APPLICATION OF THE EXTENDED FINITE ELEMENT METHOD IN THE AIM OF EXAMINATION OF CRACK PROPAGATION IN RAILWAY RAILS

Dániel Bóbis*
Péter T. Zwierczyk
Tamás Máté
Department of Machine and Product Design
Faculty of Mechanical Engineering
Budapest University of Technology and Economics
Műegyetem rkp. 3., H-1111 Budapest, Hungary
E-mail: bobis.daniel.@gt3.bme.hu
*Corresponding author

KEYWORDS
Extended finite element method (X-FEM), Ansys Mechanical APDL, Stress intensity factor (SIF), Rolling contact fatigue (RCF), Head check (HC), Crack propagation, Railway.

ABSTRACT

In this research, a rolling contact fatigue (RCF) caused crack the so-called head check (HC) was studied by the extended finite element method (X-FEM). The formation and treatment of these cracks is a major problem in the railway industry worldwide. The study aimed to explore the capabilities of the X-FEM concerning examinations of crack propagation in rolling contact conditions. The paper introduces the X-FEM from a practical point of view, with an emphasis on the application of the method, rather than focusing on the exact values. Results showed that the method can be used adequately for such complex issues as examination of crack propagation. However, the possibilities of this technique are quite limited in practice yet due to technical reasons.

INTRODUCTION

In this paper, a finite element (FE) modeling technique is introduced, called extended finite element method (X-FEM). The aim of this study is to explore the possibilities of the above mentioned method in practice regarding the examination of crack propagation in the research area of railway wheel-rail connection. On both the wheel and the rail side micro-cracks can appear for different reasons that must be inspected and handled properly. Rolling contact fatigue (RCF) is one of the main reasons for crack initiations. The phenomenon is investigated widely but still has numerous unanswered questions. The overall purpose of the research is to gain a better understanding on the nature of these cracks.

Overview of the Examined Issue

This research focuses on the so-called head check (HC), which refers to a kind of multiple hairline cracks in the railhead. The formation of the HC is a typical manifestation of the RCF-caused failures of the railway rail. They are located on the gauge corner parallel to each other at a typical slanted angle as Figure 1 depicts.

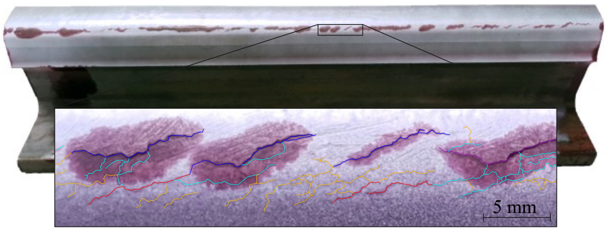

Figure 1: Typical HCs on a Part of a Rail. As a Result of a Former Liquid Penetrant Inspection, Cracks are Highlighted in Red. At the Bottom, Similar Cracks are Represented with the Same Color.

Csizmazia (Csizmazia and Horvát, 2014) summarized the initiation and propagation mechanisms of these cracks in addition to the methods used to eliminate them. In brief, HC usually appears in sharper curves on the outer rail where the wheel-rail connection is more adverse. Due to the plastic strain, significant hammer-hardening effect occurs on the upper layers of the rail. As the material hardens, its elasticity declines. When the highly concentrated loads exceed the resistance of the rail steel, micro-cracks initiate and start to grow towards the railhead. Regarding the stages of crack growth, fluid significantly influences the propagation process, since it can be forced to the crack by contact pressure, which produces hydraulic pressure inside the crack.

