AUTOMATIC 3D CRACK PROPAGATION IN COMPLEX WELDED STRUCTURES

J. Martinsson
Department of Aeronautical and Vehicle Engineering, Royal Institute of Technology, SE-100 44 Stockhom, Sweden
e-mail: jma@kth.se

Abstract
In this paper an automatic crack propagation program using the FE program ANSYS® is developed. The program evaluates the Stress Intensity Factors (SIF) along the root side of a weld line and automatically updates the size and shape of the crack front. An example of how to apply the program to a welded component and fatigue test of the component is shown at the end of the paper.

Introduction
The life prediction of welded structures based on FEA can be done in several different ways with quite different level of accuracy. For complicated structures, mesh size, type of elements, introduction of loads and boundary conditions strongly affect the result. Over the years numerous methods have been developed to predict the fatigue resistance of welded structures. These methods have naturally been developed and verified based on tests of small scales specimens. The suitability of the four most common methods for fatigue analysis of complex structures is qualitatively illustrated in figure 1.

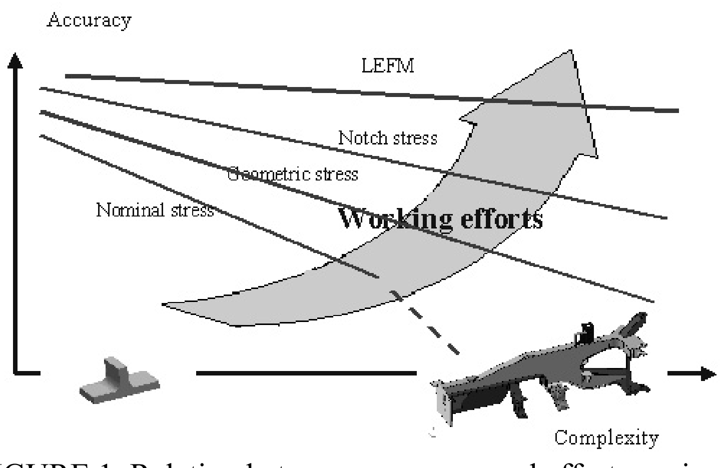


FIGURE 1. Relation between accuracy, and effort required for fatigue analysis of welded joints as a function of structural complexity.

For example, the nominal stress method upon which most fatigue design codes are based requires the least amount of computational effort but also decreases rapidly in accuracy as the structural complexity increases. The geometric or "hot spot" approach is only suitable for failures from the weld toe i.e. no failure from the root side. The LEFM (Linear Elastic Fracture Mechanics) and the notch stress based approaches can handle failures from both toe and root side and gives potentially high accuracy even for complex structures, but the need of modelling and computational effort increases.

The accuracy indicated in figure 1 is more or less verified by life predictions and tests of different complex welded structures made by the authors Byggnevi [1], Petterson [2,3] and Martinsson [4,5]. A general problem in all these investigations was the difficulties to model the crack growth from the root side. There is a need of a 3D automatic crack propagation program to increase the accuracy and speed of life predictions. Based on the author's investigations some specified demands were set up and several academic and commercial programs were examined. Although the development in this area has made large progress during the past years none of the investigated programs did fully satisfied the specified demands. In this paper an automatic crack propagation program using the FE program ANSYS® is presented. The program evaluates the Stress Intensity Factors (SIF) along the root side of a weld line and automatically updates the size and shape of the crack front. An example of how to apply the program to a welded component is shown at the end of the paper.

Program overview

The program requires one global model with or without modeled welds and one detailed submodel of the investigated weld. There are no restrictions of how many weld submodels that can be evaluated at the same time. It's also possible to apply contact elements on the crack surfaces if necessary. The sub-model is made in the GUI (Graphic User Interface) mode in ANSYS®. This will make it easy to build sub-models from an already existing global model. The global model can preferably be meshed with tetrahedral elements, which was one of the strongest specified demands. The program automatically calculates ΔK_{I}, ΔK_{II} , ΔK_{III}, crack shape and size due to the σ_1-criterion and ΔK_{eq} along the crack front. One or two load cases can be applied to the global model to get the range of ΔK along the crack front. Contact elements can be applied to the crack surfaces if necessary. A distance Δa_{max} is specified as the maximum increment of movement for the crack front at every step. Any other location along the crack front gets a smaller increment. In case of several investigated crack fronts they can be moved relatively to each other. This is a very powerful tool to study how different cracks interact with each other.

