Finite Element Analysis of an Aluminum Alloy Using Progressive Failure Algorithm

***M.F. Hossain*[1], *K.M. Shorowordi*[1], *M.S. Islam*[2]**

[1]Department of Materials and Metallurgical Engineering, Bangladesh University of Engineering and Technology, Dhaka 1000, Bangladesh
[2]Department of Naval Architecture and Marine Engineering, Bangladesh University Engineering and Technology, Dhaka 1000, Bangladesh

Abstract. This study aims to develop a reliable finite element analysis procedure to model the complete fracture of ductile specimens using the progressive degradation of the material stiffness algorithm under tensile load. The ductile specimen in this study is an aluminum alloy. The progressive failure algorithm used here is based on the assumption that the material behaves like a stable progressively fracturing solid. The stiffness reduction is carried out at the reduced integration gauss points of the finite element mesh depending on the mode of failure. A number of material properties are necessary for such simulation to carry out and experimentation of the metal are needed to evaluate these properties. The actual tensile tests data are applied to the finite element simulation. The verified simulation method has a great importance in practical design of structures and metals. A renowned finite element analysis software ABAQUS is used in this study.

1. Introduction

Aluminium is the key material in aerospace and automobile industry. Among the aluminium alloys, Al 7075 has high strength to weight ratio and is increasingly applied to manufacture of automobiles and aircrafts [1]. Recent researches shows that these aluminium alloys can attain strength as high as 1 GPa [2]. Having boron in this alloy decreases its grain size and thus improve its strength. The addition of nickel hardens the surface of the alloy and thus increases its overall property [3,4]. A 7075 alloy modified with nickel and boron can be potential to achieve excellent properties which are lucrative for the strengh related applications. Being this highly important material, it is evident that a properly developed simulation model will allow further maneuverability in its application by aiding in complex designs.To understand the material, tensile tests until fracture are often employed. Having a realistic tensile simulation of this material until fracture using damage models means that the simulation results will also, allow researchers to have an in depth understanding of this material. And also, these simulation will enable further ease of process designing such as in various drawing processes [5].

Obtaining reliable experimental results are extremely important to obtain a reliable simulation but the key challenges here are choosing the right damage criterion and the proper meshing of the specimen. Through the use of meshing in a rectangular tensile specimen EL, Amri Abdelouahid et al. showed that different size of meshing shows different results in the simulation [6]. Most of the time using finer mesh means that the result will converge to the real experiments but that also means higher computational time so a meshing should be such that it matches the realistic data

IOP Conf. Series: Materials Science and Engineering **438** (2018) 012022 doi:10.1088/1757-899X/438/1/012022

but not too much fine that it consumes unnecesssarily high amount of computaional energy. And about the fracture damage models, although there are many damage models, it is reported that the phenomenological damage models (i.e. johnson cook , ductile damage models) are the best to employ when experimental data are at hand since the continuum damage mechanics shows systematically the effects of damage on the mechanical properties of materials and structures as well as the influence of external conditions and damage itself, and thus accurately representing the real situations [7,8]

In this study a simulation of tensile test using a computer program with the aid of real experimental data is done by applying various mesh element sizes and using the ductile damage model as a representative of phenomoligical damage model.

2. Methodology

2.1 Material Preparation

The present investigation was carried out on modified 7075 Al alloy with nickel and boron whose composition is shown in Table 1. The material was cast at 850˚C in the form of rectangular bar in a metal mold.

Table 1. Chemical composition of Al 7075 alloy

Elements	Weight Percentage	Elements	Weight Percentage
Zinc	6.0	Chromium	0.3
Magnesium	3.0	Zirconium	0.2
Copper	2.0	Boron	0.01
Nickel	1.0	Aluminum	Balance

2.2 Heat Treatment

Two types of heat treatments were applied namely solution heat treatment with rolling and precipitaion hardening. Samples were machined and then Ni and B modified Al 7075 alloy was at first solution treated at 480 ˚C for 5 hours and later hot rolled at 450 ˚C to facilitate recovery and recrystallization [9]. After the reduction of about 66 percent in height the material was quenched in cold water. During the reduction the specimen was reduced from 15 mm to 5mm decreasing about 0.66mm at each rolling pass to avoid the material failure during the hot rolling. Then the sample was aged at 120 for 24 hour to impart some precipitation strengthening.