Spalling of the surface is usually caused by these micro-cracks, which can lead to the development of larger and more dangerous defects. Therefore, it is important to inspect and treat HCs properly. The treatment should be done at the early stages of the formation by removing the damaged layer of the rail, mainly via re-grinding processes. Detection systems are generally based on eddy current technology. The accuracy of these systems might be unreliable in some cases, hence the damaged layer is conservatively

Communications of the ECMS, Volume 36, Issue 1,
Proceedings, ©ECMS Ibrahim A. Hameed, Agus Hasan,
Saleh Abdel-Afou Alaliyat (Editors) 2022
ISBN: 978-3-937436-77-7/978-3-937436-76-0(CD) ISSN 2522-2414

overestimated. Since the elimination processes are highly expensive, the extent and frequency of the maintenance is a key issue from an economic point of view.

As there is a high level of uncertainty around the phenomenon, every study made on the subject can be valuable in order to understand the problem better and handle it more efficiently.

Methods

Linear Elastic Fracture Mechanics

All of the used methods are within the theory of linear elastic fracture mechanics (Anderson, 2005). It assumes that the material behavior is brittle and can be described by Hooke's law.

Cracks can be loaded by three different modes according to Figure 2. In mode I, the load is normal to the crack plane. Mode II corresponds to in-plane shear load loading and Mode III refers to out-of-plane shear.

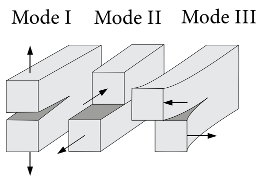

Figure 2: Crack Loading Modes (Anderson, 2005)

The stress field can be expressed via the K stress intensity factors (SIF) nearby the crack tip since it is a singular point. According to the crack loading modes, different SIFs can be distinguished. In this study, SIFs are calculated numerically.

Fatigue Crack Propagation

The typical fatigue crack growth behavior of metals is shown in Figure 3.

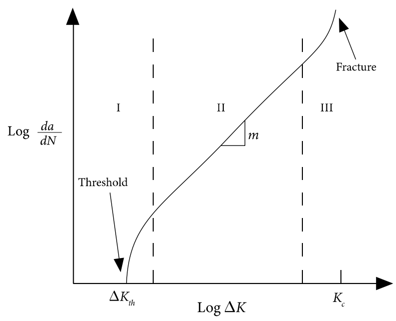

Figure 3: Fatigue Crack Grow (Anderson, 2005)

The range where this curve is linear (marked II) can be described with the commonly used formula called Paris' Law. According to the formula, the crack propagation rate is

$$\frac{\mathrm{d}a}{\mathrm{d}N} = C\Delta K^m, \qquad (1)$$

where C and m are material constants determined experimentally (Anderson 2005). Regarding this specific study, ΔK is derived from the SIFs calculated previously.

Extended Finite Element Method

The finite element method (FEM) is a very common and frequently used numerical tool applied in many areas of engineering. Its great advantage is that it can also be used for more complex geometries. However, the calculated results are strongly dependent on the quality of the mesh. Thus, in cases where the solution contains non-smooth behavior like singularities or discontinuities in the displacement field, the classical finite element method cannot be applied with sufficient efficiency. A promising method for modeling different discontinuities is the extended finite element method. The theoretical background of the method is well explained in (Koei, 2015) and in (Zhuang, 2014).

The basis of the X-FEM approach is the extension of the displacement field by special enrichment functions to describe the effect of discontinuities, including cracks. In general, the displacement field can be written as

$$\mathrm{u^h(x)} = \sum_I \mathrm{N_I(x)} \cdot \mathrm{u_I}(x) + \sum_J \Psi_\mathrm{J}(x) \cdot \mathrm{q_J(x)}, \qquad (2)$$

where N(x) denotes the shape function of the standard finite element, and $\mathrm{u_I(x)}$ is the standard degree of freedom (DOF). $\Psi_\mathrm{J}(\mathrm{x})$ is the enrichment shape function chosen for the specific discontinuous problem. In the study, this term describes the current geometry of the crack as well as the effect of the crack on the displacement field. The newly added DOFs are denoted by the $\mathrm{q_J(x)}$.

