Using ABAQUS Cracktip Submodels to Investigate Cracking in Conventional Power Generation Plant

Andrew Morris and Tony Ing

E.ON Engineering and DAMT Ltd

Abstract: In the UK, conventional fossil fired plant is operated flexibly, some has now operated in excess of 5,000 starts and there has been an increase in the occurrence of crack initiation and propagation in a variety of components and from a variety of features.

Three previous papers by the same authors have been presented on this topic which detail results for straight and curved cracks in generator rotor teeth. ABAQUS submodelling is used with a 'simple cracked' C3D8 global model driving a C3D20 submodel containing the 'crack tip' mesh.

Rotor geometries are relatively easily meshed by sweeping, this paper extends the method to use a C3D10 tetrahedral global model applicable to conventional power generation plant. Tetrahedral mesh is controlled using the external face mesh with little control of the 'internal' mesh. In this paper the solid is partitioned to ensure a fine and even mesh around the crack tip and at the driven nodes.

The 'crack tip' submodel is made by sweeping and can be fully buried, it need not extend to the surface boundary, which can be problematic if this is not normal to the crack tip. The global model can contain many cracks and the same cracktip submodel can simply be repositioned to extract crack output at as many crack locations and directions as required.

The complexity of traditional crack modelling has restricted its use to fairly simple geometry, this method however enables cracks to be effectively modelled in any general solid.

Keywords: Crack Propagation, Crack Modelling, Crack Submodel, Fracture, Power Plant, Generator rotor, Remaining Life

1. Introduction

E.ON Engineering is part of the E.ON Group and centre of excellence for engineering and scientific support. In addition E.ON Engineering, is a leading supplier of independent technical services to the power sector (coal, gas and oil fired plant) and major energy users, providing technical support to the majority of the fossil fired power stations in the UK and internationally.

E.ON UK is one of the major power generating utilities in the UK. It has a domestic operating capacity of approximately 10,900MW, which includes a mix of coal, gas, combined heat and power and renewable plant.

In the UK, conventional fossil fired plant is operated flexibly, dependent upon commercial market conditions. Consequently some of the plant has now operated in excess of 5,000 starts. During periodic inspections of the plant and rotating components cracks are sometimes identified and the severity of these cracks needs to be evaluated see Figure 1.

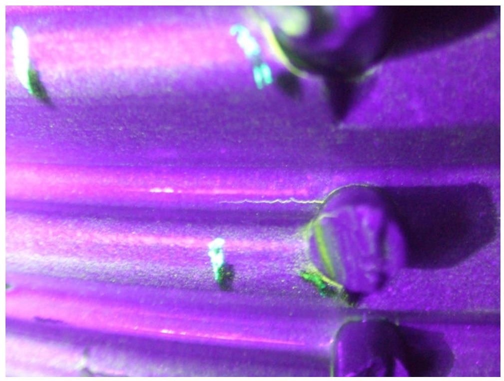

Figure 1. Crack identified using fluorescent magnetic particle inspection.

ABAQUS Linear Elastic Fracture Mechanics (LEFM) capabilities have been used to establish the Stress Intensity Factors (SIF) at the crack tip due to the applied thermal and/or pressure and/or rotational loadings. The SIF's can then be used to establish the crack growth rates and hence the severity of the crack and the remaining life of the component "(Morris, 2006)", "(Morris, 2007)" and "(Morris, 2007)".

2. Modelling the cracktip singularity in 2D

In 2D LEFM analysis the stress at a distance $\mathbf{r}$ from the crack tip is a square root singularity i.e.

$$\sigma \propto 1/\sqrt{r}$$

This stress distribution can be created in ABAQUS using most of the standard 8-noded quadrilateral 2D elements.

Figure 2 shows the node movements required for an element to model a square root singularity. In this example the collapsed edge nodes 1859, 1858 and 1857 are all moved to the cracktip location and the mid-side nodes 36,488 and 36,487 are moved to the ¼ point nearest the cracktip.

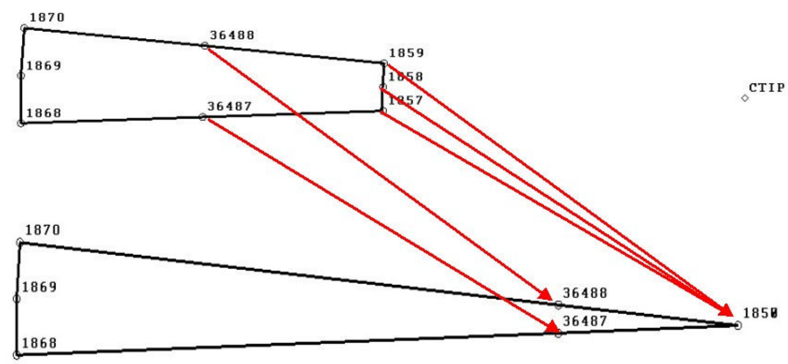

Figure 2. Shows the node movements required for a 2D element to model a square root singularity.

