

3D Fracture Mechanics In ANSYS

Ramesh Chandwani, Miles Wiehahn, Chris Timbrell

Zentech International Limited
<http://www.zentech.co.uk>

ABSTRACT

This paper will address methods of performing truly three-dimensional fracture mechanics analyses in ANSYS.

Generally available fracture mechanics techniques and their implementation and use with ANSYS for 3D analysis will be briefly discussed. Techniques include the crack opening displacement (COD) method for LEFM, crack tip opening displacement (CTOD) method for EPFM, and the J-Integral method.

A software implementation using the COD method in conjunction with ANSYS will be presented. This implementation addresses generation of cracked 3D meshes and crack growth prediction. Examples will demonstrate large-scale crack growth under generalised mixed-mode loading and the development of complex 3D crack surfaces.

INTRODUCTION

Crack propagation behaviour is a major issue in a variety of industries. Aerospace structures, gas turbine engines, pressure vessels and pipelines are obvious examples where failure could lead to catastrophic consequences and loss of life. For example, grounding of a fleet of aircraft because of cracks found in a turbine blade of the engine, or shutting down of a pipeline have huge implications.

The mere presence of a crack does not condemn a component or structure to be unsafe and hence unreliable. Whether under cyclic or sustained loading, it is necessary to know how long an initial crack of a certain size would take to grow to a critical size at which the component or structure would become unsafe and fail. Also by knowing how a crack evolves and its rate of propagation, one should be able to estimate the residual service life of a component under normal service loading conditions. The financial costs involved if an in-service component is found to contain a defect are a major factor in the search for numerical methods to predict 3D crack propagation.

Other situations where crack propagation is required include:

- Studying the effects of surface treatment such as shot peening, laser shock peening etc., to enhance the service life of a component.
- Studying the effectiveness of crack repair systems, remedial work, modifications or design changes.
- Establishing inspection and maintenance regimes.

Until recently mode I and mixed mode crack growth behaviour was generally evaluated using experimental tests. However, using the design philosophy based on reliability and “safe life” criteria, it becomes necessary to test a large number of materials and structural components in a very short period. This makes the use of experimental methods rather impractical and implies the benefit of using numerical computational methods to evaluate 3D mixed mode crack propagation from simple mode I tests.

Zentech has developed a 3D crack analysis tool called ZENCRACK (Ref. 1). ZENCRACK reads in an uncracked finite element model and produces a cracked finite element model. Stress intensities or the energy release rate are calculated automatically from the results of the cracked finite element analysis. Furthermore crack growth can be undertaken by extending the crack position. An updated finite element model is then created and run to simulate crack growth.

ZENCRACK is a mature product that was first released in 1990. However an ANSYS (Ref. 2) interface has only been created recently.

This paper discusses, and demonstrates with examples:

- methods of obtaining the stress intensity factors or energy release rate from F.E. codes and in particular, ANSYS
- topology and meshing in relation to 3D cracks
- crack growth prediction

FRACTURE MECHANICS PARAMETERS

The primary fracture mechanics parameters that may be of interest for crack propagation are:

- Stress intensity factors, K_I , K_{II} , K_{III}
- Energy release rate, G

The stress intensity factor approach was developed by Irwin in the 1950s following on from the elastic strain energy approach to brittle fracture developed by Griffith from the 1920s. Irwin's work led to the foundations for the concept of linear elastic fracture mechanics (LEFM) which is still fundamental in most crack propagation analyses.

For linear elastic analysis the concepts of energy release rate and stress intensity factors are closely linked. The stress intensity factors describe the magnitude of the elastic stress field at a crack front. The general form of the stress intensity factor is:

$$K = f(\text{load}, \text{crack length}, \text{geometry}) \quad \text{Equation 1}$$

For mode I behaviour it can be shown that:

$$K = \left(\frac{EG}{1 - (\alpha\nu)^2} \right)^{1/2} \quad \text{Equation 2}$$

where E is Young's modulus, ν is the Poisson ratio and α is a value ranging from 0 for plane stress to 1 for plane strain. In a more general form it is possible to write:

$$G = \frac{B}{E} (K_I^2 + K_{II}^2) + \left(\frac{1 + \nu}{E} \right) K_{III}^2 \quad \text{Equation 3}$$

where $B=1$ for plane stress and $1-\nu^2$ for plane strain.

Another important relationship for stress intensity factors in linear elastic analysis is based on the Westergaard equations that link the stress intensity factors to the displacement field around the crack tip (COD) giving:

$$K_I = \frac{EV_i}{4B} \sqrt{\frac{2\pi}{r}}, \quad K_{II} = \frac{EV_{ii}}{4B} \sqrt{\frac{2\pi}{r}}, \quad K_{III} = \frac{E}{2(1+\nu)} V_{iii} \sqrt{\frac{\pi}{2r}} \quad \text{Equation 4}$$

where B is defined as above and V_i , V_{ii} and V_{iii} are the relative opening displacements at a radius r from the crack front for an orthogonal system aligned with the mode I, II and III directions. This approach is widely used by practitioners of both the finite element and boundary element methods and has the benefit that it requires relatively little additional effort on top of the basic stress solution. The drawback to the method is that it requires a state of stress assumption.

Calculation of the magnitudes of energy release rate and / or stress intensity factors do not directly provide directional information regarding crack growth. A number of criteria have been developed to specify the direction. They include maximum energy release rate, maximum tangential stress and the normal to the maximum principal stress. In the context of numerical calculations of energy release rate and stress intensity factor, the two most useful criteria are maximum energy release rate and a direction based on stress intensity factors e.g.:

$$\theta = \tan^{-1} \left(\frac{K_{II}}{K_I} \right) \quad \text{Equation 5}$$

The J-integral concept was first described by Rice in the late 1960s. It is an energy based concept in which the J-integral, J, can be considered a non-linear elastic equivalent of the energy release rate, G. By definition G and J are the same for elastic behaviour. The J-integral can be calculated as a post processing exercise after completion of a finite element analysis. However, this capability is currently not available for 3D fracture mechanics in ANSYS.

For similar reasons that significant additional post-processing coding would be required, methods such as the virtual crack closure method do not present an attractive proposition.

For elastic plastic fracture mechanics the crack tip opening displacement (CTOD) is a measurement of the crack opening displacement at the crack tip. The relationship between J and CTOD (δ) is given by the equation:

$$J = m \sigma_{ys} \delta \quad \text{Equation 6}$$

where m varies between 1.15 and 2.95 (Ref. 3). For such elastic plastic cases the difficulty of using this method is choosing a valid value of m . Hence the method cannot readily be applied as a means of evaluating J from displacements for EPFM.

In the interface between ZENCRACK and ANSYS, the COD method is used for evaluating the stress intensity factors (see Equation 4) for linear elastic materials. Extensive testing has shown that COD agrees well with the J-Integral when using ZENCRACK with other f.e. codes (see verification examples later in the paper). COD can also be used for problems with residual stresses where current J-Integral implementations have shortcomings (See Ref. 7).

Note that ZENCRACK is also capable of generating meshes suitable for EPFM although in these cases there is no calculation of fracture mechanics parameters.

TOPOLOGICAL ISSUES

A critical issue that must be addressed in 3D F.E. fracture mechanics analysis is that of mesh generation. In the simplest of geometric cases where symmetry can be used, it may be possible to utilise standard mesh generation tools to produce a crack of the required size. In the general case, however, the use of standard tools poses several time consuming problems including:

- Component geometries are often complex and time consuming to model in their uncracked forms.
- Defects often occur at geometrically difficult locations e.g. corners, welds, chamfers.
- Initial cracks of the correct size and shape must be inserted into the component at the correct location.
- Cracks may develop in a non-planar fashion depending upon the loading.
- These problems are compounded if more than one crack size must be analysed or if there are multiple cracks in a component.

The approach that has been successfully adopted at Zentech is the use of 'crack-blocks' which model the details of the crack region. Figure 1 and Figure 2 demonstrate the use of the crack-block methodology in generating a cracked mesh from a user-supplied intact component. The method works by replacing one or more elements in the uncracked mesh by crack-blocks that contain sections of crack front.

