Modelling a fatigue crack propagation using eXtended Finite Element Method in Abaqus.

Kevin Cacarin Cumbal

University of Twente, Faculty of Engineering Technology, Drienerlolaan 5, 7522 NB, Enschede, The Netherlands
**corresponding author: k.m.cacarincumbal@student.utwente.nl*

ABSTRACT: The train industry relies heavily on factors such as safety, maintenance, and comfort. These factors are contributing to trains remaining a popular and busy method of transportation until modern times. Safety is a fundamental part that constantly reinforces and improves the rail service. One of the problems that most notably affects the rails is the concentration of stresses due to the continuous wagons that pass over them every day. Over time, it causes cracks and even more crucial damage to the rails. This research will evaluate the crack propagation in standardized parts UIC60 for the rail profile and wheel profile s1002.

Key words: Abaqus, crack propagation, eXtended Finite Element Method, fatigue, railways.

1 INTRODUCTION

Trains are a fundamental part of the development of society, the mobility of people, and the transportation of raw material between cities and even countries. Consequently, the train companies have increased the requirements and the regulation for safety, comfort, and capacity to satisfy the increased desires of the travellers.
The suppliers and the manufacturers of the trains are in charge of solving and bringing the practical option to increase the durability and reduce the future cost of maintenance plans. They are improving the efficiency, the velocities, and the trains' size. Trains are becoming a popular option among travellers because of the economic cost of tickets and the low carbon footprint. (The 100% percent of the trains in the Netherlands are powered by wind renewable energy [1]).
Furthermore, safety is a crucial aspect to focus on to continue ensuring the success of the railway industry. One of the principal mechanical failures is the weakening of the material properties due to cyclic load because of the contact of locomotive wheels with the rails. As a consequence, fatigue crack can initiate on the microstructure of the material until the breaking of the specimen [2]. It is a significant threat to travellers, and therefore a rise in the cost of maintenance. The fatigue can lead to costly rail grinding, replacement of rails and even complete rail failure. According to Popovic [3], the standard life cycle of the rails is reduced by 2-3 years due to fatigue problem if maintenance is not performed on time. This mechanical failure on rails is identified as rolling contact fatigue. The focus of this research is to model a fatigue crack propagation system using software and find a relation between the number of cycles and the crack length. However, it is a challenging issue to solve because of the many aspects that influenced this mechanical damage. This topic has been addressed by previous researches using different approaches; for instance: There is research in which the strain energy density factor criterion is used to evaluate the crack formation. The strain energy is introduced because it describes the change of the energy density with respect to a volume/shape change [4].
As an alternative, Palaniswamy & Knuss [5], proposed a method using the energy release rate criterion to identify the crack. The research was done in 2D and using linear behaviour. They suggested evaluating the energy balance when the crack propagates because the work done at the cohesion forces must be in equilibrium with the unloading tractions. Recent research analyse the crack propagation on rails using the maximum circumferential stress criterion. The crack propagation direction is obtained with the averages of the crack angles. This criterion follows the Weibull distribution [6]. However, using the previous researches as a start point, this research paper will focus on creating a model to analyse the effect of the crack propagation on the rails assuming non-linear mechanical behaviour. It will be modelled in Abaqus using the eXtended Finite Element Method (XFEM). This research will help contribute to the knowledge about the fatigue crack in the railway industry to help

the transition to a new maintenance plan. Using the simulations as an alternative method in combination with the traditional.

2 METHOLODOLOGY

This research aims to develop a simulation model that can describe the crack propagation with a defined initial crack location over the railways due to the cyclic load. Furthermore, find the relation of the number of cycles and the crack length in contrasting scenarios. In order to achieve the objectives of the research, the following steps were defined to complete the research successfully Figure 1.

