Crack Growth Prediction Using FEM Data

Robert A Adey, John M. W. Baynham, Sharon Mellings, Lie Zhang

C M BEASY, Ashurst Lodge, Southampton, Hampshire, SO40 7AA, UK

Tel: +44 (0)23 8029 3223 Fax: +44 (0)23 8029 2853

Summary:

As part of the engineering design process engineers have to assess not only how well the design satisfies the performance requirements but also how durable the product will be over its life cycle. A major cause of failure is the growth of cracks which grow due to fatigue loadings to the point where the product fails. This paper describes a new approach to predicting crack growth which combines the best features of boundary element and finite element technology. The crack and the crack growth are simulated using the boundary element model and the finite elements are used to represent the remaining part of the structure. Examples are presented showing how this can be applied to complex structures.

Keywords:

Fracture Mechanics, Crack growth, Stress Intensity factors, Fatigue, Damage Tolerance, Integrity Assessment

1 Introduction

The analysis of cracks within structures is an important application if the damage tolerance and life of structures and components are to be predicted. As part of the engineering design process engineers have to assess not only how well the design satisfies the performance requirements but also how durable the product will be over its life cycle. Often cracks cannot be avoided in structures; however the fatigue life of the structure depends on the location and size of these cracks. In order to predict the fatigue life for any component, a crack growth study needs to be performed.

The boundary element method is an ideal solution for performing crack growth analysis due to the high accuracy of the stress results computed on the surface of the structure and its ability to represent the stress field singularities near the crack front. In addition, since only the boundary of the body needs to be discretised for boundary element analysis, the meshing time can be significantly reduced over other analysis methods.

In many cases the engineer responsible for the durability of the product only becomes involved during the later stages of the design process when analysis models have already been developed often involving substantial parts of the structure. In order to make a crack study the engineer would have start the process of building a new model of the area of interest and identify the loads acting on the component or the part of the structure of interest. While this can be done reasonably quickly using the BEM technology it is not always that straight forward to identify the loads.

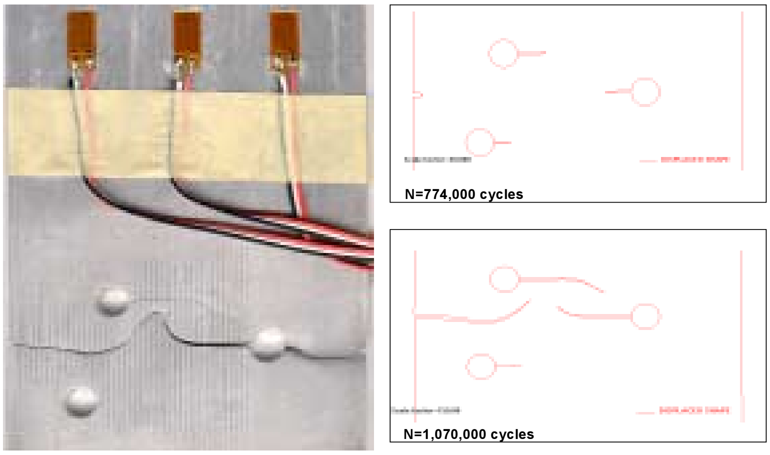
A new approach is therefore presented in this paper whereby the crack model can be automatically created from any existing FEM model and the crack growth study performed

2 Theoretical Foundations

A number of authors have studied the numerical simulation of crack growth using a variety of numerical techniques. Finite element methods have been developed[10,11,12] using mesh generation techniques and cohesive elements. They however still require a discretization of the three-dimensional volume mesh. Boundary element solutions[1-10] benefit from a surface only representation of the crack.

The model presented here uses the Dual Boundary Element Method (DBEM) to predict the stress field for cracked structures and hence to predict the stress intensity factors along the crack front. The analysis method implemented is based on the theoretical foundations developed for two-dimensional analysis by Portela, Aliabadi and Rooke[2] , and for three-dimensional analysis by Mi and Aliabadi[3]. This method has been further developed to include the effect of thermal stresses by Prasad, Aliabadi and Rooke[4] and dell'Erba, Aliabadi and Rooke[5]. In the Dual Boundary Element method, the crack in a structure is represented by special "Dual" elements that allow the stress and displacement fields to be computed on both crack faces without the need to subdivide the body along the crack boundary.

The Dual Boundary element method is a powerful solution tool for fracture mechanics, because it is a boundary only representation, the high accuracy and the methods ability to represent the high stress fields near the crack front.