The interface to ANSYS operates via the ANSYS batch file for the uncracked component. This file is read and processed by ZENCRACK. A new batch file is created for the cracked mesh. Each batch processing keyword in the ANSYS input library has been given an associated status value. When a keyword in the uncracked mesh is identified, the action taken by ZENCRACK depends upon this status value (e.g. the SFE option is checked to see if load updates are required on the crack-blocks).

"Standard" processing is performed if the batch file contains explicit element and node specifications. Alternatively, the batch file may be constructed using the ANSYS solid-modelling and mesh generation capabilities. ZENCRACK then automatically performs an initial ANSYS analysis to generate an ANSYS database file for the uncracked model. This is then queried to generate explicit node, element and boundary condition data. ZENCRACK then creates a modified batch file and interprets this data in the "standard" way.

When the cracked mesh is constructed, output requests are incorporated to allow displacements at key nodes to be extracted by ZENCRACK for use in calculating stress intensity factors.

ZENCRACK has two types of crack-blocks:

- Standard crack-blocks.
 - These crack-blocks have a "clean" face on three faces.
 - The crack-blocks are designed to replace elements in the mesh by updating element connectivities and node numbers (see Figure 1).

- The crack-blocks consist of “through” and “quarter circular” crack-blocks.
- Large crack-blocks.
 - These crack-blocks do not have “clean” faces.
 - These crack-blocks use tied contact to tie the crack-blocks to the model and can therefore straddle several standard elements (see Figure 2).
 - The crack-blocks consist of “through” and “quarter circular” crack-blocks.

The crack-blocks have a varying number of “rings” of elements around the crack front. The innermost ring contains “collapsed” elements to represent the singularity in the stress and strain field at the crack front. ZENCRACK offers full control of the nodes along the crack front and the radial nodes closest to the crack front in order to generate a singularity best suited to LEFM or EPFM (Ref. 5).

Although the crack-blocks are referenced as “quarter circular” or “through” crack-blocks, the user has control of the initial crack front shape which may be defined by fitting a spline through a series of points for the greatest flexibility in definition.

The use of crack-blocks allows loading (e.g. pressure load) and boundary conditions to be updated as the crack is incorporated (and advanced through the mesh).

CRACK GROWTH PREDICTION

By extending the mesh generation scheme described above and adding a crack growth algorithm, it is possible to carry out automatic crack growth prediction, as summarised in Figure 3.

The remeshing of a fixed region in space was discussed in by Cook et al in Ref. 8. The original method only allowed growth to occur within the volume occupied by the original element of the uncracked mesh. The current implementation in ZENCRACK allows greater flexibility by (see Figure 4):

- Shifting of the boundaries of the crack-blocks.
- Relaxing elements surrounding the crack-blocks.
- Transferring crack-blocks from one location to another.
- Allowing use of “large” crack-blocks to increase the volume in which growth may occur.

In order to complete the crack growth prediction, it is necessary to integrate using the results from a f.e. analysis along with the crack growth data. This procedure cannot be fully described here (see ZENCRACK documentation for details) but the salient features of the implementation are:

- Standard forms of crack growth data are allowed such as Paris and Walker equations in addition to tabular data (as a function of stress ratio and temperature) and a completely general user subroutine option for proprietary data or complex equation forms.

- Threshold and fracture material properties can be defined.
- The integration is carried out in such a way that a consistent dn is evaluated for all crack front nodes although in general the da values vary from node-to-node (thus allowing the possibility of complex crack shape development).
- The step size between f.e. analyses is adapted based on the accuracy of previous integration steps.
- Constant amplitude or variable amplitude fatigue loading can be analysed.
- A sustained load crack growth option is also available (i.e. da/dt vs K growth data rather than da/dn vs ΔK).
- Static load may be incorporated with the cyclic load.

STRESS INTENSITY FACTOR VERIFICATION EXAMPLES

Two different verification examples are shown to demonstrate the basic calculation of stress intensity factors:

- A SEN specimen
- A fully embedded crack

The SEN model is shown in Figure 5. A comparison of the ANSYS COD values with J-Integral values and a theoretical solution (Ref. 4) is shown in Figure 6 (a is the crack length, W is the specimen width, $K_0 = \sigma\sqrt{\pi a}$ where σ is the stress). The COD line agrees well with the theoretical and J-Integral results.

The uncracked mesh for the fully embedded crack is shown in Figure 7. Figure 8 shows the cracked mesh for a circular crack ($a=0.3\text{mm}, c=0.3\text{mm}$) and Figure 9 shows the cracked meshes for an elliptical crack ($a=0.3\text{mm}, c=0.6\text{mm}$). The results can be seen in Figure 10. Again the COD results agree well with the J-Integral and theoretical solutions (Ref. 6).

EXAMPLES OF CRACK GROWTH

Some of the crack growth capabilities of ZENCRACK are shown in three examples:

- A 43° oblique crack.
- A lug-pin interaction.
- A turbine compressor disk.

It must be noted that each analysis, once started, is completed fully automatically.

The first example is of a 43 degree slanting crack in a plate under cyclic axial tension (see Figure 11). This is based on a test case for which experimental data is published (Ref. 9). The crack face triangular facets that are defined by ZENCRACK for the initial crack, plus those calculated during the analysis, are shown in Figure 12. The growth is compared against experimental results in Figure 13 and Figure 14. It is clear that there is excellent agreement in both the calculated growth direction and

cycle history. Note that although the problem is initially mixed mode, the crack very quickly re-aligns itself into mode I behaviour.

The second example is a pin-loaded lug (see Figure 15). This problem requires contact conditions between the pin and lug in the f.e. analysis. The maximum principal stress in the uncracked mesh is used to determine the initial defect location. The initial defect is postulated and is grown all the way to the edge of the lug (note that no K_{Ic} is defined in this analysis). The calculated crack growth profiles are shown in the bottom right corner of Figure 15. The crack that develops is determined by the loading and materials data. The user does not have to make any assumptions about the crack direction, or forcing the crack to grow between the particular element boundaries.

The third example is of a crack growing from a bolt hole in a compressor disk (see Figure 16). This analysis represents a spin test in which there is bi-axial loading at the bolt holes. The model was obtained from Ref. 10. Further discussion of the problem can be obtained from Ref. 11.

CONCLUSIONS

ZENCRACK has been interfaced to ANSYS allowing state of the art 3D fracture mechanics analysis to be undertaken. COD is used to obtain the stress intensity factors. A general crack growth scheme allows crack advancement and non-planar crack growth.

REFERENCES

- Ref. 1 ZENCRACK
 Zentech International Limited, U.K.
 <http://www.zentech.co.uk/zencrack.htm>
- Ref. 2 ANSYS
 Ansys Inc, U.S.A.
 <http://www.ansys.com>
- Ref. 3 "Fracture mechanics", H.L.Ewalds and R.J.H Wanhill, Edward Arnold (Publishers) Limited, 3rd imprint 1986 (ISBN 0-7131-3515-8), pg 156.
- Ref. 4 AFGROW,
 Air Vehicles Directorate, Air Force Research Laboratory, U.S.A.
 <http://afgrow.wpafb.af.mil/>
- Ref. 5 "Use of modified standard 20-node isoparametric brick elements for representing stress/strain fields at a crack tip for elastic and perfectly plastic material", Koers, R.W.J., Int. J. Frac. 40 (1979) 79-110.
- Ref. 6 "Fracture mechanics", H.L.Ewalds and R.J.H Wanhill, Edward Arnold (Publishers) Limited, 3rd imprint 1986 (ISBN 0-7131-3515-8), pg 49.

