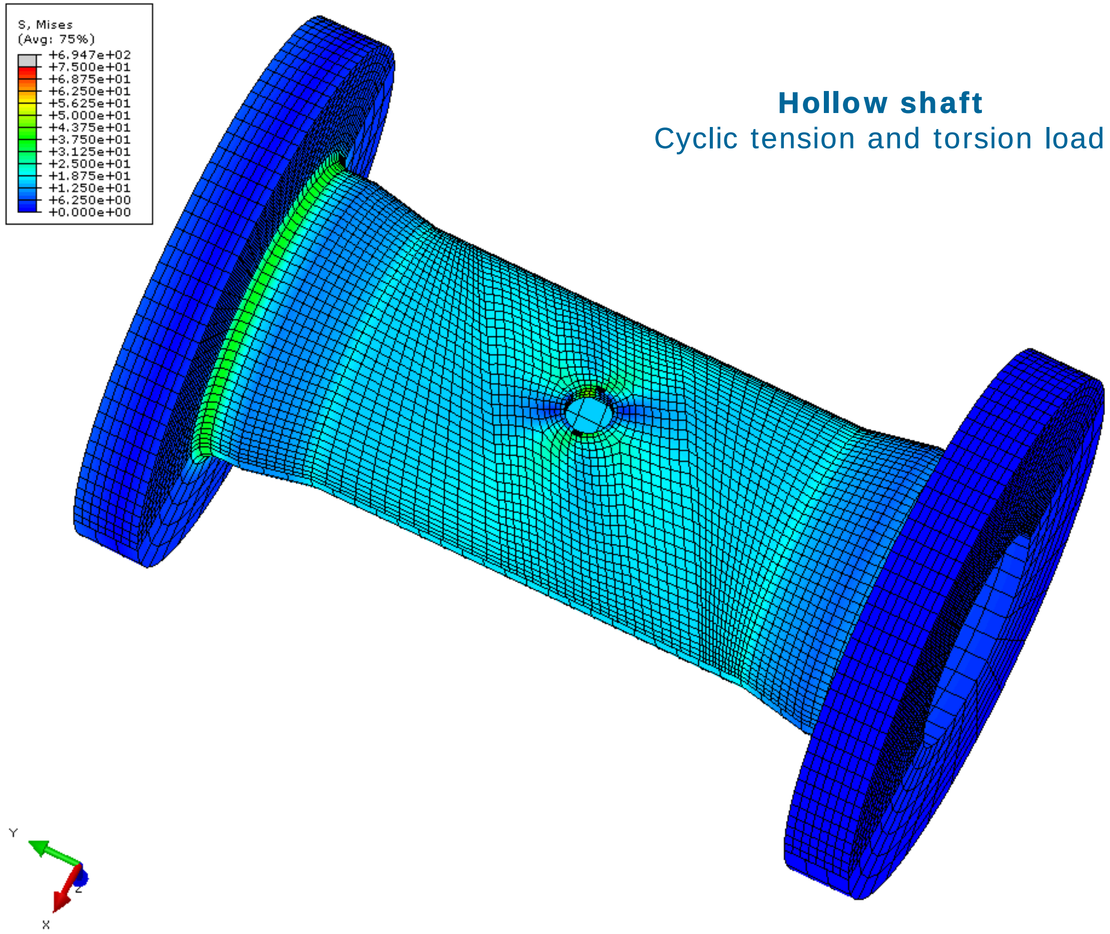
Application of Abaqus to Analysis of 3D Cracks and Fatigue Crack Growth Prediction

C. Timbrell
Zentech International Limited, UK
www.zentech.co.uk

Overview

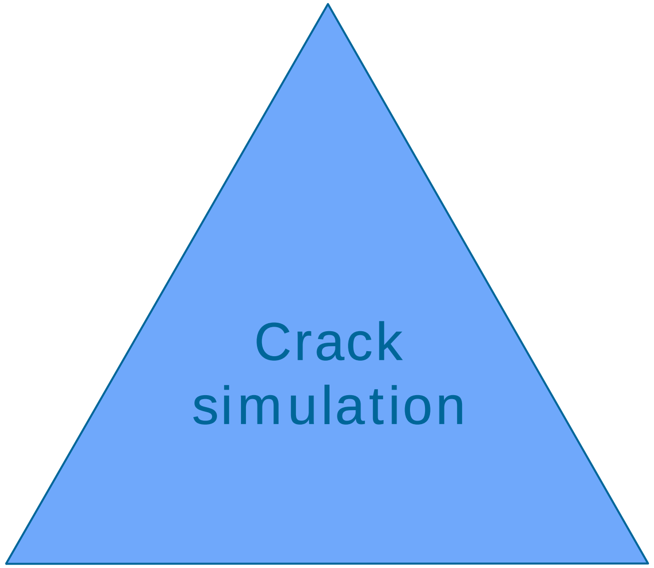
- General requirements for 3D crack simulation
- Historical perspective
  - Zencrack development
- Typical workflow and interaction of Zencrack with Abaqus
- Examples
- Zencrack GUI



Hollow shaft
Cyclic tension and torsion load

Three key areas for crack simulation

Fracture mechanics parameters
K, G, J
Closed form
Numerical evaluation - BEM, FEM

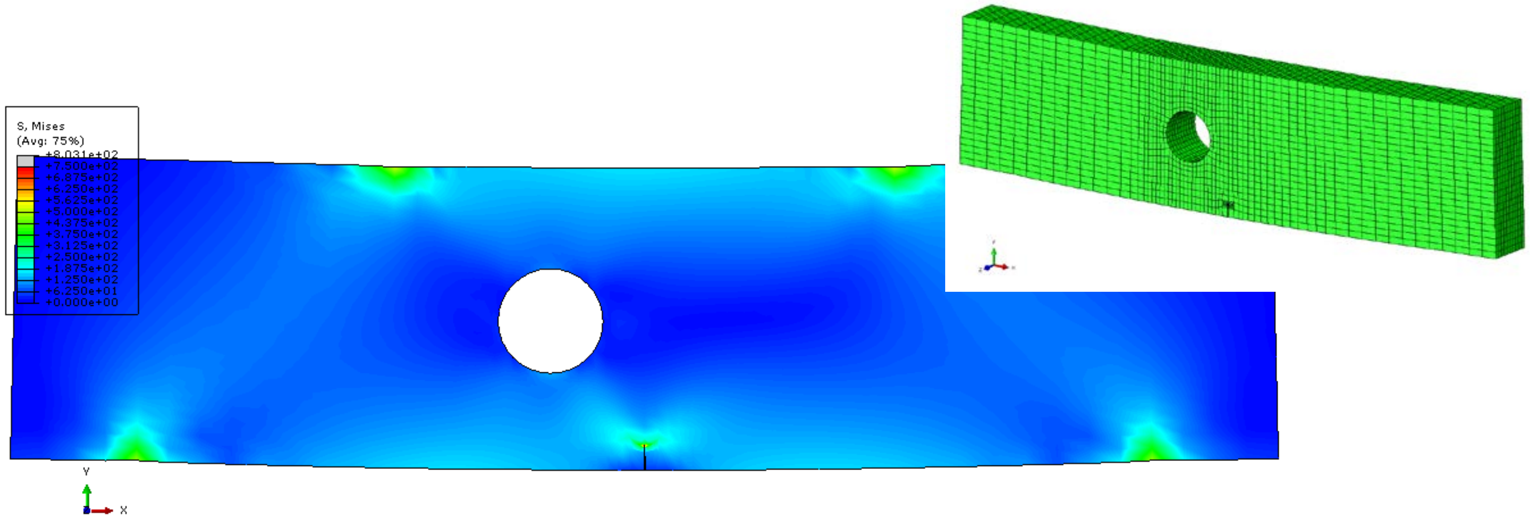


Topology issues
Initial crack
Mixed mode / non-planar growth
Crack shape development
Multiple cracks

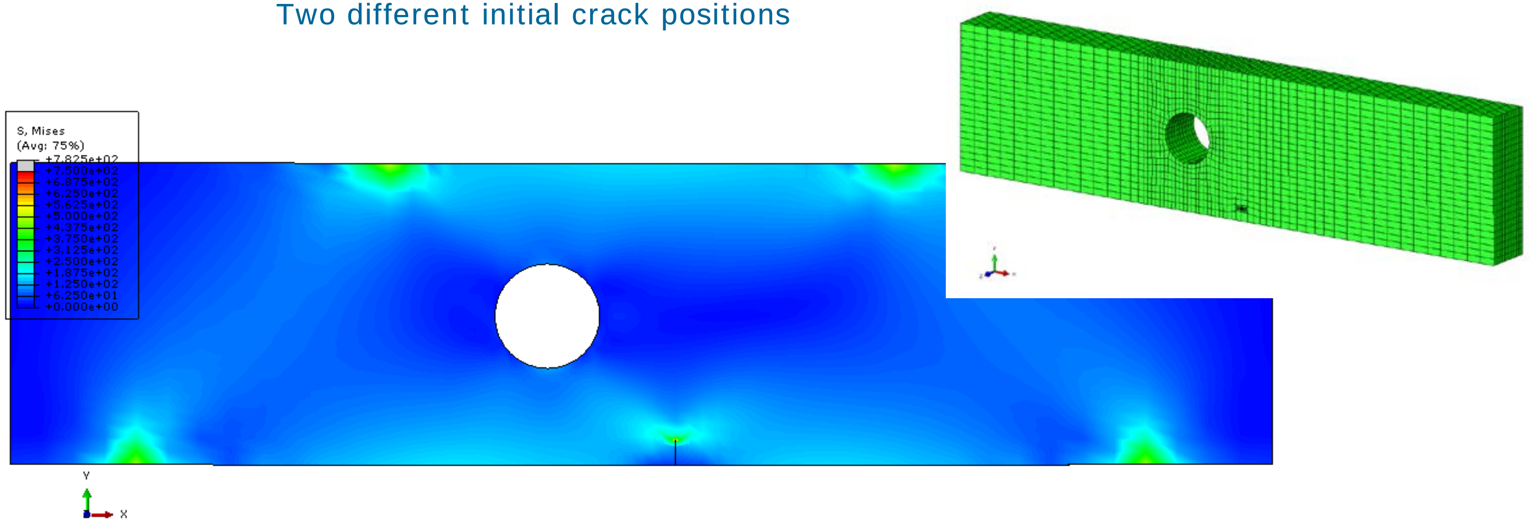
Crack growth integration
Cycle-by-cycle
Finite evaluations of fracture parameters
Crack growth data, threshold effects
Load spectrum, flight cycles

The solution provided by Zencrack

- Zencrack:
  - uses commercially available finite element codes to calculate fracture mechanics parameters
    - » e.g. Abaqus/Standard
  - has meshing algorithms for modelling and updating crack fronts in 3D components
  - has a crack growth integration scheme to allow generalised non-planar crack growth prediction



Four point bend specimen
Two different initial crack positions



Zencrack historical perspective

90 91 92 93 94 95 96 97 98 99 00 01 02 03 04 05 06 07 08 09 10 11 12 13 14 15

Zencrack v1
Marc interface
VMS & UNIX

Abaqus interface

Release of v6
First release for Windows PCs

Release of v7

Ansys interface — v7.2

Finas interface — v7.4

x86 64bit versions for Windows and Linux — v7.5

Network license manager — v7.5-8

Abaqus/CAE plug-ins : "basic" data picking — v7.6

Abaqus/CAE plug-ins : full Zencrack input file processing — v7.7

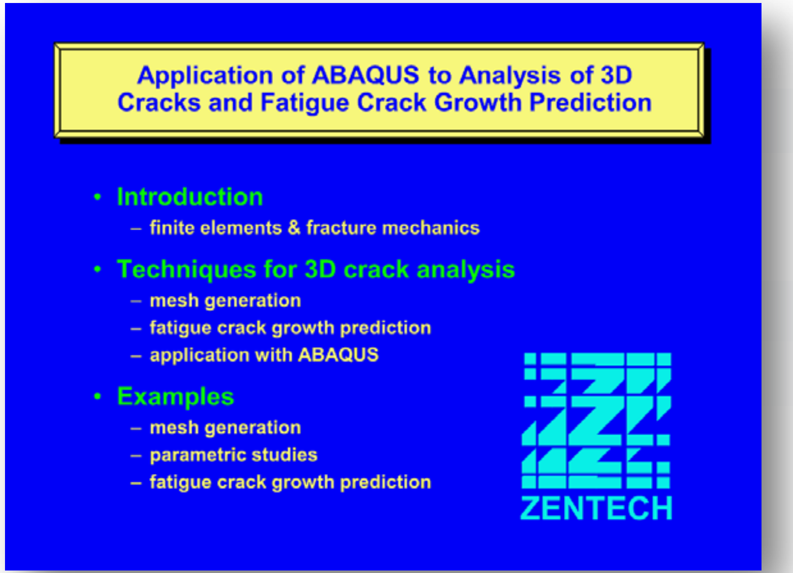
General "full cycle" fatigue & time crack growth capability — v7.8-1

Release of the Zencrack GUI — v7.9-1

NX Nastran interface — v7.9-3

3 year development for USAF at Wright Patterson

- Large crack growth capability
- "Load system" methodology
- Thermo-mechanical fatigue
- Time dependent crack growth
- Enhanced crack growth laws and threshold models



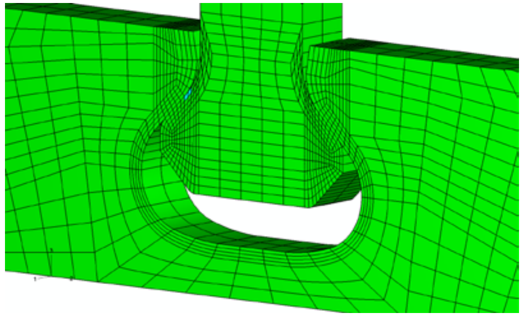
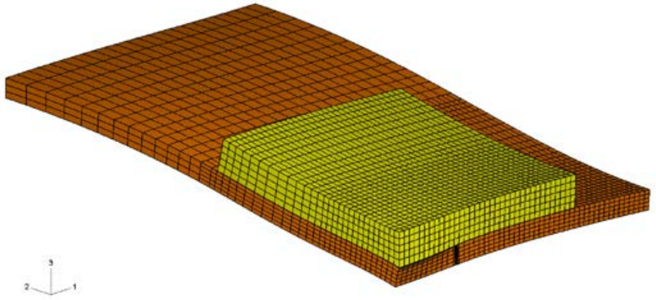
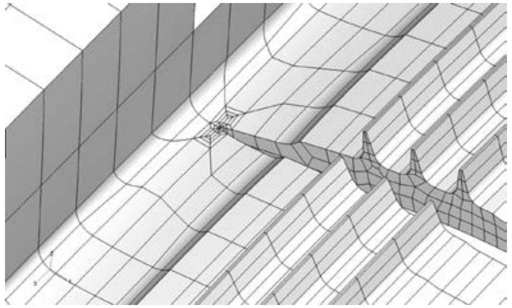
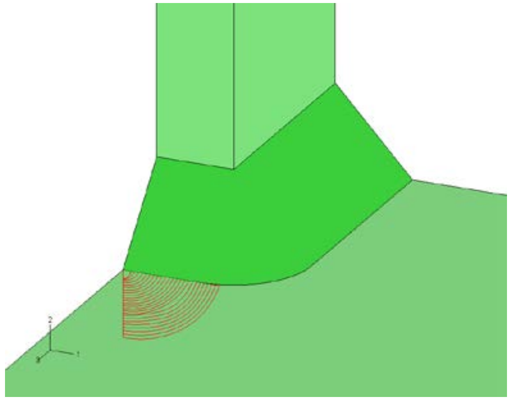
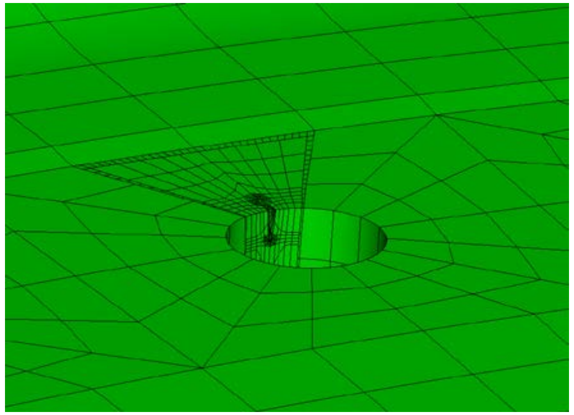
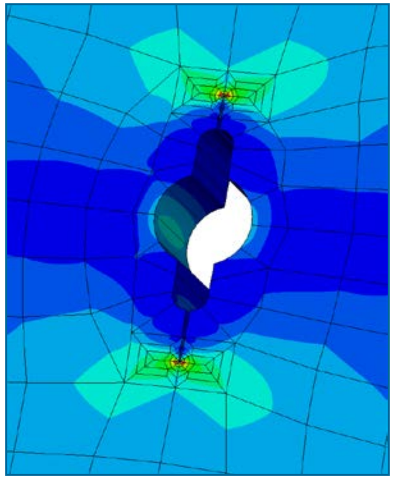
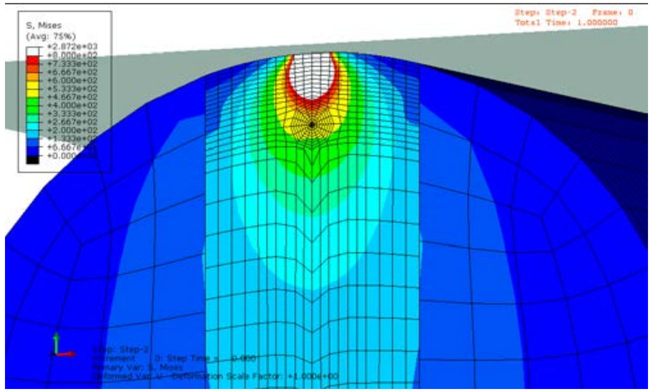
Application of Abaqus to Analysis of 3D Cracks and Fatigue Crack Growth Prediction
C.Timbrell, P.W.Claydon, G.Cook
Abaqus Users' Conference, Rhode Island, U.S.A., June 1-3 1994

3 year DISPLACE project led by Rolls-Royce ended Dec 2011

- Input from Zentech, Serco, Universities of Birmingham and Southampton
- Improved time dependent capability
- General "full cycle" capability

Applications

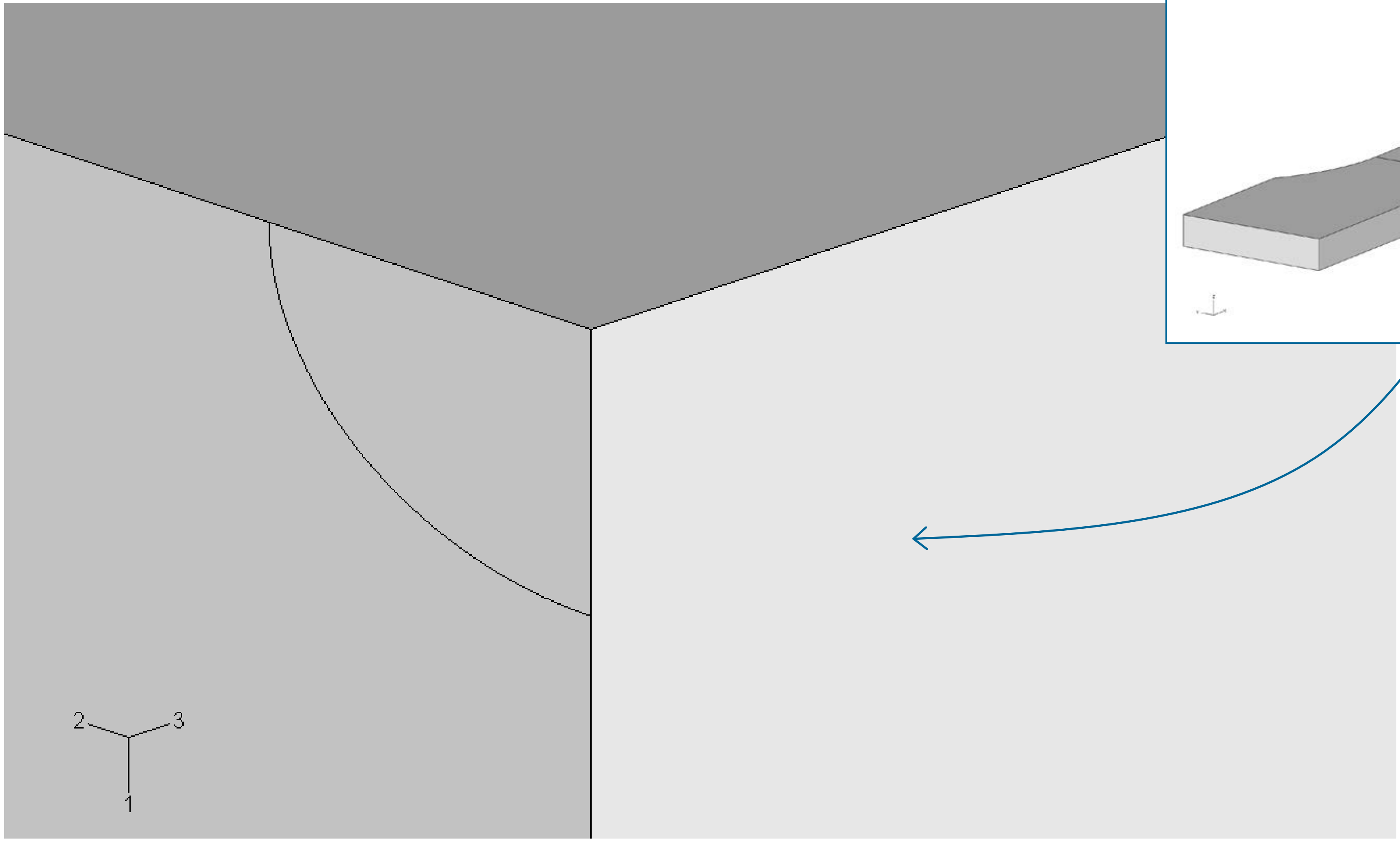
- Life extension, durability & damage tolerance assessment
- Engine disks and blades and their dovetail connections
- Power generation components
- Pipeline defects
- Welded connections
- Stiffened panels (integral & riveted)