As a result of this kind of approach to the modeling of cracks, X-FEM has notable benefits. Cracks are independent from the FE mesh, thus neither remeshing is required during the analysis, nor drastic mesh refinement is necessary near the crack tip. Figure 4 shows the significance of it by comparing the classical and extended FE methods.

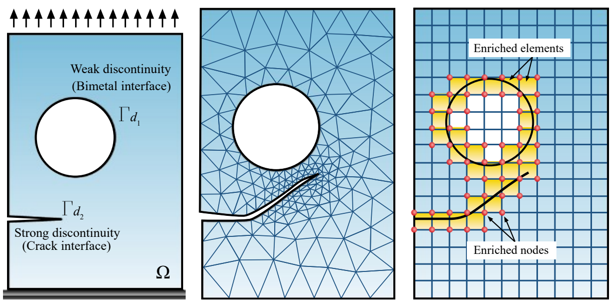

Figure 4: Modeling of Discontinuities with Different Techniques. The Picture in the Middle Shows an Adaptive (Classical) FE Mesh Which is Regenerated as the Crack Propagates. Significant Refinement is Noticeable Near the Crack tip. In the Right Picture, an X-FEM based Structural Mesh can be Seen with Far Fewer and Better Quality Elements (Koei, 2015)

Nevertheless, the use of the X-FEM modeling technique in practice is quite complex mainly due to the lack of a graphical user interface at least regarding the tools that were used in this study. Until now there is a limitation in both the available technical support and experience gained on the use of the method.

FINITE ELEMENT ANALYSIS

The aim of the research is to create a specific model by the application of the X-FEM, which is able to examine HC-type cracks in order to get a better understanding on the development of these cracks. Since the practical use of this method is quite complex, significant simplifications were taken during the research. There might be some inaccuracy in the input data applied. Consequently, the main emphasis is placed on the possibilities of the method, rather than focusing on the exact values of the results

Workflow and Software Environments

Regarding the X-FEM-based analysis, Ansys Mechanical APDL version 2020 R1 was used. Since the required functions are not yet available through the graphical user interface, the model was configured in a text file with Notepad++ and post-processed in Microsoft Excel. The workflow is shown in Figure 2. During the configuration, iteration steps were needed, because the enrichment area where the crack propagates is not known, as well as the optimal threshold value ($K_{\mathrm{eq,th}}$) where the propagation starts.

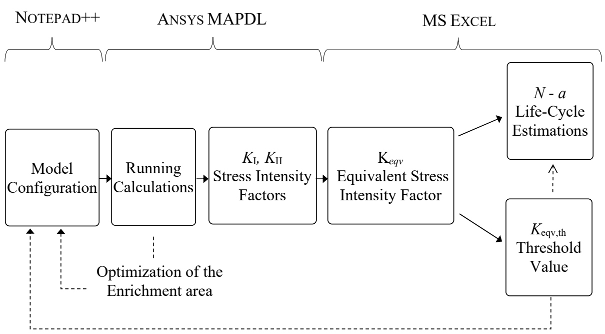

Figure 5: Software Products Used and the Workflow of the Analysis

Geometry

The examined geometry is two-dimensional in the longitudinal section of the rail with a predefined initial crack, corresponding to Figure 6 and Figure 7. The plane strain behavior of the model assumes that the extent of the crack is perpendicular to the section and relatively large. In contrast to this coarse approximation, head checks can be imagined as a countless number of spatial surfaces growing into the deeper layers of the real head at a slanted angle. In addition to this, they are typically located parallelly and close to each other. The connection between the surfaces of the initial crack is frictional. The frictional coefficient $\mu_{\text{crack}} = 0{,}15$ assumes dry conditions.

