A procedure for evaluating the crack propagation taking into account the material plastic behaviour

A. Rossetti[1], P. Zerres[1], M. Vormwald[1]

[1]Material Mechanics Group, Technische Universität Darmstadt, Petersenstr. 12, D-64287 Darmstadt, Germany Fax: +49 06151163038

rossetti@wm.tu-darmstadt.de , zerres@wm.tu-darmstadt.de , vormwald@wm.tu-darmstadt.de

ABSTRACT. *Fatigue crack growth is one of the main causes for failure in structures under predominantly cyclic loading. A current challenge in numerical simulation of fatigue crack propagation is to evaluate the crack propagation taking into account the cyclic plasticity effects. For example, the autofrettage of intersecting holes generates extremely high compressive residual stress fields following large plastic deformation. These stresses, in combination with the plastic deformations, decelerate the fatigue crack growth at the hole intersection notch. In this work a method is presented which allows the simulation of fatigue crack propagation based on finite element analysis by taking into account the elastic-plastic material behaviour. Particular attention should be paid to these effects because, especially in the area of short cracks and high loads, the crack growth is significantly influenced by plasticity effects. The procedure here presented is numerically implemented in a programming system based on the commercial finite element software ABAQUS. Within this procedure, the calculation of the crack growth life is done by integrating a crack propagation law, which is based on the effective range of the crack tip parameter, as the stress intensity factor or the J-integral. At this point, a new model with the new crack shape is created and meshed, whereas the status variables, such as the components of the back stress tensor and the plastic strains, are transferred from the old mesh to the new one. The latter numerical technique, like the conventional node release algorithms, is capable to preserve the history of plastic deformation for structures with increasing crack length. However, the proposed procedure differs from the nodal release technique because it allows the crack front to develop freely in a non-predetermined way.*

INTRODUCTION

The currently common methods for calculating the fatigue crack growth can be distinguished in simulations based on linear-elastic and based on elastic-plastic fracture mechanics. By using linear-elastic fracture mechanics the 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. 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 and to model fatigue crack growth a nodal release scheme is generally adopted. These methods are often used to model plasticity induced crack closure [1], see for example [2], [3], [4]. 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, so on one hand has to be determined during the analysis and on the other hand the crack growth rate is affected by plasticity effects.

Both methods stated above are not suited for reproducing the experimental results obtained in previous projects [5] in collaboration with MFPA (Materials Research and Testing Center at the Bauhaus-Universität Weimar). During these tests the specimens present a shape of the crack front partially unusually, as shown in Figure 1 and Figure 2. As it is possible to notice the crack length is shorter in the middle of the crack; this is caused by the residual stress field produced during the autofrettage. This residual stress field has a strong influence on crack shape, especially for small load amplitudes it decreases the rate of the crack in the middle of the specimen.
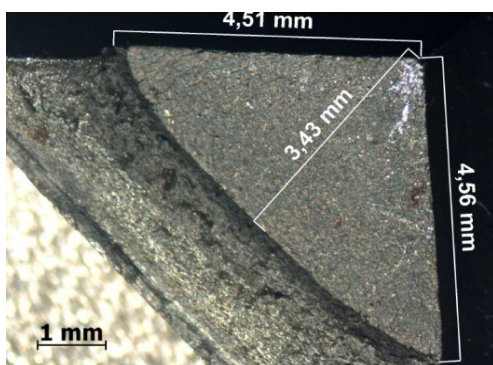

Figure 1 Fracture surface experimental test R=0 with p_{max}= 300 MPa (picture from [5])
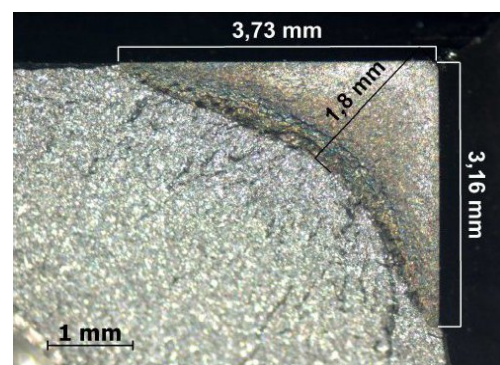
Figure 2 Fracture surface experimental test R=0 with p_{max}= 200 MPa (picture from [5])

For all these reasons, it was decided to perform the explicit simulation of crack propagation in 3D to simulate fatigue crack growth by combining the crack growth determined by adopting elastic-plastic material behaviour (Döring Plasticity model [6]) and the crack path that is determined during the simulation by an adaptive remeshing, after every increment of crack growth, in combination with the transfer of the state variables of the plasticity model from the previous model into the subsequent FE mesh.

