International Journal of Advance Research in Science and Engineering
Volume No.07, Special Issue No.01, March 2018
www.ijarse.com
IJARSE
ISSN: 2319-8354

CONTACT ANALYSIS OF RAIL WHEEL USING FINITE ELEMENT TECHNIQES - A REVIEW

Aditya Deshpande[1] , Prof. U. M. Nimbalkar[2]

[1]*M.Tech (CAD/CAM/CAE), Rajarambapu Institute of technology, Rajaramnagar, India.*
[2]*Assistant Professor, Rajarambapu Institute of technology, Rajaramnagar, India.*

ABSTRACT

Rail wheel transport system is one of the widely used systems to transport load. And Finite Element techniques shoes one of the most effective ways for simulation and stress analysis of contact system. This paper reviews the work done by various authors in field of contact analysis of rail wheel using finite element techniques. Various FE softwares such as ANSYS, NASTRAN are used for analysis purpose and numerical formulation is also done in some cases. This studied data is used as base for contact analysis of rail wheel geometry of Rail trolley used for material handling and results of analysis are predicted based upon the available data.

Keywords: Rail Wheel contact, simulation, Finite element analysis.

1. INTRODUCTION

The question about the relation of wheel-rail is very important to the vehicles and the rail system, and the research is very complex and involves many disciplines such as contact mechanics, elastic-plastic mechanics, tribology, heat transfer theory, and systems dynamics and so on. Rail transport is one of the most effective ways of transport. Rail wheels have high load carrying capacity and greater life compared to other type of wheels. Hence for these reasons many industrial material handling equipment incorporates rail and rail wheel mechanism. is one of the effective way sued The stress distribution is an important factor at the Rail–Wheel contact interfaces, i.e. two materials contacting at a rolling interface, are highly dependent on geometry of the contacting surfaces, loading conditions and material properties of the system of interest. The factors which cause the damages of wheel/rail contain the creep and collisions, shocks, vibrations and relative sliding.[1] As the rail wheels are generally subjected to high loads, it is essential to calculate stresses that are produced in wheel and rail and also deformation in the contacting bodies. Different techniques are available to calculate stress values in the bodies. But finite element techniques show more reliability as well as realistic results. Here in this paper work done by various authors in analysis of rail wheel contact problem is studied using FE technique.[2-7] Rail wheel analysis results can be used to modify the design and it can also be used to alter the material as well as working conditions of system for better results.

2. APPROACHES FOR RAIL-WHEEL CONTACT ANALYSIS

Contact analysis of rail wheel geometry is categorised depending upon the FE techniques followed for analysis and parameters considered for analysis.

2.1. Rail-Wheel contact analysis using FE software

Aleksander Sladkowski, Marek Sitarz [3] research into the area of effect of rail and wheel contact profile on the distribution of contact zone and stress. The quasi-Hertz method as well as a finite element(FE) methods were included as basis of mathematical simulation. The effects of roughness, defects and cracks are also considered in the literature. Geometric Modeling of rail and wheel can be created in any CAD environment. Fig. 1 shows an FE model of the wheel–rail contact interaction and the irregular grid of rail. The Grid/mesh between rail and wheel have been created by using following method: the area of possible contact of a wheel and rail is allocated on the surface where generation of a surface grid is carried out; the element size in the grid is kept constant. Automatic grid generation is used for remaining geometry of rail. Visual NASTRAN for Windows created the remaining FE grid of the rail head. Similar technique is used for creating grid for wheel geometry. It is preferred that the grids should be regular in rail and wheel contact area to eliminate the possible errors. Authors developed in house code to create matching between the flat surface of rail and wheel, rest of the wheel and rail automatic mesh generation was employed which resulted in an irregular mesh, as the contact area between wheel and rail is our area of interest. The wheel is fixed at the nodes on a contour of a hole in the centre of the wheel.