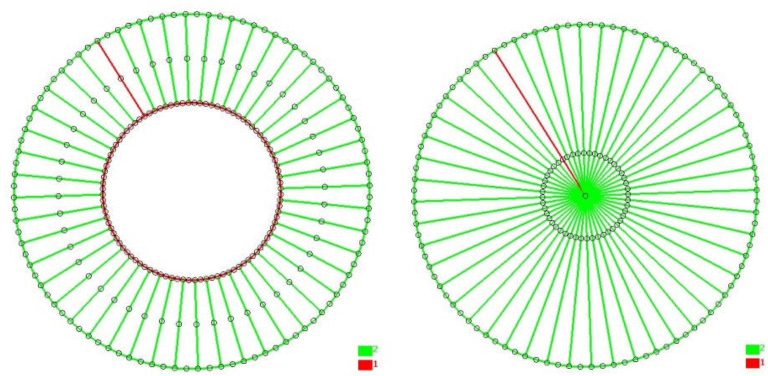

Figure 3. Shows the node movements required for all the 2D elements surrounding the cracktip.

Figure 3 shows the node movements required for all the elements surrounding the cracktip. Note that any pre-processor can make the simple cracktip topology and it is only the node coordinates that need to be modified. In ABAQUS the *NODE keyword can be used to reposition an existing node if required.

To create a square root singularity additionally all the nodes at the cracktip need to be constrained to move together. The easiest way of achieving this is to use the *MPC keyword with the TIE option.

It is therefore a fairly simple task to process a 2D cracktip topology modelled in any pre processor and transform it into the correct 2D geometry and constraints for a square root singularity. By using pre named node sets, element sets etc. this can be achieved at the ABAQUS .inp file level.

3. Outputting the SIF's in 2D

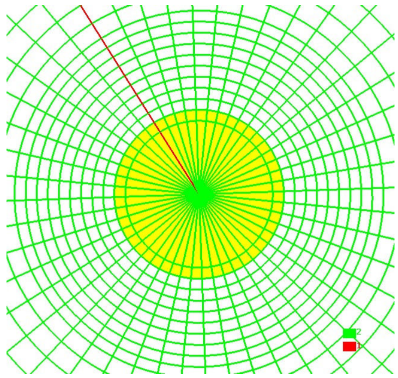

Figure 4. Shows the element contours required around the crack tip.

The 8 noded quadrilateral elements around the cracktip have to be modelled as concentric rings of elements or contours. In Figure 4 the crack tip and the first ring of elements, shown in yellow, make up contour # 1, with contour # 2 outwards being made up of increasing diameter rings of single elements.

SIF output in 2D i.e. K1 (opening), K2 (in plane sliding) and MTS (Maximum Tangential Stress) angle is then obtained using the *CONTOUR INTEGRAL output keyword, the number of contours required, the set of nodes (TIPN say) at the cracktip and the direction cosines of the assumed crack extension direction, this is a results processing request, e.g. for the above mesh.

```
*CONTOUR INTEGRAL, CONTOURS = 5, TYPE = K FACTORS

TIPN, 1.6 , -5.2
```

4. Including the 2D Cracktip mesh into a general 2D model

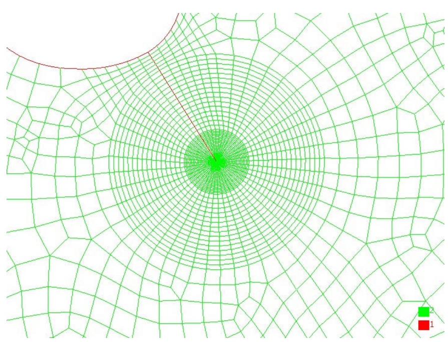

Figure 5. Shows the inclusion of a cracktip mesh in a quad8 auto paved mesh.

In most pre processors the 8 noded quadrilateral cracktip mesh can be included in a general 2D model using a mixture of auto paved and ruled meshing see Figure 5. This is the method used for most 2D cracktip models but the extension into 3D is limited to extruded mesh since general 3D hex auto meshers are not available.

