# Computer simulation of superplastic tensile test

**Vadim Mikolaenko** [1,2]

[1] National Research University Higher School of Economics, 123458 Tallinskaya 34, Moscow, Russia
[2] Correspondence: vmikolaenko@hse.ru

**Abstract.** Superplasticity is an ability of polycrystalline materials to archive extremely large deformations, which is utilized in advanced forming technologies demanded mainly in the aerospace industry. Design of such technologies needs an information of the material flow behaviour, which describes the relation of the effective stress on the strain and strain rate taking place during deformation. The most popular experimental method for investigation of the flow behaviour of superplastic materials is tensile testing. The procedure of superplastic tensile testing and interpretation of its results is described by several international standards. At the same time, it is known that the due to flow inhomogeneity in the specimen volume the accuracy of such tests may be violated. Moreover, different standards provide different ratio between the width and the height of the gauge area of a specimen. This work provides the numerical analysis aimed to study how the initial specimen geometry affects the results of tensile tests. A computer program implementing finite element method (FEM) was developed to predict the specimen deformation during the test. A flat specimen is discretized using prismatic elements with specific geometrical constraints reducing the degree of freedom to the order of a plane stressed task. The output stress and strain values were calculated as specified in the ASTM E2448 standard. The effect of the gauge length was studied focusing on the output stress strain curves. The results were compared with the experimental results available in the literature.

## 1. Introduction

Superplastic materials are used for gas forming, which makes it possible to obtain thin-walled products of complex geometric shapes. To create such a product, it is critically important to set correctly the pressure and temperature regime [1,2], which are determined proceeding from the material properties. Consequently, there has been made persistent attempts to conduct technological experiments with stress conditions similar to the real forming processes aiming to provide the information about the flow behaviour of superplastic materials [3–6]. Nevertheless, the main method for studying superplastic material properties is mechanical tensile testing at a constant strain rate. The methodology for carrying out such a test and the interpretation of its results is set out in the international standards [7,8]. However, the inhomogeneity of the strain rate distribution in the specimen is neglected as if it is an uniaxial tension. The inhomogeneity takes place because of the viscoplastic character of deformation: material flows from the grip section to the gauge section, that leads to the change in volume of the gauge section. Thus, the specimen geometry affects the results of tensile tests. The effect of specimen geometry on the predicted deformation behaviour of aluminium alloy 5083, was analysed experimentally and numerically in many studies [9–13].

Bate at al. [12] made an experimental research of the effect of the gauge length to width ratio on the tensile test results of commercial superplastic Al–45Mg alloy. The specimens were marked with a grip and photographed before and after tensile testing with a digital camera. As a result, it was found that the volume of the gauge section does not remain constant because of the griping. Moreover, reducing the gauge proportion of the Al–45Mg alloy specimen leaded to doubling apparent elongation. It was summed up that tensile test result's error depends on the mechanical behaviour of the material, so standardizing geometry of the specimen is unlikely to solve such a problem.

Extensive experimental research, involving 24 different specimen geometries, was carried out by Abu-Farha et al. [13]. Different variations of gauge length, gauge width, grip length and grip width were studied at the same conditions for the commercial AZ31B-H24 magnesium alloy. It was shown that the gauge length to gauge width ratio, as well as the grip width to gauge width ratio, should be equal to four for tensile test results to be more accurate.

Computer simulations can be used for deeper understanding of material flow in the specimen volume during the tensile test, so the inhomogeneity of the strain rate can be taken into account [11]. In this study this work the software product was developed based on finite element modeling to implement such computer simulations. By means of this product the effect of the specimen initial geometry and material properties on the tensile test results were studied. The output stress-strain curves which, were calculated following the standard ASTM E2448 procedure, were calculated and compared with the input ones in order to evaluate the accuracy of tensile testing.

## 2. Models and methods

Modeling the slow shape change of a rigid body is reduced to solving a quasi-static problem, where at each step a system of equations is solved, consisting from the equilibrium equation and boundary conditions: force boundary type on $\Gamma_\sigma$ specimen frame, kinematic boundary type on $\Gamma_u$ and mixed type on $\Gamma_{\sigma u}$.

$$\begin{cases} \sigma_{ij,j} = 0 \\ \overline{P_n} = \sigma_{ij}\, L_{n_i}\, \overline{e_i}\,, \;\; on\, \Gamma_\sigma \\ \overline{v} = v_i\, \overline{e_i}\,, \;\; on\, \Gamma_u \\ \overline{P_\tau} = \sigma_{aj}\, L_{\tau_a}\, \overline{e_j}\,, \quad \overline{v} = v_\beta * \overline{e_\beta}\,, \;\; on\, \Gamma_{\sigma u} \end{cases} \tag{1}$$

where $\sigma_{ij}$ are stresses; $\overline{P_n}$ – vector of distributed surface forces acting on the normal to the applied fragment of the surface; $L_{n_i}$ – component of the normal according to the $i$-th axis; $\overline{e_i}$ – unit vector of the local basis, outgoing from the point of application of the force; $\overline{v}$ – velocity vector; $v_i$ – component of the velocity vector in accordance with the $i$-th axis. This problem can be solved by the finite element method allowing to replace the initial formulation of differential equations with the system of algebraic equations of the following form:

$$[K] * \overline{\delta} = \overline{F_{ext}} - \overline{F_{\sigma^*}}, \tag{2}$$

where $[K]$ – stiffness matrix; $\overline{\delta}$ – velocities in the nodes of the elements which are used for discretization of the body; $\overline{F_{ext}}$ – force due to the action of distributed intensity loads on the body surface; $\overline{F_{\sigma^*}}$ – force due to accumulated hydrostatic pressure.