Program Flow

The program is written in APDL (Ansys Parametric Design Language). A flow chart of the program is shown in figure 2.

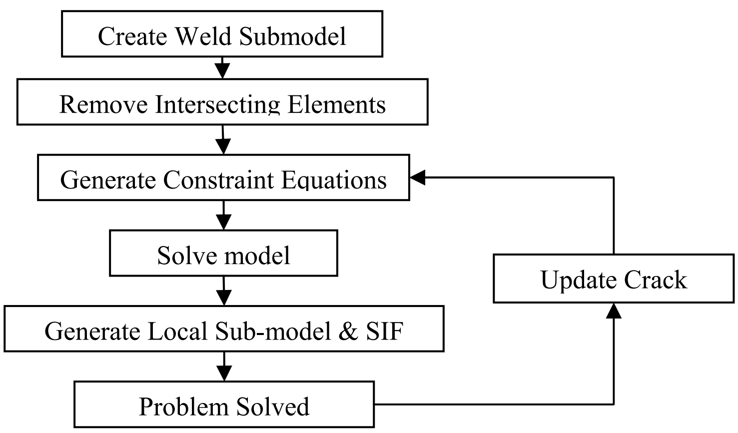
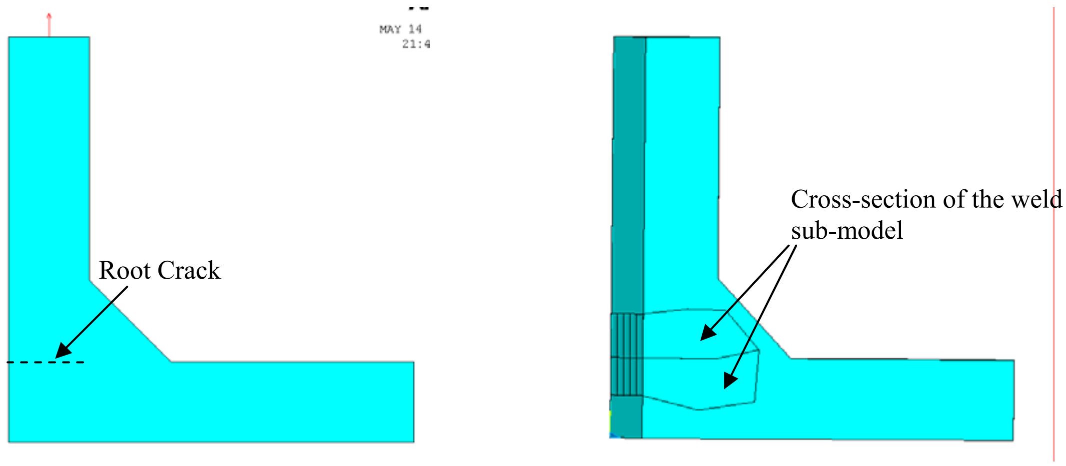


FIGURE 2. Flow Chart of the Program.

Weld Submodel

The weld submodel is very easy to create in an already existing globally meshed model. A cross-section of the weld submodel, normal to the weld geometry is made in the GUI (Graphical User Interface) mode in ANSYS®. The cross-section of the weld sub-model is swept along the weld line to create the submodel, see figure 3. This procedure is done once for every investigated weld. Several welds can be examined at the same time.



Global model Global model & Weld submodel

FIGURE 3.

The weld sub-model contains several meshed volumes with local coordinate systems along the crack front. The SIF are calculated at every location of the local coordinate systems. The distance between the local coordinate systems can be set in the program.

Remove Intersecting Elements & Generate Constraint Equations

In this procedure the elements of the global model which lies within the volume of the weld sub-model will be removed and constraint equations will be generated between the global model and the sub-model, see figure 4a.