Reproduced from paper titled ***Multiple Site Damage (MSD) crack growth: numerical evaluations and experimental tests*** by C. Cali, R. Citarella, M. Perrella, Department of Mechanical Engineering, University of Salerno

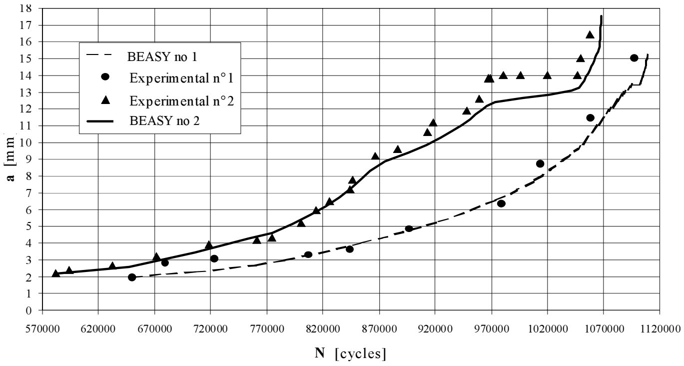
Figure 1 Simulation of multiple site damage in which the growth of four cracks is simulated. The experimental tests are shown on the left and the BEASY numerical predictions on the right.

2.1 Crack Modelling

The proposed approach can be clearly seen in multiple site damage calculation performed using BEASY by Cali, Citarella et al (14). The model and results can be seen in Figure 1, which shows close agreement between the experimental data and the numerical predictions. This example clearly shows the benefits of crack growth simulation as the crack path is predicted as well as the stress intensity and crack growth data. This example also shows the impact of load redistribution on the cracks as they grow and how the cracks interact with each other. This type of information cannot be obtained from analytical and textbook type crack growth solutions.

The crack growth model not only provides data on the crack path but also the life of the structure.

Figure 2 shows a comparison of the crack size versus the number of loading cycles with experimental data. This type of display can be used for design and prediction of life.



Reproduced from paper titled ***Multiple Site Damage (MSD) crack growth: numerical evaluations and experimental tests*** by C. Cali, R. Citarella, M. Perrella, Department of Mechanical Engineering, University of Salerno

Figure 2 Predicted crack size verses the number of loading cycles for one of the cracks shown in Figure 1. A Paris model was used for the crack growth rate.

2.2 Fatigue Crack Growth Prediction

In practical applications a simple cyclic loading is inadequate to represent the conditions the component or structure will experience during its working life. Therefore the software has been linked to a comprehensive multi-axial loading module that enables real life loading data to be applied to the model. Another important element is the crack growth model that is used to predict the crack growth rate (da/dn). The analysis code allows a range of fatigue growth laws to be represented (for example

Paris, NASGRO) and the code is linked to the NASGRO database of fatigue crack growth data for fatigue analysis. This also allows the use of retardation models for crack growth[1].

2.3 Automatic Remeshing

In the analysis of a crack, at each iteration a series of new elements are added to the crack front. These elements are formed from the positions of the old crack front and the predicted positions of the new crack front. Where the crack intersects the surface of the component or structures the surface mesh has to be modified to represent the new geometry as the crack grows.

In some models the crack re-meshing can be very complex and time constraints and model complexity can prohibit crack growth beyond a few iterations when performing the meshing “manually”. The aim of the automatic remeshing is to remove this manual work from the user to allow more detailed study of crack growth models to be performed automatically.

2.4 Crack Initiation

One important extension of this technique is that it is possible to use the same algorithm to add a crack into a model that has simply been defined to perform a stress analysis. This allows the generation of the model without having the task of modelling the crack. The crack can then be added anywhere on the model and a new data file automatically generated containing the crack.

2.5 Integration with Finite Element Data

Depending upon the application the user has a number of choices on how the FEM data can be used to simulate the behaviour of the crack. For example

- The complete model including all the loading and restraints can be automatically transferred
- A representative part of the model can be selected including the loads and restraints
- In the case where there are complex non linear stresses (e.g. Residual Stress fields) the stress field can be transformed to a part or the whole of the model

In some applications the original FEM model may not contain sufficient detail in the area where the crack is located (i.e. some of the geometrical details missing, poorly refined mesh etc). In this case the user can build a new geometric model of a sub section and transfer the loads and restraints.

The following example shows how the procedures work.

