Numerical Simulation and Parametric Analysis of Fatigue Crack in UIC60 Rail Thermite Welded Joint

Prakash Kumar Sen¹, Mahesh Bhiwapurkar² and S P Harsha³

¹ Mechanical Engineering Department, O.P. Jindal University Raigarh, India
² Mechanical Engineering Department, O.P. Jindal University Raigarh, India
³ Department of Mechanical & Industrial Engineering, IIT Roorkee, India

prakashkumarsen@gmail.com

Abstract. The mechanism of rail-wheel contact is the most essential field of study in railway engineering since it requires extensive application expertise, diagnostic skills, and a trustworthy analysis technique. In this research the fatigue life of a UIC60 rail AT weld under vertical load and its parametric effect has been studied, and for that a three-dimensional elastic-plastic finite element model is created using ANSYS space-claim software, and then finite element method is employed to analyse the full-scale model of wheel-track and weld system with realistic three-dimensional solution. Model assembly components include axle, wheel, and thermite-welded rail. Simulation of contact between wheel and UIC60 rail weld with crack on weld at angles of 30 and 60 degrees with different coefficients of friction between the weld wheel contact and between crack surfaces was carried out under vertical loadings. In general, the Hertz contact theory assumptions are taken into consideration throughout the analysis, and the impacts on fatigue life are given by using damage mechanics method. The results of the wheel/weld fatigue crack analysis have been displayed to demonstrate the influence of different parameters on the fatigue life of cracks. The purpose of this study is to identify and safeguard the rail against failure, as well as to ensure the safety of passengers and to reduce the cost of maintaining the rail system.

Keywords. Railway, fatigue, crack, FEM, ANSYS.

1. Introduction

Rails are now connected together by welding whenever it is possible. Welding is, on average, a less expensive process that generates a higher-quality connection than other methods. Every year, a considerable quantity of AT weldments is installed in rail pathways, applying customary processes and ingredients which are established primarily via field practises over the years, and a very high proportion of them found reliable.

Lengths of standard rails are welded together using flash welding methods to form long sections, which are normally six times in length of a standard rail. These sections are then brought to the construction site and joined together to form a continuous track. Alumino-thermic welding is perhaps the most often used technology for joining rails in the work field, for multiple of motives, like easy to align the tracks and other field parameters, as well as cost and time savings. Aluminium-thermite weld on rails are exactly positioned having joint-gap of approximately 35 mm between the rail ends as a regular practice [1]. For joining through gape, an exclusive ceramic mould is placed around it, which is then packed with cementitious sand, and a combine mix powder having iron oxide and its alloys with aluminium is poured through hopper in it. As the combination of this mixture is burnt, molten metal rushes downward in the
cavity, forming the joint over there. The method is also said as casting joint. After the weld-joint cooled, the mould frame is removed, and any excess metal on joint area is chipped out and by grinding operation the joint has provided surface finish so it can match the shape and profile of the parent rail. The rail-wheel assembly is considered as very complex geometry who rely on friction to convert energy. As a consequence of this rolling contact friction and frictional slip or sticky kind of action a damage mechanism comes in to picture which manifest the structure to initiate wear and growth of crack in it. This phenomenon now becomes a challenge for railway engineers to identify and solve such problems as early as possible as it arises. That’s why it’s essential to understand the fundamentals of contact theory and generation of contact stress, and its distribution to predict the fracture behaviour and propagation. In addition, for above mentioned terms the 3-D stresses parameters and locations are estimated correctly in [2, 3], so the wheel-rail wear/damage can be accurately predicted. In the literature [4-6], for some circumstances, it was difficult to find the direct solutions for stresses on the contact point hence new methods were developed for those cases and presented. These solutions were based on specific elastoplastic material properties where their qualities were improved in physically and do not consider the friction coefficient normally for wheel rail contact. It’s also found that these closed forms solutions are with very basic geometries and full of errors thus rail-wheel contact becomes quite sophisticated.