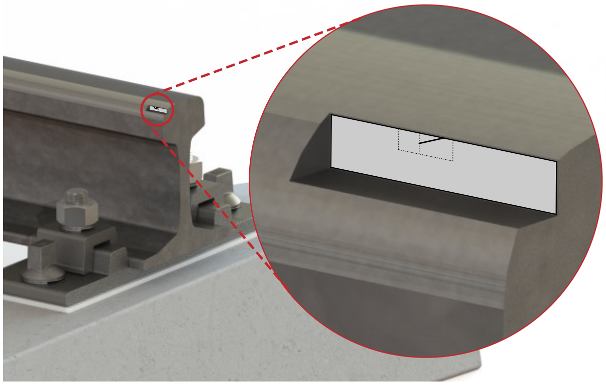

Figure 6: Interpretation of the Two-dimensional Geometry Examined

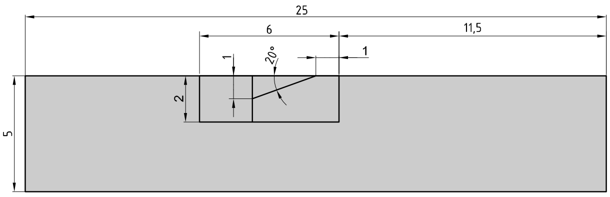

Figure 7: Dimensions of the Geometry Including a 1 mm Deep Initial Crack

Finite Element Mesh

The FE mesh consists of 0.012, 0.05, and 0.2 mm size 4-node quadrilateral-shaped elements corresponding to Figure 8. The large-scale mesh refinement was needed because loads were defined in time as a series of tiny steps, and in every load step 20 elements cracks in case the fracture criteria is satisfied. Special elements were used in order to describe the effect of the crack. Elements highlighted in yellow in Figure 4 have an enriched degree of freedom, thus they can crack. This yellow highlighted region is called the enrichment area, the extent of this region is critical. Red elements had already been cracked and were formed from the yellow elements. The green element has the crack tip which is the basis of the subsequent computations.

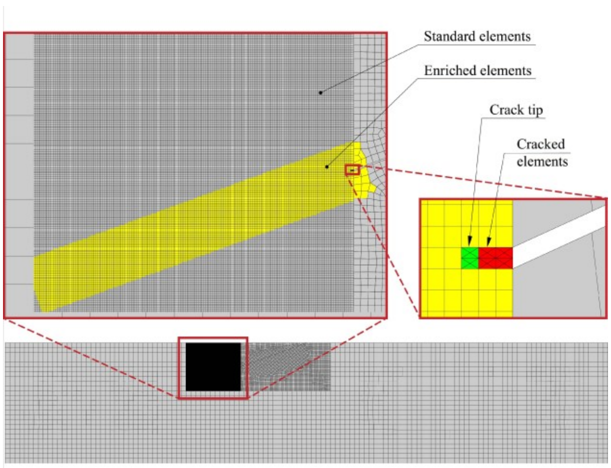

Figure 8: Finite Element Mesh with the Special Elements Describing the Crack

Loads and Boundary Conditions

The loading model assumes that the locomotive is accelerating and the frictional state between the wheel and the rail is at the limit of sticking. The contact stress distribution between the wheel of a passenger carriage and the rail was calculated by Zwierczyk in his Ph.D. dissertation (Zwierczyk, 2015). The stress distribution shown in Figure 9 was used during the analysis, assuming it does not differ significantly on the rail side.

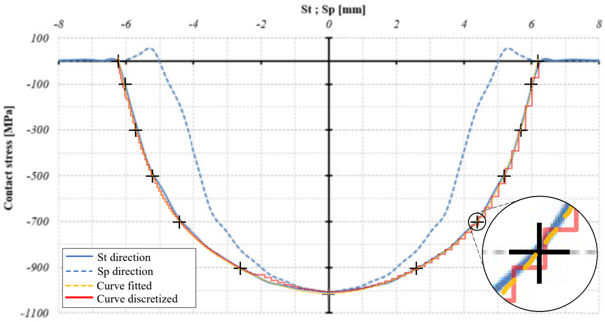

Figure 9: Contact Pressure Under the Wheel and the Discretization of it.
Adapted from (Zwierczyk, 2015)

A six-order polynomial curve was fitted to the marked points. Then, it was discretized by applying the right amount of pressure onto each FE element located on top of the examined geometry, using the equation of the curve.