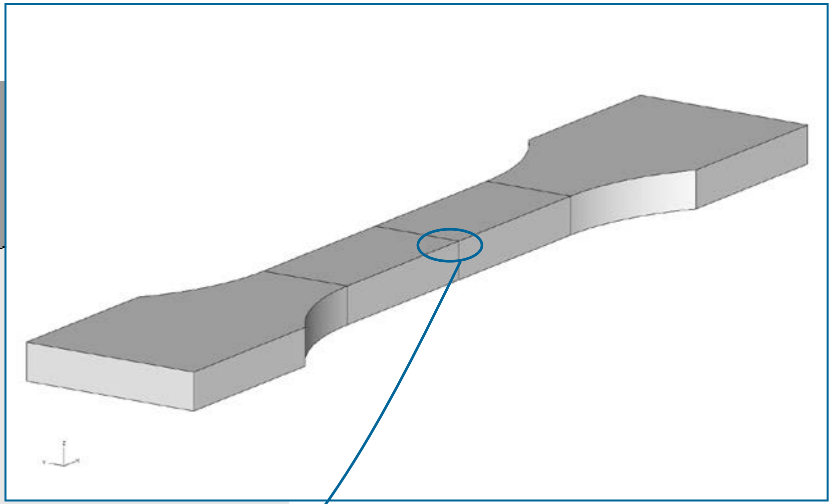








Zencrack workflow for crack growth

- Geometry model
- Uncracked mesh
- Zencrack input data
- Growth analysis
- Post-processing

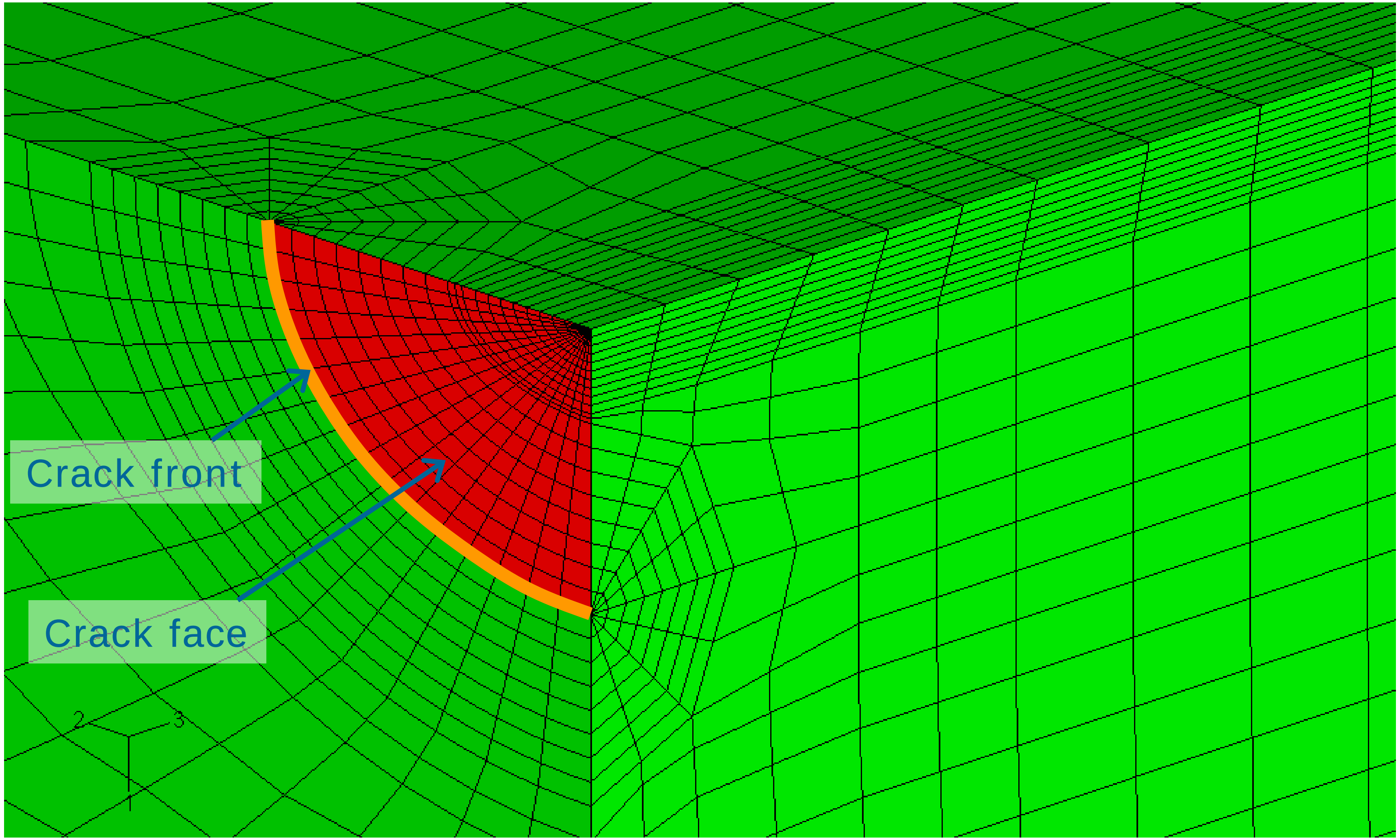
A crack in a 3D finite element mesh





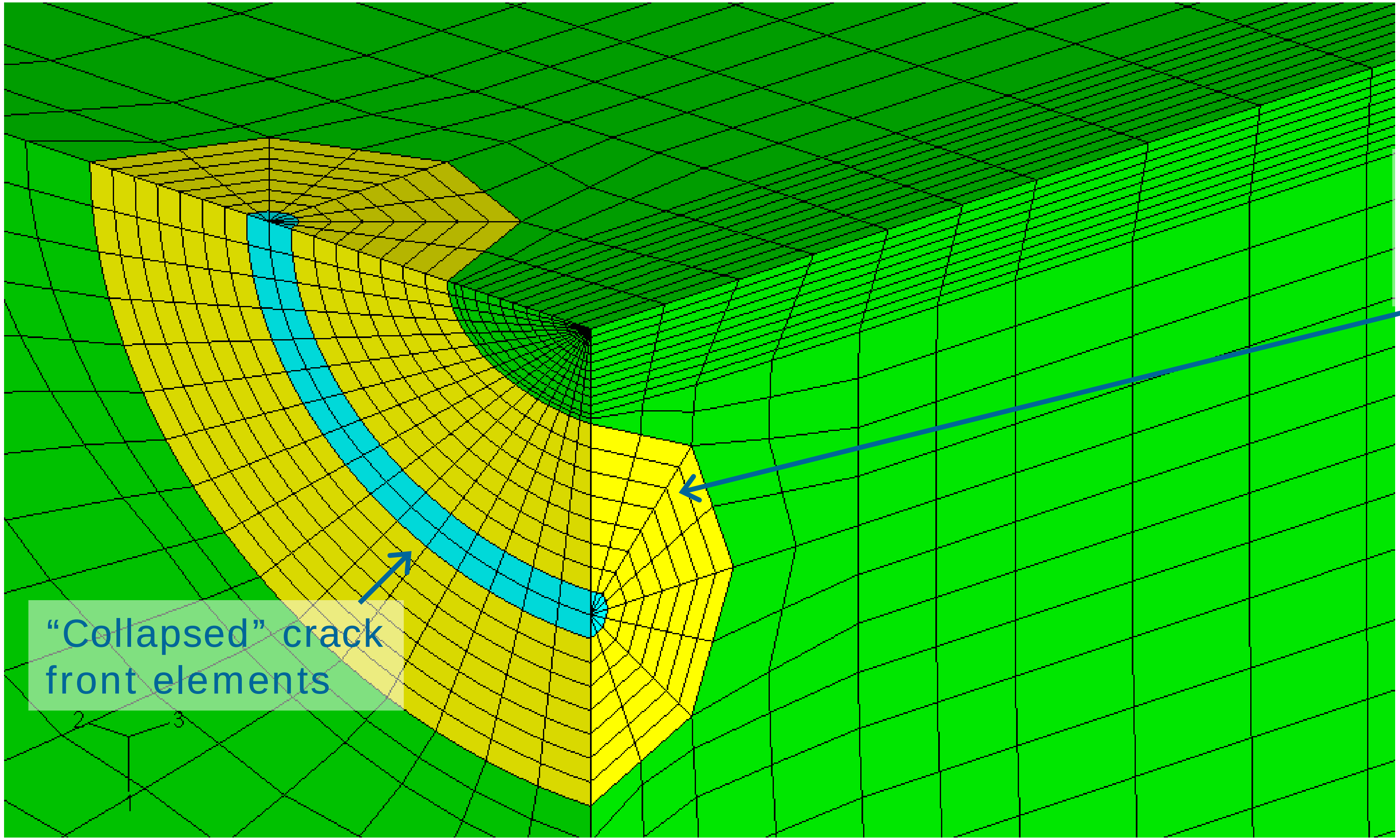
Cut-away at the mid-section showing the corner crack that must be analysed

A crack in a 3D finite element mesh



Typical finite element mesh for an elliptic crack

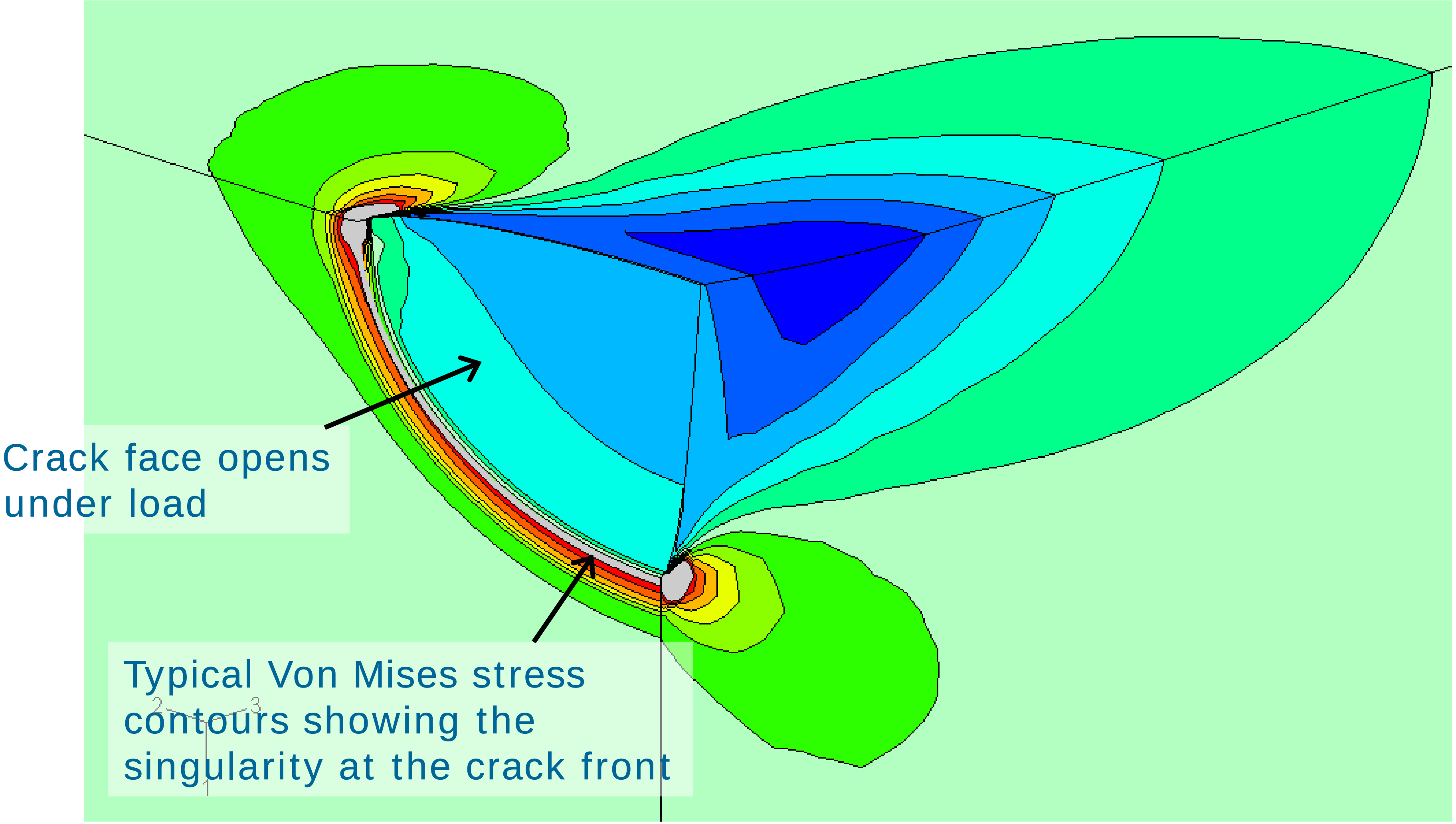
A crack in a 3D finite element mesh



Focused mesh of rings of elements around the crack front

Typical finite element mesh for an elliptic crack

A crack in a 3D finite element mesh



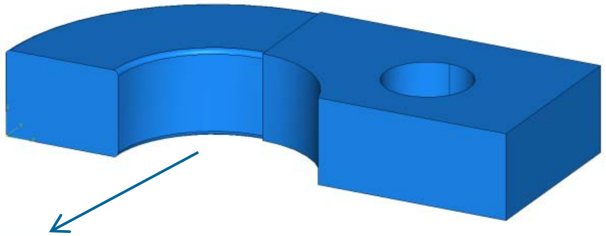
Typical finite element mesh for an elliptic crack

How does Zencrack work ?

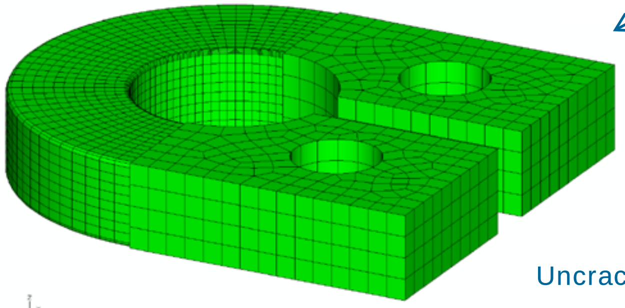
1. Mesh of intact component

The user must supply Zencrack with a mesh of an intact component e.g. an Abaqus input file (.inp):

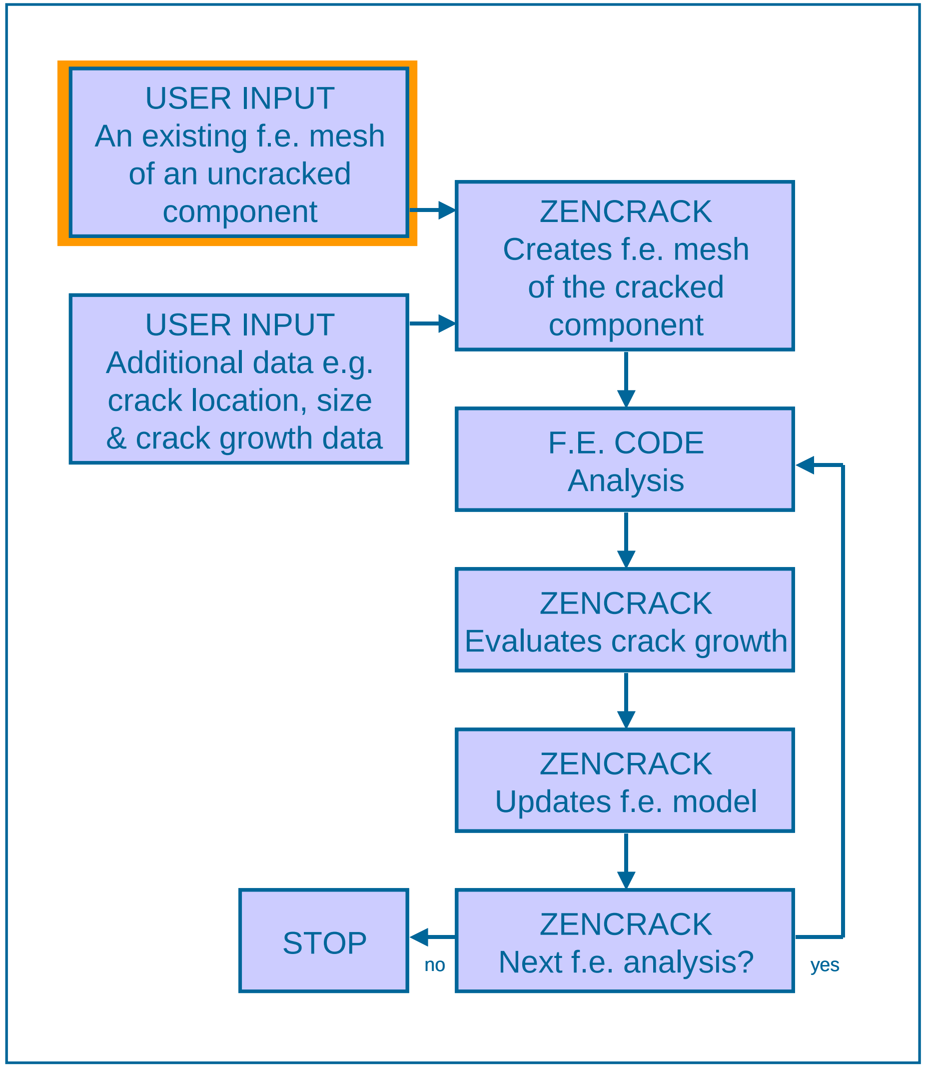
- any suitable pre-processor
- 8 or 20 noded bricks where cracks are required



Geometry model



Uncracked mesh



Simplified flowchart

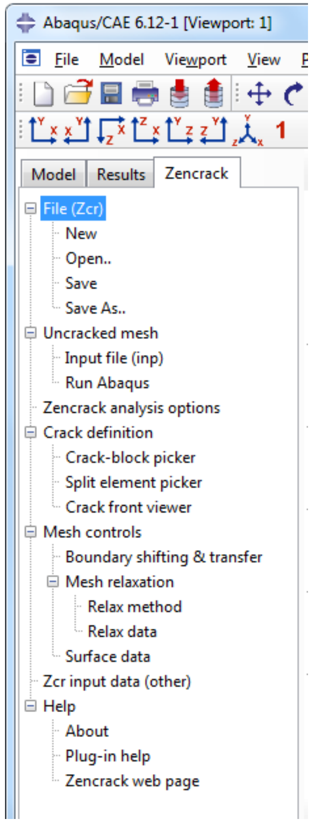
How does Zencrack work ?

