

3D FE Fracture Mechanics Analysis for Industrial Applications

C. Timbrell, G. Cook

Zentech International Limited, 103 Mytchett Road, Camberley, Surrey, GU16 6ES

Tel: 01252 376288 Fax: 01252 376389 Web: <http://www.zentech.co.uk>

Introduction

This paper gives an overview of techniques that have been used by Zentech for analysis of cracks in 3D geometries in a number of commercial applications. These techniques are incorporated into the Zencrack program (Ref. 1) which is interfaced to Abaqus (Ref. 2) and Marc (Ref. 3). Simple demonstrative examples and some consultancy applications are included. Some of the difficulties encountered in this type of analysis are noted and future areas of interest are outlined.

Crack modelling in 3D

The first issue that must be addressed in 3D FE 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 :

- meshing of a complex crack shape and / or orientation in a component
- preventing opposite crack faces from merging together

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 demonstrates this using an edge notch in a plate.

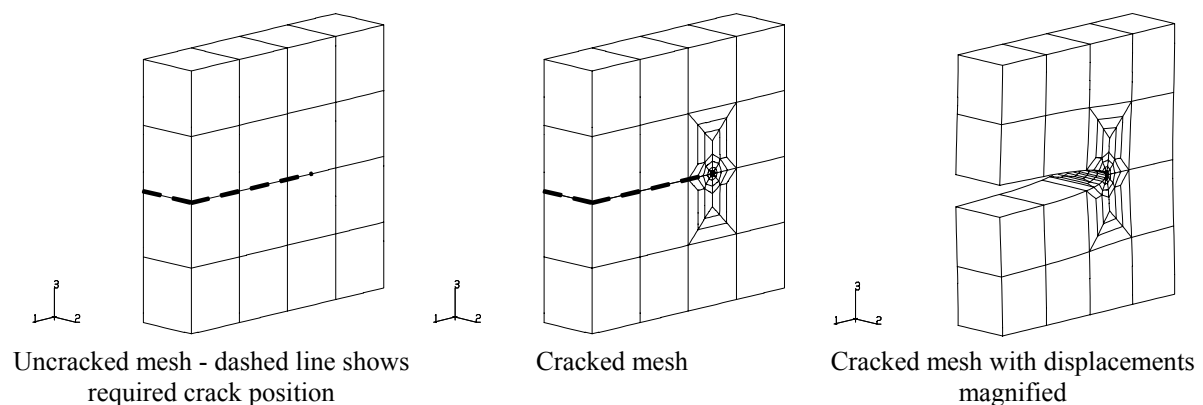


Figure 1 - Simple example of crack modelling

The meshing technique is one in which 20 noded brick elements in an uncracked mesh are replaced by crack-blocks. The size of the crack in the crack-block is user-defined and may be a straight, curved or elliptic section of crack front. Crack-blocks may be joined together as in Figure 1 to model both sides of a crack. They may also be joined to form longer sections of crack front. If there are displacement boundary conditions or pressure loads in the cracked region they can be automatically transferred onto the cracked mesh. As in this example, other element pairs in the mesh can be separated to increase the depth of the crack. All of the merging of the crack-blocks into the rest of the mesh is done automatically.

A library of crack-blocks allows the user to select a suitable mesh density at the crack front for the problem in hand. This also facilitates mesh refinement studies in the local crack region by selection of a different crack-block.

The analysis of this type of mesh using Abaqus or Marc requires specification of data for energy release rate evaluations within the FE analysis. In the general 3D case this can be awkward but is automatically taken care of in this scheme.

Small 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 2. The key part of this process is the automatic updating of the mesh to model crack advancement and the re-submission of the updated mesh for the next FE analysis. The crack growth algorithm is general and caters for non-planar crack growth. An example of calculated crack growth profiles is shown in Figure 3.

