A numerical fracture mechanics tool to help assess the structural integrity of nuclear power plant components

Chris Timbrell [a,*], Ramesh Chandwani [a], Angelo Maligno [a], Charly Ma [b]

[a] *Zentech International Limited, 590b Finchley Road, London, NW11 7RX, UK*

[b] *Consys Company Limited, Room 504 Liqin Plaza, No 1885 Duhui Road, Minhang District, Shanghai P.R.C 201108, China*

Abstract

Many nuclear power plant facilities have been operating for longer than twenty years. Along with ageing of structural materials come other issues, such as creep and corrosion, which can impact upon plant safety and the integrity of the primary circuit. Plant lifetime extension and continued safe and economic operation depends on ageing and lifetime management. To be effective, this requires an understanding of how safety may be maintained as components degrade over extended time periods under operational conditions. Potential structural integrity issues for future generation IV reactors are also paramount to further develop the nuclear industry.

This paper describes some aspects of the numerical analysis tool, Zencrack, which can assist in crack growth prediction and fitness for service investigations for a range of nuclear power plant applications.

Keywords: fad; structural integrity; fatigue crack growth; finite element analysis; residual life; crack driving force; fitness for service

1. Introduction

To provide a reliable assessment of structural integrity for existing and future nuclear power plants, it is necessary to cover several engineering aspects related to crack initiation and growth. For example:

- Environmentally-assisted cracking and thermal fatigue initiation and crack growth behaviour in nuclear reactor structural materials.
- Stress corrosion cracking, initiation and propagation in stainless steel components.
- Fatigue crack growth development under complex thermal and mechanical load cycles.
- High temperature creep effects on fatigue crack growth.

To investigate fracture mechanics problems in nuclear power plant components, Zencrack [1] represents a reliable and invaluable numerical tool. The software offers the following capabilities which are crucial for a realistic and trustworthy evaluation of structural integrity in components working in harsh environments:

- Evaluation of fracture mechanics parameters (e.g. stress intensity factors, energy release rates) under linear and non-linear material behaviour and in complex geometry.
- Three-dimensional (3D) modelling and prediction of fatigue crack growth in structural components under complex thermal mixing situations with cyclic thermal shock during fuel loading phase and also to simulate start up and shut-down scenarios.
- 3D modelling and prediction of fatigue crack growth from the micro-scale e.g.: micro-inclusions, voids (multi-scale modelling approach).
- 3D modelling and prediction of fatigue crack growth taking account of plasticity and creep.
- 3D modelling and prediction of fatigue crack under mixed mode loading conditions.

The application of Zencrack to a level 3 assessment according to the API/ASME Fitness For Service code [2,3] is used to demonstrate a typical application of the software for the nuclear industry. Some other application examples are then briefly described.

*Corresponding author.

E-mail address: chris@zentech.co.uk.

2. Level 3 fitness for service assessment

The API/ASME Fitness For Service code [2,3] describes, in section 9, several levels of assessment for components containing crack-like flaws. The assessments are based on the use of failure assessment diagrams (FADs). The most onerous of the levels is the level 3 assessment as described in paragraph 9.4.4.1 [2]. This is further sub-divided into assessment methods A to E. Of these, method C is considered in more depth here.

The level 3 method C assessment follows the level 2 assessment procedure with the difference that the FAD is generated based on the actual loading conditions, component geometry, and material properties. A procedure to construct this FAD, and to complete the assessment for a known crack-like flaw is described in Annex B1, paragraph B1.7.4 [2]. The procedure requires the use of numerical analysis techniques in the form of finite element analysis (FEA). Paragraph B1.7.3 [2] provides some general guidelines for the use of finite element analysis for crack-like flaws. Paragraph B1.7.5 [2] describes an alternative driving force method for non-growing cracks. This also requires finite element analysis.

Although the FAD approach can handle multiple primary stresses, secondary and residual stresses, it suffers from some limitations. In fact, even if a complex elastic-plastic finite element solution is undertaken for the configuration of interest, the FAD approach requires stress classification (i.e. only the primary stresses should be included in the analysis). Moreover, when multiple primary loads are present, they are assumed to increase and decrease in phase with one another. In practical applications this classification may be difficult and the driving force method provides an alternative approach in which stress classification is unnecessary. The method is sufficiently general to be able to handle any load history.

