Damage model calibration and application for S355 steel

João Ribeiro*, Aldina Santiago, Constança Rigueiro

*ISISE - Department of Civil Eng., Faculty of Science and Technology, University of Coimbra, Portugal
bISISE - School of Engineering, Polytechnic Institute of Castelo Branco, Portugal

Abstract

Current demands for improved structural integrity and risk assessment involves the evaluation of alternative unloading paths, ensuring that a whole building (or a significant part of it) remains stable when subject to an unforeseen event: natural hazards (earthquakes, foundation failure, fire,…) or even terrorist attacks. The design under these conditions requires that the structural elements and, particularly in steel framed structures, the joints connecting elements are able to undergo elevated deformations without fracturing, thus providing means of energy dissipation.

The finite element method (FEM) is nowadays a widespread practice assisting in the simulation of many physics phenomenon. Looking forward to an accurate finite element simulation of steel connections up to its fracture, the implementation of a failure criterion based on continuum damage mechanics is explored in this paper. It is done by comparing the results reached from an undamaged analysis and the ones obtained from a damaged analysis using a ductile failure with “element deletion” technique to simulate the fracture. The analyses are carried out using the finite element software ABAQUS. The establishment of the fracture strain dependency to the triaxial stress state is based on the experimental evidences reached from 12 tensile coupon (S355) tests, including both notched and unnotched flat dog-bone test specimens.

© 2016 The Authors. Published by Elsevier B.V.

Peer-review under responsibility of the Scientific Committee of ECF21.

Keywords: Structural steel; Material behaviour; Damage, FE methods; Experimental tests.

1. Introduction

Structural design solutions that can withstand the localised damaged without widespread collapse have become an important and urgent subject to study. Such design requires that the structural elements and, particularly in steel framed structures, that joints connecting elements undergo under elevated deformations without fracturing, thus providing
means of energy dissipation. Today, the Eurocodes already provide recommendations towards a ductile design; yet, the establishment of the joints limiting rotation capacity requires costly laboratory tests. Past studies have showed that finite element modelling can be used to predict the nonlinear behaviour of joints. These numerical results were mostly used to supplement test data and give a more accurate description of the loading paths in the connecting components of the joints, Swanson et al. (2002), Girão Coelho et al., 2006 and Ribeiro et al., 2016. In order to model more realistic behaviour of materials, its complete description (including failure criteria) is required, either based on experimental procedures or numerical methods. In this respect, virtual tests carried out by means of numerical modelling are increasingly replacing some mechanical and physical tests to predict and validate their structural performance and integrity due to recent developments in software-based nonlinear finite element analysis methods, as example: Kang et al. (2015) and Liao et al. (2015). This includes the computational advances in fracture modelling. This approach represents a cost effective way of exploring new, adequate and cheaper solutions (Fig. 1).

**Nomenclature**

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CTOD</td>
<td>Crack tip opening displacement</td>
</tr>
<tr>
<td>CDM</td>
<td>Continuum damage mechanics</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite element method</td>
</tr>
<tr>
<td>PEEQ</td>
<td>Equivalent plastic strain</td>
</tr>
<tr>
<td>TRIAX</td>
<td>Triaxial stress state</td>
</tr>
</tbody>
</table>

Regarding the failure criteria, the use of micromechanical models based on void growth and coalescence seems like an attractive tool to evaluate conditions for ductile fracture initiation, Lemaitre (1992). These models are able to predict ductile fracture from fundamental mechanical principles, as a function of multiaxial stresses and plastic strains.
The current study focuses on the implementation of a failure criterion on a finite element model in order to predict ductile fracture behaviour with good accuracy across the specimen geometry and material types, in terms of load-displacement curve, ultimate load and fracture initiation. To achieve this, the present paper has been broken down into the following topics:

i) Review of the micromechanical models applied to ductile fracture initiation in metal plastic deformation;

ii) Presentation of a testing campaign on unnotched and notched specimens under monotonic tensile loading, aiming to establish damage onset as a function of multiaxial stresses and plastic strains for a structural steel.

iii) Development of finite element models to simulate the experiments up to its fracture, using the commercial finite element software Abaqus. Undamaged analysis and damaged analysis using a ductile failure with “element deletion” technique are considered.