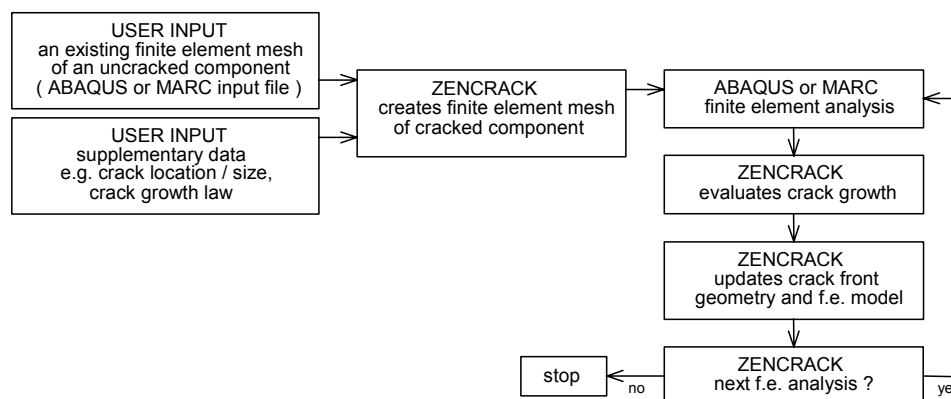


Figure 2 - Simplified flow chart for crack growth prediction analysis

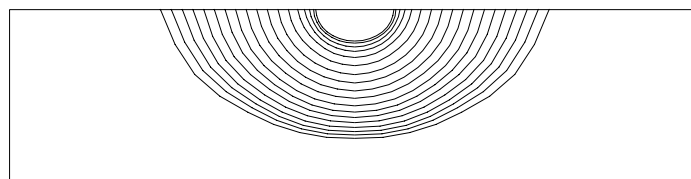


Figure 3 - Example of predicted fatigue crack growth profiles

The current scheme, originally set up for fatigue crack growth prediction, does have some limitations for crack growth calculation. The crack growth is limited to be contained within the original crack-block (a restart capability is available for growth outside this region and can be used in some cases). In general, the technique is limited to small crack growth although the crack itself may be long or deep.

Elastic, plastic, creep and thermal stress analysis

Using the methods described above Zentech has undertaken elastic, plastic, creep and thermal stress analysis of cracks in 3D FE models. The combination of the meshing scheme for the crack-blocks and finite element analysis in Abaqus or Marc gives the ability to select the appropriate type of crack front modelling depending upon the analysis to be carried out. For example:

- use of midside or quarter point nodes in collapsed crack front elements
- choice of full or reduced integration elements
- wide range of materials models

In theory the materials modelling in the FE analysis can be extended to just about any of the materials models available. The difficulty is not necessarily in doing the analysis but in understanding what it is you are trying to model and how to deal with the results.

Projects have been carried out on a variety of models including :

- Elastic and thermal stress analysis of a variety of crack configurations in meshes of 5000+ elements (e.g. Figure 4 shows a $\frac{1}{4}$ symmetry model containing a crack along the full length of the corner of the upper internal wall).
- Creep analysis of a fully embedded crack.
- Thermal stress analysis with two levels of submodelling in Abaqus and up to three distinct crack fronts (e.g. Figure 5 shows two submodels which were driven by a global model similar to the mesh shown in Figure 4. The second submodel includes detail of three separate crack fronts. The submodelling technique was necessary to reduce model sizes to manageable levels).

These techniques can be used in a wide range of industrial applications including:

- pressure vessels and piping
- shot peening (with effects of residual stress)
- turbine blades
- introduction of cracks into rubber components
- for sustained load crack growth prediction by analogy with fatigue crack growth prediction