3. Zencrack methodology

Due to space restrictions, only a very brief description is given here of the Zencrack software and the way in which it interfaces to Abaqus [4] and Ansys [5]. More information can be obtained, for example, in [1,6].

The Zencrack software places one or more user specified cracks into an uncracked mesh. This process operates a 3D finite element mesh which must contain 8 or 20 noded brick elements in the crack region (with other element types elsewhere if required). The procedure is a replacement of one or more elements of the uncracked mesh with "crack-blocks" containing the detailed rings of elements required around a crack front. This process includes any necessary boundary conditions, pressure load and temperature distribution updates for the new elements and nodes in the crack region.

The cracked mesh is then analysed using the interfaced finite element code. Contour integral evaluation in the interfaced finite element code is used as one means of calculating fracture mechanics parameters. The finite element analysis may be linear or non-linear.

Zencrack has a generalized algorithm for 3D non-planar crack growth prediction due to fatigue loading or sustained loading. The crack is advanced, the mesh updated and a further finite element analysis is carried out. This procedure repeats, as shown schematically in Fig. 1.

The crack growth capability can cater for complex load cycles and temperature dependency in materials data and the load history.

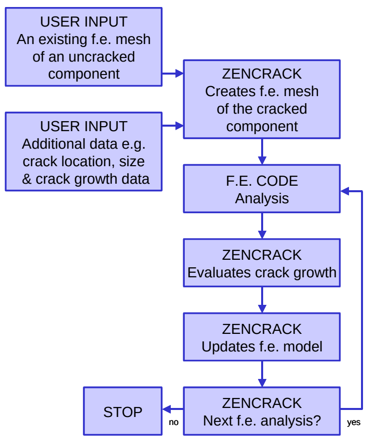

Fig. 1. Zencrack analysis procedure.

4. Use of Zencrack in a level 3C assessment

The steps required to generate the FAD for a level 3C assessment can be summarized as follows:

- Identify the primary loading and apply it to a suitable finite element model of the component

- Undertake a linear elastic FEA and determine the J-Integral values: J_{el}.
- Undertake an elastic-plastic FEA and determine the J-Integral values as a function of load, P: J_{elpl}
- Determine the reference load of plastic collapse (P_{ref})
- Draw the failure assessment diagram using points (K_r,L_r) where:

  $K_r = (J_{el}/J_{elpl})^{½}$ $\quad L_r = P / P_{ref}$

As can be seen from these requirements, the process requires an elastic and elastic-plastic solution. Although the required elastic data can almost certainly be extracted from a very low load level of the elastic-plastic analysis, in practice it is better to use a separate elastic analysis. This allows full verification of the loading and boundary conditions before commencing the non-linear analysis.

One of the difficulties associated with this type of analysis for “real” 3D structures is referred to in section 1.7.3 of the code [2]:

“… constructing meshes such as depicted in Figure B1.5 can be extremely cumbersome and time consuming. It is recommended that the analyst develop or acquire mesh generation software for this purpose.”

Zencrack is such software and provides not only help in mesh generation for 3D cracks, but also automatic processing of the fracture mechanics parameters resulting from elastic or elastic-plastic finite element analysis.

For a general 3D crack front, a separate failure assessment diagram can be constructed for any position along the crack. Therefore, the analyst must make some decision about how many such points should be assessed. Zencrack provides help in respect of the post-processing of the finite element results by providing results not only at the node positions along the crack front, but also by including an option to interpolate the results to any parametric position along the crack.

5. Finite element meshing for a level 3C assessment

A cracked nozzle is used to demonstrate the use of Zencrack for a level 3C assessment. The model is defined in section 9.10 of [7], along with a detailed description of the procedure for the assessment. Here we concentrate on the meshing problem and subsequent changes to the mesh that is generated.

