Fatigue crack growth in the contact wire of railway catenary

Si Hai Mai[1,*], **Mac Lan Nguyen-Tajan**[1], **Arnaud Verrier**[1], **Josselin Banting**[2], **Stéphane Avronsart**[3]

[1] Innovation &Research, Société Nationale des Chemins de Fer Français (SNCF), 75611 Paris, France
[2] Agence d'Essai Ferroviaire, SNCF, 94407 Vitry Sur Seine, France
[3] Direction de l'Ingénierie, SNCF, 93574 La Plaine Saint Denis, France
* Corresponding author: si-hai.mai@sncf.fr

Abstract

In this paper, we used the eXtended Finite Element Method (XFEM), implemented in the software CAST3M developed by Commissariat à l'Energie Atomique (CEA), to simulate the fatigue crack growth of the contact wire in the railway catenary.

The material characteristics and parameters of the Paris law were identified thanks to experimental tests performed in a laboratory of Société Nationale des Chemins de Fer Français (SNCF). Specimens were cut directly from the contact wire. Two mean-stress levels were considered. The stress intensity factors were calculated by Finite Elements Method.

In order to validate the model and check its robustness, we performed a parametric analysis with different crack geometries. The numerical results showed a good agreement with the experimental observations in terms of the evolution of the crack shape and its growth rate. These results show that this numerical strategy is relevant and efficient to predict the critical size and the residual lifetime of the fatigue cracks detected by maintenance operations (Ultrasonic monitoring).

Keywords: fatigue crack growth, numerical simulation, eXtended Finite Element Method (XFEM), railway catenary.

---

1. Introduction

The role of the catenary system is to transmit the electrical energy from the energy supply point to trains (Fig. 1). To ensure a good current collection quality during a train passage, the pantograph applies a vertical force on the contact wire. This upward force causes a periodic bending stress. In addition to this periodic bending stress, the contact wire is subjected to a constant mechanical tensile force depending on the maximum speed of the line. The role of this tensile force is to avoid problems of dynamic instabilities in the contact wire. Moreover, the wear due to the pantograph passages reduces the cross section of the contact wire which leads to a higher tensile stress in the wire. These conditions accelerate the risk of fatigue fracture of the contact wire.

To predict the propagation of fatigue crack, different approaches could be considered. The most popular one is Finite Element Method (FEM). However, the conventional FEM using fixed meshes can only deal with this type of problem, either if the crack path travels through mesh nodes, or if we remove mesh elements [1]. This is an extremely important limitation in industrial applications.

In order to overcome the disadvantage of FEM, we used in this paper the eXtended Finite Element

Method (XFEM), implemented in the software CAST3M [4] developed by Commissariat à l'Energie Atomique (CEA), to simulate the fatigue crack growth of the contact wire. The identification of material parameters and the numerical modeling of the contact wire are described in the next sections. The numerical results are shown and discussed in the last section of the paper.