2. Material behaviour

2.1. Material modelling

Mild steel is a ductile material; its typical constitutive behaviour is characterized by an initial linear response until the yield strength followed by a second nonlinear phase of reduced stiffness. The strain energy accumulated in the material is released beyond the instability point where the ultimate strength is attained; from this point on the material progressively loses its strength and stiffness until its rupture. Fig. 2 presents the characteristic stress-strain behaviour of a ductile material with damage degradation; the dashed curve represents a generic material response without damage definition, while the solid line corresponds to the damaged stress-strain relationship. In this figure, $\sigma_u$ and $\bar{\varepsilon}^{pl}$ are the ultimate strength and equivalent plastic strain at the onset of damage, while $\bar{\varepsilon}^{pl}$ is the equivalent plastic strain at failure, ABAQUS (2011).

Using the conventional fracture mechanics, fracture modelling requires three steps, Lemaitre (1992):

1. Stress analysis: the geometry of the structure being known, together with the history of loading and initial conditions, the stress and strain fields are firstly calculated by means of strain constitutive equations and a numerical procedures (FEM, for example).

2. Damage criterion: the most critical locations regarding fracture are determined and the load corresponding to the crack initiation at that point is calculated by integration of damage constitutive equations taking into account the local stress or strain history.

3. Fracture mechanics concepts are applied in order to calculate the evolution of the crack up to the final rupture of the whole structure. Methods as the stress intensity factor (based on linear elastic fracture mechanics), the crack tip opening displacement (CTOD) or the J-integral based on elastic–plastic fracture mechanics are considered.

However fracture mechanic methods require that a crack already exists, and focuses on establishing the local behaviour of how this crack propagates, which is not appropriate in studying failure on a structural scale, Lemaitre (1992). In contrast, when coupled with finite element deletion, a continuum damage mechanics (CDM) approach is able to deliver an approximation of the fracture pattern, suitable for a structural scale analysis, while avoiding the 3rd step described earlier.

![Stress-strain curve with progressive damage degradation](https://example.com/stress-strain.png)

Fig. 2. Stress-strain curve with progressive damage degradation, adapted from (ABAQUS, 2011).
2.2. Continuum damage mechanics

In ductile materials, as mild steel specimens without macroscale flaws or cracks, the void nucleation occurs with little difficulty, therefore the fracture properties are controlled by the growth and coalescence of those voids, resulting in failure, Anderson (1995). Damage results in softening of the material and the fracture is a consequence from the competition between hardening and damage. When damage succeeds, local strains that result in a crack are observed. The stress triaxiality and the plastic strain play an important role in this process. This type of mechanism is best described by micromechanical models. In the past, numerous criteria have been developed for ductile fracture initiation in metal plastic deformation. Kachanov (1958) was the first to introduce a continuous variable related to changes in the mechanical properties of a material. Based on several experimental tests, Rice and Tracey (1969), McClintock (1968) and Hancock and Mackenzie (1976) showed that the stress triaxiality and plastic strain are important factors in crack initiation and propagation of damage. Since then, several authors have studied and proposed other criteria and models; as example, refer, Johnson and Cook (1985), Lemaitre (1992), Bao and Wierzbicki (2004), Hooputra et al. (2004). Wierzbicki et al. (2005) provide insight of the usage of current fracture models by exploring the calibration parameters of seven of them.

Both formulations proposed by Johnson and Cook (1985) and by Hooputra and co-authors (2004) are included in the software ABAQUS package and are used in this study. The dependency between the multiaxial stresses and plastic strains is written by Johnson and Cook’s equation (Eq. 1):

$$\varepsilon_f = [D_1 + D_2 \delta \sigma\gamma][1 + D_4 \ln(\dot{\varepsilon}_p)][1 + D_5 \sigma^*]$$

(1)

Where the material parameters are defined as: $D_1$, $D_2$ and $D_3$ establish the fracture strain dependency to the triaxial stress state, $D_4$ establishes the strain-rate dependency and $D_5$ accounts for temperature softening ($D_4$ $D_5$ are not considered in the current study).