The analysis is quasi-static, hence the movement of the loads is defined separately as a series of steps. The loads are moving from the left to the right in 25 load steps, each load step representing a position as Figure 10 shows. Above that resolution, results are not differing significantly, but the analysis would be computationally much more expensive. During the analysis, loads are passing through the geometry five times.

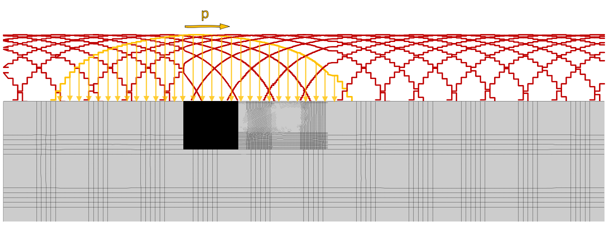

Figure 10: Representation of the Moving Loads. During the Analysis, Loads Highlighted Yellow are Changing Their Position According to the Red Curves.

Proportion to the contact pressure, frictional forces are applied as well. The direction of the frictional force is opposed to the direction of the movement. The frictional coefficient $\mu_{wheel\text{-}rail}$=0,15 assumes dry conditions.

Boundary conditions are shown in Figure 11.

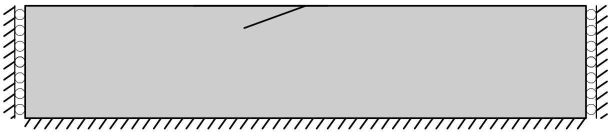

Figure 11: Boundary Conditions

At the bottom of the geometry, a fixed constraint is defined. On the sides, only the vertical DOFs are free. The latter might influence the behavior of the rail, but in this stage of the model, these constraints are acceptable. Wider geometry would require a lot more computational resources.

Material Properties

Material properties are tabulated in Table 1.

Table 1: Material Properties

Material Property	Value	Unit
Young's modulus	200 000	MPa
Poisson ratio	0.3	1
Paris constant (C)	$1.0\cdot10^{-8}$	1
Paris constant (m)	1.13	1

The values are correlating to that Zwierczyk used in his calculations (Zwierczyk, 2015). Paris constants were elected from a report by Aglan (Aglan, 2011), assuming that the rail steel is bainitic.

Further Analysis Settings

The duration of the load steps is 1 s. Since 25 load steps were defined over 5 cycles, the total time of the analysis is 125 s. For the sake of accuracy, each load step was divided into 20 equally spaced subsets. This results to be the crack propagation rate uniform over time, by splitting the same number of elements per step. Furthermore, it provides a satisfactory convergence of the nonlinear analysis.

The method calculates the SIFs in each step, and in case the predefined value is exceeded, then the splits of the elements result in a Δa crack increment. Life-cycle estimations are based on Equation (1), using the above-mentioned variables. Since the procedure gives a large number of ΔN cycles in each increment, the calculated lifetime is not directly correlating to the number of how many times the loads are passed through the geometry or to the size of the FE elements. The direction of the propagation is arbitrary and always turns into the critical angle which is recalculated in every moment.

RESULTS

The propagated crack in the fifth load cycle is shown in Figure 12. For the sake of illustration, it is shown in the position where t=105 s which is not the critical point.