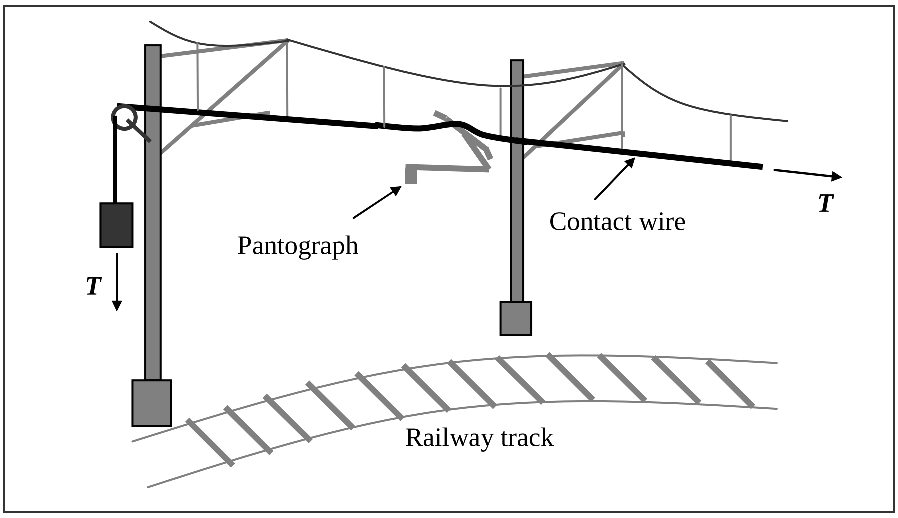


Figure 1. Railway Catenary

2. Fatigue Tests

2.1. Fatigue test specimen

In order to identify the mechanical characteristics and the fatigue crack growth law of copper contact wire, we had carried out fatigue tests with specimens cut directly from the contact wire (Fig. 2). An initial crack of 1mm depth was created on the contact size in the middle of the specimen.

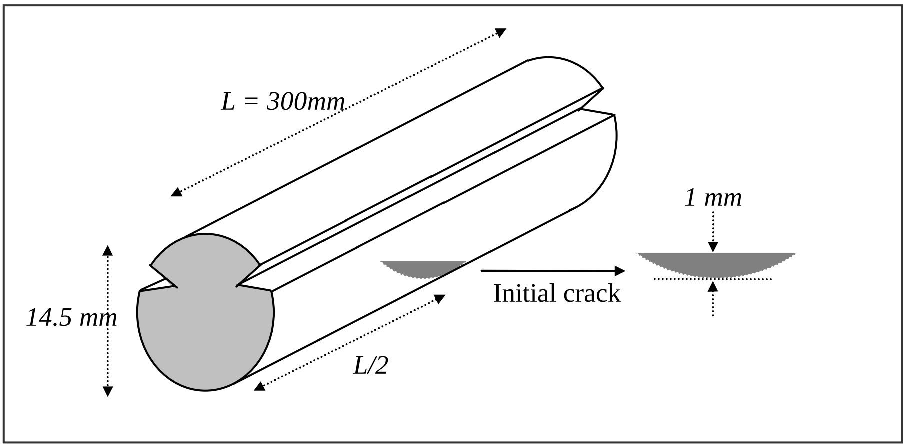


Figure 2. Specimen for fatigue tests (copper contact wire)

2.2. Fatigue crack growth test rig and conditions

The fatigue crack growth test rig is illustrated in Fig. 3. These tests were conducted in air at room temperature on a servo-hydraulic test machine having a load capacity of 50 tons with a frequency of 10 Hz under constant amplitude loading. These tests were conducted at different stress ratio R.

