Study of Fracture Mechanics Assessments in Industrial Applications

Bartolomeus Irwanto[1,*], **Thorsten Prothmann**[1]

[1] Alstom, Zentralstrasse 40, 5242 Birr, Switzerland
* Corresponding author: bartolomeus.irwanto@power.alstom.com

Abstract

Fracture mechanics has been widely used for mechanical integrity assessment in industrial applications. Different approaches in the fracture mechanics assessment have been developed to achieve more accurate prediction of component lifetime. This paper describes fracture mechanics assessments by applying the 2D and 3D crack growth analyses of a real mechanical component under cyclic load as an example from the power industry. In 2D crack growth analysis, both an appropriate equivalent geometry as well as the original geometry with an explicit crack model are investigated. A limitation of 2D crack growth analysis in the example is to consider load redistribution, which follows from the local stiffness change attributed to the cyclic crack growth. This can be overcome by 3D crack growth analysis with an explicit crack model. Comparison between 2D and 3D assessment results are given. The concept of local limit load and failure assessment diagram (FAD) are reviewed and studied by using finite element (FE) analysis.

Keywords fracture mechanics, crack growth, load redistribution, limit load

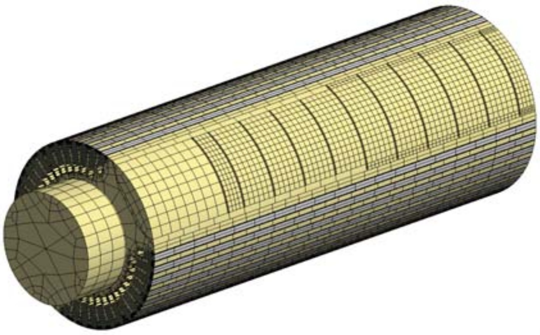
1. Introduction

Industries have been challenged to come up with better and less costly products within shorter development cycles. Thus, product development processes have also needed to be improved. This has lead to a tremendous challenge for engineers, to continuously develop improved approaches including fatigue life time assessment of a component. In the lifetime assessment it is important to consider both crack initiation and crack propagation. Fatigue failure is caused by microscopic damage in the material that after continued cycling develops into a crack that may finally lead to a component failure. Generally the design should avoid the initiation of fatigue cracks during the design life time. However one has to assume the existence of initial flaws in the raw material, e.g. forgings. In order to determine criteria for the non-destructive testing of forgings, appropriate investigations of crack propagation are required. This paper focuses on these numerical crack propagation investigations.

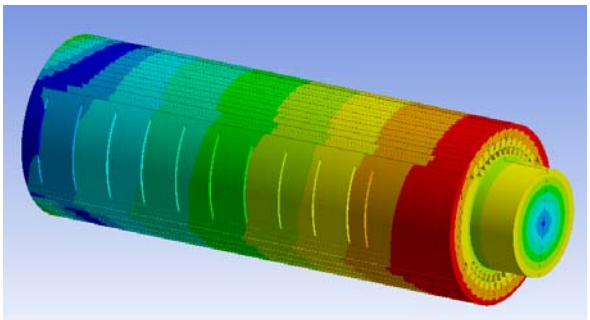
In this paper, the application of fracture mechanics for mechanical integrity assessment is shown in industrial applications. Different approaches in the fracture mechanics assessment have been developed to achieve more accurate prediction of component lifetime. This paper describes fracture mechanics assessments by applying the 2D and 3D crack growth analyses of a real mechanical component under cyclic load as an example from the power industry. Some limitations in the application of crack growth simulation in 3D analysis and the potential use of a failure assessment diagram (FAD) are discussed.

2. Mechanical model

In the shaft trains of large steam turbine generator sets, the mechanical integrity of each component of the shaft train has to be verified. The rotating parts such as generator and steam turbine rotor have to be particularly verified with accurate lifetime assessment methods. In this work, a generator rotor is taken as an example for fracture mechanics assessment. Figure 1 shows the FE-model of a generator rotor and one example of analysis results by using ANSYS [8]. A generator rotor has complicated geometries and components such as insulation, conductors and wedges. The calculation of a complete 3D rotor model is very time consuming. Therefore, a 2D model can be used to reduce the computational efforts. In 2D crack growth analysis, a stress analysis should be performed beforehand. A 2D model of generator rotor is depicted in Fig. 2. The corresponding 2D elements in ANSYS are applied in the mesh generation.