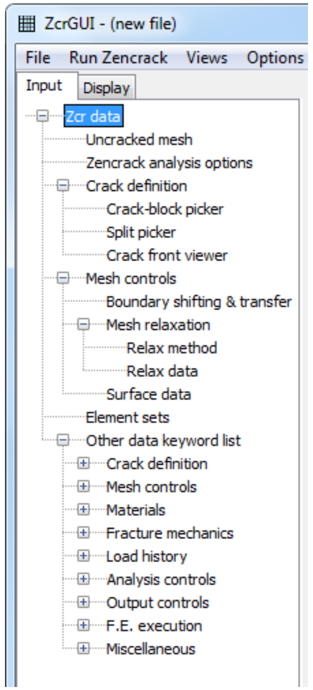
2. Zencrack input file

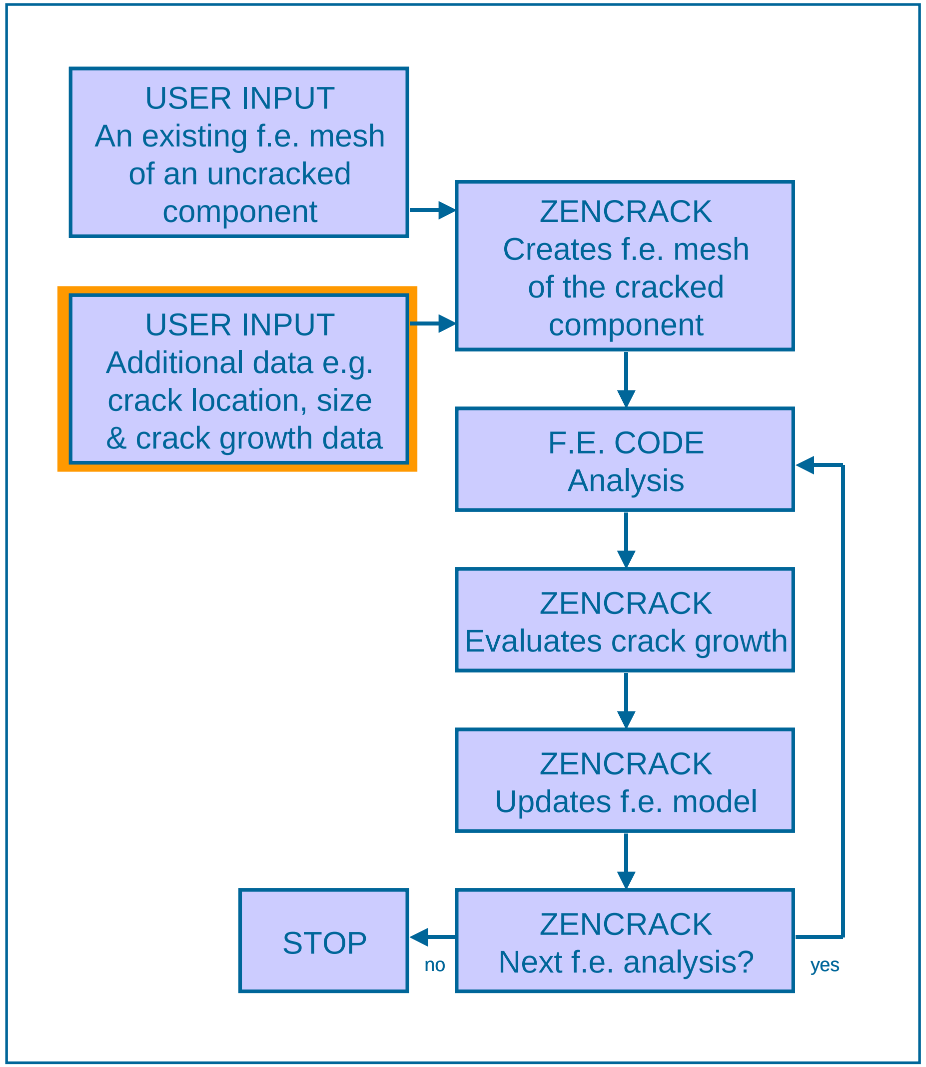
This includes the required location, orientation and size of the initial crack front.

For crack growth prediction additional data such as the crack growth law data is required.

Creation of the input file via plug-ins for Abaqus/CAE is now replaced by the Zencrack GUI.





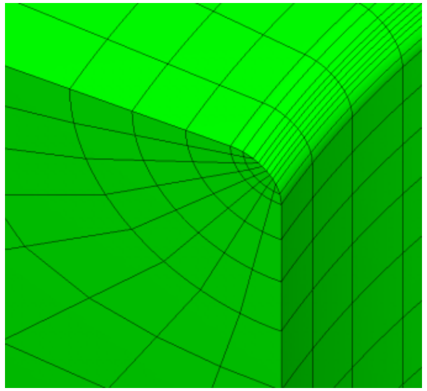


Simplified flowchart

How does Zencrack work ?

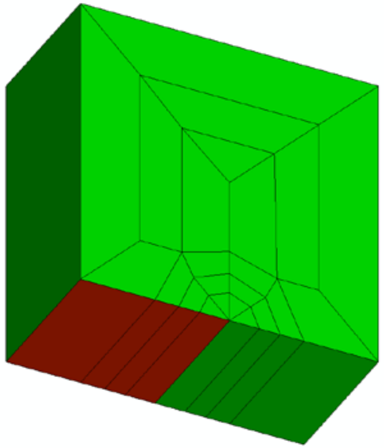
3. Zencrack creates mesh with initial crack

Target elements in the uncracked mesh are replaced by “crack-blocks” to generate the desired initial crack front(s)



Uncracked mesh

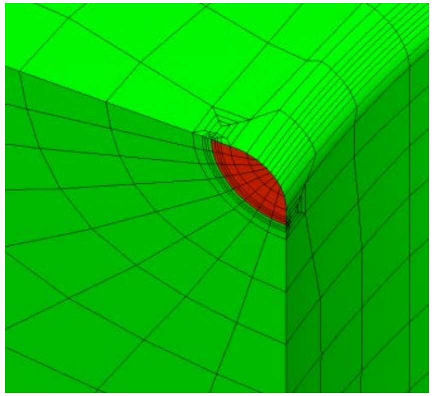
+

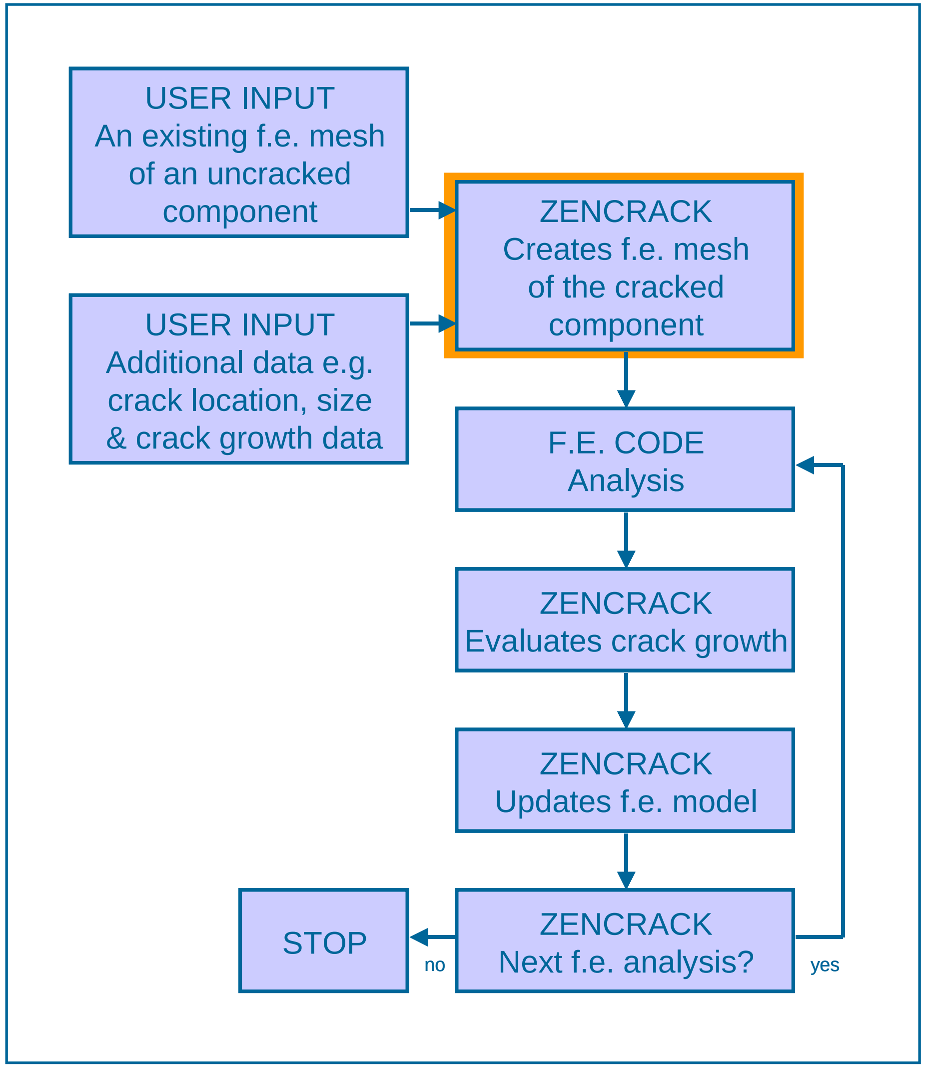


Crack-block(s)

→

Cracked mesh





Simplified flowchart

How does Zencrack work ?

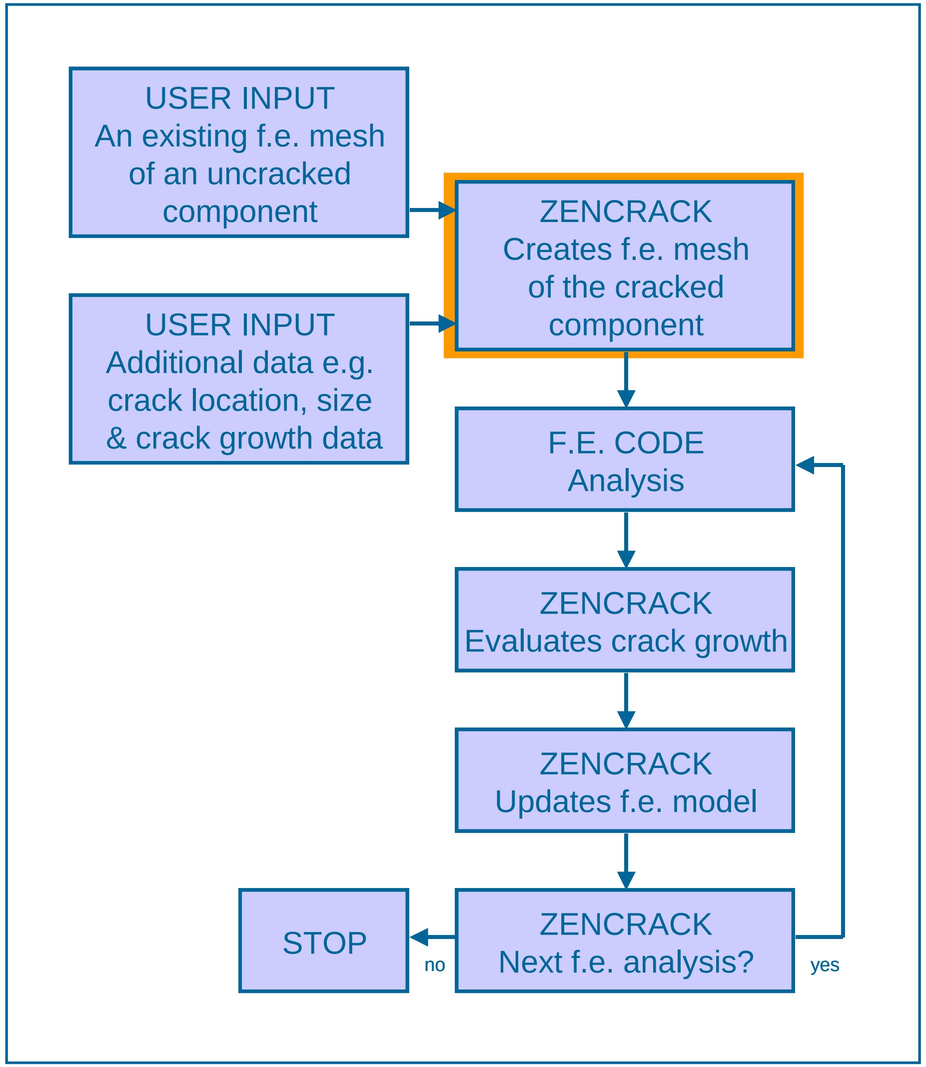
3. Zencrack creates mesh with initial crack

Boundary conditions, pressure loads, temperatures and surfaces are updated near the crack front if necessary

i.e. updates to keywords including:

*SURFACE
*DLOAD
*BOUNDARY
*TEMPERATURE
*INITIAL CONDITIONS, TYPE=TEMPERATURE

Contour integral and displacement requests at the crack(s) are generated for the new mesh



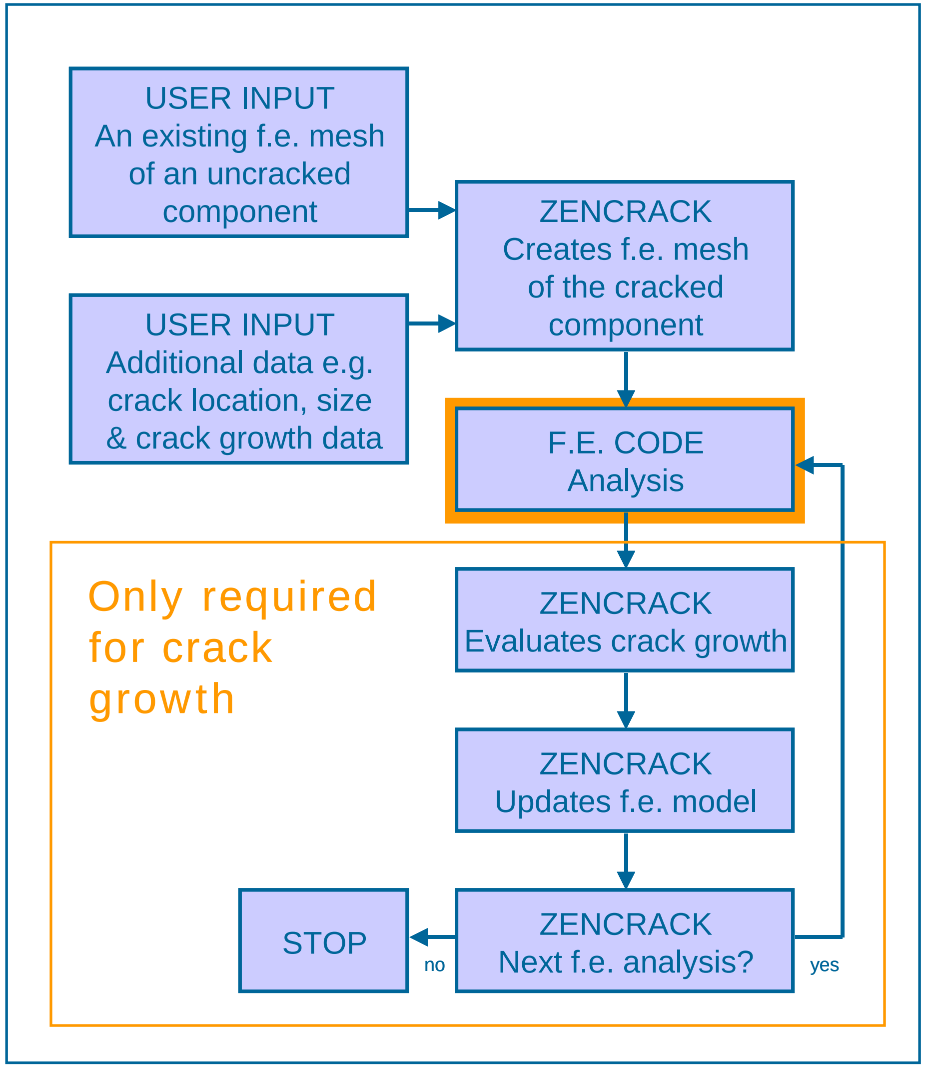
Simplified flowchart

4. Finite element analysis

The cracked component mesh is automatically submitted for analysis in Abaqus/Standard.

When the f.e. run completes, results are extracted from the .odb file and processed to allow reporting of fracture mechanics parameters for the crack(s).

This is the last step if the user only wants to evaluate stress intensity factors or the energy release rate for a given crack.



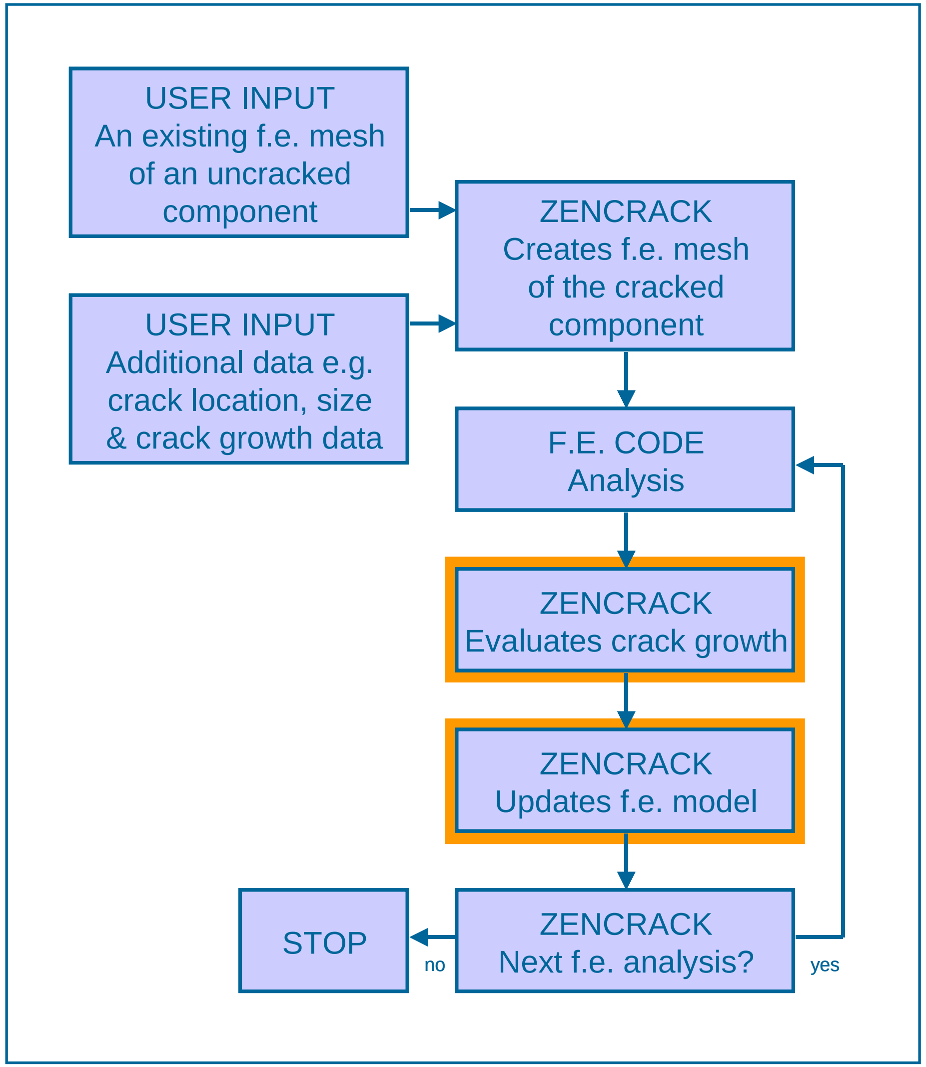
Simplified flowchart

5. Crack growth prediction

If crack growth is to take place the information from the f.e. analysis is combined with cyclic load history data and the crack growth law given by the user.

An integration scheme calculates an updated position for each crack front.

The mesh is updated to contain crack(s) in the new position(s).