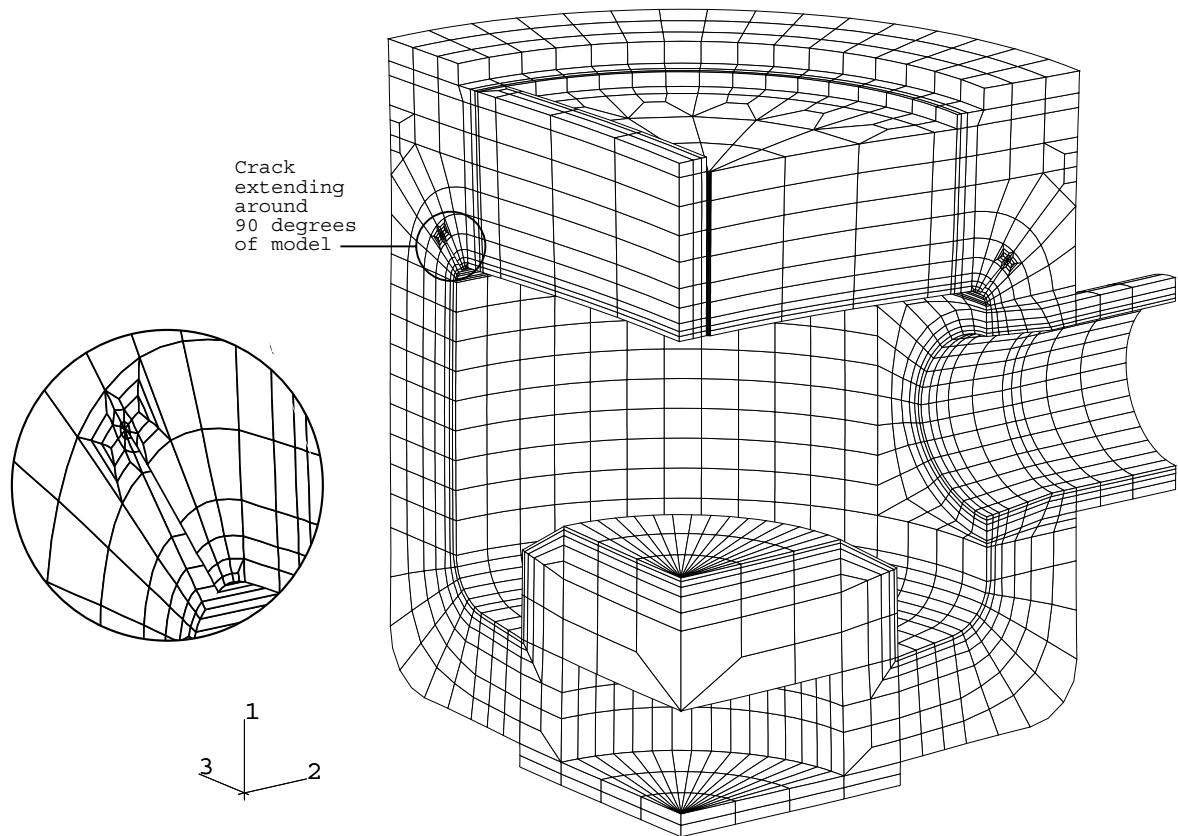


Figure 4 - Cracked 3D FE model with displacements magnified and close-up at one end of the crack front to show crack opening

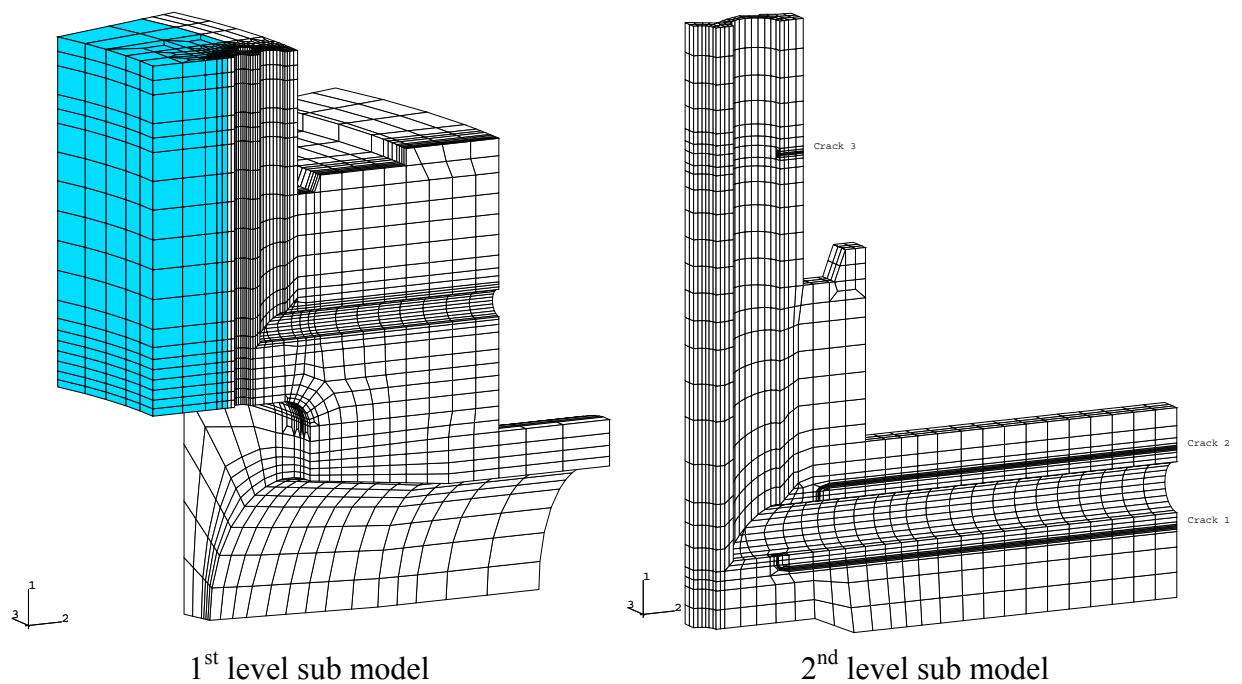


Figure 5 - Crack modelling around a hole

Future requirements

Powerful as they are, these techniques have restrictions. Difficulties may arise from many sources and can broadly be split into geometry and materials related problems although both may occur together.

