Zencrack

3D fracture mechanics simulation

State-of-the-art software for 3D fracture mechanics simulation

Zencrack is an advanced engineering analysis tool for 3D fracture mechanics assessment and crack growth simulation. The program uses finite element analysis to allow calculation of fracture mechanics parameters such as energy release rate and stress intensity factors. This is achieved by automatic generation of focused cracked meshes from uncracked finite element models.
A general mixed-mode crack growth capability allows non-planar crack growth prediction for fatigue and time-dependent load conditions via automated adaptive meshing techniques.

Overview

Zencrack provides flexibility with two levels of simulation capability - Standard and Professional.
For industries where static loading is important the Standard version can be used to evaluate stress intensity factors using energy release rate and nodal displacement methods. For thermal transients, the instantaneous stress intensities may be evaluated through the transient to steady state conditions. Collapse analyses may be undertaken to allow, for example, generation of data for failure assessment diagrams.
The second level of capability is introduced through Zencrack Professional and provides a facility for 3D non-planar crack growth prediction for cases of fatigue and time-dependent loading. This includes several options for crack growth data definition and a flexible “load system” approach for defining complex load spectra.

Application areas

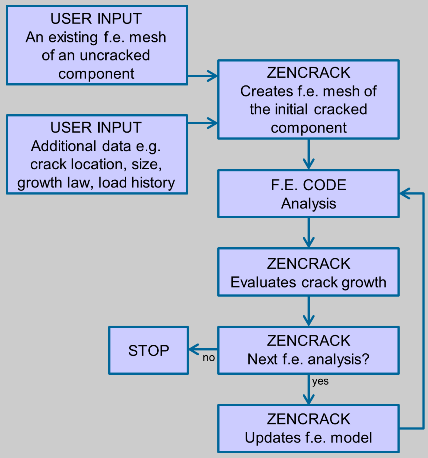
Zencrack can be applied in any industry in which knowledge of crack behaviour, crack growth prediction and residual life calculation are important. Zencrack is relevant in many situations, e.g.

- Post-failure, forensic and accident investigations
- Leak before break studies
- Determination of inspection periods
- Determination of residual life
- Durability of additive manufactured components
- Assessment of adhesively bonded lap joints and composite patch repairs
- Assessment of brittle-ductile failure using failure assessment diagrams.

F.E. interfaces

By interfacing to commercial finite element codes rather than using a proprietary finite element solution, Zencrack is able to take advantage of the many man-years of development within these codes and their associated pre and post-processors. Users have at their disposal all the capabilities within these codes and may include any number of non-linear features in an analysis, in addition to being able to carry out a “standard” linear elastic fracture mechanics analysis. For example:

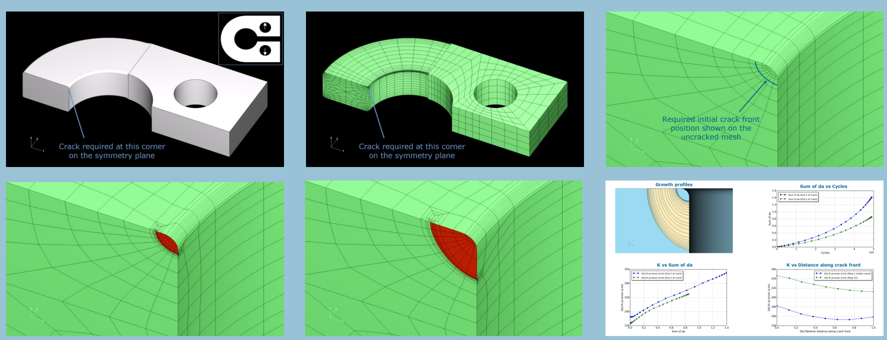
- Non-linear materials
- Large deformation (e.g. rubber, rolling)
- Contact (between components and/or crack face contact)
- Plastic collapse.



Zencrack takes an uncracked finite element mesh and inserts one or more user-specified cracks before submitting the model for analysis. Results are processed automatically and the mesh updated ready for the next finite element analysis of the advanced crack position.

Zencrack has interfaces to Abaqus/Standard, Ansys/Mechanical APDL and Simcenter Nastran.

A typical crack growth analysis procedure



A typical crack growth analysis begins with a geometry model created in the user’s pre-processor. This is meshed as an “uncracked model” in preparation for the insertion of the defect(s) by Zencrack. The insertion of the initial defect (shown in this panel using a crack-block example) results in a cracked mesh ready for analysis in the interfaced finite element code. The analysis results are extracted for a fully automatic crack growth simulation in which re-meshing technologies allow the crack to be advanced through the structure. Typical results from an analysis include crack growth profiles and plots such as crack growth vs load cycles and stress intensity factor vs crack size.

Cost benefit

In the past, the generation of finite element meshes for 3D fracture mechanics applications was both time-consuming and difficult. The evaluation of stress intensity factors was generally limited to simple planar cracks in 3D structures, usually under mode I loading. Non-planar crack growth prediction in 3D structures was virtually impossible. Zencrack removes these limitations and provides simulations that generate cost-effective solutions to otherwise intractable problems. The use of interfaces to commercially available f.e. codes allows users to take full advantage of their existing software investments.

Meshing procedure

The generation of 3D finite element models suitable for analysis of cracks requires special attention in and around the crack region where focused rings of hex elements are used. In addition, the initial crack front may be straight, a simple elliptic section, or a general curve in space.