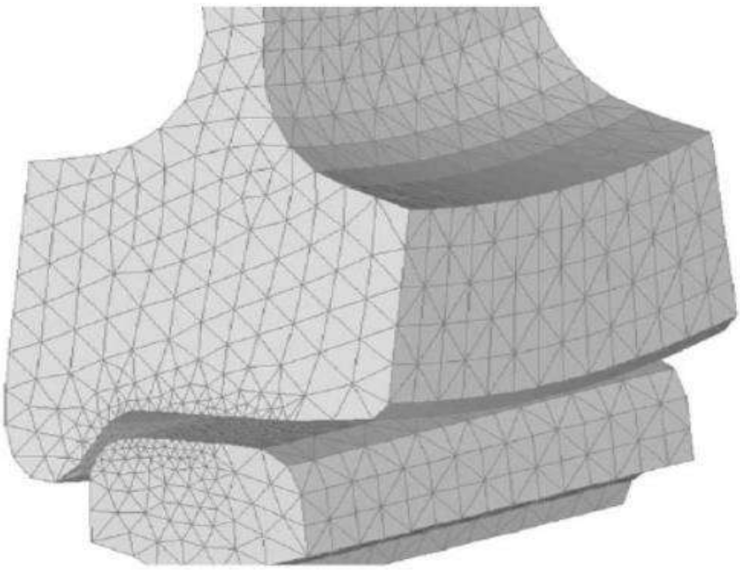

Fig 1: Contact between wheel and rail

Standard mechanical properties for rail and wheel are considered for analysis purpose. Dense FE mesh is used for analysing the contact problem and the corresponding boundary conditions were applied to rail and wheel. Next, the considered areas of the wheel and rail were positioned. Vertical and lateral force were applied on the borders of these areas and the location and size of the contact zones was derived along with the distribution of stresses in an iterative manner. The magnitude of force acting on the wheel was chosen 125kN static or 200kN quasi-static and for the lateral force it was chosen from 10 kN to 100 kN. After carrying out analysis it is seen that the wheel rim becomes slightly warped under the influence of a normal force. The contact between rail and wheel is generally one point due to standard profiles of rail and wheel but in some cases two point contact is also possible, which is also considered here. Such two point contact of a wheel and a rail takes place in the case of the displacement of a wheel to the lateral side of the track relative to its initial position and hence, the wheel flange comes in contact with the lateral surface of the rail head wheel results in significant lateral forces. This causes the increase in size and magnitude of stresses in flange area. The advantage of using FE techniques is, it is possible to analyse different relative positions of rail and wheel. Authors developed new profile for wheels based upon results of the research by varying the flange thickness of wheel. This new profile helps in decreasing the intensity of wear and also the slip between the wheel and rail.

2.2. 3D analysis of Rail-Wheel contact geometry

Mehmet Ali Arslan , Oguz Kayabas [4] performs the 3D analysis of rail wheel contact. As seen in other works wheel geometry of 1000mm diameter and corresponding rail and axel geometries are created and assembled in CAD environment. Geometrical model is meshed with SOLID45 8-noded hex elements. To define contact, ANSYS Contact173 elements placed on wheel and ANSYS Target170 elements placed on rail. It is considered that total 80Tonn is acting on the wheels, which is considered to be distributed equally on each of eight wheels. And the lateral loads on the system are considered to be negligible. Friction coefficient, 0.3 is used. Cut view of rail and wheel geometry, along normal direction is considered and boundary conditions are applied on it.

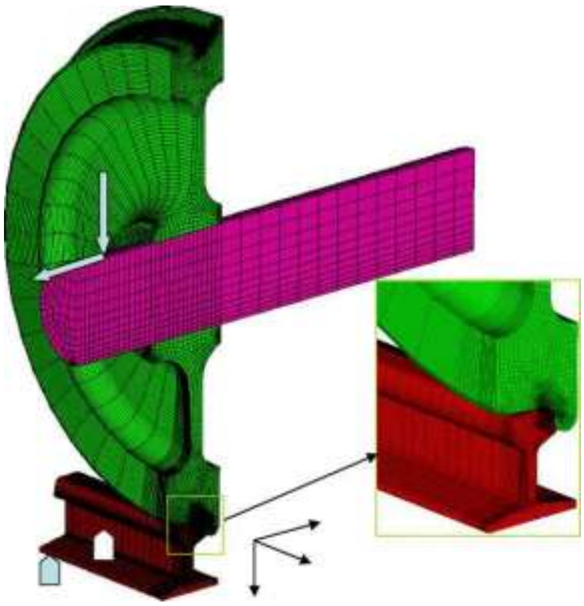

Fig 2: Rail-Wheel contact geometry

Rail is held fixed at traverse locations at all directions. Instead of applying load directly on the wheel, in this case it is applied on the bearing location. Also consideration of axel in the contact assembly makes the FE loading and boundary conditions more realistic. After observing the results obtained by analysis, it is seen that for wheel the maximum Von Mises stress value is at 2.73mm inside the outer surface and value of stress is above the yield limit. And similarly for rail maximum stress is found 1.53 mm inside of the outer surface and value of stress lies well above yield limit of rail material. most of the plastic deformation occurs at the Rail, on the contrary almost very small plastic deformation occurs at the Wheel. After visualising results of 3D analysis it is seen that most of the plastic deformation is in the rail and comparatively very small plastic deformation occurs at wheel.