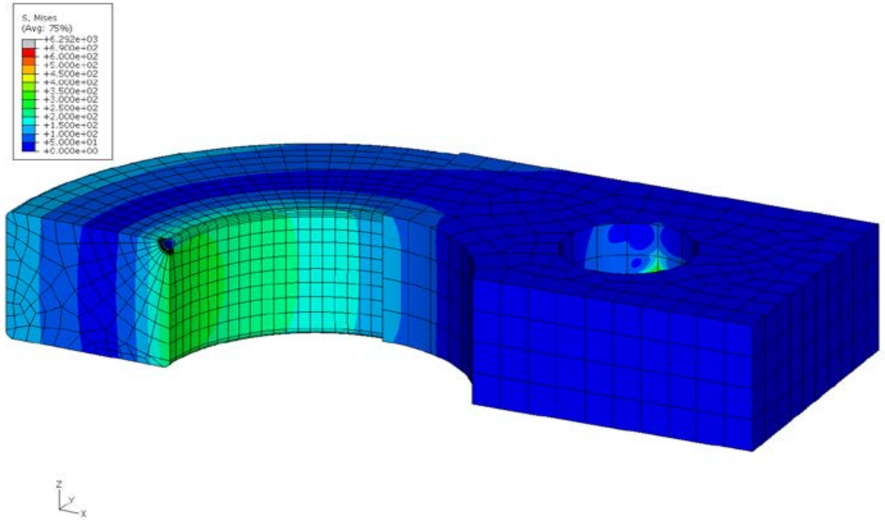


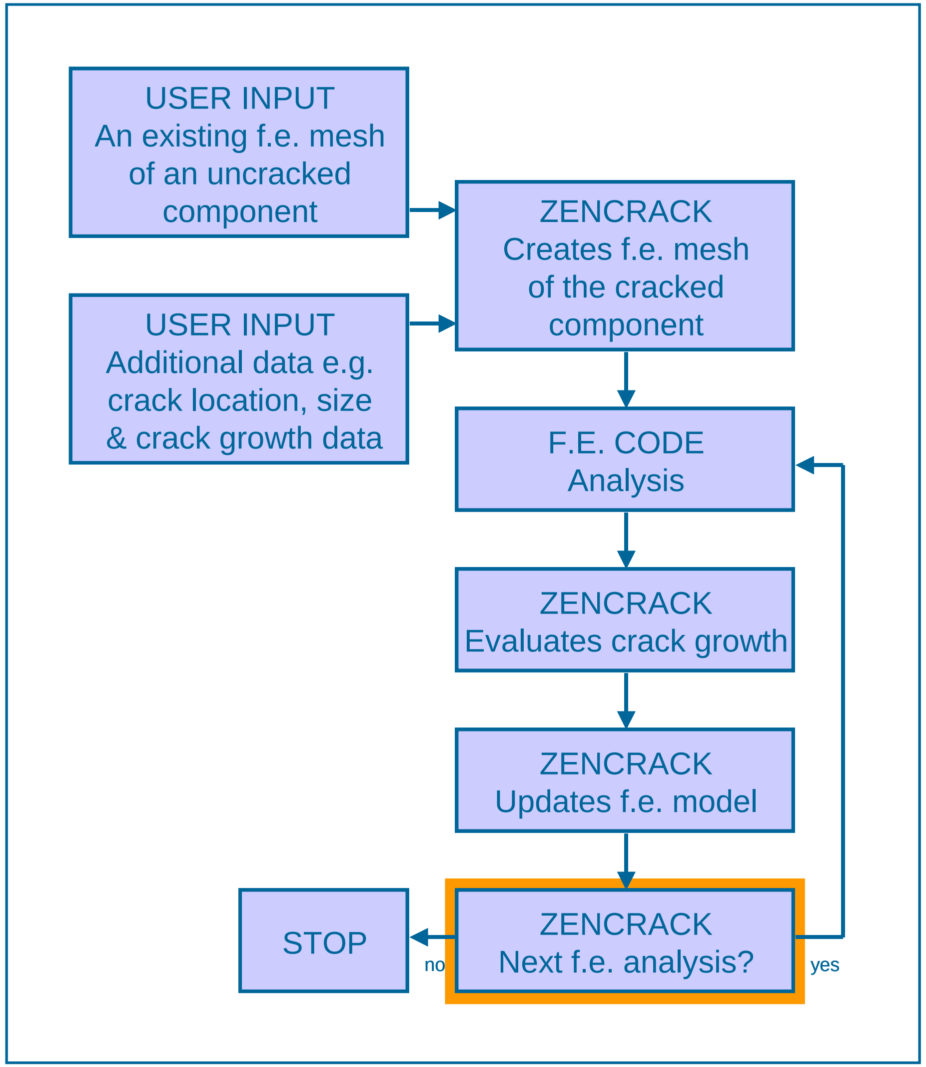
Simplified flowchart

6. Another f.e. analysis ?

If limits defining the end of the analysis have not been reached, a new finite element job is submitted.

Analysis and update is repeated until user defined limits or failure are reached e.g Kic has been reached.

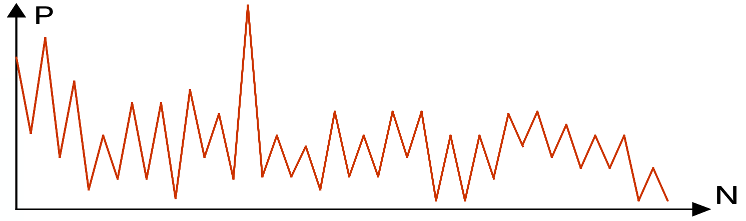


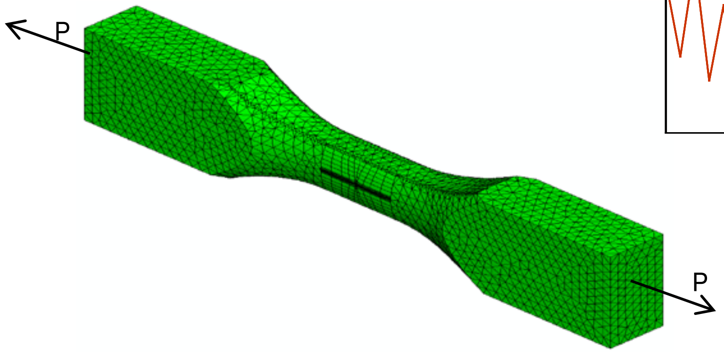


Simplified flowchart

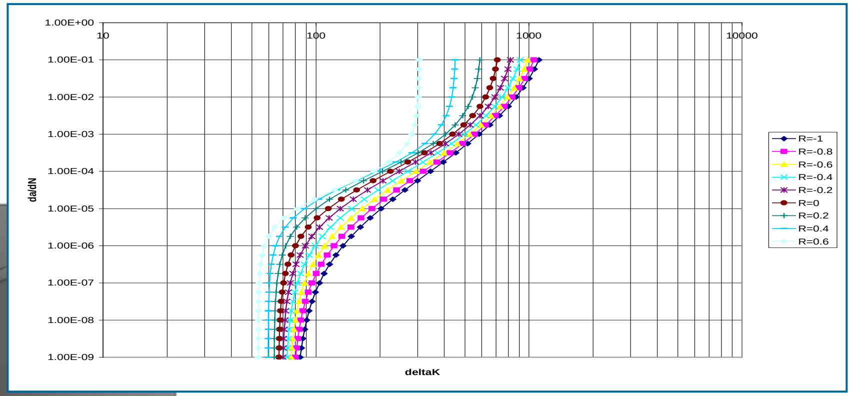
Overview of fatigue calculations

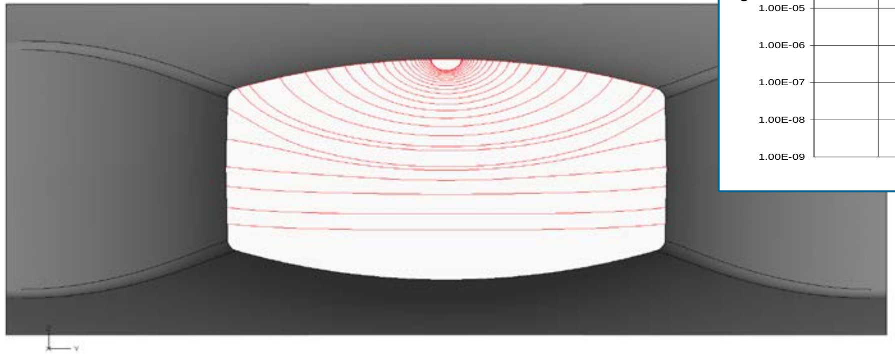
- A “linear elastic fracture mechanics” approach is most often used for crack propagation
  - Loading & history
    - » Calculate SIF range, ΔK
  - Crack growth law
    - » Relates ΔK to the growth rate, da/dn
  - Advance the crack by da over the next dn cycles
  - Each crack front node may have different “local” conditions allowing for generalised crack shape development





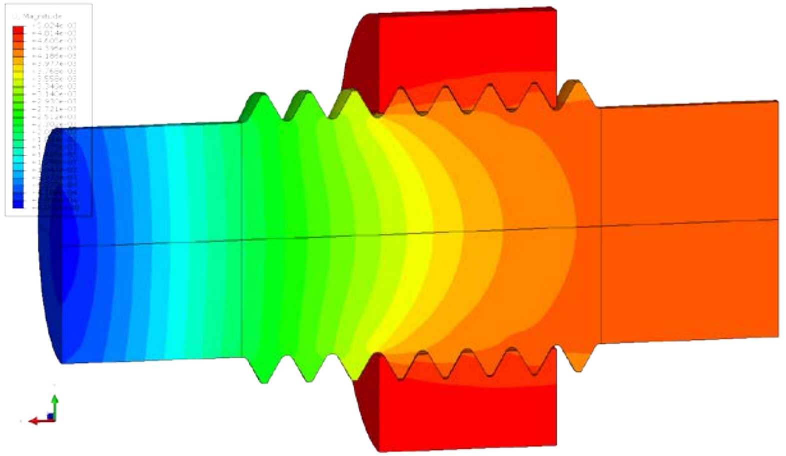
$$\frac{da}{dN} = f(\Delta K, R, T, f....)$$



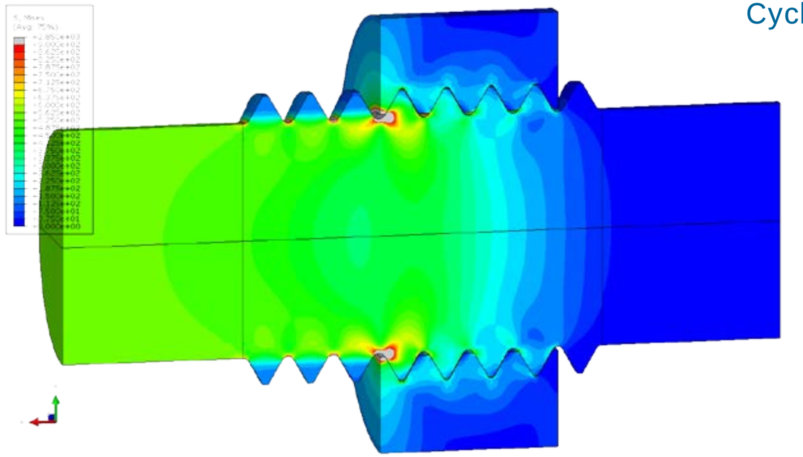


Load systems

- Many different load types may need to be analysed e.g.:
  - Thermal transients, internal pressure, aerodynamic loads, contact forces
- The f.e. analysis may have
  - A single step with one increment
  - Multiple steps each having multiple increments



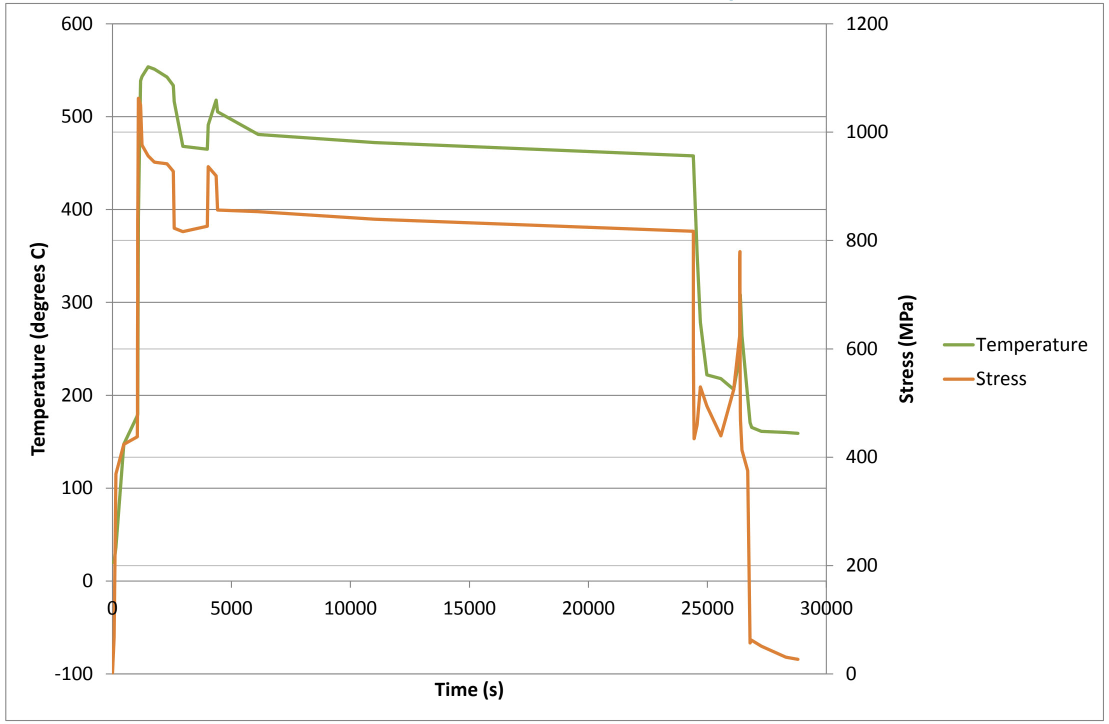
Threaded connection
Cyclic load on nut

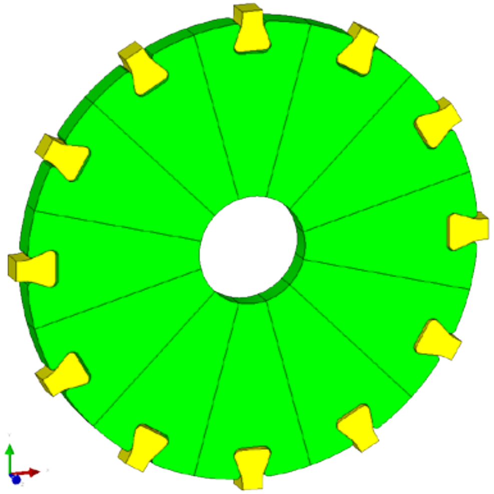


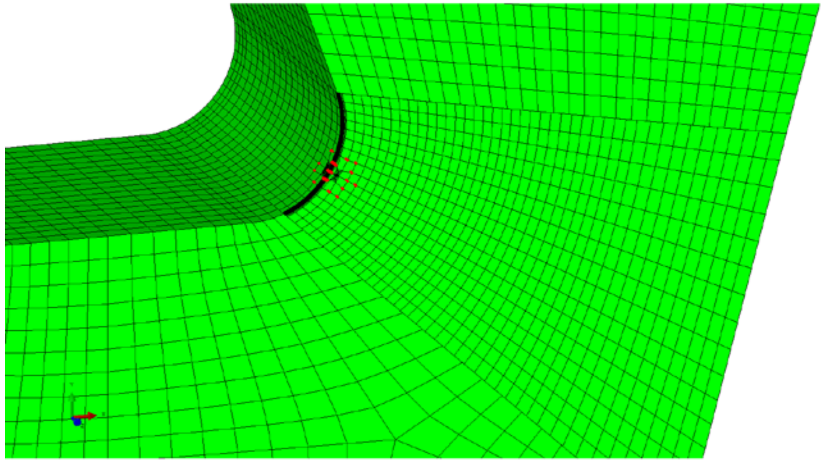
- Zencrack uses a “load system” methodology to relate the f.e. results to the load history needed for crack growth prediction

- The load system approach is extremely powerful and flexible
- It can handle simple cases such as constant amplitude loading through to general load cycles in which stresses and temperatures may be varying out-of-phase
- Fatigue and/or time dependent load scenarios can be considered

Typical aero engine data with out-of-phase stress and temperature time histories

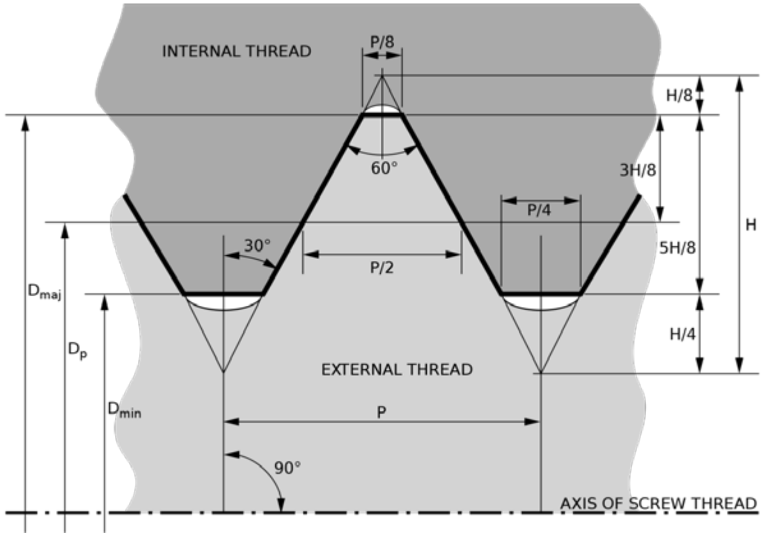


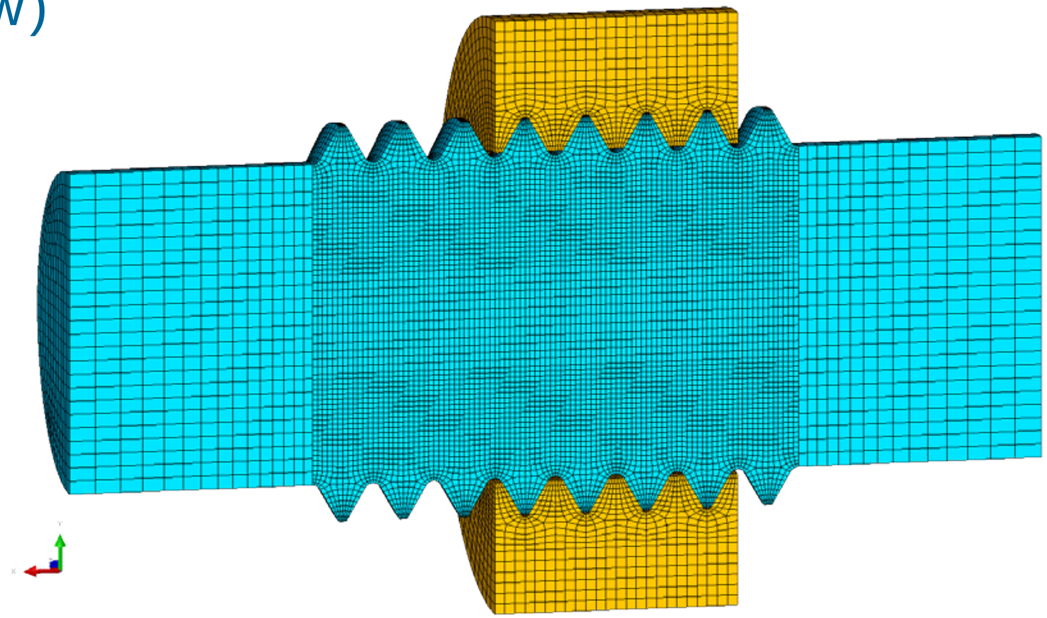




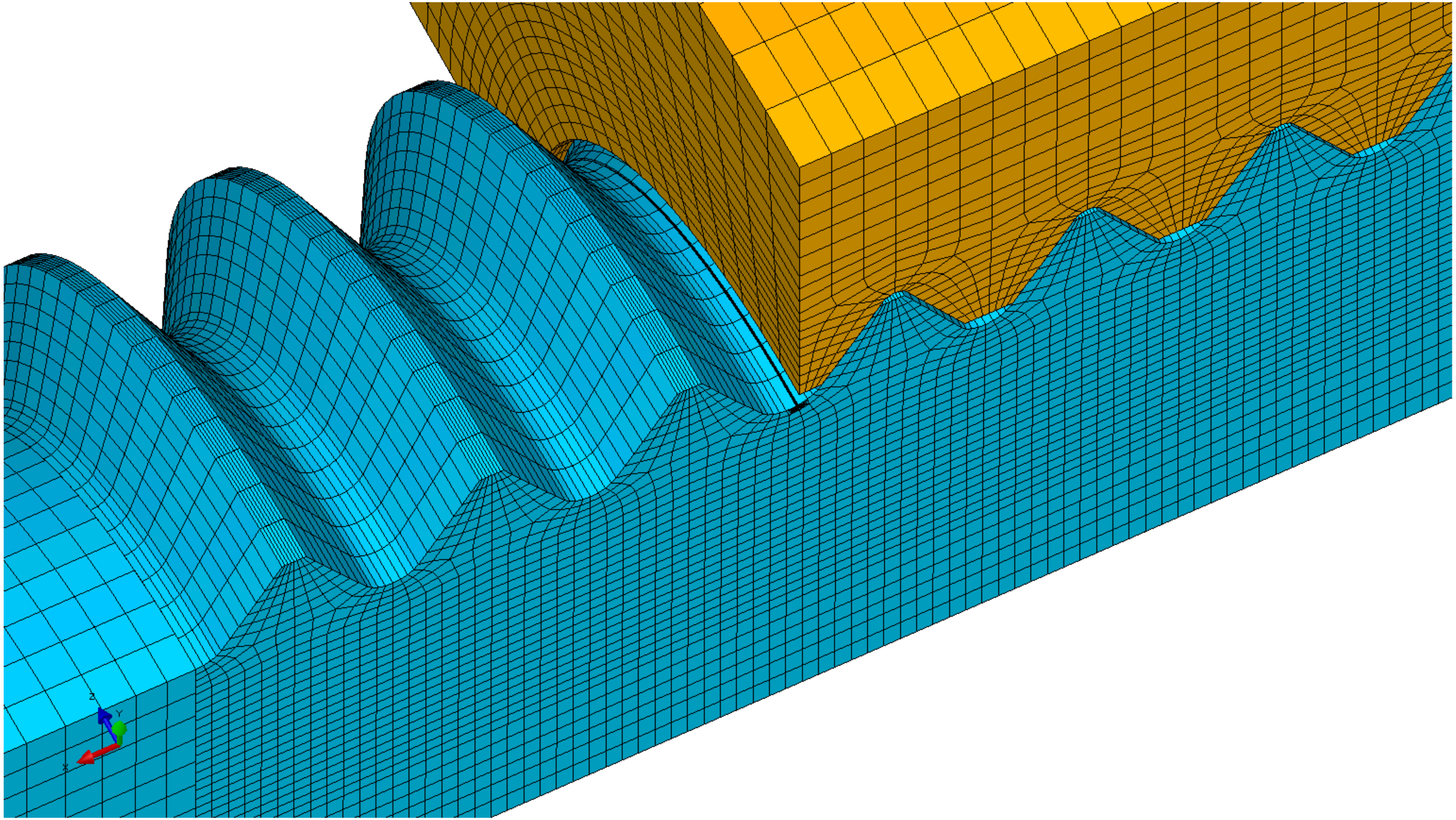
Example – threaded connection

- For this example, the geometry is defined for a coarse (UNC) 0.5 inch major diameter thread
  - the major diameter is Dmaj=0.5”, pitch is P=1/13”
- The model is a simplified one in which the circumferential effect of the pitch is not included
- A 90° sector is modelled with the crack extending around the full 90 degrees of the model (180° expansion is shown below)