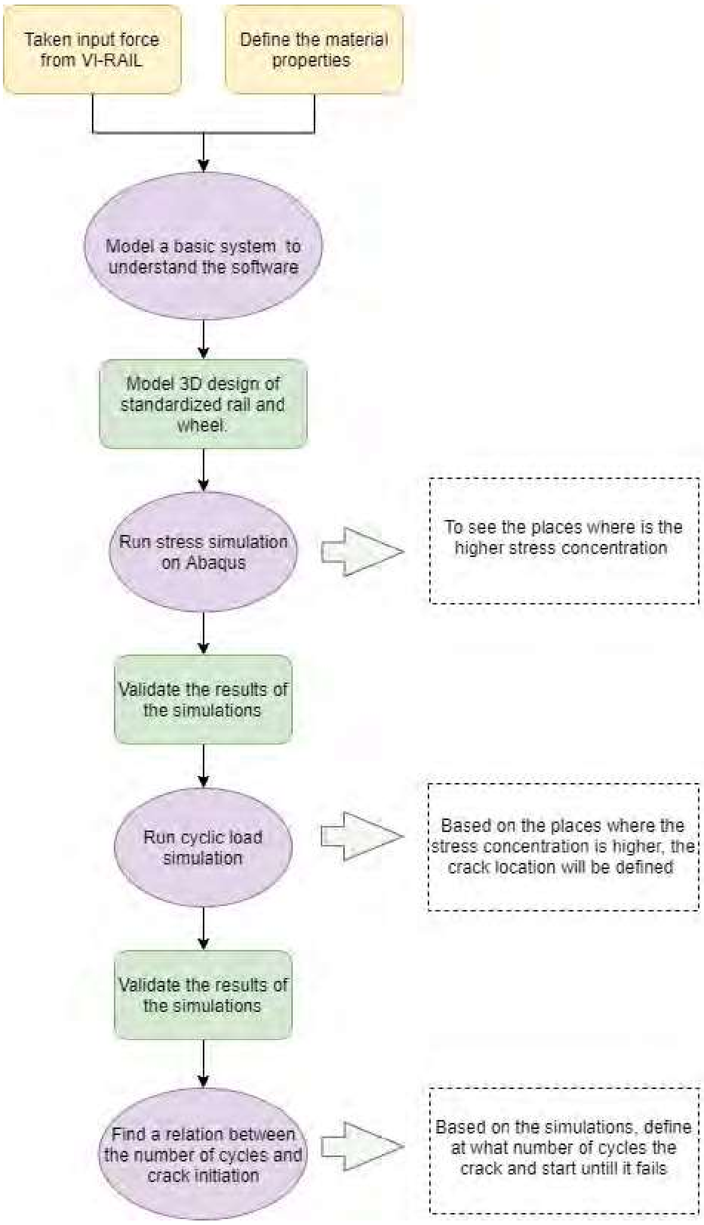

Fig. 1: Methodology scheme.

2.1 Material Properties

To select the appropriate material for the simulation. Standardized parts are used for this research. The rail profile UIC60 was selected with a steel grade R260, and for the wheel a s1002 profile. The chemical composition of the steel grade R260 is given in Table 1, and the mechanical properties are shown below [7]:

- Yield stress $\sigma_y = 533$ MPa.
- Ultimate tensile strength, $\sigma_{UTS} = 924$ MPa.
- Elongation at fracture 12%.
- Strain hardening coefficient $n = 0.243$

Table 1: Chemical composition.

Symbol	**% by mass**
C	0.62/0.80
Si	0.15/0.58
Mn	0.70/1.20
P	$\leq$ 0.025
S	0.008/0.025
Cr	$\leq$ 0.15
Al	0.004
V	$\leq$ 0.03

2.1.a Parameters

For the simulation in Abaqus additional parameters are necessary to describe the effect of the fatigue crack propagation. The data required is the damage evolution parameters, those values provide information of the material that characterizes the evolution of the damage until reaches complete failure. Those parameters can be obtained from the tensile test curve.

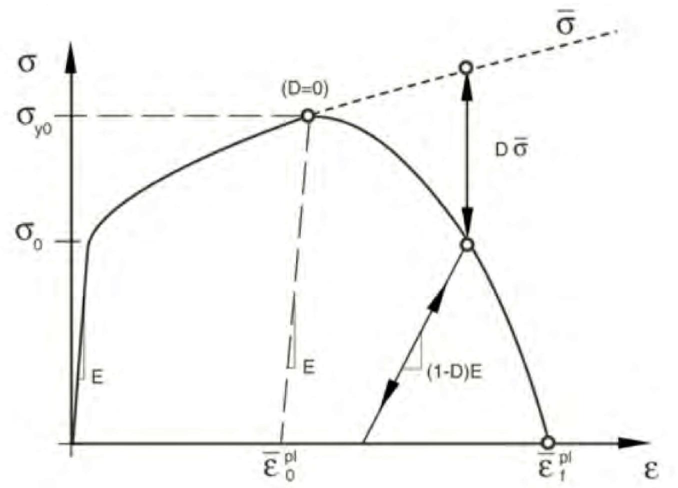

Fig. 2: Stress-strain curve with progressive damage degradation [8].