CRACK GROWTH PROCEDURE

As described in the previous paragraph, with the existing programs it is not possible to include plasticity effects during fatigue crack growth simulation in general load cases,

when the crack path is unknown. For this reason, it will be presented a method and its technical implementation program, through which it is possible to simulate fatigue crack growth with an initially unknown crack path, taking into account elastic-plastic material behaviour. The calculations are based on the method of finite elements and the crack propagation can be simulated explicitly using adaptive remeshing [7].
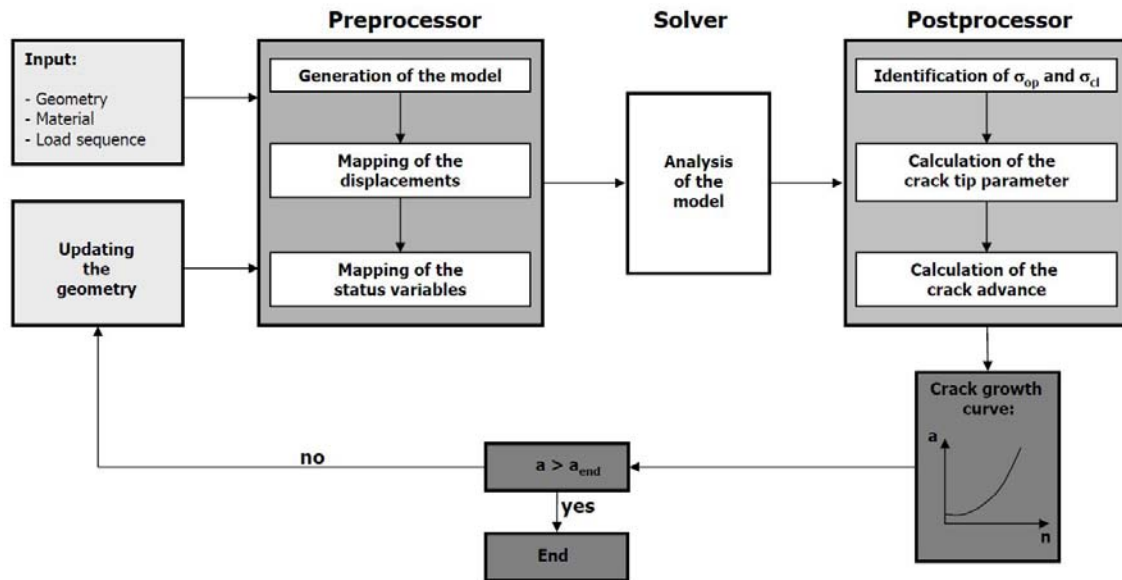

Figure 3 Crack growth procedure (picture from [5])

Figure 3 provides an overview of the modules of the method developed for the crack propagation simulation. The advantage of the modular structure is that the individual components can be each replaced without affecting the overall functionality. The required input data for the simulation are the geometry, including the information regarding the crack (location and length), the parameters for the material used in the model and the load sequence.

Since the used material model significantly influences the crack opening and closure level, and hence the crack growth behaviour (see e.g. [4], [8]), a model which is able to realistically describe cyclic plasticity effects such as ratcheting and mean stress relaxation has to be used. In this work, the advanced material model of Döring [6] is used. The constitutive equations of the model are implemented in a user subroutine to the commercial finite element software ABAQUS coupled by the UMAT interface with the plasticity model.