To address these issues, Zencrack removes the onus of modelling the crack region from the analyst and requires instead that an uncracked mesh is supplied.

A “crack-block” or “remeshing” approach is then used to introduce crack fronts into the mesh (see adjacent panels).

In a crack growth analysis the shape of the crack that develops is a function of the geometry, loading history and material properties. These parameters result in a variation of stress intensity factors along the crack front and, in general, different growth magnitude and direction along the crack front.

The remeshing of the crack-region takes places automatically to allow the correct crack shape development in the structure. No shape assumptions are forced onto the developing crack shape.

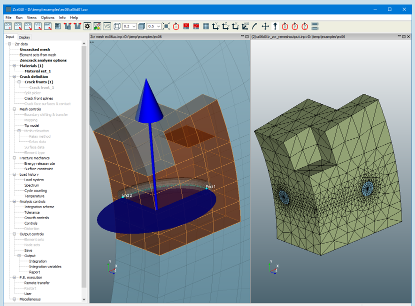
Fracture mechanics parameters

The fracture mechanics parameters for the cracked mesh are calculated using two methods:

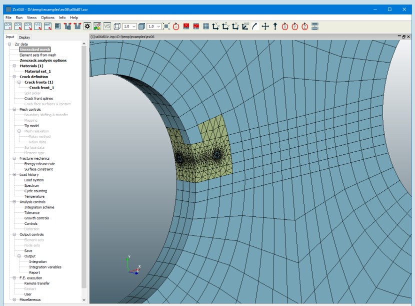
- The contour integral capability in the interfaced f.e. code e.g.
  - j-integral
  - SIF integral
  - Ct-integral
  - T-stress.
- Conversion of displacements on the crack faces to stress intensity factors. This requires a linear isotropic material.

Remeshing method

The remeshing method, introduced in Zencrack 9.0-1, simplifies the initial crack definition and provides more generality during crack advancement than the crack-block method. The initial crack definition is geometric and independent of the mesh. Once the initial crack geometry is defined, a “remesh region” is created in the mesh. Elements in this region, whose extent can be calculated automatically, are removed and replaced by new mesh. The new mesh includes hex elements formed into rings along the crack front, surrounded by tet elements. The various mesh parts are tied together to construct a valid cracked mesh. Element densities in the tet region can be calculated automatically to give transition from the rings to the surrounding mesh with optional manual control if desired. The remesh region is recalculated at each step of the crack growth process.



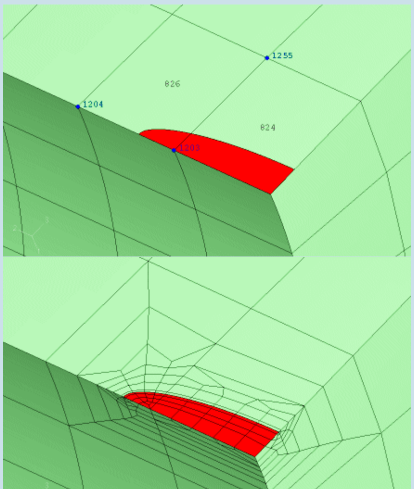
A preview of a geometric elliptic crack showing the crack front rings, remesh region and remeshed region.



The cracked mesh after full insertion of the remeshed region into the surrounding mesh.

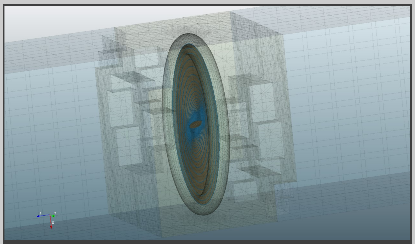
Crack-block method

The term crack-block refers to a collection of brick elements stored as a unit cube. In their unit cube form each crack-block contains either a quarter circular or through crack front on one face. The meshing procedure is one of replacement of one or more 8 or 20 noded brick elements in a user-supplied uncracked mesh by crack-blocks. During the mapping process to introduce the crack-blocks the user can control the size and shape of the generated crack front section for each crack-block. Crack -blocks can be connected together to form one or more distinct crack fronts of the required size in the cracked mesh.

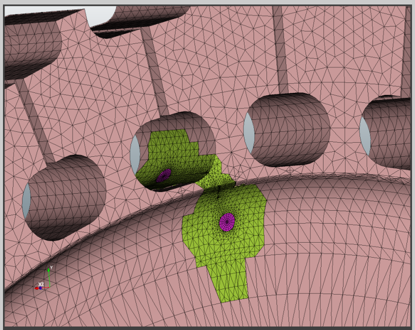


Two elements in an uncracked mesh are replaced by crack-blocks to model a quarter-elliptic surface crack.

