175 Finite Element Study of Effect of Constraint on Fracture in a Ductile Material

S.M. Tonge[1a], C. Simpson[1], T.J. Marrow[2], M. Mostafavi[1]

Queen's Building, University Walk, Clifton, Bristol,BS8 1TR, University of Oxford, Department of Materials, Parks Road, Oxford OX1 3PH

[1] st16173@bristol.ac.uk

Abstract To verify an experiment carried out on four ductile samples Finite Element Analysis was performed. A tensile test on four Aluminium Titanium (AlTi) metal matrix composite samples was simulated. Four different samples with varying levels of in and out of plane constraint were investigated. The Stress Intensity Factor (K) was determined for both experimental and Finite Element Data. K values from the experiment found to range between 38 and 52 MPa√m for the simulation between 40 and 62 MPa It will be possible using the full-field measurements of the stress field to determine the region of plasticity around the crack tip. A method of quantifying constraint such as the Anderson-Dodds method could then be applied.

Introduction

In the nuclear industry the fracture of components is an important consideration due to the risk of releasing radioactive materials, this is known as the fundamental nuclear safety hazard. Many components in the nuclear industry from boiler tubes to waste containers have thin section steels in their structures. The failure of thin walled structures has normally been assessed by considering plastic collapse rather than fracture. However, embrittlement of these structures occurs over their lifetime. For this reason, using fracture mechanics in failure assessments is relevant, particularly when assessing the integrity of structures that have been considerably aged.

Constraint of the crack tip is an important consideration in fracture mechanics assessments. Parameters to quantify constraint including Q, T and T_z, which are specific to either in or out-of-plane constraint. In-plane constraint is defined by the distance in front of a crack tip that is yet to fracture and out of plane constraint is defined by the thickness of the sample. A method of accounting for out-of-plane constraint is necessary for assessing thin section materials. It is not currently considered in R6, the UK nuclear industry standard fracture assessment code. Currently, no parameter exists that can quantify both in and out of plane constraint.

Method

A tensile test performed for four samples with dimensions chosen to give four different levels of constraint as shown in table 1. The material used was an Aluminium Titanium (AlTi) metal matrix composite In-situ X-ray tomography and diffraction was performed, and a series of bespoke codes used to extract stress intensity factor (K) and Crack Opening Displacement (COD)[1]. A Finite Element Simulation was used to validate the analysis of the test. The test was modelled in Abaqus 6.14, using a quarter of the model by symmetry as shown in Fig 2. These parameters were obtained from an experimental tensile test. The simulation used a rigid body to represent the arm of the testing rig, an equal load to the experiment was applied in displacement control, by displacing the reference point of the rigid body by a fixed distance. A Ramberg-Osgood model for the material was used with the following parameters: Young's modulus (E) = 100.2 GPa, Poisson's ratio = 0.3, Yield Stress = 103 MPa and Yield Offset = 2.75. This model uses the additive relation of the elastic and plastic components of strain Eq. º1.

$$\varepsilon = \varepsilon_e + \varepsilon_p \ (1)$$

Ramberg and Osgood combined the equation relating linear elastic stress and strain and an expression relating plastic stress and strain, Eq. º2.

$$\sigma_p = H\varepsilon_p^n \ (2)$$

This gives Eq º3, where σ_0 is yield stress and α is a dimensionless constant, α is the co-efficient of strength and n is the strain hardening exponent.

$$\frac{\varepsilon}{\varepsilon_0} = \frac{\sigma}{\sigma_0} + \alpha\left(\frac{\sigma}{\sigma_0}\right)^{-n} (3)$$

	Thick sample	Thin sample
Long notch	a/W=0.5, b=20mm,14.9kN	a/W=0.5, b=5mm, 4kN
Short notch	a/W=0.1, b=20mm,15kN	a/W=0.1, b=5mm, 4.5kN

Table 1: Dimensions of samples and loads used at failure, 2W=8mm for all samples.

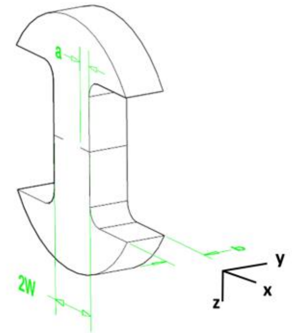

Fig. º1 Diagram of sample dimensions in Table 1

The sample size was reduced by using a quarter model, symmetry boundary conditions were enforced on the x and y axes of symmetry. The mesh, shown in Fig. º2 was optimised around the crack by the use of partitions. Linear 3D stress elements were used from the Abaqus 6.14 standard library. A history output request was used to determine the stress intensity factor K, a measure of the crack driving force. The same number of contours were used in evaluating K from FE data as experimental data to ensure consistency.

Results and Discussion

The K values were plotted through the thickness of the sample for each sample. The values found from FE simulations are shown in Fig. º3. Higher K values were found in the thick sample and the long-notched samples than thin and short notched samples. The same trend was seen in the results derived from the tomograms taken during the experiment, for which the K values ranged between 38 MPa√m for the thin sample with long notch and 52 MPa√m for the thick sample with short notch. Therefore, a relatively good agreement has been found between the experimental and FE values derived at the middle of the sample thickness. However, the FE results showed the values in the centre of the sample at a maximum, whereas experimentally derived samples where at a minimum in the centre. A possible reason for this is that the FE analysis was conducted using the entire thickness of the sample whereas the values derived from tomography considered each tomogram slice individually.