(a)



(b)

Figure 1. FE model (a) and displacement results (b) of a complete 3D generator rotor

The dominating load in a generator rotor is due to centrifugal forces, i.e. the main contribution for cyclic rotor loading is related to the start/stop cycles of the shaft train. The number of design start/stop cycles is specified in IEC 60034-3. Thus, the centrifugal load at rated speed is considered for stress analysis. For 3D crack growth analysis, a simplified 3D model is created as shown in Fig. 3, where a rotor tooth is modeled. The influence of conductors and wedges is replaced with equivalent pressure applied at the rotor tooth flank.

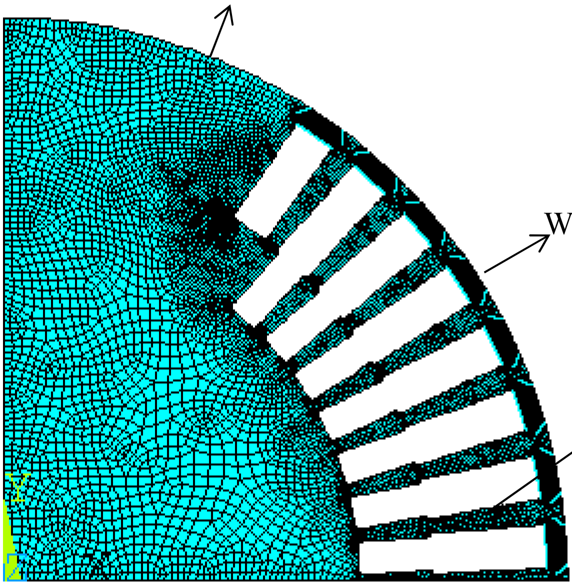


Figure 2. 2D FE model of generator rotor

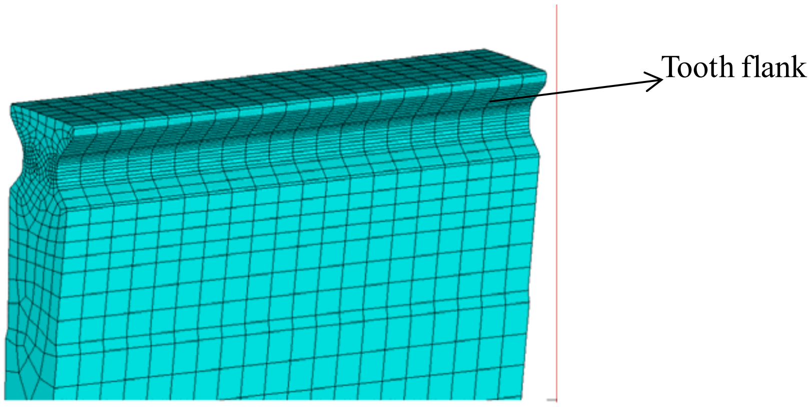


Figure 3. Simplified 3D FE-model for calculation and crack growth analysis

3. Crack growth analysis

3.1. 2D crack growth calculation

To calculate 2D crack growth the program AFGROW [1] is used in this example. AFGROW is an analytical fatigue crack growth prediction program initially developed in AFRL for airframe applications and based on the principles of Linear Elastic Fracture Mechanics (LEFM) [1]. AFGROW calculates fatigue crack growth by considering known stress intensity solutions for a particular geometry, under an applied stress field. A stress intensity solution has the following general form:

$$K = \beta\sigma\sqrt{\pi a} \qquad (1)$$

in which ‘K’ is the stress intensity factor, ‘a’ is a characteristic crack length, ‘σ’ is the applied stress, ‘β’ is the geometrical factor. As the stress intensity solutions are dependent on the geometry of the components, AFGROW provides certain standard predefined geometries (crack models) for predicting crack propagation.