- Ref. 7 "Residual Stress in a 3D Finite Element Fracture Mechanics Analysis", C. Timbrell, R. Chandwani, FENET Technology Workshops - Durability and Life Extension, Palma, Majorca, Mar 25-26 2004.
Available at http://www.zentech.co.uk/zencrack_papers.htm
- Ref. 8 "Automatic and adaptive finite element mesh generation for full 3D fatigue crack growth", G.Cook, C.Timbrell, P.W.Claydon, STRUCENG & FEMCAD Conference, Grenoble, France, 1990.
Available at http://www.zentech.co.uk/zencrack_papers.htm
- Ref. 9 "Fatigue crack propagation under general in-plane loading - I: Experiments", M.A. Pustejovsky, Engineering Fracture Mechanics 11 (1979) 9-15.
- Ref. 10 "Prediction of crack growth from bolt holes in a disc", W. Z. Zhuang, International Journal of Fatigue, 22 (2000) 241-250.
- Ref. 11 "The Application of 3D Finite Element Analysis to Engine Life Prediction", G. Cook, C. Timbrell, B. Browning, AeroMat 2001 - 12th Advanced Aerospace Materials & Processes Conference & Exhibition, Long Beach, CA, U.S.A., June 11-14 2001.

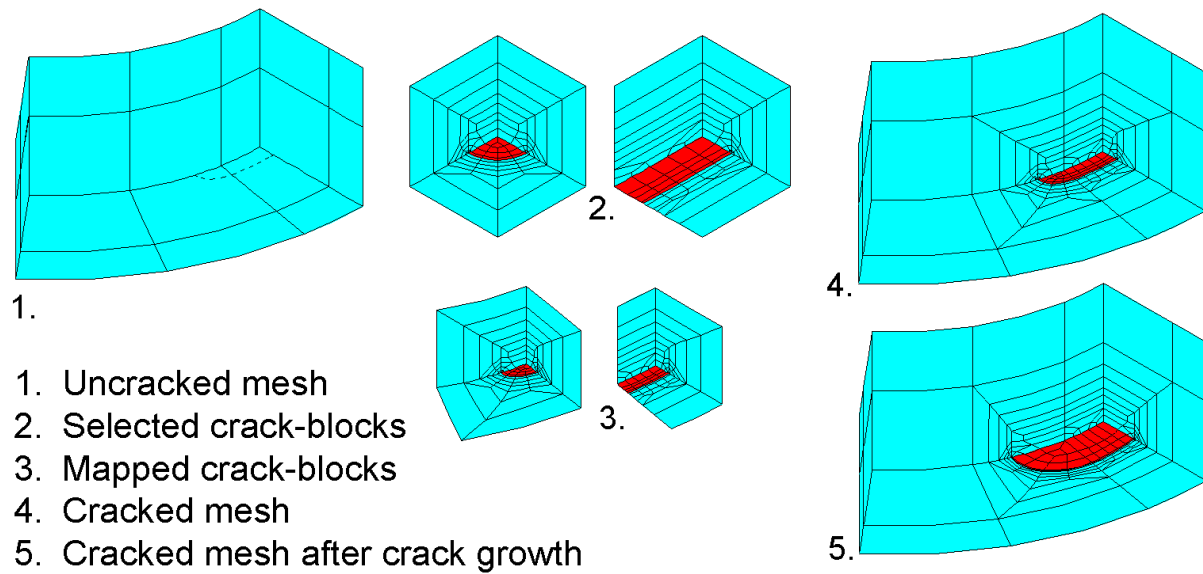


Figure 1 - Summary of crack insertion and mesh update process using standard crack-blocks

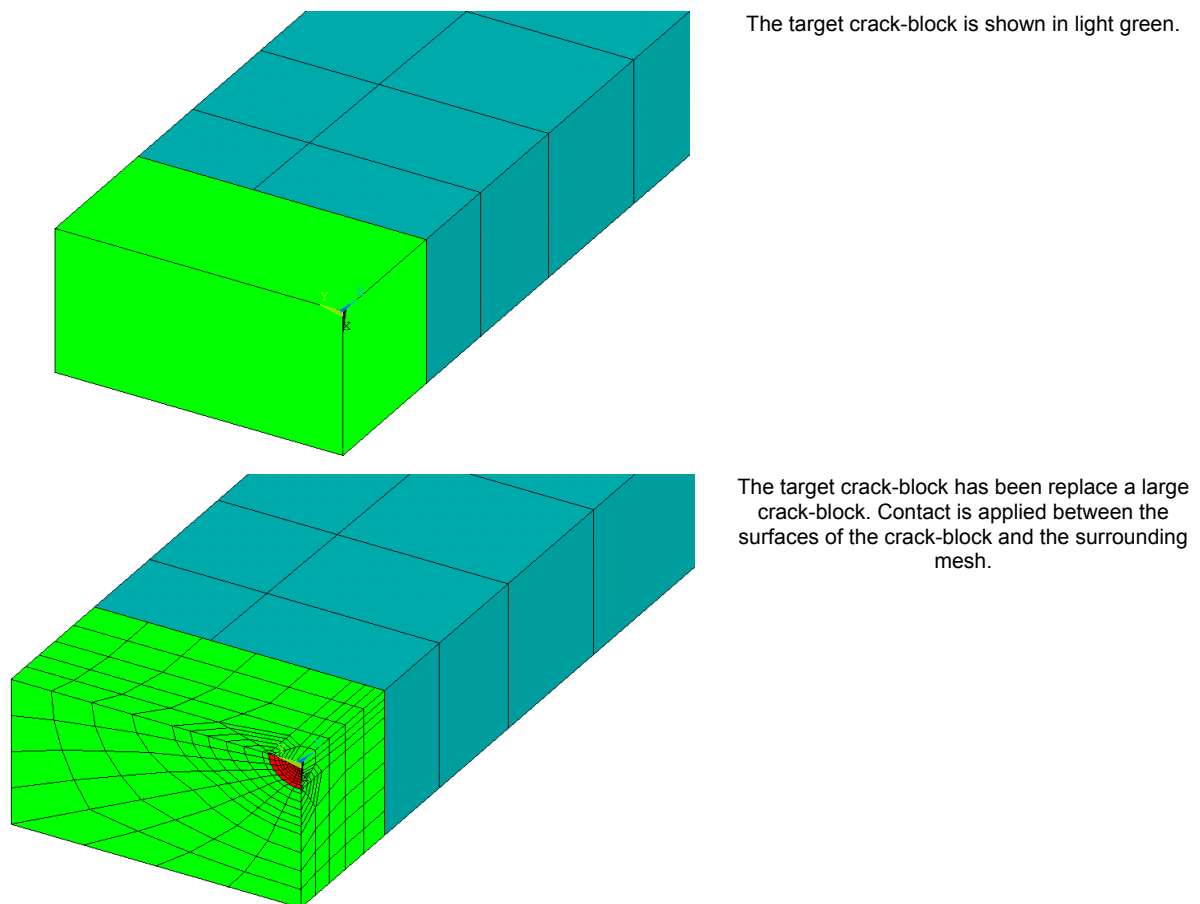


Figure 2 - Example of a large crack-block

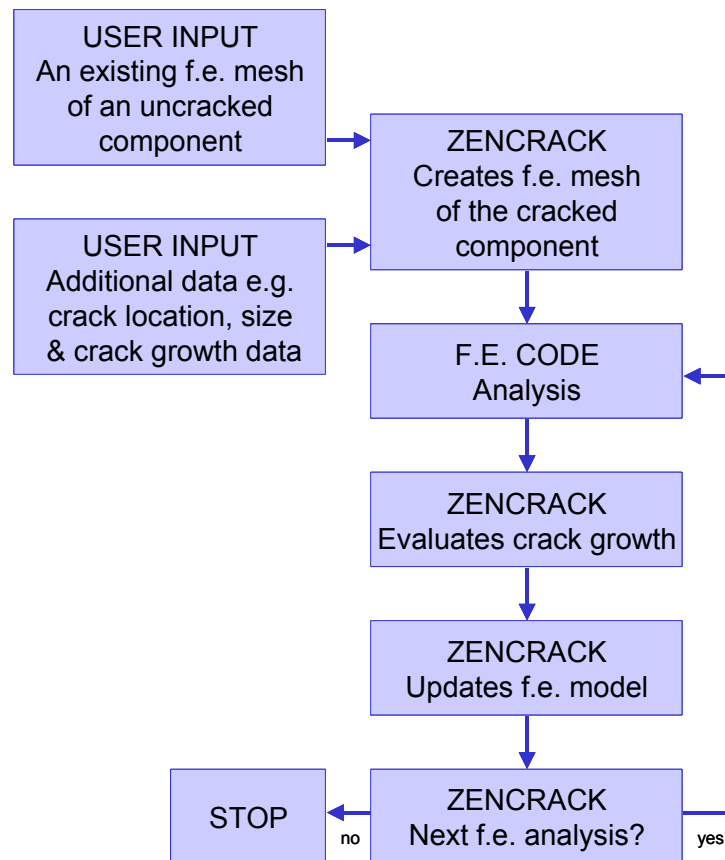
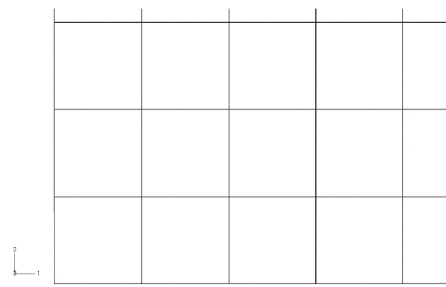
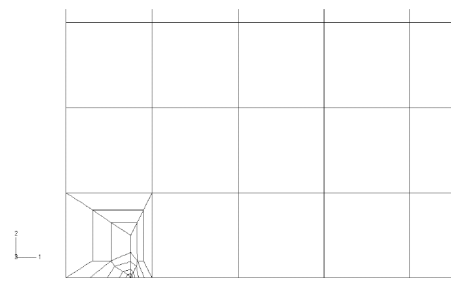


