Finite element modelling replication information for *"On the interaction of crack-like defects in ductile fracture"*

Author: Harry Coules

Contact: harry.coules@bristol.ac.uk

Document last updated: 31/01/2018

Preamble

This document describes finite element analyses performed as part of the study reported in: "On the interaction of crack-like defects in ductile fracture" by H. E. Coules. Used along with the main article, this document is intended to provide a further description of the modelling procedure used, to the extent that the models may be replicated by other researchers.

Description

The FE models examine the internal pressurisation of a pipe containing a twin pair of crack-like semi-elliptical internal flaws in the axial-radial plane. Three material models are used: an elastic → nonlinear-plastic material which approximates the behaviour of a typical pressure vessel steel (from which crack tip J-integral are calculated), an equivalent linear-elastic material (for calculating SIFs), and an elastic → perfectly-plastic material (used for calculating limit pressures). In each case, models of a pipe with a single defect are also performed in order to compare the result with that of the twin-cracked pipe and hence judge the effect of crack interaction. There are also models for examining the effect of inter-crack spacing and plastic hardening modulus on the J-integral.

Modelling information

Unit convention

- In all models, a consistent set of units is used. No unit conversions should be necessary. The system of units is as follows, with derived quantities in square brackets:
    - Length: mm
    - Force: N
    - [Pressure]: MPa
    - [Energy]: mJ
    - [SIF units]: MPa √mm
    - [Crack driving force units]: N mm^{-1}
    - Time: s
    - Mass: ton

Modelling software and hardware

- Dassault Systemes Abaqus/CAE v6.12 was used for model pre- and post-processing.
- The Dassault Systemes Abaqus/Standard v6.12 finite element solver was used for model solution.
- Models were solved using a machine with 12 Intel Xeon 5670 CPUs and 50 GB of RAM running under CentOS Linux 6.8.

Material model

J-integral models

- The material being modelled is ASTM A533B Class 1 ferritic steel.
- It is modelled using incremental theory of plasticity.

- The pipe is assumed to be initially free from residual stresses.
- The pipe is assumed to be isothermal throughout the modelled time interval.
- A von Mises yield locus is assumed.
- The material is assumed to exhibit non-linear isotropic hardening (i.e. proportional expansion of the yield locus during strain-hardening).
- The material is assumed to be elastically isotropic.
- The material's Young's modulus was measured to be 210 GPa. The Poisson's ratio is assumed to be 0.3.
- The yield stress (0.2% offset) is assumed to be 450 MPa.
- The yield offset parameter is assumed to be $\alpha = 0.9333$.
- Post-yield, the uniaxial true stress-strain curve is assumed to follow a Ramberg-Osgood relationship with a hardening modulus of n=7.6. This is shown in Figure 1.
- Note that the Ramberg-Osgood model used is a fit to uniaxial tensile test data for this material, as given previously in:

R. Hurlston, J. K. Sharples, and A. H. Sherry, "Understanding and accounting for the effects of residual stresses on cleavage fracture toughness measurements in the transition temperature regime," *International Journal of Pressure Vessels and Piping*, 2015.
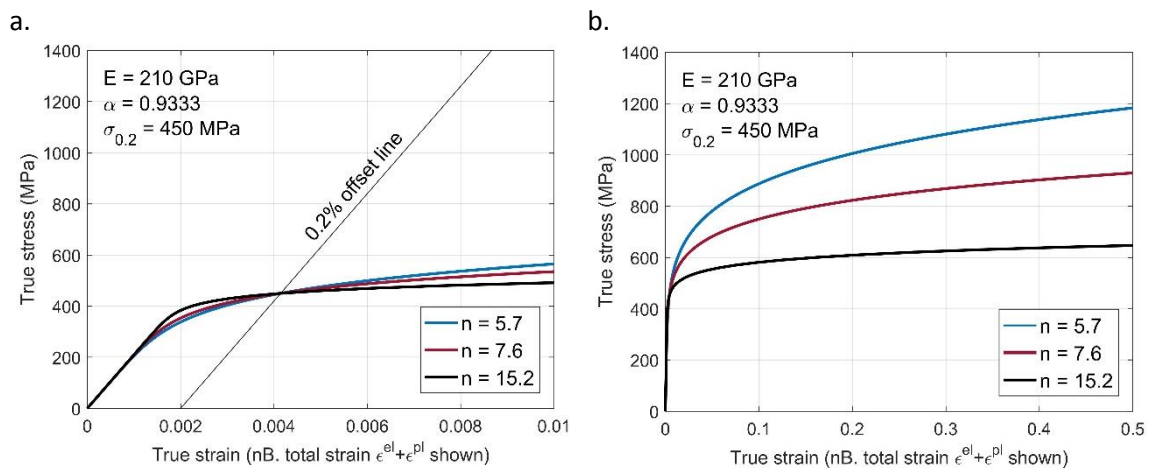
a.   b.