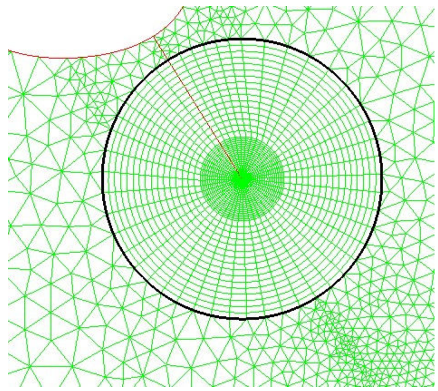

Figure 6. Shows the inclusion of a cracktip mesh in a auto tri6 mesh.

The 8 noded quadrilateral cracktip mesh can also be included in a general 2D model using a mixture of auto tri6 mesh and ruled quad8 mesh see Figure 6. This requires that the boundary between the tri6 and quad8 elements, shown in black, be joined e.g. by common nodes or by a TIED contact. Figure 7 compares the maximum principal stress P1 stress between a quad8 and a TIED quad8/tri6 model. This method has the advantage that it can be extended to general 3D models since general 3D tet10 auto meshers are available in most pre processors.

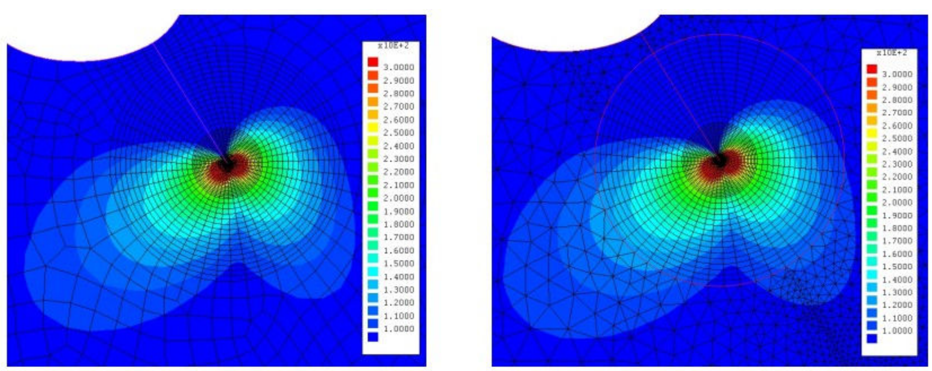

Figure 7. Compares the P1 stress between a quad8 and a TIED quad8/tri6 model.

A cracktip can also be included in an auto tri6 mesh as a simple stress singularity, away from the crack tip the stresses and displacements will be similar to those obtained using a square root singularity. See Figure 8.

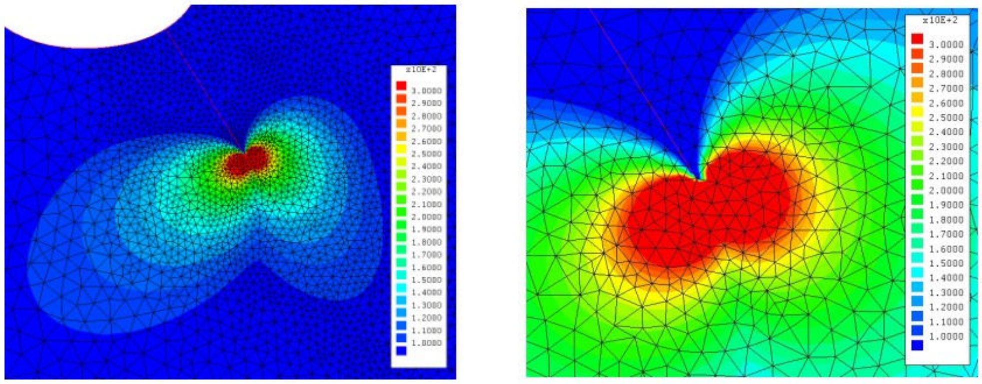

Figure 8. Shows the P1 stress on a tri6 model.