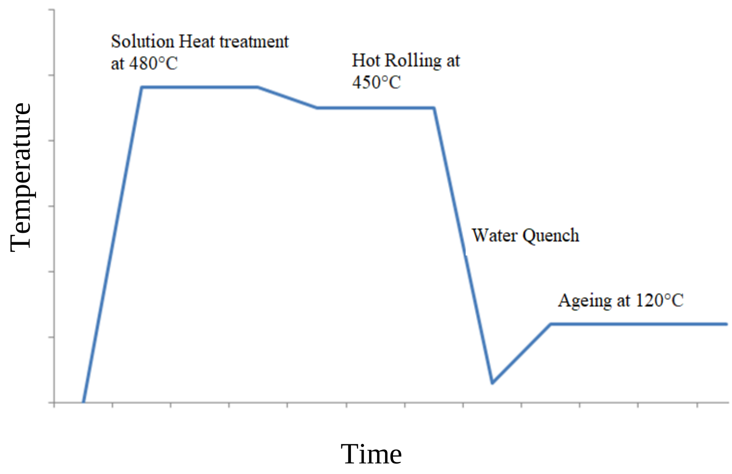

Figure 1. Heat treatment cycle of modified Aluminum 7075 alloy

2.3 Simulation Data

2.3.1 Density, Plastic and Elastic parameters. To simulate tensile damage and failure we have collected our tensile test data in the load elongation form. This data is later converted into stress and strain form. But to use these data in finite element software we need true stress-true strain data because with the true stress data we can simulate realistic results which can be obtained by the equations below

$$\sigma = S(1+\delta) \tag{1}$$
$$\in = \ln(1+\delta) \tag{2}$$

- σ = True stress
- $\in$ = True strain
- S = Engineering stress
- δ = Engineering strain

From the true stress true strain curve modulus of elasticity is calculated with the equation below:

$$\text{Modulus} = \text{True Stress at Yield} / \text{True Total Strain at Yield} \tag{3}$$

We calculated our youngs modulus to be 73 MPa. Densty was calculated 2.4g/cm^3 and the poisons ratio was taken from literature which is .33[10]

In order to include plasticity within Abaqus, the stress-strain points past yield, must be input in the form of true stress and logarithmic plastic strain. The logarithmic plastic strain required by Abaqus can be calculated with the equation given below.

$$\in^{plastic} = \ln(1+\delta) - \frac{\sigma}{\text{Youngs Modulus}} \tag{4}$$

Now, this true stress plastic strain data is good to apply in Abaqus as input data.

