Paper 128

Proceedings of the Eleventh International Conference
on Computational Structures Technology,
B.H.V. Topping, (Editor),
Civil-Comp Press, Stirlingshire, Scotland

Simulation of Fatigue Crack Growth of a Contact Wire in a Catenary System

S.H. Mai and M.L. Nguyen
Innovation and Research
Société Nationale des Chemins de Fer Français, Paris, France

Abstract

The role of the catenary system is to transmit the electrical energy from the energy supply point to trains. To ensure a good energy capture during the passage of a train, the pantograph applies a vertical force on the contact wire. This upward force causes a periodic bending stress which can lead to a fatigue fracture.

To predict the propagation of this fatigue crack, different approaches could be considered. In this paper, the extended finite element method (XFEM), implemented in the software CASTEM developed by the Commissariat à l'Energie Atomique (CEA), to simulate the fatigue crack growth of the contact wire.

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

Two geometries of crack were studied. 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 preliminary results show that this numerical strategy is relevant and efficient to predict the critical size and the residual life of fatigue cracks detected during maintenance operations.

Keywords: catenary, contact wire, fatigue, fracture, numerical simulation, extended finite element method, fatigue test.

1 Introduction

The role of the catenary system is to transmit the electrical energy from the energy supply point to trains (Figure 1). To ensure a good energy capture during the

passage of a train, the pantograph applies a vertical force on the contact wire. This upward force causes a periodic bending stress which can lead to a fatigue fracture.

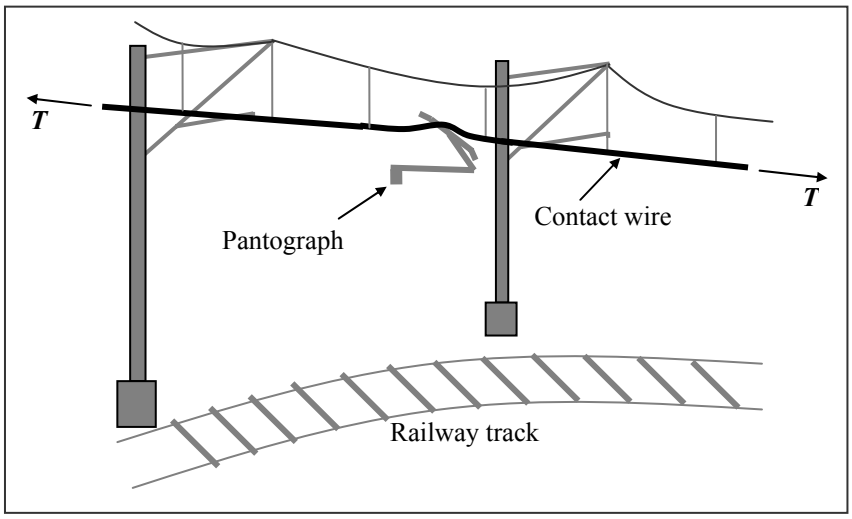

Figure 1: Catenary system.

In addition to the periodic bending stress, the contact wire is subjected to a constant tensile force defined by the speed of the train. The role of this tensile force is to avoid problems of dynamic instabilities in the contact wire. Moreover, the wear induced by friction of the pantograph reduces the cross section of the contact wire which leads to a higher tensile stress in the wire. These conditions accelerate the risk of fracture of the contact wire.

To predict the propagation of fatigue crack, different numerical approaches could be considered. The most popular one is finite element method (FEM). Within this approach, a unique mesh is used to model the structure and the crack, which leads to meshes with a very high number of degrees of freedom and also difficulties to have relevant meshes. And to model appropriately the crack growth, it is generally needed to incrementally modify the mesh. This procedure is very time-consuming.

In order to overcome the disadvantage of FEM, we used in this paper the extended finite element method (XFEM), implemented in the software CASTEM [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 modelling of the contact wire are described in the next sections. The numerical results are shown and discussed in the last section of the paper.

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, tensile fatigue tests were carried out on specimens directly

removed from the contact wire (Figure 2). An initial crack of 1mm depth was implemented on the contact size in the middle of the specimen.