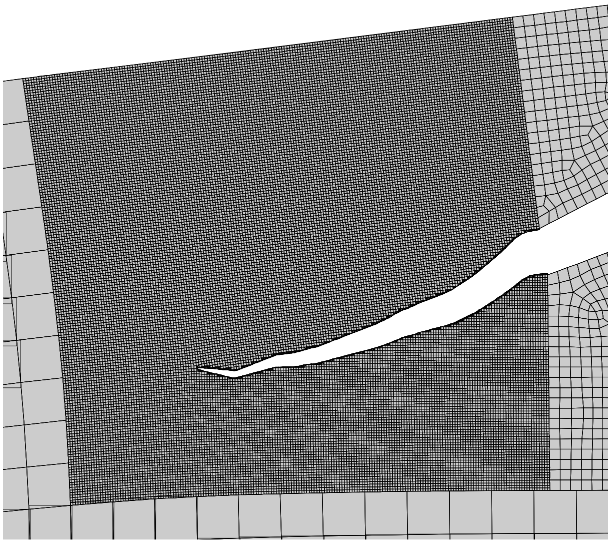

Figure 12: Crack Propagation in the Fifth Load Cycle. Scale of Deformations is 50:1

The characteristic of the distribution of equivalent von Mises stress around the crack tip is shown in Figure 13. It is shown in the critical time instant of the fifth load cycle. The values are not representing the reality, indeed, since the crack tip is a singularity and the material behavior is linear elastic.

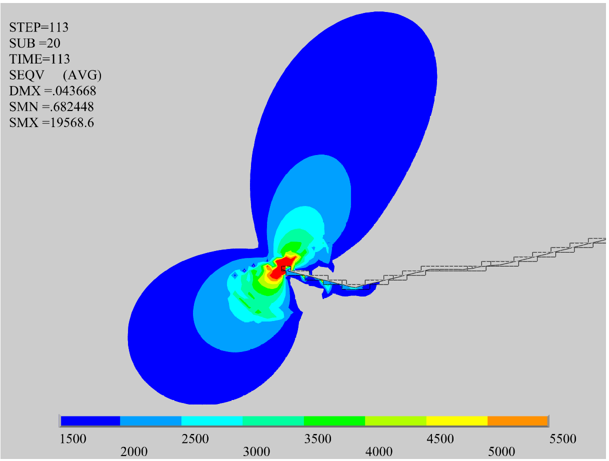

Figure 13: Equivalent von Mises Stress in MPa Around the Crack Tip

The stress state of the crack tip can be described via the SIFs. Changes in SIFs during the simulation are shown in Figure 14.

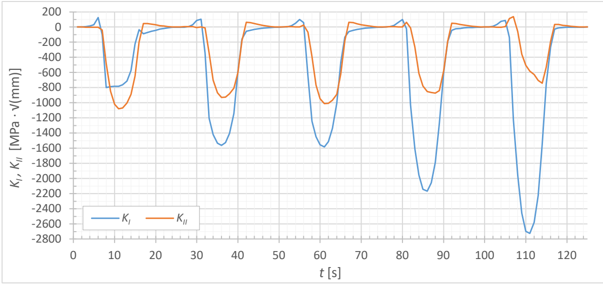

Figure 14: Stress Intensity Factors in Function of Time

It can be seen that the compressing Mode I crack loading is dominant, and the magnitude of it is increasing together with the crack length. As a consequence of it, the peak value of the equivalent stress intensity factor (K_{eqv}) is decreasing over time.

The equivalent stress intensity factor is calculated from SIFs. The positive values of the K_{eqv} are shown in Figure 15 in the function of the subsets which are proportional to time.

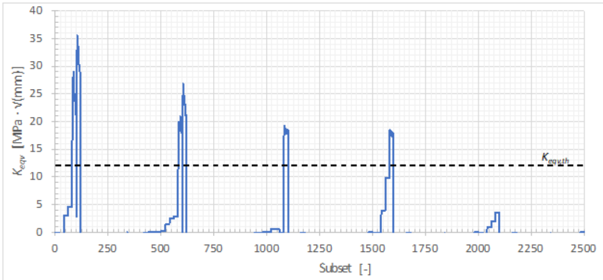

Figure 15: Equivalent Stress Intensity Factor in the Function of Subsets. Jagged Line Shows the Threshold Value Above Which the Crack Propagates.