To create the initial crack, a surface crack model is selected as shown in Fig. 4. As input data, the corresponding model geometry and dimensions should be defined. The crack growth rates are expressed in the form of the Walker equation, which is simplified to a Paris equation. In AFGROW the stress gradient can be included in the analysis. The analytical solution for this crack model is described in [2], which is implemented in AFGROW. It is assumed that the first principal stress is always normal to the crack plane. The calculated first principal stress from the 2D finite element model described previously is shown in Fig. 5, where the stress path can be seen. Using the stress gradient along the stress path, the crack growth calculation is performed and the normalized results are depicted in Fig. 6. The crack size ‘a’ corresponds to the crack growth in the circumferential direction and crack size ‘c’ to the axial direction.

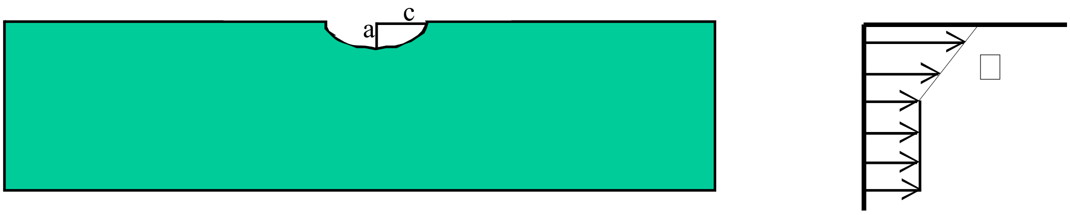


Figure 4. Crack model in AFGROW and stress gradient (schematic)

The application of the crack model has to satisfy a limit that the ratio of crack size ‘a’ and plate thickness ‘t’ must be less than or equal to 0.8 ($a/t \leq 0.8$). The reason of this limitation is according to Glinka that the reference solutions used for the derivation of the weight function were valid for this range (see also [2]). Further investigation shows that the reference solution is related to the local limit load which causes local crack-ligament yielding somewhere along the crack front [5]. The local limit load will be further discussed in the next section.

2D crack growth analysis is a straight-forward approach, where the FE results can be used directly in the crack growth calculation. Its numerical effort is also acceptable. However, 2D crack growth analysis is limited by the fact that for larger crack extension compared to the geometrical dimension of the analyzed component possible load redistribution cannot be considered. The load redistribution occurs as the local stiffness around the crack changes more significantly as the crack becomes larger. Moreover, the assumption that the 1st principal stress is always normal to the crack plane may lead to a conservative result. The load redistribution may have a significant impact as demonstrated in the next section with the results of the 3D crack growth analysis of the same component.

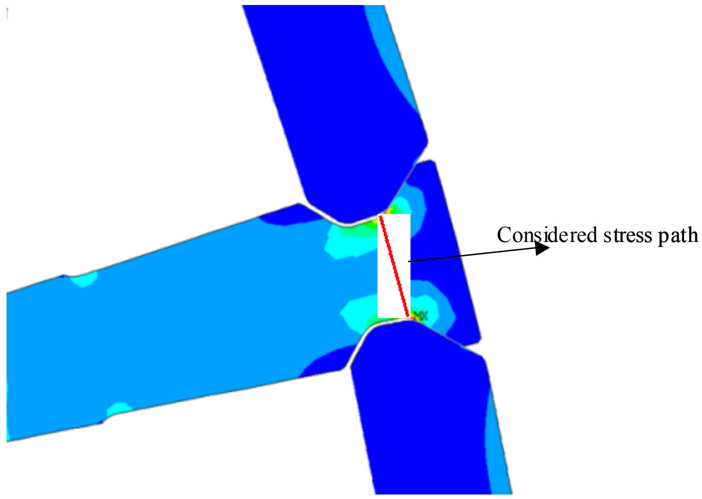


Figure 5. 1st principal stress distribution of rotor due to centrifugal load

3.2. 3D crack growth calculation

A critical issue in 3D crack growth calculation is the mesh generation. In general cases, the use of a standard FE program faces time-consuming challenges such as:

- Component geometries are complex and time consuming to model, where cracks can occur at geometrically difficult locations, e.g. chamfers, corners.
- Initial cracks of the correct size and shape must be inserted in the component at the correct location.
- After several load cycles and depending on the loading, initial planar crack may develop into a non-planar crack.