In Figure 2 illustrates a traditional tensile test curve, where four regions can be identified. The first region is the undamaged or elastic region. The limit of this region is the yield stress σ_{y0}. The next region start at yield stress and end in the ultimate tensile strength σ_{UTS}. It is called the damage initiation criterion zone. Beyond this point, the stress is reduced, and the curve depend on the material properties until it breaks. The damage is represented by D while the dashed line means zero damage (No damage evolution). $\bar{\epsilon}_0^{pl}$ and $\bar{\epsilon}_f^{pl}$ are the plastic strain and the plastic strain at failure, respectively.
The tensile test curve for steel grade 260 can be plotted using the mechanical properties defined in section 2.1. With the intention to obtain accurate values, the Ramberg-Osgood relation (Equation 1) is used. This equation is used to approximate the stress-strain curve as a function of stress.

$$\epsilon = \frac{\sigma}{E} + 0.002(\frac{\sigma}{\sigma_y})^{\frac{1}{n}} \tag{1}$$

In Equation 1 using the yield strength σ_y, Young modulus E, the strain hardening coefficient n and a range of stresses the curve is found. In this case, the yield strength is assumed as 0.2% of the total elongation.

$$\epsilon_{yield} = \frac{\sigma_{ty}}{E} + 0.002 \tag{2}$$

$$\epsilon_{UTS} = \frac{\sigma_{UTS}}{E} + \epsilon_f \tag{3}$$

Furthermore, following the Equation 2 and Equation 3 the strain value at yield strength and at ultimate strength are found. With this information, the tensile test curve is plotted. The Figure 3 is the actual stress-strain curve extracted from [7].

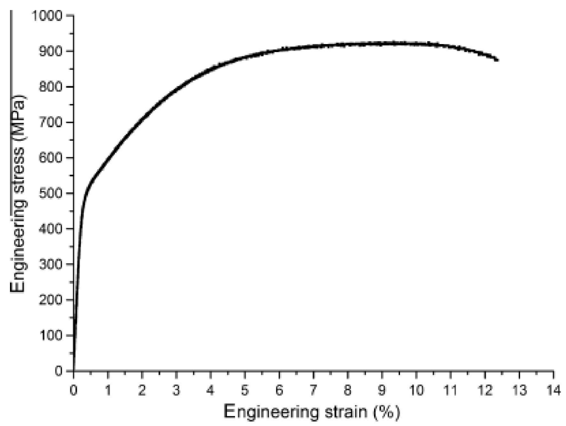

Fig. 3: Engineering stress vs strain [7].

From this curve, the value of fracture energy can be found. The values are approximations because the tensile curve is an estimation of the real data. The fracture energy is the area under the strain from the ultimate tensile strength until the maximum strain and multiplies by the characteristic length of the meshing [9].

- Fracture energy = 9153 N/m

The resulting value is the required information to simulate the crack propagation in Abaqus.

2.2 *Software*

All the mechanical parameters for the simulation were defined. However, there is still a missing input parameter. Which is the cyclic load applied over the rails. The input forces will be obtained from a simulation software. In this section, All the software used in this research will be briefly explained.

2.2.a VI-Rail

It is a software focused on realistic railcar simulations. The users can study, improve, and refine the performance of the rail-wheel contact. It allows to create a parameterized model of a real train.
Static analysis, dynamic analysis, track load, analysis of passenger comfort can be done on VI-Rail. In this research, a wagon, bogies and rails were selected from the predefined library of VI-Rail. The parts were assembled to form a basic train system.

After dynamic simulation, the normal contact forces can be extracted. For this research, different scenarios were selected to evaluate the effect of fatigue crack propagation in distinct conditions. The scenarios will be changed in parameters to see the influence on the crack propagation. The parameters that will change are:

- 1. Speed $[km/h]$
- 2. Wheel radius $[m]$
- 3. Rail profile [-]
- 4. Coefficient of friction [-]
- 5. Total mass $[kg]$
- 6. Curved radius$[m]$

For this research, 5 scenarios will be analysed to evaluate the influence of the parameters mentioned above. The scenarios are shown in Table 2.

Table 2: Scenarios table.

PARAM	**SCENARIOS**				
	1	2	3	4	5
1	130	130	80	130	130
2	0.42	0.42	0.42	0.42	0.42
3	72250	72250	72250	86700	72250
4	54E1	54E1	54E1	54E1	54E1
5	0.4	0.4	0.4	0.4	0.6
6	5000	500000	0	0	0

After applying the conditions of the different scenarios in VI-Rail. It results in a set of forces that is composed by the wheelset and its location. For this research, the first front wheelset from the left side is considered because from the results, it is the highest value in comparison with the other wheel sets locations. The overview of the input force extracted from VI-Rail is illustrated in Figure 4. These are the inputs forces need to simulate the effect of the fatigue crack on the rails in Abaqus.