The damage evolution law proposed by Hooputra and co-authors (2004) is used to compute the damage variable in the software. Damage evolution description based on linear displacement requires the definition of the effective plastic displacement $\bar{\varepsilon}^p = L \cdot \dot{\varepsilon}^p$, where $\dot{\varepsilon}^p$ is the equivalent plastic strain at failure and $L$ is the characteristic length of the finite element; due to strain localization in elements placed in the necking development zone, the progressive damage response is mesh dependent, ABAQUS (2011). As elements reach a user defined level of degradation (for instance, the maximum degradation of $D = 1$, Fig. 2) following $\sigma = (1 - D) \sigma$, elements may be either kept or removed from the mesh. Hooputra et al. (2004) advise that the procedure is suitable to predict crack initiation zones, but element removal should be regarded as preliminary assessment for crack propagation simulation.

3. Experimental tests

3.1. Experimental campaign

In this study, an experimental campaign on 12 tensile steel coupons (dog-bone tests of unnotched and notched specimens) is carried out to support the calibration of the numerical simulations. The tests are performed as close as possible to the recommendations of CEN EN10002-1 (2001). The material to build test specimens has been provided by Martifer Construction; the plate with approximate dimensions of 1000 x 300 x 19 mm$^3$ is S355 JR+N. The tests specimens are flat specimens with the dimensions presented in Fig. 3. The label follows the specimen geometry: FN-#- stands for “Flat + Notch + (--) notch value + (#-) specimen number. Different notches sizes are considered to obtain different triaxial stress states needed to establish its relationship to the fracture strain; 3 repetitions are considered for each specimen geometry.

The tests are carried out on a general purpose hydraulic tensile testing machine on a displacement based setup with a low velocity (0.001 mm/s). Additionally the data acquisition setup includes a Data Logger (TDS-530) to read the data obtained from the tensile testing machine and from the mechanical axial extensometer with an initial gauge length of $L_0 = 50$ mm placed at half length of the specimens.
3.2. Results

Fig. 4 presents the strain-stress relationship obtained from the flat specimens under tensile load. The unnotched dog-bone tests – FN00 (in blue) presents a common steel plateau in the plastic transition and elevated ductility flow with great amounts of deformation after the maximum strength is attained (as signalled by the markers in all tests). The results indicate that the material meets the Eurocode requirements, CEN, EN 1993-1-1 (2010) to be classified as an S355: i) \( f_y \geq 355 \text{ MPa} \); ii) The ratio \( f_u/f_y \) is over 1.10; iii) Elongation at failure is not less than 15%; iv) \( e_u \geq 15 \varepsilon_y \).

The results of the notched dog-bones tests (FN02, FN04 and FN06) shows that the yield plateau vanishes and ductility is diminished as the notch dimension is increased (i.e. smaller cross-section area). An apparent increase in the maximum strength is observed as a consequence of the rapidly changing area within the notched zone of the specimens and data measurement constraints:

- strains are gathered from measurements of the mechanical axial extensometer considering its initial gauge length of 50 mm for all specimens, despite straining becomes more and more localized as the notch size increases;
- stresses, rely on the force readings from the tensile machine and consider the initial area of the reduced cross-section of the specimens (as presented in Fig. 3).

However, these limitations do not affect the success of this work as the different geometries provide different triaxial states, allowing establishing the fracture locus to be introduced in the numerical model.

Table 1 presents the initial and final specimen dimensions for the axial length. It can be observed that with the notch size increase that the final length decreases thus yielding a lower overall strain; the unnotched specimens develop up 10% strain over the whole specimen while with a notch of 6 mm only around 1.8 % is observed.

![Fig. 4. Stress-strain relationship obtained from mechanical axial extensometer for flat specimens.](image)

<table>
<thead>
<tr>
<th>Thickness [mm]</th>
<th>Width [mm]</th>
<th>Li [mm]</th>
<th>Lf [mm]</th>
<th>Strain Global</th>
</tr>
</thead>
<tbody>
<tr>
<td>FN00</td>
<td>14.8</td>
<td>19.9</td>
<td>300.0</td>
<td>330.8</td>
</tr>
<tr>
<td>FN02</td>
<td>14.6</td>
<td>16.1</td>
<td>300.0</td>
<td>312.3</td>
</tr>
<tr>
<td>FN04</td>
<td>14.5</td>
<td>12.1</td>
<td>300.0</td>
<td>306.5</td>
</tr>
<tr>
<td>FN06</td>
<td>14.8</td>
<td>8.0</td>
<td>300.0</td>
<td>305.3</td>
</tr>
</tbody>
</table>
4. Numerical model