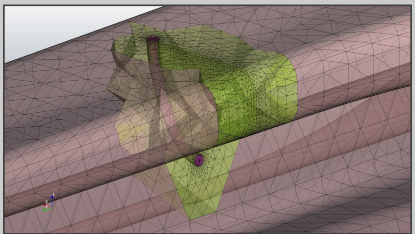
Sample applications



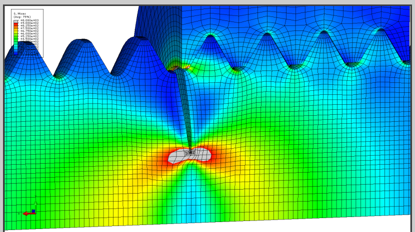
Growth of an initially mixed-mode embedded crack



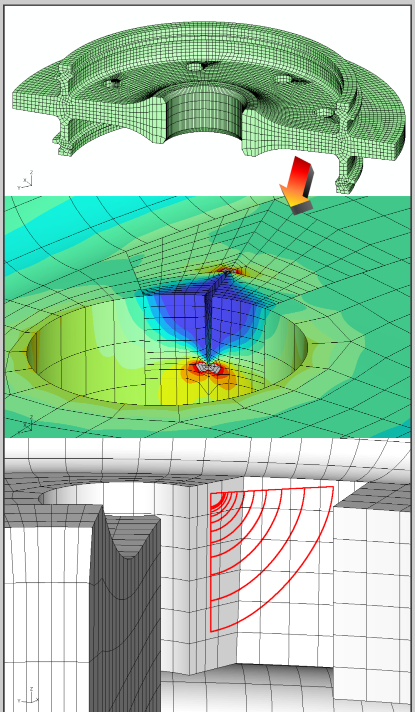
Crack in a minidisk model



Crack in a connecting rod



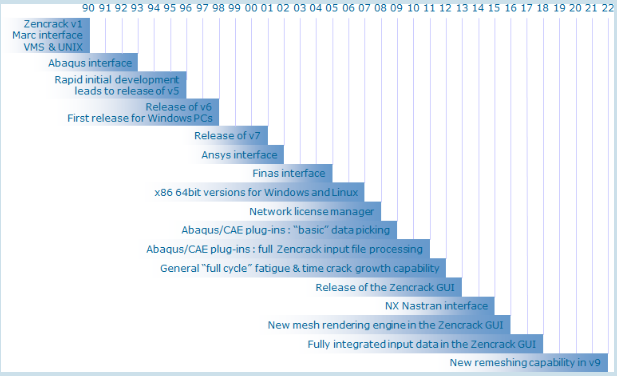
Crack growth in a threaded connection with multiple



Crack growth at a bolt hole in an engine disk includ-

Historical perspective

Zentech was contracted by RAE Farnborough (late 1980s) to develop methods for the analysis of elliptic corner cracks in rectangular bars. Finite element work was undertaken with the Marc code for which Zentech was then U.K. agent. Upon completion of that project Zentech embarked on a new development using some of the RAE-based concepts to create a general software package - Zencrack. The first release of Zencrack in 1990 operated on VAX/VMS hardware and was interfaced to Marc (K4). In 1994 Zencrack was released with the first interface to Abaqus (5.2). Development of the code progressed over the next decade with ports to a range of platforms and enhancements to the program capabilities. As finite element analysis moved onto the PC platform, so the first Windows version of Zencrack was released in 1998. In the early 2000s, significant developments to Zencrack were undertaken for the U.S.A.F. at Wright Patterson Air Force Base. Part of this work led to an interface to Ansys. Involvement in a three year research project led by Rolls-Royce, with a particular interest in high temperature effects, resulted in a general "full cycle" fatigue and time dependent crack growth capability in 2012. The release of version 7.9 in 2013 included the first Zencrack GUI, providing the same features for all Zencrack users regardless of their chosen finite element interface, allowing input data generation and post-processing. An update in 2015 provided a Zencrack interface to NX Nastran. The release of version 8.0 in 2016 brought dedicated input screens to the GUI for all Zencrack keywords with version 8.3 in 2018 adding a new rendering engine for meshes. A new tet-based remeshing methodology was introduced in Zencrack version 9.0 in early 2022.



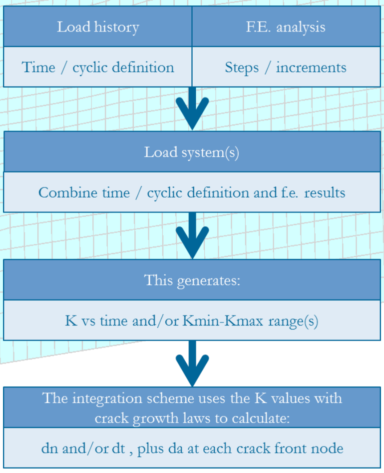
Load handling

The loading requirements in a fracture mechanics analysis vary considerably depending upon the goal of the analysis. Some examples include:

- Parametric study of crack sizes to determine stress intensity factors
- Evaluation of j-integrals at multiple load increments in a non-linear analysis
- Fatigue crack growth analysis with constant amplitude loading
- Fatigue crack growth analysis with spectrum loading
- Sustained load crack growth analysis with load as a function of time
- Combined fatigue and time dependent crack growth in which cyclic and time effects both contribute to the overall crack growth.

These requirements are tackled in Zencrack by the use of a "load system" methodology. Load systems provide a framework that cross-references the finite element results to the applied loading spectrum. Energy release rates or stress intensity factors for the load sequence can then be used in the crack growth integration scheme.

This allows crack growth analysis ranging from constant amplitude loading to complex load cycles. Out-of-phase temperatures and stresses or non-linearities such as contact may be included. In the most complex cases a full load cycle is modelled in the f.e. analysis to generate a time history of stress intensity factors for each node on each crack front. These time histories can be treated by using the major cycle only, or they may be cycle counted to extract minor cycles. This type of time history presents a framework which allows fatigue-only, time-only or combined fatigue and time dependent crack growth prediction.



