Influence of geometric elements on the pressure distribution and the state of tension at non-Hertzian wheel-rail contact

I Blanari¹, V Goanță¹ and M Alexandrov¹

¹Mechanical Engineering Department, “Gheorghe Asachi” Technical University of Iasi, Iasi, Romania

E-mail: blanariigor@gmail.com

Abstract. In view of the trends of increased traffic speeds, increase axle load, but also the reduction of the losses of material from the wheel and rail due to wear phenomena, the appearance of undulating wear, fractures of the wheel and rail, it is necessary to know the pressure distribution and the state of tension at wheel-rail contact. This can be obtained either by the finite element method (FEM). Using an elastic body modelling, a very rapid FEA model was obtained for solving the pressure distribution at concentrated wheel-rail contact, for real surfaces, modelling absolutely necessary for any study requiring analysis of the state of tension at such a request, for different situations of this interaction, modifying only the input data: geometric model of wheel and rail, longitudinal elasticity modulus, transverse contraction coefficient, load on wheel, etc.

1. Introduction

Taking into account trends in traffic speeds, increasing the axle load, but also reducing the loss of material from the wheel and rail, because of the wear phenomena, the appearance of undulating wear, it is necessary the knowledge of gradient of pressure and the state stress on the wheel-rail contact. One of the most important and difficult part of the railway engineering, is the interaction between wheel and rail, the kinematics of the vehicle in move as a result of an impact that generates a high force from the contact, which is a phenomenon highly complex and non-linear. The calculation of the forces in the contact manually, which takes place at the interface of wheel and rail, it is not possible, due to the complexity of the algorithm and the number of calculations. Fortunately, in the last three decades, it has developed and modernized the techniques for the numerical modelling of contact and of the resources of the computer, it has become possible to calculate the intricate problems of the wheel-rail interface.

Now, there are a wide variety of algorithms, from the modelling presented in the literature [1-5], which can solve the normal contact and tangential, and it also provides a detailed description of the surfaces in contact. The modelling tools are used for the calculation the state of stress of the contact at the interface of the wheel and rail, called the finite element Method (FEM). This method has advantages and limitations, and it may give you an understanding of the improved the different aspects of the interaction between the wheel and rail. In the last few years there has been a lot of attention to the improvement of the solution of the general problem of the wheel/rail contact. In this regard, the FEA provided an opportunity for detailed modelling, which permits analysis of the geometry of the realistic 3D geometries of the rail and wheel, will be presented in this paper. Moreover, it is not limited by the theory of linear as well as the theory of Hertz and the assumptions on the half-space. In
the calculations with the FEM, the geometry is meshing into a number of parts (finite elements) that are connected by nodes. Each element is made of three sets of equations: the equations of compatibility, which reports to the displacements Figure 1, the equations of the constituents, which links the stresses and strains.

The wheel-rail geometry Figure 2, dramatically influences the rates of wheel-rail wear and contact fatigue. In the wheel-rail contact, the separation between the contacting surfaces depends on a lot of variables what would it be: wheel profile, rail profile, rail inclination, track gauge, lateral shift of the Wheel Set, rail inclination and also.

Figure 1. Wheel-rail loads and relative movements [1].

Figure 2. Geometric elements defining wheel-rail contact [6].

Solving this set of equations, depending on the size of the mesh generally results in a fairly accurate description of the stresses and strains of the parts are modelled. Even though a complete from the wheels up with rails, sleepers, ballast, can be modelled only a very small part of the railway. In addition, since the contacts are of a magnitude to be very high in a very small area, the size of the elements in the zone of potential contact have to be very small, enough to achieve good accuracy. This
makes the analysis of the time-consuming, because for each element, we have solved a set of equations. Also, it has been shown that the results of the analysis of the FEM depends on the size of the elements of the mesh [7-11], which means that the size of the grid has to be chosen very carefully in order to obtain more accurate results. FEM is used to analyse the interaction between wheel and rail, and in order to evaluate the contact with the rolling stock in a variety of situations contacts parametrization, the geometric model and the mechanical properties of the material.