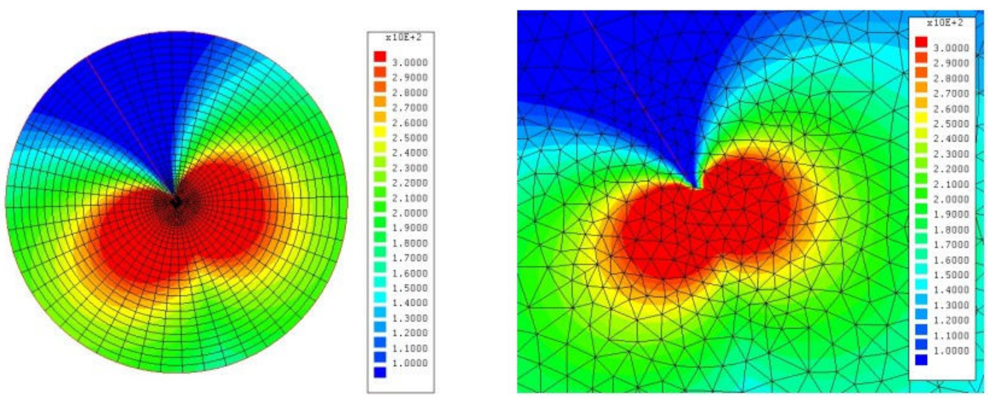

Figure 9. Compares the P1 stress on a cracktip quad8 submodel with the global tri6 model.

Extracting the SIF's from a tri6 model can then be achieved using the ABAQUS node based submodelling technique. The displacements in the global tri6 model shown in figure 8 are used to drive a quad8 cracktip submodel see Figure 9. Note how the witness mark of submodel boundary in the global model mesh helps to compare maximum principal stress P1 stress contours.

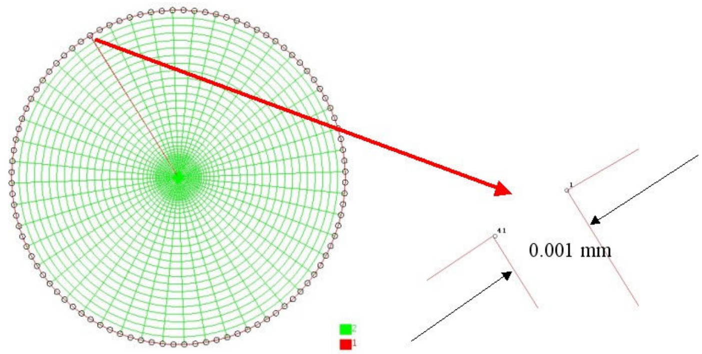

Figure 10. Shows the cracktip submodel mesh and driven boundary nodes.

The cracktip submodel is driven by the set of boundary nodes shown in Figure 10. Since the crack faces are coincident in the tri6 global model a gap, in this case 0.001 mm, is introduced between the crack faces on the submodel boundary. This ensures that the correct crack face displacement is picked up from the global model.

Defining the submodel boundary as node set SUBM the submodel is driven using:

```
*SUBMODEL, ABSOLUTE EXTERIOR TOLERANCE = 0.0005

SUBM

*STEP

*STATIC

*BOUNDARY, SUBMODEL, STEP = 1

SUBM, 1, 2
```

Note the global model .fil or .odb file to be used in a submodel analysis is supplied on the ABAQUS execution command line.

Auto tri6 meshers typically use the face edge density to define the mesh and this means the user has little control over mesh inside the face. Faces however can be subdivided to ensure a suitable mesh is achieved in the area of the driven nodes. This creates the witness mark of the submodel boundary on the global model as seen in Figure 9. This witness mesh also allows easy comparison of results between the global model and the submodel.

This method also has the advantage that a global model could contain many cracks modelled as simple singularities and the submodels would basically be the same model moved and possibly rotated to various locations.

5. K Factor estimates for various modelling methods

ABAQUS K FACTOR ESTIMATES FOR CONTOUR #5

		CAX8 PAVED	CAX6/CAX8 TIED CONTACT	CAX6/CAX8 COMMON NODES	CAX6 GLOBAL MODEL CAX8 SUBMODEL
SIF - K1	MPa√m	11.865	11.865	11.865	11.735
SIF - K2	MPa√m	-0.506	-0.507	-0.507	-0.476
MTS	deg	4.865	4.872	4.872	4.798

Table 1. Compares ABAQUS K Factors for the various modelling methods.

Table 1 details the ABAQUS K factor estimates for the various 2D modelling methods discussed. All the modelling methods provide similar results and the one chosen would probably depend on the capabilities of the pre processor. The submodel method would certainly be simpler in a multi cracked structure.

6. Modelling the cracktip singularity in 3D

In 3D LEFM analysis the stress at a perpendicular distance **r** from the crack front is a square root singularity i.e.

$$\sigma \propto 1/\sqrt{r}$$

This stress distribution can be created in ABAQUS using most of the standard 20-noded hexahedral 3D elements.