Conversion of loading to meaningful data for crack growth integration

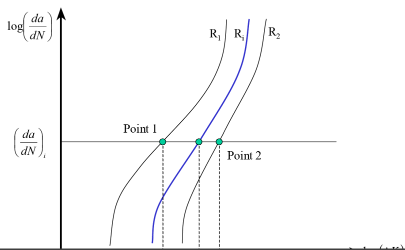
Material Data

Zencrack provides several options for input of material and crack growth data. This includes temperature dependency which can be important for thermo-mechanical simulations.

For a fatigue crack growth analysis the crack growth data can be input in standard Paris, Walker, Forman and Hartman-Schijve forms. A piecewise curve option is also available. For more complex data a tabular option allows da/dn vs ΔK curves as a function of stress ratio. All these options can include temperature dependency.

In addition to the crack growth data, options are available for threshold definition and fracture toughness.

For users with proprietary data or other non-standard data, a number of user subroutine options are available for definition of both the crack growth data and/or the threshold condition. The user subroutine options also allow, for example, use of a NASGRO crack growth law.



da/dn vs ΔK curves as a function of stress ratio (can also be a function of temperature)

Time dependent or sustained load crack growth analyses can be carried out with Zencrack and these require data in the form of da/dt against K or C_t. As with fatigue crack growth data, several options provide flexibility of input from simple single curve data through to user subroutines. The COMET crack growth law embodies the high temperature effects of Creep, Oxidation, Microstructure, Environment and Temperature in a temperature dependent K-based crack growth law.

Verification & QA

Zencrack has been verified against theoretical solutions, other software and experimental data. Independent assessment by users has given further confidence that Zencrack produces reliable and meaningful simulation results across a broad range of problems. A large suite of analyses is used for regression testing during development and prior to software release.

Zencrack GUI

The Zencrack GUI helps in the tasks of creating and reviewing a Zencrack analysis. The GUI is independent of the interfaced finite element code and provides the same level of capability to Abaqus, Ansys and Simcenter Nastran users on both Windows and Linux platforms. The GUI main screen consists of five regions:

- The left pane with the 'Input' tab describing the zcr input file, and the 'Display' tab with options for visualisation of results and recall of saved viewpoints.
- The graphics area for display of meshes and results in multiple viewports.
- The toolbar region with icons for important operations.
- The drop-down menus with standard options such as File & Help and replication of some toolbar operations.
- The status bar at the bottom left of the window for providing information messages.

Capabilities for Zencrack input data generation and review include:

- Import of an Abaqus, Ansys or Simcenter Nastran uncracked mesh; as with a Zencrack analysis, the origin of the uncracked mesh file is not important - it may have been generated in any pre-processor.
- Extraction of node and element data required in the zcr file from the uncracked mesh by direct picking from a mesh e.g. crack-block location and orientation, nodes for geometric crack positioning etc.
- Visual confirmation that picked data are correct.
- Other types of input data are entered via dedicated screens which include help links and information about defaults.
- Comprehensive warning and error checking of input data.
- Interactive preview of remeshing options to check the remeshed region before running a job.

For Zencrack analysis submission:

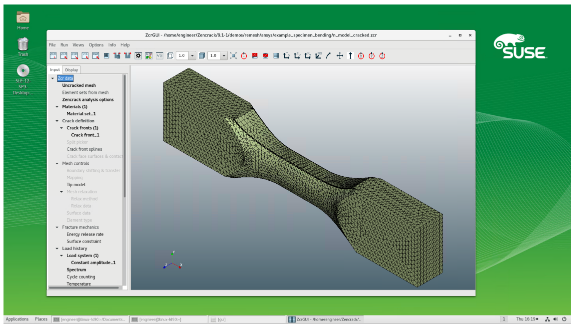
- Create and display the initial cracked mesh for a given set of input data.
- Submit a full Zencrack analysis.

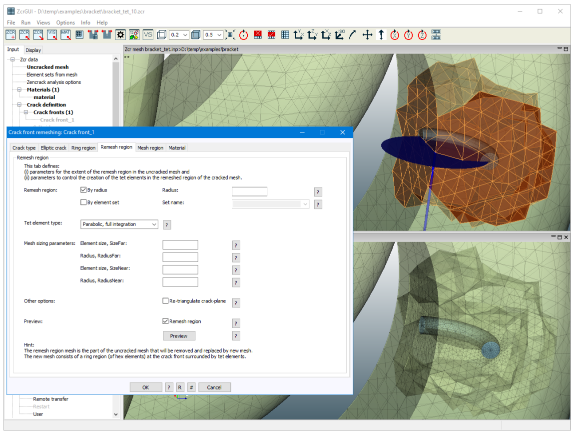
For post-processing:

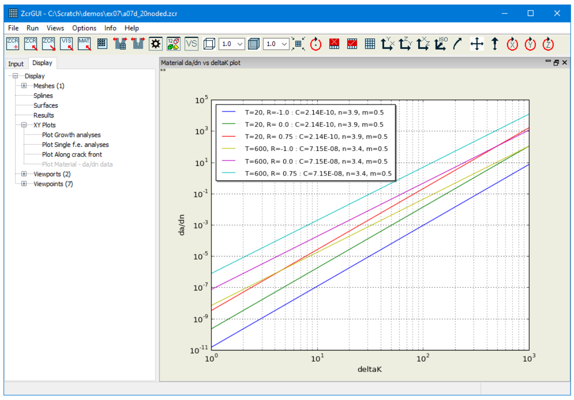
- Import multiple uncracked or cracked meshes.
- Import crack surface and growth profiles from Zencrack analyses and display them alone or superimposed on one another or on any imported mesh.
- Create a range of xy plots: along a crack front, from crack growth results (e.g. a vs N) or from the multi-increment results of a single f.e. analysis.
- Create material data plots.
- Create, modify and display splines from stand-alone data or from selected profiles from a Zencrack growth analysis.