Many scholars have observed the contact between a rail and a wheel in various contexts and with various rail and wheel geometries in order to find the contact and its behaviour, the generated stress and its distribution. Result shown by the clarifies about surface fatigue fracture and damage mechanism and plastic deformation on structure and other vibrational parameters. The proper study and simulation of these systems allowed us to understand thorough comprehension of the specific knowledge about real time interaction of wheel and the rail. These study have been investigated in the literature [7-11] through the use of some laboratory data, and finite element simulations with analytical assumptions under variety of scenarios [12-15]. This research will not provide any new insight into the literature review or the fundamentals of the area of interest because the objective of this work is to merely illustrate a well-known finite element analysis approach on an assembly of 3-D rail–weld. Following are the steps involved in the 3-D contact study of rail weld–wheel contact that were taken into account in this work, and they are depicted in the Figure 2 below in the following order:

- Rail-weld assembly layout and design with Elastic-plastic material model.
- Fixing the boundary and loading conditions.
- Parameters in which the varied coefficients of friction on contact are taken into consideration.
- Solve the problem and post-process the results

![Figure 1](image1.png)  
**Figure 1.** (a) Rail weld with wheel contact (b) AT weldment with crack [18]

2. FEA model, material and methodology: Pre-Processing
Specifically, a mono-block wheel with a diameter of 1150 mm and the matching rail UIC 60 [16] and with its axle and a connection of realistic AT weld with side embossed and grinded surface have been
modelled in accordance with the Indian Railways criteria indicated in Figure 3(a) for this study. This assembly model is shown in Figure 3(b) after the geometries have been produced in ANSYS space-claim 2020 student edition [17].

![Figure 3. (a) Rail and wheel geometry (b) Rail weld with wheel assembly model and (c) Mesh](image)

The 3-D model assembly was imported into the ANSYS mechanical workbench and then mesh was provided with 8 node hexagonal elements type of solid185, as in Figure 3(c), along with weld zone meshing as shown in Figure 3(d). The frictional surface contact is defined using conta174 type elements, which are on trade surface of wheel, and other surface as the target170 elements, which are placed on the rail and its weld. There were a total of 190572 nodes and 204846 elements in the FEA model mesh for all geometry. The loading conditions are depicted in Figure 4, where a vertical force of 55 kN is applied to the centre of the wheel at its centre of rotation. The rail and weld bottom are anchored at the transverse site in all directions to prevent the body from moving. The effects of the rotating wheel and lateral force were not taken into consideration. The FE model is anticipated to reflect the weld in a more realistic manner by including a 1mm-wide side extruded portion of the weld and the assumption that the head of the weld has been grounded.

### Table 1. Parameters of elastic plastic weld model

| Mechanical property          | Value   |
|------------------------------|---------|
| Poissons Ratio              | 0.3     |
| Young’s Modulus (GPa)        | 207 GPa |
| Ultimate tensile strength (MPa)| 996.7 MPa |
| Yield strength (MPa)         | 675.7 MPa|
| Percentage reduction of area | 4.22    |
| Percentage Elongation        | 3.09    |

### Table 2. CAD parts material properties

| Part name | Modulus of elasticity(GPa) | Modulus of plasticity (GPa) | Yield stress (MPa) | Poisson’s ratio |
|-----------|---------------------------|-----------------------------|--------------------|-----------------|
| Rail      | 206.9                     | 22.7                        | 483                | 0.295           |
| Wheel     | 205                       | 22.7                        | 640                | 0.3             |
| Axle      | 205                       | -                           | -                  | 0.3             |

Both Tables 1 and 2 represent the relevant material attributes that were used in the ANSYS material library’s, material database and then assigned to the CAD assembly model in workbench. Figure 4 depicts the assembly with a view of the AT weld on rail as a joint, where two fractures are postulated for the purpose of analysis under vertical loading, as shown in the assembly. The position of the fracture is thought to be the region where the greatest amount of stress distribution may generate. For the purpose of individual analysis, the orientation of cracks has been considered to be at an angle of 30° and 60°. The coefficients of friction used in this calculation of stress between contacts are 0.25, and the coefficients of friction used in the parametric analysis of crack are 0.1 to 0.4, respectively, with vertical loads of 92, 120, and 140 kN in the case of the contact.
3. Simulation results: Post-Processing

The assembly is applied with boundary conditions with loading and simulated as post-processing, allowing for the visualization of stress distributions. Figure 5 depicts the maximum stress distribution at various points in contact. Figure 5(a) shows the pictorial view of simulation having a maximum Von-Mises stress value as 597 MPa under the frictional coefficient of 0.25 just beneath the head surface about 2.89 mm as expected to be occurred. It may be observed that the induced stress is above the value of yield strength of parent rail material and quite below the yield strength of the rail weld material. For both weld part and contact region stress contour may be seen in Figure 5(b).