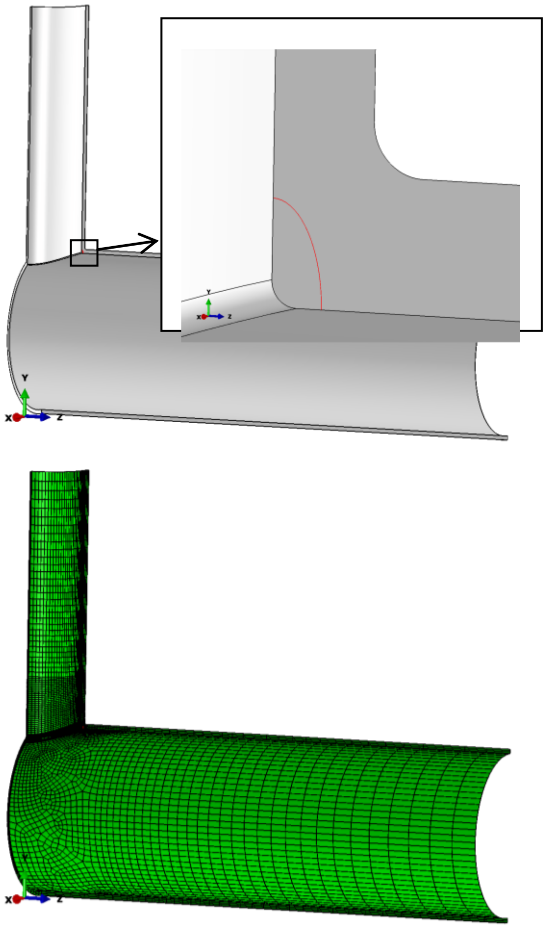

Fig. 2. Crack location and uncracked mesh.

The component consists of a 25mm thick main pipe with OD 1000mm. The nozzle is 20mm thick with OD 500mm. Fillet radii of 10mm and 5mm exist between the pipe and nozzle on the outside and inside respectively. The crack is an internal 20mmx10mm corner elliptic crack as shown on the quarter symmetry geometry model of Fig. 2, which also shows an uncracked mesh consisting of 20 noded elements.

The applied load is an internal pressure with end cap effect on the pipe and nozzle. The pressure load is also applied on the crack face.

A possible uncracked mesh in the crack

region and the cracked mesh generated by Zencrack are shown in Fig. 3, in which the crack front position is indicated by the red line. Within this mesh generation process, the user may specify the type of tip model along the crack front. Two tip models are generally used, as discussed in section B1.7.3a)1) of [2]:

- Single node at each crack front position, radial midside nodes moved to the quarter point positions – used for linear elastic analysis ($r^{-½}$ singularity).
- Multiple nodes at each crack front position allowing the crack front to open up under load – often used for elastic-plastic analysis (r^{-1} singularity).

The former is the default option is Zencrack but the latter is simply requested with one option in the Zencrack input: TIP MODEL, EPFM. This allows elastic and elastic-plastic analysis of the cracked component to be carried out using the same uncracked mesh, subject to a small number of input changes:

- In the Zencrack input file the type of crack tip model can be changed by using TIP MODEL, EPFM.
- In the uncracked mesh the non-linear material data is added and the load step controls are changed to be suitable for the gradual application of load.

If a more detailed mesh is required, then the re-meshing process has two possibilities. If more detail is required only in the crack region, a different crack-block may be available with more elements and nodes at the crack front (e.g. more rings of element around the crack front). Then only a change in the Zencrack input file is necessary.

In other cases, re-meshing of the uncracked model may be necessary. Such an example is shown in Fig. 4; in which the general element density is increased and there are 15 elements along the crack front compared to 9 in the original model. Starting from the model in Fig. 3, about 15 minutes is required to obtain a completed (elastic) crack mesh analysis of the mesh in Fig. 4. The changes required are re-seeding of key edges in the uncracked model and re-meshing (here using Abaqus/CAE), and re-definition of a small amount of Zencrack input data.