Figure 1: Uniaxial stress-strain curves for Ramberg-Osgood materials used in the FE models. a). For the range 0≤ε≤0.01, b.) for the range 0≤ε≤0.5.

J-integral models (with different hardening moduli)

As above, but with hardening moduli of n=5.7 and n=15.2 (see Figure 1).

Elastic SIF models

As J-integral models, but with elastic properties only.

Limit-load models

As J-integral models, but with:

- Elastic → perfect-plastic material, instead of a Ramberg-Osgood relationship for plastic strain.
- A yield stress of 360 MPa. *Note*: this requires that all limit load results from these models to be scaled by 450/360 = 1.25 if they are to be applicable to A533B, which has a yield stress of 450 MPa.
- Note that, as in the J-integral models, a von Mises yield locus is assumed in the limit-load calculations.

Loading and boundary conditions
- A half-model is used, with symmetry plane bisecting the pipe at the crack plane.
- Pressurisation is simulated by applying a linearly-increasing pressure to the internal surface of the pipe and to the crack faces. A tensile surface traction is also applied to one end of the pipe cross-section, which the simulates the axial stress applied to a long thin-walled pressure vessel by pressure acting on its closed ends (since the closed ends are not explicitly modelled here). The ratio of the internal pressure to the applied cross-section stress is 1:2.25, which is what would be applied for this ID/OD ratio according to the usual thin-walled pressure vessel equations. The other end of the pipe is held fixed using a symmetric boundary condition.
- No inertial forces or other body forces are modelled. The loading of is assumed to be quasi-static.

Modelling sequence and procedures
- All models comprise a single analysis step, during which linearly-increasing pressurisation occurs.
- For J-integral and elastic SIF models, contour integral results are calculated from the deformation field by Abaqus. The contour integral results show good path-independence.
- It is important to note that although J-integral results are provided in the .dat files of limit-load models, they are not meaningful for this material model.
- Local limit loads were extracted manually from the limit load model .odb files. This was done by viewing the von Mises equivalent plastic strain and observing when the plastic strain reached the outer wall of the pipe.
- The global limit load was judged to have been reached when the model could not continue due to plastic instability.

Spatial modelling
- The pipe ID is 400 mm and its OD is 500 mm. The length of pipe modelled is 4000 mm. The boundary conditions imply a pressurised vessel that is closed at the ends but arbitrarily long.
- The FE mesh is composed of three main regions. There is a crack tip region containing reduced integration linear brick elements (Abaqus type C3D8R), an unevenly-shaped region of quadratic tetrahedral elements (Abaqus type C3D10) surrounding it, and the outermost region which comprises most of the pipe is composed of reduced integration linear brick elements (Abaqus type C3D8R). a single layer of linear wedge elements (Abaqus type C3D6) are used at the crack tips themselves.
- When incompatible element types occurred next to each other (specifically, adjacent quadratic tetrahedral and linear brick elements) tie constraints were used to join them.
- Each set of J-integral, elastic SIF and limit load models with the same flaw geometry used the same mesh.
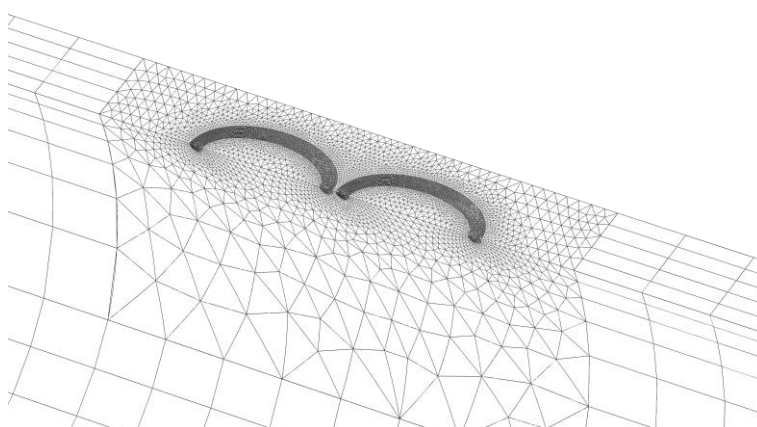
Images showing the FE mesh are given below:
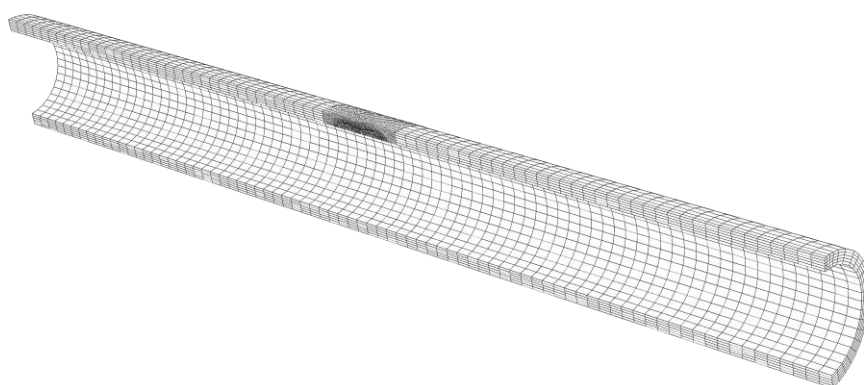
Figure 2: Overview of the complete mesh.
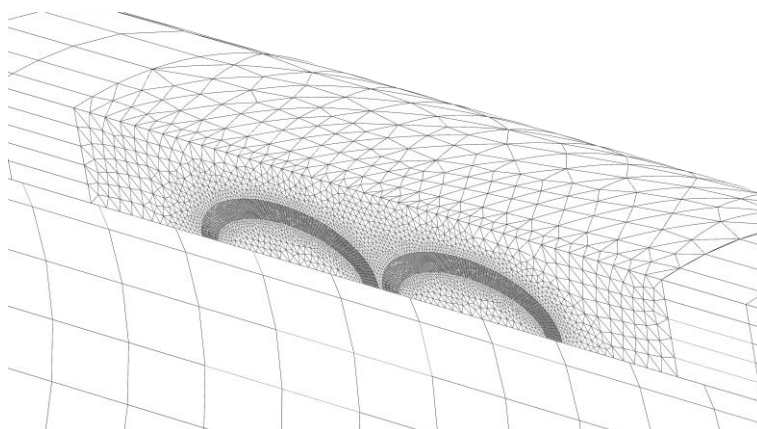
Figure 3: Close-up view of the twin crack mesh.

