DYNAMIC FINITE ELEMENT ANALYSIS OF THE WHEEL – RAIL INTERACTION ADJACENT TO THE INSULATED RAIL JOINTS

Tao Pang# & Manicka Dhanasekar↑

# Rail CRC Postgraduate Student, Centre for Railway Engineering, Central Queensland University, QLD 4702, Australia.
↑ Director, Centre for Railway Engineering, Central Queensland University, QLD 4702, Australia.

m.dhanasekar@cqu.edu.au

ABSTRACT

Through a 3D dynamic wheel/rail rolling contact model, the contact impact at the wheel/rail interface adjacent to insulated rail joints (IRJ) is reported in this paper. An elasto-plastic finite element method based on explicit algorithm is employed to simulate the dynamic rolling contact impact phenomenon from which numerical results are determined. It is shown that the explicit algorithm provides appropriate solution only when steady state static solution obtained from the standard implicit algorithm is imported into it. Contact impact force history as well as contact pressure distribution parameters are investigated. The results indicate that in the vicinity of the IRJ, the contact pressure distribution is very different from the zones away from the IRJ. The numerical results are compared with the classical Hertz theory and conclusions drawn.

1. INTRODUCTION

Insulated rail joints (IRJ) are regarded as points of weakness in rail track network. The discontinuities induced by the IRJ to the rail, which is otherwise regarded as a continuum for stress analysis, produce high frequency dynamic impact forces that cause localised damage and failure. This mechanism of failure is believed to cause significant reduction to service lives – as low as 200MGT (whilst in general the continuously welded rails provide 1000MGT service life) - of IRJ with subsequent increasing demand for maintenance to ensure safety [1]. The failure of IRJs within short service span, especially in heavy haul lines, has therefore become a significant economic problem that is technically very challenging to solve. This is because, although the classical mechanics provide solutions for linear and nonlinear contact and impact of solids that are idealised as elasto-plastic continuum, the mechanics of contact-impact close to the edges of solids remain relatively not well explored [2].

In recent times some information on the behaviour of IRJs has started emerging in the open literature. Deflection and stresses near the IRJ have been investigated by Kerr and Cox [5]. Using a modified beams on elastic foundation model, the rail and joint bars have been modelled as linear elastic beams, and the epoxy-fiberglass insulation has been simplified as spring layers using the Zimmermann hypothesis. They have validated their static model using a three point bend test of a simply supported IRJ. Whilst their model could improve design principles of new rails, failure of existing IRJs could not effectively be analysed. Recently Yan and Fischer [3] and Chen and Kuang [4] have reported, through a 3D finite element analysis (FEA), that the contact pressure parameters vary significantly from the conventional Hertzian contact theory (HCT), when the rail is loaded either at locations where there exists change in curvature of at least one of the contacting bodies, or close to the rail joints.

Contact problems in rail are popularly solved using HCT as reported in several studies [6-9] that focus on the fatigue and fracture of railhead. The application of HCT for wheel-rail contact problems is however questioned in [4,10]. Pau et al. [11] used an ultrasonic experimental method to examine the contact area in wheel/rail contact and compared the experimental results with that of the finite element method and HCT.

Chen and Kuang [4] also carried out a static analysis of an IRJ subjected to vertical wheel loadings through an
elastic 3D FE rail model. They have found that the traditional HCT was not valid for predicting the contact pressure distribution near the rail joints and have listed several idealized elliptical contact dimensions (major and minor axes). In their later paper [10], Chen and Chen have used a 2D FE model in which different traction and braking forces were applied to investigate the contact stress and maximum shear stress distribution in the railhead. Their model was based on a static analysis and provided the conclusion that both Hertz and Carter’s theory were not valid near the IRJ due to edge effects [12]. None of these studies have considered the dynamic effects of the wheel passing over the IRJ, and thus are valid only for static loads or slowly moving quasi-static loads.

Wen et al [13] performed a dynamic elasto-plastic finite element analysis of the standard fish plated rail joints containing a gap. They employed a coupled implicit - explicit technique that imported the initial steady state implicit solution prior to impact into the explicit solution to determine the impact dynamic process. They reported that the impact load varied linearly with the static axle load but was largely insensitive to the speed of travel of the wheel. The impact force history presented in the paper exhibited three peaks, which were difficult to comprehend given the model allowed for only a single wheel. Impact force factors as high as 2.6 was reported. This paper reports a 3D elasto-plastic finite element analysis of contact impact between the wheel and the rail with particular reference to IRJ and contact pressure distribution in the vicinity of IRJ using the ABAQUS [14] FE package. The results contained in this paper is a part of an ongoing research on the failure of IRJ at the Centre for Railway Engineering (CRE), Central Queensland University supported by the Rail CRC.