2.2.b ABAQUS

Abaqus is a simulation software suitable for finite element analysis, fluid dynamics and crash impacts. The nonlinear effects can be represented and analyse. The purpose of this research is to find the location of

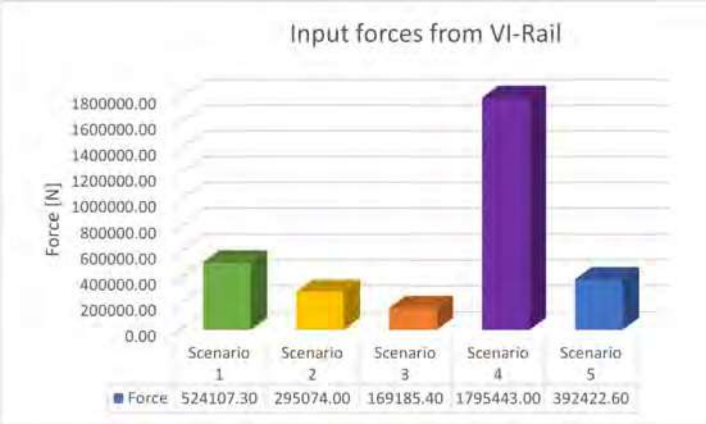

Fig. 4: Forces from the different scenarios.

crack initiation and propagation based on the number of cyclic load. In order to obtain those results, three simulations cases will be performed.

- **Wheel-Rail contact** to analyse the stress concentration on the rail and defined the location of the *crackset.*
- **Cyclic load** apply cyclic loads over the rails to see the effects.
- **Fatigue crack** Simulate the crack propagation over the rail varying the number of cycles and the scenarios.

Wheel-rail contact analysis

The fundamental part of the research analysis is the contact between the wheel and the rail. The contact will be simulated using the properties defined before to evaluate the location of higher stress concentration. The model is formed by the geometry of rail profile UIC60 rail and the wheel profile s1002. Abaqus will perform the simulation. To reproduce the motion of the wheel over the rail, a dynamic explicit option was used. Dynamic explicit is an Abaqus option to analysis, model that have low dynamic response times.

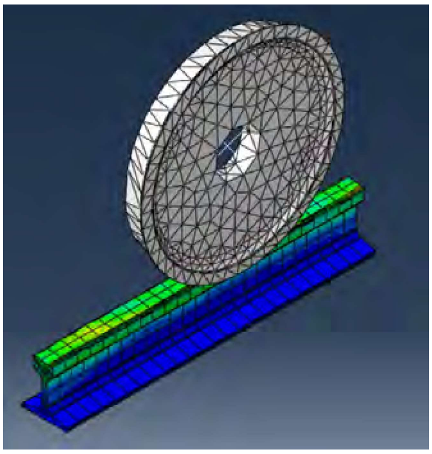

Fig. 5: Wheel rail contact simulation.

In Figure 5 the schematic representation in Abaqus is shown. To simulate the effect of the wheel-rail contact, a constant velocity was applied to the wheel of a magnitude of 200 km/h. The total time of the simulation was set to 10 seconds. Using the meshing tool of the preprocessor of Abaqus, the rail is divided into 3100 small squares to perform the simulations and the wheel into 23282 elements. The number of elements applied is related to the accuracy of the results and the computational time.
After applying the already defined condition, the analysis was done. Due to meshing, the rails are divided into various elements. It means that each element will be analysed separately, by cause of the complexity of the results and the understanding of the resulting graphs, this research will focus on a specific set of elements. This set will be selected base on the location of the higher stress concentration. This set will be called *crackset* in this research.

Cyclic Load

With the aim of simulate the fatigue crack on the rails, it is necessary to apply a cyclic load to the sample. The cycle load will be represented using a sine function. A period of the function is equivalent to one cycle. After a X number of cycles, the crack can initiate and even propagate. For this simulation, the statics method is selected. It can be used for linear and non-linear effects where the inertia is neglected. This method was selected because of the simplification of calculations and the lower computational time.
The total time for the simulation analysis was set to 10 seconds. A sine function with amplitude of one and period of $2 \cdot 10^{-11}$ $[1/s]$ is used to represent the cyclic load, it results in $5 \cdot 10^{11}$ cycles applied to the sample.
From the VI-Rail, the normal contact force was taken for the simulation. The effect of load on the sample will be described in the next section.