This also shows that the effect of geometry in providing different levels of in and out of plane constraint, has a strong on the values of K. It was also found that the values of ε_{22} for the thin sample with short notch shown in Fig. º4, which were derived from FE, corresponded well with the values derived by analysis of strain data from X-ray diffraction [4].

Future Work

It is intended to use the volume of the plastic region as a means of quantifying the level of constraint. Two methods will be tested using the Finite Element and experimental data. The Anderson-Dodds method involves finding the area inside a contour of plasticity is used to define the plastic region [5]. The size of the area bounded by the contour is used to determine high or low constraint. Recently, Seal and Sherry proposed a method of quantifying in and out of plane constraint with a single parameter, this method will be tested by evaluating the parameter for the data in this experiment [6].

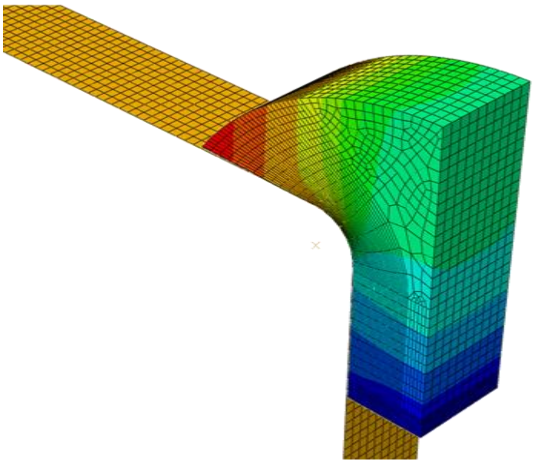

Fig. º2 Image of geometry of simulation .showing FE mesh

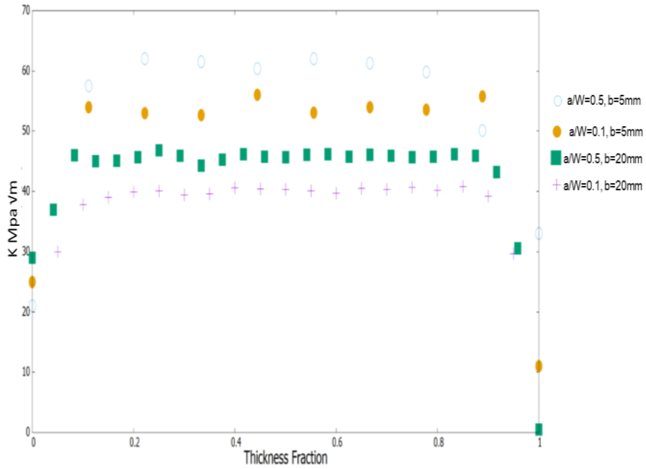

Fig. º3 Graph of K values through thickness from FE

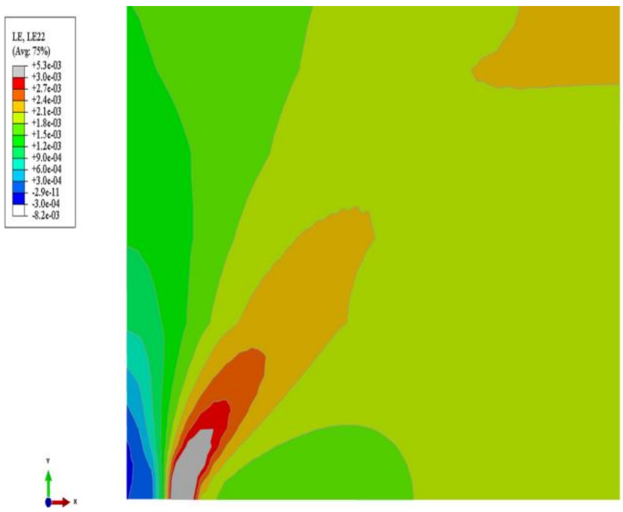

Fig. º4 Linear Elastic Strain ε_{22} shown for a/W=0.1,b=5mm

References

[1] A. F. Cinar *et al.*, "An autonomous surface discontinuity detection and quantification method by digital image correlation and phase congruency," *Opt. Lasers Eng.*, vol. 96, pp. 94–106, 2017.

[2] T. L. Anderson, *Fracture Mechanics: Fundamentals and Applications*, vol. 58, no. 1. 2012.

[3] S. M. Barhli *et al.*, "Obtaining the J-integral by diffraction-based crack-field strain mapping," *Procedia Struct. Integr.*, vol. 2, pp. 2519–2526, 2016..

[4] C. Simpson et al, Validating 3D two-parameter fracture mechanics for structural integrity assessments, ECF22, 2018

[5] Wolfenden, A., Anderson, T. L. and Dodds, R. H. (1991) Specimen size requirements for fracture toughness testing in the transition region. J. Test. Eval., 19, 123–134..

[6] C. K. Seal and A. H. Sherry, "Predicting the effect of constraint on cleavage and ductile fracture toughness using area contour toughness scaling," Eng. Fract. Mech., 2017.