3 Example Application

In this application an aircraft fixing lug is to be assessed for crack growth near one of the supporting pillars. A finite element model already exists of the overall fitting but without any crack information as the model was simply developed to obtain predictions of the general stress levels. The next few steps describe how this model can be used to obtain information about the stress intensity factors and how the crack will grow.

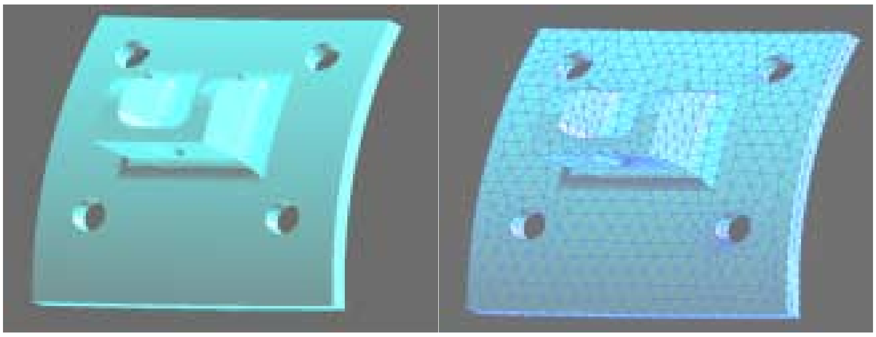


Figure 3 On the left is the geometric model of the fitting. On the right is the FEM model mesh

In this case the user has decided that the crack is sufficiently small that only a small subsection of the fitting is required to be part of the crack model. Therefore a subsection of the model is chosen in the modelling system (in this case MSC PATRAN) by selecting the finite elements to be included. In this way critical areas can be quickly assessed by zooming into areas of the FEM model and simulating the crack behaviour.

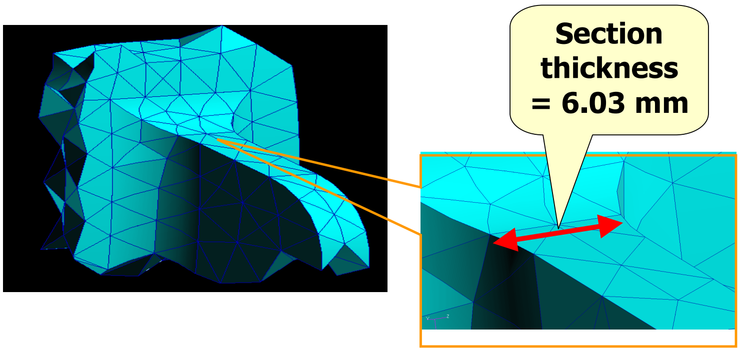
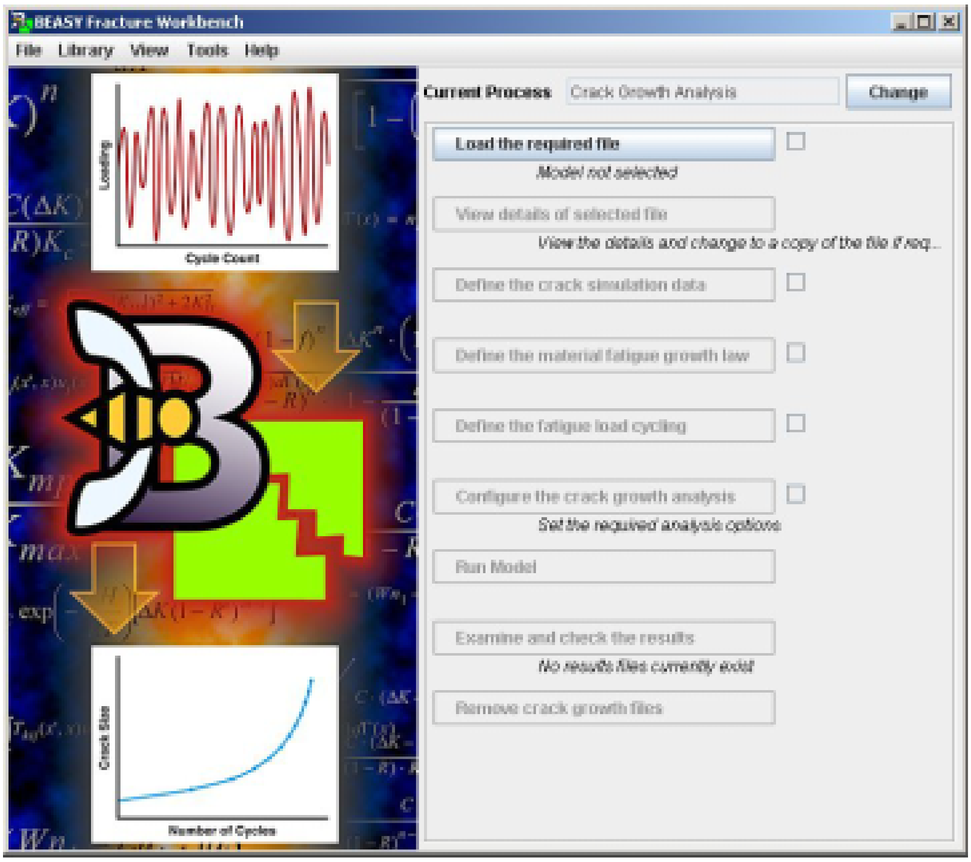