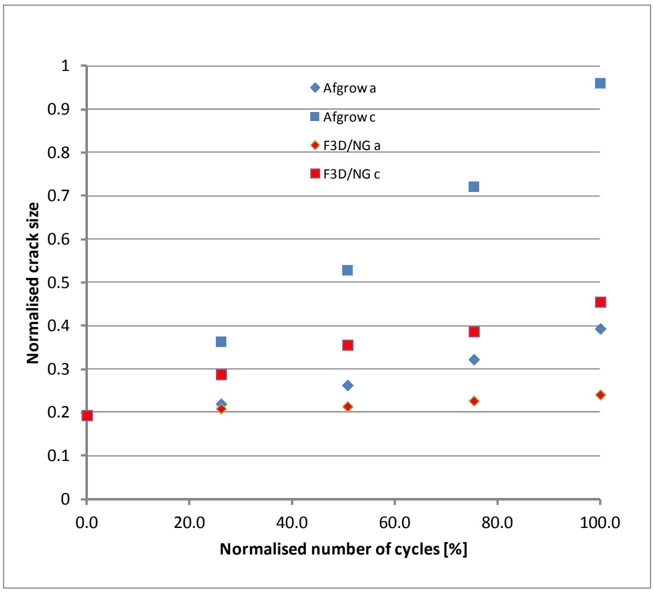


Figure 6. 2D and 3D crack growth calculations and their comparison

To alleviate the above challenges, there are several programs existing that allow users to generate/insert a crack into the FE model. In this work, the program FRANC3D/NG (short name F3D/NG) version 4.0 has been used. The methodology of F3D/NG has been documented in [3]. In F3D/NG the stress and displacement fields in the structure of interest are obtained from a Finite Element program such as ANSYS or ABAQUS [9].

The results of the FE analysis as well as geometry are read directly into F3D/NG in a text format. The F3D/NG program uses the FE program as solver to obtain the stresses and displacements after growing the crack at a user defined step size. Hence, by updating the stresses and displacements for each crack growth step with the FEA method, the simulation can more closely represent the crack's influence on the structure. The crack insertion and all re-meshing are carried out within F3D/NG in the submodel created before by using the FE program. The F3D/NG will produce a neutral format file that can be read back into the FEA code and re-analysed.

In F3D/NG the stress intensity factors of Mode I, II and III are calculated by using M-Integral (interaction integral) for isotropic and anisotropic materials [3, 4]. F3D/NG can also compute stress intensity factors using a displacement correlation approach. In this approach, the calculated displacements for nodes on the crack faces are substituted into the theoretical expressions for the crack-front displacement fields as functions of the stress-intensity factors. The applied propagation direction criterion is based on the maximum stress criterion. In this method, depending upon user's choice, the mode I and mode II with/without mode III values are combined to yield both magnitude and direction of the propagation at the various discrete points along the crack front.

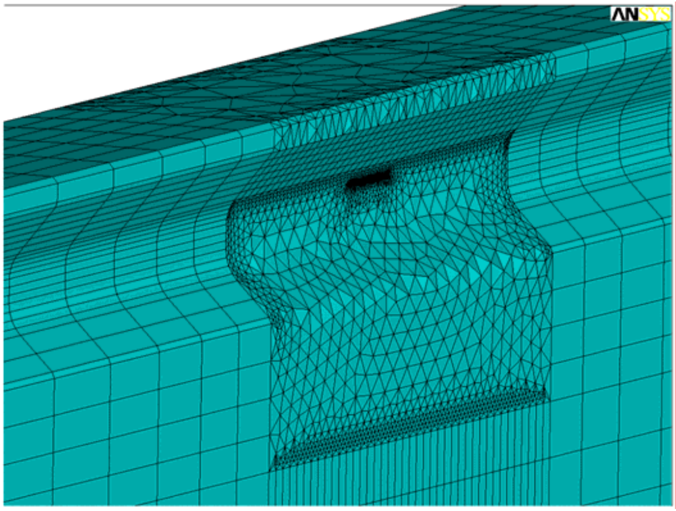


Figure 7. Example showing the submodel with finer mesh and inserted crack

Fig. 7 shows the global model and the submodel having finer mesh and inserted crack. The applied element type is an element with a quadratic shape function. The calculated equivalent stress field is shown in Fig. 8. By using the same parameter for the Paris equation as for 2D crack growth, 3D crack simulation with the same initial crack size is carried out. Small size steps and other meshing options are defined in order to get optimal results. The calculated crack sizes in both directions are presented in Fig. 6. It is clearly observed from Fig. 6 that there is a significant difference in the results from the 2D and 3D crack growth calculation yielding an overestimation of the cyclic crack growth with the 2D model. This demonstrates that the influence of load redistribution as well as the stress direction to the crack plane is significant for crack growth analysis.