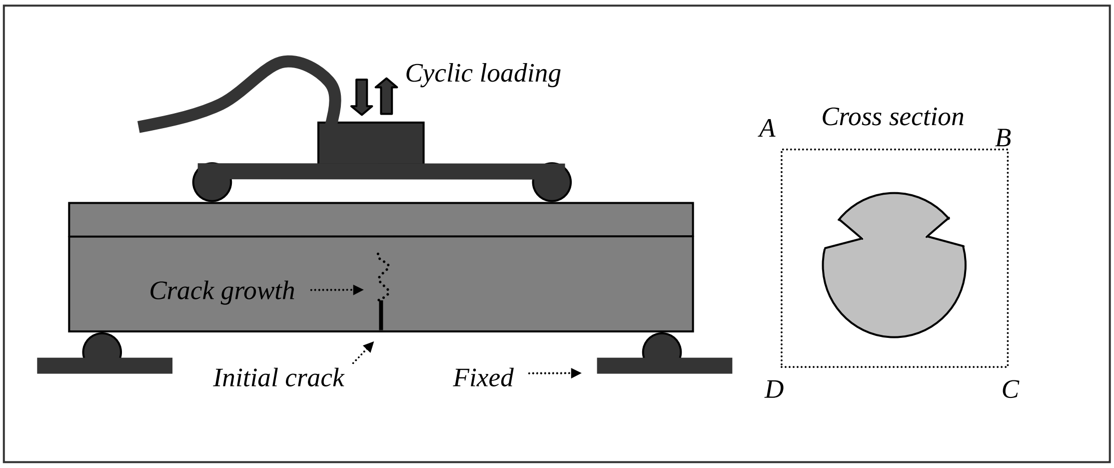
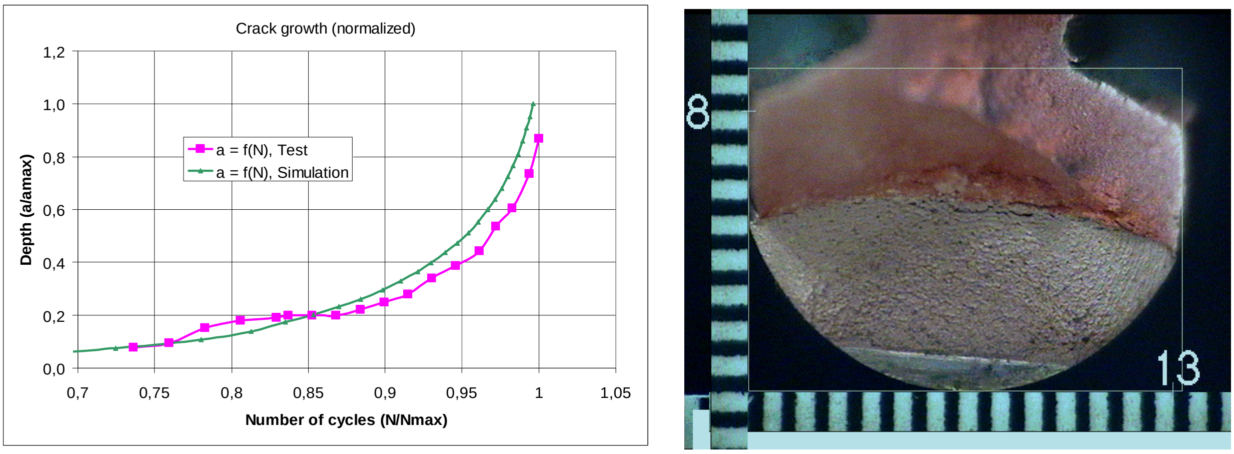


Figure 3. Fatigue crack growth test rig.

2.3. Test results

The test results are shown in the figure 4. Using the test results coupled with numerical calculations (FEM), we identified the parameters of Paris law (m, C) and the toughness (K_{1c}) of the copper material.

Paris law:
$$\frac{da}{dN} = C.\Delta K^{m} \tag{1}$$



a. Crack growth rate b. Crack geometry

Figure 4. Fatigue crack growth in the contact wire

3. Modeling

3.1. XFEM Model

In this paper, we used the eXtended Finite Element Method for linear elastic fracture mechanics (LEFM), in which an enrichment basis is added to the classical finite element basis approximation. This is done using the partition of unity method developed in Babuska and Melenk [2]. The enriched basis shape functions are associated to new degrees of freedom and the displacement field can be written (see Moes *et al.* [1]):

$$U = \sum_{i \in \mathrm{N}} N_i(x) U_i + \sum_{i \in \mathrm{N}_{cut}} N_i(x) H(x) a_i + \sum_{i \in \mathrm{N}_{branch}} \sum_{\alpha} N_i(x) F_\alpha(x) b_{i,\alpha} \tag{2}$$

N is the set of the standard finite element nodes, N_{cut} the set of nodes which belong to elements completely cut by the crack and N_{branch} the set of nodes containing a crack front. N_i are the standard finite element shape functions, $H(x)$ is a Heaviside function which value is 1 if x is above the crack surface and -1 if x is under the crack surface. $[F_\alpha]$ is derived from the LEFM asymptotic displacement field:

$$[F_\alpha] = \left[\sqrt{r}\sin\left(\frac{\theta}{2}\right), \sqrt{r}\cos\left(\frac{\theta}{2}\right), \sqrt{r}\sin\left(\frac{\theta}{2}\right)\sin(\theta), \sqrt{r}\cos\left(\frac{\theta}{2}\right)\sin(\theta) \right] \tag{3}$$

3.2. Numerical algorithm

The eXtended Finite Element Method (XFEM), implemented in the software CAST3M developed by Commissariat à l'Energie Atomique (CEA), was used in this paper to simulate the fatigue crack growth of the contact wire.