Figure 4 The sub model selected is shown. This will be automatically converted to use in the crack growth study

3.1 Crack Wizard

Predicting crack behaviour using fracture mechanics requires the user to address a number of additional factors such as the crack growth model, crack growth material properties, the loading history or spectra in addition to the task of meshing the crack in the model. To help the user a Crack Wizard has been developed which automates the process and guides the user through the necessary steps. The first screen of the crack wizard can be seen in Figure 5. The first step is to select the objective of the simulation, for example “calculate stress intensity factors” or “perform and crack growth analysis. Once selected the wizard will guide the user through the steps and automate many of the processes.

The next step is to identify the location of the crack initiation point and the size and shape of the initial crack. In this case it has been decided to initiate the crack on the corner in the area with the highest stress.



Current Process: Crack Growth Analysis

Model not selected

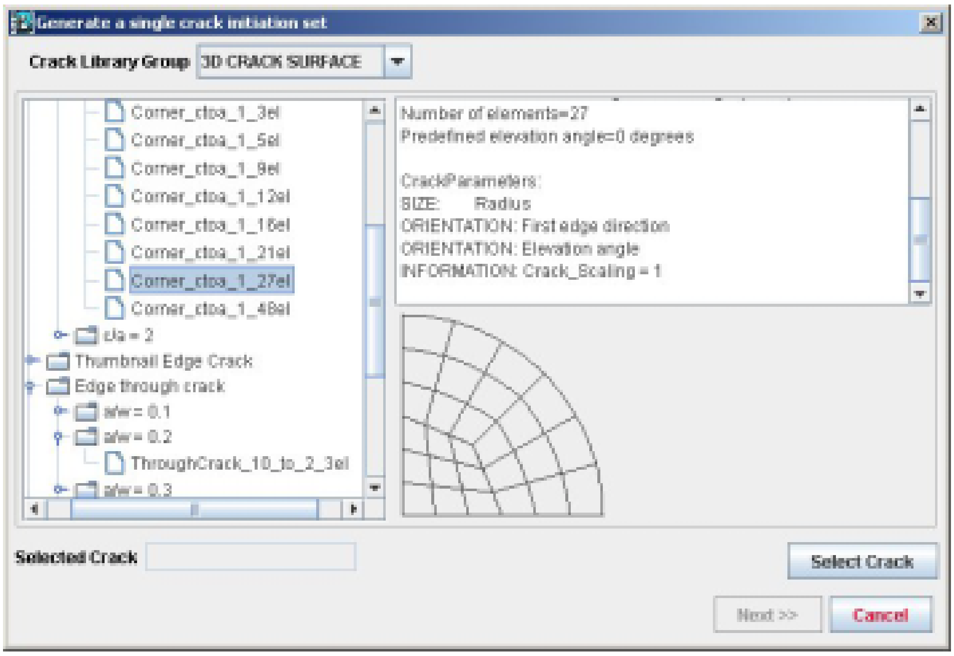
View the details and change to a copy of the file if req...

Set the required analysis options

No results files currently exist

Figure 5 The BEASY Crack Wizard automates the process and guides the user

The crack to be considered in the model can be selected from the crack library which contains a number of typical crack shapes and mesh types which the user can choose. Given the node in the model where the crack is to be located and the crack size and orientation the mesh is automatically modified to create a model capable of computing the Stress Intensity Factor data.



Number of elements=27
Predefined elevation angle=0 degrees

CrackParameters:
SIZE: Radius
ORIENTATION: First edge direction
ORIENTATION: Elevation angle
INFORMATION: Crack_Scaling = 1

Figure 6 Sample screen view of the crack wizard where the user selects the size and shape of crack from the crack library

The final step is therefore to run the analysis to predict the stress intensity factors, the crack growth rate and the crack growth path.