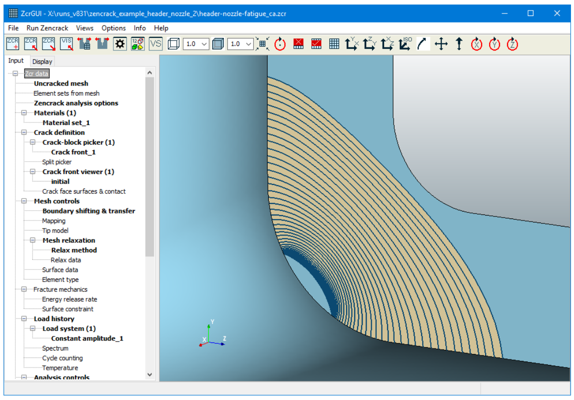
All mesh import includes the following features:

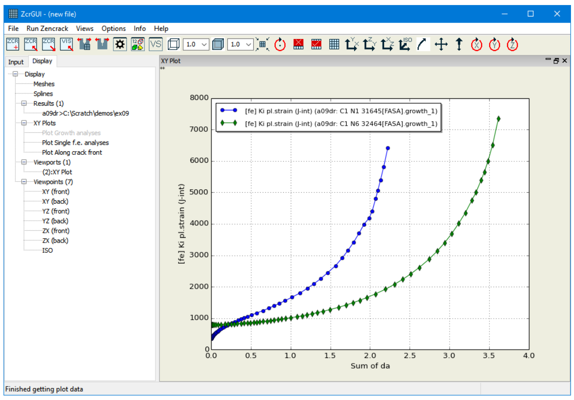
- Display of first and second order tet, prism and brick elements.
- Import of node/element sets (Abaqus) or node/element components (Ansys).
- Option to import or ignore data from "included" input files (i.e. data defined via Abaqus *INCLUDE or Ansys /INPUT options).









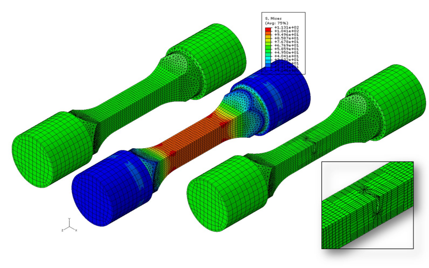


Modelling idealised and observed crack shapes in test specimens

Zentech was involved in a collaborative research project called "DISPLACE" between Jan 2009 and Dec 2011: Developing Improved Service Propagation Lives in Arduous Cyclic Environments. The project was part funded by the UK Technology Strategy Board Project (reference TP/8/MAT/6/I/Q1525K). The total project cost was £1,692,589 with the TSB providing approximately half of that amount in funding. The project was led by Rolls-Royce (Derby) with partners:

- University of Birmingham
- University of Portsmouth
- Serco
- Zentech International Limited

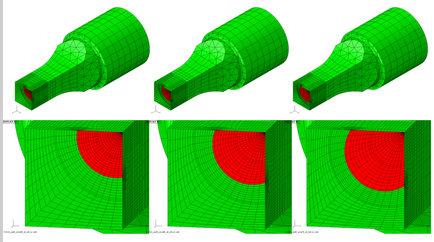
As part of the project involvement, Zentech carried out simulations related to crack shape development in corner cracked test specimens. Investigations involved modelling of idealised tunnelled crack shapes and actual crack shapes observed during high temperature dwell tests. The Zencrack capabilities for high temperature time dependent crack growth prediction were also enhanced during the DISPLACE project.

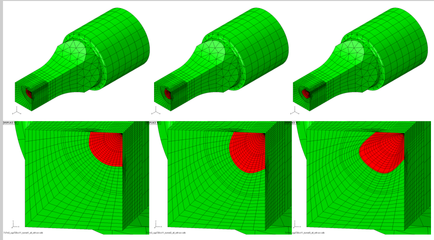

Uncracked and cracked test specimen models

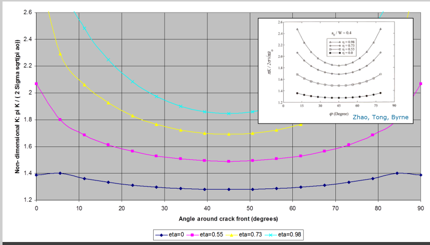
The geometry used for test and simulation was a corner crack specimen with a 7mm cross section. Idealised tunnelled crack sizes were modelled by offsetting a circular profile along the diagonal of the specimen cross-section. This is easily achieved with the user defined initial crack front option in Zencrack. Similarly, the user defined crack front option can be used to create a model with an observed non-symmetric crack shape.

Part of the investigation compared stress intensity factors from Zencrack with reference solutions.