2. NONLINEARITIES – A REVIEW

The nature of dynamic contact impact problems introduces several nonlinearities; this paper particularly is concerned with material and contact nonlinearities.

2.1 Material Nonlinear Solution

Static problem is represented as follows:

$$\mathbf{K} \mathbf{u} - \mathbf{P} = 0$$  \hspace{1cm} (1)

in which $\mathbf{K}$ is the stiffness matrix of the structure that is affected by the state of strain at element level; $\mathbf{u}$ is the displacement vector and $\mathbf{P}$ is the external force vector. Newton Raphson method is a popular technique that is widely used in the finite element approximation for the solution of the equilibrium equations. In static problems, the solution is obtained through a series of increments with each increment involving a number of iterations until convergence is obtained for that increment.

ABAQUS/STANDARD was used in this paper to deal with the nonlinear static problem, which has the capability to apply the whole load in one “step” with automatic incrementation and iteration through powerful arc-length algorithms to deal with severe nonlinearities.

2.2 Dynamic Nonlinear Solution

As the current problem involves impact, high frequency wave propagation is central to the solution that demands small time increment. Explicit time integration methods are used for this purpose. The dynamic equilibrium equation is written as follows:

$$M \ddot{u} + C \dot{u} + K u - P = 0$$  \hspace{1cm} (2)

in which $\mathbf{M}$ and $\mathbf{C}$ are mass and damping matrices and over-dot stands for time derivatives. Eq. (2) is solved using central difference scheme as shown in (3).

$$\ddot{u}_0 = M^{-1}[-C\dot{u}_0 - K(u_0) + P_0]$$  \hspace{1cm} (3)

To evaluate the $u_{n+1}$, $u_0$ and $\dot{u}_0$ are used as initial conditions. ABAQUS/EXPLICIT simplifies the solution of Eq. (3) by ignoring $\mathbf{C}$ and $\mathbf{K}$ matrices; a large number of very small increments are carried out to obtain the required level of accuracy. This procedure requires the time increment to be less than the material wave propagating speed through the characteristic length of the smallest element for the efficiency of the solution.

2.3 Contact Nonlinearity

Contact condition introduces boundary nonlinearity because of its inequality boundary condition. To resolve the contact problem in the finite element method, detecting for active boundary constraint is essential. ABAQUS uses contact algorithm that employs Newton-Raphson iteration technique to detect the active boundary constraint.

For two bodies coming into contact, one of the contact bodies is defined as slave and another as master with the permission of the master surface penetration into the slave surface but not vice-versa. The first step is examining the state of contact at the start of each increment to establish whether the slave point is open or closed. If closed, further detecting has to be applied to determine whether it slides or sticks. When closed, active constraint is applied and where open state exists, the contact constraint is removed.

Prior to equilibrium checks, ABAQUS searches for any changes in contact condition. If the contact gap clearance becomes negative or zero after the iteration, the contact condition is set closed. If the contact pressure becomes negative, the contact condition is changed from closed to
open. Any change detected in the current iteration will terminate the current iteration. When no changes exist in the current iteration, equilibrium checks will be carried out in these iterations until convergence is obtained. Any contact condition change will resort back to contact check within the equilibrium check.

3. FINITE ELEMENT MODEL

The objective of this analysis is to develop a finite element model by which the rolling contact impact between the wheel and the rail in the vicinity of an IRJ could be simulated to investigate the contact impact behavior. To realise this objective, a dynamic contact model was developed.

There are two major difficulties in establishing the FE model: the first is the requirement of fine mesh near the IRJ versus the need for considering the full length of the rail that is affected by the influence of the wheel; and the second is the requirement of obtaining steady state rolling of the wheel prior to crossing the IRJ.

As the first demands much larger length of rail to minimise the effect of the longitudinal boundary condition, it is simply not possible to adopt 3D solid modelling of the rail for the full length; in this research, beam elements and solid elements with appropriate coupling were used.

The second demands sufficiently larger length of rail modelled with solid elements for absorbing the initial condition effects of the rolling wheel. The length of rail modelled with solid elements was determined with the prime focus of the second issue.