Fatigue crack

In order to simulate the crack propagation, different scenarios (see Table 2) will be analysed in this research. The selected model used the geometry of the rail profile UIC60 and the wheel profile s1002. It is chosen because of the simplicity and the reduced simulation time, the quasi-static analysis is selected to perform the analysis. It is used to analysis models with time-dependent material response and the effect of non-linearity is allowed, furthermore, Abaqus only work with quasi-static analysis to run XFEM tool.
Abaqus is a simulation software capable to simulate the crack formation and propagation over a material. For this research, the method selected to analyse the fatigue crack is the extended Finite Element Method.

eXtended Finite Element Method (XFEM)

It is a numerical method which is an extension of the traditional finite element method (FEM). The advantage of XFEM is the use of discontinuous function to solve differential equations. XFEM was developed in 1999 by Ted Belytschko and collaborators [10].

$$\mathrm{u}^h(x) = \sum_{I\in N} N_I(x)\left[u_I + \left[sgn(x)a_I\right]_{I\in N_H} + \left[\sum_{\alpha=1}^{N_{tip}} F_\alpha(x) b_I^\alpha\right]_{I\in N_{tip}}\right]$$

The equation above describe the mathematical definition for XFEM, where $N_I(x)$ describe the shape function used in elements, u_I is the degrees of freedom (DOF) of the nodes, a_I and b_I^α are enriched DOF in the elements used for the implementation of discontinuities. The Heaviside distribution is represented by $sng(x)$. The nodes that are part of the element cut by a crack is N_H, while N_{tip} are the nodes part of element where the crack tip lies. The crack tip asymptotic function is $F_\alpha(x)$ [11].
The main benefit of using XFEM in comparison with the traditional FEM in domains such as: interior boundaries, discontinuities, singularities and failures is the not remeshing process. FEM uses a uniform meshing, it produces difficulties and incoherence to solve problems [12]. XFEM is an added tool on abaqus environment. It is commonly used to model

the propagation of mechanical failures of materials, such as strong deformations(cracks) and weak (material interfaces).The failure start to growth element by element [13]. Therefore, less computational time is needed to complete the whole model. XFEM option is only available in quasi-static analysis. This approach uses the discontinuous function, which is used to capture the singularity of the stress field near the crack tip, it is illustrated in Figure 6.

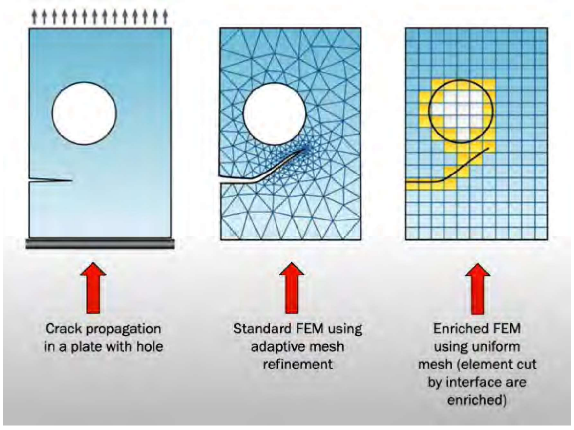

Fig. 6: Crack propagation using FEM.

Simulation analysis

The analysis for the crack propagation is performed on a basic beam that uses the dimension of the UIC60 rail with a length of 1 meter. The crack was defined in the position of the *crackset*. It is the location of the highest stress concentration. The crack is located in the *crackset* with an initial length of $1 \cdot 10^{-4}$ m and a depth of $1 \cdot 10^{-}3$ m. Moreover, the boundary conditions were defined at the bottom, with the constraint of translations and the rotation. The load is applied on the top of the beam with the already defined sine function.

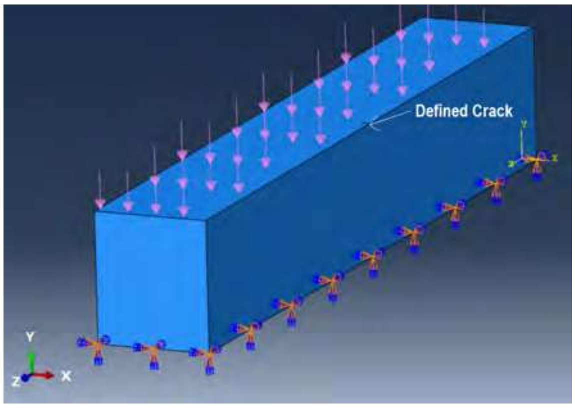

Fig. 7: Constrained beam with applied load and crack.

3 RESULTS

The simulations for all the scenarios were performed using the mechanical properties and the parameter calculated in section 2.1.a. In this section, the results obtained will be presented and evaluate.

