NUMERICAL SIMULATION OF CRACK GROWTH UNDER CYCLIC LOADING
WITH A FOCUS ON ELASTIC-PLASTIC MATERIAL BEHAVIOUR

P. Zerres¹, M. Vormwald¹

¹) Materials Mechanics Group, Technische Universität Darmstadt, Petersenstraße 12,
D-64287 Darmstadt, Germany

ABSTRACT

In this paper a procedure to simulate the crack growth under consideration of elastic-plastic
material behaviour by means of the finite element method is described. Within this
procedure, after each increment of crack advance, the structure is re-meshed and the status
variables, such as the components of the backstress-tensor and the plastic strains, are
transferred from the old mesh to the new one. The crack growth life is determined by
integration of a crack growth law which is based on the effective stress intensity factor range.
The results obtained by the procedure are compared with experimental results and finite
element results based on a nodal release scheme for autofrettaged Diesel-engine injection
tubes. A good agreement between the two numerical analyses can be stated. Also the
significant dependence of the results on the used material model is shown.

KEYWORDS

Crack growth, crack closure, finite element method

INTRODUCTION

The currently common methods for calculating the fatigue crack growth can be distinguish in
simulations based on linear elastic and based on elastic-plastic fracture mechanics.

By using the linear elastic fracture mechanics effects due to plasticity of the material are not
explicitly captured but implicitly in line with the crack growth law. Here, small scale yielding
has to be assumed. Beside the aspired realistic estimation of the fatigue crack growth life,
the focus of the scientific works on this field concentrates on the description of the crack
path. The fundamental parameter is the range of the stress intensity factor and its connection
to the crack growth rate, whereas this connection has to be empirically determined. For the
calculation of the stress intensity factors, the finite element (FE) and the boundary element
(BE) method are established. This approach is also implemented in many two- and three-
dimensional, commercial and non-commercial software tools, such as FRANC2D [1, 2],
FRANC3D [3, 4], ZENCRACK [5, 6], ADAPCRACK3D [7, 8], BEASY [9].

Methods based on elastic-plastic fracture mechanics are favoured, if load sequence or mean
stress effects should be captured or if the assumption of small scale yielding is evidently
violated. By incorporating elastic-plastic material behaviour deeper insights in the
mechanisms of fatigue crack growth should be achieved. The crack path is commonly
prescribed and is assumed to be straight-lined. To model fatigue crack growth a nodal-
release scheme is generally adopted. These methods are often used to model plasticity
induced crack closure [10], see for example [11-13]. A drawback of these methods is that the
actual fatigue life is calculated after the simulation by integrating an empirically determined
crack growth law.
For a crack growing in a complex structure under a high cyclic loading level, the crack path is not known a priori on the one hand and has to be determined during the analysis. On the other hand the crack growth rate is affected by plasticity effect. Thus, both methods stated above are not suited for this problem.

The purpose of this paper is to present a procedure to simulate fatigue crack growth by combining the two methods described above. So the crack growth is determined by adopting elastic-plastic material behaviour and the crack path is determined during the simulation by remeshing the structure after every increment of crack growth.

CRACK GROWTH PROCEDURE

In this section the developed crack growth procedure, which is shown in Fig. 1, is presented. All the modules of the procedure are implemented as user routines to the commercial finite element program ABAQUS.

Fig. 1: Crack growth procedure

As an input to the procedure, the geometry of the structure, information of the material (parameters of the used material-model) and the load sequence must be provided.