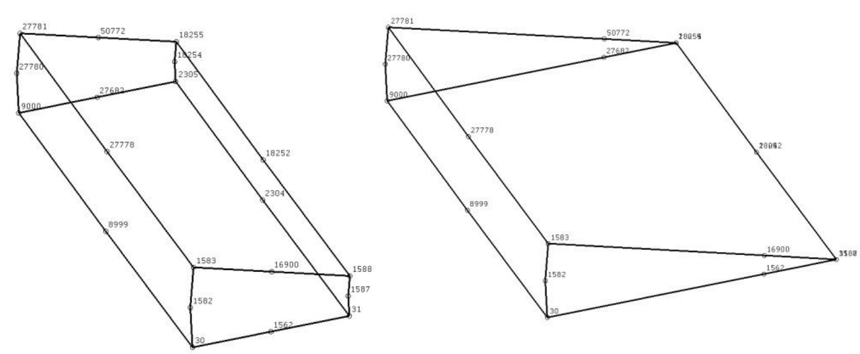

Figure 11. Shows the node movements required for a 3D element to model a square root singularity.

Figure 11 shows the node movements required for an element to model a square root singularity in 3D. This is the logical extension to the movements detailed in 2D Figure 2.

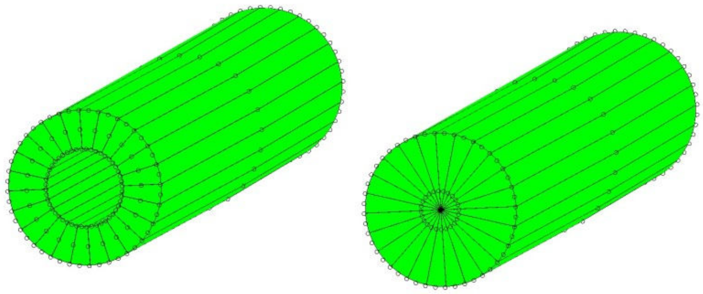

Figure 12. Shows the node movements required for all the 3D elements surrounding the cracktip.

Figure 12 shows the node movements required for all the elements surrounding the crack front. To create a square root singularity additionally all the coincident nodes along the crackfront need to be constrained to move together. In this example there will be three node sets and the easiest way of achieving this is to use the *MPC keyword with the TIE option.

It is therefore a fairly simple task to process a 3D cracktip topology modelled in any pre processor and transform it into the correct 3D geometry and constraints for a square root singularity. By using pre named node sets, element sets etc. this can be achieved at the ABAQUS .inp file level.

7. Outputting the SIF's in 3D

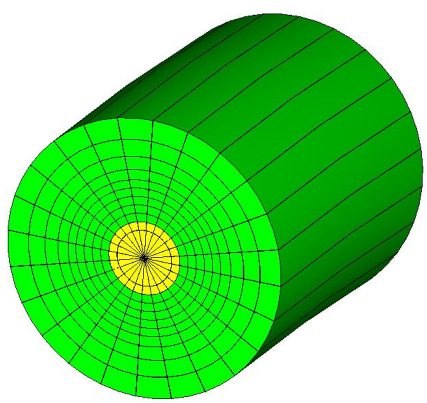

Figure 13. Shows the element contours required around the crack tip.

The 20 noded hexahedal elements around the cracktip have to be modelled as concentric cylinders of elements or contours. In Figure 13 the crack tip and the first cylinder of elements, shown in yellow, make up contour # 1, with contour # 2 outwards being made up of increasing diameter cylinders of single elements.

SIF output in 3D i.e. K1 (opening), K2 (in plane sliding), K3 (out of plane tearing) and Maximum Tangential Stress MTS angle is then obtained using the *CONTOUR INTEGRAL output keyword, the number of contours required, the coincident sets of nodes at the cracktip, these must be ordered along the crack front, and the direction cosines of the assumed crack extension direction for each node set e.g. for the above mesh with node sets T1, T2 and T3.

```
*CONTOUR INTEGRAL, CONTOURS=5, TYPE=K FACTORS
T1    ,      0.000,    -48.035,    -63.397,
T2    ,      0.000,    -47.554,    -63.758,
T3    ,      0.000,    -47.071,    -64.116,
```

Note that this is a results processing request only and that to achieve stable SIF values the sections through a cracktip mesh should be planar and also finer at the start and end of the crack front, this effectively limits a crack mesh to swept/extruded meshes or combinations of swept/extruded meshes.