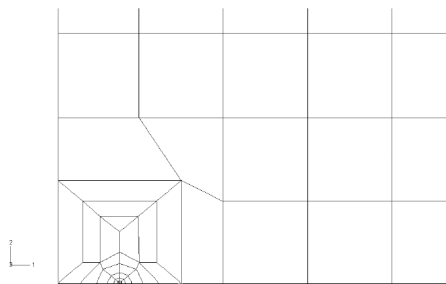
Figure 3 - Simplified flow chart for crack growth prediction analysis



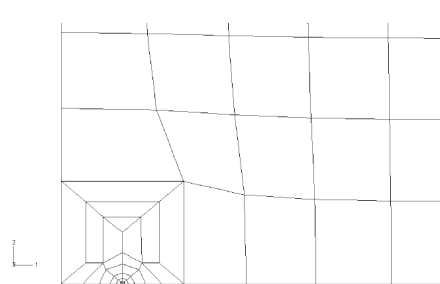
Un-cracked mesh



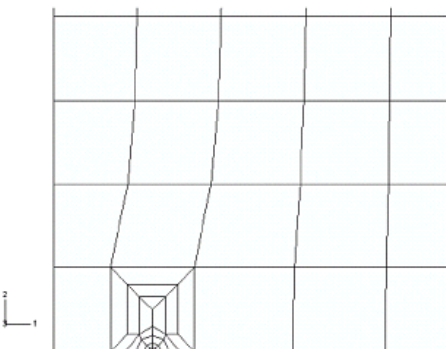
Crack-block inserted into original element definitions



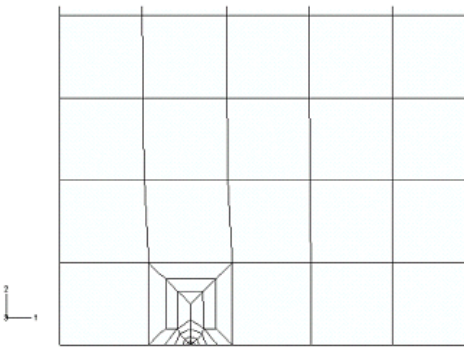
Crack-block inserted with boundary shift



Crack-block inserted with boundary shift and relaxation of surrounding elements

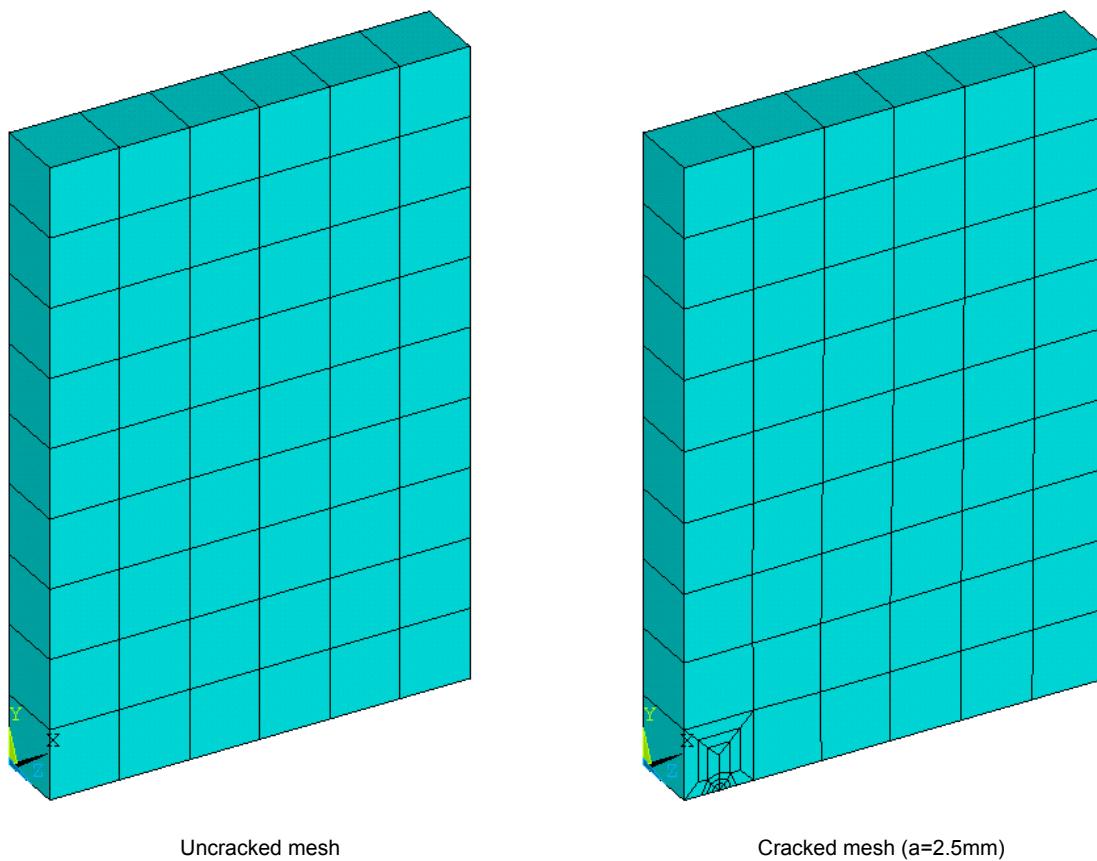
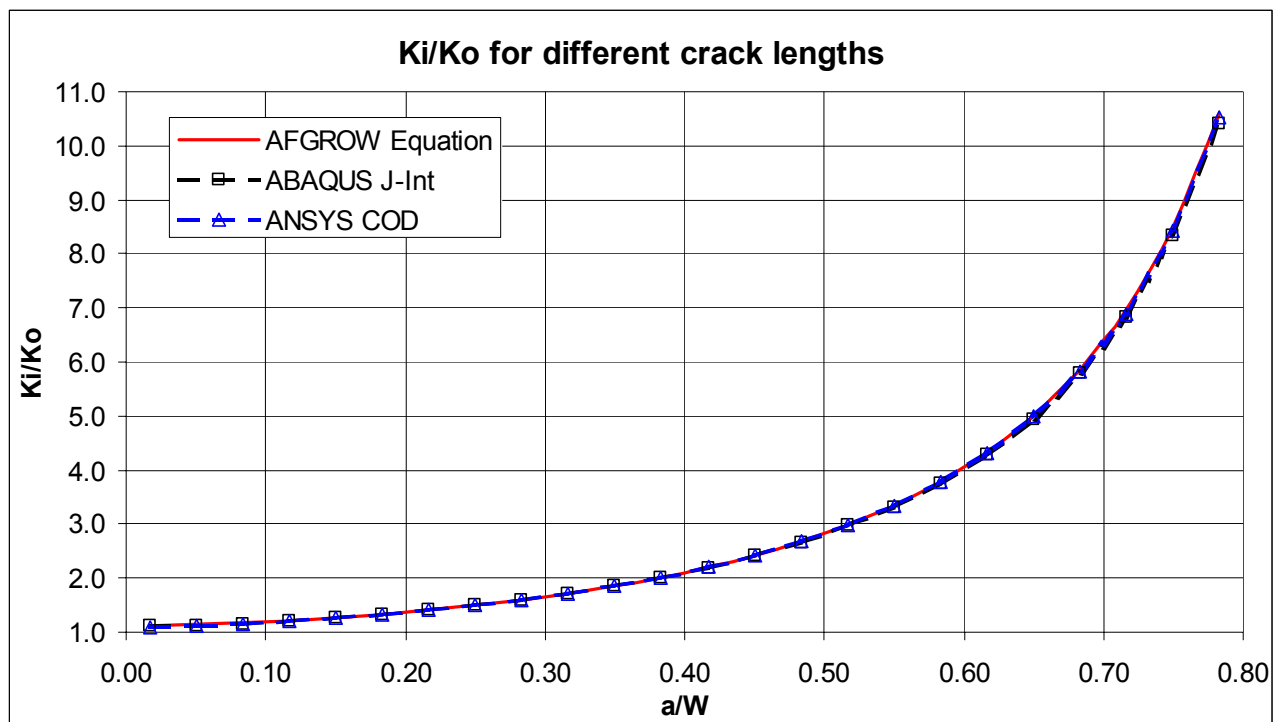


