Proceedings of the 29th Polish-Slovak Scientific Conference on Machine Modelling and Simulations 2024, Poland

Determination of Physical Fracture Conditions of AW-1050A Aluminum Specimens Under Different Material Weakening Geometries

Z. Saternus*, T. Domański and M. Kubiak

Czestochowa University of Technology, Faculty of Mechanical Engineering, Dąbrowskiego 69, 42-201 Częstochowa, Poland

Doi: 10.12693/APhysPolA.145.802 *e-mail: zbigniew.saternus@pcz.pl

The work concerns the experimental and numerical determination of the physical conditions of the cracking process of samples made of AW-1050A aluminum. The AW-1050A material is low-alloy aluminum and is widely used in the food industry, architecture, and energy industry. The tested samples are subjected to axial external loads until the material ruptures. In the conducted research, the main attention is focused on the physical conditions in the crack zone for various material weakening geometries. The experimental tests are the basis for the verification of the developed 3D discrete models. The geometries in the numerical model and the developed material model are based on experimental research. The ductile damage model is used to predict the initiation of material cracking. The obtained results of numerical simulations are compared with the experiment. The main aspect of the results is the comparison of the fracture zones in the material.

topics: low-alloy aluminum, cracking process, numerical model, ductile damage

1. Introduction

Knowledge of the strength parameters of construction materials is important information for designers and technologists [1–4]. The strength tests provide all the necessary information about the material parameters [3–6]. The introduction of holes into structural elements can cause a significant weakening of the material compared to solid material.

Conducting experimental studies to check the effect of hole shape on strength is time-consuming. Therefore, scientists are developing numerical models that allow for a preliminary assessment of the influence of hole geometry on material weakening. At the initial stage of developing such numerical models, it is necessary to verify the simulation results by comparing them with experimental data [7, 8].

In the presented work, the main attention was focused on the numerical modeling of fracture during a static tensile test. Simulation calculations were performed in the commercial engineering software package Abaqus FEA. In the Abaqus/CAE module, a three-dimensional (3D) discrete model was developed, which was based on a real object. As part of the work, experimental studies were performed. The tests were carried out for three standardized measuring samples made of low-alloy AW-1050A aluminum. The samples contained different material weakening geometries: two samples with holes and one solid sample. The conducted experimental

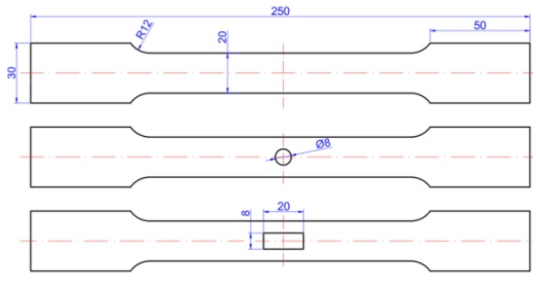

Fig. 1. Geometric dimensions of samples.

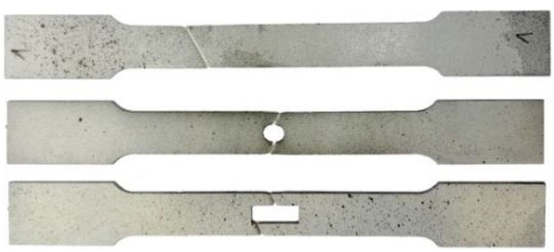

Fig. 2. Set of damaged samples.