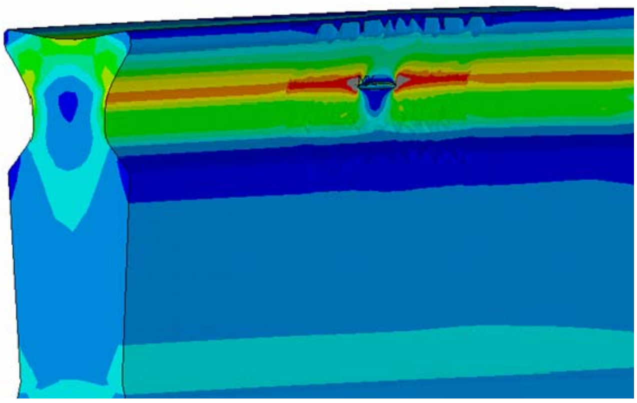


Figure 8. Stress field of model with a crack

Conventional limit load analysis computes the global (net-section) limit load, at which displacements become unbounded. However, the global limit load might be taken conservatively for a part-through crack. Thus, local limit load is proposed and defined as the load needed to cause a local crack-ligament yielding somewhere along the crack front [5]. In the paper, the local limit load is derived by finite element analysis of a plate with surface crack. In order to check the local limit load of the rotor, elastic-plastic analysis with ideal plastic material behavior is performed on the model with sequentially increased crack sizes. The result analysis shows that the local load limit is achieved at a/t=0.8. Strain distribution at ligament yields that the limitation of up to a/t=0.8 is also valid for the component assessment.

3.3. Failure Assessment Diagram (FAD)

The failure assessment diagram (FAD) method is the broadly accepted methodology for the analysis of components containing a crack-like flaw. The FAD method is described in the engineering code, for example API 579 and FKM guideline for fracture mechanics [6]. The FAD method uses brittle fracture ratio (K_r) and plastic collapse ratio (L_r). In a FAD the non-dimensional ratio L_r is on the horizontal-axis and K_r (brittle fracture ratio) on the vertical-axis. The plastic collapse ratio is a parameter measuring the proximity to plastic collapse. The brittle fracture ratio is a parameter comparing the stress intensity factor at the crack and the fracture toughness of the material. The FAD curve can be computed if the stress-strain curve exists. In this case, the plastic collapse ratio is determined by using the reference stress, which can be computed using the J-integral results from the elastic-plastic analysis. The brittle fracture ratio is computed from the crack front stress intensity, obtained by an elastic analysis.

Fig. 9 shows a graph of FAD and the evaluation point for the example calculated in the previous section, where the ratio a/t is less than 0.5. In Fig. 9, the evaluation point inside the FAD curve indicates an acceptable crack. The evaluation point represents the quasi static condition of a crack/flaw at certain size and loading. The FAD curve is not created by using a specific structural component and a stress-strain curve. The FAD curve is derived from the following equations [6]:

$$F1(L_r) := \begin{cases} \left(1 + \frac{L_r^2}{2}\right)^{\frac{-1}{2}} & \text{if } L_r < 1 \\ \left(\lambda + \frac{1}{2\lambda}\right)^{\frac{-1}{2}} & \text{if } L_r = 1 \\ \left[\left(\lambda + \frac{1}{2\lambda}\right)^{\frac{-1}{2}} \cdot L_r^{\frac{N-1}{2N}}\right] & \text{if } L_{rmax} > L_r > 1 \\ 0 & \text{if } L_r \geq L_{rmax} \end{cases} \tag{2}$$

in which λ and N are parameters given in [6], which depend on material yield strength as well as ultimate tensile strength. The above equations are based on basic level assessment, which is more conservative than the use of stress-strain curve in FAD. However, the computed FAD curve with stress-strain curve can provide a better representation of the particular material effect and the particular structural component geometry.