Furthermore, the contact nonlinearities pose a severe problem of running the entire analysis of rolling wheel – rail model in the explicit algorithm. To solve this problem, steady state contact solution was first obtained from a static implicit analysis and the results imported into the dynamic explicit analysis. In the first step, static analysis, bolt pretension, rail/wheel static contact and wheel rolling condition are analysed by using the FE code ABAQUS/STANDARD. The static solution was then transferred to the dynamic analysis code ABAQUS/EXPLICIT to determine the contact impact behavior at the IRJ. It is notable that ABAQUS/Explicit deals with this dynamic problem with relative ease due to the explicit integration algorithm it employs to solve the equilibrium equations. With this concept, a practical FE model was developed with some assumptions and simplifications.

In this research, the geometry of GIJs are considered as per the Australia Standard (AS 1085). 60 kg rail steel (AS 1085.1) and other necessary components and their assembly as shown in Fig.1 included in the standard are adopted for the study.

Figure 1: typical insulated joint assembly (AS 1085, 2002)

Fig. 1 shows a typical insulated joint assembly. The main theme of the IRJ is to electrically insulate all metallic components through the joint insulation and the end post materials. To overcome the structural weakness induced by the IRJ to the rail system as reflected by maintaining the vertical and the lateral alignments and bending stiffness as close as possible to the uncut original rail, the two sections of the rail are rigidly joined by a pair of supporting plates (known as joint bars) electrically separated by the flexible insulating materials from rails and fastened by bolts.

As the analysis is limited to the rail/wheel contact impact in the vicinity of the IRJ, for effective finite element modelling the geometry of the IRJ was simplified to just one part model with several partitions for material definition by ignoring the interaction details of contacting surfaces of various components. The simplified geometry that defines the whole IRJ as one instance is shown in Fig. 2. A slit was included in the instance to represent the bolt hole; this allowed specification of bolt tension in the solid medium where the bolt would be present in the original model. The distance between the two slits was
kept the same as that of the diameter of the bolt. The nuts and washers were ignored as they do not contribute to the stiffness of the joint.

Bolt pretension was applied through the internal cross section of the “bolt shank” connecting the joint bars as represented in between the slits; an expression (Eq. (1)) provided in Wen et al [13] was used for this purpose.

\[ P_b = \frac{T}{K_bD} \]

Where \( P_b \) is the bolt pretension, \( T \) is the bolt torque moment, \( D \) is the bolt diameter, and \( K_b \) is a coefficient of the bolt torque moment (\( K_b = 0.19-0.25 \)). The selected values for the analysis were: \( T = 500 \text{Nm} \), \( D = 27 \text{mm} \), and \( K_b = 0.2 \).

To simulate the impact caused by the rail/wheel contact near the IRJ, a single wheel symmetrically rolling on the rail that was resting on discrete viscoelastic supports as shown in Fig.3 was employed [16]. The wheel was subjected to one-eighth of the wagon mass \( M_w \) through a primary suspension simplified as a spring \( K_e \) and a dashpot \( C_s \) in the vertical direction. The elastic support was simplified as a spring and dashpot system with stiffness \( K_s \) and damping \( C_s \) positioned at the location of sleepers. The wheel surface was defined to contact with the railhead surface as it rolls with a pre-defined velocity.

According to the Sun[17], the effect of a single loaded wheel on a rail is felt for a distance of 6m on each side of the wheel. To reduce computational costs and increase the efficiency of the FE model, the full 12m length of the rail was divided into three parts: the middle fish-plated rigid IRJ of length 2.4 m was discretised using solid elements, for the rail that extended 4.8 m on each side of the middle IRJ section was modelled using beam elements. Coupling method was used to ensure continuity at the section of connection between the solid elements and beam elements (Fig. 4).

![Figure 2 Simplified model of IRJ](image1)

Special attention was paid to the spring support boundary conditions in the explicit dynamic analysis. The “import” function in ABAQUS provides the capability to import a deformed mesh and its associated material state whereas the boundary conditions, loading, and contact interactions could not be imported and hence have to be redefined in the explicit model.

As the newly defined supporting and suspension springs should contain the initial forces obtained in the steady state analysis, we defined the forces in the springs using the relationship between spring force \( F_b \) and displacement \( x \) of springs as shown in Eq. (4).

\[ F_b = K_s x + F_b^0 \]  

(4)

in which \( F_b^0 \) is the spring force obtained in the static analysis.