8. Including the 3D Cracktip mesh into a general 3D model

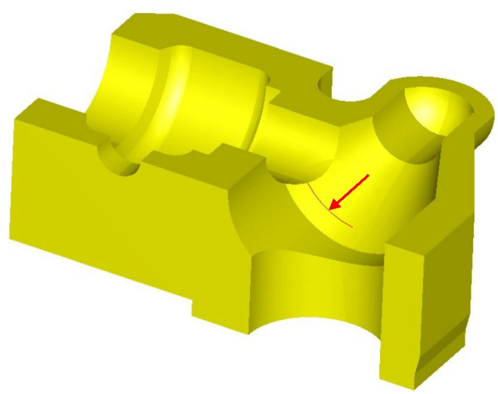

Figure 14. Shows the location of a circular crack in part of a steamchest.

Fig 14 shows part of a typical power generation plant component with an observed crack location. The component is too difficult to economically hex mesh so we need to consider an auto tet mesh based solution. Including a C3D20 cracktip region explicitly in the model and using TIED contact to connect it to a C3D10 region is a geometrically complex modelling task.

The cracktip submodel approach simplifies the geometric modelling considerably i.e. the global model is made up of auto meshed C3D10 elements and the submodel is made up of swept C3D20 elements and this is the method described here.

9. Including a simple crack singularity into the global model

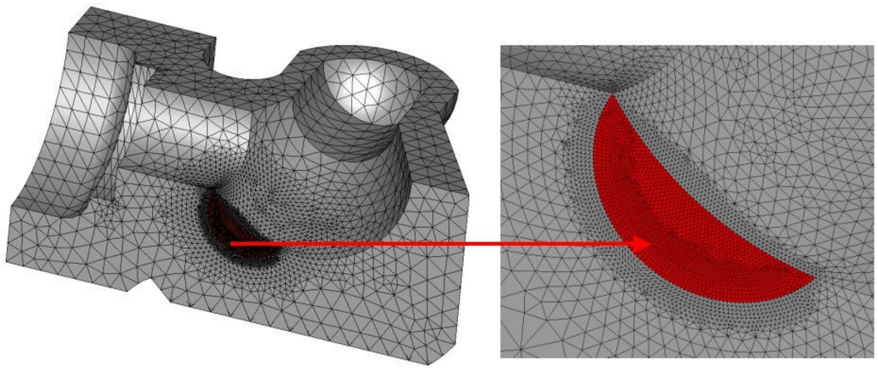

Figure 15. Shows a section through the global model at the crack.

Including a simple crack singularity into the global model is achieved by splitting the relevant solid/solids in the crack direction, defining the required crack profile on the cut section, in this case part circular, and then merging selected geometry to achieve the crack envelope shown in red in Figure 15.

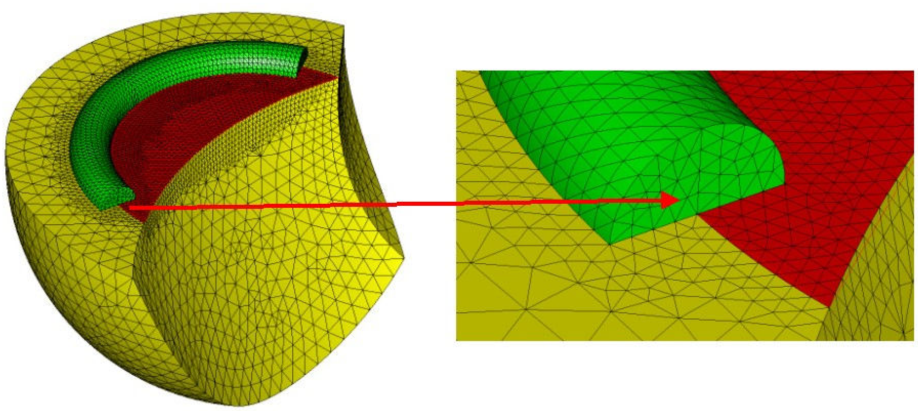

Figure 16. Shows the C3D10 mesh in the region of the driven nodes.

This mesh also shows that by adding more control geometry a mesh transition from coarse to fine at the cracktip driven nodes can be achieved. The witness of the cracktip submodel is shown in green in Figure 16 is used to control the driven node density and for verifying the submodelling analysis. Note that for this crack profile the submodel has been fully buried since it can not extend to the steamchest surface boundary with the crack sections remaining planar.

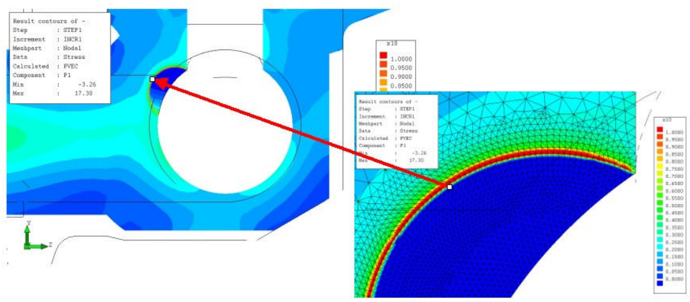

Figure 17. Shows the P1 stress on a section through the global model on the crack plane.