The wheel-rail (with the complex geometry of the contact) using the friction in order to turn it into a power; in the movement, in such a system, the degradation of the wear and tear of elements in contact, occurs by the phenomena of the contact of the rolling and the sliding during the movement, in the form of wear and tear, or the opening of cracks and their growth. The purpose of a railway engineer is to define and accurately modelling the problem. Therefore, an understanding of the mechanism of the contacts (stress/pressure distribution), it is important to estimate the behaviour and growth of cracks. The analysis of the contact of rail and wheel studied by many researchers in a variety of contexts, with different geometries of the rail and wheel to calculate the behaviour of the Hertzian contact, and in order to find out the distribution of stress in the area of interest. Consequently, they are being investigated, and the mechanisms of deterioration, such as cracks in the surface, plastic deformation, wear, and road noise. The deal the accurate simulation of these mechanisms, it serves to be clearly understood, a detailed knowledge of physical interaction between wheel and rail. In the literature [11-14]; some of these problems have been studied by using several experimental observations, analytical calculations, and the calculations in the FEA, in a variety of contexts. In order to make the numerical simulation efficient and robust, it is applied the following procedure, shown in Figure 3.

Figure 3. FEM Flow Chart.
Basically, the procedure consists of the interconnection between the three well-known programs, Autodesk Inventor 2020 for students, Space Claim for students and the ANSYS Work Bench for students, by bringing the modelling to be realistic and flexible in that it can be set up in all of the three levels. First of all, the profile of the wheel and the rail are modelled and parameterized in a CAD program, Autodesk Inventor 2020, strategy for the creation of the 3D model, which is able to detect all the contact properties. After that, the data model is imported into the software Space Claim for meshing of the 3D model with hex finite elements (FE). Subsequently, the pattern of the FEA is to be imported in the Ansys Work Bench (static structural) modelling static and the interpretation of the results (static structural). The results of the pressure and the state of stress are analysed, interpreted, and presented in this paper.

2. FEA Modelling and results

In the finite element model, elastic model is adopted to describe the elastic characteristic of wheel-rail material, which obeys Von Mises yield criterion. The Von Mises yield criterion is expressed by

\[
\sigma_e = \sqrt{\frac{1}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2]}
\]

where \(\sigma_e\) is the equivalent stress; \(\sigma_1, \sigma_2,\) and \(\sigma_3\) are the principal stresses; and \(\sigma_e\) is the yield stress of the material.

In this study were used the following input data, monobloc-wheel with 460 [mm] radius with profile: S78 [16], S1002 [17] and ВНИИЖТ[18], rail profile: [19], UIC 60 [19]and R65 [20], gauge: normal (1435 [mm]) [16], rail inclination: 1/20 rad (France, Romania, Republic of Moldova, Russia), 1/40 rad (Germany, Austria), 1/30 rad (Sweden, Norway), the distance between the inner faces of the two wheels: 1360 [mm], angle of attack: 0°, elastic material model: longitudinal elasticity modulus: 2.0•10^5 [MPa] (rail), 2.1•10^5 [MPa] (wheel), Poisson's ratio 0.28 (rail), 0.30(wheel), density: 7850 [kg/m^3], load on wheel - 90 [KN], rail is fixed on travers location, friction being null on the two directions (f_x = f_y = 0.0).

![Figure 4. CAD assembly of wheel – rail.](image1)

![Figure 5. Meshing of assembly](image2)

![Figure 6. Boundary and loading conditions.](image3)
The corresponding state of stress, strain and pressure distribution wheel/string contact are plotted in figures 7-9 for rail inclination 1/20 rad, wheel profile S78, lateral shift 3mm, elastic material model, load on Wheel 90 [kN].

Figure 7. The state of tension to a lateral shift of the wheelset of 3 mm.

Figure 8. The state of strain to a lateral shift of the wheelset of 3 mm.

Figure 9. The distribution of pressure on the 3D, a lateral shift of the Wheelset to 3 mm.

The lateral shift of the wheel and the rail inclination are a major factor in influencing the shape of the contact area, distribution of pressures, the state of stress/strain and location of the maximum stress/strain point. These influences are described in figures 10-11.
Figure 10. The comparison of maximum of pressure registered for different rail inclination and lateral shift of the wheelset the to -5, 0 to 3 mm.
Figure 11. The comparison of maximum of state of tension registered for different rail inclination and lateral shift of the wheelset the to -5, 0 to 3 mm.
3. Conclusions
Using an elastic body modeling, a quick FEA model was obtained for solving the pressure distribution, the state of tension at the concentrated wheel-rail contact, for real surfaces, a preliminary model absolutely necessary for any tribological study by modifying only the input data: the longitudinal elasticity module, the Poisson’s ration, the geometric model of the wheel-rail assembly.