3.1 Wheel-rail contact results

The results of the simulation are shown in Figure 8. The stress variation on the rail is visible by the different colours, where the red colour is the place of the highest stress concentration. The location is indicated by an arrow. This will be the selected set for further analysis, called *crackset*.

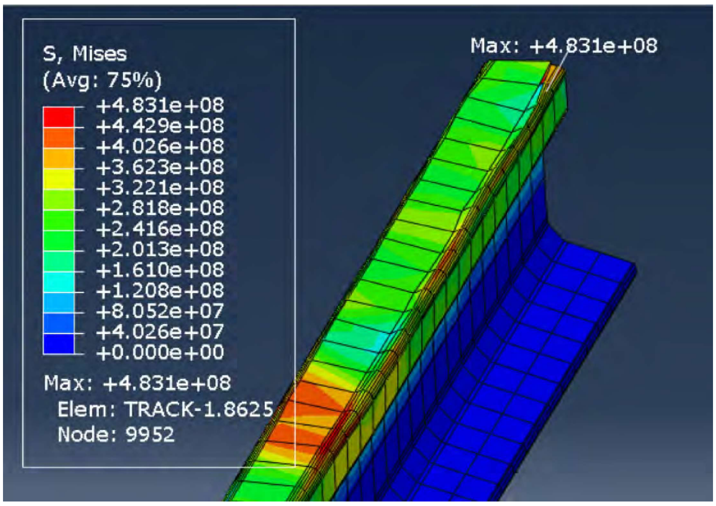

Fig. 8: Stress concentration on rail UIC60.

As it was explained before, the *crackset* will be selected to narrow the analysis into a specific area instead of processing all the elements. The *crackset* will be selected on the corner of the rail, because the stress concentration on that part is the highest in comparison with other points.

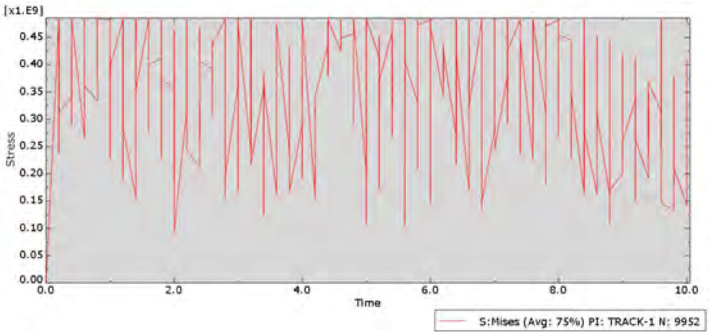

Fig. 9: Von Misses stress vs time of the *crackset*.

The variation of the stress in the *crackset* is illustrated in Figure 9. From this figure, the maximum Von Mises stress value was extracted. It is $4.83 \cdot 10^8$ Pa. Following the research of Aalami, Anari, Shafighfard and Talatahari [14], which is focused on the stress calculation in three-dimensional real size geometries with Hertzian contact theory.

The stress distribution on the rail was found. To validate the results of this research, the same initial conditions were applied to compare the outputs. From the research [14], the maximum Von Mises stress is about $5.09 \cdot 10^8$ Pa. The results obtained in the wheel-rail simulation in this paper are close (less than % 10 of difference). However, the exact solution could not get found because of the missing information about density and velocity in the paper of Aalami and others. In order to get a more precise solution, the same inputs may be used. Therefore, the results of this simulation are considered as valid. The *crackset* is defined and in the further analysis will be focus on it.

Using the scenarios shown in Table 2, The same analysis is performed to locate the highest stress concentration point on the other scenarios. Following the result of the simulation for the wheel-rail contact, the location of the highest stress concentration in each scenario is comparable to the Figure 8. This behaviour might be produced by the velocity, because it is the only input value used to run the simulations. Furthermore, the geometry of the rail play a fundamental role. The rail is modelled as beam, from definition the stress concentration have the maximum value close to irregularities such as: notches, fillets and in this case corners [15].

3.2 *Fatigue crack and cyclic load*

With the definition of the *crackset* for the different scenarios, in this section, the results for the cyclic load effect on the rails will be described. The time duration and the number of cycles for each specimen were defined in the section. Using the scenarios from Table 2), the number of cycles necessaries to propagate the crack will be analysed based on the variation of the initial parameters. The results are shown in Figure 10.