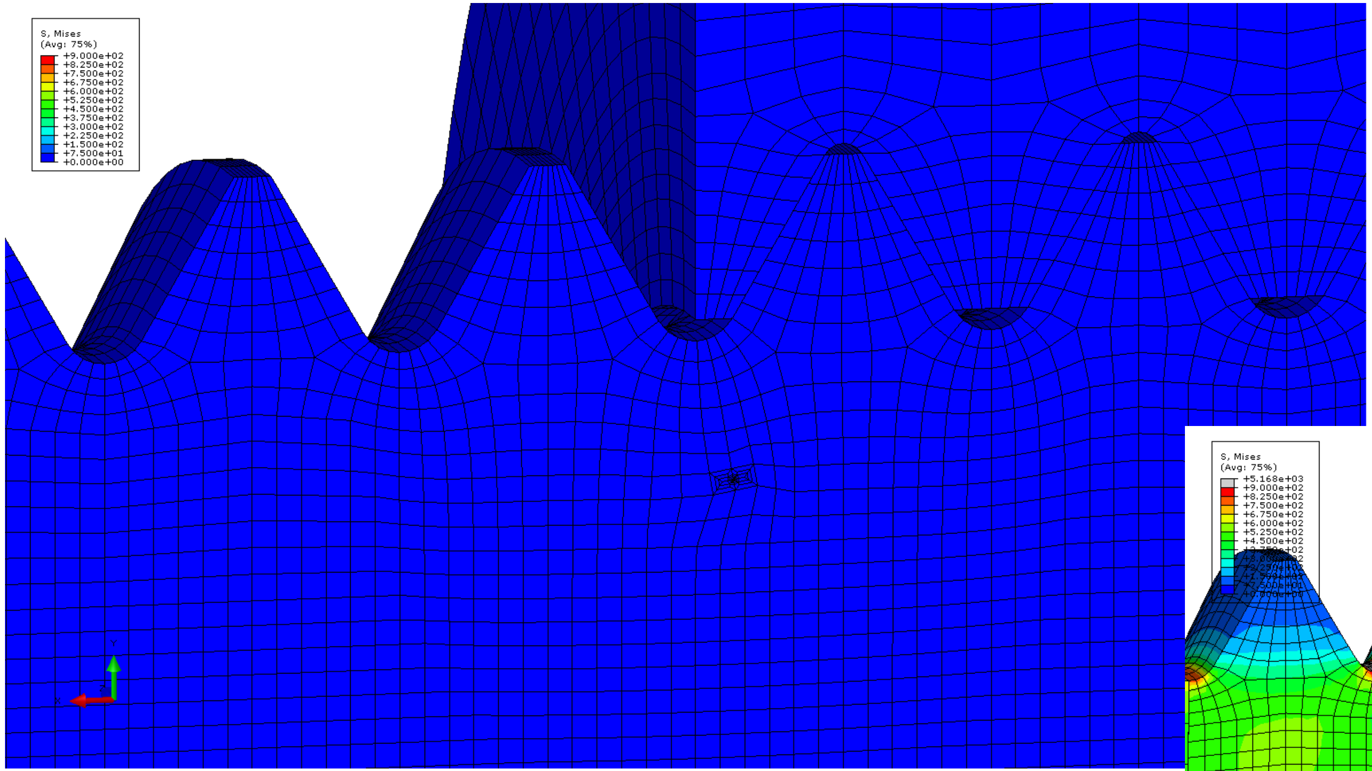




Example – threaded connection

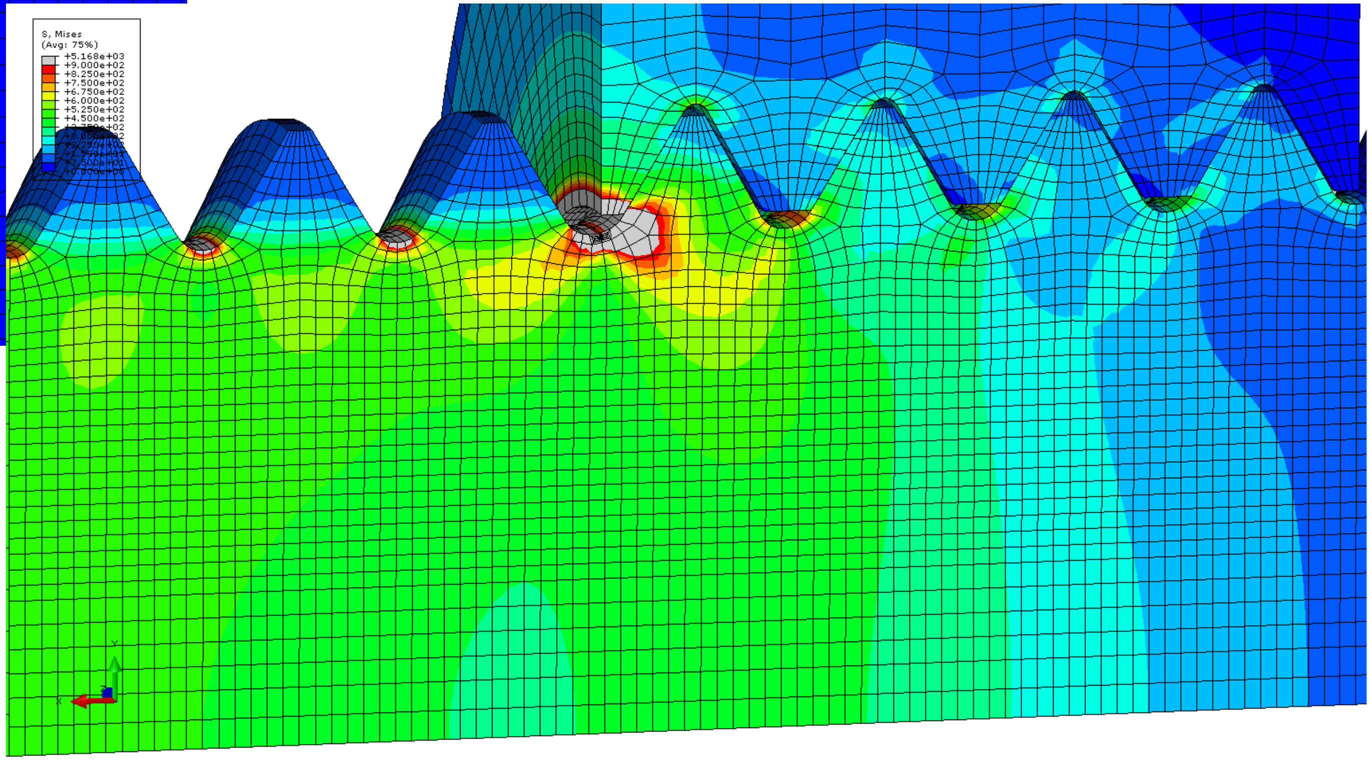


Example – threaded connection



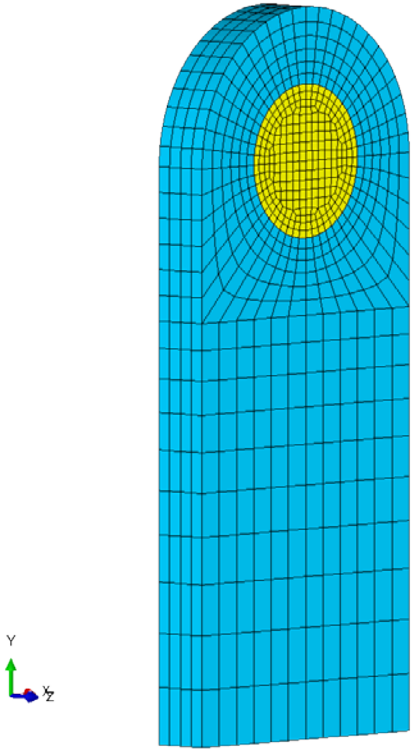
Contours of a complete load cycle for
one crack position during the growth sequence

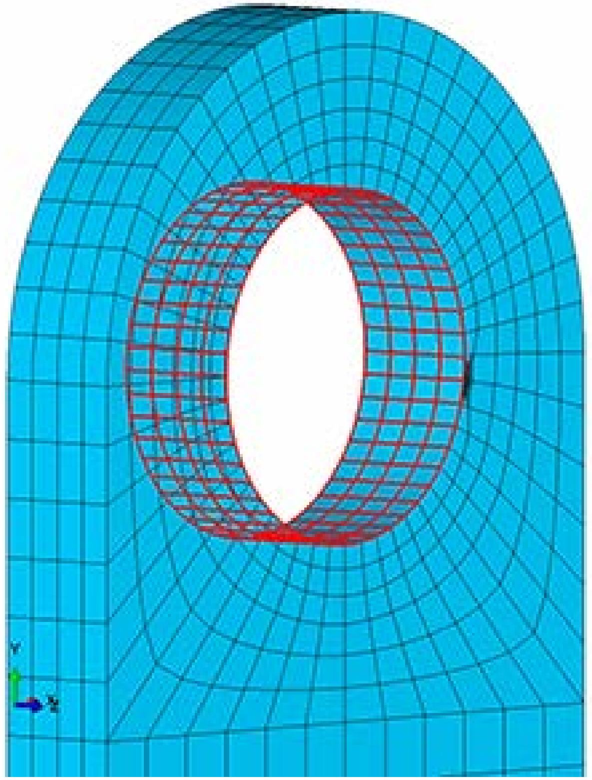
Contours of snapshots at the maximum
load condition as the crack grows



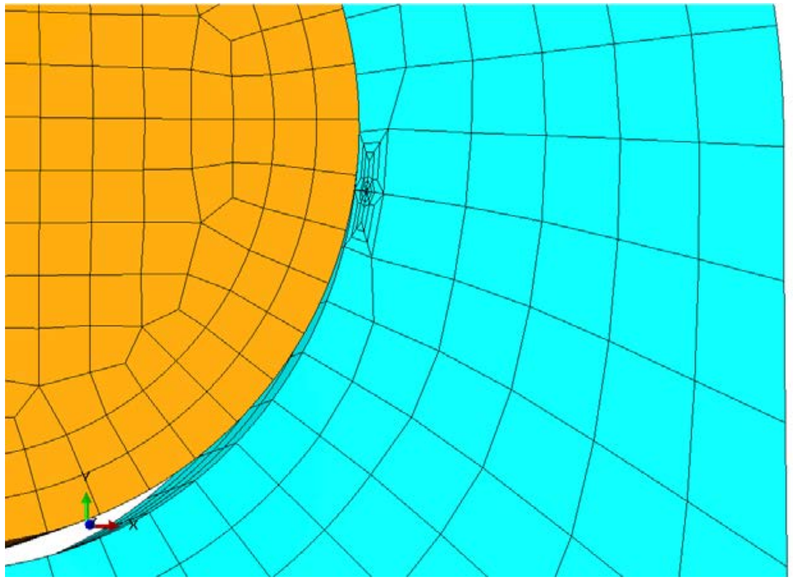
Example – pin / lug interaction

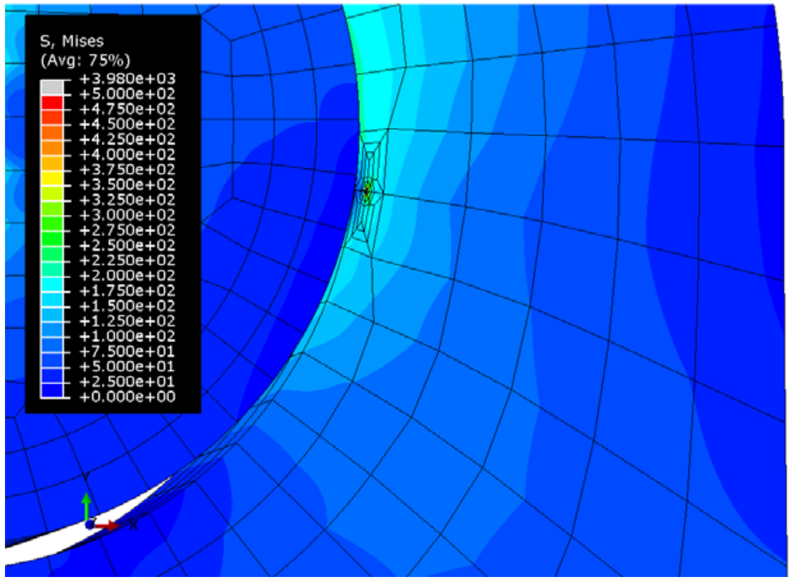
- Cyclic loading applied on a tight-fitting pin in a lug
- Initial crack is a through crack located in the lug