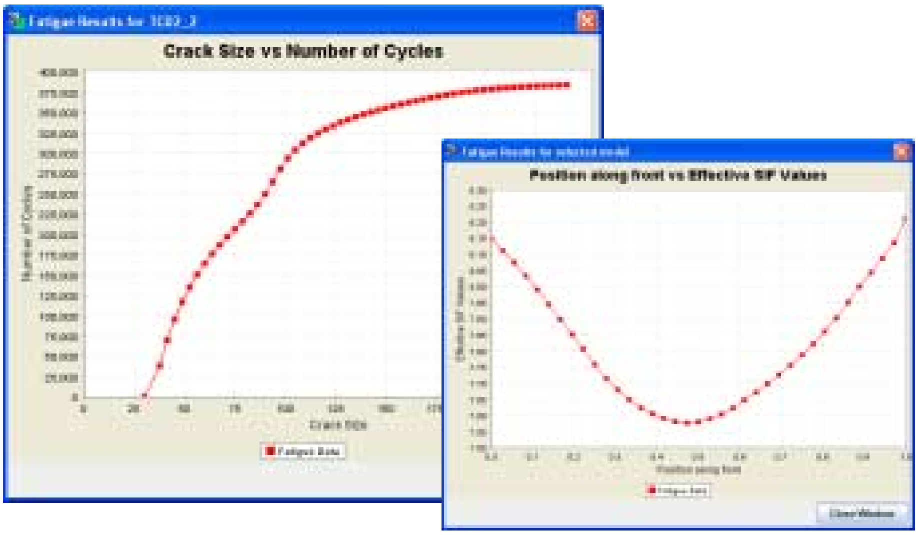


Figure 7 Predicted Stress Intensity Factors and Crack Growth data

3.2 Zooming in to compute fracture data

In this second example data is required on a potential crack located at the intersection of a tubular member and a plate. The original model was constructed using shell elements which were adequate to represent the overall deformation and load distribution but were inadequate to predict the local stress behaviour at the joint as the local fillet detail was not included in the model.

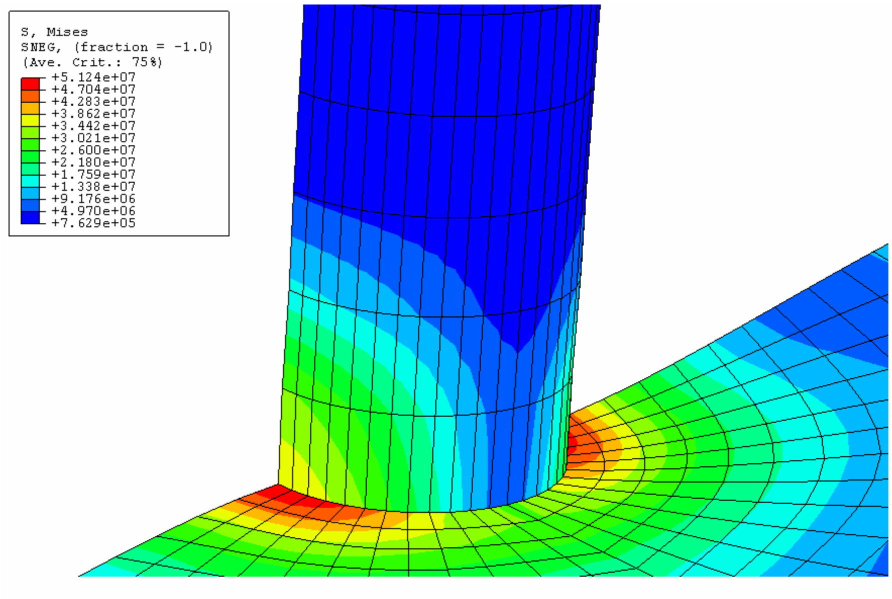


Figure 8 Shell model of joint detail without the filet geometry

The first step is therefore to create a more detailed model of the joint which correctly represents the joint geometry. The result of this is shown in Figure 9.

