

3D Crack Analysis Using MSC.Marc

Chris Timbrell, Gerry Cook, Ramesh Chandwani
Zentech International Limited, 103 Mytchett Road, Camberley,
Surrey, GU16 6ES, U.K.
<http://www.zentech.co.uk>

Introduction

This paper describes software which is interfaced to MSC.Marc for analysis of arbitrary 3D cracks. The software, Zencrack, can be used in three ways:

1. For generating 3D finite element meshes containing multiple crack fronts from a finite element model of an uncracked component.
2. For determining the distribution of the maximum energy release rates and stress intensity factors along crack fronts.
3. For automatically calculating fatigue or sustained load crack growth in a general 3D body under arbitrary loading.

Items 2 and 3 use the results from the LORENZI option of a MSC.Marc analysis. The raw data from this option is processed to calculate energy release rates along the crack front.

There are many complex issues to be addressed in 3D crack growth prediction. This paper presents the key concepts of the software implementation and the interface to MSC.Marc. A high level overview of the procedure is given in Figure 1.

Mesh generation for 3D cracks

A schematic diagram of the steps taken to generate a cracked mesh is given in Figure 2. The user must generate a finite element mesh for an uncracked component. This must consist of 20 noded brick elements in the region(s) where the crack(s) are required. An ancillary file is used to define the crack related data.

A crack-block approach is used to introduce the crack front. The term crack-block refers to a collection of 20 noded brick elements stored as a unit cube. The arrangement of these crack-blocks is such that they contain either a quarter circular or through crack front on one face. Part of this face is allowed to open up under loading forming the opening crack face within the crack-block.

Examples of unit cube crack-blocks are shown in item 2 of Figure 2 with their opening crack faces shaded for clarity.

The elements forming the crack front in each crack-block are modelled using collapsed 20 noded brick elements, with an option to control the midside node positions extending radially from the crack front to allow generation of quarter point nodes if required. The crack front itself is seen as a line of nodes on the crack-block surface. The internal mesh of the crack-block coarsens away from the crack front such that some of the crack-block faces can be matched with a standard 20 noded brick element. The other highly populated faces can be left as free surfaces, symmetry surfaces, or connected to other compatible crack-blocks.

The meshing procedure is one of replacement of one or more 20 noded brick elements in the user supplied uncracked mesh by crack-blocks, as shown in Figure 2. The elements to be replaced can be of arbitrary shape, but the faces which become the initial crack plane should be as flat as possible to minimise element distortion. During the mapping process to introduce the crack-blocks the user can control the size of the generated crack front section for each crack-block. Crack-blocks can be connected together to form distinct crack fronts of the required size in the cracked mesh. In the first example of Figure 3 two crack-blocks have been merged together to form a single semi-elliptic crack front. This is done by replacement of elements 29 and 30 in the uncracked mesh. The crack orientation is defined by key node positions 177, 162, 219. The initial crack size is defined by ratios along the key edges defined by these nodes.

In the simplest case a cracked mesh may contain only a single crack-block (and therefore a single crack front). If there is a single crack-block then only one side of the crack (i.e. one crack face) is modelled and symmetry constraints should be applied. These can be applied in the uncracked mesh and automatically updated by Zencrack. Figure 4 shows an example of update of symmetry conditions from the uncracked mesh to the cracked mesh for a semi-elliptic crack. The program can also update pressure loads in the crack region as shown in the example of Figure 5. (Note that in the updated mesh each element is listed in a separate entry on the DIST LOADS option since the face numbers vary from one element to the next – this results in multiple boundary condition sets when the cracked mesh is read by Mentat).

If both sides of the crack (i.e. both crack faces) are to be modelled, then pairs of crack-blocks are used with a face-to-face match of the crack-blocks. This approach also allows modelling of fully embedded cracks. Examples of embedded cracks in a cube block are shown in Figure 6 (with one side of the model removed to allow the cracks to be seen).

All merging of crack-blocks with one another and with the uncracked mesh is carried out automatically to create a new finite element mesh containing the required crack fronts. If requested, this new cracked mesh can be submitted for f.e. analysis or a crack growth analysis can be carried out with this mesh as the starter crack.

If the crack grows too large to be contained within the crack-blocks a restart analysis can be carried out. This takes the history of crack growth calculated in the analysis and continues growth with a newly modelled mesh containing larger crack-blocks.

Crack growth prediction

Two parameters are generally used in L.E.F.M. (linear elastic fracture mechanics) to describe conditions at a crack front:

K_I K_{II} K_{III} the stress intensity factors for mode I, II and III loading respectively.

G the energy released at a crack front due to an increase in crack surface area.

It is known that for mode I loading, K and G can be related by the material properties (E and ν) and the state of stress :

$$\frac{G}{1 - (\alpha \nu)^2} = \frac{K^2}{E}$$

where :

$\alpha=0$ for plane stress

$\alpha=1$ for plane strain

Crack growth is generally evaluated using one of the parameters K or G. For fatigue crack growth the simplest equation relating loading and material data to crack growth and fatigue cycles is the Paris equation of form:

$$\frac{da}{dn} = C \Delta K^m$$

where:

a = crack growth

n = number of fatigue cycles

C, m = material constants

ΔK = stress intensity factor range

Generalising these expressions allows the crack growth rate to be written as a function of ΔK or ΔG :

$$\frac{da}{dn} = f(\Delta K) \quad \text{or} \quad \frac{da}{dn} = f(\Delta G)$$

Zencrack always uses the second of these forms. The energy release rate G can be accurately calculated using the finite element method. Since the energy release rate is a fundamental quantity describing the formation of new crack surfaces under any state of stress, the difficulties of assessing the state of stress do not arise.

A Paris equation in terms ΔK for mode I loading can be re-formulated in terms of ΔG . It is argued that when other modes of loading exist, the energy release rate can always be used to describe crack growth from the mode I test data.

The direction of crack growth in a general case is assumed to be in the direction of maximum energy release rate. The energy release rate approach has an added advantage in that it automatically takes care of effects such as anisotropy, plasticity, creep strains and other material non-linearities.

In order to take full account of such effects, the LORENZI extended J-integral virtual extension technique is used in MSC.Marc.

Using the energy release rate distribution, the direction and magnitude of crack growth is evaluated using an integration scheme and the crack growth law. This integration is controlled by tolerances on the allowable “da” and “dn” increments

between one finite element analysis and the next. Figure 7 shows a typical curve of growth against load cycles for a point on a crack front. It is clear from this figure that early on in the analysis the increment should be controlled by “dn” and that the control should switch to “da” as the crack extends.

An automatic crack advancement scheme in Zencrack updates the crack front and generates a new finite element model containing the updated crack front.

Interface to MSC.Marc

Update of the uncracked model to incorporate the crack front(s) requires special treatment of a number of MSC.Marc keywords:

SETNAME	updated to allow for new crack related node and elements sets
COORDINATES	new nodes added
CONNECTIVITY	new elements added, replaced elements removed
FIXED DISP	update supports in crack region (e.g. Figure 4)
DIST LOADS	update pressure loads in crack region (e.g. Figure 5)

A key feature for the cracked mesh is the LORENZI option. To allow analysis of an arbitrary 3D crack the treatment of data for this option is non-trivial. Several virtual crack extension directions are defined in order allow calculation of the maximum energy release rate and its direction. This is shown schematically in Figure 8. In addition, the raw values generated by the LORENZI option are local nodal energy release rate values. These must be integrated along the crack front paying attention to the element shape functions and the applied virtual crack extension. The scheme developed by Zentech for this processing is described in Ref. 1.