Crack-block transferred to new location after growth



Advanced position in new location

Figure 4 - Demonstration of boundary shifting, relaxation and crack-block transfer

**Figure 5 - SEN model****Figure 6 - Comparison of COD with theory for a SEN specimen**

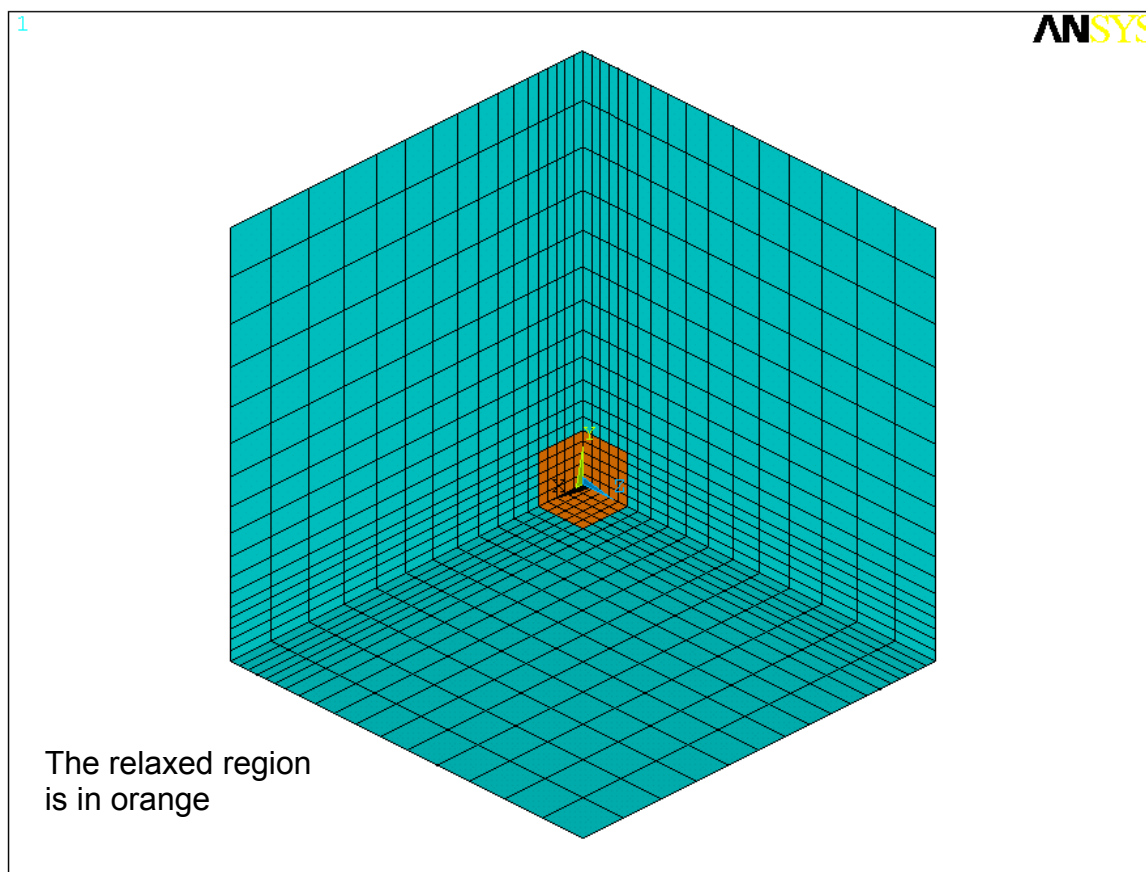
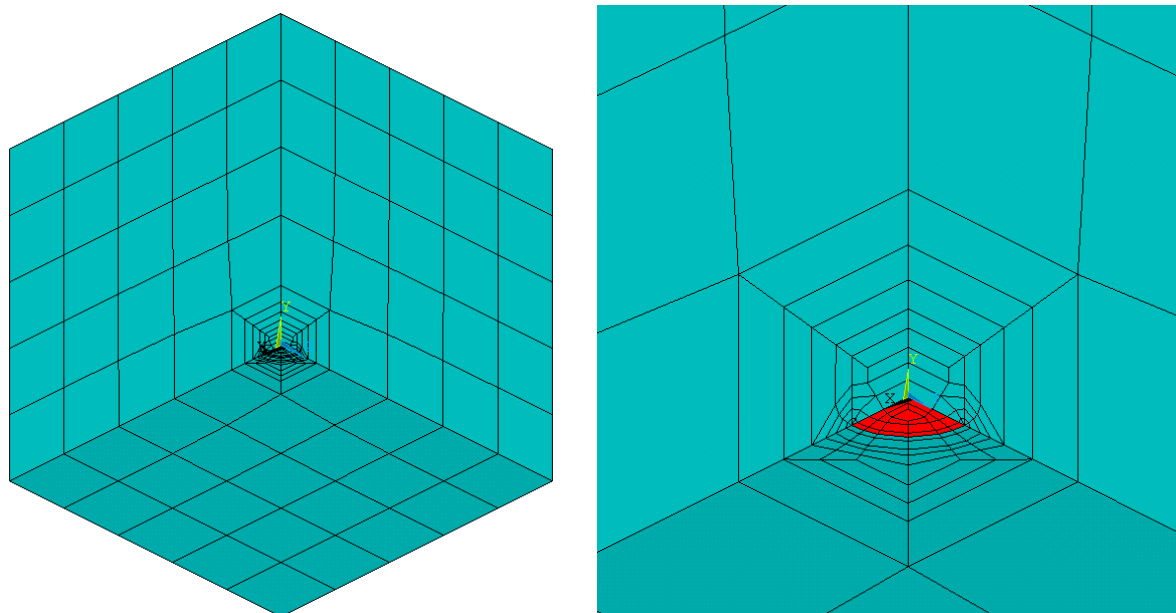