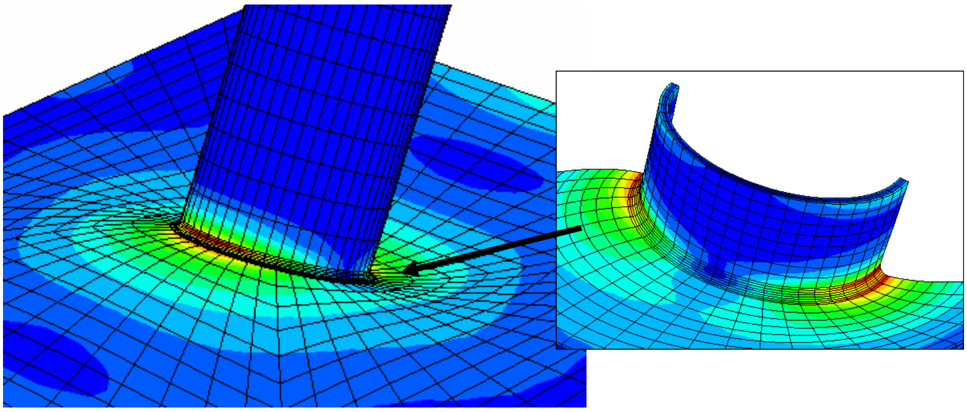


Figure 9 Close up view of the joint now modelled with 3D solid elements

The model is now sufficiently accurate to capture the stress field and load distribution near the crack and now is ready to use the Model Generation Wizard to create the sub model which will be used to simulate the crack. The user simply defines the region near the crack by selecting all the elements within a specified distance from the crack or by explicitly selecting a group of elements. The wizard then automatically creates the BEASY model including the loads and boundary conditions.

The Crack Wizard can now be used to insert the crack and remesh the model. The submodel can be seen in Figure 10 and the initiated crack can be seen in Figure 11.

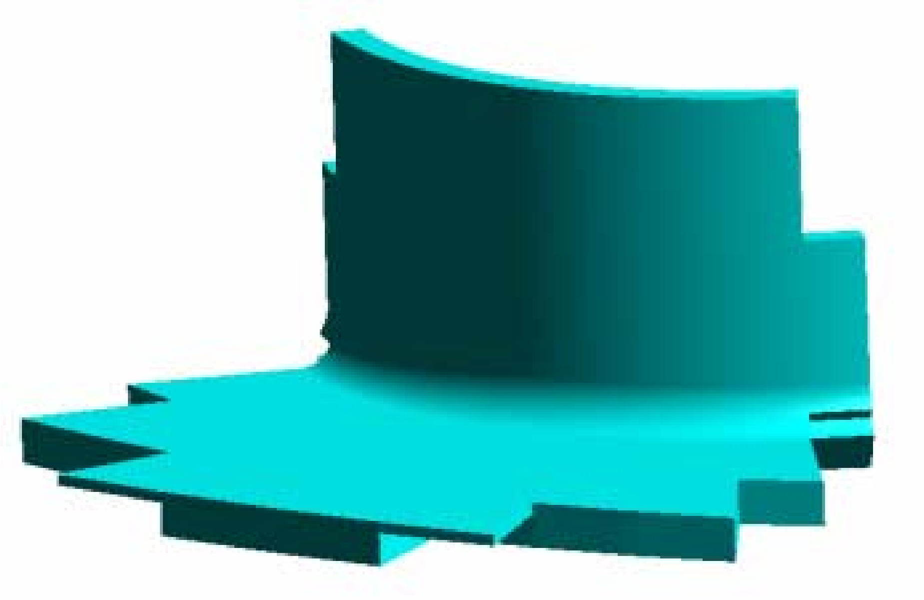


Figure 10 BEASY Sub model at region of maximum stress

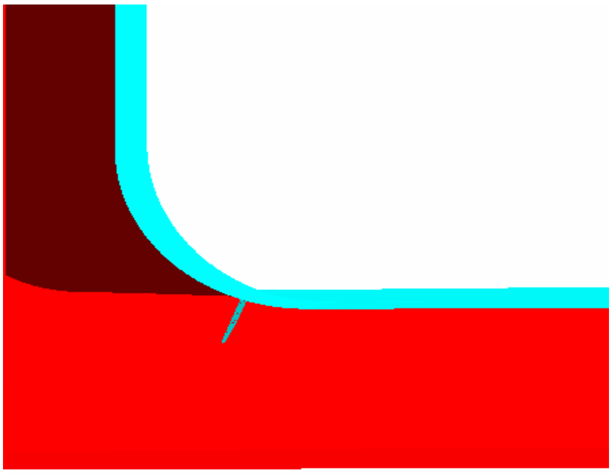


Figure 11 BEASY Sub model with thumbnail crack located in the filet at the point of maximum stress

In this study we also wish to predict how the crack grows so the Crack Wizard is used to select the crack growth model, material parameters and the loading spectra.