Mesh of the rail/wheel set is shown in Fig.5. Fine mesh was used in the contact region on the top of the railhead and the surface of the wheel. A total of 97,063 nodes, 84,010 eight node solid reduced integral elements (C3D8R) and 48 beam elements were used.

Definition of rail/wheel contact interaction in ABAQUS/STANDARD is very sensitive to iteration convergence, result accuracy, and computational time. Thus accurately defining the rail/wheel contact is a key issue in the impact dynamic analysis.
Surface contact method of ABAQUS is chosen throughout the whole static and dynamic analyses. Surface of the wheel was defined as the master contact surface, while the rail head was defined as the slave contact surface. The contact surface pair was allowed finite sliding. Friction parameter between them was set as 0.3.

For static analysis, hard contact based on the penalty method was chosen with the contact pressure-overclosure relationship in ABAQUS/STANDARD. Iterations continued until convergence of the solution was obtained. If a slave node penetrated into the master surface by more than 0.1% of the characteristic interface length, the contact pressure was “augmented” and another series of iterations were executed until convergence was achieved. Only when the penetration tolerance requirement was satisfied was the solution accepted.

For the rolling contact impact simulation in ABAQUS/EXPLICIT, the kinematic contact algorithm was used. This contact algorithm searched for slave node penetrations in the current configuration. Contact forces that were a function of the penetration distance were applied to the slave nodes to oppose the penetration, while equal and opposite forces acted on the master surface at the penetration point. When the master surface was formed by element faces, the master surface contact forces were distributed to the nodes of the master faces that were being penetrated.

At the beginning of the surface based contact analysis, it is difficult to get perfectly matching surfaces; small gaps or penetrations caused by numerical round off, bad assemblies, or various other reasons invariably exist. Adjusting of the initial position of the slave contact surface was therefore taken into consideration to eliminate these gaps or penetrations. Where such care was not taken, slave nodes that were over-closed in the initial configuration remained over-closed at the start of the impact simulation, which caused convergence problems. In the static implicit analysis, an adjustment zone was defined by specifying an adjusting depth. In the explicit impact analysis, contact controls parameter approach was used to activate the viscous damping in the vertical direction such that numerical difficulties associated with rigid body motion was avoided.

4. NUMERICAL RESULTS

Numerical examples considered 60kg/m rail and 460 mm radius wheel containing 1/20 conicity. Wheel load was taken as 150 KN and the wheel velocity was set as 120km/h.

Elastic and plastic material properties of the rail steel and nylon insulation material used in the analysis are listed in Table 1 and Table 2 respectively. Parameters of supporting and suspension spring and dashpot sets are listed in Table 3.

| Table.1. Elastic properties of Materials |
| --- | --- | --- |
| Material | E [MPa] | ν |
| Rail Steel | 207000 | 0.3 |
| Nylon66 | 1590 | 0.39 |

| Table.2. Plastic properties of Rail Steel |
| --- | --- | --- |
| σy (MPa) | 830 | 1230 | 1240 |
| εp | 0 | 0.01 | 0.1 |

| Table.3. Stiffness and damping parameters for support and suspension systems |
| --- | --- | --- | --- |
| Kb (MN/m) | 26.8 | 220 | 14.5 |
| Ks (KN/m) | 1230 | 220 | 14.5 |
| Cb (KNs/m) | 1240 | 14.5 | 138 |
| Cs (Ns/m) | 0.01 | 0.1 |

Impact dynamic analysis of an IRJ was performed using ABAQUS/EXPLICIT; the contact force between the
railhead and the wheel obtained from the analysis is shown in Fig. 6.

![Rail/Wheel contact force history](image1.png)

Figure 6. Rail/Wheel contact force history

This figure shows that, at the beginning of the analysis, the contact force increases sharply from zero to 157KN. The zero initial force was due to static contact analysis that commenced with the condition of the wheel not in contact with the rail. At the end of the static analysis, the contact force reached 150kN that was equal to the static load imposed on the wheel. At the beginning of the explicit analysis, the imported static contact force increased further to 157kN due to the effect of the initial condition. After a short period of decline, the contact force stabilised at 150kN just after 2 milliseconds of travel time. From 6.6 milliseconds of travel, the wheel entered into the vicinity of the IRJ. The contact force decreased gently from 150kN to 138kN before it diminishes rapidly to 130kN. The impact happened at 9.3 millisecond and the vertical contact force peaked at 179KN during this period. Vibration continued after the impact and stability was regained at 15.5 millisecond of travel.