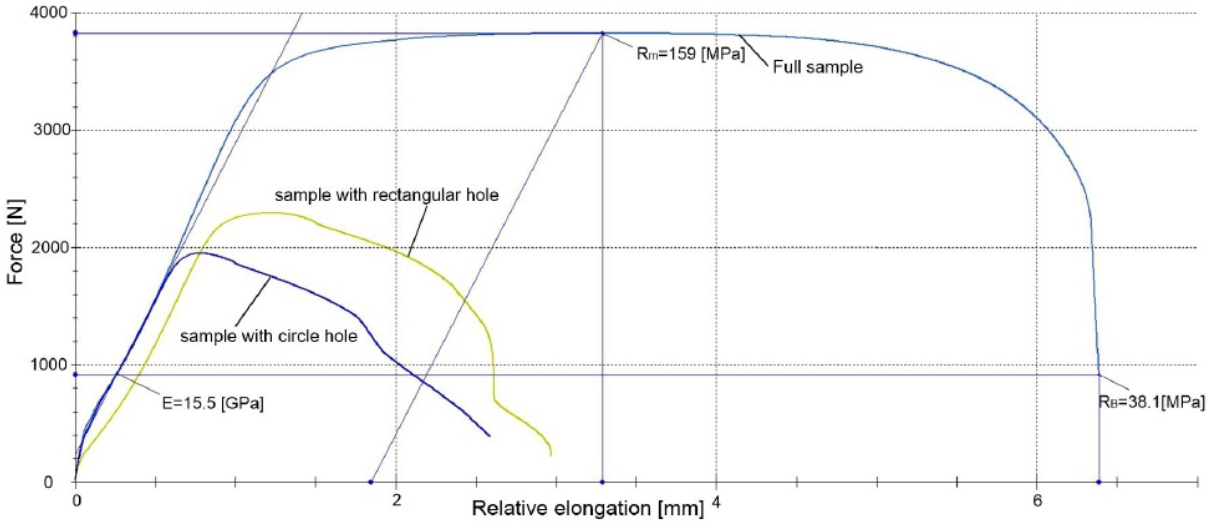

Fig. 3. Tensile graph for three different samples.

studies were necessary to verify the developed numerical models. The experimentally obtained stress–strain diagram for the full sample was used to develop a numerical model of the material. The ductile damage model [2–5, 7] was used to model the cracking in the Abaqus software. The obtained results of numerical simulations were compared with experimental studies. Then, appropriate conclusions were drawn.

2. Experimental research

Experimental tests of axial tensile strength of samples with different cross-section weakening were performed on a universal testing machine Zwick/Roell Z100 with a maximum load of 100 kN and precision of 1 N force/0.01 mm displacement [8]. The tests were conducted in accordance with the PN–EN 10002-1 standard. The material from which the samples were made is low-alloy aluminum AW-1050A, which is characterized by high plasticity. The chemical composition of the aluminum alloy is: ≤ 0.05 Mg, ≤ 0.05 Mn, ≤ 0.4 Fe, ≤ 0.05 Si, ≤ 0.05 Cu, ≤ 0.07 Zu, ≤ 0.05 Ti, ≥ 99.5 Al [%].

The tests were carried out for three different flat samples obtained from a 1.2 mm thick sheet. The samples were made using the water jet cutting technique. The diagram of the prepared samples is shown in Fig. 1.

Tensile tests were performed for a flat sample without a hole and for samples with rectangular and circular holes. Figure 2 shows a set of samples after testing.

Tensile stress–strain diagrams were obtained for each test and are presented in Fig. 3.

The stress–strain diagram for the solid sample was used as an input data set to prepare the material model in Abaqus software.

3. Numerical material model

An important element of conducting numerical simulations is processing the data of the material from which the element is made. This has a significant impact on the quality of the obtained numerical simulation results. The experimentally obtained stress–strain diagram of a flat sample (see Fig. 4) can be effectively used as input data in numerical calculations. In order to reduce the amount of data entered into the Abaqus program module (the graph from the testing was machine-generated over 6 000 points), the number of points was reduced to the necessary minimum so that the quality of the stress–strain curve was preserved (≈ 100 points).