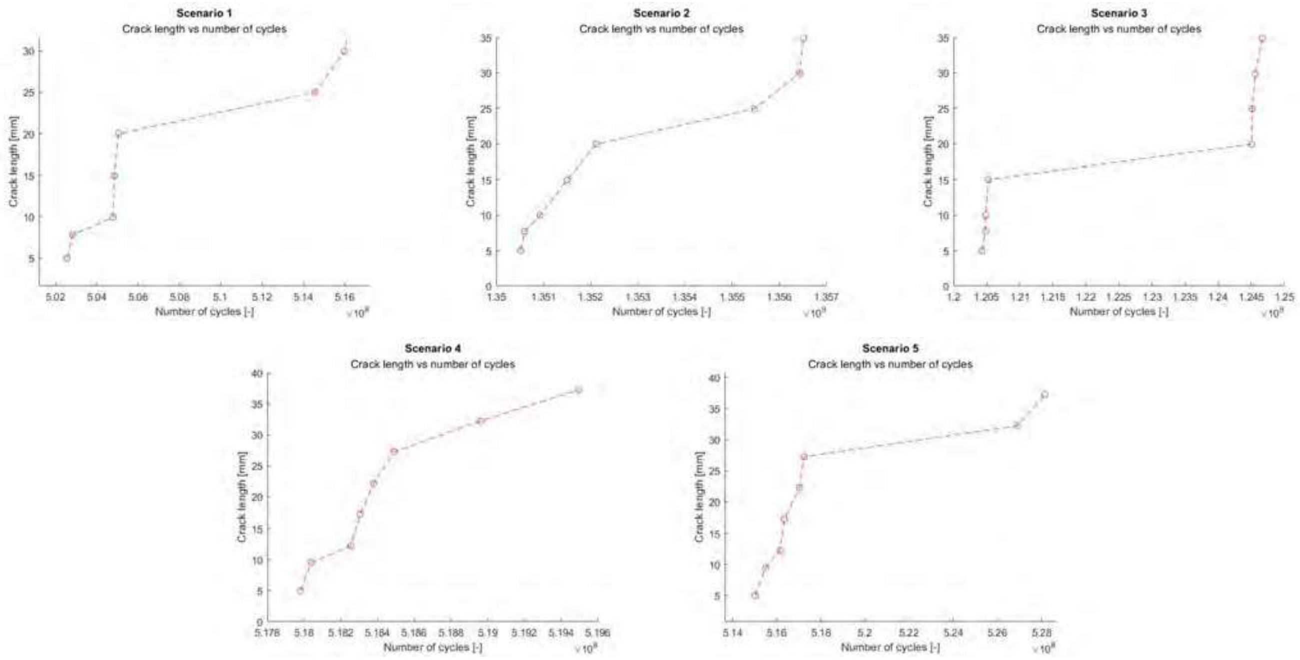

Fig. 10: Crack length vs number of cycles from different scenarios.

For each scenario, nine samples were selected to plot the graphs. The reason of the reduced number of samples. It is the limited software capacity available to run the simulation. The crack length is considered until it reaches a 35 mm. It is considered as visible for the naked eyes. The Figure 11 illustrated the amount of cycles necessary to reach 35 mm for each scenario.

Fig. 11: Number of cycles to reach 35 mm.

The representation of the crack propagate path is illustrated in Figure 12. In this figure is used PHILSM function which is used to define the path of the crack between the mesh elements.

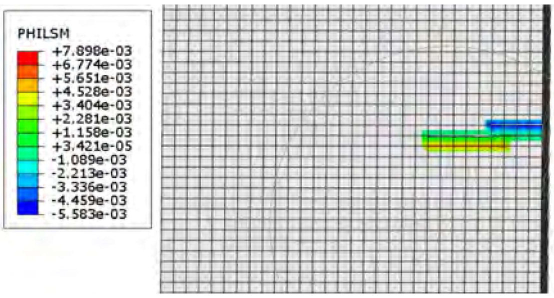

Fig. 12: Crack path.

The effectiveness of the results calculated using XFEM tool on abaqus is supported by [16]. In this research, the XFEM technique in Abaqus is tested using different parameters in 2-D and 3-D elastic cases. The advantages of the use of XFEM is explained in relation with mesh size when the crack propagates is simulated.

4 CONCLUSIONS

The objective of this research was to create a model in Abaqus using the eXtended Finite Element Method to simulate the effect of the fatigue crack propagation on distinct scenarios. Based on the results and the processes explained in this paper. It can be concluded that the model was created, and it works correctly. The results from the simulation are given in Figure 10. From these results, the effect of each parameter can be evaluated to see which affect more in the crack propagation.