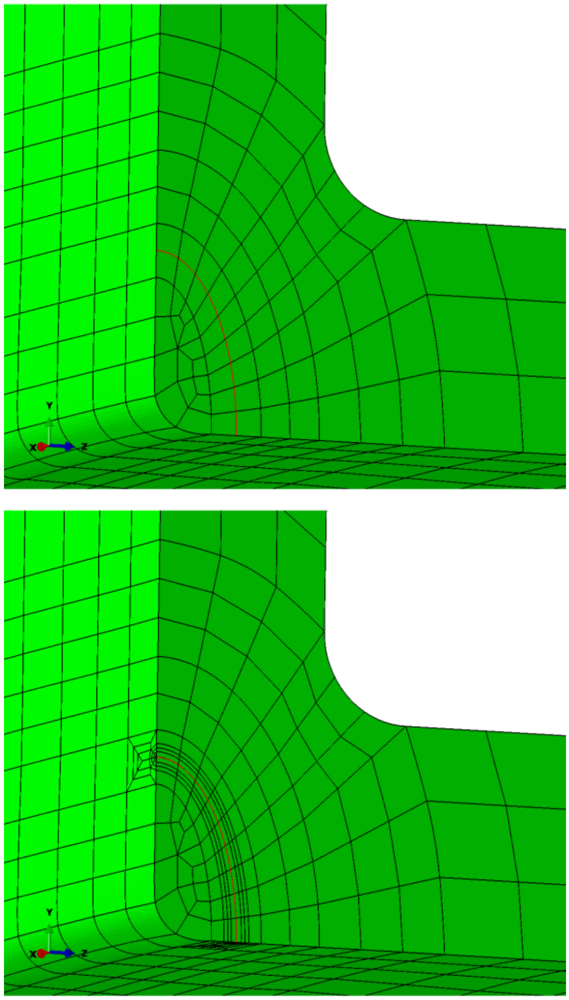

Fig. 3. Uncracked mesh and the cracked mesh generated by Zencrack.

The level of simplification required in the model is one which can also quickly be investigated via re-meshing. For example, the analysis in [7] does not include the fillet radius on the inside of the nozzle. Again, a 15 minute re-meshing exercise from the mesh of Fig. 3 to that of Fig. 5 can produce a cracked model analysis in which the fillet radius is not included (in fact a very small radius of 0.1mm has been included).

This type of investigation may be feasible due to the removal of the time consuming manual re-meshing of the details of the crack front.

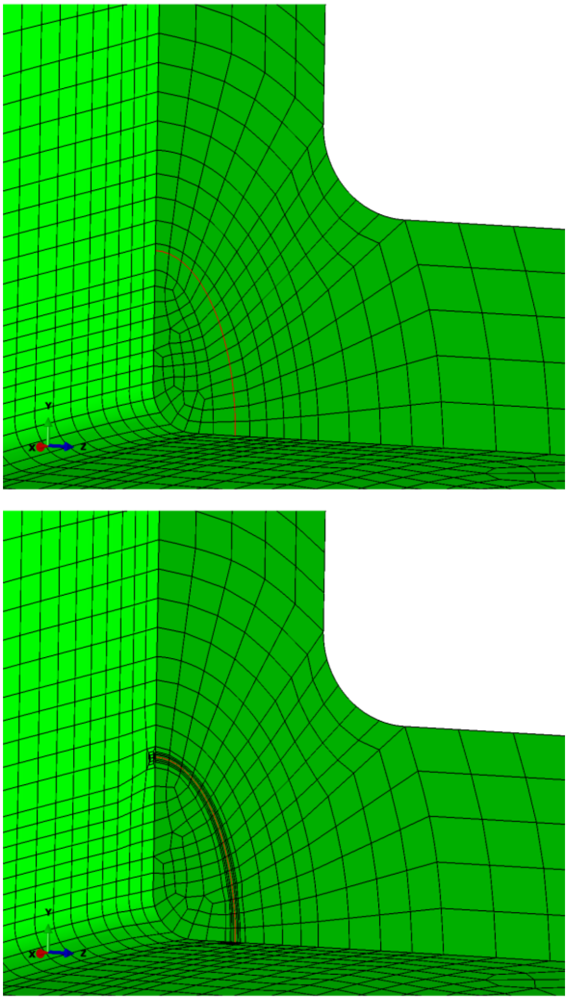

Fig. 4. Refined uncracked mesh and the cracked mesh generated by Zencrack.