The modeling of the elastic range in the program is based on the classical Hooke's law. The experimentally determined Young's modulus will be used in the calculations. The Poisson's ratio was 0.33. When entering plasticity data, Abaqus FEA requires a true stress–strain graph [2, 7]. Using the following equations, the experimentally obtained engineering graph σ_{En} should be converted to the real system σ_{True}

$$\sigma_{\mathrm{En}} = \frac{F}{A_0}, \tag{1}$$

$$\sigma_{\mathrm{True}} = \frac{F}{\Delta A} = \frac{FL}{A_0 L_0} = \frac{F}{A_0}\left(1+\varepsilon_{\mathrm{En}}\right), \tag{2}$$

$$\sigma_{\mathrm{True}} = \sigma_{\mathrm{En}}\, m\left(1+\varepsilon_{\mathrm{En}}\right), \tag{3}$$

where F denotes forces [N], A_0 — initial cross-sectional area [mm^2], ΔA — actual cross-sectional area [mm^2], L — measurement length [mm] and L_0 — initial measurement length [mm], and ε_{En} — actual displacement. The determination of true strain (ε_{True}) on the basis of the real displacement (ε_{En})

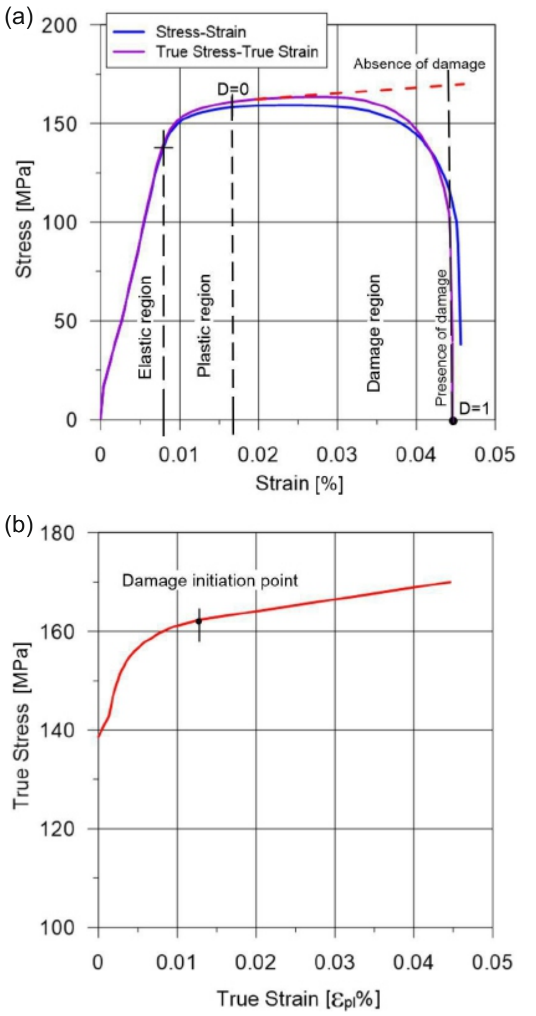

Fig. 4. Comparison of engineering diagram with stress–strain diagram (a), plastic deformation diagram (b).

Fig. 5. Discrete model developed for a flat sample.

is performed as follows

$$\varepsilon_{\mathrm{En}} = \frac{\Delta L}{L_0} = \frac{L}{L_0} - 1, \tag{4}$$

$$\varepsilon_{\mathrm{True}} = \int_{L_0}^{L} \frac{\mathrm{d}L}{L} = \ln\left(\frac{L}{L_0}\right) = \ln\left(1+\varepsilon_{\mathrm{En}}\right). \tag{5}$$

The calculated true stress–strain diagram is shown in Fig. 4.

The Abaqus program uses the plasticity curve presented in Fig. 4 until the tensile strength limit Rm is reached. After exceeding this limit, the program starts using the ductile damage cracking model [9]. The ductile damage model is responsible for modeling the formation of a neck on the