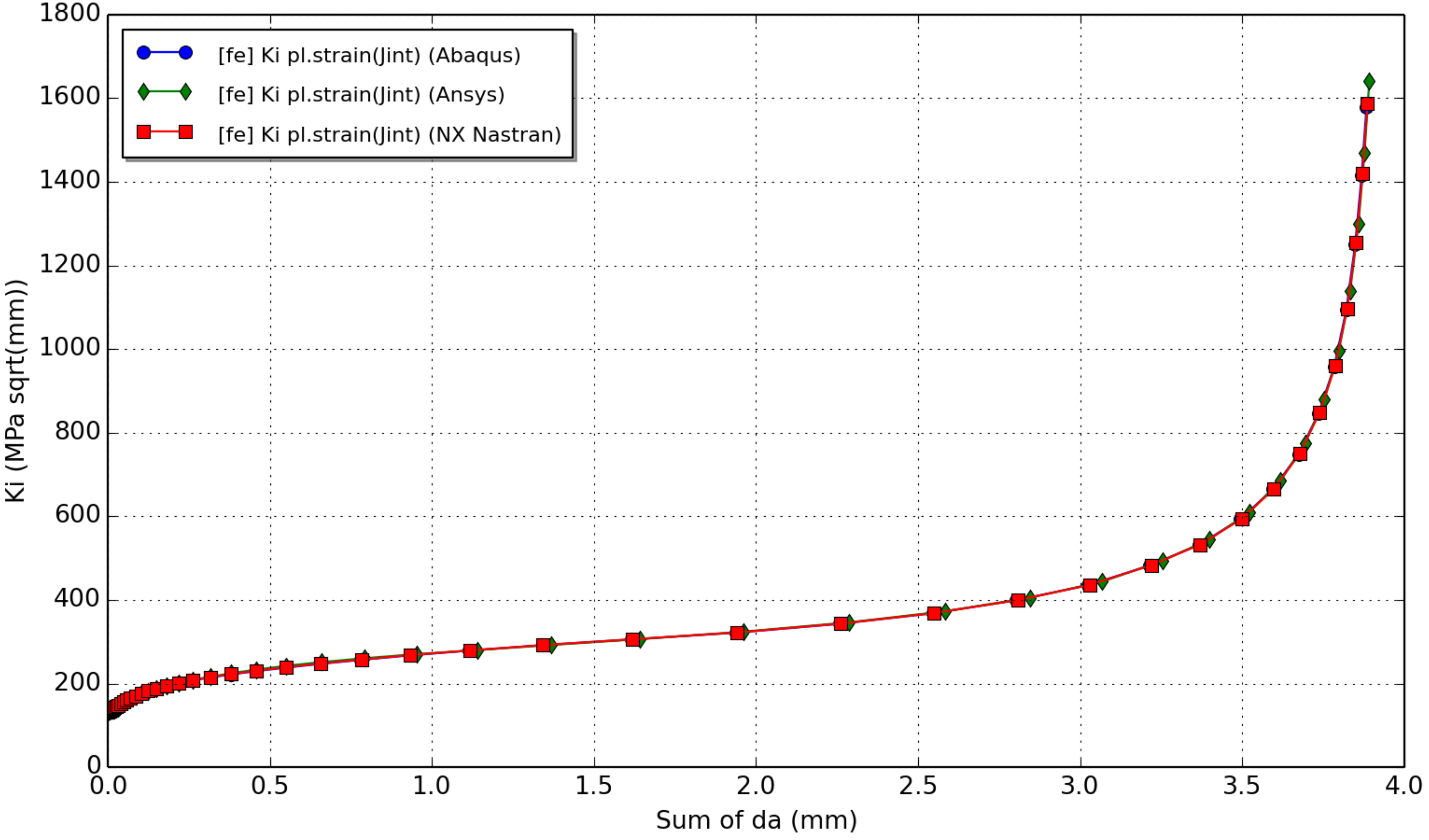


Contact surface updated on the lug side for the initial cracked mesh

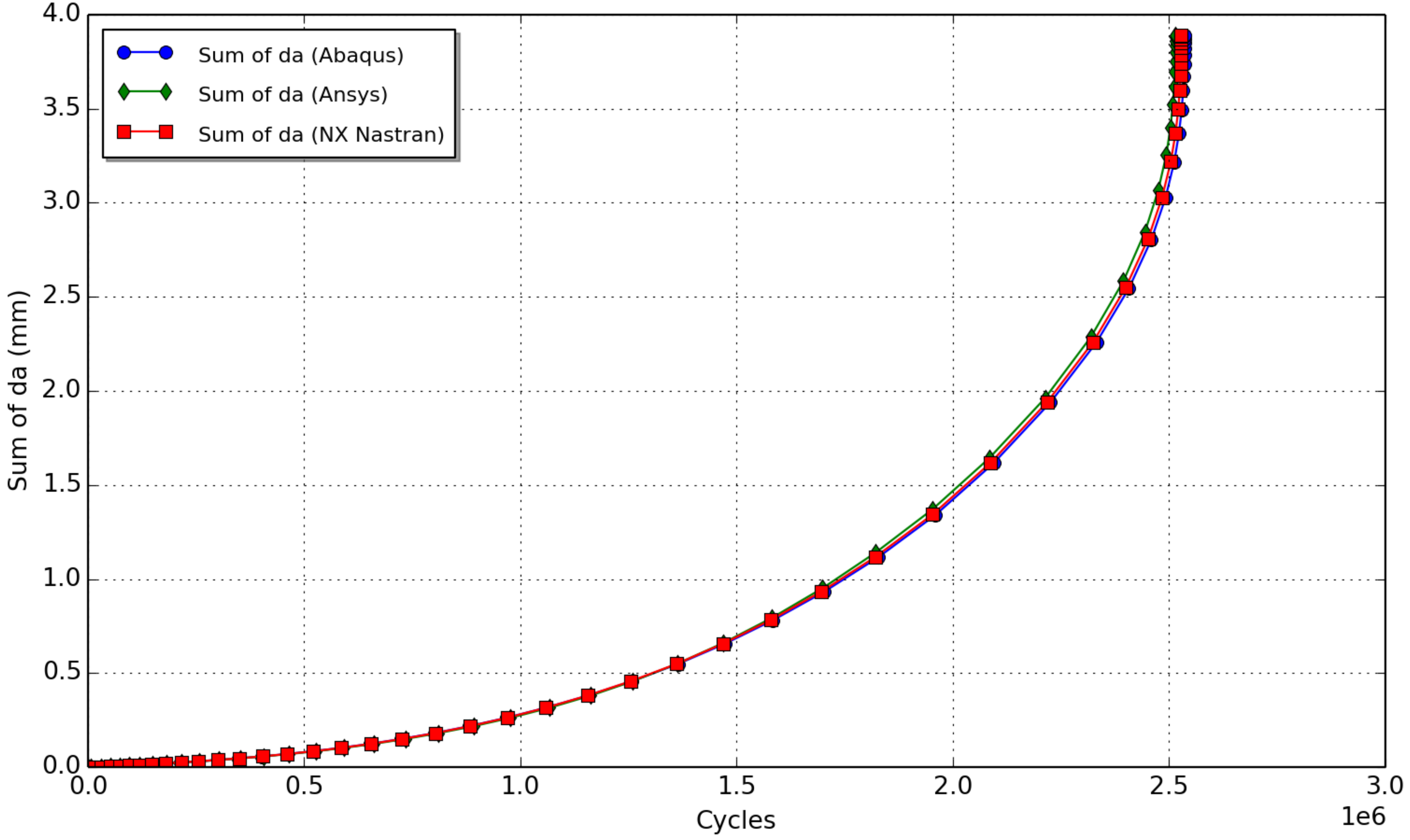




Example – pin / lug interaction

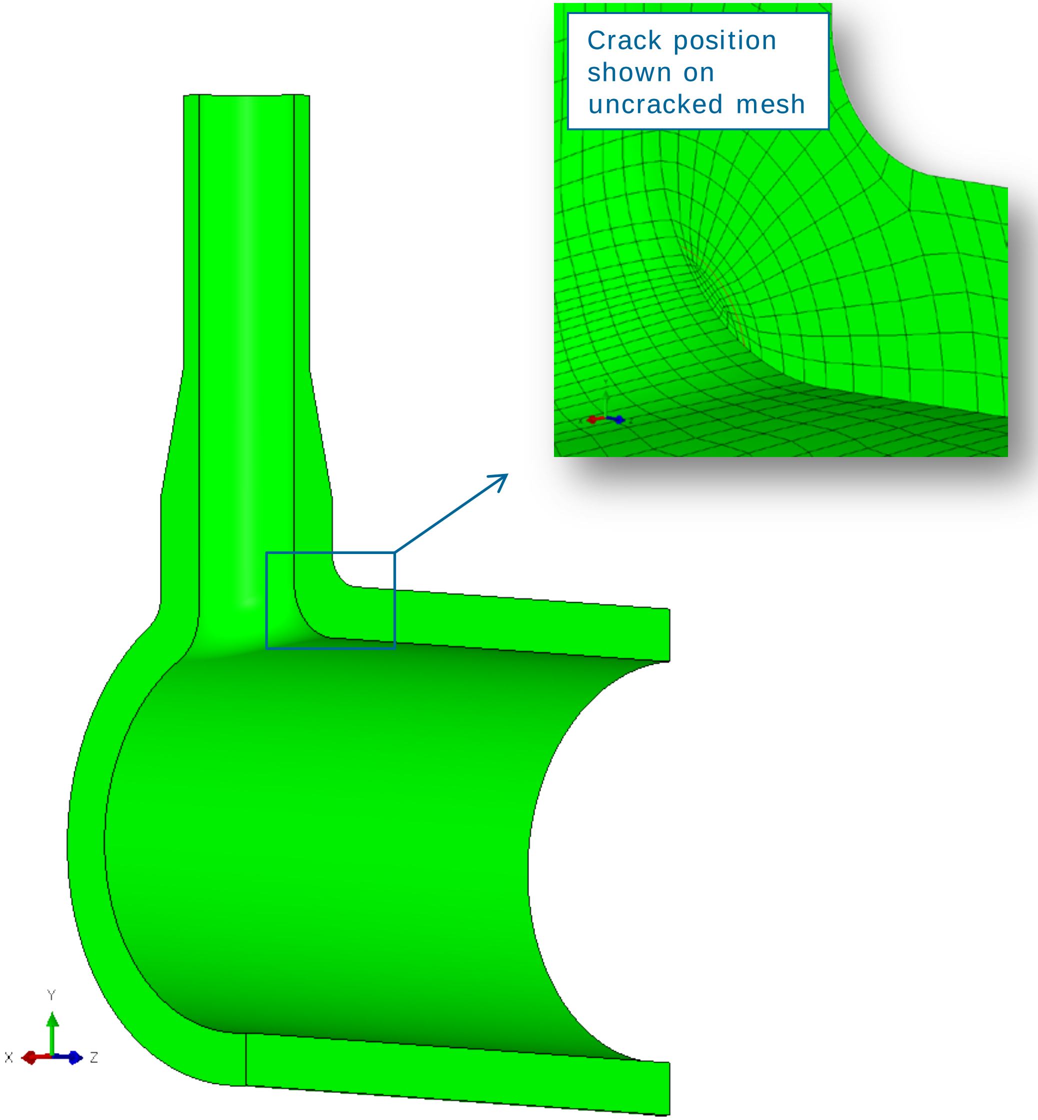


Example – pin / lug interaction



Example – header / nozzle intersection

- This example considers a crack in a pressurised header nozzle connection
  - Response to a pressurised thermal shock
  - Crack growth for a cyclic pressure load case
- The quarter symmetry model has the following main dimensions
  - Header
    - » OD 30”, wall $3^{1}/_{8}$”
  - Nozzle
    - » OD $10^{5}/_{8}$”, wall $1^{5}/_{16}$”



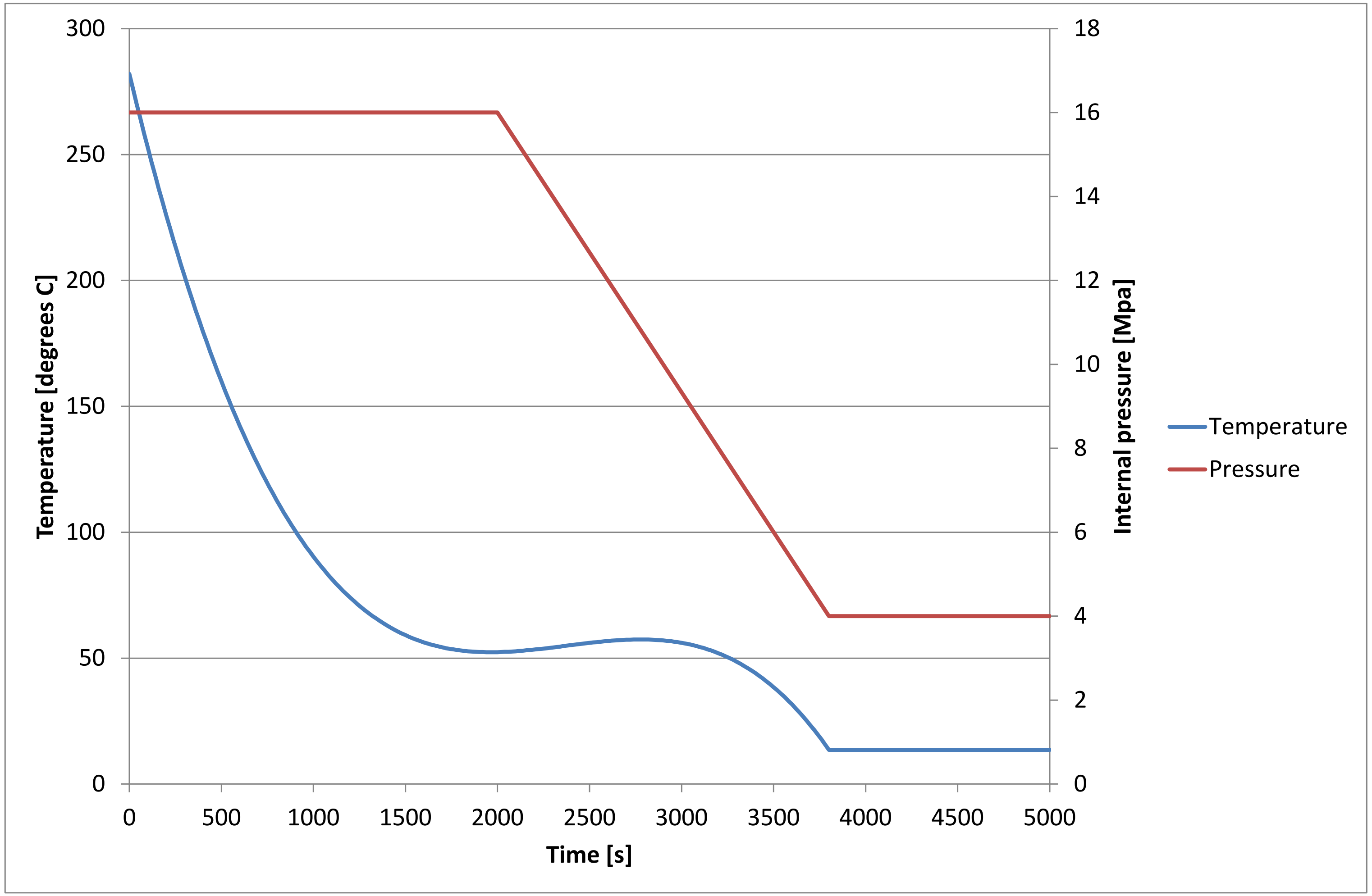
Example – header/nozzle intersection

- Temperature dependent steel data has been used
- The data for Young's modulus and Poisson ratio is also part of the Zencrack input data
- The transient analysis is carried out as a sequential thermal-stress analysis:
  - Heat transfer run to develop spatial temperature distribution as a function of time
  - Stress analysis using the heat transfer results as one of the inputs, plus pressure loading
  - Abaqus allows interpolation of temperatures from one mesh to a dissimilar mesh
    - » an uncracked model heat transfer analysis can provide the temperature input for the Zencrack stress analysis of the cracked component
    - » assumes the crack does not influence the temperatures

```
*Material, name=Steel
*Conductivity
 0.0518, 20.
 0.0503,100.
 0.0476,200.
 0.0445,300.
 0.0414,400.
*Density
 7.85e-06,
*Elastic
209000., 0.3, 20.
205000., 0.3,100.
199000., 0.3,200.
191000., 0.3,300.
181000., 0.3,400.
*Expansion, zero=20.
 1.15e-05, 20.
 1.17e-05, 50.
  1.2e-05,100.
 1.23e-05,150.
 1.27e-05,200.
  1.3e-05,250.
 1.33e-05,300.
 1.35e-05,350.
 1.38e-05,400.
*Specific Heat
460., 20.
480., 50.
500.,100.
520.,150.
530.,200.
540.,250.
560.,300.
570.,350.
590.,400.
```

Example – header / nozzle intersection

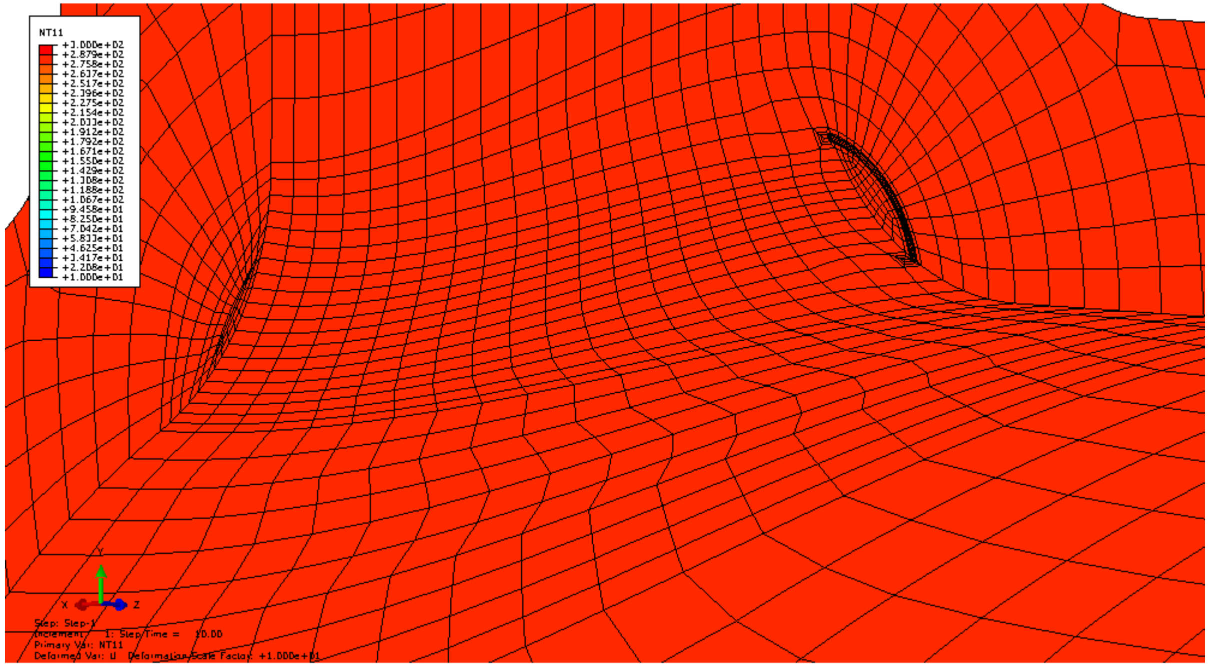
- Transient definition for pressurised thermal shock



- Internal temperature is defined in the heat transfer analysis using an Abaqus *AMPLITUDE definition.
- Internal and end cap pressures are defined during the stress analysis using an Abaqus *AMPLITUDE definition.
- Pressure load is also applied on the crack face.