In this XFEM model, the mesh of the structure (three dimensions) without crack is fixed during the crack growth. The position of the crack inside the structure is identified thanks to an independent crack mesh (two dimensions) which needs to be updated after each crack growth step.

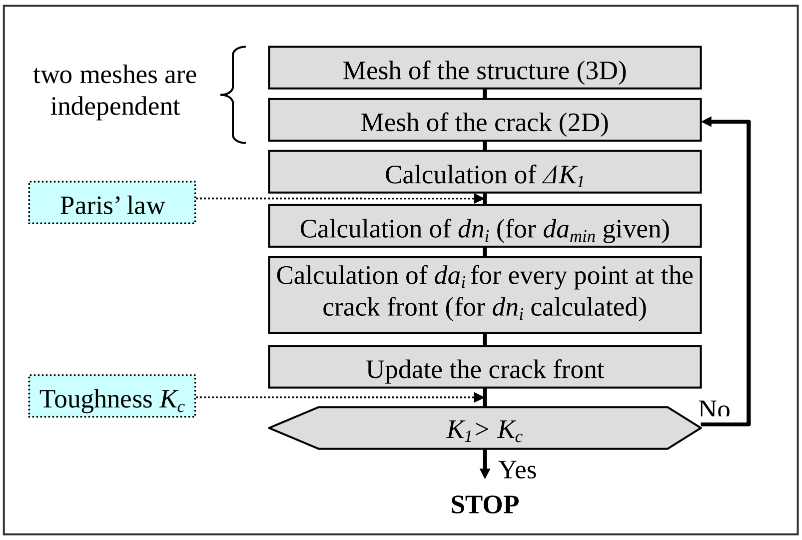


Figure 5: Numerical algorithm for fatigue crack growth simulation.

The numerical algorithm is shown in Fig. 5. The Paris' law and the toughness are material inputs which are identified in the previous section. The calculation of stress intensity factors K_i is done by using the domain integral method (J integral). The crack growth step da_{min} is a numerical input. A convergence study for this numerical input is necessary.

The simulation is stopped if the stress intensity factor K_1 is greater than the toughness K_c of copper material. Otherwise, the crack front is updated and a new crack mesh is constructed.

4. Numerical Results

4.1. Inputs

In this section, we study the fatigue crack growth of a contact wire in flexure using the XFEM presented in previous section.

The meshes of the structure (worn contact wire without crack) and the two initial cracks are shown in Fig. 6. In the cross section of the structure, QUAD elements are used. The final mesh (3D) of the structure is obtained by extrusion of the cross sectional profile. Special 3D XFEM elements are used in the crack zone. Whereas, the elements used to mesh the cracks are classical 2D triangular elements.