Some of the key areas where there are gaps in current capabilities or new techniques are becoming practical are described briefly below.

Limited theoretical understanding

At the simplest level is the statement that if a process is not understood from a theoretical standpoint it cannot be modelled. Fracture mechanics has not yet produced an accepted description of creep crack growth, for example.

Crack tip plasticity and crack closure

Linear elastic fracture mechanics forms the basis for much work on fatigue crack growth prediction. Attempts have been made to incorporate effects of crack tip plasticity and crack closure by defining an effective stress intensity factor. Possibilities are however opening up for including these effects directly into analysis:

- development of rezoning techniques to maintain the plastic zone in the wake of the crack during crack advancement
- use of contact algorithms for including effects of crack closure

Large crack growth

The techniques described earlier are limited to small crack growth. A simple example of where this may be inadequate is a crack growing from an inclined notch in a plate under tension (Figure 6). In this example the amount of crack growth that can be calculated using the procedures described earlier (without using restart techniques) is limited to about 20% of the initial defect length. Schemes for large crack growth may depend upon better adaptive meshing or the use of submodelling techniques. Two difficulties that are immediately obvious are those of maintaining the geometric integrity of the FE model as the crack grows and updating boundary conditions.

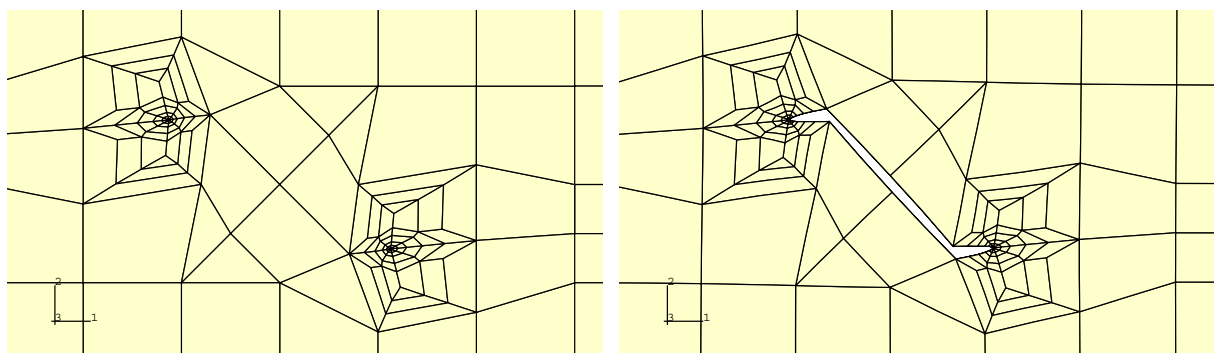


Figure 6 - Example of inclined crack modelling

Thin materials and stiffened plates

There are several issues to be considered here, including:

- large crack growth (again)
- through thickness modelling
- crack development at stiffeners
- ductile fracture
- use of R curves

Merging of two separate defects into one defect

One or more defects can readily be modelled, as shown in Figure 5. However, cases where two defects are close together and tend to join into one larger defect present difficulties. Two surface elliptic cracks may form one larger elliptic crack, for example. The situations before and after the merging of the separate cracks can be modelled, but the process of two crack fronts joining remains an area where engineering judgement is required.

Other areas

- Cracking driven by gas diffusion.
- Crack growth across material boundaries.
- Use of 8 noded bricks or 2D / shell elements in automated crack growth schemes.

Conclusions

Techniques are available which make many types of 3D fracture mechanics analyses possible using the finite element method. Analyses which were previously too difficult and expensive have become practical only to be replaced by an even more difficult set of problems.

Current meshing and analysis techniques can be extended to cater for many of these more difficult problems. In some cases, however, analysis is restricted by a lack of accepted theoretical models.

References

- Ref. 1 Zencrack software, Zentech International Limited, Surrey, U.K.
Ref. 2 Abaqus finite element program, H.K.S. Inc., Pawtucket, RI., U.S.A.
Ref. 3 Marc finite element program, Marc Analysis, Palo Alto, CA., U.S.A.

Acknowledgements

Figure 4 and Figure 5 are from consultancy projects and are used with client permission.