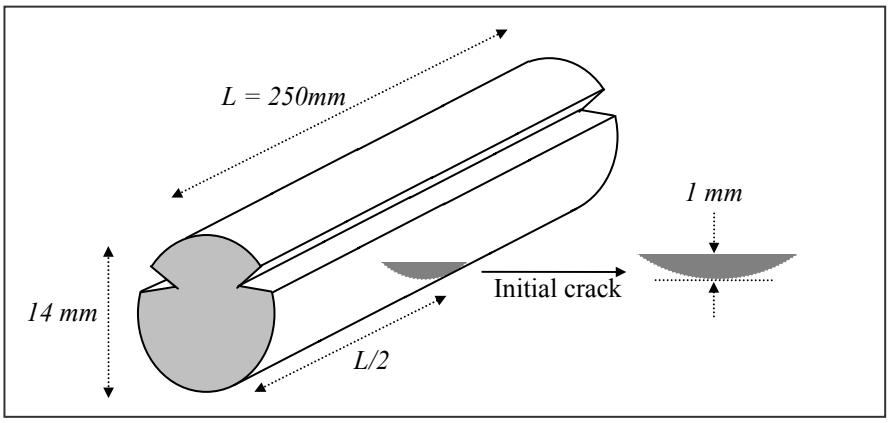

Figure 2: Specimen for fatigue tests (copper contact wire)

2.2 Fatigue crack growth test rig and conditions

The fatigue crack growth test rig is illustrated in Figure 3. These tests were conducted in air at room temperature on a servo-hydraulic test machine with a load capacity of 50 tons with a frequency of 5 Hz under constant amplitude loading. These tests were performed for different stress ratio R=Smin/Smax.

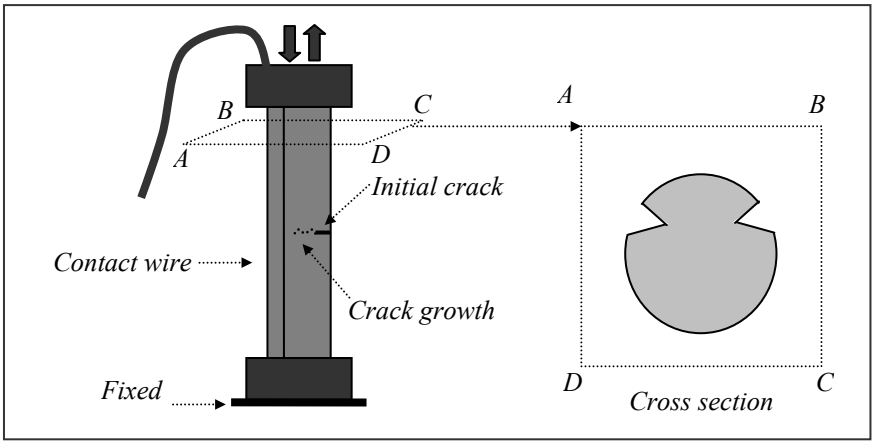

Figure 3: Fatigue crack growth test rig.

3 Modelling

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]):

$$U = \sum_{i \in \mathrm{N}} N_i(x) U_i + \sum_{i \in \mathrm{N}_{cut}} N_i(x) H(x) a_i + \sum_{i \in \mathrm{N}_{branch}} \sum_{\alpha} N_i(x) F_\alpha(x) b_{i,\alpha} \tag{1}$$

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:

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

3.2 Numerical algorithm

The extended finite element method (XFEM), implemented in the software CASTEM 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.

The numerical algorithm is shown in Figure 4. 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 stops when the stress intensity factor K_I is greater than the toughness K_c of copper material. Otherwise, the crack front is updated and a new crack mesh is built.

4 Numerical Examples

4.1 Input data

In this section, the fatigue crack growth of a worn contact wire submitted to a bending moment using the XFEM presented in previous section is studied. The boundary conditions and the dimensions are illustrated in Figure 5.