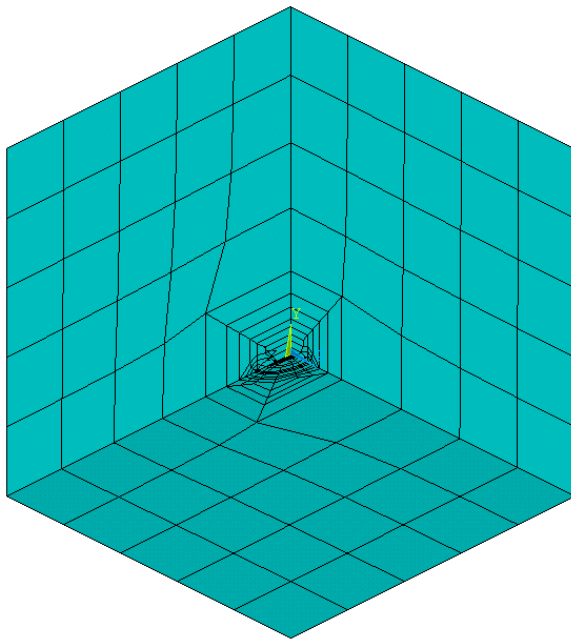
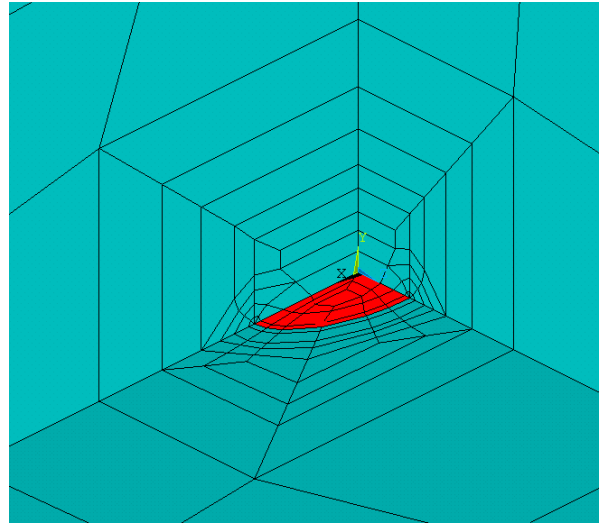
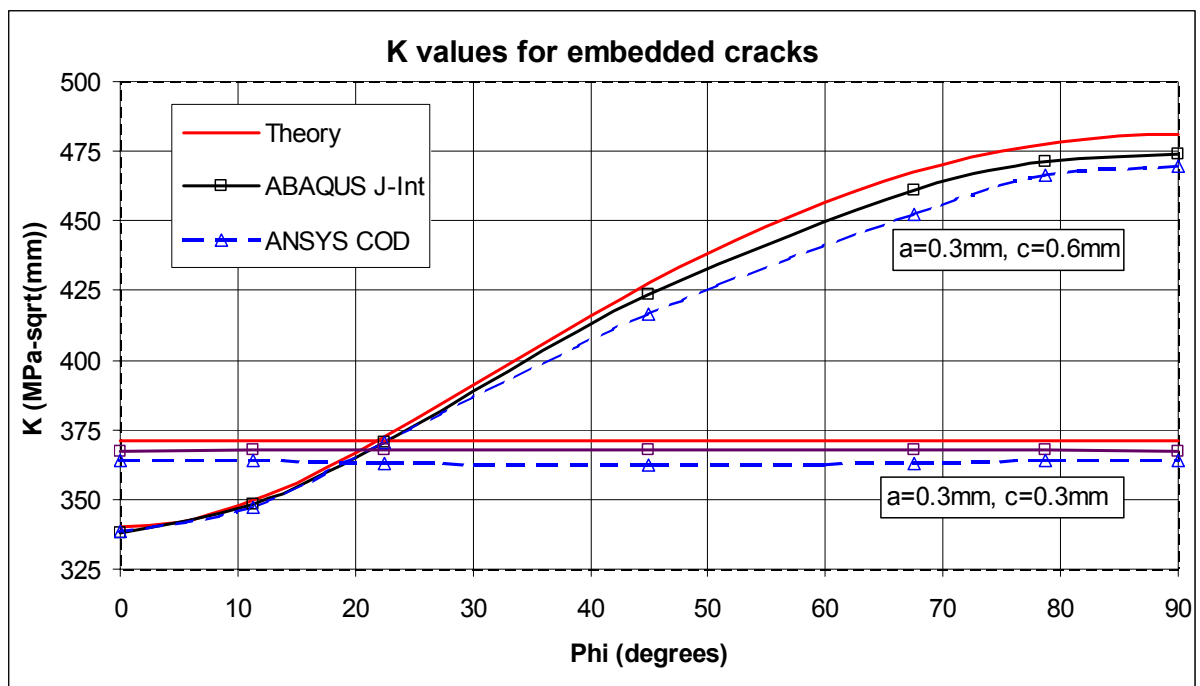
Figure 7 - Uncracked mesh for the embedded crack



Relaxed region of the mesh (see Figure 7)

Close-up of crack region

Figure 8 - Cracked mesh with a single standard quarter circular crack-block ($a=0.75\text{mm}$, $c=0.75\text{mm}$)

Close-up of crack region ($a=0.3\text{mm}, c=0.6\text{mm}$)Close-up of crack region ($a=0.3\text{mm}, c=0.6\text{mm}$)**Figure 9 - Embedded elliptical crack ($a=0.3\text{mm}, c=0.6\text{mm}$)****Figure 10 - Comparison of COD with theory for a fully embedded crack**