The finite element model is generated on the basis of the imported geometry and the load sequence. This model is already obtained from a modified geometry due to crack progress, as is explained below. Then both the displacements and the state variables from the model before the modification are transferred to this. The resulting model, with the material parameters, can then be processed by the solver of the finite element program. After that, within the postprocessor, the crack opening and closure level are determined and the effective range of the crack tip parameter is calculated for the subsequent determination of crack propagation. Based on these parameters, the crack

growth direction and the number of cycles to reach a given crack advance (or the crack advance for a prescribed number of cycles) can be computed by integrating an appropriate crack growth law. With that, the geometry is updated and the new model is generated. To re-establish the stress state prior to the crack advance, the displacements and the status variables have to be mapped from the old mesh to the new one. After the mapping of the variables the structure is analyzed again 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 the mapping of the variables, i.e. for linear-elastic material behaviour, the procedure was successfully validated in [9] for the determination of the crack growth path under mixed-mode loading.

To reduce the number of iterations the initial model contains an initial semi-circular crack; the length of crack initiation is set to 0.25 mm. Of course, there are cycles in order to reach this initial crack length, and these cycles produce also plastic deformations at the crack tip. To take into account these cycles, a model with an initial semicircular crack with radius a_i=0.1 mm must be created; this model has to be subjected to the autofrettage cycle (p_{af}=600 MPa), and after that single cycle a sequence of 250 cycles is performed; during the analysis a maximum pressure equal to 200 MPa and a minimum pressure of 5 MPa were used; by doing this the ratio of the cycles was R≈0. In Figure 4 the procedure to reach the crack initiation length is represented.
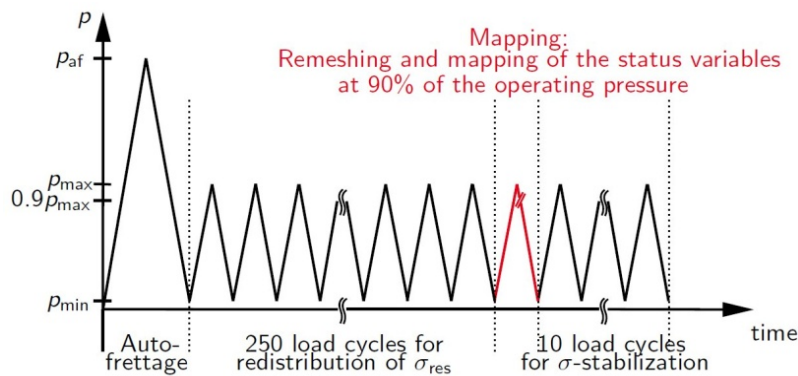

Figure 4 Load history during the crack growth procedure (picture from [5])

This sequence of 250 cycles had the purpose to induce the stress redistribution in the model. In fact, the effect of autofrettage for the service life extension can only come into play when the residual stress reduction reaches a nearly stabilized value. The numbers of cycles to reach this stabilization point is dependent on the material. Döring's constitutive model offers an opportunity to simulate this phenomenon, however, for the sake of a large numerical expense. Interrupting this simulation after 250 cycles was due to the observation that further residual stress changes became very small for one cycle and were continuously decreasing. After these 250 cycles, the displacements and the status variables have to be mapped at 90% of max operating pressure, to ensure to be in the elastic part of the material behaviour. At this point the crack length is increased by a fix increment of 0.05 mm and a new model with the new crack length is created and loaded until the 90% of p_{max}, where the variables and displacements of the old mesh are transferred into the new model, which is then submitted to other 10 load cycles. The crack length has to be incremented by this fix value of 0.05 mm for other two times until the crack reaches the length of 0.25 mm.

At that point the *real* crack growth simulation can start. In fact, from this point the effective J-integral along the crack front has to be evaluated and the crack growth is determined using the J-integral results. A new model with the actual crack front is then created, remeshed, remapped and submitted to other 10 load cycles to induce the plastic deformation and stress redistribution due to ratcheting and cyclic mean stress relaxation at the crack front. The effective J-integral is then evaluated and all the previous passages are repeated. The simulation is stopped when the number of cycles to reach the new crack length is very small. In this case, the analysis can be aborted because it is more or less a static crack growth.