Plastic dissipation history at the IRJ is shown in Fig. 7. There is a sudden surge during impact and then the dissipation remained constant after the impact. This phenomenon shows that the impact force was sufficient to stress the material in the vicinity of IRJ beyond the elastic range causing unrecoverable plastic strain. Fig. 8 showed that, at the time of impact, the kinetic energy sharply increased and oscillated after the impact.

At the time of impact the contact force reached its maximum magnitude, and the corresponding Von Mises stress was investigated as shown in Figs. 9 & 10. The contours indicate the Von Misses stress on the surface of railhead was 531 MPa and the maximum stress of 840 MPa (the rail material plastic stress) occurred at 4.98mm below the railhead surface and 10.8mm away from the rail endpoint.

![Plastic energy dissipation history](image2.png)

Figure 7. Plastic energy dissipation history

![Kinetic energy history](image3.png)

Figure 8. Kinetic energy history

![Von Mises stress contour (top of rail view) just after impact](image4.png)

Figure 9 Von Mises stress contour (top of rail view) just after impact

![Von Mises stress contour (longitudinal vertical symmetric plane through the rail) just after impact](image5.png)

Figure 10 Von Mises stress contour (longitudinal vertical symmetric plane through the rail) just after impact
Contact pressure distribution at the top of the railhead obtained during one of the increments of the rolling of the wheel, with the wheel residing 57.1mm away from the IRJ is shown in Fig. 11. The shape of the contact pressure zone shown in Fig. 11 appears elliptical with the major axis oriented along the longitudinal direction (shown by the single headed arrow) of travel.

The area of the contact pressure distribution zone obtained at each increment of the explicit analysis is plotted as a time history in Fig. 12. In this figure the contact area predicted by the HCT is also shown (as the horizontal straight line). Whilst the HCT predicts the area as 160 mm$^2$, the explicit dynamic analysis predicted area varied around a value of approximately 260 mm$^2$ prior to impact, and has registered a sharp increase in the contact pressure area to 450 mm$^2$ at the time of impact, and just after the impact it varied around 320 mm$^2$.

The variation of the maximum contact pressure along with the prediction of HCT is shown in Fig. 13. Except for the influence of the initial conditions, the $P_0$ determined from the explicit FE analysis coincided with that of the HCT analysis until the wheel was 129mm away from the IRJ and then started deviating from the HCT prediction as the wheel approached the IRJ. $P_0$ shows a steep increase for this magnitude after crossing over the IRJ.

When the wheel just crossed the IRJ, effectively both rails shared the wheel contact, and as such the contact pressure contour was defined by two ellipses, one for each of the rails.

Fig. 14 shows that the contact area was divided into two parts and the middle part corresponding to the end post material was out of contact. Clearly HCT could not determine such situations clearly. (The single headed arrow denotes the travel direction)
5. CONCLUSIONS

The vertical contact force history that occurs between the wheel and the rail through the rolling contact procedure was investigated in this finite element model and impact forces were also determined. The effect of the IRJ on the contact pressure distribution was also explored.

From the simulation results, the following conclusions are made:

1. This FE model determines the impact force by developing a rolling contact procedure between the wheel and the rail. The impact force can be observed at the time when the wheel rolls over the IRJ. Impact factor was determined to be around 1.2 for the type of problem investigated.

2. For the wheel/rail rolling contact away from the IRJ, the shape of the contact area remained elliptical, consistent with the HCT determination. The size of the contact area was, however, larger than the HCT evaluated values. The maximum contact pressure at the original point of contact patch P₀ determined by this simulation was also proven to be larger than the one advised by HCT.

3. The IRJ plays a significant role in the contact pressure distribution when the contact patch is close to the IRJ. Both the shape and the size of the contact area are different from HCT which is based on the elastic material and infinite half space assumptions.

4. The maximum contact pressure P₀ is significantly larger just after the impact of the wheel even though the associated contact area also increased.

6. ACKNOWLEDGEMENTS

The authors express their gratitude to Rail CRC - the Cooperative Research Centre for Railway Engineering and Technologies - which has funded the research. Sincere thanks are also due to Ian Marks of QR, Chair of the steering committee of the GIJ Failure Mechanism research project for the constant support. Inkind support is provided by QR, CQU and ARTC.

REFERENCES