(Stress intensity factor K and the elastic T-stress for corner cracks, Zhao, Tong, Byrne, International Journal Of Fracture, 109: 209-255, 2001).

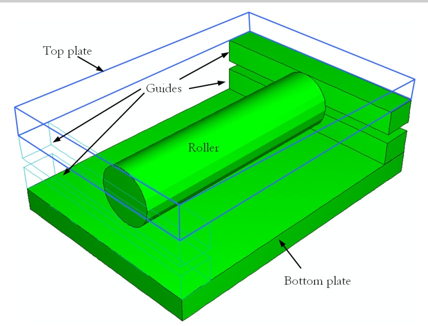

Models with idealised tunnelled crack shapes, $a_o/W=0.4$


Models with observed tunnelled crack shapes



K distributions for idealised tunnelled crack shapes

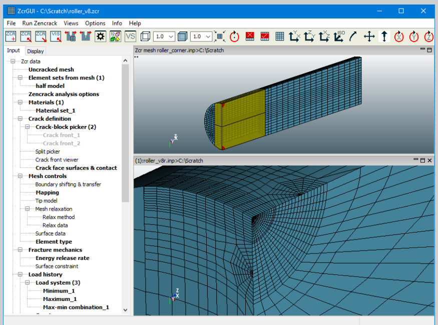
Forensic failure analysis of bridge roller bearings

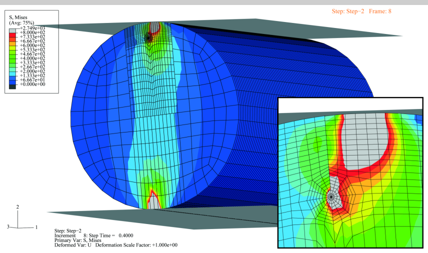


Schematic of roller, plates and end guides

This example shows how Abaqus was combined with Zencrack to predict 3D crack growth as part of an investigation to explain failure of single cylinder bridge roller bearings. The rollers in question, part of a key bridge structure on the UK motorway network, began to fail several years after a major refurbishment programme. Finite element analyses were conducted to gain an understanding of the stresses caused during operation and to explain the possible cause of crack growth resulting in failure. The fatigue load cycle is generated by repeated rolling contact between the roller and plates due to ambient temperature changes.

A number of analyses were performed to investigate growth of initial cracks at one end of the roller. Of those analyses, model plots are shown here for analysis of a perfectly aligned roller (i.e. no contact with end guides) having diameter 120mm, length 480mm and initial corner cracks of radius 10mm.

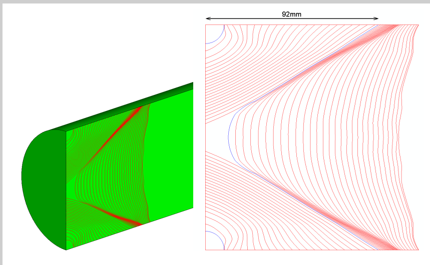

Crack position verification in the Zencrack GUI


Von Mises stress for a model with a single defect

Key features of the analysis included:

- Use of mode II crack growth law due to dominance of shear cracking.
- Rigid surfaces for plates: bottom plate fixed and top plate given a vertical load followed by horizontal translation.
- Contact between the crack faces.

More details: Prediction Of Crack Growth In Bridge Roller Bearings, N.K. Prinja, J.M. Bushell, AMEC Nuclear UK Ltd., R. Chandwani, C. Timbrell, Zentech Int. Ltd., NAFEMS World Congress, Crete, Jun 16-19 2009.



Calculated profiles: corner and through crack phases

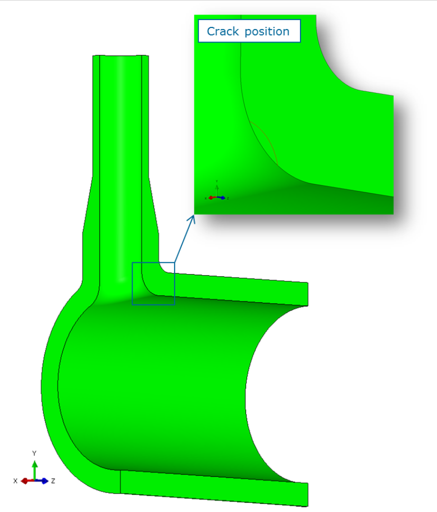

Photograph of one of the failed roller bearings

Pressurised thermal shock in a cracked header-nozzle intersection

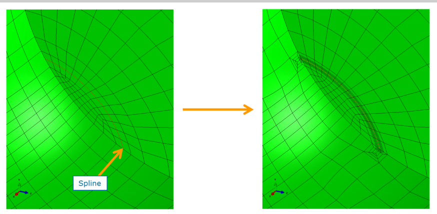
This example shows analysis of a defect through a typical pressurized thermal shock transient for a small-break loss-of-coolant accident in a header-nozzle intersection. The header pipe has a 30 inch outer diameter and $3^{1}/_{8}$ inch wall thickness. The nozzle has a $10^{5}/_{8}$ inch outer diameter and $1^{5}/_{16}$ inch wall thickness. The aim of the analysis is to determine the time history of stress intensity factor at the ends of the crack front and on a 45 degree line through the component cross-section.