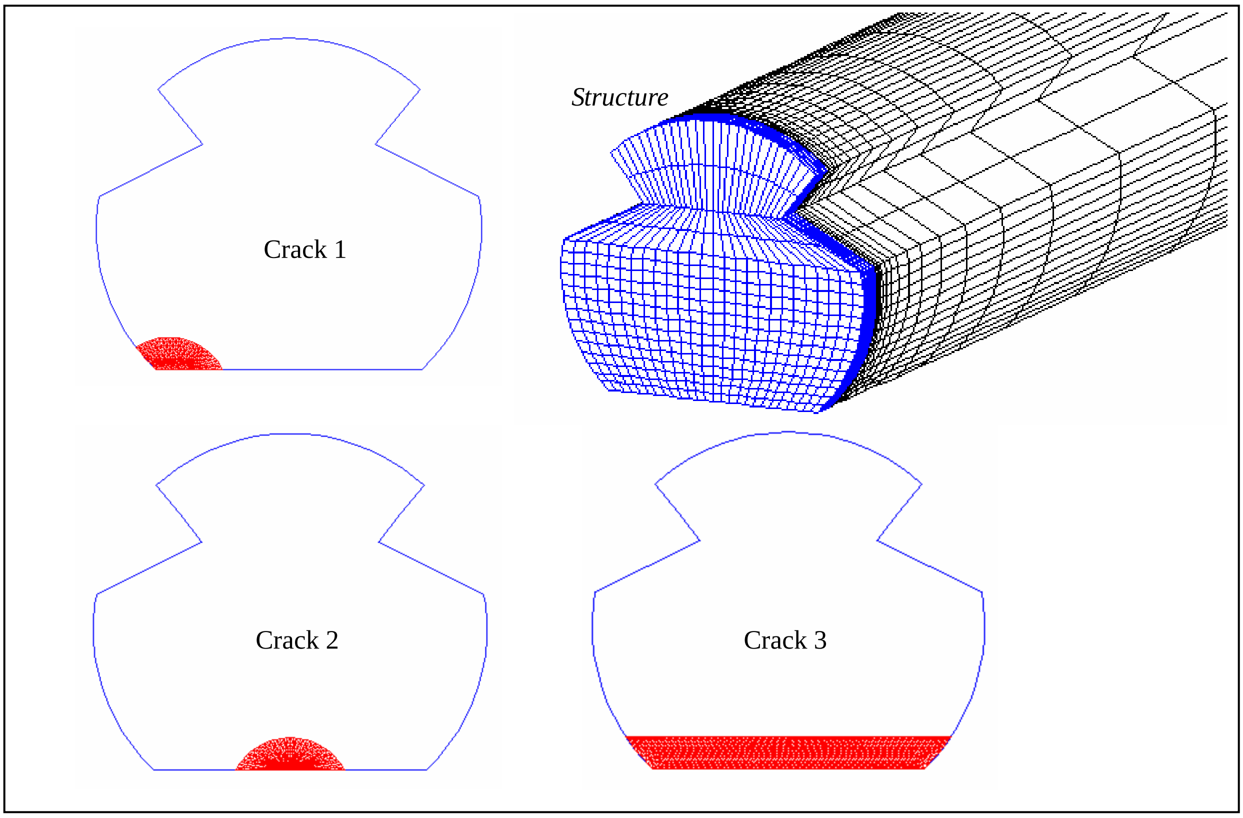


Figure 6: Meshes of the structure and initial cracks

The material parameters (Paris' law and toughness) are identified in the section 2 and the numerical model is presented in the section 3.

4.2. Results

Figure 7 shows the stress intensity factors calculated at the crack front for different types of initial crack. We can see from these results that the mode I is dominant ($K_2 \sim 0$ and $K_3 \sim 0$) in all cases. Thus, the Paris' law is applicable for our model.