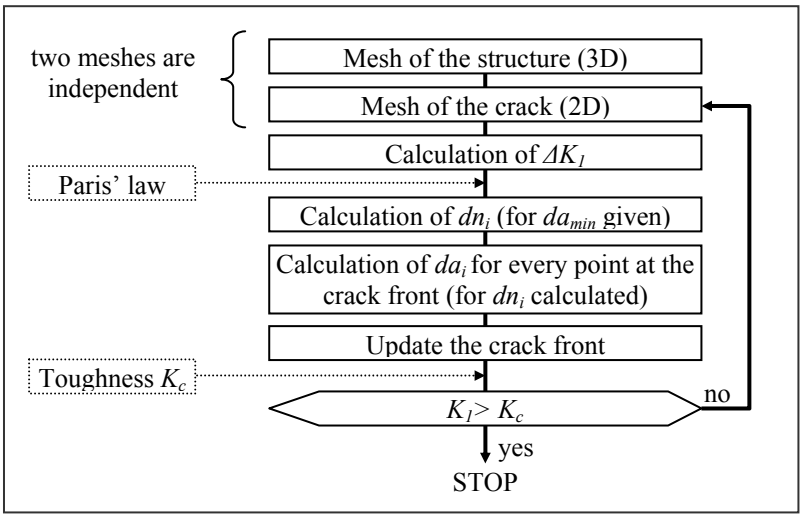

Figure 4: Numerical algorithm for fatigue crack growth.

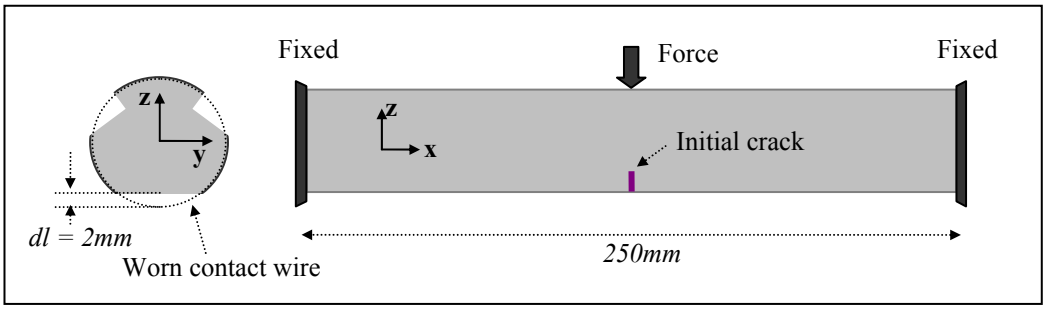

Figure 5: Contact wire submitted to a bending moment.

Two different geometries of initial crack are considered. The first one is an elliptical crack and the second one is a straight crack.

The meshes of the structure (without crack) and the two initial cracks are shown in Figure 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.