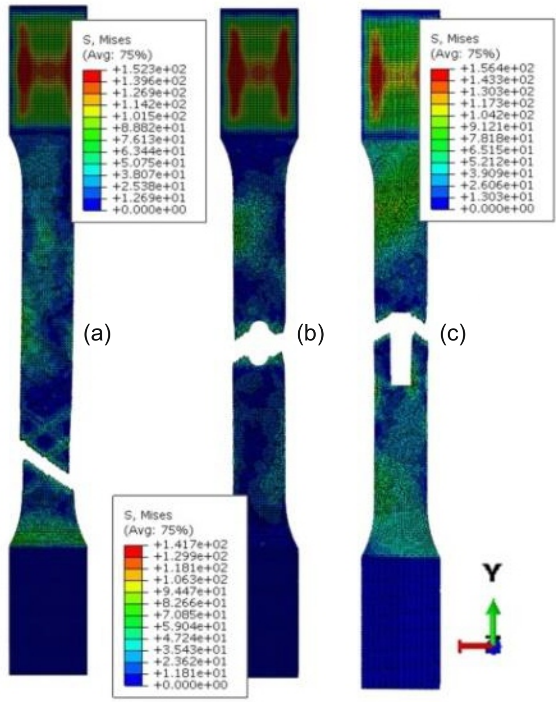

Fig. 6. Distribution of reduced stresses in the sample: (a) flat, (b) with a round hole, (c) with a square hole.

sample and the rupture of the material. According to the software documentation, the model assumes that the equivalent plastic strain at the onset of damage is a function of stress triaxiality and strain rate [9].

4. Numerical model

Numerical calculations were performed in the Abaqus/Explicit calculation module. Three-dimensional discrete models of the analyzed elements were developed according to the dimensions in Fig. 1. The models use the material data for low-alloy aluminum samples presented in Sect. 3. The specified boundary conditions reproduced the conditions prevailing in the experiment. For each of the samples within the finite element mesh measurement zone, a finite element method (FEM) mesh refinement of 0.5×0.5 mm^2 was used (Fig. 5). The mesh element dimension in thickness was 0.4 mm. The sample loading parameters were adopted in accordance with the experiment.

5. Results and discussion

In the Abaqus calculation module, calculations were performed for all three types of samples. Figure 6 shows the obtained results of numerical simulations.

The numerically estimated crack locations are comparable with the results obtained experimentally (compare Fig. 2 and Fig. 6). The differences between the numerically obtained results and the experimental results may result from factors such as the accuracy of the entered material data, the adopted plastic damage model, and the density of the FEM mesh used.

6. Conclusions

Numerical prediction of the physical conditions of the fracture process of samples made of AW-1050A aluminum requires the introduction of many specific data into the numerical analysis. By analyzing the obtained simulation results along with the experimental tests, it can be concluded that the fracture locations of the materials are comparable.

The presented numerical model can be effectively used to conduct analyses for samples with various shapes of material weakening. The developed numerical model of the material (low-alloy aluminum AW-1050A) can be effectively used for further numerical analyses.

References

[1] Q. Chen, Z. He, *Composites Communications* **45**, 101801 (2024).

[2] J. Chen, O.V. Magdysyuk, X. Li, P.J. Withers, K. Yan, *Eng. Fract. Mech.* **293**, 109711 (2023).

[3] A.V. Shutov, V.S. Klyuchantsev, *Eng. Fract. Mech.* **292**, 109656 (2023).

[4] Ch. Zeng, X. Fang, N. Habibi, S. Münstermann, J. Lian, *Eng. Fract. Mech.* **306**, 110217 (2024).

[5] Z. Wei, S. Gerke, M. Brünig, *Int. J. Mech. Sci.* **276**, 109380 (2024).

[6] M. Wojtaszek, T. Śleboda, A. Czulak, G. Weber, W.A. Hufenbach, *Arch. Metall. Mater.* **58**, 1261 (2013).

[7] M. Khademi, M.J. Mirnia, H.M. Naeini, *Int. J. Solids. Struct.* **301**, 112947 (2024).

[8] T. Domański, W. Piekarska, Z. Saternus, M. Kubiak, S. Stano, *Materials* **15**, 3243 (2022).

[9] SIMULIA Abaqus FEA theory manual. Version 6.7, Dassault System (2007).