After the fourth load cycle, the threshold value is not reached, therefore the crack stopped growing. This shows consistency with the reality, as the development of HC is driven by the trapped fluid (which was neglected in this study), by which hydro pressure tends to tear up the crack. In other words, the compression of the cracked surfaces is not that dangerous in the case of dry conditions from the crack propagation point of view.

The equivalent stress intensity factor is used directly in the lifetime estimations (K_{eqv}=K) based on Equation (1). The overall result of this is shown in Figure 16.

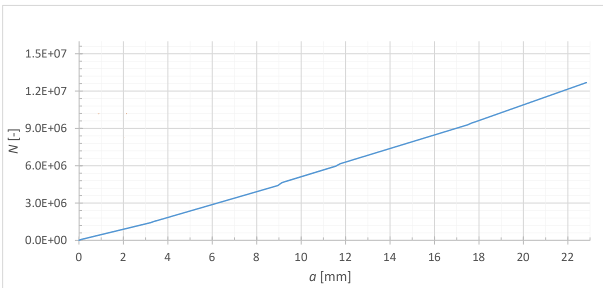

Figure 16: Life-Cycle Estimation Based on Paris' Law. N denotes the theoretical cycles, a is the crack length.

In Figure 17 a comparison of results of different examinations on HC can be seen. At the top, a non-destructive, liquid penetrant examination shows cracks on the rail surface. At the bottom, HC is shown in a cross-section of the rail under a microscope.

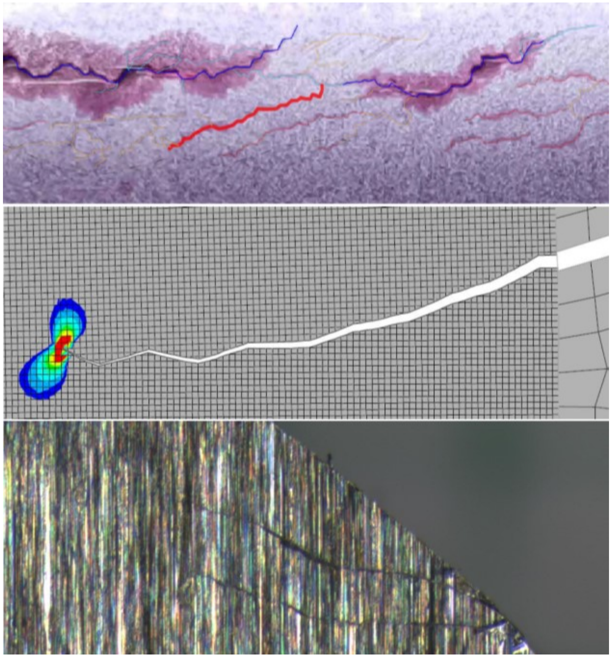

Figure 17: Comparison of the Laboratory and X-FEM Results

Even though these results are interpreted from different aspects and the numerical results are made with simplifications, the similarities are promising and prove that the phenomenon is worth to be examined from this kind of approach as well.

CONCLUSIONS

In conclusion, applying the extended finite element method in practice has met the expectations of the theoretical background. Based on the results of the above study this method has been proven to be adequate to examine such complex problems as the propagation of HC-type cracks. Using the extended finite element method arbitrary crack growth simulations can be implemented without the need of complicated meshing techniques and cracks can be examined in detail.

The software environment used (Ansys Mechanical APDL) is programmable, providing the possibility to define complex loading conditions and other specific parameters, which were essential in the study. The above environment provides the possibility to make fatigue-based calculations, so the method is able to predict lifetime as well.

However, it is important to mention that the method has numerous limitations in practice.

- Branching the crack is not possible, as such element-types are not supported.
- The contact of the cracked surfaces within an FE element is non-configurable, which might cause stability problems. By default, this is a frictionless, normal penalty.
- Loads and boundary conditions cannot be applied on cracked elements, which makes it difficult to take into account the effects of the trapped liquid.
- Regarding fatigue-based calculations, only Paris’ law can be applied..
- Material model must be isotropic linear and elastic.