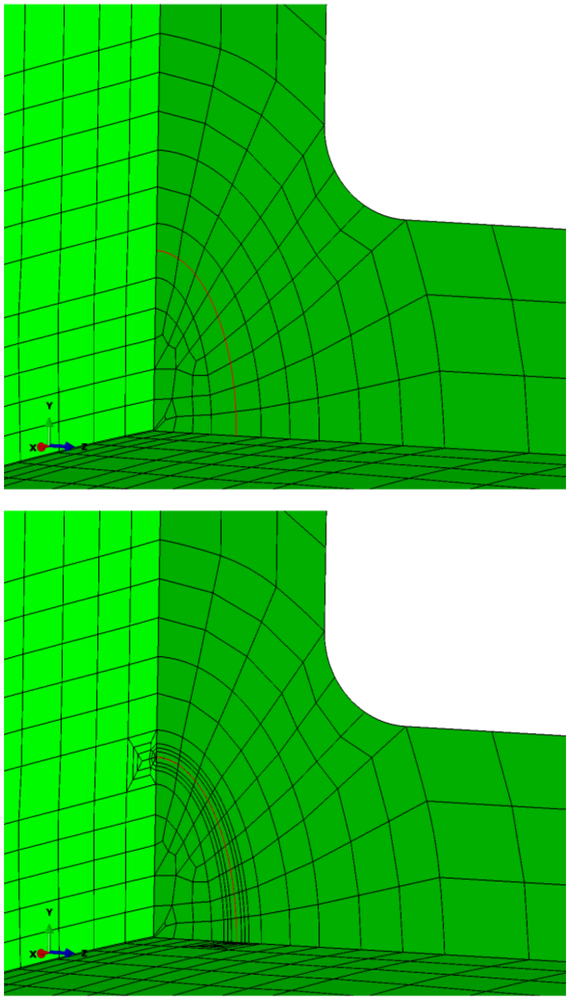

Fig. 5. Uncracked mesh modified to remove the internal fillet radius and cracked mesh generated by Zencrack.

The uncracked mesh can also be used to analyse smaller or larger cracks. Depending upon this requirement, some additional care may be required in creating the uncracked mesh so that it is suitable for re-use for different crack sizes.

The uncracked mesh can also be used if crack growth prediction is required. An advanced crack position from such an analysis is shown in Fig. 6, in which the crack is has grown to a more circular type of shape. There is no assumption about the shape development of the crack – it is merely a consequence of the loading, geometry and crack growth data.

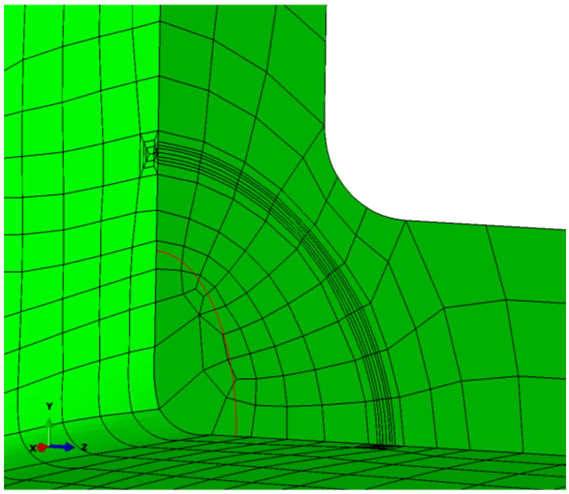

Fig. 6. A cracked mesh during a growth analysis.

6. Other practical applications

The previous section demonstrates how the meshing capabilities in Zencrack can help speed up the process of crack modelling. There is also significant time-saving during post-processing.

However, this application is just one example of use of the software within the nuclear industry. The generalized approach to loading and the ability to cater for temperature dependent materials and non-linearities means that there are many possible applications. These include:

- Multiple cracks e.g.: stress intensity factor calculation in a high pressure waterbox with three cracks and a mixture of pressure and thermal transient loading (Fig. 7).