The interoperability of rolling stock between countries is affected by different profiles of rail/wheel and the varies of canting of the rail.

The lateral shift of the wheel and the rail inclination are a major factor in influencing the shape of the contact area, the distribution of pressures, the state of stress/strain and the location of the maximum stress/strain point. FEA model showed the greater canting of the rail provide greater maximum pressures and stresses.

It was noted that the maximum pressure value and the maximum von Mises stress are greater for the situation where the S1002 profile is used on the UIC60 rail with the 1/20 rad canting of the rail, compared to the example in which the canting of the rail by 1/40 rad. For the canting of the rail 1/20 rad the good results present the wheel profile S78. Good results on the R65 rail profile presented the wheel profile VNIIJT (in Russian).

4. References
[1] Spiridon C and Constantin-Ioan B 2009 A Numerical Solver For The Wheel-Rail Contact Subjected To Combined Loads The annals of university “Dunărea de Jos” of Galați pp 73-78
[2] Wiest M, Kassa E, Nielsen J and H. Ossberger 2008 assessment of methods for calculating contact pressure in wheel-rail/switch contact WEAR vol. 265 pp 1439-1445
[3] Enblom R and Berg M 2008 Impact of non-elliptic contact modelling in wheel wear simulation WEAR vol. 265 pp 1532–1541
[4] Iazikov-Vladislav H 2004 Application of the model of non-Hertz wheel-rail contact to assess the dynamic properties of a freight diesel locomotive (Bryansk: PC “Bryansk Machine-Building Plant LLC”) (in Russian)
[5] Ayasse J B and Chollet H 2006 Determination of the wheel rail contact patch in semi-Hertzian conditions Vehicle System Dynamics: International Journal of Vehicle Mechanics and Mobility 43-3 pp 12
[6] Spiridon C2009 Contactul Concentrat Elastic-Plastic (Iaşi: Polytehnium).
[7] Ansys “Ansys” 2020 [Online] Available: https://www.ansyshelp.ansys.com [Accessed 25 Februarie 2020]
[8] Nam-Ho K 2015 Introduction to Nonlinear Finite Element Analysis (Springer)
[9] Daryl-L L 2016 A First Course in the Finite Element Method (Amazon.com Services LLC)
[10] ŞTEFAN-I M and DIANA-A B 2007 Introduction to the Finite Element Method, Iași: CERMI, (in Romanian)
[11] Ma Y and Markine V L 2015 A Numerical Procedure for Analysis of W/R Contact Using Explicit Finite Element Methods 10th International Conference on Contact Mechanics, Colorado USA
[12] Jingmang X, Ping W, Li W and Rong C 2016 Effects of profile wear on wheel–rail contact conditions and dynamic interaction of vehicle and turnout Advances in Mechanical Engineering, VIII-1 pp 1-14
[13] Sunil-Kumar S and Anil K 2015 Dynamics Analysis of Wheel Rail Contact Using FEA 12th International Conference on Vibration Problems, Uttarakhand
[14] Yongming L, Liming L and M. Sankaran 2007 Analysis of subsurface crack propagation under rolling contact loading in railroad wheels using FEM Engineering Fracture Mechanics 74 pp 2659–2674
[15] Yongming L, Brant S and Sankaran M 2005 Fatigue crack initiation life prediction of railroad wheels International Journal of Fatigue 28 pp 747–756
[16] STAS 7110 1994 Rolling stock for normal gauge railway wagons. Dimensions (in Romanian)
[17] UIC ORE S1002 Profile of Wheel
[18] GOST 10791 2011 All-rolled wheels. Specifications
[19] BS EN 13674-1 2011+A1:2017 Railway applications. Track. Rail. Vignole railway rails 46 kg/m and above
[20] GOST 8161-75 Railroad rails of the P65 type. Design and dimensions (with amendments N 1, 2, 3) (in Russian)