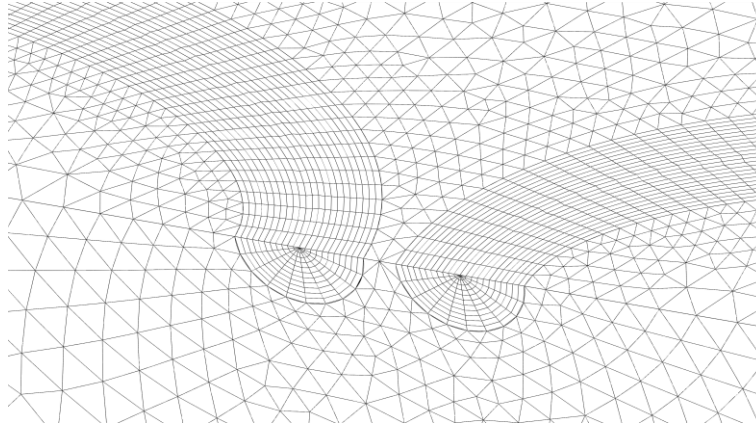*Figure 4: Different close-up view of the twin crack mesh.*

Figure 5: Very close view of the mesh at the cracks' closest approach.

Preliminary studies

Mesh refinement

Preliminary models were performed to investigate the required level of refinement in the finite element mesh. Refinement of the mesh beyond the level shown in Figure 2 - Figure 5 was found to give no visible change in the stress and strain fields, and no significant difference in contour integral results. It was found that the J-integral results exhibited good contour-independence throughout the pressurisation range used.

Geometric nonlinearity and bulging

To reduce computation time and ensure that regular model increment durations were easily achieved, the J-integral models were performed as "small-displacement" or geometrically-linear analyses (Abaqus option *STEP, NLGEOM=NO). However, the effect of geometric nonlinearity (and hence bulging at the flawed region) on the result was considered by running preliminary analyses with NLGEOM=YES and found to be small. This is shown in Figure 6. Subsequently, it was assumed that differences in bulging would have no significant effect on the interaction for this case.
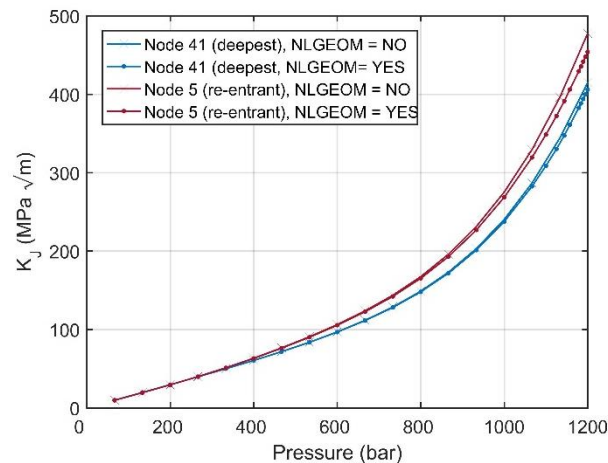

Figure 6: The effect of considering geometric nonlinearity on the calculated elastic-plastic strain energy release rate for a pipe containing twin cracks with d = 0.25c = 12.5 mm.

Data provided in this package

In each directory, input files (.inp) used for model definition are provided. Each .inp file is sufficient to completely define the model input for Abaqus/Standard v6.12. In addition, all files resulting from the Abaqus analysis are provided.