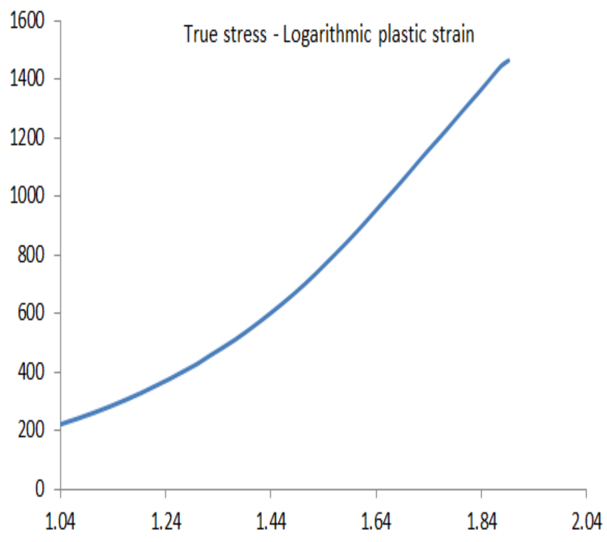

Figure 2. True stress-Logarithmic plastic strain data

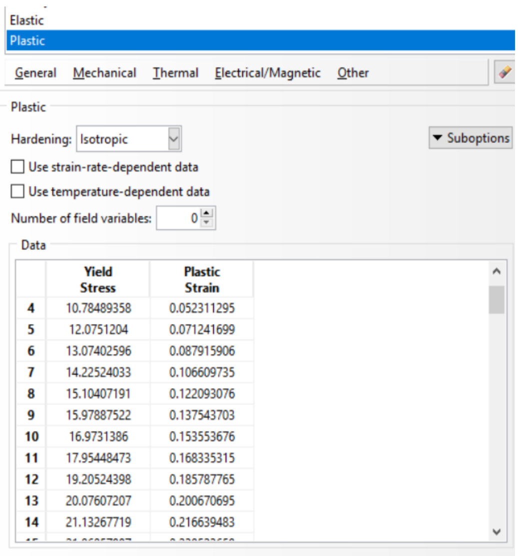

Figure 3. Data imported to Abaqus

2.3.2 Damage Parameters. In the damage parameters strain rate can readily be obtained from the experiment itself . In our case the strain rate was 0.25mm/s. For stress triaxiality as a practical approach, the stress triaxiality is used to identify the state of stress and is defined as follows:

$$\text{Stress triaxiality} = \frac{\sigma(\text{mean normal stress})}{\sigma(\text{effectice stress})} \tag{5}$$

$$\sigma_{\text{mean normal stress}} = (\sigma_x + \sigma_y + \sigma_z)/3 \tag{6}$$

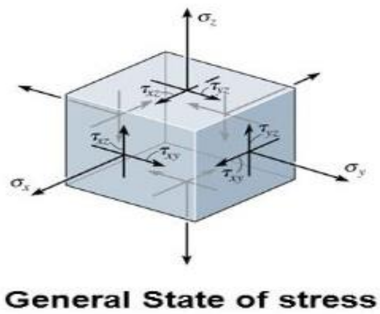

Figure 4. State of stresses in various directions

$$\sigma_{\text{effective stress}} = \frac{1}{\sqrt{2}}\sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2)} \tag{7}$$

- $\sigma_x, \sigma_y, \sigma_z$ = stress in three directions.
- τ = shear stresses.

The unique situation for uniaxial test is that it can be assumed until necking all of the stress state except in y and z directions are zero and also the scalar value of the stresses in y and z direction are equal with having opposite signs. Using the equation 5, 6 and 7 the value of stress triaxiality leads to 0.33. So, we can assumed our stress triaxiality value as 0.33 upto necking[11]. And for fracture strain it is equals to the equivalent plastic strain and under tensile loading, the equivalent plastic strain exactly equals to the tensile plastic strain at rupture from stress-strain diagram. Al 7075 properties are given in Table 2.

Table 2. Properties of Al 7075 alloy

Property	Value
Density	2.4 g/cm^3
Youngs modlus	73000
Poissons ratio	0.33
Fracture strain	0.6
Stress triaxiality	0.33
Strain rate	0.25 mm/s
True stress- plastic strain	Data imported in abaqus in tabular form

IOP Conf. Series: Materials Science and Engineering **438** (2018) 012022 doi:10.1088/1757-899X/438/1/012022

2.3.3 Specimen geometry. The test specimen is modelled on Abaqus in accordance with the tensile sample as much as possible. Dimension of the sample is given in Table 3 and the samples are shown in figure 4.

