railway rail using finite element analysis (FEA) was carried out in this paper. A three dimensional model was developed using CATIA to investigate the mechanical stress distribution under various loadings. The analysis was performed using ANSYS software to determine the stress response under loading of rolling contact, braking and cornering. In this study, the interaction between rail and wheel at some point was analyzed under static loading. The fracture risk via safety factor was then calculated based on the fracture strength of the wheel's material. The results showed that the braking load condition contributes to the lowest possibility of fracture on the surface of rail compared to the rolling contact and cornering loads.

1. Introduction
Railroads has a critical influence on the transportation framework of a nation and was assumed as an imperative part in managing a solid economy. Thus, railroad is adapted to redesign and overhaul its framework to take care of future demand of developing activity. The rolling contact between wheel and rail can be influenced by certain factors such as train speed, loading, wheel/rail profiles, and the response of the track parts. Railway track can be divided into several structures that play an important role in ensuring the smooth running of the train. Rail track commonly was classified into two main parts; substructure and superstructure ¹. Railway track structure comprises of rails, rail pads, sleepers, fastenings, ballast, sub-ballast, and subgrade. Therefore, two subsystems of a ballasted track structure can be recognized: the superstructure, made out of rails, sleepers, stabilizer and sub-ballast, and the substructure made out of an arrangement layer and the ground. An investigation on mechanical stress distribution is important in order to predict fracture or safety factor of the railways rail.

Numerous studies ²–⁴ have investigated the mechanical stress distribution on the railways rail. Ertz and Knothe ⁵ reported that the stress distribution is an important factor for the rail-wheel contact interface analysis. It clearly shows that two materials contacting at rolling interface, are highly dependent on the geometry of the contacting surfaces, loading conditions and material properties of the railway system ⁶. Analysis of rail–wheel contact was studied by many researchers in different context such as different rail/wheel geometries to calculate the Hertzian contact behavior and determine the stress distributions at the area of interest. As a consequence, damage mechanisms such as surface cracks, plastic deformation, wear and operating rolling noise were also investigated ⁷. For braking load condition ⁸, most of the previous studies investigated the thermal analysis between wheel and the brake pad. For instance, a modelling thermal effects in the braking systems of railway vehicles was studied by Arslan ⁹. It is necessary to derive an analytical model of a thermal analysis that describes the heating transfer of the heat generated by friction at surfaces which are in contact, between a railway wheel and braking blocks, through the wheel and blocks, as well as heat outflow of the whole braking system due to cooling of the surrounding air ¹⁰. Vo et al. [3] studied a nonlinear cornering model of a bogie. The nonlinearities, such as the ability to model stick-slip (or strictly roll-slip) between a wheel and a rail, are found to have a very significant influence on the development of corrugation in cornering. In particular, the periodic occurrence of complete sliding of the wheels of the leading axle makes possible
corrugation growth associated with a lateral mode of vibration of the track that is not significantly excited until this sliding occurs.

On the other hand, most of the studies on the mechanical stress of railways rail have utilised the finite element method (FEM). Applications of FEM in structural analysis have been used extensively for macro and micro levels. Even the use of FEM was not limited to regular microstructure but also for random microstructures. Hence, this study aims to develop a finite element model of railway rail in order to predict the safety factor under various loading during service. Investigation on mechanical stress distribution was performed under three types of loading which is the rolling contact loading, braking loading and cornering loading.

2. Methodology

2.1 Development of railway rail model

A rail with 1000 mm length was created using CATIA. The assembly model of the railways rail-wheel and its half symmetrical model are shown in Figures 1 and 2, respectively. The three dimensional model was exported to ANSYS workbench for finite element analysis. The rail was made of 900A steel. The present analysis was tested by using a standard of 60 kg/m Rail Profiles-Types UIC 60 which provided by local train company.

![Figure 1: Rail-wheel assembly in CATIA.](image1)

![Figure 2: Cross section of rail-wheel contact in ANSYS.](image2)

2.2. Assign material properties

The information regarding to the railways wheel and rail was provided by the local train company, including the wheel diameter of 965 mm (38 in), the vertical load (maximum design load = 78.48 kN), and the mechanical properties of wheel (yield strength = 438 MPa; Young's modulus = 205 GPa; friction coefficient = 0.3; material density = 7850 kg/m³; rotational velocity = 24.6 rad/s; ultimate tensile strength = 1080 MPa).