Zencrack recovers the results from the LORENZI option via the user subroutine IMPD:

- During mesh generation Zencrack writes a file specifying the number of nodes on the crack front and the number of LORENZI evaluations per node
- During the Marc analysis this file is read during the first call to IMPD in order to establish the quantity of LORENZI output to be extracted from each analysis increment

- LORENZI results data is extracted directly from the relevant common block and written to a new file
- On completion of the Marc analysis all the LORENZI values are read from the generated data file by Zencrack

Example - Compact Tension Specimen

An ASTM standard compact tension specimen has been modelled using dimensions shown in Figure 9. A straight crack has been generated in the model as can be seen in the mesh plots Figure 10 and Figure 11. The mesh is a quarter symmetry model (half length and half thickness). Load is applied as displacement control to the rigid surface load pin. A deformable contact body is defined around part of the load hole. The load magnitude is recovered by summation of reaction forces. The analysis provides an energy release rate distribution which can be converted to stress intensity factor if an assumption is made regarding stress conditions along the crack. The plane strain and plane stress bounds are shown in Figure 12 compared with a reference solution (Ref. 2).

Example – Growth of an inclined crack in a plate

This analysis demonstrates crack growth prediction in a plate under cyclic axial load. The plate contains an initial defect inclined at 43 degrees to the load direction as shown in Figure 13. The full specification for this problem is given in Ref. 3.

A displaced plot of the initial defect mesh is shown in Figure 14. Contour plots showing the crack at two points during the growth prediction are shown in Figure 15. As may be expected the crack very quickly turns to take up a mode I configuration.

Comparison with experimental results is shown in Figure 16 and shows excellent agreement. A typical plot of growth magnitude against number of load cycles is shown in Figure 17.

References

- Ref. 1 “Maximum Energy Release Rate Distribution From A Generalised
3D Virtual Crack Extension Method”, P.W.Claydon, Zentech
International Ltd., Engineering Fracture Mechanics, 42 (6), 1992
- Ref. 2 Ewalds, H.L. & Wanhill, R.J.H., Fracture Mechanics, Edward
Arnold (Publishers) Limited, 3rd imprint 1986 (ISBN 0-7131-3515-
8)
- Ref. 3 “Fatigue crack propagation in titanium under general in-plane
loading : I – Experiments”, Pustejovsky, M.A., Eng. Frac. Mech.,
Volume 11, pg 9-15, 1979

