Effect of thermal and mechanical parameter’s damage numerical simulation cycling effects on defects in hot metal forming processes

Abdelouahid El Amri¹*, Mounir el yakhloufi Haddou¹ and Abdellatif Khamlichi²

¹Department of Physics, Faculty of Sciences, Tétouan 93030, Morocco
²Departments TITM, National School of Applied Sciences, Tetouan 93030, Morocco

*Corresponding author email: abdelouahid26@gmail.com

Abstract: Damage mechanisms in hot metal forming processes are accelerated by mechanical stresses arising during Thermal and mechanical properties variations, because it consists of the materials with different thermal and mechanical loadings and swelling coefficients. In this work, 3D finite element models (FEM) are developed to simulate the effect of Temperature and the stresses on the model development, using a general purpose FE software ABAQUS. Explicit dynamic analysis with coupled Temperature displacement procedure is used for a model. The purpose of this research was to study the thermomechanical damage mechanics in hot forming processes. The important process variables and the main characteristics of various hot forming processes will also be discussed.

1. Introduction
Manufacturing technologies of hot metal forming is rapidly developed in recent years. Nowadays, deep drawing, incremental hot forming, and sheet forming become very popular in hot metal forming. Ductile damage is the process of metallic material damage in conditions of monotonic loading. Evolution of the damage follows plastic straining and ends by fracture of component. Several investigations were conducted to investigate the hot forming process [1-9]. Ying et al. [1] investigate the hot forming process of 22 MnB5 steel. In metal forming processes, the plastic strain significantly outweigh the elastic strain, in many cases the rigid-plastic, or rigid-viscoplastic material behaviour is acceptable. One of the most important steps in the simulation of hot metal forming processes was the inclusion of the effects of strain-rate and temperature in material properties and the thermal coupling in forming solutions [10].

The development and application of damage models was focused on hot forming process fracture. However, in recent years, several different formulations for a variety of materials and processes have been presented, such as elastic-brittle [11, 12], brittle [22, 18], creep [17, 13], fatigue [17, 18] among others. Problems of ductile damage play significant role in industry, for example in optimization of technological processes, evaluation of safety in automotive and aeronautic industry, analysis of steel civil structures. Therefore an accurate hot forming process simulation has to be thermomechanically coupled and has to incorporate flow stress and strain rate dependency on temperature [23].

Two types of material models can be distinguished. Uncoupled models are separate plastic response and ductile damage and failure. Coupled models modify plastic response in dependence on damage evolution. Process control is easier and the variety of usable hot forming process metal alloys is wider. A thermo-mechanical analysis with a three-dimensional finite element
model is performed to compute stress-strain distribution. Therefore numerical simulation using the Finite Element Method (FEM), which is nowadays almost indispensable for the design of a hot metal forming process, is even more important for hot forming. As mentioned above, it has been proofed that in hot forming it is preferable to set the flange temperature higher than the punch temperature. That means that there are time dependent temperature gradients in the hot forming process metal. In addition, one finite element solver that offers adequate methods for advanced simulation of thermomechanically coupled processes is ABAQUS Software Commercial. The main aim of this study is to achieve hot forming process using a finite element analysis method. The other aim is focused on investigation into microstructure and thermal and mechanical influences on specimen and it mechanical properties. The results showed that the thermo - mechanical Manufacturing damage and microstructure of hot forming process especially Al5083-O aluminum alloy conformed to the literature.

2. The selected material
Currently, in the automotive and aeronautics industries, for instance, one has been very common to find plastic components submitted to the most diverse loading conditions. Automotive door knobs, window cranks and snap fits, which are usually manufactured with commodities, are good examples. In order to perform the experimental literature tests with the numerical, a Al5083-O aluminum metal alloy has chosen due to one reason, that can find many applications of plastic components subjected to ductile failure and manufactured with kind of material. The main mechanical and thermal properties of tested specimen are summarized in table 1. The geometry of the tested studied is shown in ‘Figure 1’ To obtain a thermal stress loading under homogeneous uniform temperature distribution, the steel 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 $\alpha$. Poisson’s coefficient does not depend on temperature and takes the constant value $\nu = 0.3$. 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.