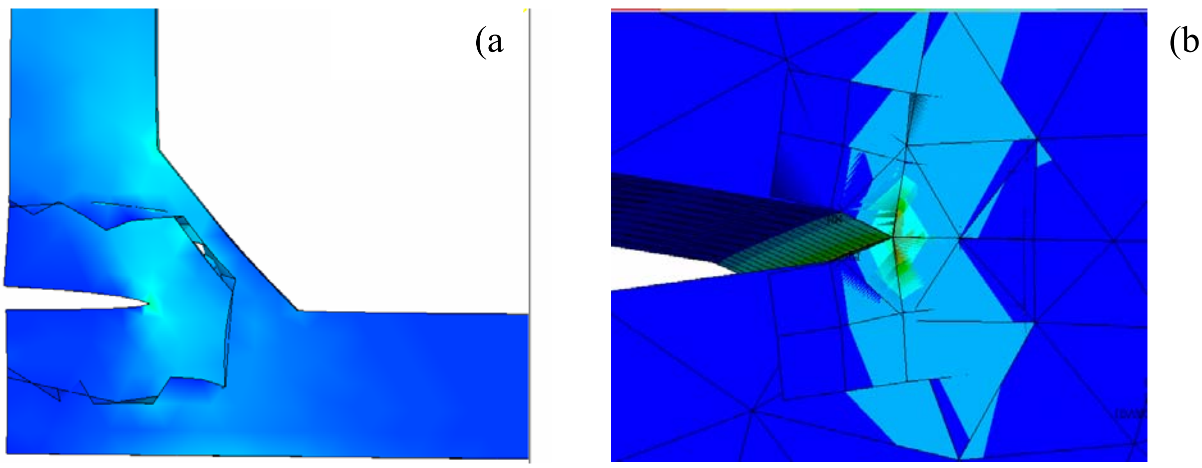


FIGURE 4. Global & Submodel tied together with constraint equations.

This is different from the traditional sub-modeling technique, where the displacement field of the global model is applied as boundary conditions on the sub-model. If traditionally technique is used the global model must be geometrically correct. This means that the global model must contain the actually crack length, otherwise the SIF will be incorrect due to the reduced stiffness of the sub-model. This is often a very time consuming task to both update the global model and the sub-model. If pure tension is applied to a rectangular block with a cross section of 10x100 mm with a through crack of 2-10 mm, the SIF calculated by the developed program showed ~6 - 7 % lower SIF values than the Fracture Mechanics Program AFGROW [6].

Generate Local Submodel & SIF Solutions

The new modified model, including global and sub-model is solved in ANSYS®. When the modified model is solved, the SIF along the crack front can be evaluated at the local coordinate systems of the crack front. However the problem is that the elements around the crack front are tetrahedral elements which make it impossible to achieve the SIF. To solve this problem a small local sub-model with 20-nodes hexahedral elements with singular elements [7] is generated along the crack front. Figure 4b shows the 20-nodes sub-model and the weld sub-model.

Due to the geometrically correctness, the traditional displacement based submodelling technique can be used to solve the local submodel. The following quantities are then calculated along the crack front, K_I, K_{II}, K_{III}, φ, ΔK_{eq} & Δa. ΔK_{eq} is the equivalent stress intensity factor due to Eq. (1), purposed by Richard [8], φ is the kink angle in the K_I & K_{II} plane, due to Eq. (2), purposed by Erdogan & Sih [9].

$$\Delta K_{eq} = \frac{\Delta K_I}{2} + \frac{1}{2}\sqrt{\Delta K_I^2 + 4(\alpha_1 \Delta K_{II})^2 + 4(\alpha_2 \Delta K_{III})^2} \qquad (1)$$

,where α_1=1.155 and α_2=1.0.

$$\varphi = -\arccos\left(\frac{3K_{II}^2 + K_I\sqrt{K_1^2 + 8K_{II}^2}}{K_1^2 + 9K_{II}^2}\right) \qquad (2)$$

The growth increment Δa along the crack front is calculated according to the following procedure. The number of necessary loading cycles due to Paris law, da/dN=f(ΔK_{eq},R), can be approximated by Ni=Δa_{max}/ f($\Delta K_{eq,max}$,R). The application of this N_i to the local crack growth rate at any other location along the crack front results in smaller increments Δa_i=N_i*f($\Delta K_{eq,i}$,R) at those locations. If two load cases are applied to achieve ΔK_{eq}, the R-values will be calculated at every local coordinate system along the crack front by Eq. (3), and the effective stress intensity factor $\Delta K_{I,eff}$ will replace ΔK_I in Eq. (1), where $\Delta K_{I,eff}$ is calculated by Eq. (4), see Maddox *et al* [10].