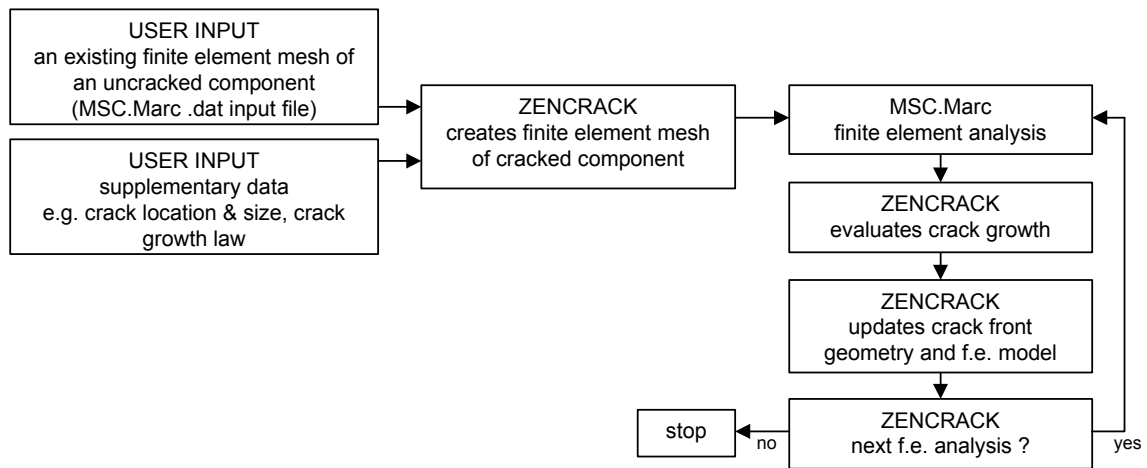


Figure 1 – Overview of crack growth scheme

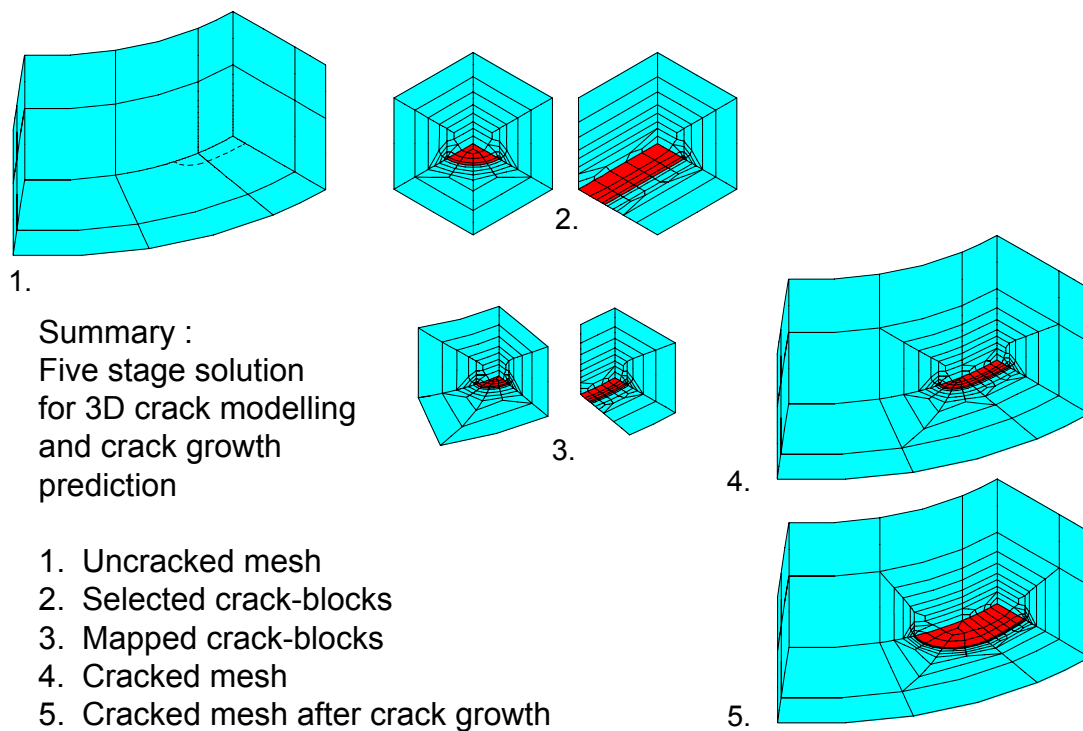


Figure 2 – Example of meshing using crack-blocks

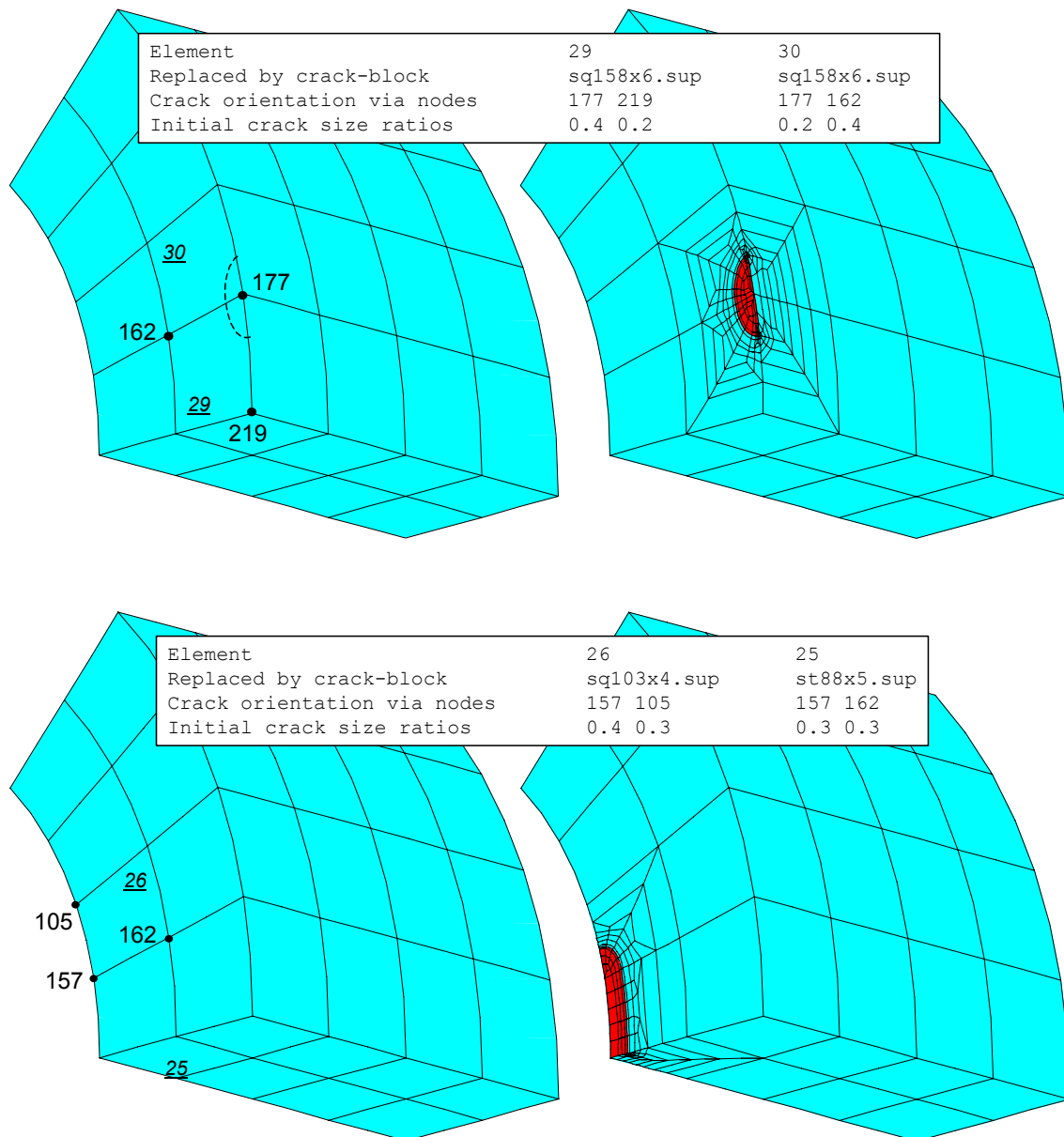
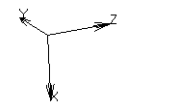
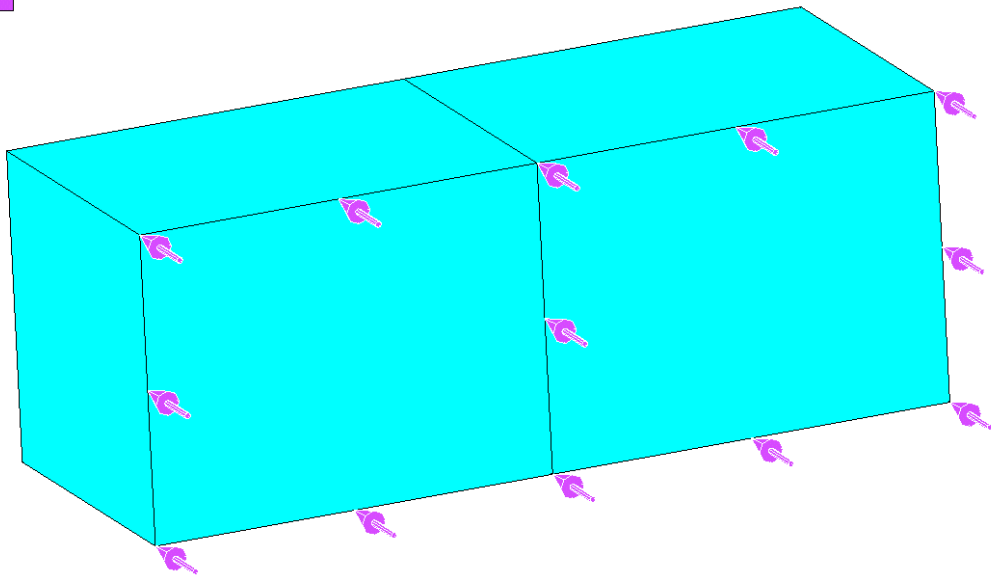