Table 3. Dimension of the sample

Gauge length	Width	Thickness	Fillet radius	Overall length	Length of reduced section	Length of grip section	Width of grip section
25 mm	15 mm	4 mm	6 mm	80 mm	32 mm	22 mm	7.5 mm

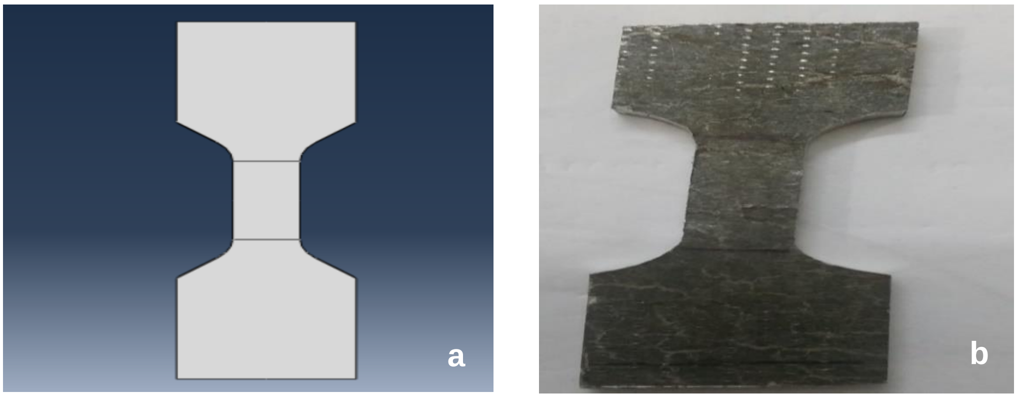

Figure 5. Showing the tensile specimen for (a) simulation and (b) real test

2.3.4 Meshing. As discussed above meshing is an important part for obtaining proper results in the simulation. Meshing of the sample was done in four different ways with C3D8 8-node linear brick, hourglass control mesh elements. The data obtained by meshing in the simulation of the material are tabulated in Table 4 and meshing is shown in Figure. 5.

Table 4. Element size and Simulation run time

Model No.	Total Number of Elements	Total number of nodes	Mesh size	Simulation run time required (s)
Model 1	1674	2496	1.6	118.23
Model 2	2973	4312	1.2	245.33
Model 3	11780	14940	0.8	1795.3
Model 4	95790	108273	0.4	38770.8

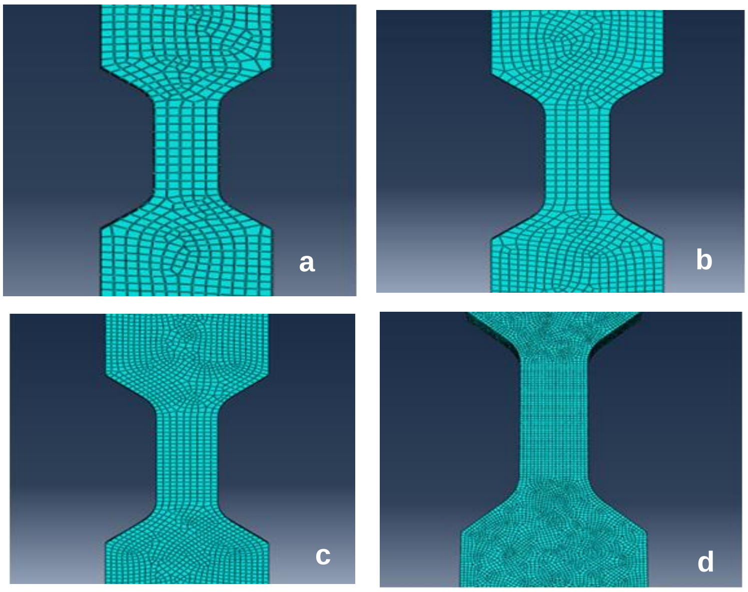

Figure 6. Showing different meshing style of the sample 3

3. Results and Discussion

The data obtained from the simulations show that as the mesh becomes finer the simulation time increases dramatically. On the other hand with the finer mesh size the simulation starts to converge with the real result. Simulation with 95790 elements showed very good convergence with the real data which is apparent in figure 6(d) whereas the model 1 with the lowest number of elements showed poorest result as evident in figure 6(a). So, it can be concluded that in the simulation particularly in tensile test mesh size plays an important role and with the higher mesh density, we got more realistic result.