4.1. Introduction

The numerical model is built using the general purpose finite element software Abaqus. Due to the simplicity of the model and the rather unsymmetrical shapes of the fractured shapes, the entire geometry is modelled. The models are discretized with C3D8R finite elements. Generally, a finite element mesh size of 2 mm is used; however, it has been reduced to 1.25 mm within the notched zones – Fig. 5. Explicit/dynamic algorithm is used to solve the non-linear problem. The load is applied as a velocity ramping from 0 to 4000 mm/s within 0.01 seconds on the top of the specimen, while the bottom end is fixed.

Material nonlinearity is included by specifying a non-linear stress-strain relationship for material hardening in the form of true stress-true strain; von Mises criterion is considered to establish the yield surfaces with the associated plastic flow for isotropic materials. The non-linear true stress - true strain relationship is introduced taking into account the results from the unnotched test specimen (FN00) as presented in Fig. 6; the horizontal dashed plateau represents the software assumption when it runs out of data points to describe the material behaviour.

Two numerical analyses are carried out and compared: i) Analysis without considering damage behaviour to establish the triaxial stress state given by the experimental tests; ii) Based on the triaxial vs. PEEQ response obtained from the analysis without damage, Eq. 1 is used to establish the triaxial dependency, which will be introduced in the damaged numerical analyses.

4.2. Numerical results without damage

In order to establish the triaxial stress state for the different specimens, an analysis without considering damage behaviour is run; the strain stress results (Fig. 7) have been obtained in the same manner as the experimental tests: considering the initial gauge length of 50 mm and the initial cross-section area of the specimens. Again, an increase in the ultimate strength and reduction of the ductility is apparent with the increase in the notch size. The simulations are able to deliver a softening behaviour due to the plateau in the stress strain material input (Fig. 6) which inhibits the finite elements of carrying more strength, but will overestimate the fracture strain observed experimentally, as the simulation continues running without failure. The strength reduction is reflected in the deformed shape of specimen FN00 in Fig. 8 as a clear necking zone is developed.

In order to obtain the triaxial stress states needed to establish the damage behaviour of the specimens, the analyses are receded to the experimentally observed fracture strain (Fig. 8), and it’s triaxial vs. equivalent plastic strain (PEEQ) is plotted for a central finite element, allowing to define the dependency to be used in an analysis considering damage.
4.3. Numerical results considering damage

Based on the triaxial vs. PEEQ dependency plotted for the middle finite element (continuous curves in Fig. 9), D1, D2 and D3 of Eq. 1 are calibrated. Now, this equation is used to define the dependency to be introduced in the simulation with damage (dash-dot line in Fig. 9). Fig. 10 compares the strain-stress results obtained experimentally (solid line) with the numerical ones with damage, (dash-dot line), assuming an effective plastic displacement of $\bar{u}^P = 0.1$; it can be observed that the fracture strain prediction is improved and the full separation of the specimens is reached using the deletion technique (Fig. 11).
5. Conclusion

A tensile testing campaign comprising four differently notched specimens of a structural steel S355 JR+N allowed the calibration of a relationship between the fracture strain at the onset of damage and the triaxial stress state. The implementation of this relationship within the ductile damage material model available in Abaqus coupled with finite element deletion, resulted in a good approximation of the fracture of the specimen, with improvements over analyses not considering damage.

Despite allowing establishing a fracture pattern, the approach used is found to be somewhat indeterminate as it depends on the users input to define both:

i) the onset of damage, i.e. the fracture strain at which damage is initiated, and,

ii) the damage evolution, i.e. the how much an element will elongate before its removed from the mesh, which by itself is mesh dependent. Should the user use a lower TRIAX vs. PEEQ relationship and a higher $\alpha^{pl}$ to define the linear damage evolution law, one could obtain the same result, increasing the difficulty of establishing definitive values to be used in different cases with the same material.

Acknowledgments

This work is financed by FEDER funds through the Competitivity Factors Operational Programme - COMPETE and by national funds through FCT – Foundation for Science and Technology within the scope of the project POCI-01-0145-FEDER-007633.

References