$$R = \frac{\Delta K_{I,\min}}{\Delta K_{I,\max}} \qquad (3)$$

$$\Delta K_{I,eff} = U \cdot \Delta K_I \qquad \text{where, } U = 0.72 + 0.28 \cdot R \qquad (4)$$

Update Crack Geometry

Combining the crack increments Δa_i and the deflection angle φ_i will generate a new crack front, see figure 5.

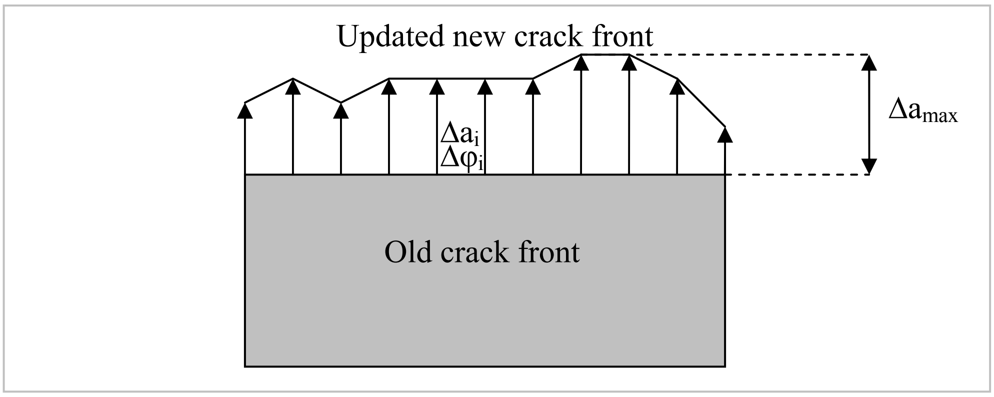


FIGURE 5. Updated new Crack front.

The angles φ_i, along the crack front, are defined after the first complete load cycle. The crack angles are then kept constant and will not be updated during the crack growth process. The in plane modeled crack front in ANSYS® contains 4 lines and 5 KeyPoint's (KP) with a local coordinate system at the crack tip, see figure 6.

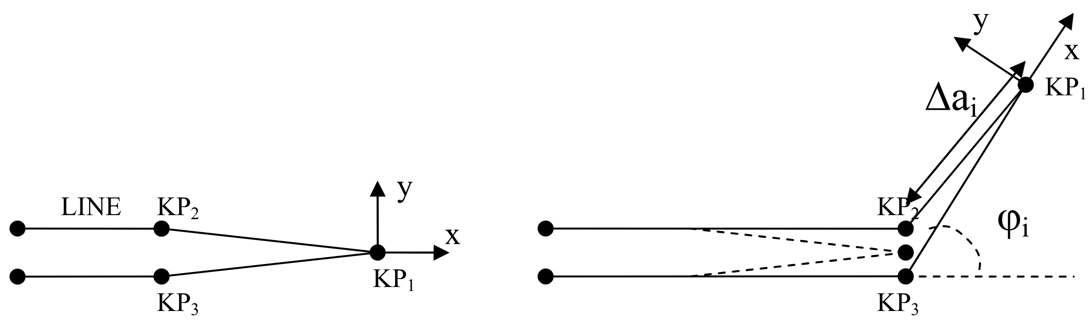


FIGURE 6. Modeled cross-section of the Crack in ANSYS®.

Figure 6 also shows the crack before and after a specified crack growth increment. The KP´s 2 & 3 are moved forward to the old crack front, while KP 1 is moved to the new crack front. The smallest crack growth increment applied in every load step is equal to the size of the local sub-model. The new updated weld sub-model is then remeshed with tetrahedral elements and the program returns to block 3 in figure 2.