2.3. Numerical formulation and software based Dynamic Analysis of rail wheel contact

Sunil Kumar Sharma, Anil Kumar [5] used numerical and mathematical model for the analysis of rail wheel contact. For the analysis purpose contact between rail and wheel of fait bogie is taken under consideration. The material model of wheel and rail is assumed to be linear-elastic. Coefficient of friction is taken as 0.78 between the contacting surfaces. For the analytical model of contacting surfaces numerical formulation for principal stresses is done depending upon the load on the wheel and dimensions of the contact area and velocity of wheel. Along with the analytical model, a FE model for rail wheel contact is also made in Ansys. For the analysis purpose between the contact surfaces the hexa-dominating meshing is done. Frictional force is calculated by varying the lateral displacement. One of the major factors which can affect the real contact area dimensions and the plastic pressure distribution is the rail inclination. It can be noticed that a higher rail inclination provides a higher maximum pressure. It can be observed that the frictional force is higher in the absence of creepages, but as the lateral creepages increase the frictional force gets reduced. by varying the lateral creepages, its effects on friction force is explained and its value were taken from 0 to 0.1 and. Rail wheel contact problem was investigated by varying contact profile geometries to estimate the contact pressure and contact stress. A comparison of Hertzian calculation and FE-simulation shows a close agreement in contact pressure and contact stress for different contact patches. From the results of the present model it is seen that the increment of rail profile causes contact stress reduction and also the effect of increasing wheel profile radii increases the width and decreases the length of the contact area ellipse causes higher sliding friction. There is also influence of wheel taper on contact area, it increases the length and reduces width of contact area which causes reduction in sliding friction. The lateral displacement of the wheel considerably alters both, the shape of the real contact area and the maximum value of the pressure distribution. Hence, it is observed that the Shape of wheel is major factor which affects the pressure distribution and shape of contact area.

2.4. Thermo-Mechanical Analysis of rail wheel contact area

Lei Wu, Zefeng Wen, Wei Li, Xuesong Jin [6] considered deformation due to thermal effect produced during sliding movement of wheel on rail along with deformation due to other stresses. A finite element method (FEM) is designed to study thermal-elastic–plastic deformation and residual stress after wheel sliding on a rail. The research is made on effect of frictional temperature on deformation as well as the residual stresses using the Thermo-Mechanical FE method. The heat flux perunit time when the wheel is sliding over the rail with constant velocity V_s in contact area can be given as:

$$q(x') = f\,V_s\;p_t\,(x')$$

The influence of the heat convection and radiation on the wheel/rail temperatures distribution is implemented by a heat transfer coefficient. The thermal boundary condition on the wheel/rail contact patch is assumed to be the boundary condition (heat flux) and the other free surface is subjected to the boundary condition (heat convection). To get the accurate results of the stess and strain values on and near the contact surface, very fine mesh is used to carry out analysis. The wheel/rail contact patch is assumed to be heat flux boundary and the other running surface of the rail is assumed to be heat convection boundary. Under the combined effect of the thermal load and the wheel load, the residual displacements at the rail surface in the longitudinal direction are larger than those in the vertical direction. influence of thermal effect on the development process of equivalent plastic strains is significant, but that on the stabilized values of the plastic strain is insignificant. Result of analysis shows that when the thermal effect for every rail-wheel contact is considered, the residual displacement, the equivalent plastic strain and the residual von Mises stress on the rail surface are the highest than other loading cases. Hence the effect of thermal stresses is very much significant on residual stresses and strain values.

3. Proposed Work

Based upon the above mentioned literature FE software is used for meshing and boundary conditions are applied on the wheel of Rail Transfer trolley. The model of wheel is shown below:

Fig 3: Model of wheel for Rail transfer trolley

For the analysis of the wheel the meshing of wheel and rail is done in Hypermesh software. The wheel has bearing inside it which rests on the shaft, to have free rotation about shaft. Hex element is used for the meshing of wheel and rail. Fine mesh is preferred for better accuracy of results. For defining the Bearing between when and shaft Locus of nodes onside wheel surfaces is created and similar is done for nodes on shaft. Then these two points are connected using 1D rigid element having only one rotation free about the axis. To define the contact between wheel and rail, frictional surface to surface contact is used with 0.2 as coefficient of friction. Total load on trolley will be 3000KG and total mass of trolley components will be acting on wheels. This weight is assumed to be equally distributed over 4 wheels thus approx. 10kN on each wheel. Now boundary conditions for geometry are decided depending upon actual working conditions. Rail is fixed from bottom and sideways to avoid displacement in that direction and load is applied on the shaft, as it is the actual component carrying the load. Standard material properties are used for wheel and rail as well. The meshed model with applied boundary conditions is shown below:

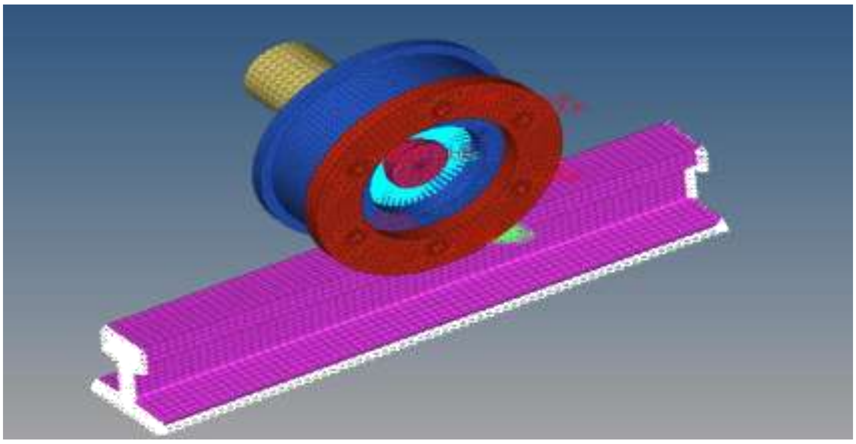

Fig 4: Rail wheel contact geometry with meshed surfaces

As per above literature we can say that, the expected results for above mentioned analysis can be small deformation in the rail surface and wheel surface as well as high stress development in both the surfaces. The deformation will be comparatively less as the thermal effect of contact between rail and wheel is not considered.

4. CONCLUSION

Researchers have done tremendous work in field of contact analysis of rail and wheel. Various factors that must be considered for analysis purpose changes depending upon the working environment and hence alter results of analysis. Finite Element software enables us to predict life of rail and wheels very accurately, when real working conditions are neatly applied. Numerical formulation can also be done using FE methods to campare software and analytical results. But due to the accuracy of it, finite elements techniques require more time for processing and also it is very complex to apply.

REFERENCES

[1] Arthur Anyakwo, Crinela Pislaru, Andrew Ball, A New Method for Modelling and Simulation of the Dynamic Behaviour of the Wheel-rail contact, 9(3), June 2012, 237-247.

[2] Qi Wanming, Zhang Jun, Sun Chuanxi, Wang Shengwu, Finite element analysis of wheel-rail thermal-contact coupling when the wheel's sliding, December 2009.

[3] Aleksander Sladkowski, Marek Sitarz, Analysis of wheel–rail interaction using FE software, 258 (2005) 1217–1223.

[4] Mehmet Ali Arslan, Oguz Kayabas, 3-D Rail–Wheel contact analysis using FEA, 45 (2012) 325–331.

[5] Sunil Kumar Sharma, Anil Kumar, Dynamics Analysis of Wheel Rail Contact Using FEA, 144 (2016) 1119 – 1128.

[6] Lei Wu, Zefeng Wen, Wei Li, Xuesong Jin, Thermo-elastic–plastic finite element analysis of wheel/rail sliding contact, 271 (2011) 437–443.

[7] L. Maa, C.G.He, X.J.Zhao, J.Guo, Y.Zhu, W.J.Wang, Q.Y.Liu, X.S.Jin, Study on wear and rolling contact fatigue behaviors of wheel/rail materials under different slip ratio conditions, April2016.

[8] A. Roviraa, A. Roda, M.B. Marshall, H. Brunskill, R. Lewis, Experimental and numerical modelling of wheel–rail contact and wear, 271 (2011) 911– 924.

[9] Zefeng Wen, Lei Wu, Wei Li, Xuesong Jin, Minhao Zhu, Three-dimensional elastic–plastic stress analysis of wheel–rail rolling contact, 271 (2011) 426–436.

[10] Sajjad Z. Meymand, Alexander Keylin & Mehdi Ahmadian, A survey of wheel–rail contact models for rail Vehicles, Jan 2016.