THE MODEL

Geometry

The experimental tests, carried out by MFPA [5], have been performed using specimens shown in Figure 5. The material used was 42CrMo4 with R_m=938 MPa and $R_{P0,2}$=842

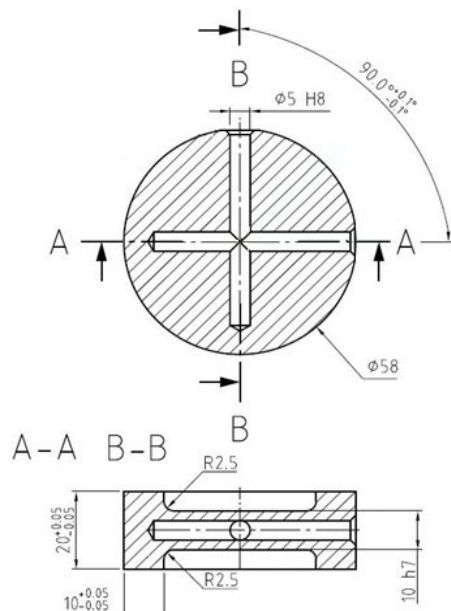MPa. The Q&T treatment has been performed after fabrication of the bore holes and then at the end it has been autofrettaged with 600 MPa. By taking advantage of existing symmetries, the model of the specimen was reduced to 1/16 (1/2 of thickness and 1/8 of circumference) and some FE analysis were performed to investigate the residual stress field after the autofrettage process. As it is possible to notice from Figure 6 and Figure 7, in which the stresses normal to the symmetry plane on the half of the thickness (S22) are shown, the autofrettage induces a residual compressive stresses zone 0.5 mm behind the corner, generated by the two crossing holes, with a maximum stress of circa -780 MPa and a high residual tensile stress zone, circa 590 MPa, at a distance of more or less 5 mm.

Figure 5 The specimen used for the experimental tests (picture from [5])

Döring Plasticity model

The residual stress field has been calculated by a finite element simulation of the loading and unloading part of the autofrettage cycle using Döring's constitutive model [6]. This model is able to capture the transient changes of the material's stress–strain curve as a function of prior maximum strain within one set of material parameters. The parameters themselves have been identified by calculating the model's response to experimentally determined material stress–strain curves. Additionally, this constitutive model is able to take into account ratcheting and mean stress relaxation during fatigue cycling. This option is exploited in the simulation of 250 cycles from zero to maximum

operating pressure following autofrettage. During this stage the initially introduced residual stresses are redistributed. The final result of that simulation is the residual stress field. In Figure 6, the resulting residual stress distribution along the bisector between the holes is displayed.
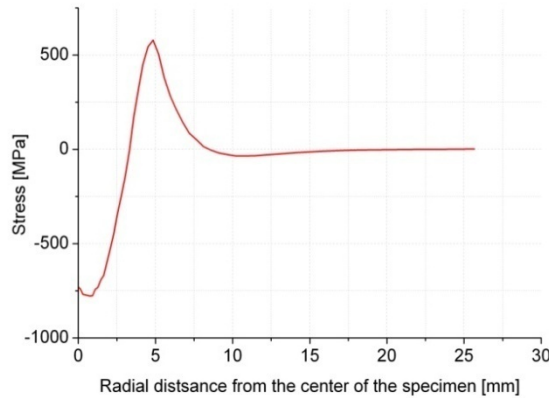

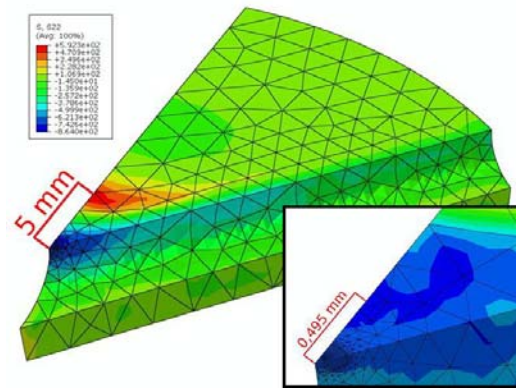**Figure 6 Residual stress normal to the crack surface along the symmetry edge.**

Figure 7 Residual stress normal to the crack surface after autofrettage and magnification near the crack.