![Figure 4](image1.png)

**Figure 4.** (a) AT weldment crack position (b) Rail weld with wheel assembly (c) weld part having crack positions (d) Loading conditions (f) simulation view of wheel contact on weldment

![Figure 5](image2.png)

**Figure 5.** (a) Stress AT weldment contact (b) Stress showing at weld and at contact point

It has been observed in many studies that the majority of the distortion due to plastic deformation happens on the wheel body, and then rail and its weldment experience essentially no specific plastic deformation. According to Figure 5, stress concentration where the greatest equivalent deformable stress value occurs just having same node, is approx. 462 MPa. An examination of the Von-Mises stress contours of the wheel and rail-weld reveals the presence of a fracture that develops over the running cycle exactly in the area depicted in Figure 4(a). The result has been plotted between frictional coefficient and equivalent stress is in Figure 6(f).

For the study of effect of different parameters on weld life due to fatigue is explored using the proposed damage mechanics approach and carried out by numerical simulation, and their plots are drawn. The effective parameters considered here are loads which are vertically applied and coefficients of friction...
variations having two condition first in between the contact surfaces of wheel and weld and second in between the crack surfaces. Here two cracks are named as crack 1 and crack 2 as discussed previously. Now for parametric analysis with varying loading conditions i.e. 92, 120, 140 kN through wheel has applied on weld and observations have been made on weld crack 1 with a constant opening angle 60° and coefficient of friction 0.3. The welds cracks fatigue life for crack 1 and crack 2 under various vertical loads is shown in Figure 6 (a-b) respectively for different crack lengths and coefficients of friction.

Figure 6. Vertical loads effects on weld fatigue life (a) for crack 1 (b) for crack 2, Influence of Coefficient of friction on fatigue life (c) between wheel and weld contact (d) between crack surfaces, Influence of initial crack length on fatigue life (e) for both crack 1 and 2 (f) Von-Mises Stress on contact

When the weld is subjected to a vertical load on a crack having fixed initial length of 1 mm and at an angle 30° for both crack 1 and crack 2 under coefficients of friction 0.1, 0.2, 0.3, and 0.5, the weld crack fatigue life is estimated for both crack 1 and crack 2. Graph in Figure 6 (c) represent the weld life for
different coefficient of friction. Figure 6 (d) represents the estimation of welds crack life under fatigue for coefficients of friction between the surfaces of crack taking in to account as 0, 0.1, 0.2 and 0.3 respectively, considering a fixed vertical load of 92 kN and an opening angle of 60° and fixed initial crack length of 6 mm with wheel-weld surface friction as 0.3, and plotted for both crack 1 and crack 2. Figure 6 (e) illustrate the life of fatigue for weld with initial crack lengths of 1, 3, 6 and 10 mm respectively.

A good agreement between results of experimental fatigue test and FEM based solution on two different surface crack growth, performed by Yang Liu et al. [19] can be co-related with this wok for satisfactory and concluding remarks.

4. Conclusions
At wheel weld contact the Von- Mises equivalent stress condition has been achieved in this work, using the elastic-plastic modelling and interaction has been developed between wheel and UIC60 weldment were with different coefficients of friction. The Von Mises equivalent stress state at the wheel-rail-weld contact has been achieved using the elastic-plastic model in this work. The numerical method and finite element software ANSYS were used to obtain the results, and then analysed. The presence of frictional forces on the contact zone results as following:

- Von Mises stress is distributed in an asymmetric manner.
- Increment in Von Mises stress value at narrow depths on weldment occurs.

It has been shown that increasing value of Von Mises equivalent stress close to the contact surface have a negative impact on wheel and weld fatigue endurance when operating under the rolling contact phenomenon.

The linear elastic fracture mechanics used to perform the parametric analysis of the cracks, which represents operational factors such as vertical load, coefficient of friction between the wheel and weld contact, and surface friction for two types of surface cracks. On observational basis it is possible to draw the following conclusions:

- The greater the coefficients of friction at the contract surface between weld and wheel, results in higher growth of crack 1 and, as a result, the reduction in the growth of crack 2.
- The friction between the fracture surface and the friction between the weld and wheel have dramatically opposed effects on life of fatigue. The fatigue life of a fracture grows as the friction between the crack surface and the surrounding material increases. When the frictional constant between wheel and the weld increases, the fracture has a tendency to grow in a progressive manner. As a result, the duration of wear can be minimised.
- Crack 1 expands toward the rail depth as friction increases, and eventually transforms into transverse cracks as the rail depth decreases.