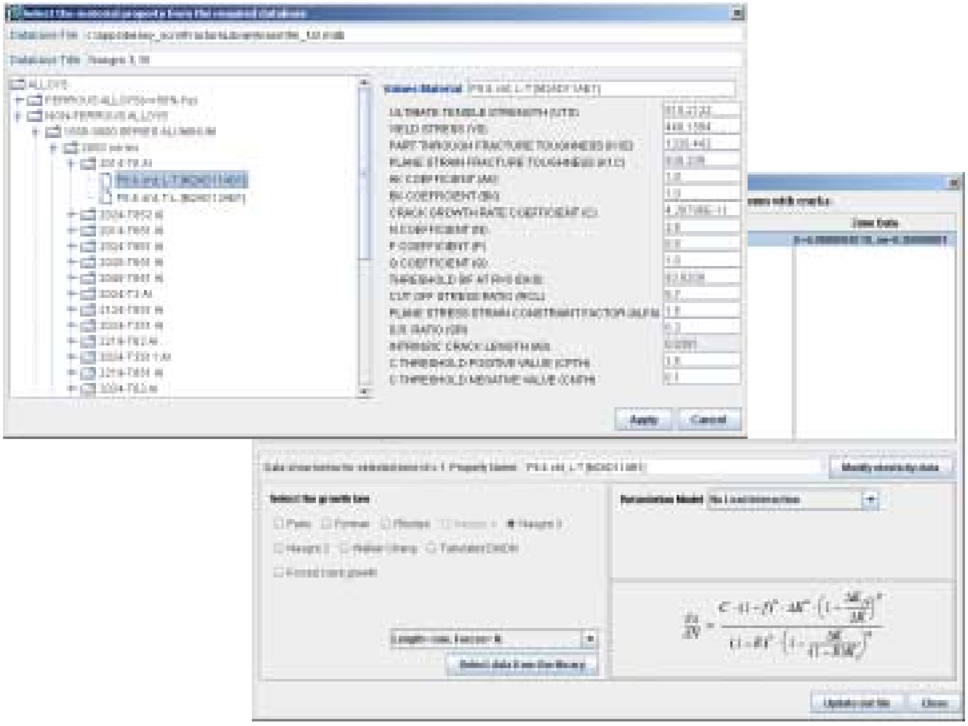


Figure 12 Selecting the crack growth model and material parameters using the Crack Wizard

The model is then automatically run by the wizard and the stress intensity factors computed. When predicting the crack growth the stress intensity factors along the crack front are used to identify the crack growth direction using either the principle stress direction or the minimum strain energy. Once a new crack front has been predicted the software automatically remeshes the model to represent the new crack shape and an new analysis performed to predict the Stress Intensity Factors for the new crack shape. To follow the crack growth path the whole process is repeated until failure is identified or sufficient life has been modeled.

The results of the simulation including the stress intensity factors and the crack size vs number of cycles can be displayed interactively or exported to an EXCEL spreadsheet.