Mesh

Concerning the mesh of the model, as is possible to notice from the Figure 8, both tetrahedral and hexahedral elements have been used. In particular, linear hexahedral elements (C3D8) have been used near the crack tip and quadratic tetrahedral elements (C3D10) have been used in the rest of the model. The mesh has been also refined at the crack tip due to high stress gradients.
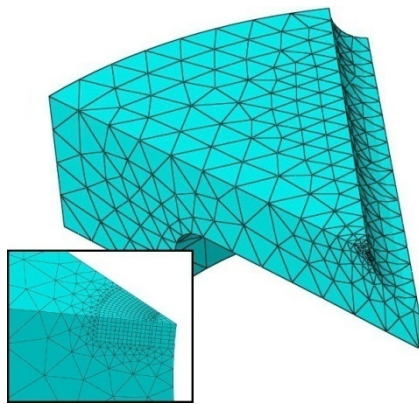

Figure 8 Mesh

During the crack growth procedure, after every crack increment a new model is generated. For transferring the state variable from one model to the other ABAQUS overlaps the two meshes and evaluates the values of the variables in each node of the new mesh using the results of the previous model; for doing this it was used an interpolation through the old nodes that are near the new node.

The crack growth

The crack growth is calculated by using the J_{eff} evaluated in the nodes that belong to the crack front and imposing a maximum increment (da_{MAX}). The crack increment is evaluated by using the Paris equation:

$$\frac{da}{dn} = C\,(\Delta J_{\text{eff}})^m$$

In this equation C and m are two material parameters and dn is correlated with da_{MAX} and ΔJ_{effMAX}. For each node of the crack front, the output is an increment da perpendicular to the crack front. The maximum increment da_{MAX} is not a fix value but a

percentage of the actual crack length. It has been fitted to this specific model through the adoption at the beginning of an increment of 10% that gradually has been reduced to 3,5% during the approximation of the zone with the maximum compressive stress.

Another element that has been evaluated is the distance between the nodes of the crack front. At the beginning, the crack front is not wide and for this reason in the first model there are only 15 nodes along the crack front; however, during the whole procedure the crack front becomes wider and the distance between two nodes tends to increase and furthermore all the node are no longer equidistant along the crack front. This can produce first of all a bad evaluation of the J-integral, if the mesh is not homogeneous before and beyond the crack tip [10]; then, if a node is too distant to the next one, it is also possible that the crack front approximation becomes very poor in precision so that it would be no more possible to appreciate curvatures and perturbations in the crack front. For all these reasons an allowed maximal distance between two nodes has been set, and if these max distance between two node is not respected, a new crack front node is added in the midpoint between the two. This maximal distance is based on the actual crack length and some geometrical parameters of the specimen.

ANALYSIS RESULTS

In the following figures the obtained results are shown. In particular in Figure 9 the crack fronts after every increment are represented. It is possible to notice how the crack length is shorter near the symmetry axis in concordance with the experimental results shown in Figure 9. In Figure 10 is represented the relationship between the crack length and the number of cycles.
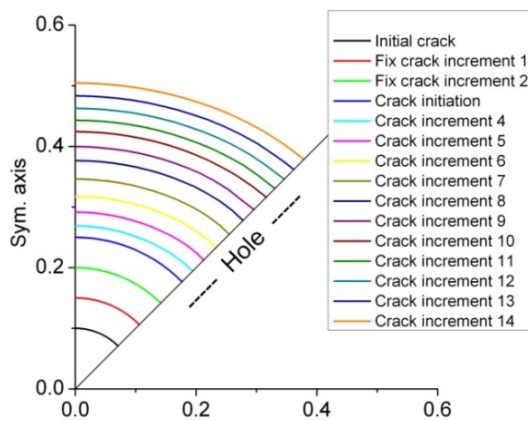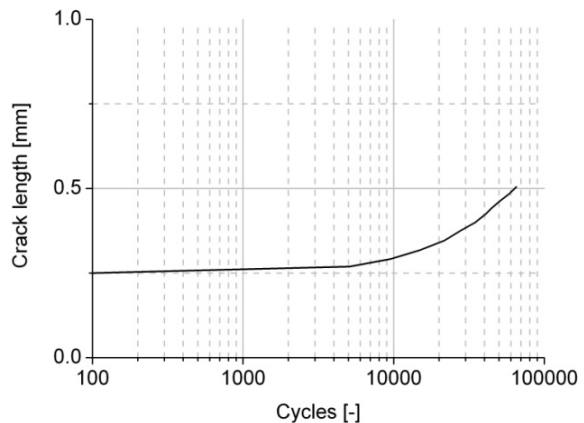
Figure 9 The crack fronts during the procedure   **Figure 10 Crack length versus number of cycles**