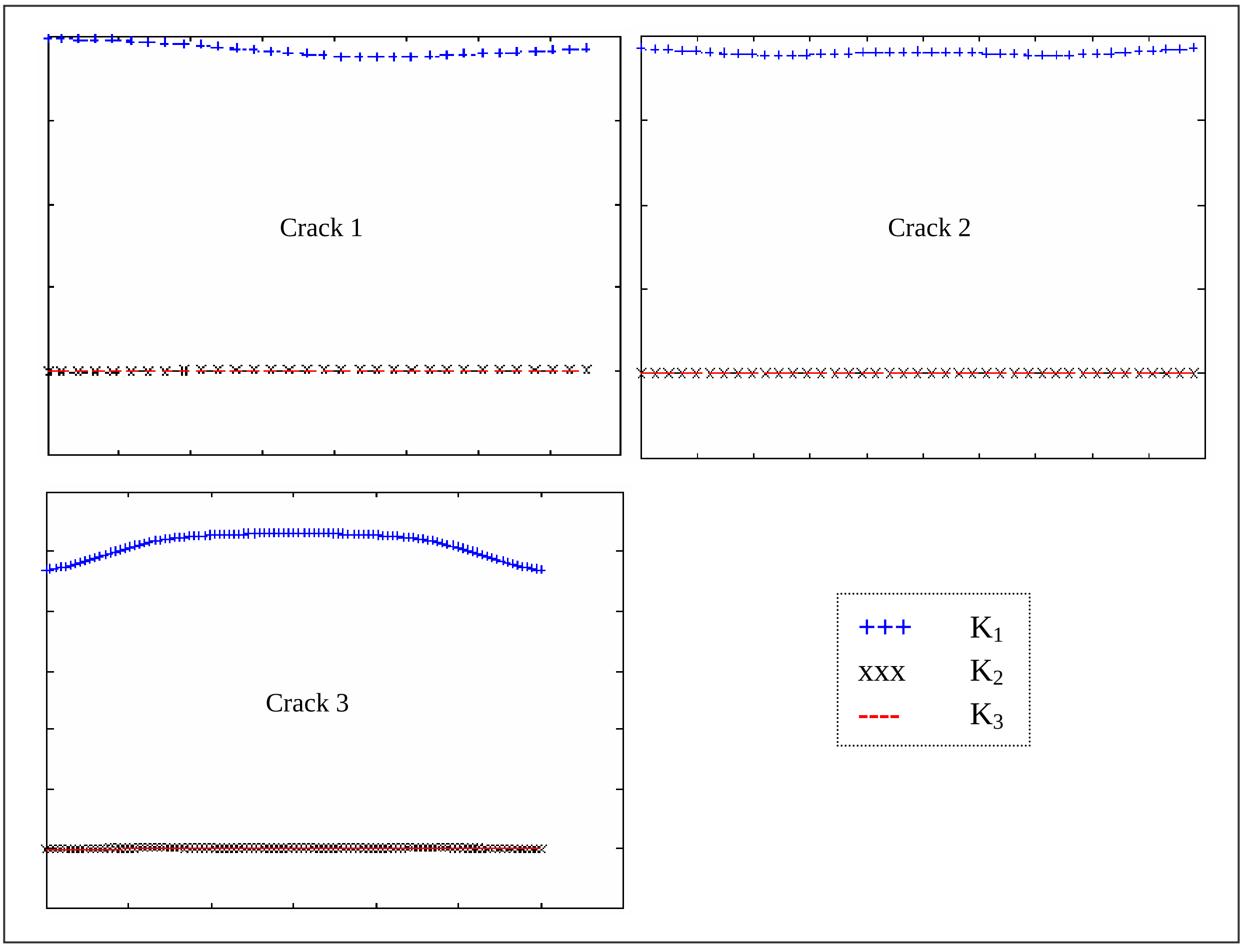


Figure 7: Stress intensity factors for three initial cracks

Using the XFEM model described above, we've performed different crack growth simulations. We can see in the figure 8 a comparison between the numerical simulation and the test result. The numerical results showed a good agreement with the experimental test in terms of the evolution of the crack shape and its growth rate.

In the figure 9, we show the crack growth simulated for three types of initial crack. Figure 10 shows the evolution of the crack depth in function of the loading cycles. This type of curve could be used in combination with the ultrasonic measurements to optimize the maintenance planning of the contact wire. For example, if we detect a crack of contact wire (point A) at the moment *t*, using the numerical curve (Fig. 10) we could estimate the action timescale before reaching the critical crack (point B). To follow the real crack growth, some additional measures could be performed between twos points A and B. If necessary, the critical point B could be modified after each measurement.