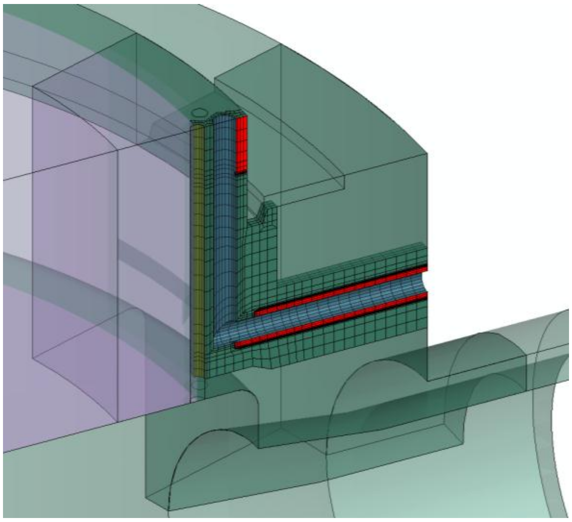

Fig. 7. Cracks in a waterbox model.

- Superimposing steady state conditions and transients for cracked ligaments in superheater tubeplate outlet pipes:
  - ○ Pressure and pipe end loads
  - ○ Multiple thermal transients: startup, reactor trip, refuel transients
  - ○ In-phase and out-of phase load combinations
  - ○ Fatigue and creep crack growth

- Mixed mode crack growth in rotating machinery components.

- Pressure, thermal transient and creep analysis for embedded defects, including crack face contact if necessary (Fig. 8).

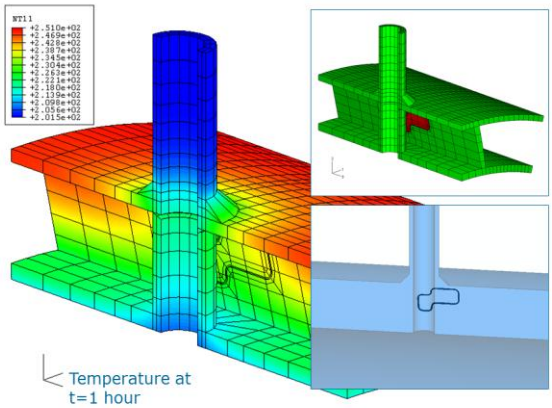

Fig. 8. Embedded crack in a reheater drum wall.

- Replication of specimen tests to help with material modelling and development (Fig. 9).

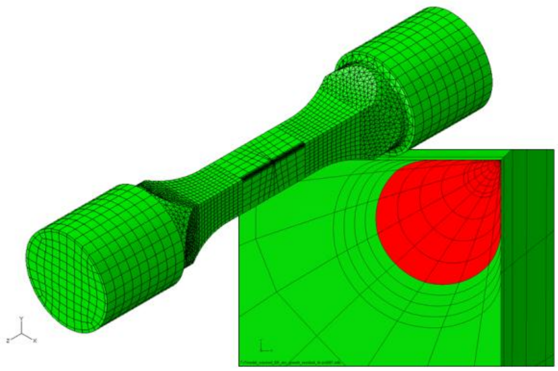

Fig. 9. Burrowing in a corner crack test specimen.

References

[1] Zencrack software, Zentech International Limited, www.zentech.co.uk/zencrack.htm.

[2] API 579-1/ASME FFS-1 2007 Fitness-For-Service, June 2007.

[3] API 579-1/ASME FFS-1 2007 Fitness-For-Service Errata, February 2009.

[4] Abaqus software, Dassault Systèmes Simulia Corp., www.simulia.com.

[5] Ansys software, Ansys Inc., www.ansys.com.

[6] Maligno AR, Rajaratnam S, Leen SB, Williams EJ. A three-dimensional (3D) numerical study of fatigue crack growth using remeshing techniques. Engng Fracture Mech, 2010, 77(1): 94-111.

[7] API 579-2/ASME FFS-2 2009, Fitness-For-Service Example Problem Manual, August 2009.