![Figure1. Geometry of Al5083-O aluminum sheet metal alloy.](image)

| Al5083-O aluminum sheet metal alloy | Values                   |
|------------------------------------|--------------------------|
| Density                            | 2660 kg/m³              |
| Young’s modulus                    | 71 GPa                   |
| Poisson’s ratio                    | 0.33                     |
| Friction coefficient               | 0.1                      |
| Specific heat capacity             | 900 J/kg.K               |
| Thermal conductivity               | 117 W/mk                 |
| Thermal expansion coefficient      | $2.38 \times 10^{-5}$ K⁻¹|
3. Ductile damage model

Generally speaking, product defects in metal forming process can be addressed by employing two different categories of approaches. The first category, which is referred to as the traditional approach, consists of stress– and strain based methods and traditional fracture mechanics. Although this type of approach has been widely used in the literature, it had some limitation in predicting fracture initiation [24–27]. The second type of approach, ductile damage criterion, is a powerful tool in prediction of material failure which has been successfully employed in different forming processes including deep drawing [28].

In this paper, for prediction of tearing of the blank, the ductile damage model proposed by Hooputra et al. [29] is employed in this study. In this model, it is assumed that the equivalent fracture strain, $\varepsilon_{fr}$, is a function of stress triaxiality, $\eta$, in the form of Eq. (1) [29]:

$$\varepsilon_{fr} = d_1 \exp (-c\eta) + d_2 \exp (c\eta)$$

(1)

where $d_1$, $d_2$ and $c$ are material parameters which should be determined through experiments. Also, $\eta$ is defined in Eq. (2):

$$\eta = \frac{\sigma_{hy}}{\sigma_{eq}}$$

(2)

In which $\sigma_{hy}$ is the hydrostatic stress and $\sigma_{eq} = \sqrt{\frac{3}{2}} \sigma_{bj} \sigma_{bj}$ is the Misses equivalent stress. According to ductile damage model, the damage parameter is defined as:

$$D = \int_{\varepsilon_f}^{\varepsilon_f} \frac{d\varepsilon}{\varepsilon_{fr}(\eta)}$$

(3)

Moreover, the fracture criterion is met when $D$ is equal to a critical value $D_cr$.

In this study, the material damage parameters were determined according to the experimental procedure proposed by Bai et al. [30]; flat-grooved tensile specimens were used to investigate the effect of stress triaxiality on fracture strain. Figure 4 shows a flat-grooved specimen from both copper and steel side views. The thickness of the specimen at the groove is represented by $t$ and the radius of the groove is $R$. the groove causes stress concentration at the center of the specimen and different values of groove radii give rise to different values of stress triaxiality [30]. In order to determine the material damage parameters, one specimen with groove radii of 4mm were designed and manufactured from the produced copper/stainless steel clad sheet. The stress triaxiality at the center of the specimen is given by the modified Bridgman equation as follows [30]:

$$\eta = \frac{\sqrt{3}}{3} \left[ 1 + 2 \ln \left( 1 + \frac{t}{4R} \right) \right]$$

(4)

where $t$ is the ligament thickness of the specimen and $R$ is the groove radius. Moreover, the equivalent strain to fracture in the necking cross section of a flat-grooved specimen can be approximately determined using the logarithmic measure of strain [30]:

$$\varepsilon_{fr} = \frac{2}{\sqrt{3}} \ln \left( \frac{t_0}{t_f} \right)$$

(5)

where $t_0$ and $t_f$ are the initial and fracture ligament thickness of the specimen, respectively. This formula defines the average strain through the cross section.