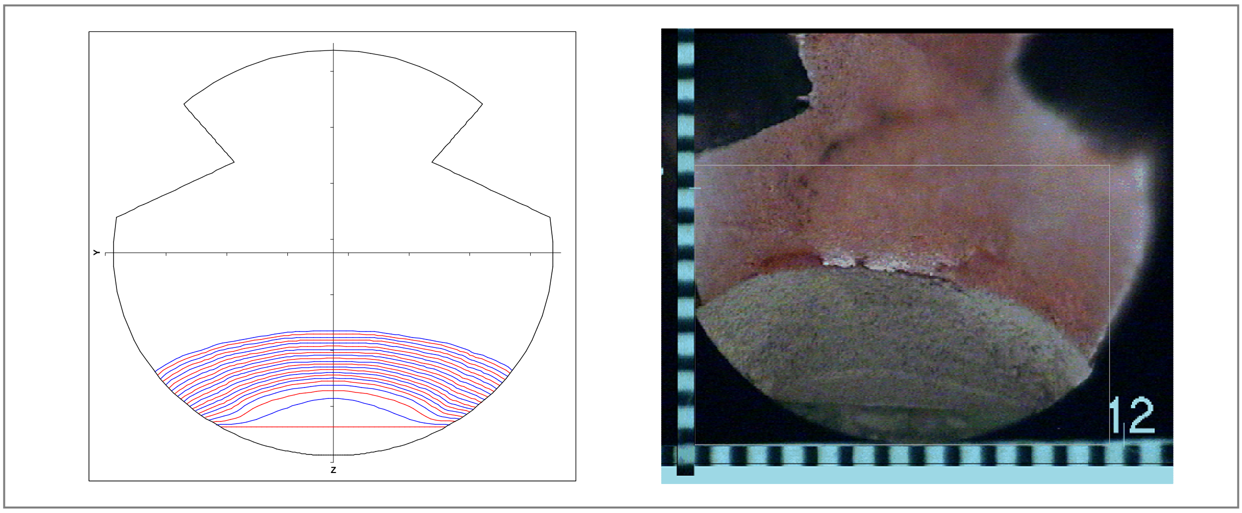


Figure 8: Comparison between the numerical simulation and the test result

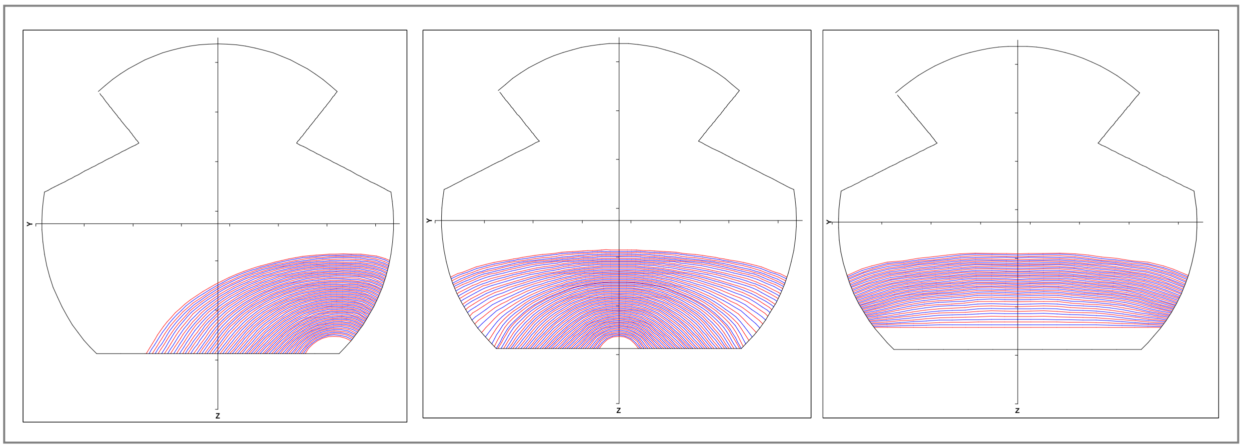


Figure 9: Crack growth for different initial cracks

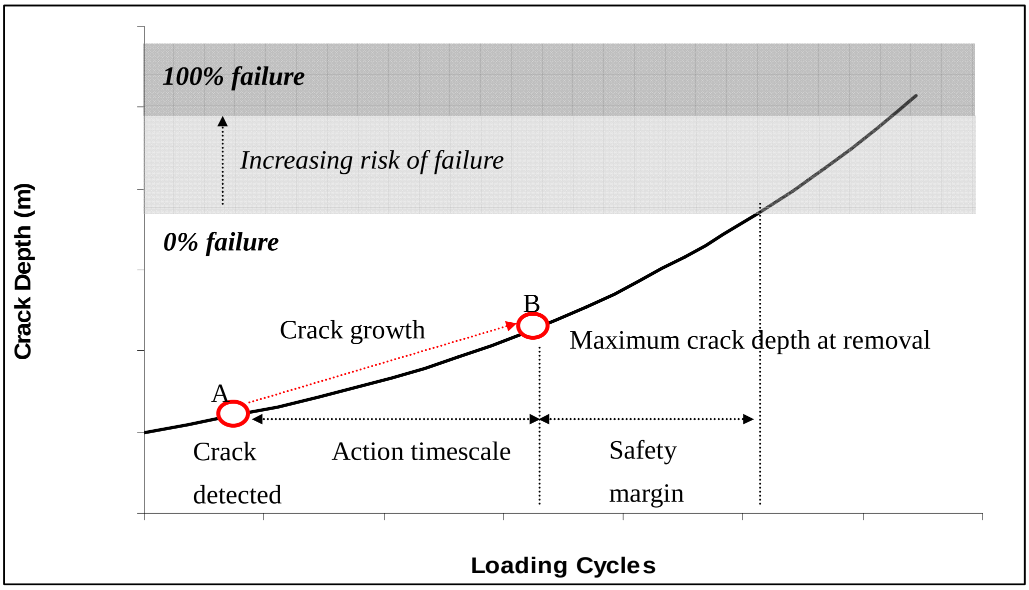


Figure 10: Evolution of the crack depth

5. Conclusion

In this paper, we used the eXtended Finite Element Method (XFEM), implemented in the software CAST3M developed by Commissariat à l'Energie Atomique (CEA), to simulate the fatigue crack growth of the contact wire.

The material characteristics were identified thanks to experimental tests performed in a laboratory of Société Nationale des Chemins de Fer Français (SNCF) with specimens cut directly from the contact wire.

Different geometries of crack were studied. The numerical results showed a good agreement with observation in term of the evolution of the crack shape and its growth rate. Theses preliminary results show that this numerical strategy can be used to predict the critical size and the residual life of fatigue cracks detected by maintenance operations.

A more detailed study on the real cracks detected in the contact wire will be the target of the next step.

Acknowledgements

The authors gratefully acknowledge Jean-Pierre MASSAT from Innovation and Recherche of SNCF for discussions about dynamical behaviour of pantograph-catenary system, engineers from Ingénierie Technologie of SNCF (SNCF-IG-TE-ZC) for discussions about maintenance method of contact wire, Benoit PRABEL from CEA for discussions about XFEM simulation and the Direction SNCF-CSC for the financial support of this project.

References

[1] N. Moes , J. Dolbow, T. Belytschko, A finite element method for crack growth without remeshing, International Journal for Numerical Methods in Engineering ,1999; 46(1):133–150.
[2] I. Babuska, J.M. Melenk , The Partition of unity method, International Journal for Numerical Methods in Engineering 1997; 40:727–758.
[3] S. Pommier , A. Gravouil, A. Combescure, N. Moes, Extended Finite Element Method for Crack Propagation, ISTE Ltd 2011.
[4] Software CAST3M, website http://www-cast3m.cea.fr/.