The numerical simulation of the fatigue crack growth is subdivided into the following steps, as shown in Fig. 1. First the model is generated. The mesh is refined at the crack tip to accurately describe the stress state. Afterwards the structure is analysed. In the postprocessing the crack opening and closure level is determined and the effective range of the crack tip parameter is calculated. With that the crack growth direction and the crack advance for a prescribed number of cycles (or the number of cycles to reach a given crack advance) can be computed by integrating an appropriate crack growth law. The geometry is updated by extending the crack in the calculated direction by the crack increment. Then the new model is generated. To re-establish the stress state prior to crack advance the displacements and the status variables have to be mapped from the old mesh to the new one. The displacements are transferred by using the isoparametric shape functions of the elements. For an isoparametric element, the coordinates $\mathbf{x}$ and the displacements $\mathbf{u}$ can be described as follows:

$$
\mathbf{x} = \sum_{j=1}^{n} N_j \cdot \mathbf{x}_j \quad \text{and} \quad \mathbf{u} = \sum_{j=1}^{n} N_j \cdot \mathbf{u}_j.
$$

(1)
In equation (1) \( n \) is the number of nodes in the element and \( \bar{x}_j \) and \( \bar{u}_j \) are the coordinates and displacements at the j-th node respectively. \( N_j \) are the shape functions. They are equal to one at the j-th node and zero at all other nodes, therefore

\[
N_j = f(\bar{g}) \quad , \quad \bar{g} = \begin{pmatrix} g \\ h \end{pmatrix} \quad , \quad -1 \leq g, h \leq 1 ,
\]

with \( g \) and \( r \) being the coordinates within the local element coordinate system. Re-arranging equation (1) leads to

\[
\bar{F} = \sum_{j=1}^{n} N_j \cdot \bar{x}_j - \bar{x} = 0
\]

For every node in the new mesh the corresponding element in the old mesh is detected by solving equation (3) using a Newton's method implying equation (2). As a result the local coordinates \( \bar{g}_{\text{new}} \) of the node in the old mesh are known and the displacements can be calculated by using equation (1). The deformed coordinates of the node in the new mesh are generated by shifting the undeformed coordinates by the calculated displacements.

Afterwards all the status variables, i.e. the stresses, strains, backstresses and other state variables (depending on the applied material model) are transferred by using routines, which are implemented in ABAQUS. The procedure is then continuing to calculate the next increment of crack advance. The procedure is repeated until the crack length reaches a user defined critical value.

It should be noted that without mapping of the variables, i.e. for linear elastic material behaviour, the procedure was successfully verified in [14] for the determination of the crack growth path under mixed-mode loading.

**EXAMPLE**

The examined problem is a pressurised tube with closed ends. The problem was investigated by Herz et al. [12, 15], where a nodal release scheme and an advanced material model [16, 17] was used.

**Geometry**

To avoid the difficulties and computational cost of a three-dimensional analysis, the straight pressurised closed tube is modelled as a completely axis symmetric pressurised torus with a very large radius of 600mm. The outer diameter of the tube is \( d_o = 6.0 \text{mm} \) and the inner diameter is \( d_i = 2.5 \text{mm} \). Axial symmetric elements with linear shape functions and full integration are employed. Unlike in [12, 15] the whole cross section area is modelled and no symmetry boundary conditions for \( y = 0 \) are used, see Fig. 2. The initial crack length is set to \( a_o = 10 \mu\text{m} \) according to experimental observations. Despite of the fact that crack advance is assumed to be only in \( x \)-direction, with the model shown in Fig. 2(b) the possibility of an arbitrary crack advance should be demonstrated.
In this investigation two different material models are used. For the nodal-release calculations an advanced material model, which was originally introduced by Döring [16, 17] (Döring-model), and a common model with a non-linear kinematic hardening rule according to Armstrong and Frederick [18] (AF-model). For the mapping analysis only the AF-model is used due to numerical difficulties with the Döring-model, which have not been solved at the moment. The material parameters for the Döring-model for the investigated material S 460 nbk are shown in Table 1 and the material parameters for the AF-model are σ = 270 MPa, c = 320 and r = 200 MPa. Additionally the isotropic elastic constants are equal to E = 208.5 GPa and ν = 0.3. The parameters of the models are described in [17].