The crack modelling is verified by cutting a section through the global model on the crack plane and viewing the P1 stress see Figure 17.

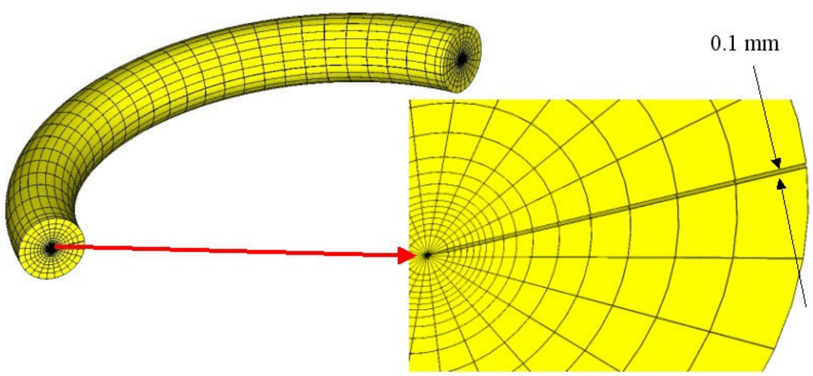

Figure 18. Shows the cracktip submodel.

The cracktip submodel is driven by the set of boundary nodes on the external free faces i.e. the torus surfaces and the end caps shown in Figure 18. Since the crack faces are coincident in the tet10 global model a gap, 0.1 mm in this example, is introduced between the crack faces on the submodel boundary only i.e. internal submodel crack faces remain coincident. This ensures that the correct crack face displacement is picked up from the global model.

Defining the submodel boundary as node set SUBM the submodel is driven using:

```
*SUBMODEL, ABSOLUTE EXTERIOR TOLERANCE = 0.05

SUBM

*STEP

*STATIC

*BOUNDARY, SUBMODEL, STEP=1

SUBM, 1, 3
```

Note that any loadings applied in the global model in the region of the submodel have to be applied to the submodel and in this case a pressure loading has to be applied to the submodel crack faces.

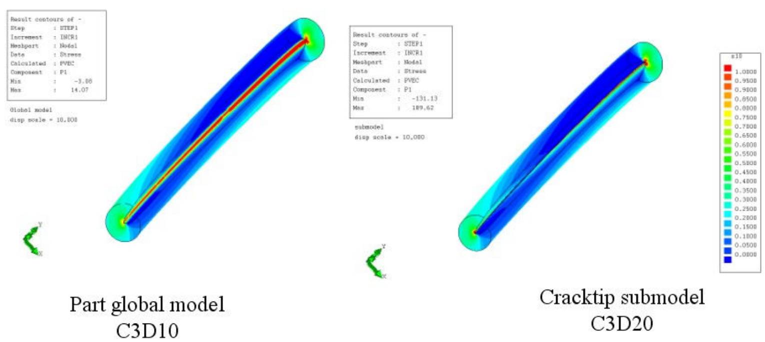

Figure 19. Compares global and submodel P1 stresses with a scaled deformation applied.

Displacements and stresses in the global model witness region can be readily compared with those of the submodel to verify the submodelling technique is working correctly. Figure 19 verifies the model by comparing global and submodel P1 stresses with a scaled deformation applied.

SIF data will be output by ABAQUS for every node set supplied along the crack front, 65 sets in this example, and for the number of contours requested, contours values above #2 and values between node sets should vary smoothly. Figure 20. Shows the variation in K1, K2, K3 and MTS at contour 5 along the crack front. Note that all the curves are smooth and that the mesh at the ends of the submodel has been refined in Figure 18 to allow the end effects to settle as quickly as possible. In this example the submodel is fully buried and spans most of the crack front i.e. 140 deg. Note that as long as the end effects settle then a reduced submodel, say 17.5 deg, could be used to extract SIF data at a specific location on the crack front, this makes the submodel geometry simpler for more complex crack front geometries.

