Numerical simulation of damage evolution for ductile materials and mechanical properties study

A El Amri¹, I Hanafi², M E Y Haddou¹ and A Khamlichi³

¹Condensed Matter, Radiation and Numerical Simulation Laboratory, Faculty of Sciences, M’hannech II 93002, Tetouan, Morocco
²National School of Applied Sciences (ENSAH), BP 03 Ajdir, Al Hoceima, Morocco
³Systems of Communications and Detection Laboratory, Faculty of Sciences, M’hannech II 93002, Tetouan, Morocco

E-mail: abdelouahid26@gmail.com

Abstract. This paper presents results of a numerical modelling of ductile fracture and failure of elements made of 5182H111 aluminium alloys subjected to dynamic traction. The analysis was performed using Johnson-Cook model based on ABAQUS software. The modelling difficulty related to prediction of ductile fracture mainly arises because there is a tremendous span of length scales from the structural problem to the micro-mechanics problem governing the material separation process. This study has been used the experimental results to calibrate a simple crack propagation criteria for shell elements of which one has often been used in practical analyses. The performance of the proposed model is in general good and it is believed that the presented results and experimental-numerical calibration procedure can be of use in practical finite-element simulations.

1. Introduction

There is a strong interest in the use of aluminium alloy sheet for vehicle applications, particularly the body, where resistance spot welding is the principal joining method. It is important that the particular discontinuities that are often found in aluminium alloy spot welds do not adversely affect the weld properties. The 5182 Aluminium alloy have a wide range of applications in the aerospace and automotive industries. This is mainly due to its advantageous characteristics that include high electrical and thermal conductivities, high ductility, high specific strength, ease of casting, low density, high strength to weight ratio and reasonable corrosion resistance [1]. The effect of high temperature on mechanical properties such as the tensile strength, the yield stress and young’s modulus depend on temperature is linked to transformations of the material structure due to various processes imply that inelastic deformation can occur more easily at elevated temperatures, so more plastic deformation and creep occur in the plastic zone of a fatigue crack. In this work, the 5182 H111 Aluminium alloy was studied in order to find out the effect of tensile strength. Both the malleability and the ductility properties relate to the plasticity of the material. Malleability refers to the ability of plastic deformation under compressive loads, while ductility refers to plastic deformation under tensile loads.

Percentage elongation before the tensile test begins two punch marks are made on the stem of the tensile test piece, distance between these marks is noted and is known as gauge length 10. After the tensile test piece fractures in two pieces, the two pieces are retrieved and placed together as close to
each other as possible. Now the distance between the two punch marks is measured and noted again. Let this distance be \( l_1 \). The \% elongation is calculated as

\[
\left( \frac{l_1 - l_0}{l_0} \right) \times 100
\]

(1)

Elongations indicate that material is very ductile. Low values indicate that material is brittle and has low ductility. For mild steel, the percentage elongation usually is 20% or more. The aluminium can replace the steel in the construction of the hull structures in order to reduce the total weight of the ship [2], the aluminium and the steel behave as brittle and ductile material, respectively, and their impact strength should be compared to guarantee the safety of the design.

The progressive or sudden deterioration of their mechanical strength due to loading, thermal or mechanical effects; Young’s modulus and also micro-hardness are among these mechanical properties. Hence, studying the variation of young’s modulus and micro-hardness are two of the proposed indirect methods that can be used to measure the damage parameters in ductile metals [3]. The numerical simulation is a widely used tool to optimize the sheet metal forming process and to reduce the development time and therefore the finale cost of products in manufacturing such as, automotive and aerospace. The identification step precedes the design of the manufacturing process.

Ductile fracture process is controlled by nucleation, growth and coalescence of micro voids, so it is natural to link material fracture behaviour to the parameters that describe the evolution of micro voids rather than conventional global fracture parameters [4]. Damage parameter is not incorporated into the constitutive equation and it is assumed that presence of voids does not significantly alter the behavior of the material. The von Mises criterion is most frequently used as yield criterion in uncoupled models, Damage parameter is incorporated into constitutive equation and crack growth simulation is automatically performed using a complete deterioration of elements in front of the crack tip [5].

Finite element modelling (FEM) has been widely introduced into the design of manufacturing products because of its high efficiency in predicting several problems and major defects occurring in sheet metal forming manufacturing process like necking. In order to predict those defects within a virtual manufacturing system [6, 7], an accurate description of the sheet metal mechanical behavior is an essential requirement. One of the most widely used models is Johnson-Cook model for ductile materials. Numerical simulation has been used to study the mechanical properties such as the tensile strength, the yield strength and Young’s modulus depend the temperature because Young’s modulus of some tempered steels increases slightly at mid temperatures before decreasing at high temperature on mechanical properties is linked to transformations of the material structure due to various processes [8, 9]. In order to describe the cyclic behaviour of the material for analysis with finite element method (FEM) based analysis code ABAQUS, the test data, i.e. stress-strain curves, have to be processed. This model describing fatigue behaviour of 5182 H111 Aluminium alloy under cyclic loading and their applicability for modelling of low-cycle-fatigue are discussed in this report.

2. Experimental details

The material considered in this study is the 5182 H111 Aluminium alloy. The main mechanical and thermal properties of tested specimen are summarized in table 1. The specimen geometry is shown in figure 1. Specimen is specially prepared for thermographic measurements. It is polished and cleaned to remove any oxide and grease.