The reference initial geometry is taken from the ASTM E2448 standard [7]. Figure 1 illustrates 1/8 of the whole specimen that was used in computer simulations with the symmetric boundary conditions at the faces: XY, XZ, and YZ. Since the specimen is thin, it can be discretized using prismatic elements with the following geometric constraints imposed on its nodes: nodes $i_0,\ j_0,\ m_0$ lie in the face $XY$; nodes $i,\ j,\ m$ lie on the specimen surface; ribs $i_0 i,\ j_0 j,\ m_0 m$ parallel to the axis $Z$. This constrains allows to reduce the number of degrees of freedom in discretized task. Figure 2 illustrates finite element described above. The distribution of velocities inside the prismatic element is set in such a way that the velocities corresponding to the X and Y axes at any point inside the element do not depend on the z coordinate, and the velocity corresponding to the Z axis depends on the $z$ coordinate linearly. For the considered elements, the stiffness matrix $K$ and the vectors of the right-side for the main equation of the finite element method (2) were calculated.

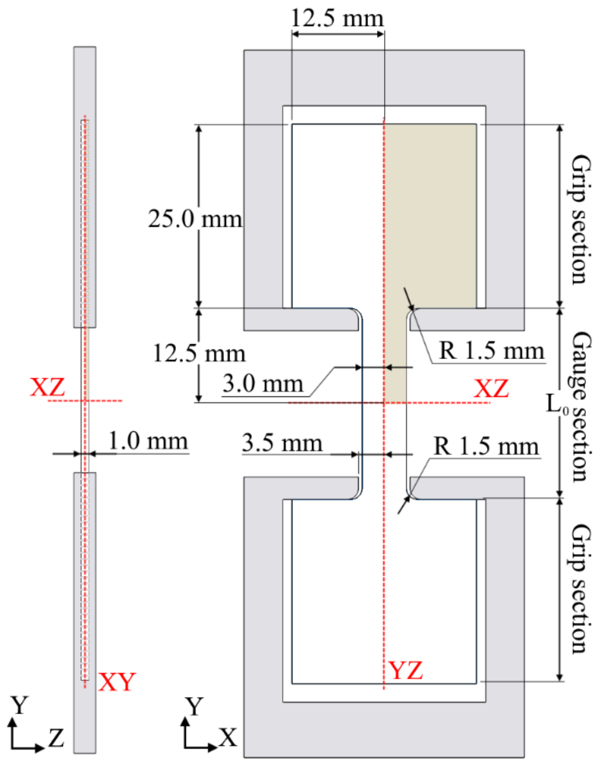

**Figure 1.** Geometry of a specimen.

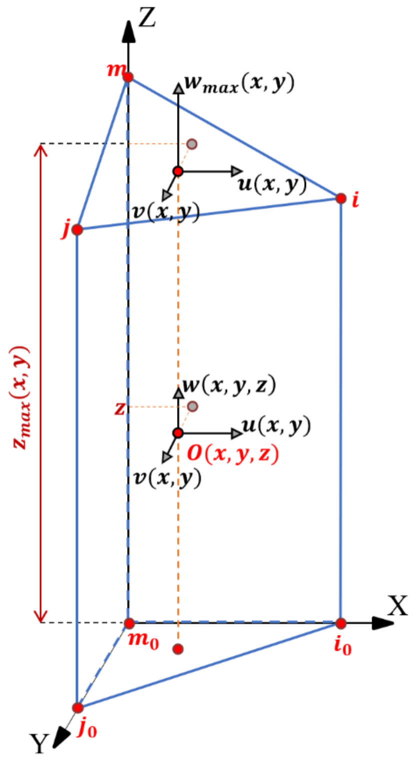

**Figure 2.** Prismatic element.

The software product was developed based on the finite element method, discretizing the specimen using specific prismatic elements as it is described above. To maintain the constant strain rate ($\dot{\varepsilon} = 0.01$), the grip was simulated as a rigid body moving with the increasing velocity ($v$) as prescribed in the standard for superplastic tensile testing [7]:

$$v = \frac{1}{2}\dot{\varepsilon}L_0\exp(\dot{\varepsilon}t). \tag{3}$$

In the developed software, the force of the specimen on the grip ($F$) and the length of the gauge section (L) can be monitored. The volume of the whole specimen ($V$) is considered to be constant. The output stress and strain values were calculated based on these parameters, according to the ASTM standard:

$$\varepsilon_{out} = \ln\left(\frac{L}{L_0}\right),\ \sigma_{out} = \frac{FL}{V}. \tag{4}$$