Simulation Example

Investigated component, a link

In this example a fatigue-loaded link from a wheel loader is analysed, see Martinsson [4]. The link is a part of the loading unit, see Figure 7 and 8, and the function of the link is to transfer force from a hydraulic cylinder into the bucket. The link is manufactured from five gas-cut plates in high strength steel. The overall length is about 1200 mm. Figure 8b shows the different types of welded joints in the link, W1-W3. There are approximately 9x2 (2 according to the symmetry condition) critical points (toes, corners and roots) were fatigue cracks can start.

FIGURE 7. Wheel loader.

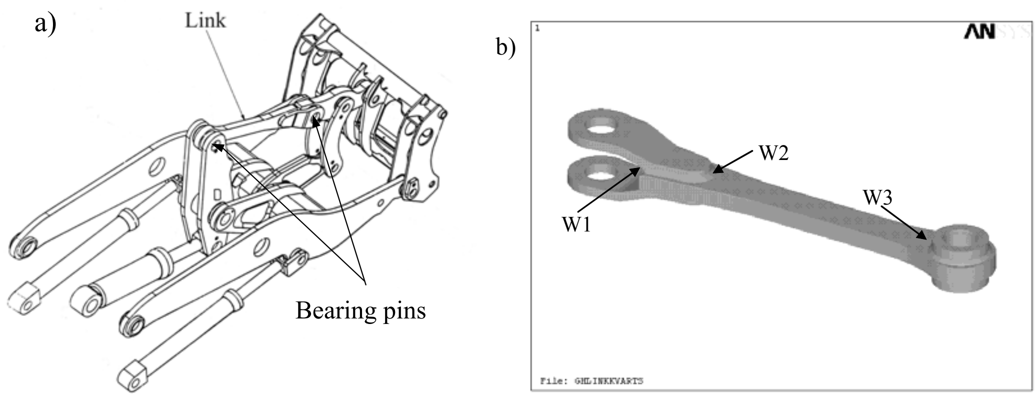


FIGURE 8. Schematic picture of the placement of the link and the critical welds. W1-W3 are the welds that will be investigated according to fatigue.

Martinsson [4] contains the stress analysis, the life predictions and the variable amplitude test of the link. In this investigation the root sides of the welds W1-W3 are examined with the proposed method. Figure 9 shows the ¼ global model with the investigated weld submodels W1-W3

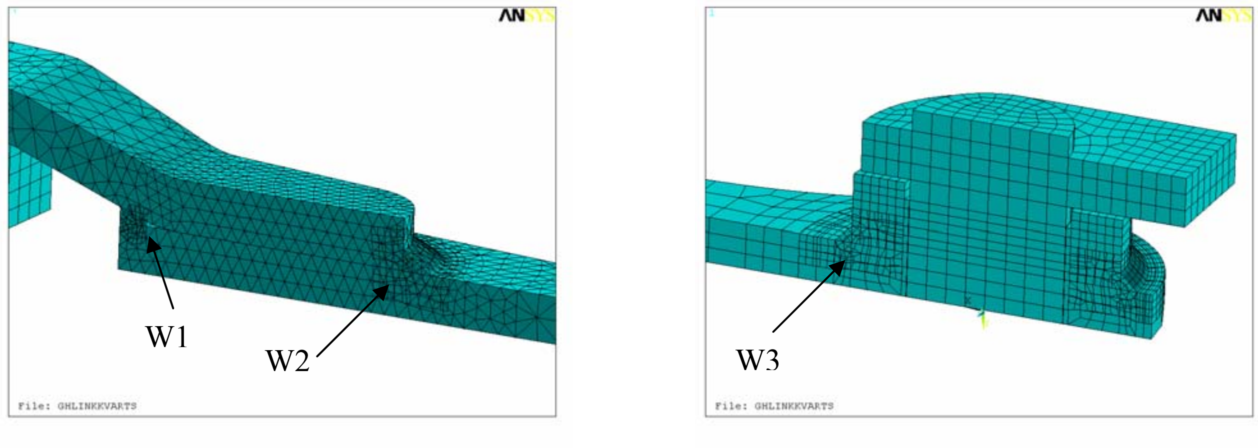


FIGURE 9. Global model with weld submodels.

The fracture occurred in the base material of the fork, see figure 10, and from the root side of weld W3 (ear), see figure 11.