Figure 3 – Definition of an initial crack

apply1



apply1

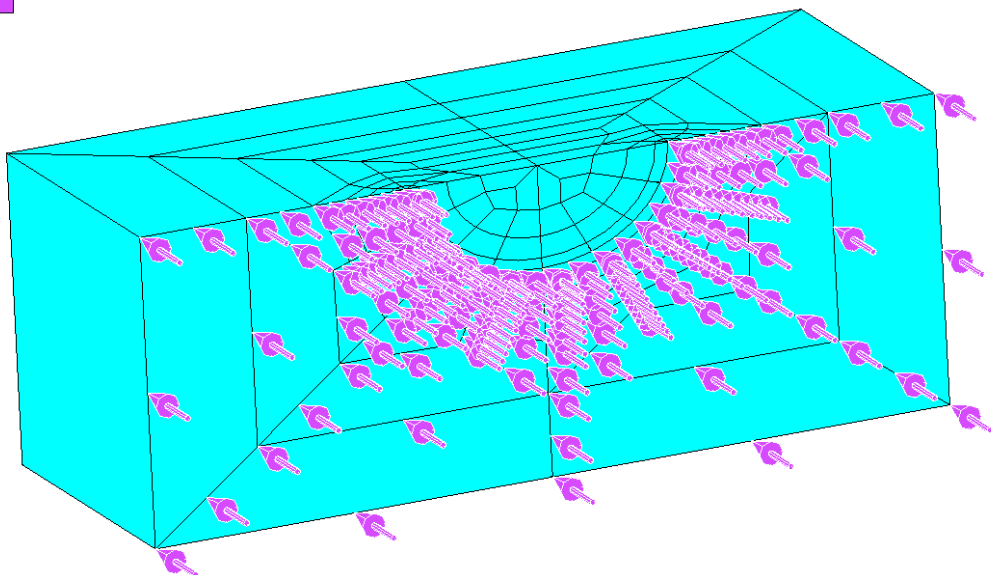


Figure 4 – Update of FIXED DISP data for a cracked mesh

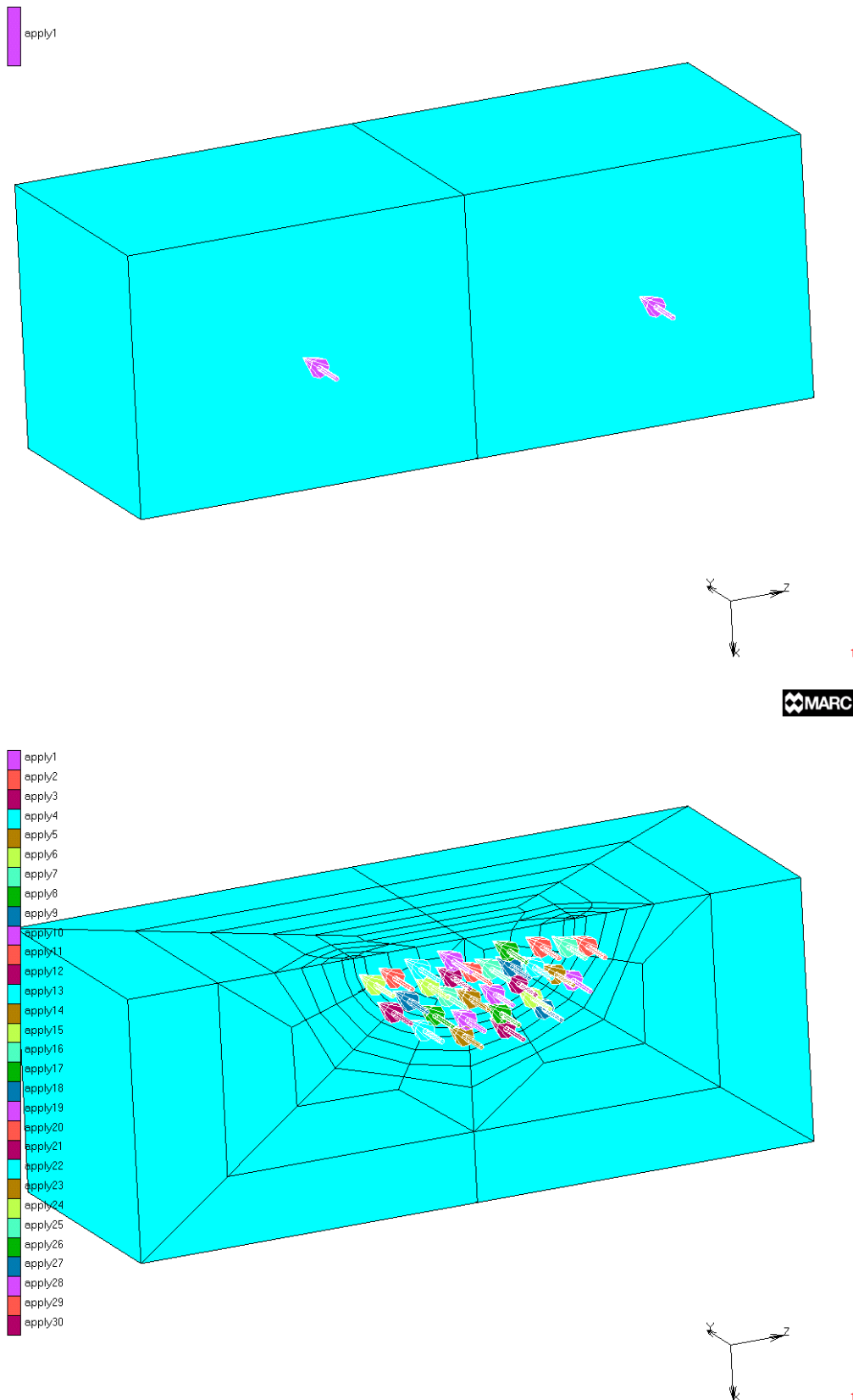


Figure 5 – Example of update of DIST LOADS data for a cracked mesh

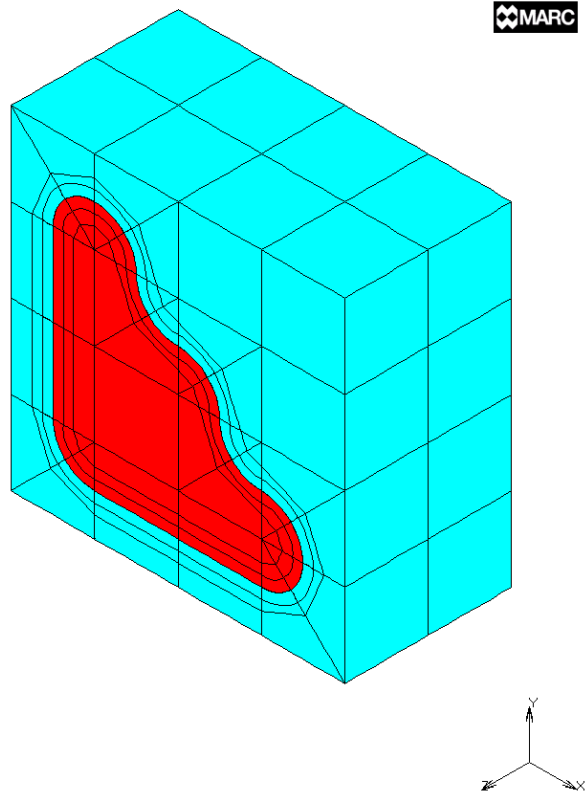
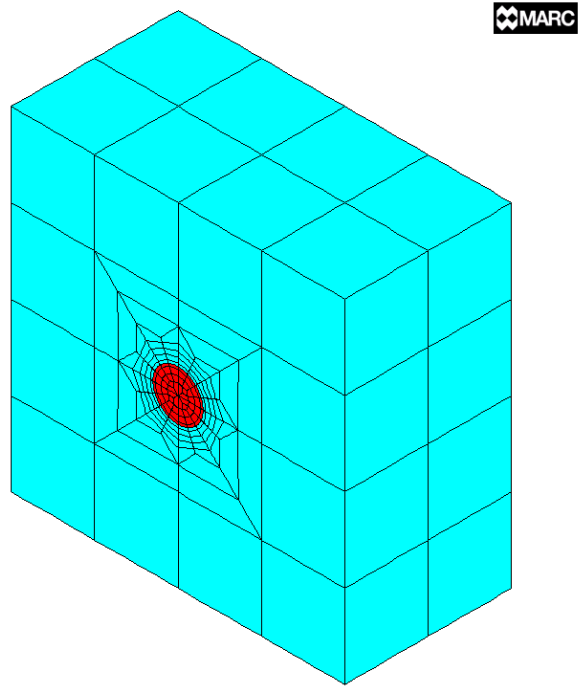


Figure 6 – Fully embedded cracks (one side shown)