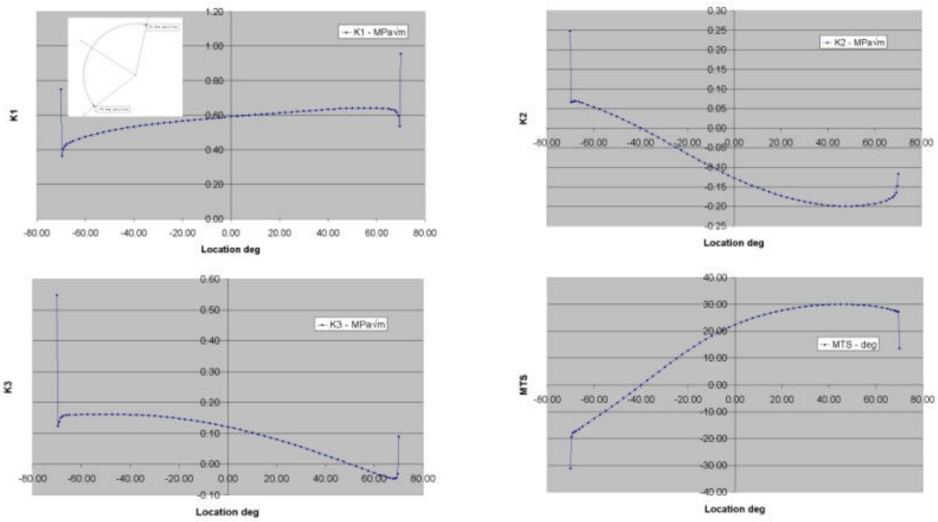

Figure 20. Shows the variation in K1, K2, K3 and MTS at contour 5 along the crack front.

10. Using SIF data

In static LEFM the SIF associated with the fracture toughness of a material is called the critical SIF – *Kc*, which is obtained experimentally, and for a crack to be stable under static opening loading K1 should be less that *Kc*.

In more general cyclic or transient loading we need to consider the SIF ranges i.e.

$$\Delta K_I = K1_{max} - K1_{min} \qquad \Delta K_{II} = K2_{max} - K2_{min} \qquad \Delta K_{III} = K3_{max} - K3_{min}$$

These ranges can be reduced to an equivalent mixed-mode cyclic stress intensity factor e.g. $\Delta K_{eq}^{(MS)}$, "(Marquis, 2003)" is defined as:

$$\Delta K_{eq}^{(MS)} = [\Delta K_I^2 + \Delta K_{II}^2 + (1+\nu)\Delta K_{III}^2]^{0.5}$$

Where ν is Poisson's ratio

It is generally accepted that ΔK_{eq} can then be used in the conventional Paris fatigue crack growth equation to establish the crack growth rate as follows

$$da/dN = C(\Delta K_{eq})^m$$

Typically $C=10^{-12}$ m/cycle and m=3, with ΔK_{eq} measured in MPa √m "(Brown1989)".

11. Conclusions

This paper has detailed the model changes required around a crack to create a LEFM square root singularity in both 2D and 3D and how to output SIF contour data in ABAQUS. It has detailed the ways in which a cracktip mesh can be included in both 2D and 3D models. It proposes that a tet10 meshed global model driving a hex20 cracktip submodel enables cracks to be effectively modelled in any general solid.

The complexity of traditional crack modelling has restricted its use to fairly simple geometry, this 'cracktip submodel method' will allow E.ON Engineering Technology Centre perform crack growth assessments on a wider range of plant and machinery than has previously been practical.

12. References

Brown, Gladwin, Priest 1989. ;The effect of torsion on the mode I fatigue response of a 3 1/2 NiCrMoV rotor steel, National Power - Nuclear, Technology Division Report TD/STM/89/10049/N, November 1989

Marquis, Socie 2003. 'Multiaxial Fatigue', in Comprehensive Structural Integrity, Vol. 4 (Cyclic Loading and Fatigue), Elsevier, Oxford, U.K. (Edited by I. Milne, R. O. Ritchie and B. L. Karihaloo), Elsevier, 221-252.

Morris, Ing, Johnson 2006 - EPRI - Louisville, Kentucky, October 16-18, 2006CRACKING IN 500MW GENERATOR ROTORS - RECENT UK EXPERIENCES.

Morris, Ing, Johnson 2007 - NAFEMS World Congress - Vancouver, May 22 to 25, 2007 FINGER AND POLE TOOTH CRACKING IN LARGE GENERATOR ROTORS - A CASE STUDY

Morris, Ing, Johnson 2007 ABAQUS UK Regional User Meeting, November 14 and15, 2007 USING ABAQUS CRACKTIP SUBMODELS TO INVESTIGATE FINGER TOOTH CRACKING IN LARGE 500MW GENERATOR ROTORS

13. Acknowledgement

The author would like to acknowledge the support of his co-author in preparation of this paper, and E.ON Engineering, Technology Centre for permission to publish this work.