Table 1: Material parameters for the Döring-model

<table>
<thead>
<tr>
<th>η</th>
<th>c_{\text{in}}</th>
<th>ω</th>
<th>c_T</th>
<th>c_A</th>
<th>a_{\text{in}}</th>
<th>a_n</th>
<th>b_n</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>5</td>
<td>1</td>
<td>50</td>
<td>80</td>
<td>1.74</td>
<td>-0.26</td>
<td>100</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>i</td>
<td>c^{(i)}</td>
<td>r_{\text{in}}^{(i)} [MPa]</td>
<td>a_1^{(i)}</td>
<td>a_2^{(i)}</td>
<td>a_3^{(i)}</td>
<td>a_4^{(i)}</td>
<td>Q^{(i)}</td>
</tr>
<tr>
<td>0</td>
<td>-</td>
<td>201</td>
<td>1.1</td>
<td>0.0</td>
<td>-0.1</td>
<td>0</td>
<td>-</td>
</tr>
<tr>
<td>1</td>
<td>1260</td>
<td>69.4</td>
<td>-1.0</td>
<td>-0.4</td>
<td>0.7</td>
<td>0</td>
<td>18</td>
</tr>
<tr>
<td>2</td>
<td>408</td>
<td>49.6</td>
<td>-0.7</td>
<td>-1.0</td>
<td>1.2</td>
<td>0</td>
<td>5</td>
</tr>
<tr>
<td>3</td>
<td>126</td>
<td>69.9</td>
<td>0.0</td>
<td>-0.3</td>
<td>-0.5</td>
<td>0</td>
<td>3</td>
</tr>
<tr>
<td>4</td>
<td>40.8</td>
<td>79.2</td>
<td>0.0</td>
<td>0.0</td>
<td>-0.5</td>
<td>0</td>
<td>3</td>
</tr>
<tr>
<td>5</td>
<td>12.6</td>
<td>163</td>
<td>0.0</td>
<td>0.0</td>
<td>-0.6</td>
<td>0</td>
<td>18</td>
</tr>
</tbody>
</table>

Load sequence and crack growth procedure

The crack growth procedures for the nodal-release and the mapping analysis are akin and adopted from [12]. They are divided into two phases.

The first phase is the autofrettage process. There, the specimen is loaded up to autofrettage pressure and then unloaded to \( p_{\text{min}} = 50 \) bar. Afterwards 250 load cycles with the operating...
pressure for the redistribution of the residual stress field due to the combined effect of ratchetting and mean stress relaxation are applied.

The second phase is the crack growth phase. Within this step small differences between the two types of analysis appear. First the specimen is loaded up to operating pressure. For the nodal-release analysis the specimen is unloaded to 90% of the operating pressure and in the same time boundary condition of fixed displacement in y-direction (according to Fig. 2(a)) of the node representing the actual crack tip is released and by this a crack advance of one element, which is $\Delta a = 2.5\mu m$ in this example, is realised. For the mapping analysis the specimen is also unloaded to 90% of the operating pressure. At this stage the new model is generated and the mapping of the variables is performed. Afterwards – for both types of analysis – the specimen is further unloaded up to $p_{min}$. Between two crack propagation steps, 10 additional load cycles are applied to stabilise the stress state at the crack tip. Within the last of these cycles the values needed for the calculation, which are the displacements in y-direction for the nodal-release and the crack opening for the mapping analysis, are read out. This cycle is subdivided into at least 20 increments for loading and unloading in order to obtain enough values for an accurate calculation.

Results and Discussion

In Fig. 3 the calculated crack opening pressures for the two different types of analysis are depicted. The autofrettage pressure $p_{af}$ is 4700bar and the operating pressure $p_{max}$ is 2740bar, leading to a pressure range of $\Delta p = 2690bar$, which is the experimentally observed endurance limit pressure range $\Delta p_{E,50%}$. For the proposed procedure three different minimal element lengths $l_e$ of 2.5$\mu m$, 0.83$\mu m$ and 0.625$\mu m$ at the crack tip are employed to examine mesh dependencies.