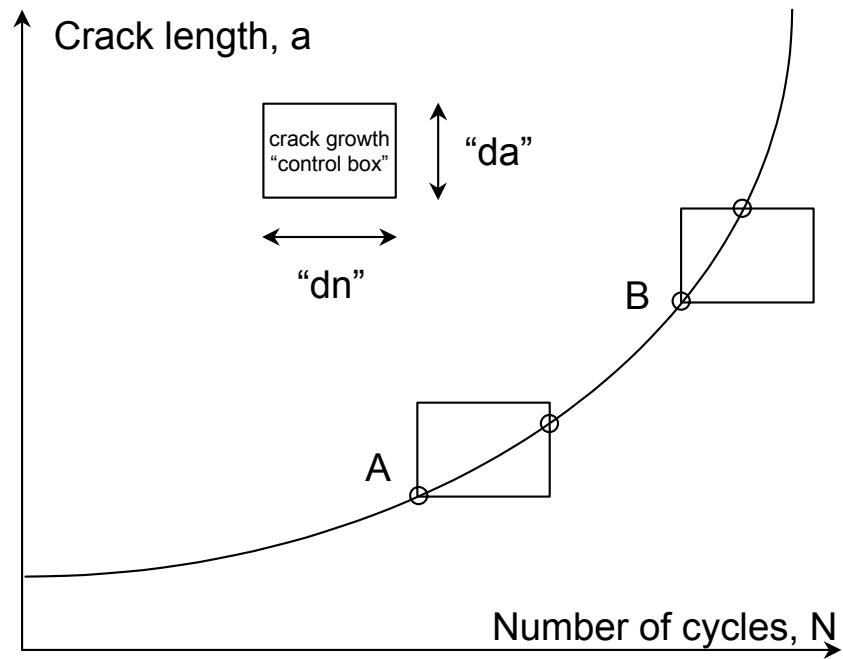


Figure 7 – Controls on crack growth increment

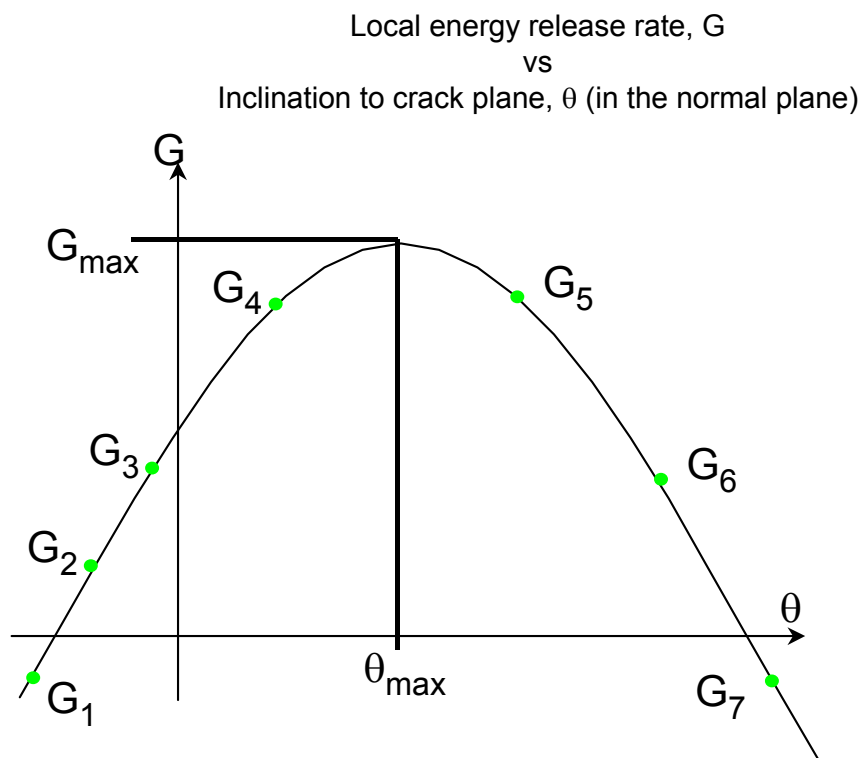
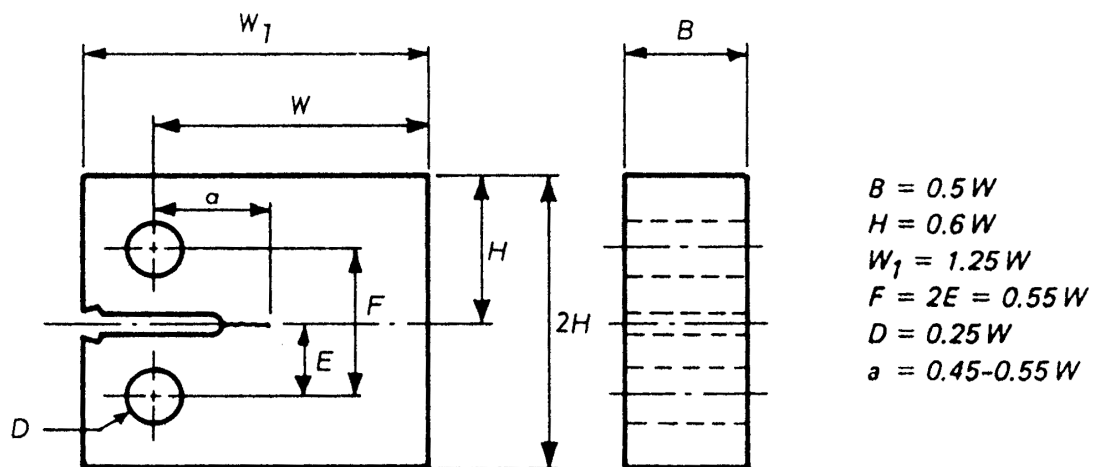


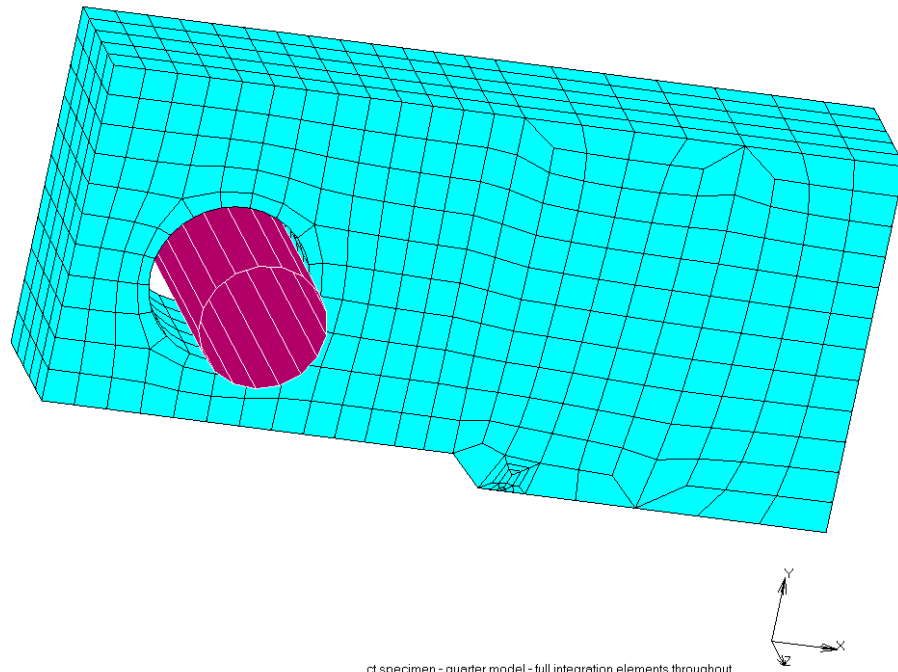
Figure 8 – Distribution of energy release rate around a crack front