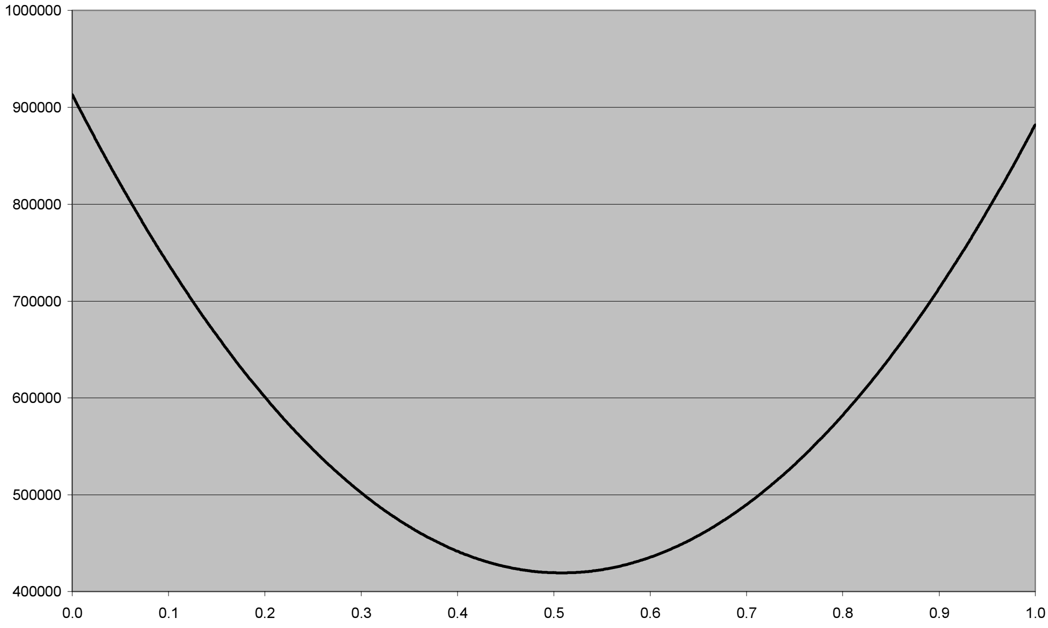


Figure 13 Stress Intensity Factor data along the crack front

4 Conclusions

A general approach to modelling the impact of cracks on the performance and life of structures has been developed which combines BEM modelling of the cracks with FEM modelling of the structure. The approach enables simple and accurate prediction of stress intensity factors and the automatic simulation of single and multiple crack growth.

The process provides the data required for Integrity and damage Tolerance assessment.

The method is ideally suited for modelling crack growth in components and structures and example applications have been presented demonstrating the capability.

5 References

[1] BEASY User Guide, Computational Mechanics BEASY Ltd, Ashurst, Southampton, UK, 2003.

[2] Portela, A.; Aliabadi M.H; Rooke, D.P., "The Dual Boundary Element Method: Efficient Implementation for Cracked Problems", International Journal for Numerical Methods in Engineering, Vol 32, pp 1269-1287, (1992).

[3] Mi Y; Aliabadi M.H., "Three-dimensional crack growth simulation using BEM", Computers & Structures, Vol. 52, No. 5, pp 871-878, (1994).

[4] Prasad, N.N.V.; Aliabadi M.H; Rooke, D.P., "The Dual Boundary Element Method for Thermoelastic Crack Problems", International Journal of Fracture, Vol 66, pp 255-272, (1994).

[5] dell'Erba D.N.; Aliabadi M.H; Rooke, D.P., "BEM analysis of fracture problems in three-dimensional thermoelasticity using J-Integral", International Journal of Solids and Structures, Vol. 38, pp. pp.4609-4630, (2001).

[6] Neves A; Niku S.M., Baynham J.M.W., Adey, R.A., "Automatic 3D crack growth using BEASY", Proceedings of 19th Boundary Element Method Conference, Computational Mechanics Publications, Southampton, pp 819-827, 1997.

[7] Aliabadi M.H., Rooke D.R., Numerical Fracture Mechanics, Computational Mechanics Publications, SOUTHAMPTON, U.K. 1991.

[8] Erdogan, F., Sih, G.C.; "On the Crack Extension in Plates under Plane Loading and Transverse Shear", J.Basic Eng., 85, pp 519-527, (1963).

[9] Sih, G.C.; "Mechanics of Fracture Initiation and Propagation", Kluwer Academic, Dordrecht, (1991).

[10] Chen,C., Wawrzynek P.A., Ingraffea, A.R.; "Residual Strength Prediction of Fuselage Structures with Multiple Site Damage",2nd Joint NASA/FAA/DOD Conference on Ageing Aircraft, (1999).

[11] Wells, G.N, Sluys, L.J.; "A New Method for Modelling Cohesive Cracks using Finite Elements", International Journal of Numerical methods in Engineering, Vol 20, pp 2667-2682, (2001).

[12] Dhondt et al, "Computation Fracture Analysis of Different Specimens Regarding 3D and Mode Coupling Effects", Engineering Fracture Mechanics, Vol 68, pp.383-401, (2001).

[13] NASGRO. NASA Johnson Space Center, Houston, Texas. USA

[14] Calì C., Citarella R., Perrella M., *Multiple Site Damage (MSD) Crack Growth: Numerical Evaluations and Experimental Tests,* Atti del Convegno Internazionale "6ICB/MF&F, 25-28 giugno 2001, Lisbona, Portogallo.

[15] "Durability Prediction Using Automatic Crack Growth Simulation", S Mellings, J Baynham, R A Adey and T Curtin, International Committee on Aeronautical Fatigue, Toulouse, France, June 2001.