![Fig. 3: Calculated crack opening pressures, $p_{af} = 4700bar$, $\Delta p = 2690bar$, S 460 nbk](image)

By looking at the nodal-release analyses for the two different material models, the results for the crack-opening pressures for the AF-model are up to 40% lower than for the Döring-model. In [12] it was shown that the results obtained by using the Döring-model are in very good agreement with the experimental findings, leading to the conclusion that the crack opening pressures obtained by using the AF-model underestimate the experimental ones considerably.
Comparing the crack opening pressures obtained by the nodal-release analysis with the results obtained by the proposed procedure a good agreement can be stated. For the coarse mesh with $l_e=2.5\mu m$ at the crack tip, which is the same as for the nodal-release analysis, the distribution of the crack opening pressure is quite discontinuous. For the two finer meshes the distribution is continuous. This leads to the conclusion that a refinement of the mesh for the nodal-release analysis would also lead to a more continuous distribution and a minimization of the difference between the two types of analysis. Another reason for the difference is due to the mapping of the variables. In doing so, inaccuracies in the stress state – especially in the region of high gradients – are not avoidable.

Despite of the fact that the crack opening pressures are considerably underestimated a crack growth analysis with the proposed procedure is conducted. In this analysis the advance per crack growth step is set to 15% of the actual crack length. The range of the stress intensity factor is calculated by integrating the weight function solution presented by Ma et al. [19]. The constants for the crack growth law are set to $C=8.8\times10^{-9}$ and $m=3.15$. Fig. 4 shows the meshed geometries for crack lengths of about 0.0175mm, 0.1mm and 0.5mm and in Fig. 5 the results of the calculation are depicted.

![Fig. 4: Meshed geometries for crack lengths of about 0.0175mm, 0.1mm and 0.5mm](image)

![Fig. 5: Crack growth results, $p_{af}=4700bar$, $\Delta p=2950bar$, S 460 nbk](image)

In Fig. 4 the advantages of the remeshing technique are demonstrated. By advancing the crack only the crack tip region, where high stress gradients are expected, is refined and the number of elements is not significantly changing during the analysis. Here, at a crack length...
of about 0.0175mm the number of elements is 2630, while for the crack lengths of about 1mm and 5mm the numbers are 3324 and 2843, respectively.

Referring to Fig. 5 most of the crack growth lifetime is associated to a crack length up to 0.2mm.

Five experiments have been conducted under these loading conditions. The corresponding lifetimes are 451000, 493400, 582000, 616000 and 755600 cycles. Comparing these results with the calculated results shown in Fig. 5, a considerable underestimation by the calculation can be stated. This underestimation can be explained by the underestimation of the crack opening pressures presented above (Fig. 3). Using the Döring-model would lead to a more precise calculation of the lifetime.

SUMMARY AND CONCLUSIONS

In this paper a procedure to simulate fatigue crack growth in elastic-plastic materials is presented. The main constituent of this procedure is the remeshing and subsequent mapping of the status variables from the old mesh to the new one after each step of crack advance.

For the verification of the procedure results on autofrettaged Diesel-engine injection tubes are used. The results obtained by this procedure using a material model with a nonlinear kinematic hardening rule are compared with experimental results and finite element results based on a nodal-release scheme using two material models – the same material model on the one hand and an advanced material model on the other hand. The following two main conclusions can be drawn:

1. A good agreement between the results of proposed procedure and the nodal-release scheme using the same material model can be stated.
2. The material model with a nonlinear kinematic hardening rule has not the ability to describe the complex stress state in an autofrettaged Diesel-engine injection tube under cyclic loading which leads to a considerable underestimation of the lifetime predicted with the proposed procedure.

REFERENCES


Corresponding author: zerres@wm.tu-darmstadt.de