$a=48.7, W=100$

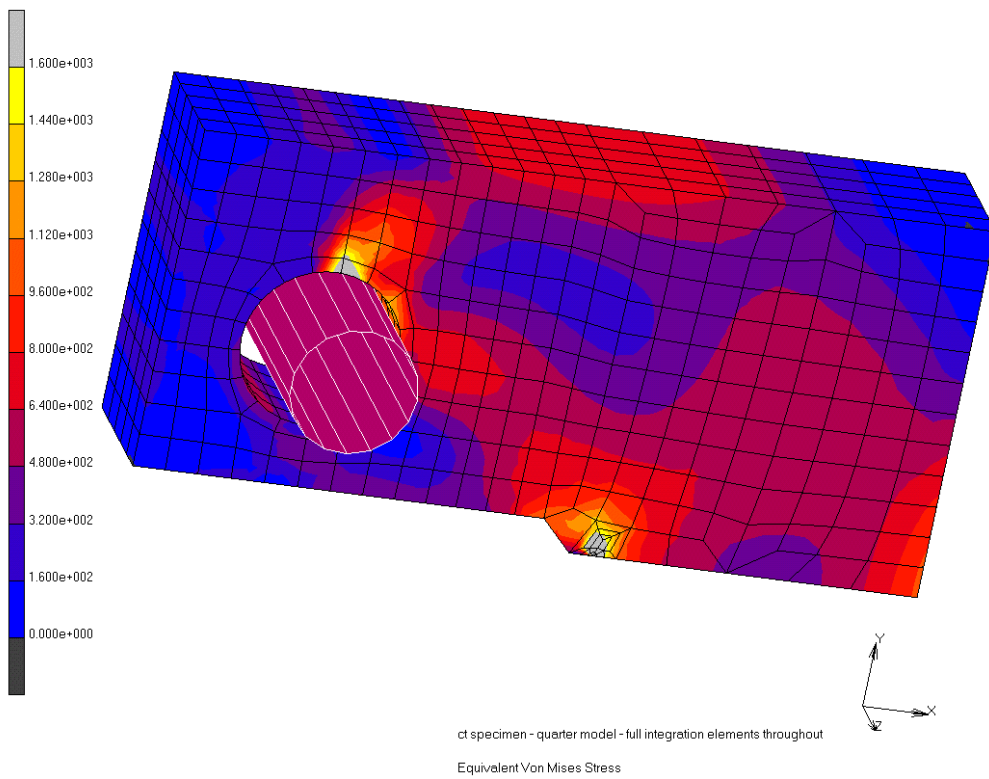
Figure 9 – ASTM standard compact tension specimen

Inc : 1
Time : 1.000e+000



1

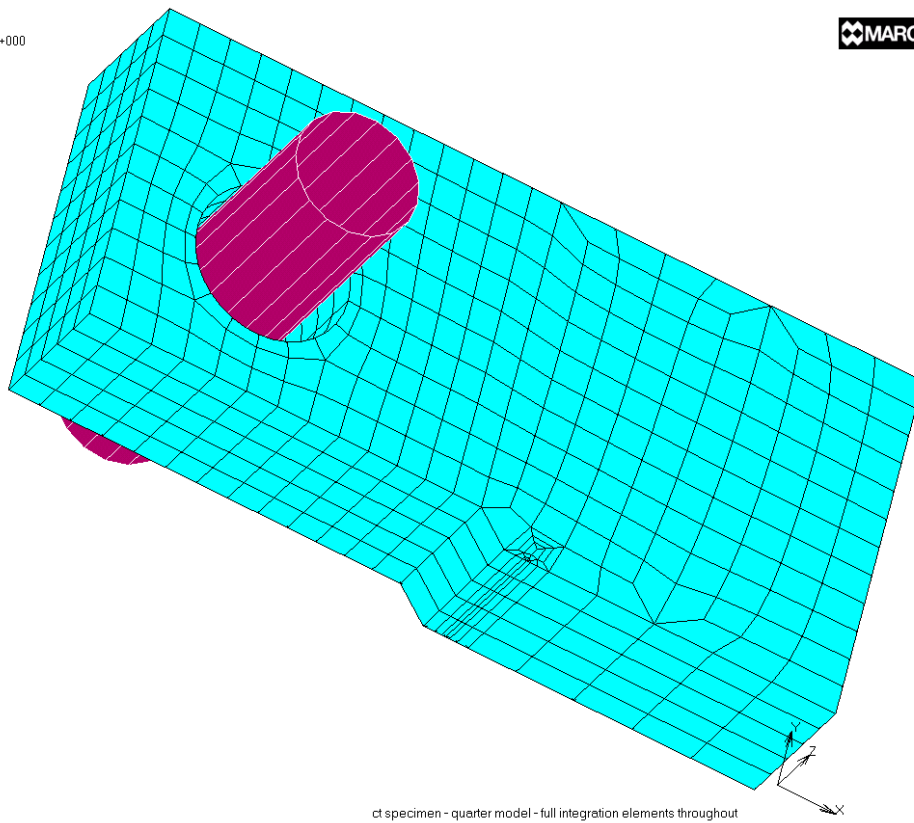
Inc : 1
Time : 1.000e+000



1

Figure 10 – Compact tension specimen

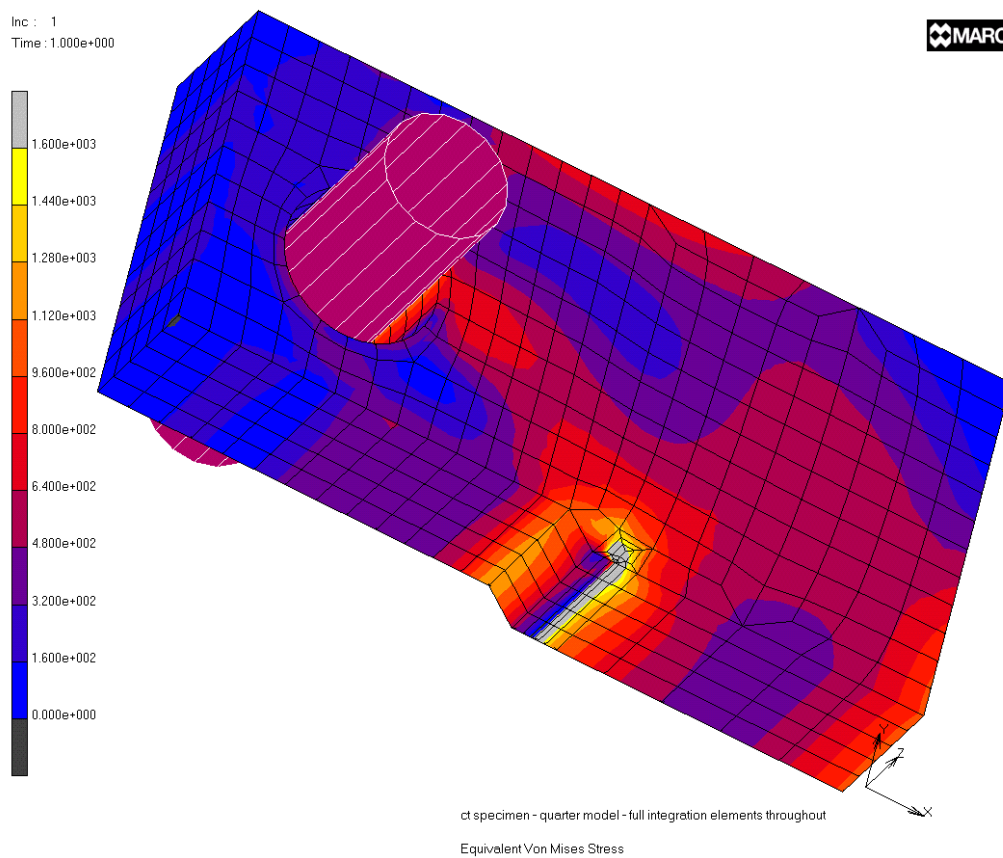
Inc : 1
Time : 1.000e+000



ct specimen - quarter model - full integration elements throughout

1

Inc : 1
Time : 1.000e+000



ct specimen - quarter model - full integration elements throughout
Equivalent Von Mises Stress