The uncracked component, the crack position and the mesh with the crack in place are shown in the figures below. The time history of internal pressure and temperature are derived from data in two refences:

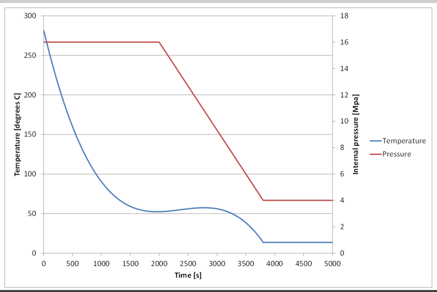
- Variable Flaw Shape Analysis for a Reactor Vessel Under Pressurized Thermal Shock Loading, Yang & Bamford, Fracture Mechanics Vol. 17, ASTM STP 905, 1986, pp41-58.
- Influence of Pressurised Thermal Shock on Nuclear Pressure Vessels, Lisha, Hongguang, Huiji, (Tsinghua University), Research Reports of Structural Engineering and Vibration 7, Tsinghua University Press Ltd, 2006.



Uncracked geometry and the required crack position



Uncracked and cracked meshes at the crack position

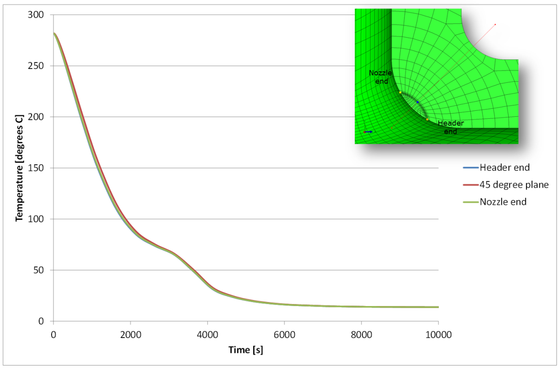


Internal temperature and pressure time histories (inputs)

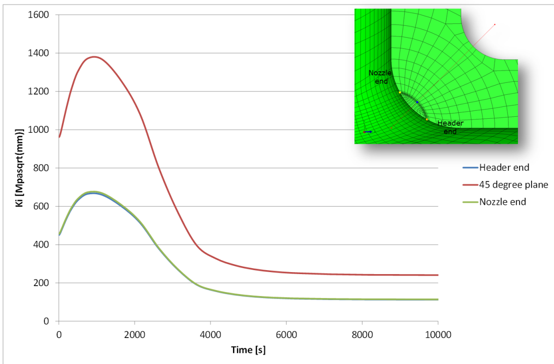
Typical temperature dependent steel data for Young's modulus and Poisson ratio has been used in the f.e. analysis and the Zencrack input data. The transient analysis is carried out as a sequential thermal-stress analysis. Abaqus allows interpolation of temperatures from one mesh to a dis-similar mesh so an uncracked model heat transfer analysis provides the temperature input for the Zencrack stress analysis of the cracked mesh.

Results are extracted at the two end points of the crack and at the intersection of a 45 degree line with the crack front, as shown on the right. The results at this intersection point are generated by Zencrack's post-processing program, PROCESS, by interpolating data from the adjacent crack front nodes.

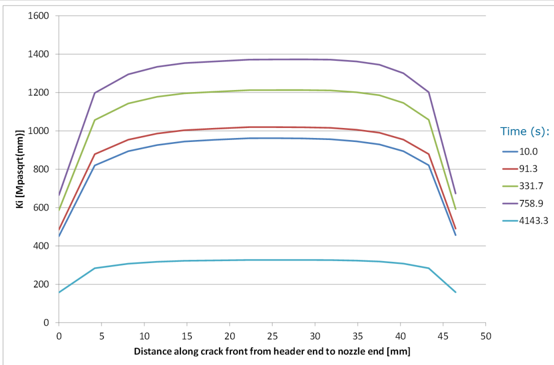
The temperature time histories for the three output points are virtually the same although, as might be expected, the point on the 45 degree line has a small time lag compared to the surface points. The 45 degree line shows a significantly higher K value than the two end points, reaching almost 1400MPamm½ at a time of around 1000 seconds. The final figure in the column above shows distribution of K values along the crack front at different times through the analysis. Five time points have been plotted.



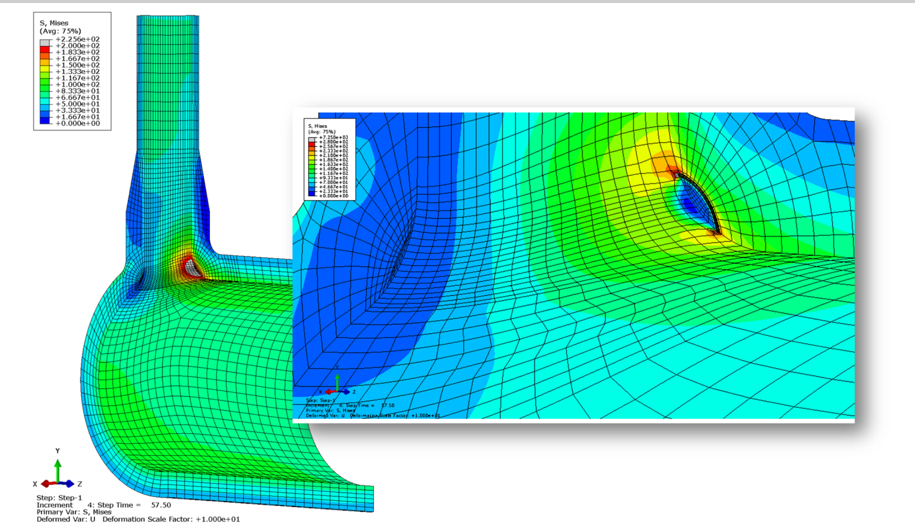
Temperature time histories at three points on the crack (outputs)