Tensile tests were performed on the grooved specimen and the ligament thickness of each specimen was measured after fracture. The test result are summarized in table
4. **Numerical methods description**

The explicit finite element calculation software, ABAQUS/Explicit, was employed for the simulation of traction deformation and fracture of the Al5083-O aluminum sheet metal alloy. The size of mesh in the element model was 0.02mm×0.28mm and 0.2mm×0.12mm for the 1mm and 3mm sample, respectively. The mesh was then transitioned toward the boundaries using triangular elements and quad elements where bilinear mapping could be employed. The finite element analysis results are critical for identifying damage propagation parameters during thermal and mechanical cycles. The selected element type was an 8-node linear brick, of reduced-integration elements (C3D8R). In the finite element model, the specimen was simplified as a rigid body. The displacement load was applied on the top specimen along the 3-axis direction with the other degrees of freedom fixed. In the traction test, the Steel cylindrical rod suffered damage. The boundary element method (BEM) simplifies the meshing process and has the ability to correctly characterize the singular stress fields near the manufacturing front. The boundary conditions applied to this model were the result of a global/local approach where displacements and temperatures were taken from the results of the global model and applied to the boundaries of the local sub-model studied here. The thermal portion of the analysis should consist at least of two steps. The first step is a steady-state analysis in which the blade is brought from an initial temperature of 21.11°C to a steady operating temperature distribution and then second step simulates the cool down.

5. **Results and discussion**

Figure 2 shows the influence of the heat exchange factor on temperature field can be discussed by change the constant value. The total time is limited within 300s. Isotropic hardening contributes more large thermal effect than isotropic hardening when the heat exchange factor is fixed as 0.9. The more temperature change occurs with the increased the heat exchange factor at a fixed time. When the temperature increases, a large number of micro-cracks will inevitably be generated within the material and gradually expands with increasing temperature, significantly reducing the Young’s modulus for macro issues, while the ability of resistance to deformation decreases, which indicates that thermal stress for high temperature effect, can cause damage to materials, namely thermal damage. In our study, we used the Young’s modulus as a variable of damage, characteristic of the effect of temperature on the properties of rock damage.

![Temperature variation with time test](image)

**Figure 2.** Temperature variation with time test.
The stress field of high-power traction with varied thermal load was analyzed. The highest stress point and its position correspond to the failure point. The simulation results show in the Figure 3, indicate that specimen damage is always higher when the mechanical and thermal loads increase respectively. The specimen stress increases apparently when the temperature increases with the time, the specimen damage is varied with different loads, which corresponds with the particle temperature variation of high-power traction steel under various loads. The stress at which a material exhibits a specified deviation from proportionality of stress and strain (flow stress), that point means the manufacturing.

![Image](Figure 3. Stress variation with true strains at different temperature.)

When the homologous temperature is relatively high Figure 4, large amounts of micro-structural of material will inevitably be produced within the material, which gradually expand as the homologous temperature rises, resulting in a significant decrease in the Young’s modulus, suggesting that homologous temperature can cause failure to the steel. The basic cause of fracture strain at failure is the decrease in the Young’s modulus and we can see that the modulus is a function of temperature. Therefore, we choose the Young’s modulus as a damage variable to describe the effect of temperature on the mechanical properties of specimen. The damage variable becomes larger with an increase in temperature, indicating that the thermal expansion can cause a much faster rate of formation and expansion of micro-cracks within the rocks than before.

![Image](Figure 4. Evolution of equivalent strain at failure with T*)
Figure 5 shows the increase of elongation with increasing damage at different diameter and the significant drop in strength is caused by increasing over ageing, which reduces the effectiveness of the precipitates as obstacles to dislocations, as well as the increased thermal activation of dislocations, given that temperature and stress put the tests in the regime of dislocation creep. During the development of the tensile tests, one could observe three aspects. First, at the beginning of each test, due to the molecular chains orientation process, the narrower section of the specimens started becoming white and thinner, characterizing a pre-neck, prior appearing the neck, which will lead to the cold drawing process. The second aspect regards the remarkable ductility exhibited by the material, before failure. Lastly, one could observe that the fractured regions have occurred at the stretched end of the gauge section. The total elongation is 2.5 mm and the maximum damage when the displacement reached the plastic deformation.