The experimental setup for testing the tensile feature of specimen is illustrated in down image tensile testing and thermographic measurements were performed simultaneously. The testing of specimens was carried out on the electromechanical testing machine HOYTON TN-D, with the maximum displacement 210 mm and the strain control at room temperature.
The extension was registered using a double extensometer. The precision of the extensometer measurement is ± 0.001 mm. The thermal CAM FLIR T335 infrared camera has been used for recording thermograms. The camera resolution is 320x240 pixels. The thermal measurement is performed at a frequency of 150 Hz. It was positioned at a distance of 0.5m from the sample surface. The camera sensitivity is 50 mK at 30 °C. The camera is provided with the automatic correction of emissivity and atmospheric transmission based on the distance, atmospheric temperature and relative humidity. It simultaneously makes video and thermographic recording or tracks.

The FLIR Quick Report 1.2 SP2 is capable of measuring temperature at sports, on lines and in select areas of various shapes and dimensions, as well as of showing isotherms using the gradation of grey or the palette of various colours and shades.

3. Numerical method
In the context of thermo mechanical model with internal variables, and to get a more complete understanding of the Mechanical properties of materials during testing, numerical simulations of the aluminium 5182 H111 alloy were performed [8], to evaluate thermal loads at different locations on the surfaces and at different set as well as within the specimen. The numerical analysis using explicit dynamic finite element based on the ABAQUS Software, in order to reproduce the surface thermal loads and to obtain the temperature evolution in the specimen. The material considered in this study are the 5182 H111 Aluminium alloy, with length L= 250 mm and the depth D=5mm and the width 57 mm. Finite element modelling is performed by assuming 3D deformation. The main mechanical and thermal Properties of tested specimens are summarized in table 1 but the specimen’s geometries is shown in figure 1 and its meshing is shown in figure 2.

| Table 1. Mechanical and thermal properties of the 5182 Aluminium alloy. |
|---------------------------------------------------------------|
| Tensile modulus MPa | 2700 |
| Tensile strength (Nmm^-2) | 284 |
| Yield stress (MPa) | 154 |
| Mass thermal capacity (Jkg^{-1}K^{-1}) | 909 |
| Thermal conductivity (Wm^{-1}K^{-1}) | 209-232 |

To obtain a thermal stress loading under homogeneous uniform temperature distribution, the 5182 H111 alloy was restrained against axial expansion by creating an interaction boundary condition at its outside edge. In order to determine thermal strain in the analysis we have need to the thermal expansion coefficient, \( \alpha \) [10]. Poisson’s coefficient does not depend on temperature and takes the constant value, \( \nu = 0.3 \).
Fine meshes with 0.0012 mm elements and coarse meshes with 0.012 mm elements were used. The two meshes used are shown in (figure 2).

![Figure 2. Fine meshes 0.012 (upper) and coarse meshes 0.0012 (lower).](image)

### 4. Load and mechanical analysis of numerical simulation

A displacement boundary condition was applied at the outside extremity of the tensile specimen. The displacement was smoothly ramped up in the first portion of the test and then held constant as shown in figure 3.

![Figure 3. Load and boundary condition.](image)

### 5. Results and discussion

The tensile behavior of the specimens was determined as the load–time relation during tensile loading. Furthermore, a change in the loading curve during time was obtained. Load start, i.e. elastic strain start, then plastic strain start point where the maximum force is reached, i.e. the end of homogenous plastic strain. The results presented in figure 6 prove that the maximum elongation before final failure line started. The results presented in figure 5 prove that in the first 600 seconds the force reaches 70kN.
The beginning of plastic deformation, reaching maximum, force 95 KN, the homogeneous plastic deformation to the final fracture of the specimen. The analysis of the results presented in Figure 5 show that in the first 350 seconds the force reaches 15 kN which the elastic strain start, The maximum tensile force is about 95 kN, point that the plastic strain and final failure start. Finally, the figure 7 (a) shows that the increase damage criterion at different temperature with the time additionally the tensile temperature increase gradually with increase in the time figure 7(b).

The performance of the investigated crack propagation criteria is in general very good compared with the experimental results [8].

6. Conclusion
This paper has presented a comprehensive experimental and numerical study aimed at developing and validating a predictive model of fracture damage in plate structures under plastic conditions. The aim of the work has been to develop and use a coupled experimental–numerical procedure where the effect and modelling of fracture is not disturbed by other complex effects like buckling, contact and friction. The obtained results confirm that it is very useful to use Johnson-Cook model based on ABAQUS
software for early diagnostics of complex metal structures in the exploitation or service conditions. The obtained results prove that the Johnson-Cook model offers the possibility of non-destructive and real time testing to observe the physical process of metal degradation and to detect the occurrence of energy dissipation. A large-scale grounding experiment is also simulated showing very good agreement with measurements.

References
[1] Ilangovan S, Srikanthan R, Vyass G V International Journal of Research in Engineering and Technology 5 57-61
[2] Liu B, Villavicencio R, Soares C G 2013 Analysis and Design of Marine Structures 167
[3] Lemaitre J 2012 A course on damage mechanics Springer Science & Business Media)
[4] Abendroth M, Kuna M 2003 Computational Materials Science 28 633-44
[5] Velay V, Persson A, Bernhart G, Bergström J, Penazzi L 2002 Proc. of the 6th International Tooling Conference Karlstad 667
[6] Bauvineau L, Burlet H, Eripret C, Pineau A 1996 Le Journal de Physique IV 6 C6-33-C6-42
[7] Junis B, Rakin M, Medo B, Sedmak A 2010 FME Transactions 38 197-202
[8] Cabrera F, Manzanares J, Hanafi I, Arenas A, Garrido I, Toledano V, et al. 2014 International Review of Applied Sciences and Engineering 5 1-8
[9] Smerd R. Constitutive behavior of aluminum alloy sheet at high strain rates [Master Thesis]: University of Waterloo; 2005.
[10] Zineb A, Abdellatif K, Cabrera F M 2010 American Journal of Engineering and Applied Sciences 3 740-8