Furthermore, the functions required for this method are not available from the graphical user interface at the moment, the model has to be programmed by text, which makes the configurational process quite complex and time-consuming.

Results in this paper may not fully cover the real behavior of HCs, since the aim of the research is to gain experience on the X-FEM. Due to the above mentioned difficulties some significant simplifications were taken during the analysis, which may have an impact on the behavior of the HC:

- Cracks were examined in plane
- Only an extracted crack was examined, the possible effects on each other were neglected.
- The effect of the trapped and pressurized fluid was neglected

Besides above-mentioned simplifications, the validity of Paris' law might be also questionable, since the RCF phenomenon differs from the classical fatigue problems at some point. Regarding future research, results should be further validated

Theoretically, the method and tools allow the extension of the simulation to three-dimensional space. However, based on this study, such complex tasks as the examination of the HCs expanded in space can be managed in this way only taking into account considerable doubts. The study showed clearly how significant and important it is to support researchers and engineers with user-friendly software products.

Due to the fact that the presented model is coarse and significantly simplified, numerous recommendations can be identified for further development of the model. For instance, it would be important to model the trapped and pressurized fluid between the cracked surfaces. More cracks should be defined and examined simultaneously, as well as the input data should be determined more precisely, especially in case of the loads and material properties.

Since it is hard to find better numerical tools for this kind of problems, it is worth implementing the above-recommended developments.

The relevance of the FE analyses is that they could provide specific information for more accurate maintenance plans in the future.

REFERENCES

Aglan H. A. 2011. ''Fatigue Crack Growth and Fracture Behavior of Bainitic Rail Steels''. Technical Report. Tuskegee University, Tuskegee. 57 p. Available: https://rosap.ntl.bts.gov/view/dot/23497/dot_23497_DS1.pdf?

Anderson, T. L. 2005. ''Fracture Mechanics: Fundamentals and Applications''. Boca Raton: CRC Press. 640 p. ISBN 978-1-4200-5821-5.

Csizmazia F. and Horvát F. 2014. (In Hungarian) ''A sínfej-hajszálrepedések műszaki és gazdasági alapú kezelése''. *Sínek Világa.* 56 no. 5. pp. 13-21.

Khoei A. R 2015. ''Extended Finite Element Method: Theory and Applications''. Pondicherry: SPi Publisher Services. 584 p. ISBN 978-1-118-45768-9.

Zhuang Z. et. al. 2014. ''Extended Finite Element Method''. Waltham: Academic Press. 286 p. ISBN 9780124078567

Zwierczyk P. T. 2015. ''Thermal and stress analysis of a railway wheel-rail rolling-sliding contact''. Ph.D. theses, Budapest University of Technology and Economics, Budapest.

AUTHOR BIOGRAPHIES

DÁNIEL BÓBIS is a final year M.Sc. student at the Budapest University of Technology and Economics, studying mechanical engineering. He started to investigate cracks in railway rails by participating at a Students' Scientific Conference in 2020 and is willing to continue the research. His email address is bobis.daniel@gt3.bme.hu.

PÉTER T. ZWIERCZYK is the deputy head of the Department of Machine and Product Design and assistant professor at Budapest University of Technology and Economics where he received his M.Sc. degree and then completed his Ph.D. in mechanical engineering. His main research field is the railway wheel-rail connection. He is member of the finite element modelling (FEM) research group. His e-mail address is z.peter@gt3.bme.hu.

TAMÁS MÁTÉ is a PhD student at the Budapest University of Technologies and Economics Department of Machine and Product Design where he received his M.Sc. degree in mechanical engineering. His research field is engaged to crack propagation in railway wheels and rails. His e-mail address is mate.tamas@gt3.bme.hu.