![Elongation vs Time](image.png)

**Figure 5.** Damage variation at different temperature traction test.

6. Conclusions
A finite element model based on ABAQUS software was used to study the effect of thermal and mechanical parameters on Al5083-O aluminum sheet metal alloy damage. The fully coupled elastic-plastic-damage model was developed and implemented into an explicit code. It is concluded that finite element analysis (FEA) be applied to predict fracture for Al5083-O aluminum sheet metal alloy. A validation study has been performed comparing Finite Element Analysis methods based on Abaqus software of notched bar subjected to impact thermal and mechanical loadings with literature experimental results obtained in a Split Hopkinson Tension Bar. Fracture strain was calculated on the base of the specimen extension at material failure. More accurate ductile fracture description in range of calibration experiments was reached using Johnson-Cook model. Both material models successfully describe ductile damage of calibration specimens that are commonly presented in the literature.

References
[1] Ying C, Zhao-huan M, Liang Y, Xiao-dong L, Ning M and Ping H 2011 *Research International* 18 59 – 63
[2] Yanagida A and Azushima A, 2009. *Annals Manufacturing Technology* 58 247 – 250
[3] Merklein M, Lechlér J and Stoehr T 2009 *International Journal of Material Forming* 2 259–62
[4] Abdulhay B, Bourouga B and Dessain C 2011. *Applied Thermal Engineering* 31 674–85
[5] Hongtu S, Ping H, Ning M, Guozhe S, Bo L and Dinglu Z 2010 *Chinese Journal of Mechanical Engineering* 23 1–6
[6] Liu H, Bao J, Xing Z, Zhang D and Song B, Lei C 2011 Journal of Materials Engineering and Performance 20 894–902
[7] Liu H, Liu W, Bao J, Xing Z, Song B and Lei C 2010 Journal of Materials Engineering and Performance 20 1–10
[8] Bai-liang Z, Zhong-de S, Chao J and Wen-juan R 2010 Proc. of Int. Conf. on Advanced Technology of Design and Manufacture
[9] Ying X, Zhong-de S, Mi-lan Z and Chao J Proc. of the Int. Conf. on Advanced Technology of Design and Manufacture
[10] Boër C R, Rebelo N, Rystad H and Schröder G (1986) Process Modelling in Metal Forming and Thermo-mechanical Treatment (Berlin: Springer-Verlag)
[11] Murakami S 1988 Journal of Applied Mechanics 55 280–6
[12] Murakami S and Kamiya K 1997 International Journal of Mechanical Sciences 39 473–86
[13] Krajcinovic D and Fonseka G U 1981 Journal of Applied Mechanics 48 809–15
[14] Krajcinovic D. 1983 Journal of Applied Mechanics 50 355–60
[15] Simo J C and Ju J W 1987 International Journal of Solids and Structures 23 821–40
[16] Simo J C and Ju J W 1987 International Journal of Solids and Structures 23 841–69
[17] Chaboche J L 1988 Journal of Applied Mechanics 55 59–64
[18] Chaboche J L 1988 Journal of Applied Mechanics 55 65–72
[19] Lemaitre J 1984 Nuclear Engineering and Design 80 233–45
[20] Krajcinovic D and Selvaraj S 1984 Journal of Engineering Materials and Technology 160 809–15
[21] Saanouni K, Chaboche J L and Bathias C 1986 Engineering Fracture Mechanics 25 677–91
[22] Murakami S. 1990 Role of continuum damage mechanics in fracture analysis Inelastic solids and structures (Swansea: Pineridge Press) p 67–80
[23] Boogaard A H 2002 Thermally Enhanced Forming of Aluminium Sheet (PhD. thesis) University of Twente