1

Figure 11 – Compact tension specimen

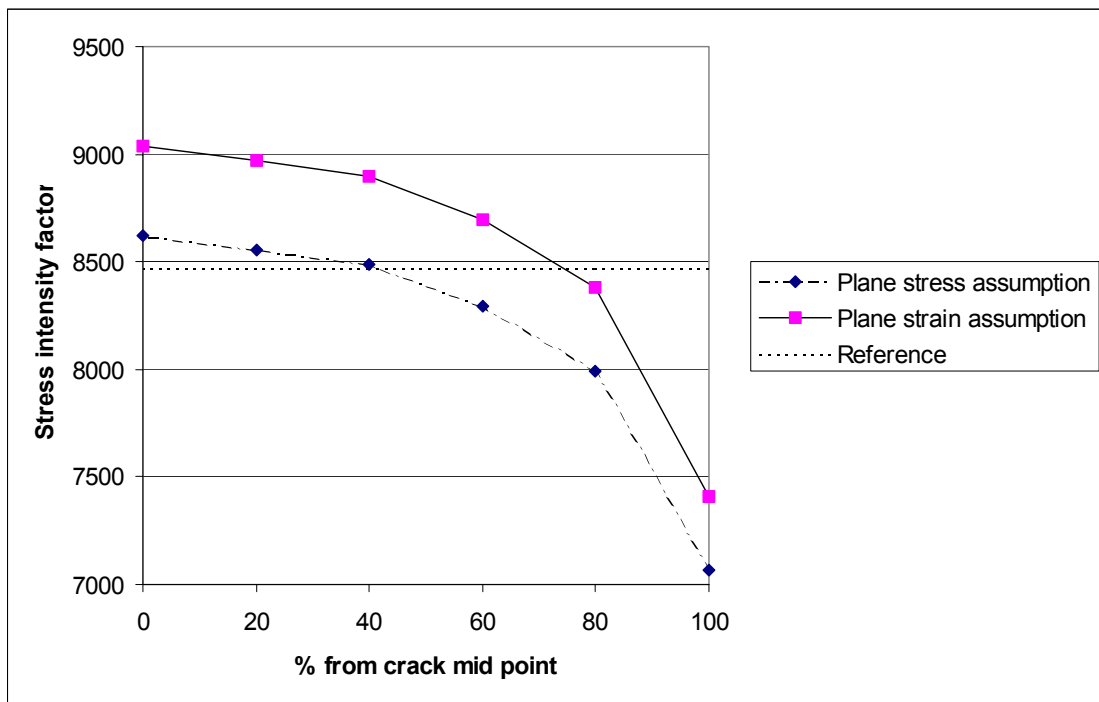


Figure 12 – Stress intensity factor for compact tension specimen

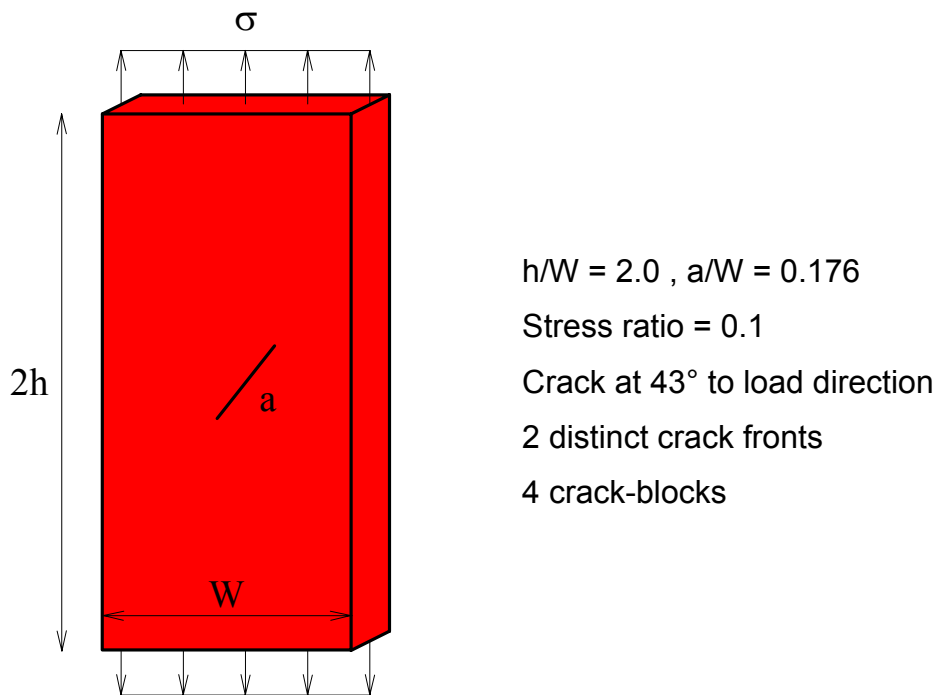


Figure 13 – Plate with inclined crack

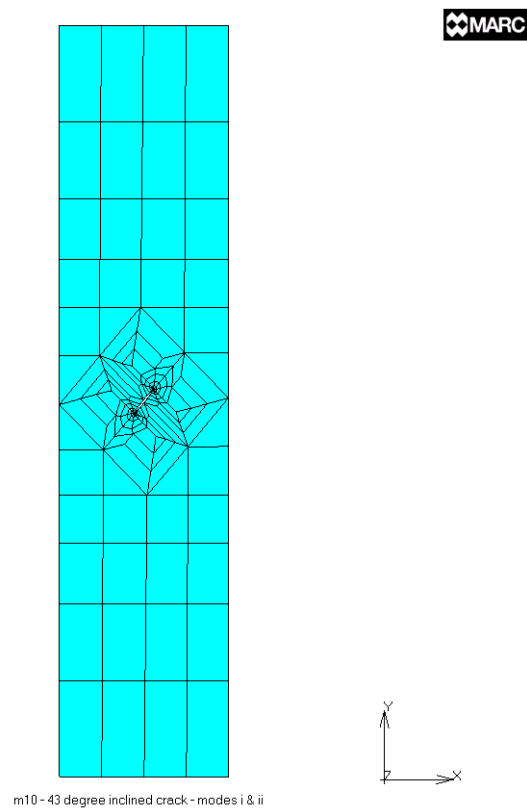


Figure 14 – Initial defect for 43 degree inclined crack model

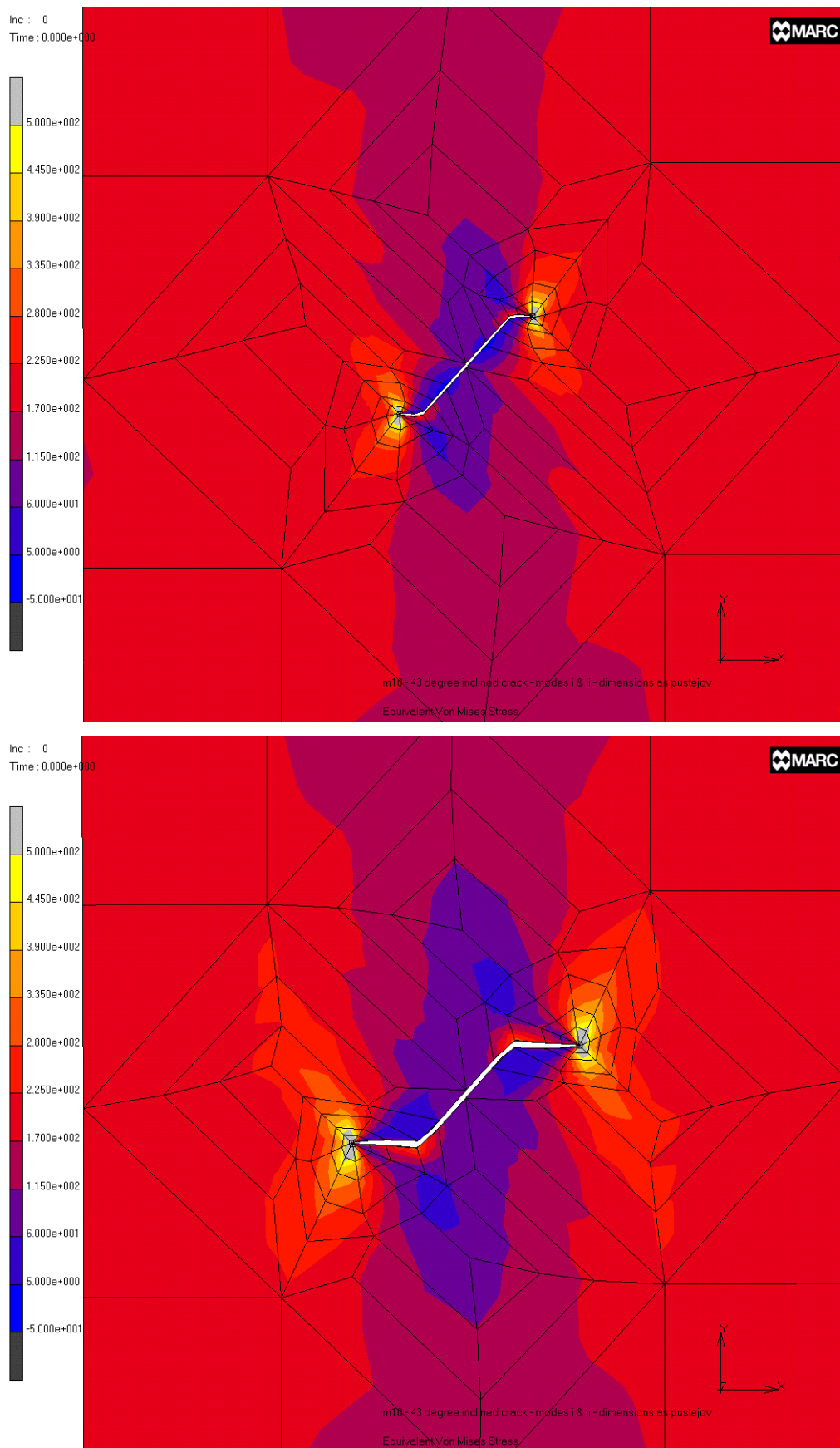