2.3. Meshing

The meshed models of both rail and wheel were generated in ANSYS with the default mesh as shown in Figure 3. A selection of meshing size of rail was checked by verifying the quality of mesh which is available in ANSYS software. The contact in mesh between wheel and rail is depicted in Figure 4. A total of 197,530 nodes and 173,200 elements (including structural and contact elements) have been generated in the present models.

2.4. Boundary conditions

The boundary conditions were applied based on real conditions of the railway system during service. Figure 5 shows the boundary conditions that was applied to the rail-wheel in contact surface. In order to investigate the relationship between the variable vertical loading towards the stress response and fracture risk of railway rail, a set of different vertical loads were applied in the finite element analysis.
2.5. Prediction of fracture risk
The maximum equivalent von-misses stress obtained in the analysis was considered so that the risk of fracture on the rail can be predicted. The fracture risk also known as safety factor, was calculated based on eq. (1).

\[
\text{Fracture risk} = \frac{\text{Yield Strength}}{\text{Maximum Equivalent Von-Misses stress}} \quad \text{Eq. (1)}
\]

3. Results and discussion
Finite element models were simulated successfully to visualize stress, deformation and strain levels. Static load analysis was performed in order to predict the fracture risk of railways rail. In Figure 6, distribution of Von Mises stress on the rail was shown. The maximum stress was found at the outer surface of the rail-wheel contact. Figures 7 and 8 show that the maximum of Von Misses stress distribution under rolling contact and cornering load conditions, respectively.
In Figure 6, the graph indicates decrement in fracture risk of rail under rolling contact load when the increase of load was applied. As shown in Figure 10, when the load reached 100 kN, there is possibility of fracture going to happen because the fracture risk below than one. However, the rail was found safe under braking load if the load applied between 20 to 100 kN as shown in Figure 10. In contrary, under cornering load as depicted in Figure 11, the possibility of fracture going to occur on the rail if the load reached at 80 kN.

**Figure 6**: Maximum Von Mises stress distribution under braking load

**Figure 7**: Maximum Von Mises stress distribution under rolling contact load

**Figure 8**: Maximum Von Mises stress distribution under cornering load

**Figure 9**: Fracture risk under rolling contact load

**Figure 10**: Fracture risk under braking load.
Figure 11: Fracture risk under cornering load

4. Conclusion
This study was carried out to investigate the mechanical stress distribution and the fracture risk of the railway rail subjected to various loadings. Variable loads were applied to represent the actual load during service. Three types of load (rolling, brake, and corner) were applied. The results showed that the highest fracture risk was obtained under braking load whereas more critical conditions were found during loads of cornering and rolling.

5. References
[1] Kaewunruen, S. & Remennikov, A. 2008. in New Research on Acoustics 197–220.
[2] Aalami, M. R., Anari, A., Shafighfarad, T. & Talatahari, S. Adv. Mech. Eng. 2013, pp. 1–9.
[3] Ertz, M. & Knothe, K. A. 2002. Wear 253, pp. 498–508.
[4] Vo, K. D., Zhu, H. T., Tieu, A. K. & Kosasih, P. B. 2015. Wear 323, pp. 61–75.
[5] Sladkowsi, A. & Sitarz, M. 2005. Wear 258, pp. 1217–1223.
[6] Arslan, M. A. & Kayabosi, O. 2012. Adv. Eng. Softw. 45, pp. 325–331.
[7] Cai, W., Wen, Z., Jin, X. & Zhai, W. 2007. Eng. Fail. Anal. 14, pp. 1488–1499.
[8] Donzella, G. & Petrogalli, C. 2010. Int. J. Fatigue 32, pp. 256–268.
[9] Halim, S. A., Basaruddin, K. S., Daud, R., Safar, M. J. A. & Rahman, A. S. A. ARPN J. Eng. Appl. Sci. 10, pp. 9353–9358.
[10] Basaruddin, K. S., Kamarrudin, N. S. & Ibrahim, I. 2014. Lat. Am. J. Solids Struct. 11, pp. 755–769.
[11] Yoshiwara, Y., Clanche, M., Basaruddin, K. S., Takano, N. & Nakano, T. 2011. J. Biomech. Sci. Eng. 6, pp. 270–285.
[12] Basaruddin, K., Takano, N., Yoshiwara, Y. & Nakano, T. 2012. Med. Biol. Eng. Comput. 50, 1091–1103.