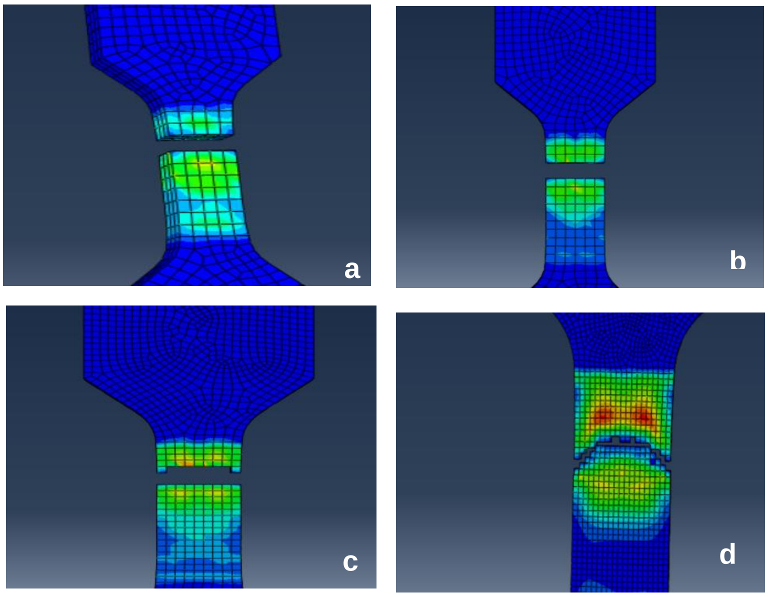

Figure 7. Simulations showing the place of failure and fracture geometry (a) model 1, (b) model 2, (c) model 3, (d) model 4.

Also, the place of the fracture in all the simulations was nearly as same as the real experiments. So, it is evident that the simulation is quite accurate in the context of fracture geometry and the place where the fracture occurred. And that is obtained just by inputting values of only a few variables obtained from a tensile test. So, using this simulation results it is possible to predict what may happen in the complex structures that uses this material by using a lot less experimental resources that would otherwise be required.

There is, also, a discrepancy with the real results and simulation in the portion where the crack propagated. The simulation shows how ideally fracture should look like and initially in the real experiment, the crack started to propagate as it should be. But as there are lots of defects in the real sample, the crack had a lot of possible pathways to propagate. So, as evident in the figure 7, the crack path differed at the later portion of the specimen. Nevertheless, from figure 7(c) it can be seen that the final fracture geometry is somewhat similar to the simulation.