Figure 15 – Crack model at two stages during crack growth prediction

Experimental results 12080, 15960
 (Dotted blue line represents a reference
 calculated solution)

Zencrack results 12075, 15793

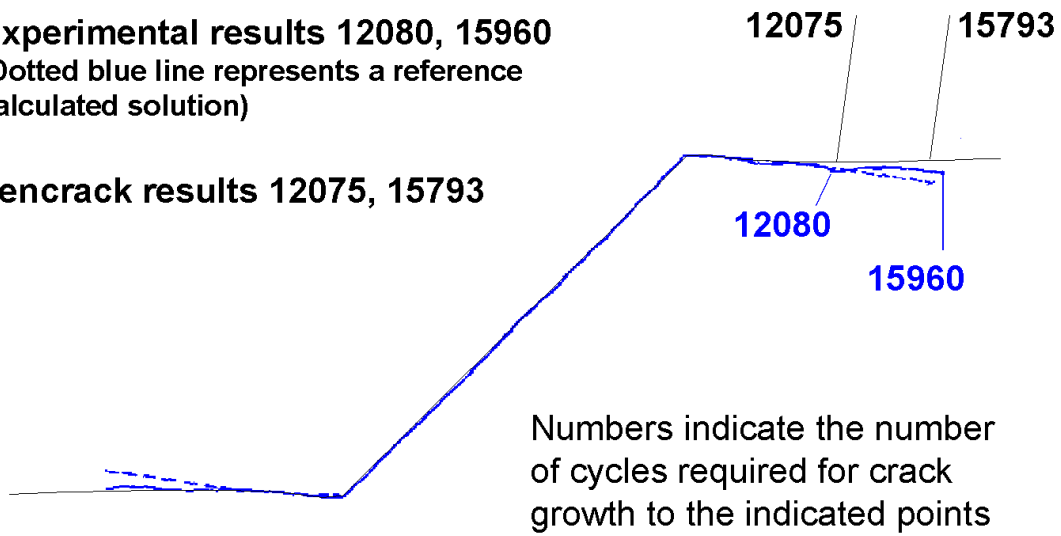


Figure 16 – Comparison with experimental results

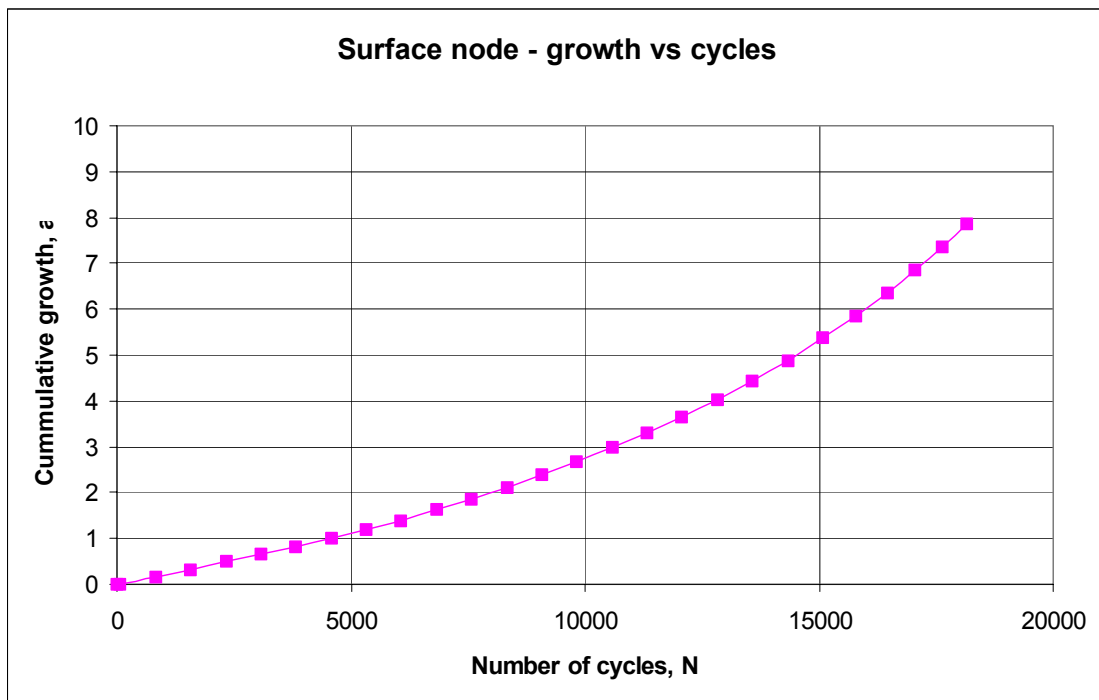


Figure 17 – Typical growth curve