Results for novel parametric variables like stress due to manufacturing process, bending loads, and rail/weld hardening, may be obtained through future research for crack formation.

Acknowledgment
Authors appreciate the support from ANSYS for using the student edition for the simulation work and also express our gratitude for RDSO, Lucknow from the Indian railways to make use of the available resources.

References
[1] Indian Railways year-book 2020 , www.indianrailways.org.
[2] Cui X L, Chen G X, Yang H G, Zhang Q, Ouyang H and Zhu M H 2015 Effect of the wheel/rail contact angle and the direction of the saturated creep force on rail corrugation (Elsevier B. V.)
[3] Xin Z, Xiaogang Z, Chao L, Zefeng W and Xuesong Jin 2016 A study on dynamic stress intensity factors of rail cracks at high speeds by a 3D explicit finite element model of rolling contact (Elsevier B. V.)
[4] Creţu S 2009 Contactul concentrat elastic-plastic” (Editura „Politehnium”, Iaşi) [2]
Vo K D, Tieu A K, Zhu H T and Kosasih P B 2014 A tool to estimate the wheel/rail contact and temperature rising under dry, wet and oily conditions COMPRAIL 14th Conference on Railway Engineering Design and Optimization United Kingdom: WIT Press pp. 191-201
[5] Zhang J, Shouguang S and Xuesong J 2009 Numerical simulation of two-point contact between wheel and rail (AMSS Press, Wuhan, China).
[6] Wang W J, Lewis R, Yang B, Guo L C, Liu Q Y, and Zhu M H 2016 Wear and damage transitions of wheel and rail materials under various contact conditions (Elsevier B. V.)
[7] Peixoto D F C et al. 2007 Non-linear analysis of the wheel/rail contact (Departamento de Engenharia Mecânica, Faculdade de Engenharia da Universidade do Porto Portugal)
[8] Busquet M et al. 2003 3D Finite element investigation on the plastic flows of railheads. Correlation with micro structural observations. In: 6th International conference on contact mechanics and wear of rail/wheel systems CM2003 in Gothenburg, Sweden.
[9] Zili L, Xin Z, Coenraad E, Rolf D and Molodova M 2008 An investigation into the causes of squats—Correlation analysis and numerical modelling (Elsevier B.V.)
[10] Wen-jian W, Wen-juan J, Heng-yu W, Qi-yue L, Min-hao Z and Xue-song J 2016. Experimental study on the wear and damage behavior of different wheel/rail materials (Journal of rail and rapid transit)
[11] Masoudi Nejad, R., Shariati, M., and Farhangdoost, Kh. (2016), “Effect of Wear on Rolling Contact Fatigue Crack Growth in Rails,” Tribology International, 94, pp 118–125.
[12] Liu, Y., Liu, L., and Mahadevan, S. (2007), “Analysis of Subsurface Crack Propagation under Rolling Contact Loading in Railroad Wheels Using FEM,” Engineering Fracture Mechanics, 74, pp 2659–2674.
[13] Liu, Y., Liu, L., Stratman, B., and Mahadevan, S. (2008), “Multiaxial Fatigue Reliability Analysis of Railroad Wheels,” Reliability Engineering and System Safety, 93, pp 456–467.
[14] Liu, Y., Stratman, B., and Mahadevan, S. (2006), “Fatigue Crack Initiation Life Prediction of Railroad Wheels,” International Journal of Fatigue, 28, pp 747–756.
[15] Cornell Fracture Group, Cornell University. Available at: http://www.cfg.cornell.edu (accessed February 16, 2014).
[16] Anderson, T. L. (1994), Fracture Mechanics, Fundamentals and Applications, 2nd ed. CRC Press: New York, NY.
[17] ANSYS Workbench help.
[18] http://www.railroadpart.com/news/fishplate-joint-and-rail-joint-welding-in-indian-railway.html
[19] Yang Liu, Kin Shun Tsang, Hsin Jen Hoh, Xiaopeng Shi, John Hock Lye Pang, Structural fatigue investigation of transverse surface crack growth in rail steels and thermite welds subjected to in-plane and out-of-plane loading, Engineering Structures, Volume 204,2020,110076, ISSN 0141-0296, https://doi.org/10.1016/j.engstruct.2019.110076.