After the analysis, the simulation for the five different scenarios where the parameters mentioned in section 2.2.a were changed to see the effect on the fatigue crack propagation. It can be concluded that vary the mass of the train system has a big impact on the input force as is shown in Figure 4 where scenario 4 has the highest value in comparison with the other scenarios with reduced masses. Using the input forces and the crack propagation rate, a relation is found. The scenarios 1,4 and 5 have the higher input force but also the crack propagates faster than in other scenarios. This relation is displayed in Figure 11. Where the number of cycles to reach a crack length is less in comparison with the other scenarios. From the Figure 4, increasing the mass of the scenario 2 by a 16.6% give as result the scenario 4. This increment in mass result in huge increment in input force, however, in scenario 4,there is also a reduction in the curved radius. To summarize, the speed and the mass of the trains are proportional to the input forces. The increment of those parameters can produce a huge increment in the input forces, and also resulting in fast crack propagation. Meanwhile, the curved radius is inverse proportional to the input force. On the other hand, scenario 2 and scenario 3 required a higher number of cycles to reach 35 mm of crack length. The curved radius play a role in the crack propagation rate. Despite the effort spent in this research, the limited time and the limitations of software, this research was focus on a reduced number of scenarios, and not the conditions were covered.Other types of rails, different scenarios, shear stress, inertia and bending were not covered within this research, Those are interesting topic to analyse in the future researches.

ACKNOWLEDGEMENTS

First of all, I would like to express my sincere gratitude to SENESCYT to financially support me during my three years of study. Further, I would like to thank my supervisor Annemieke Meghoe for the thoughtful comments and recommendations on this dissertation. I am also thankful to the University of Twente and all its member's staff for all the considerate guidance and knowledge taught. I cannot forget to thank my family despite the distance, they always encouraged me to achieve my dreams. To conclude, I deeply thank my friends that became my new family in the Netherlands.

REFERENCES

1. L. Chow, *Dutch Trains Are World's First to Run on 100% Wind Power*. EcoWatch, Jan 2017.
2. J. Schijve, ed., *Fatigue as a Phenomenon in the Material*, pp. 13–58. Dordrecht: Springer Netherlands, 2009.
3. Z. Popovic and et al, *RAIL DEFECTS HEAD CHECKING ON THE SERBIAN RAILWAYS*. 2012.
4. G. C. Sih, *Experimental evaluation of stress concentration and intensity factors*, vol. 7, p. 10–16. MARTIN US NUHOFF, 1981.
5. K. Palaniswamy and W. G. Knauss, *Propagation of a crack under general, in-plane tension*, vol. 8. Springer Science and Business Media LLC, Mar. 1972.
6. X. Jiang, X. Li, X. Li, and S. Cao, *Rail fatigue crack propagation in high-speed wheel/rail rolling contact*, vol. 25. Springer Science and Business Media LLC, July 2017.
7. P. Christodoulou, A. Kermanidis, and G. Haidemenopoulos, *Fatigue and fracture behavior of pearlitic Grade 900A steel used in railway applications*, vol. 83. Elsevier BV, June 2016.
8. M. Smith, *ABAQUS/Standard User's Manual, Version 6.9*. United States: Dassault Systèmes Simulia Corp, 2009.
9. C. Li, D. E., and N. Yi, *Analysis on fracture initiation and fracture angle in ductile sheet metal under uniaxial tension by experiments and finite element simulations*, vol. 31. Cambridge University Press, 2016.
10. T. Belytschko, R. Gracie, and G. Ventura, *A Review of Extended/Generalized Finite Element Methods for Material Modelling*, vol. 17. 04 2009.
11. T. Belytschko, R. Gracie, and G. Ventura, *A review of extended/generalized finite element methods for material modeling*, vol. 17. IOP Publishing, apr 2009.
12. A. Singh, J. Kumar, V. Dhull, and D. Bhardwaj, *Comparative study of FEM and XFEM*, vol. 4(2). et, 2013.
13. Z. zhong Du, *eXtended Finite Element Method (XFEM) in Abaqus*. Dassault Systèmes Simulia Corp.
14. M. R. Aalami, A. Anari, T. Shafighfard, and S. Talatahari, "A robust finite element analysis of the rail-wheel rolling contact," *Advances in Mechanical Engineering*, vol. 5, p. 272350, Jan. 2013.
15. "Stress concentration analysis and design," in *Peterson's Stress Concentration Factors*, pp. 457–512, John Wiley & Sons, Inc.
16. A. Riccio, U. Caruso, A. Raimondo, and A. Sellitto, "Robustness of xfem method for the simulation of cracks propagation in fracture mechanics problems," *American Journal of Engineering and Applied Sciences*, vol. 9, no. 3, p. 599–610, 2016.