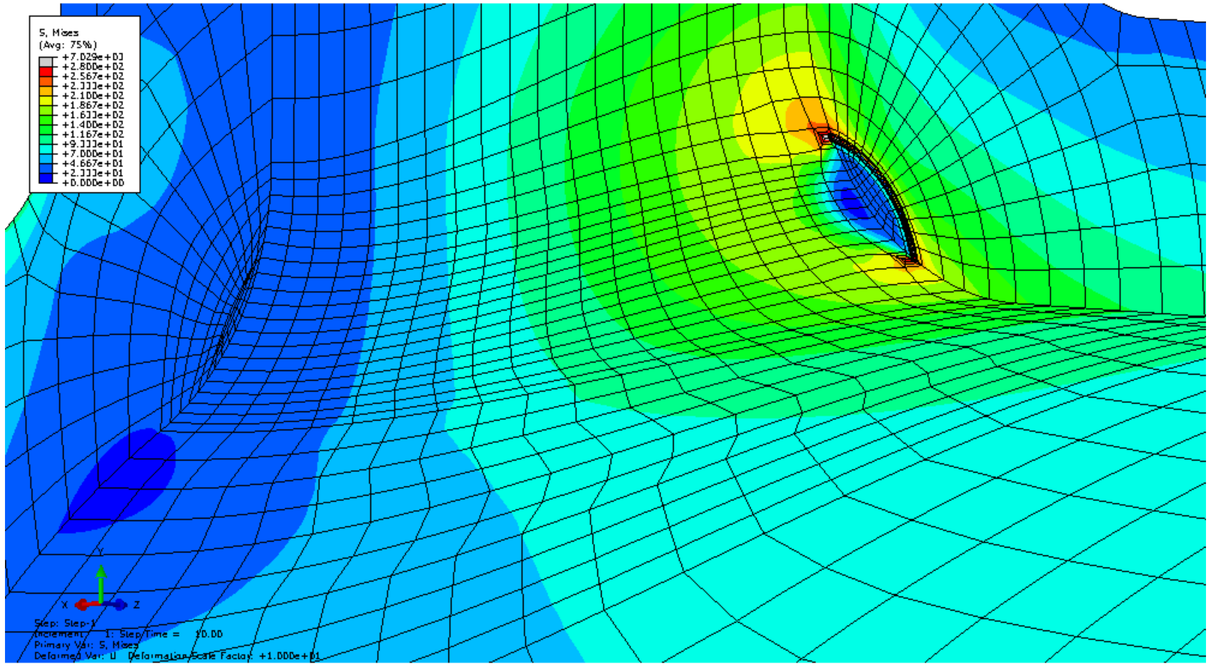
Example – header / nozzle intersection

Cracked model transient analysis

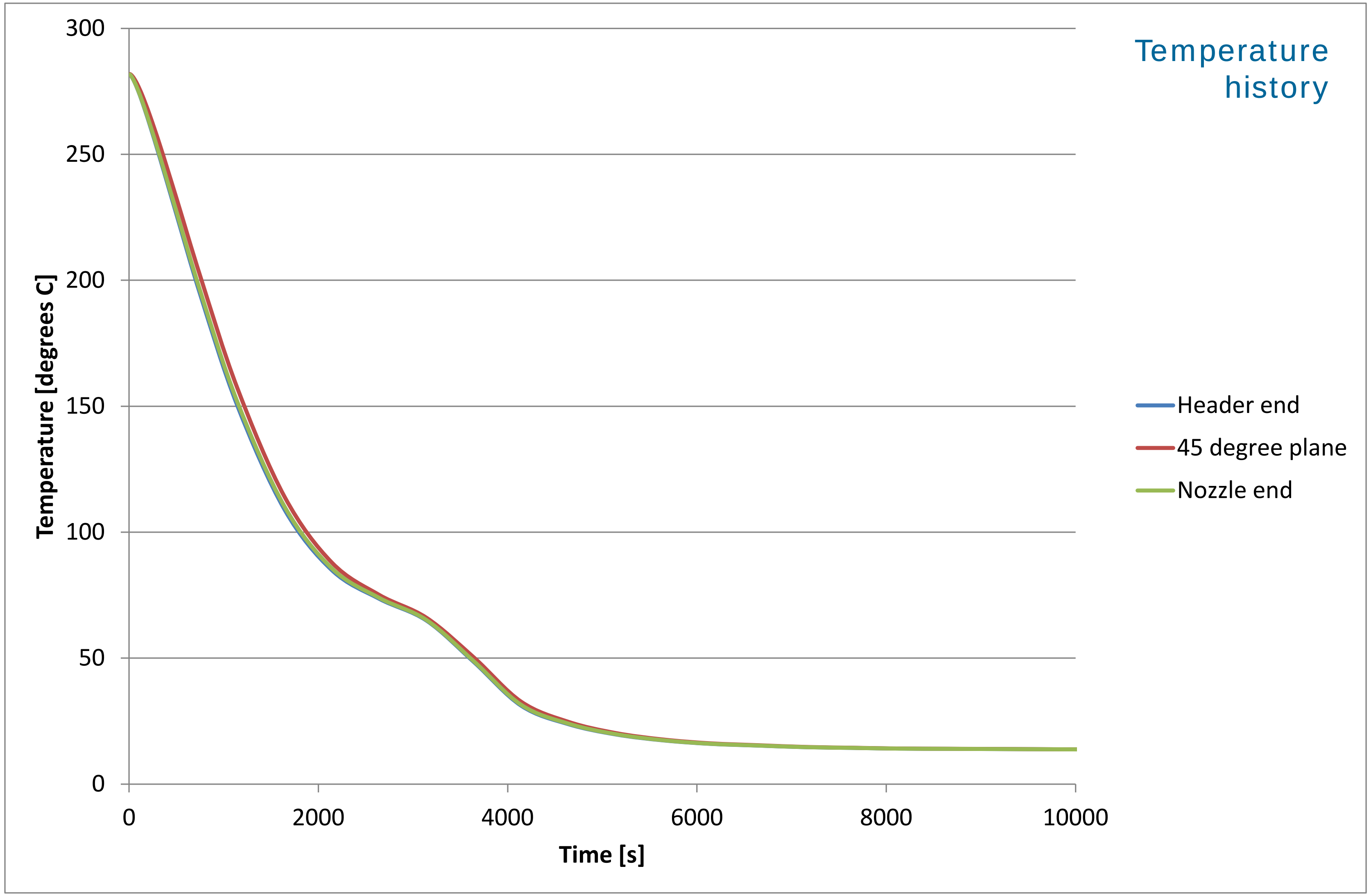


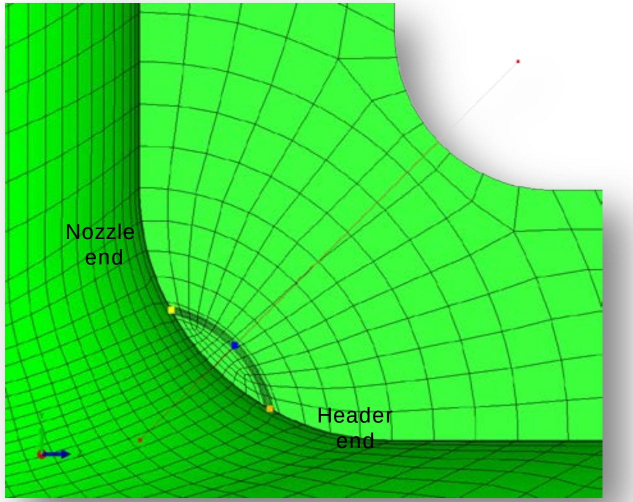
Temperature

Von Mises stress

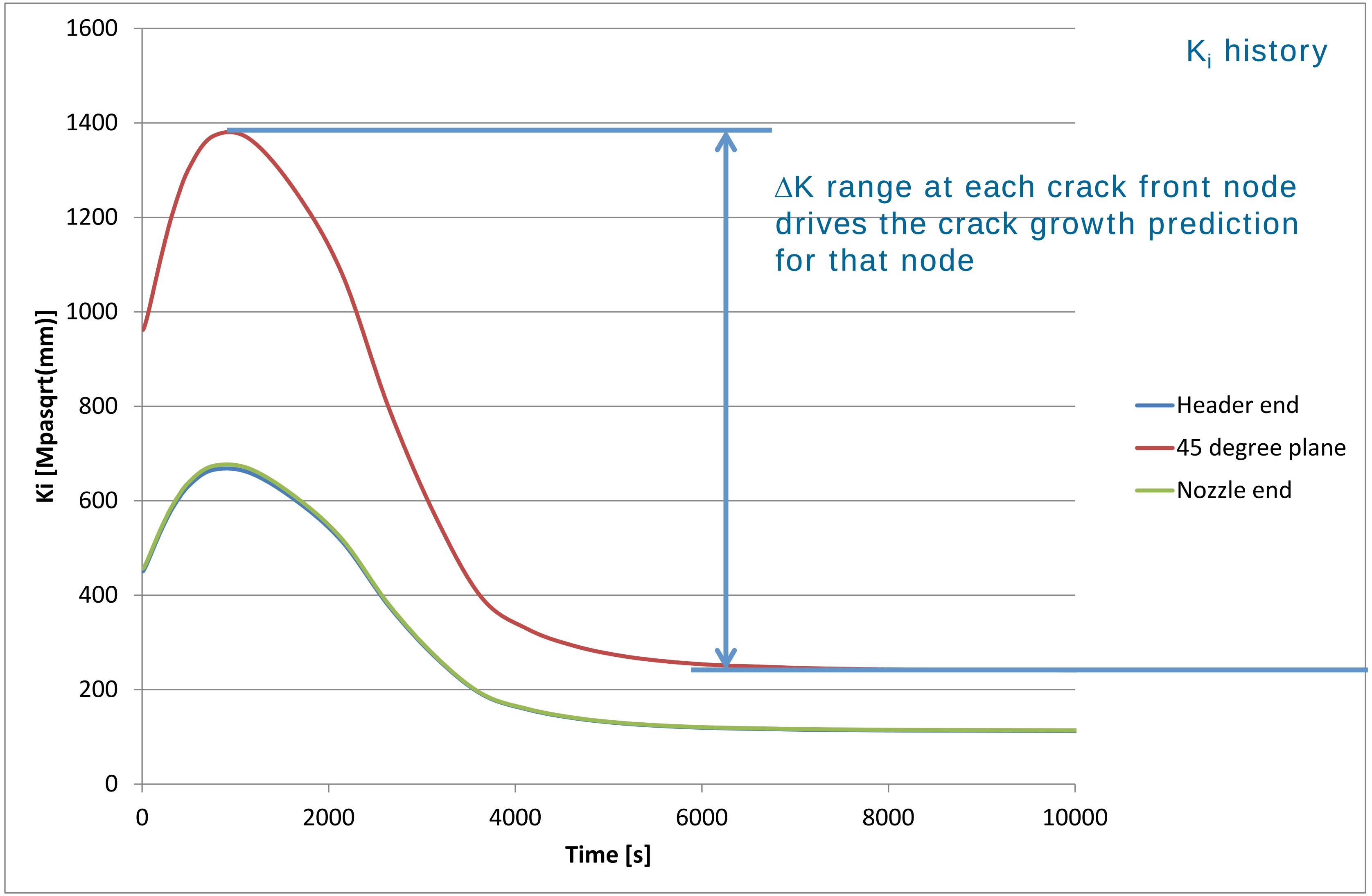


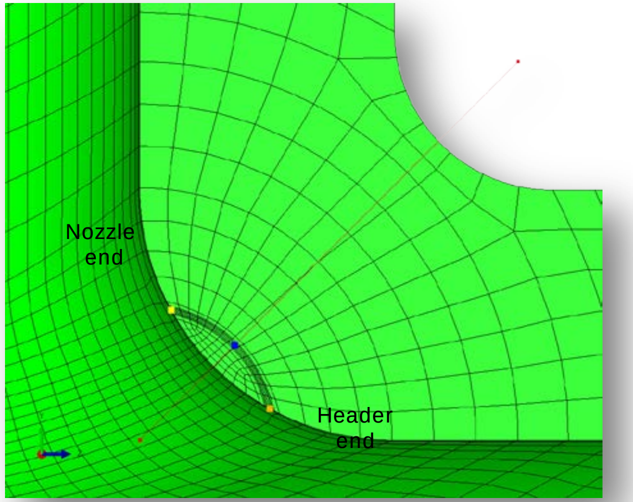
Example – header / nozzle intersection



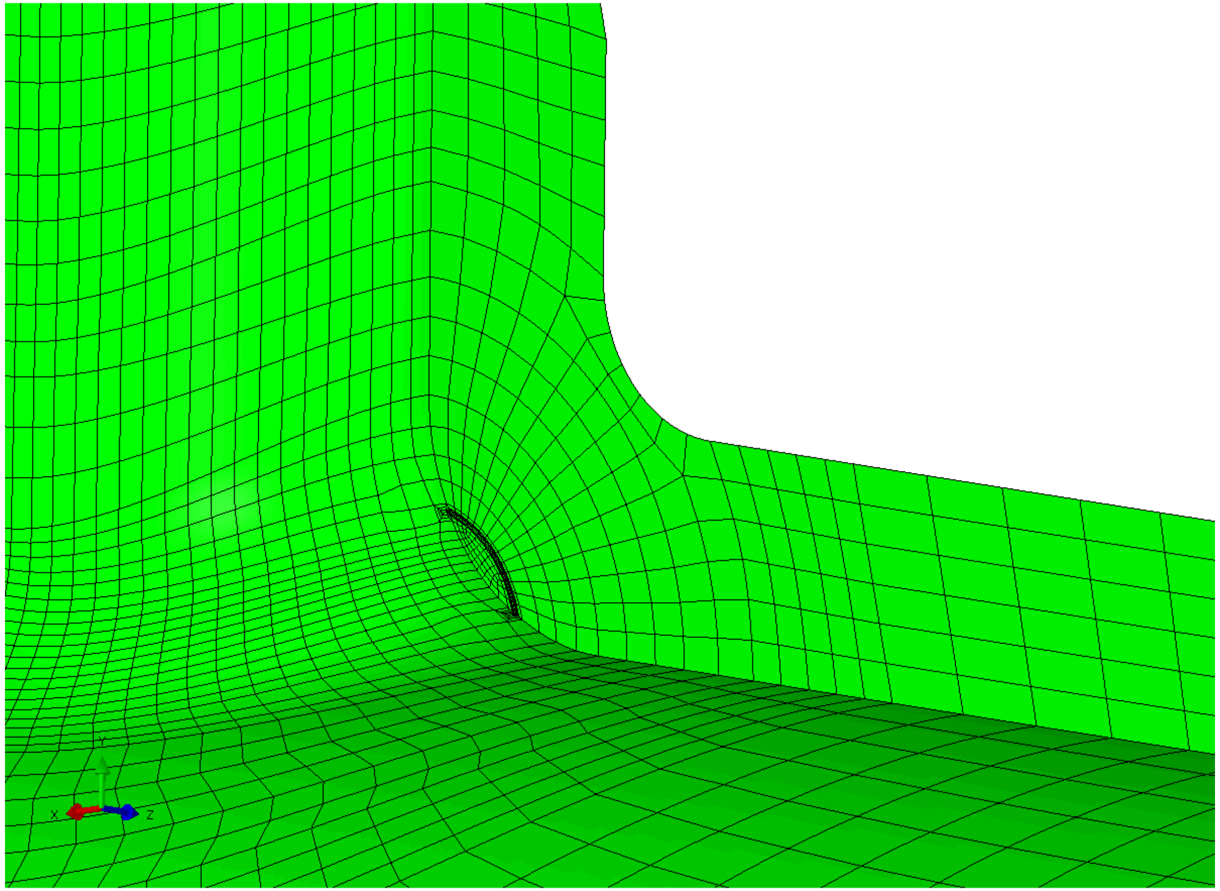


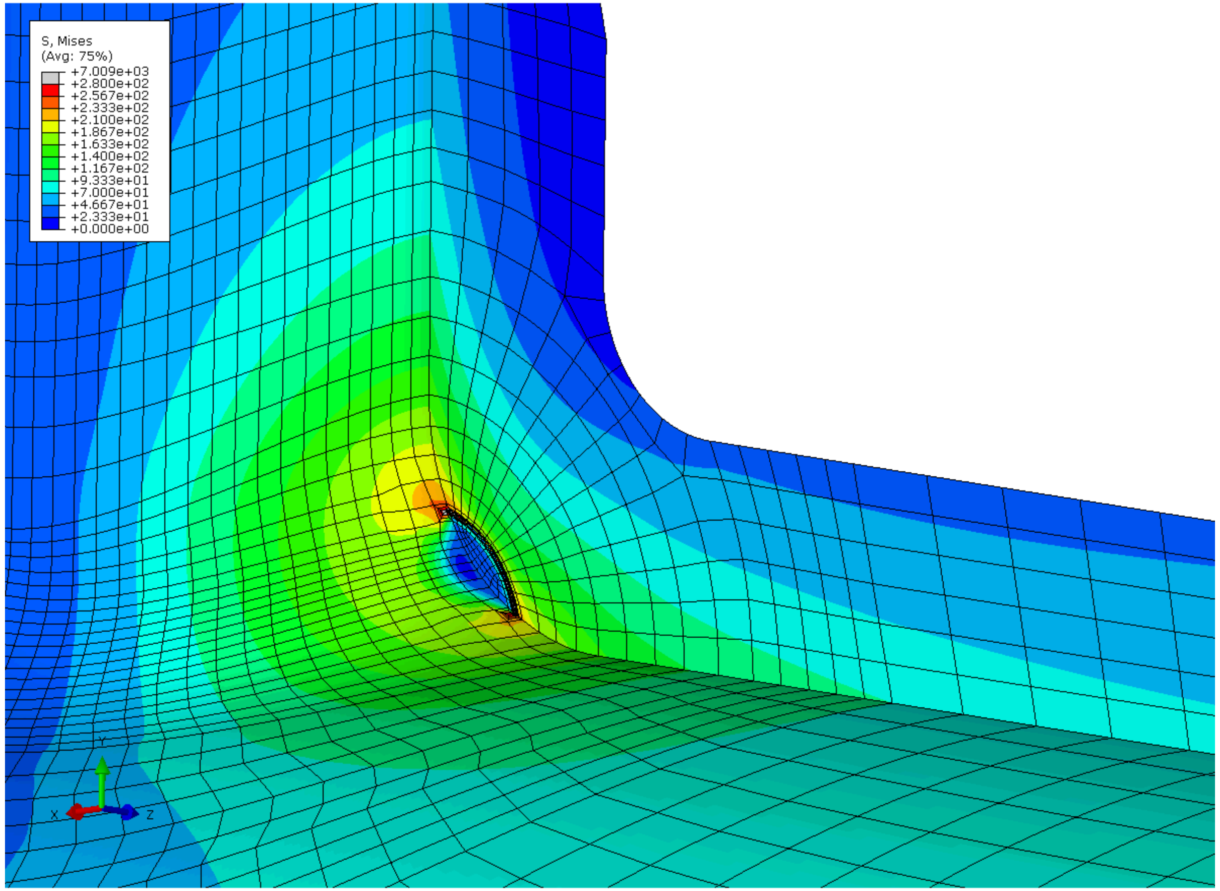
Example – header / nozzle intersection





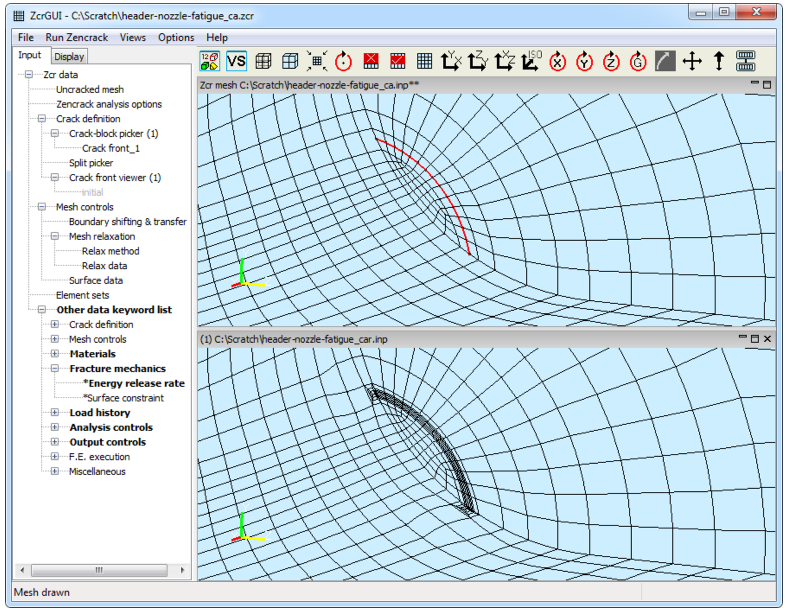
Crack growth due to cyclic internal pressure load of 1.6MPa to 16MPa

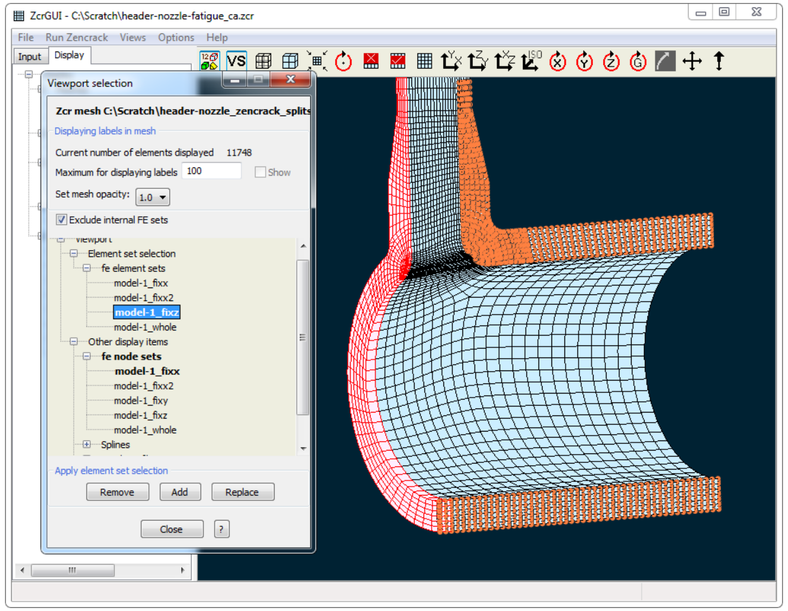


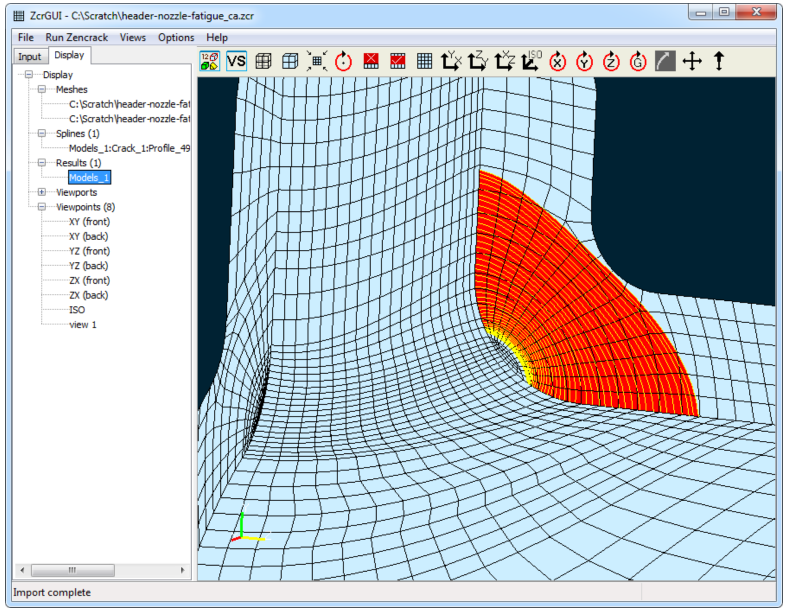


Zencrack GUI

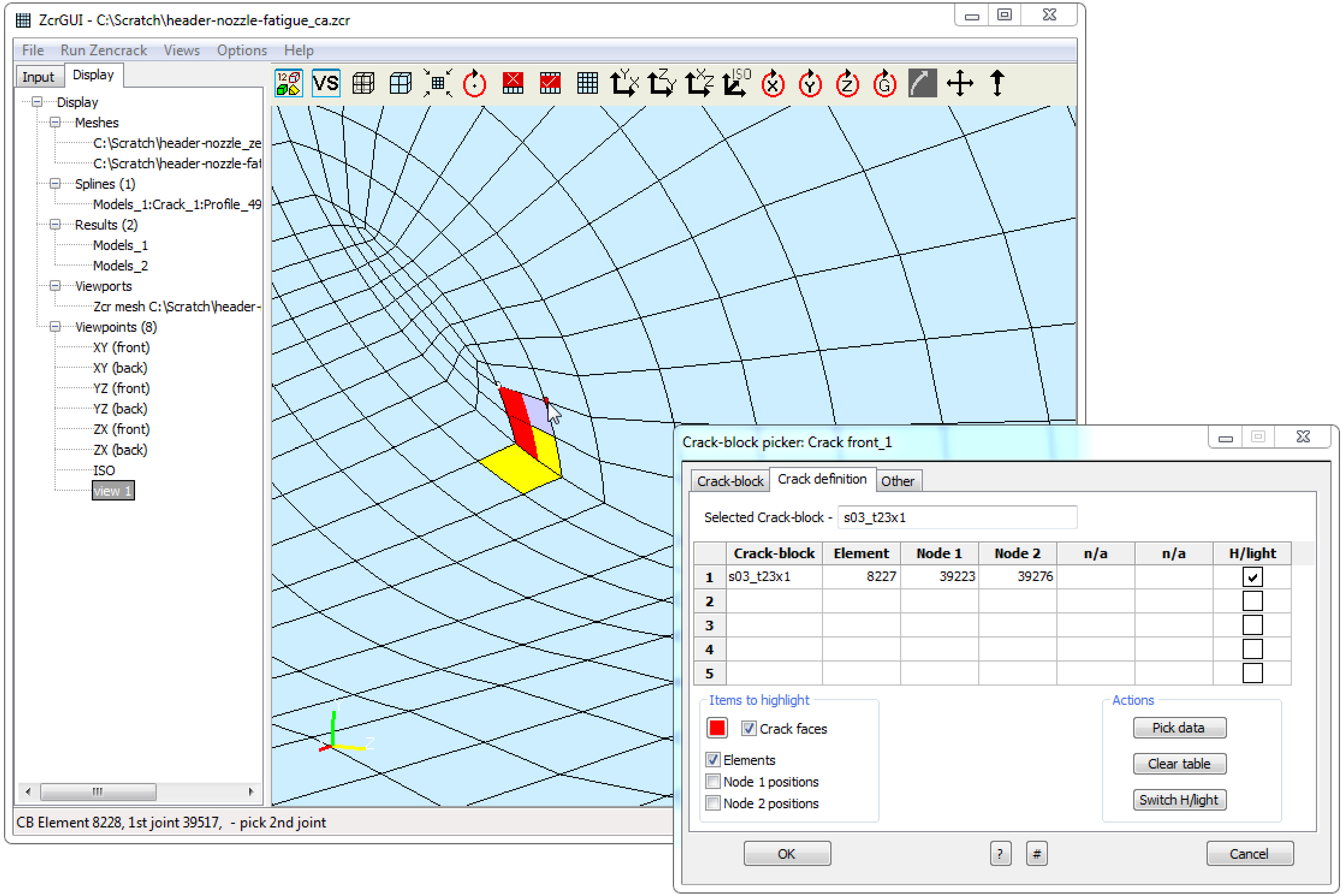
- The Zencrack GUI allows:
  - creation, review and modification of input files
  - submission of Zencrack jobs
  - visualisation of crack growth profiles and creation of XY plots
  - import of one or more uncracked or cracked meshes
- Replaces the earlier plug-ins for Abaqus/CAE