If the evaluation point is above the FAD curve, the crack is unacceptable and this can indicate a predicted structural failure. The evaluation point can be updated as the crack grows to determine the end of life when the evaluation point reaches the FAD curve. Thus, an evaluation point on the FAD curve can be useful to determine predicted critical crack sizes.

4. Conclusions

This paper demonstrates the application of fracture mechanics in industrial practice. 2D crack growth analysis provides significantly less computational effort but can yield over conservative results, since the load redistribution and the actual stress state are not taken into account.

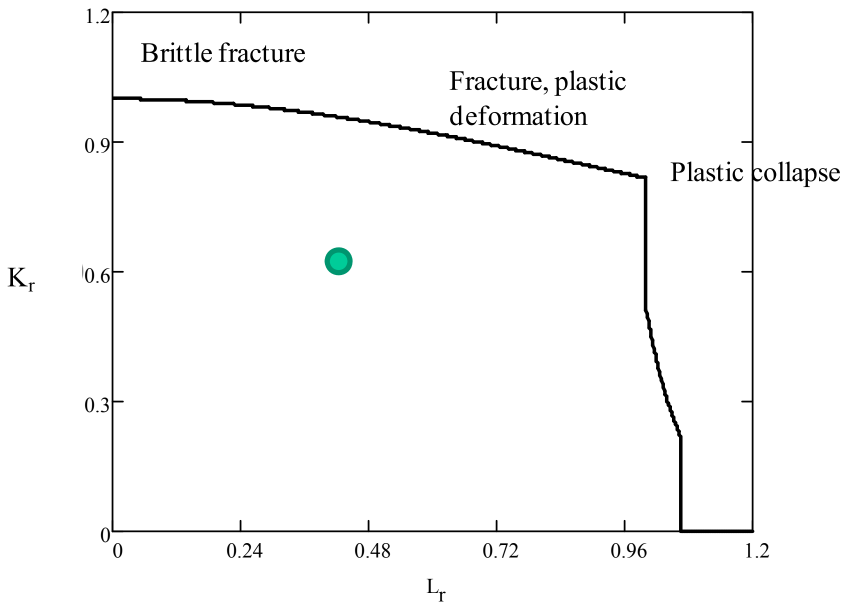


Figure 9. Calculated FAD showing the acceptable evaluation point

These weaknesses can be overcome by 3D crack growth analysis but it requires significant computational efforts. The use of submodelling in 3D crack growth analysis may reduce the computational time; however, from the practical point of view it is still a big challenge to use this method for fast development projects. Application of XFEM in the 3D crack growth calculation might offer an opportunity to extend the use of 3D crack growth analysis in lifetime assessment. However, the use of commercial XFEM code should be reviewed critically due to the result accuracy. Different strategies are currently developed for damage calculation to improve the accuracy and convergence rate of XFEM. Instead of re-meshing in the region of crack tip elements, a higher order element approach, different enrichment functions and integration rules can be applied (see e.g. [7]). Finally, the FAD assessment can be used to ensure that a component with a certain crack/flaw size is acceptable for the design load.

Acknowledgements

The authors would like to thank Alstom for allowing this publication.

References

[1] James A. Harter, AFGROW User Guide and Technical Manual, Air Force Research Laboratory, June 2006.
[2] G. Shen, A. Plumtree, G. Glinka, Weight function for the surface point of semi-elliptical surface crack in a finite thickness plate, Engineering Fracture Mechanics, Vol. 40 (1), 167-176, 1991.
[3] Fracture analysis consultants, FRANC3D/NG Reference Manual, 2007.
[4] P. A. Wawrzynek et al., Advances in Simulation of Arbitrary 3D Crack Growth using FRANC3D/NG, ICF 12, Ottawa, 2009.

[5] I. Sattari-Far, P. Dillström, Local limit load solutions for surface cracks in plates and cylinders using finite element analysis, International Journal of Pressure Vessels and Piping 81, 57-66, 2004.
[6] FKM guideline Fracture mechanics proof of strength for engineering components, 3th Edition, 2006.
[7] E. Sala-Lardles, et al.: Optimally convergent high order X-FEM for problems with voids and inclusions, ECCOMAS 2012, Vienna.
[8] ANSYS Inc., ANSYS User's Guide, 2007.
[9] Dassault Systèmes, Abaqus/CAE User's Manual, 2012.