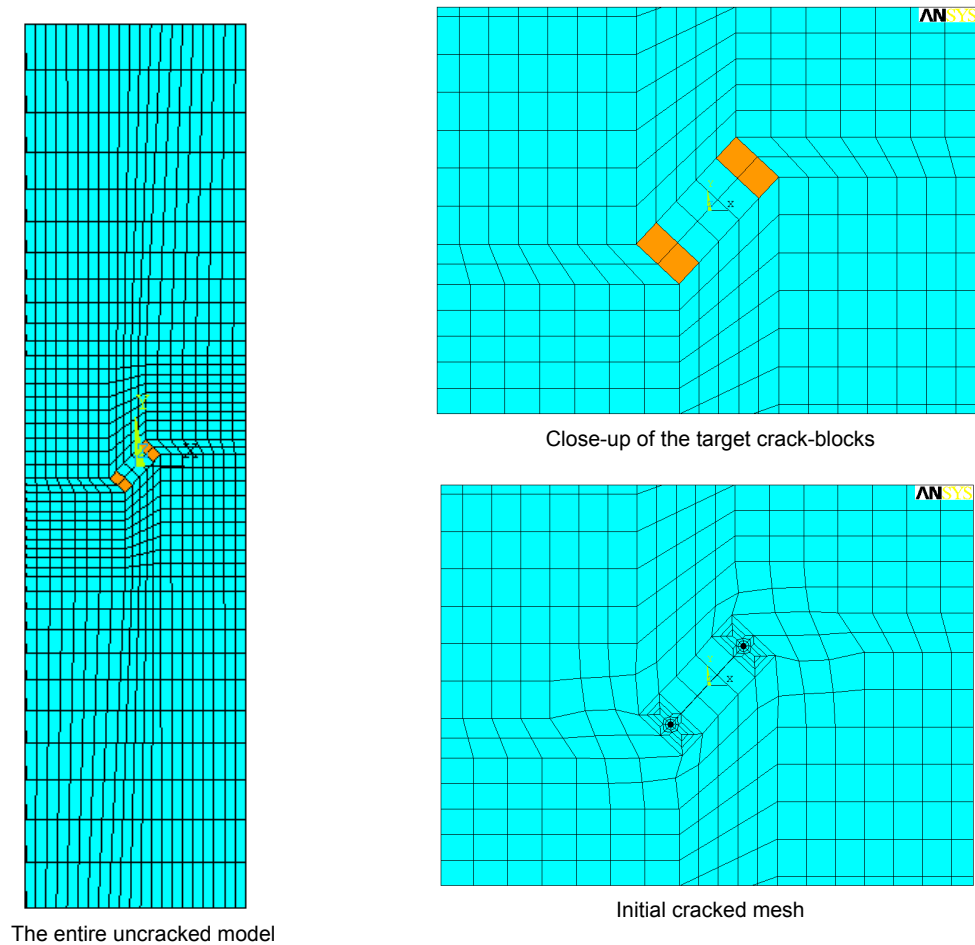


Figure 11 - Mesh for an oblique crack at 43°

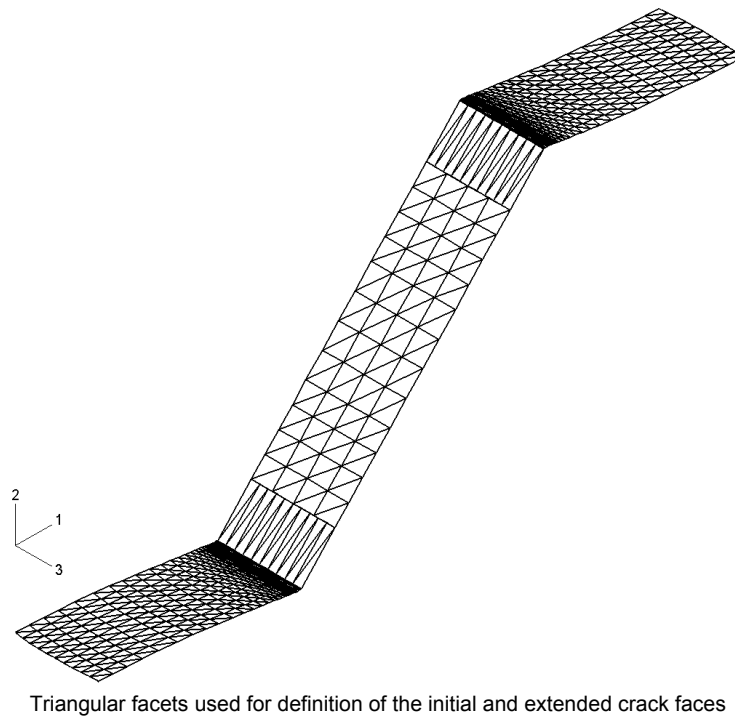


Figure 12 - Crack growth profiles for the oblique crack at 43°

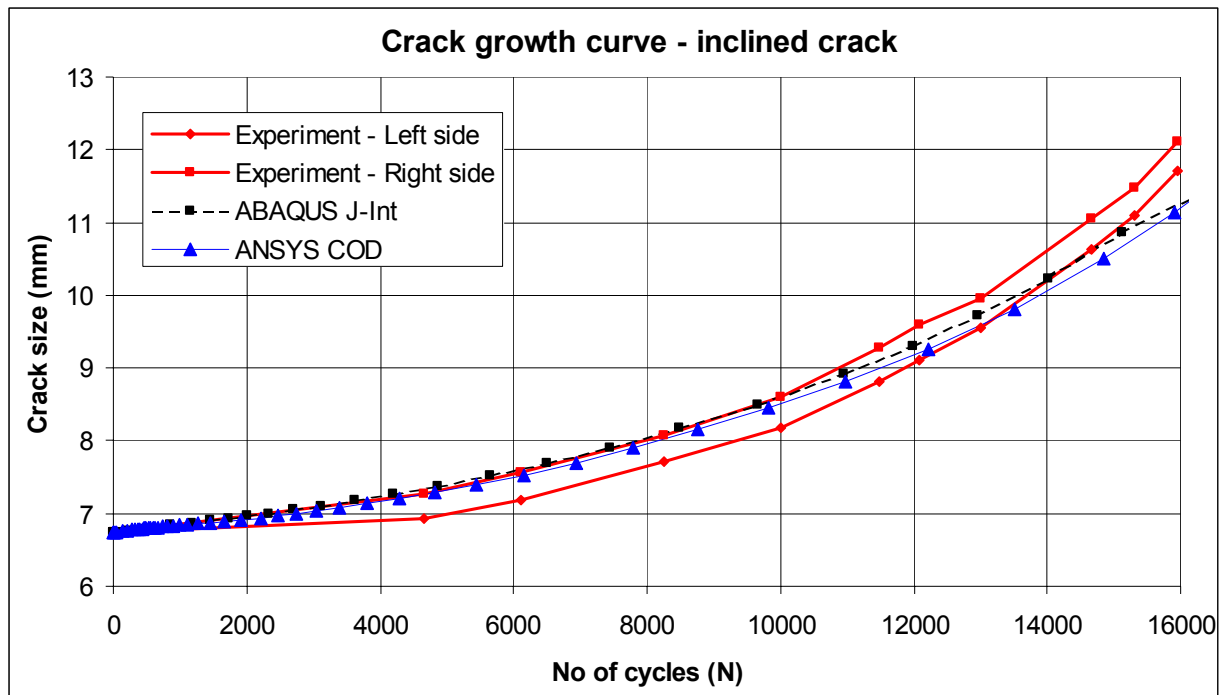


Figure 13 - Crack growth rate for the oblique crack at 43°

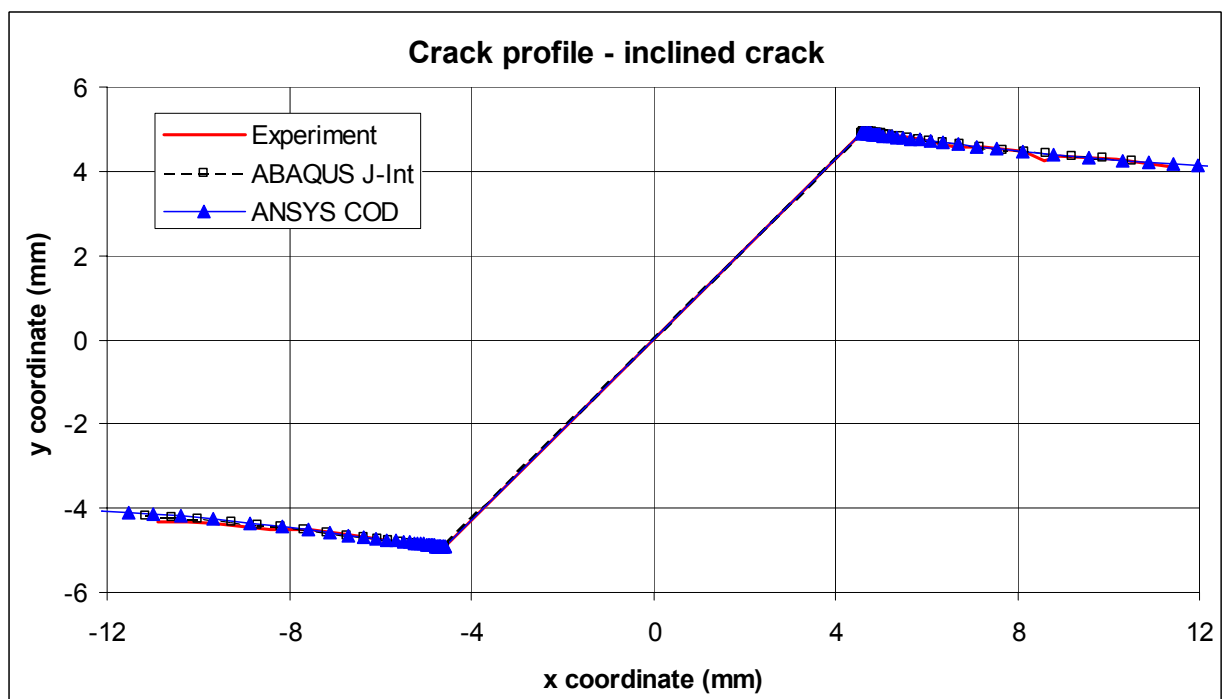
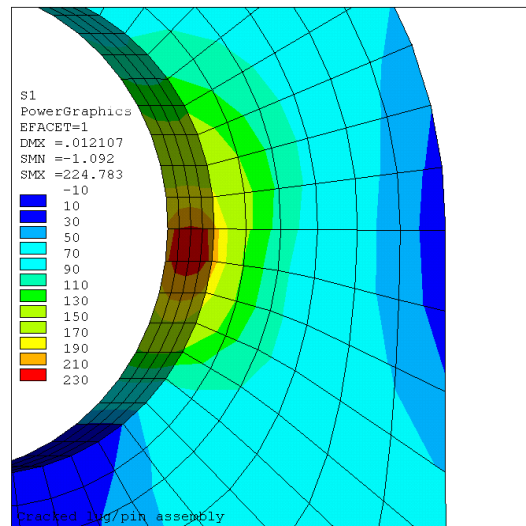
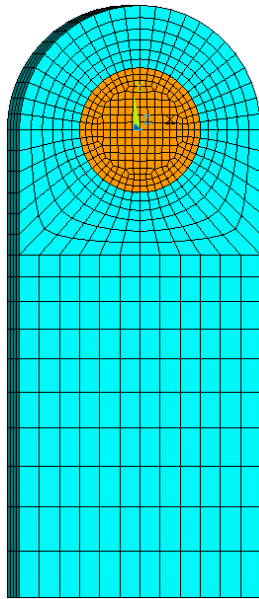
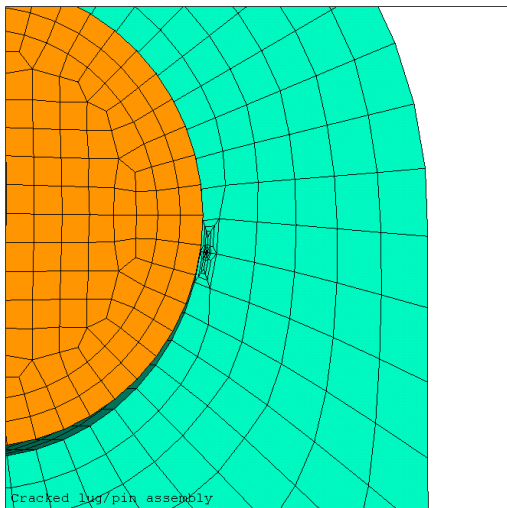