4.2 Results

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

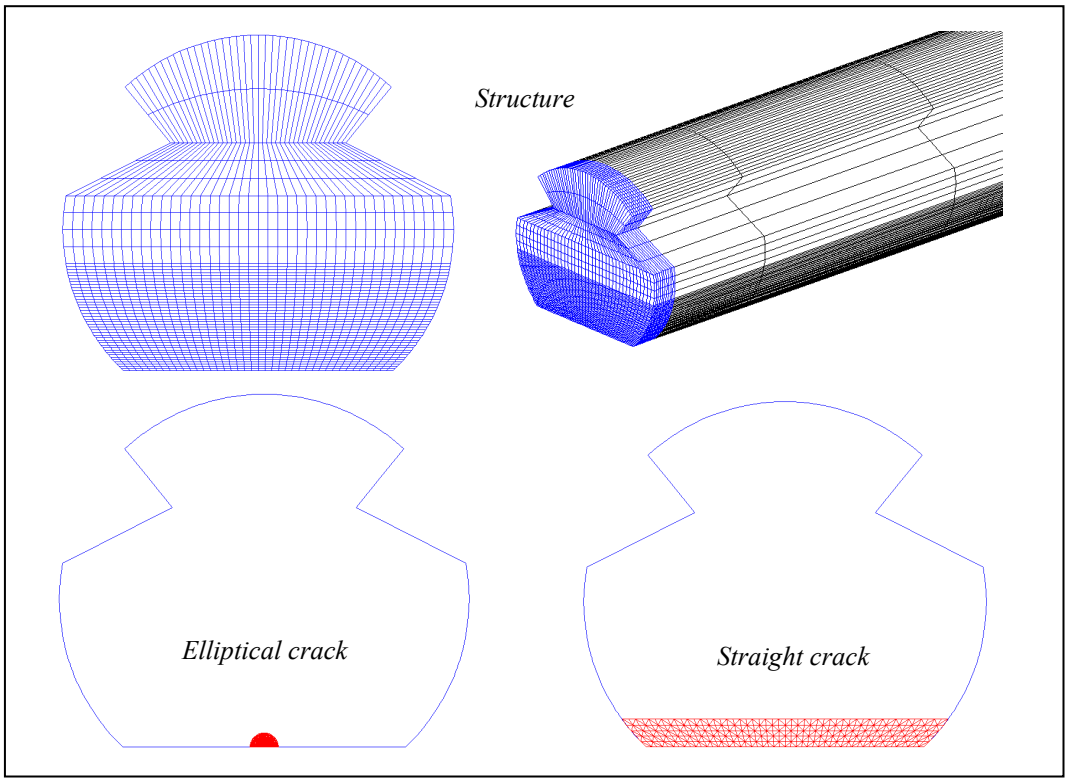

Figure 6: Meshes of the structure and two initial cracks

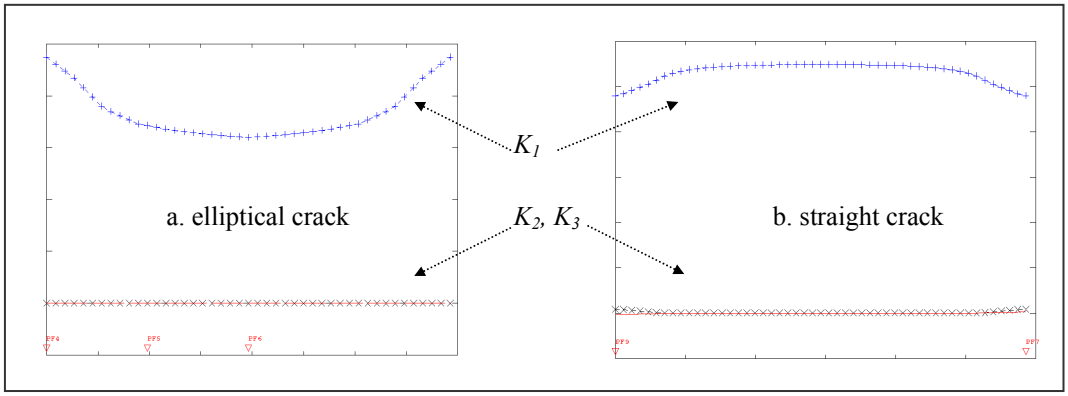

Figure 7: Stress intensity factors; a. elliptical crack; b. straight crack

In case of elliptical crack, the stress intensity factor K_1 is higher at the boundary than in the middle of the crack front. Consequently, the elliptical crack will propagate more quickly in the lateral direction than in the vertical direction and it becomes larger and larger (Figure 8a). However, when the crack reaches the lateral boundary of the contact wire, it grows more quickly in the middle than at the boundary (Figure 8b).