SIF time histories at three points on the crack



SIF along the crack at different times during the transient



Contours of Von Mises stress at 57.5 seconds into the transient

Thermo-mechanical fatigue and time dependent crack growth in an aero engine disk

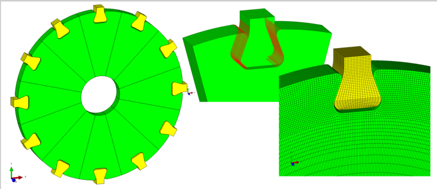
Metal alloys are continuously being improved to provide greater strength against component failure and also to increase resistance against crack propagation. This is a particular issue in nickel based alloys used in aerospace engines which experience high temperatures under both sustained and cyclic loading conditions. Traditional analysis techniques account for fatigue crack growth effects but time dependent crack growth is generally more difficult to evaluate. The “full cycle” capability in Zencrack allows combined fatigue and time dependent crack growth prediction.

The proprietary nature of aerospace engine components and material data means that “real” geometry and material data are replaced with “representative” data for the purposes of this example. The geometry used is a single sector of the disk. Two contact regions are defined for the blade/disk interaction. A spatial temperature distribution is defined which varies with time. The engine rotation time history is such that maximum temperature occurs after maximum engine speed.

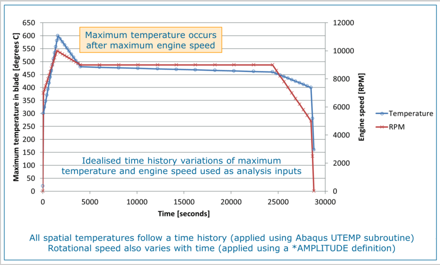
Key features of the analysis include:

- Temperature dependent Young's modulus and Poisson ratio
- Fatigue crack growth data with temperature dependency (Walker equation)
- Time dependent crack growth data with temperature dependency (COMET equation)
- Modelling of the time history of temperature and engine speed for one flight cycle in the f.e. analysis
- Extraction of f.e. results of the flight cycle for input to crack growth calculations
- Handling of different temperatures at K_{max} and K_{min} conditions when calculating da/dn

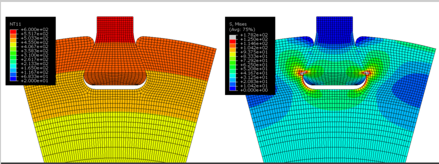
Some analysis results for a through crack position are shown. This demonstrates that the inclusion of time dependent crack growth in the analysis reduces the predicted life, in this example by about one third. The detailed integration output through the flight cycle shows the relative effects of fatigue and time dependent crack growth.



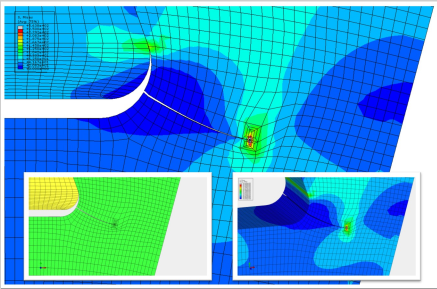
Cyclic symmetry uncracked model



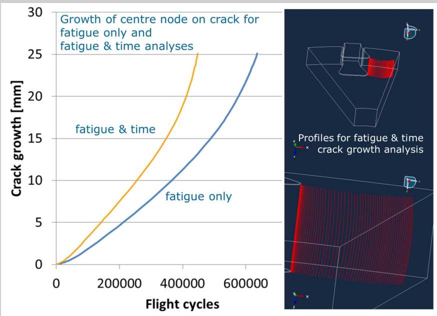
Temperature and engine speed time history



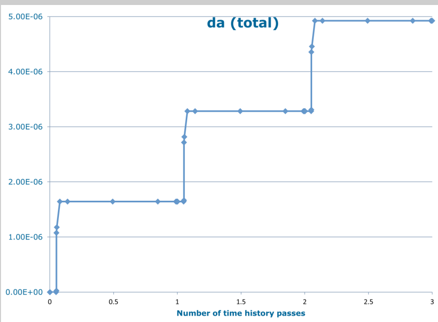
Snapshots of temperature and Von Mises stress on the uncracked mesh at one time point



One crack position during the analysis



Results profiles and comparison of a vs N with and without the effect of time dependent crack growth



Detailed extraction of growth through three flight cycles to identify relative importance of fatigue and time dependent contributions to the overall growth

Evaluation version

A free evaluation version of Zencrack can be download from:
www.zentech.co.uk/zencrack.htm

Contacts

Development work for Zencrack is carried out by Zentech in the U.K.
A network of agents provides marketing support around the world.

For more information please contact Zentech or your local agent.

U.K.
Zentech International Limited
590b Finchley Road, London,
NW11 7RX
Tel: +44 (0)1252 376388
info@zentech.co.uk
www.zentech.co.uk

China
Consys Company Limited
info@consysgroup.com.cn
www.consysgroup.com.cn

India
ProSIM R&D Center
info@pro-sim.com
www.pro-sim.com

2022-01

ZENTECH