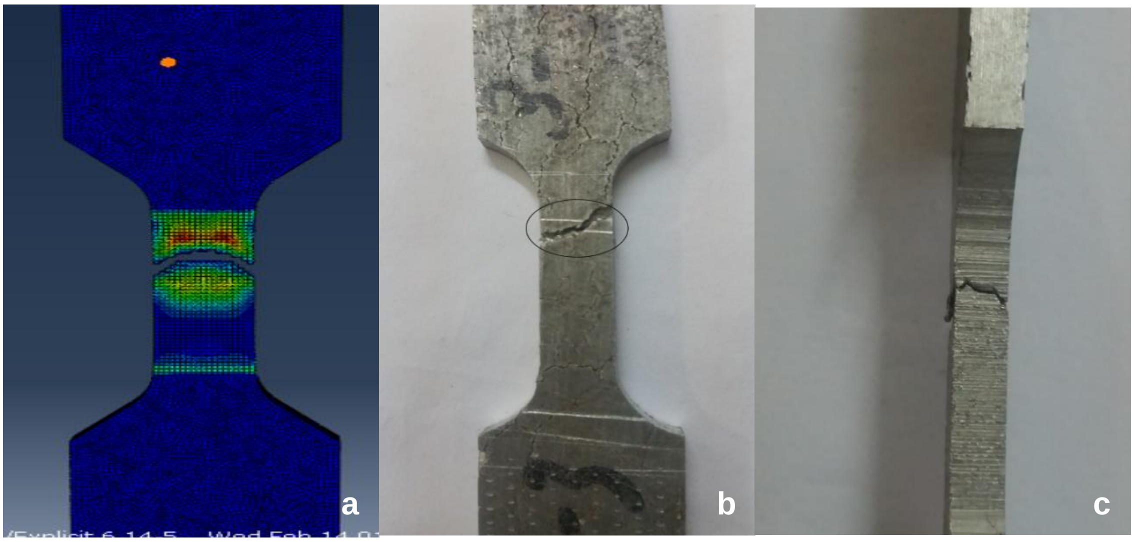

Figure 8. Fracture geometries. (a) Simulated fracture geometry (b) Fracture geometry from the front view (c) Fracture geometry from the side view.

4. Conclusion

Finite element analyses of uniaxial tensile tests were conducted using ABAQUS software. With the tricky choice of mesh size and the damage model it could quite accurately predict the place of necking and fracture geometry. Although, there were some differences in crack propagation due to the presence of defects in the real sample it can be said that the simulation did almost match the real scenario. Also, it is evident that the numerical accuracy of the simulation increased with the decreasing mesh size but the computing time increased with the decreasing mesh size. Overall, this simulation showed high convergence to the real data so the data from the simulation can be used in the process and structural areas where these materials are used.

5. Reference

[1] E.A.StarkeJr 1996 Progress in Aerospace Sciences (Virginia, Charlottesville) vol 32 pp 131-172.

[2] Liddicoat, P., Liao, X., Zhao, Y., Zhu, Y., Murashkin, M., Lavernia, E., Valiev, R. and Ringer, S. 2010 Nanostructural hierarchy increases the strength of aluminium alloys. Nature Communications 1(6), pp.1-7.

[3] Jasim M. Salman1 , Shaymaa Abbas Abd Alsada2 and Khadim F. Al-Sultani 2013 *Research Journal of Material Sciences* vol 1(6) pp 12-17.

[4] Brunelli K, Dabalà M, Martini C. 2006 Surface hardening of Al 7075 alloy by diffusion treatments of electrolytic Ni coatings *Metallurgia Italiana* p37.

[5] Venkateswarlu, G., Davidson, M. and Tagore, G. (2011). Influence of process parameters on the cup drawing of aluminium 7075 sheet *International Journal of Engineering, Science and Technology* vol 2(11).
[6] EL Amri Abdelouahid 2015 22ème *Congrès Français de Mécanique* (Lyon,France)
[7] Lemaitre, J. 1985 *A continuous damage mechanics model for ductile fracture*. J. Eng. Mater. techno. 107 pp 83–89.
[8] Lemaitre, J., 1985 *Coupled elasto-plasticity and damage constitutive equations* Comp. Meth. appl.Mech. Eng. pp 31–49.
[9] Abolhasani, A. Zarei-Hanzaki, A., Abedi, H. and Rokni, M. 2012 The room temperature mechanical properties of hot rolled 7075 aluminum alloy. *Materials & Design*, 34, pp.631-636.
[10] Asm.matweb.com.(2018). *ASM Material Data Sheet*. [online] Available at: http://asm.matweb.com/search/SpecificMaterial.asp?bassnum=MA7075T6.
[11] Mirzajanzadeh M, Canadinc D. A 2016 Microstructure-Sensitive Model for Simulating the Impact Response of a High-Manganese Austenitic Steel *Journal of Engineering Materials and Technology* vol 138(4).