CONCLUSIONS AND OUTLOOK

In conclusion, it is possible to assume that with the actual work some important results for the explicit simulation of 3D crack growth has been achieved. Using the Döring material model it is possible to perform a set of simulations for the prediction of the crack front during the load history when the crack path is unknown.
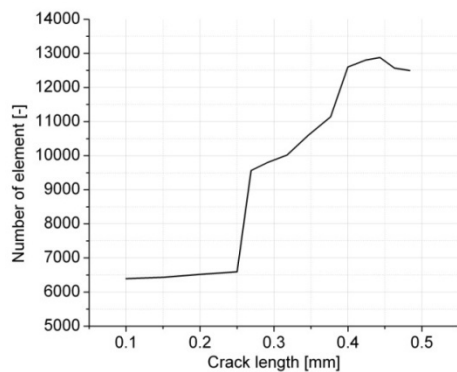
Figure 11 Number of element after every crack length increment

Still by using a preliminary set of material parameters, the procedure works, and it is possible to simulate an unusual crack front advance that is congruent with the residual stress field given by the autofrettage. Unfortunately, the procedure is still time consuming, (15 hours for every increment) and is strictly connected with the number of element in the model that increases when a new node in the crack front is added, Figure 11.

Due to the overwhelming numerical expense, it is still not possible to simulate crack paths like the one in Figure 2 in which the crack increment in the center of the specimen is very small. Crack stop happened in tests where the pressure is very low and consequently the stresses at the crack tip are very close to the threshold values of the material. The low pressure and the residual stress field due to autofrettage induce a situation in which the crack length is almost stopped in the center of the specimen and continues to increase near the edges of the model.

BIBLIOGRAPHY

1. Elber, W. (1970) *Eng Fract Mech* **2**, pp. 37-45.
2. Antunes, F., Borrego, L., Costa, J., Ferreira, J. (2004) *Fatigue Fract Eng Mater Struct* **27**, pp. 825-835.
3. Herz, E., Thumser, R., Bergmann, J., Vormwald, M. (2006) *Eng Fract Mech* **73**, pp. 3-21.
4. Jiang, Y., Feng, M., Ding, F. (2005) *Int J Plasticity* **21**, pp. 1720-1740.
5. Thumser, R., Herz, E., Hertel, O., Bergmann, J.W., Vormwald, M. (2009) *Betriebsfestigkeit gekerbter Hochdruckbauteile ohne und mit Autofrettage* / Forschungsvereinigung Verbrennungskraftmaschinen e.V. (FVV). (Heft 876).
6. Döring, R., Hoffmeyer, J., Seeger, T., Vormwald M. (2003) *Comput Mater Sci* **28**, pp.587–96.
7. Zerres, P., Vormwald, M., (2012) *Comput Mater Sci,* doi: 10.1016/j.commatsci. 2012.01.018
8. Rodrigues, D.M., Antunes, F.V. (2009) *Eng Fract Mech* **76**, pp. 1215-1230.
9. Zerres, P., Vormwald, M. (2009) *Modelling of crack propagation under mixed mode loading with advanced remeshing*. Proceedings of the 12th International Conference on Fracture.
10. Okada, H., Higashi, M., Kikuchi M., Fukui, Y., Kumazawa, N. (2009) *Eng Fract Mech* **72**, pp. 1717-1737.