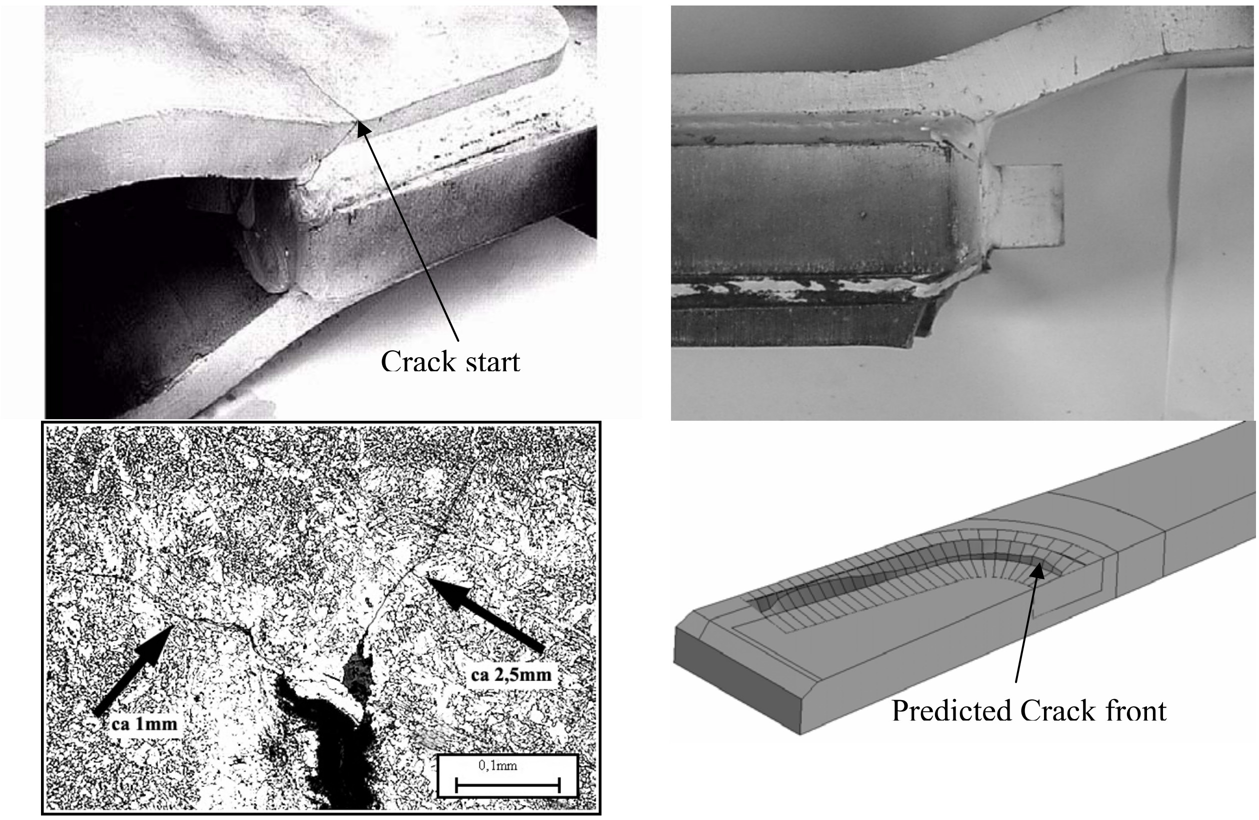


FIGURE 10. Fracture and predictions of the Fork end

The SIF in W1 was very low compared to SIF in W2 & W3 i.e. no crack growth in W1. According to the predictions, fatigue cracks in the root of W2 were expected. However no visible cracks were detected. The link was cut into the middle of W2 to be able to see if there were any fatigue cracks. Figure 10 shows that there are small fatigue cracks in the root of W2, as expected. Predicted crack growth in the middle of the fork is ~1.25 mm, which is in order with the laboratory test. As can be seen in figure 10 the crack growth in the fork has started in the base material. If a semi elliptical crack with a_0=0.3 and c_0=0.3 is applied at the edge of the fork (Crack start, figure 10) using AFGROW [6], a ΔK_I in same size as ΔK_{eq} at the weld line is achieved.