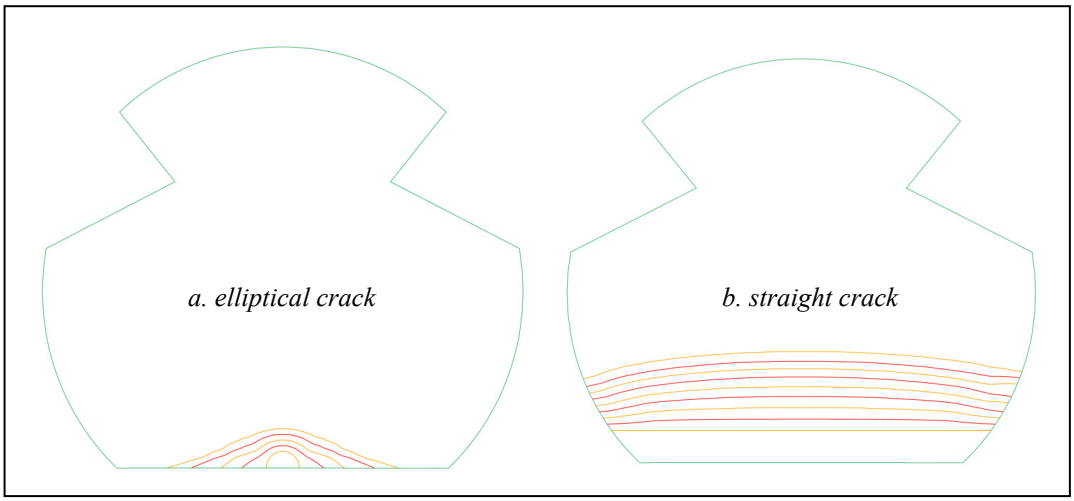

Figure 8: Evolution of the fatigue crack shape

Figure 9 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 optimise the maintenance planning of the contact wire. For example, if we detect a crack (point A) in the contact wire at the moment *t*, using the numerical curve (Figure 9) we could estimate the action timescale before reaching the critical crack (point B).

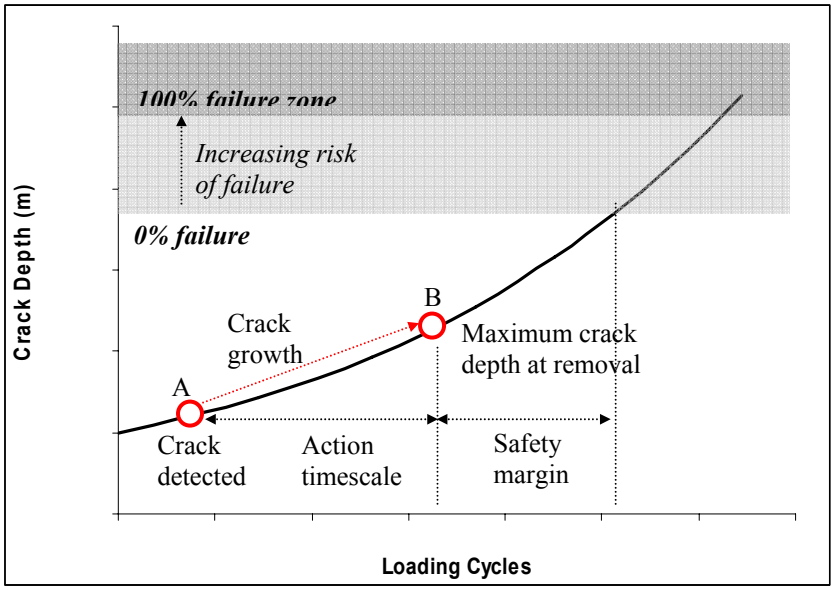

Figure 9: Evolution of the crack depth

5 Conclusion

In this paper, we used the extended finite element method (XFEM), implemented in the software CASTEM 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 directly removed from the contact wire.

Two geometries of crack were studied. 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 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 shape and position of cracks in the contact wire will be the target of the next step.

Acknowledgements

The authors gratefully acknowledge Josselin BANTING from Agence d'Essai Ferroviaire (AEF) for the realization of fatigue tests, Benoit PRABEL from Commissariat à l'Energie Atomique (CEA) for his collaboration concerning the XFEM model, 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, and the Direction SNCF-CSC for the financial support for this project.

References

[1] Moes N, Dolbow J, Belytschko T, "A finite element method for crack growth without remeshing", International Journal for Numerical Methods in Engineering ,1999; 46(1):133–150.

[2] Babuska I, Melenk JM, "The Partition of unity method", International Journal for Numerical Methods in Engineering 1997; 40:727–758.

[3] Pommier S, Gravouil A, Combescure A and Moes N, "Extended Finite Element Method for Crack Propagation", ISTE Ltd 2011.

[4] Software CASTEM, website http://www-cast3m.cea.fr/.