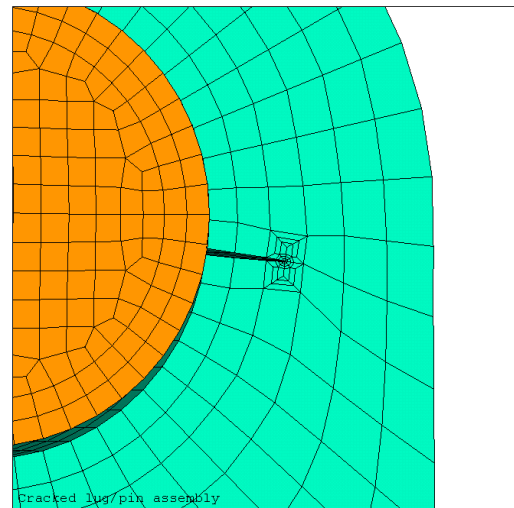
Figure 14 - Comparison of the ZENCRACK and experimental crack growth path for the oblique crack at 43°



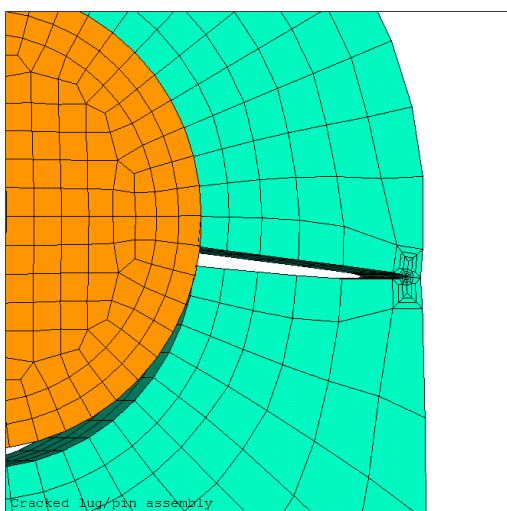
Maximum principal stress in the uncracked mesh used to determine an initial defect location



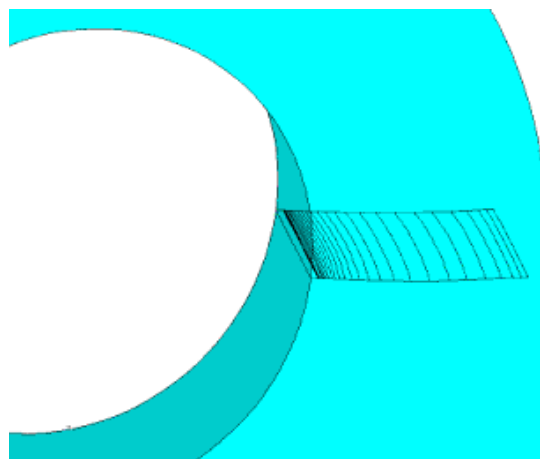
Initial defect position



Crack-blocks transferred into the third row of elements



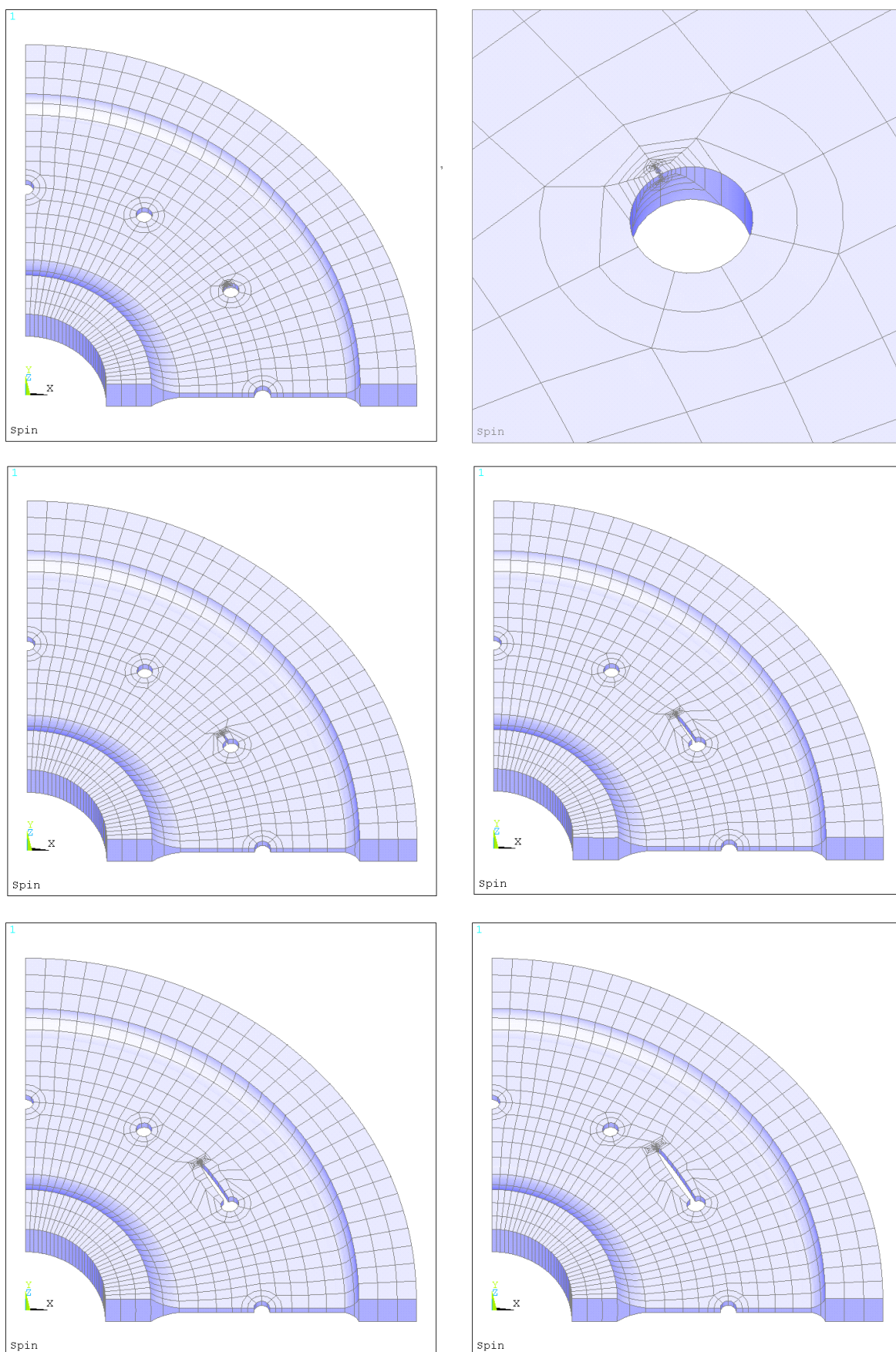
Crack-blocks transferred into the sixth row of elements



Calculated crack profiles

An initial through crack grows through the lug. Note the crack path is independent of element boundaries in the uncracked mesh and that the mesh surrounding the defect is modified as the crack advances.

Figure 15 - Pin lug example

**Figure 16 - Compressor disk example**