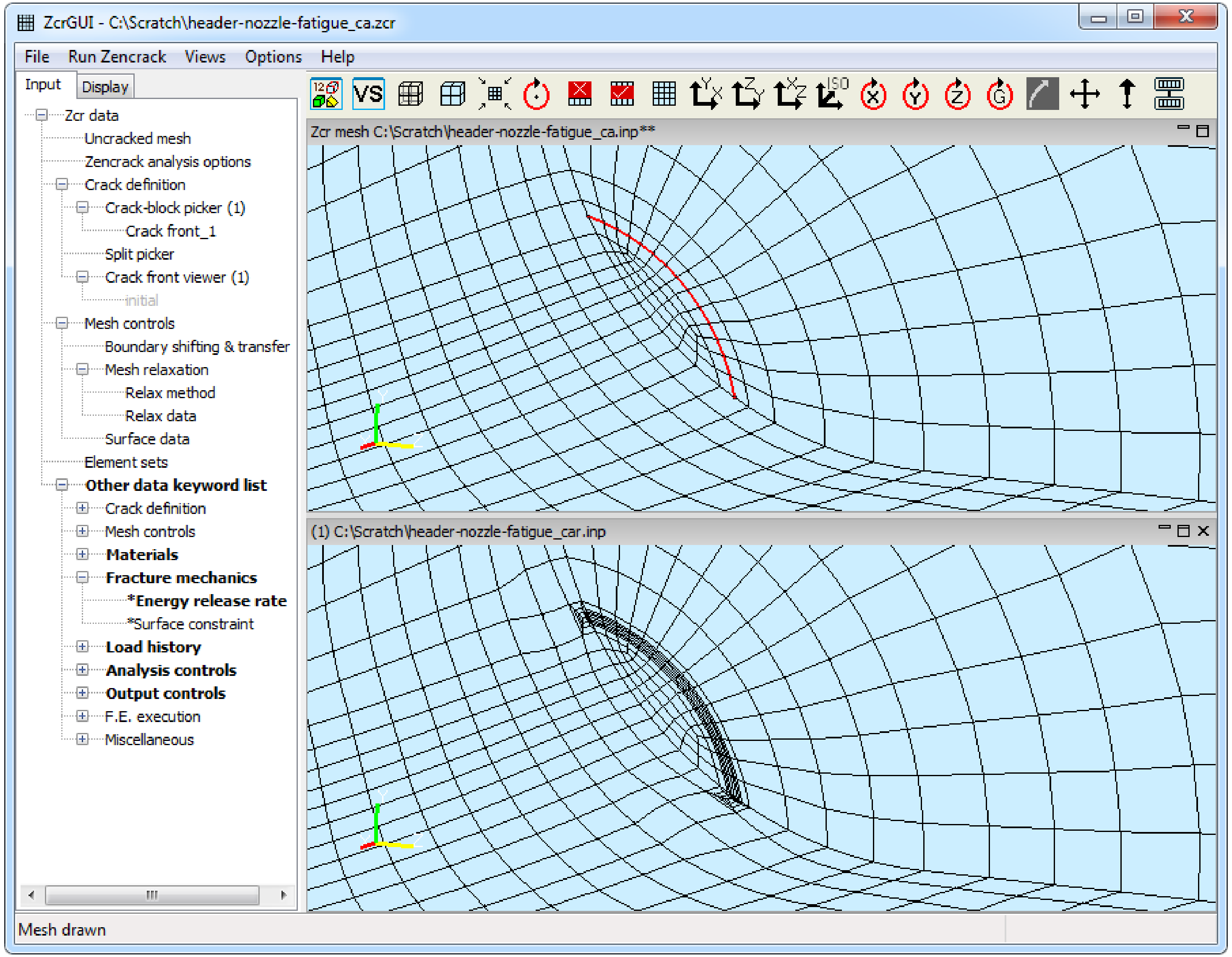




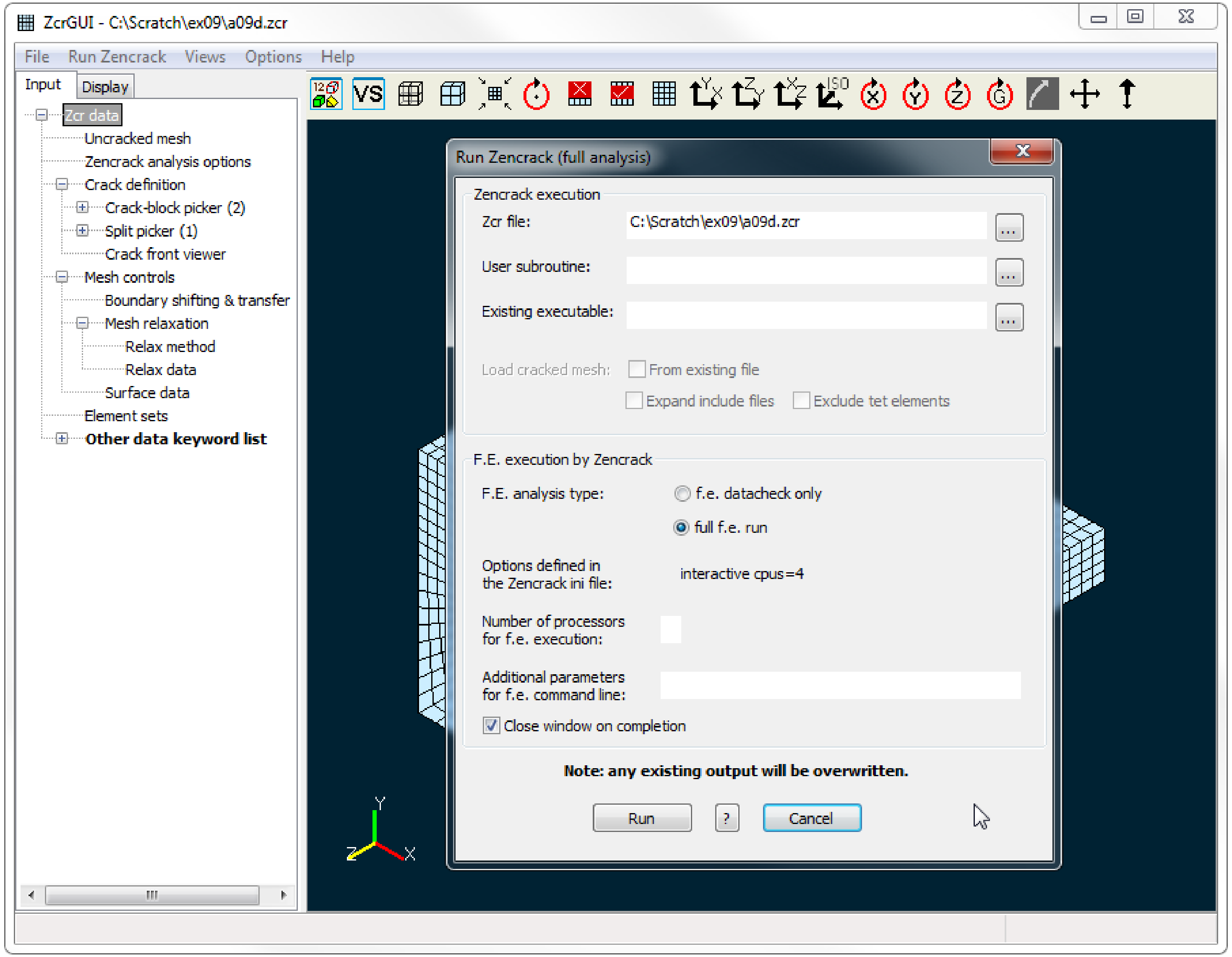
Define a crack front



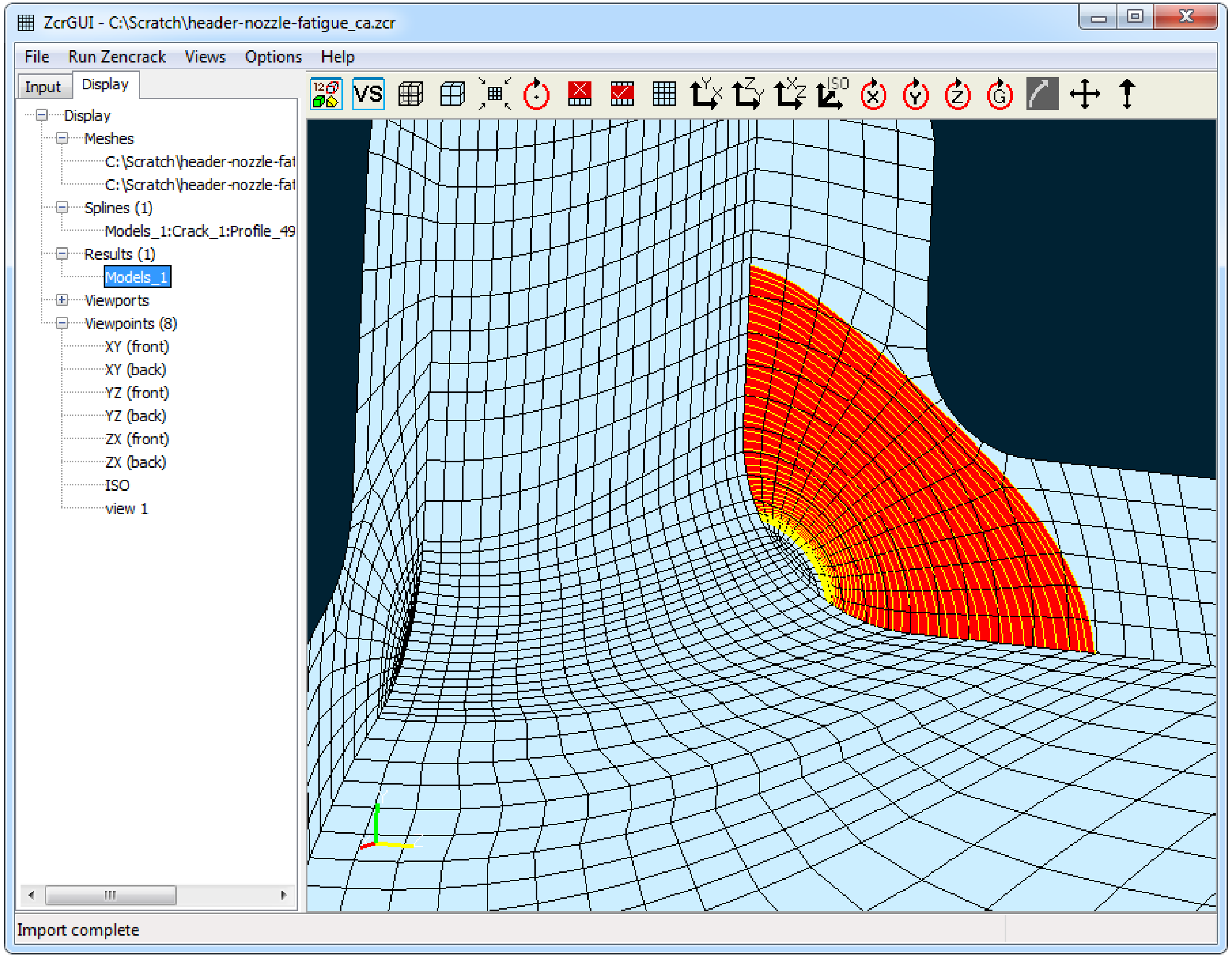
Preview the cracked mesh



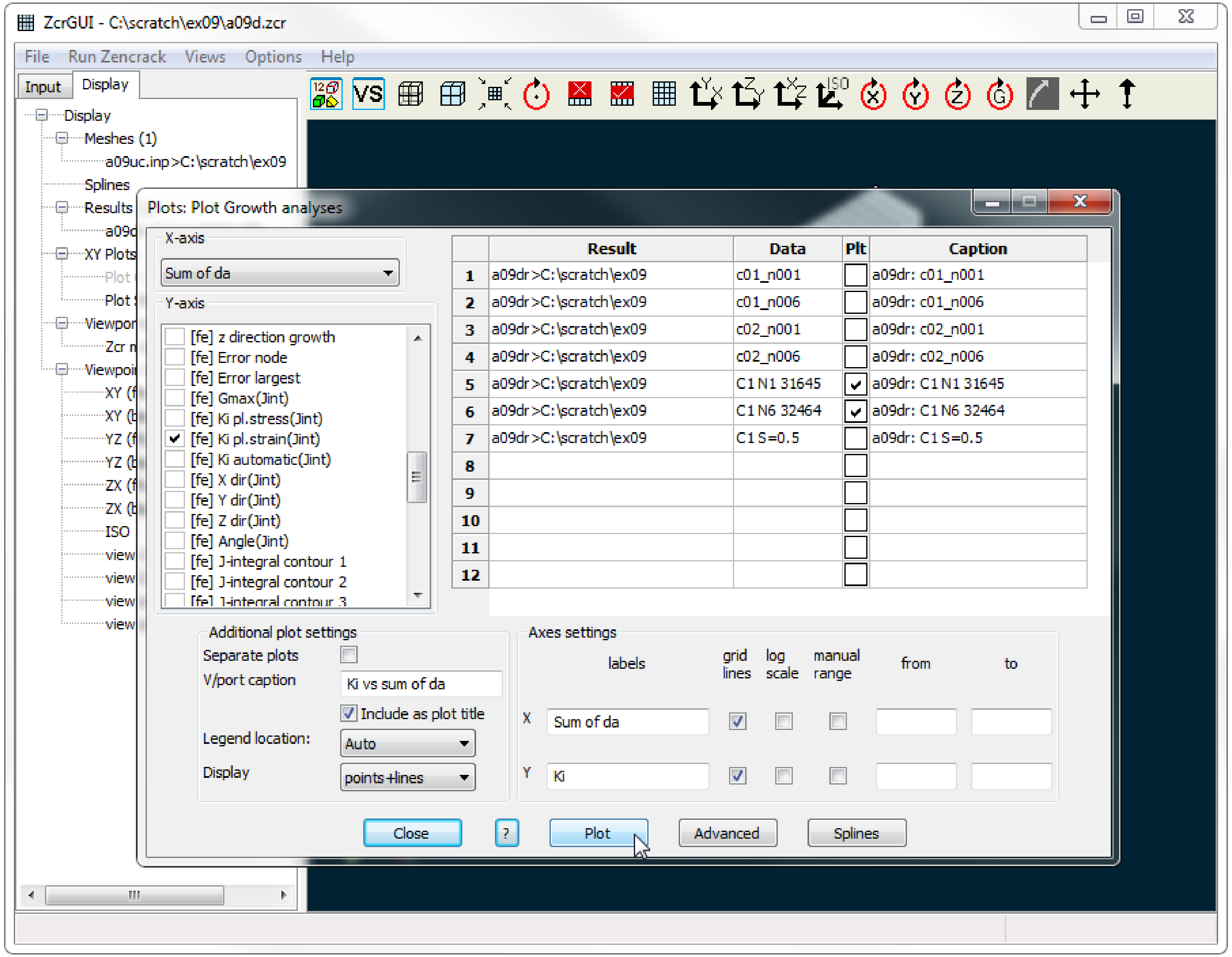
Submit analysis



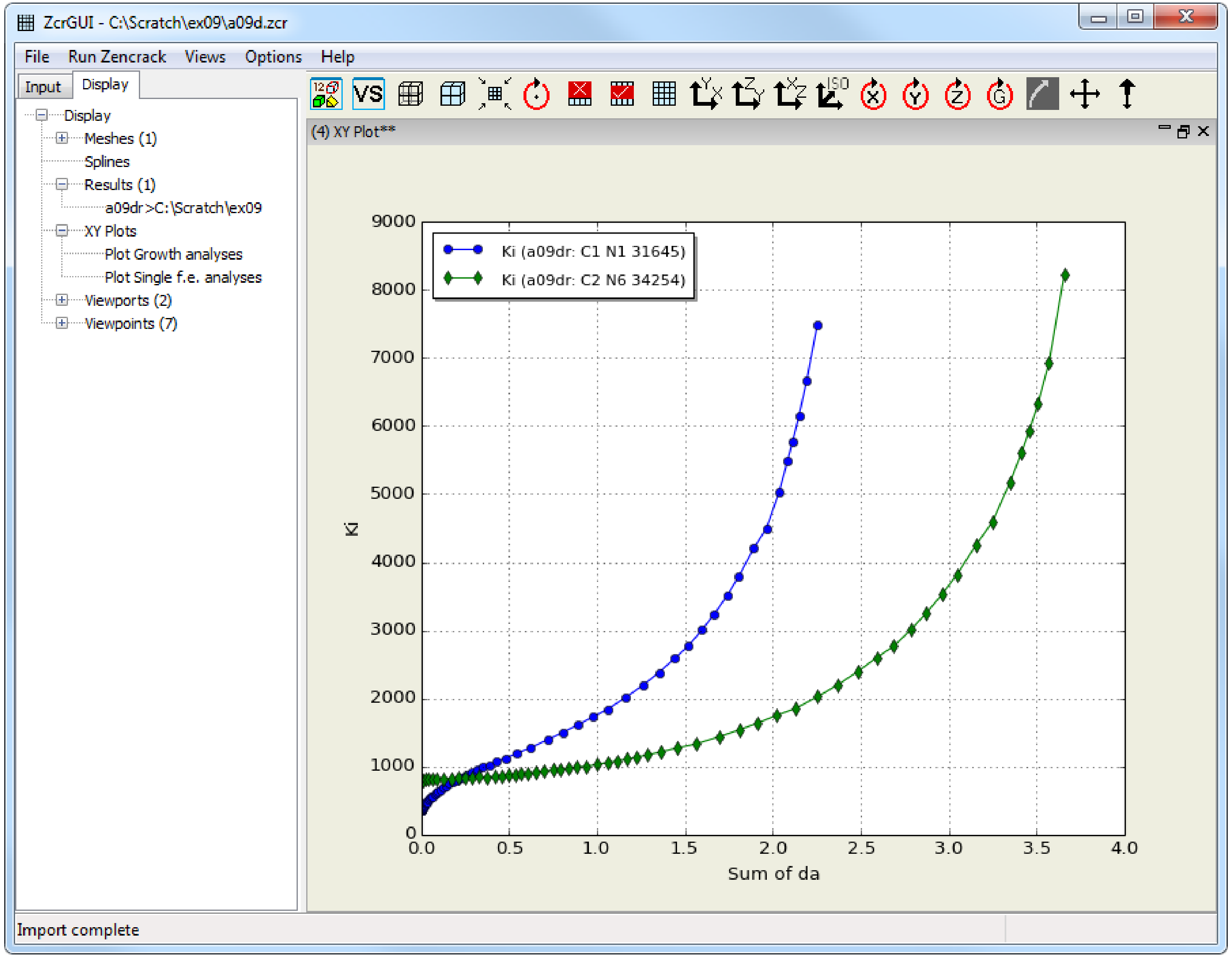
Visualise crack profiles and surface



Create XY plot

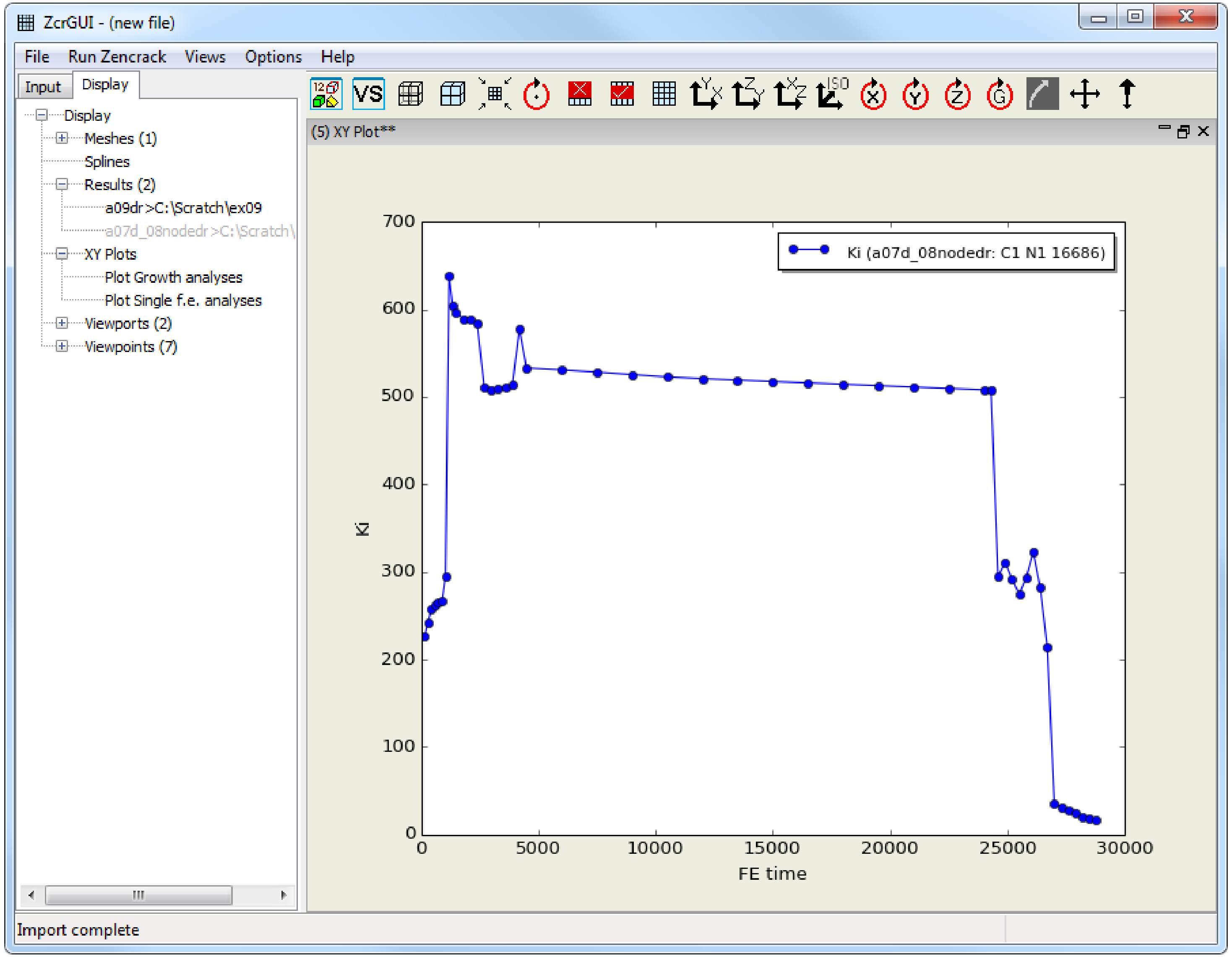


“Growth” XY plot



Example of results plotted through an entire crack growth analysis

“FE increment” XY plot



Example of results plotted through multiple increments of a single f.e. analysis

ZENTECH

Thank you.

Any questions?