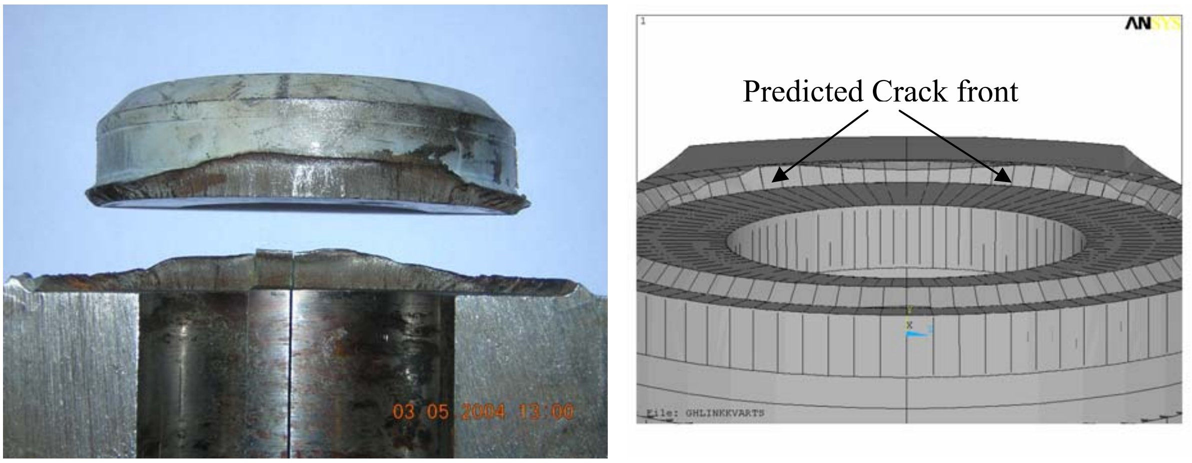


Figure 11 Fracture in W3 (ear).

Table 1 shows predicted life vs. average in laboratory test (5 samples).

Table1. Predicted and tested life of the W1-W3 in lab. hours.

Weld	Predicted (h)	Test (h)
W1	No fracture	No fracture
W2	~100 h	~210 h
W3	~200 h	~210 h
Base material in fork (AFGROW)	~200 h	~210 h

Discussion & Conclusion

The program is an good tool for fatigue analysis of root side of welded joints. The following preliminary conclusions can be made:

- It's easy to build the sub-model from an already existing FE-model.
- No restriction of the element shape functions (hexahedral or tetrahedral) in the global model is needed and the preparation of the global model is almost none.
- It's also easy to modify the weld geometry i.e. penetration deep, leg-length. The result of this type of modification of the weld geometry is often very valuable information in the design process.

Further Work

The work continues and the following investigations are planned:

- Further verification on complex joints
- Implementing the notch stress theory method on toe side

References

1. Byggnevi, M., In *Design and Analysis of Welded High Strength Steel Structures,* edited by J. Samuelsson, EMAS, Stockholm, 2002, 455-483.

2. Petterson, G., In *Design and Analysis of Welded High Strength Steel Structures,* edited by J. Samuelsson, EMAS, Stockholm, 2002, 391-412.
3. Pettersson, G., In *Design and Analysis of Welded High Strength Steel Structures,* edited by J. Samuelsson, EMAS, Stockholm, 2002, 413-436.
4. Martinsson, M., Samuelsson J., In *Design and Analysis of Welded High Strength Steel Structures,* edited by J. Samuelsson, EMAS, Stockholm, 2002, 303-334.
5. Martinsson J., *Comparisons between different contemporary FCG programs on welded components*, IIW doc. XIII-1994-03.
6. http://afgrow.wpafb.af.mil/about/history.php
7. Shih, C.F., DeLorenezi, M.D., *Int. Journal of Fracture,* vol. 12, 647-651, 1976.
8. Richard, H.A. (2001) In: CD-Rom Proceedings of ICF10, Honolulu, USA.
9. Erdogan, F., Sih, G.C., *J. Basic Engng. 85*, 519-525, 1963.
10. Maddox, S. *et al., An investigation of the influence of applied stress ratio on Fatigue Crack Propagation in Structural Steels*, Welding Institute Members, Report No. 72, 1978.
11. Samuelsson J., *Fatigue design of vehicle components methodology and applications*, The Royal Institute of Technology, Report No. 88-23, 1988.