The force ($F$) was calculated using $\sigma_{yy}$ – components of stress tensor which correspond to the $Y$ axis and $S$ – the area of the specimen cross-section in the face $XY$, as follows:

$$F = \int_S \sigma_{yy}\, dS. \tag{5}$$

A power law approximation of the flow behavior, which is widely used for simulations of superplastic forming technologies [14], was used as the constitutive equation characterizing the mechanical properties of the material:

$$\sigma = K\varepsilon^n\dot{\varepsilon}^m, \tag{6}$$

where $\sigma$ – effective stress; $\varepsilon$ – effective strain; $n$ – is a material constant responsible for the strain hardening effect; $m$ – is a strain-rate sensitivity index, $K$ – material constant. If the strain hardening index ($n$) is zero, Equation (6) takes the form of classical Backofen equation [15] used for description of ideal superplastic behavior.

## 3. Results and discussion

### *3.1. The inhomogeneity of strain rate in the sample*

Figure 3 demonstrates the mesh of a specimen at different moments of the simulation. The mesh in the specimen is defined in such a way that there are at least 5 elements wide in the gauge section. Moreover, the remeshing is realized, so the elements do not stretch too hard by the end of the modeling. Due to the strain rate distribution, which is demonstrated on Figure 4, the local strain rate maximum is located near the grip section at the beginning of a test, while after some time it moves to the center of the specimen and remains there till the end of the simulation.

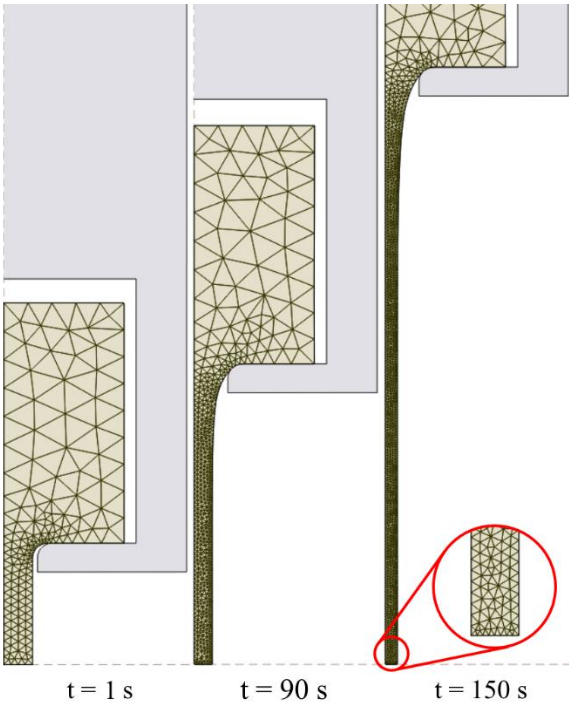

**Figure 3.** Mesh of the specimen.

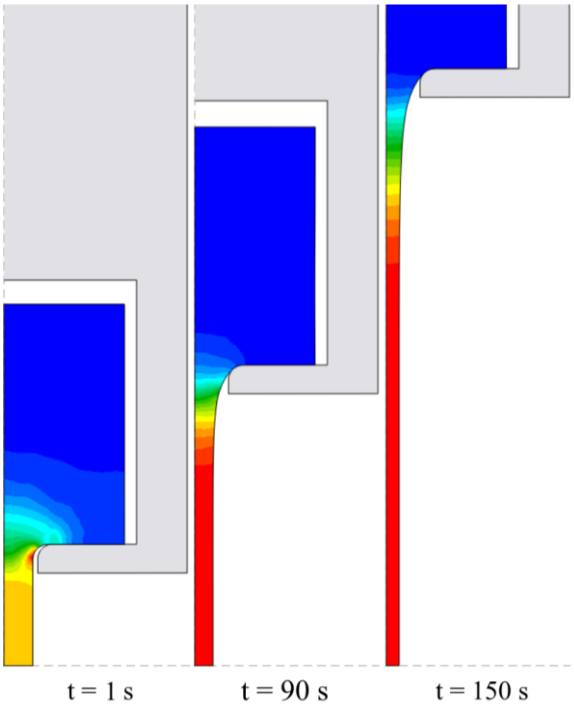

**Figure 4.** Strain rate distribution in the specimen.

### *3.2. The verification of the developed product*

Verification of the developed system was conducted in the commercial finite element modeling system for engineering calculations Abaqus/CAE, where the full three-dimensional modeling was carried out. As the Abaqus/CAE does not have built-in functionality that allows to fix the force of the specimen on the grip, a custom post-procedure Python script was written, which allows to get such a force after the simulation. The force ($F$) was calculated by Equation (5) as in the developed program. Figure 5 demonstrates the results of the simulation both in developed software product using special prismatic elements and in Abaqus/CAE using tetrahedral elements for discretizing the specimen. Material flow behaviour was described by the Backofen law from Equation (6) ($K = 300$, $n = 0.0$, $m = 0.5$ and constant strain rate $\dot{\varepsilon} = 0.01\ s^{-1}$).

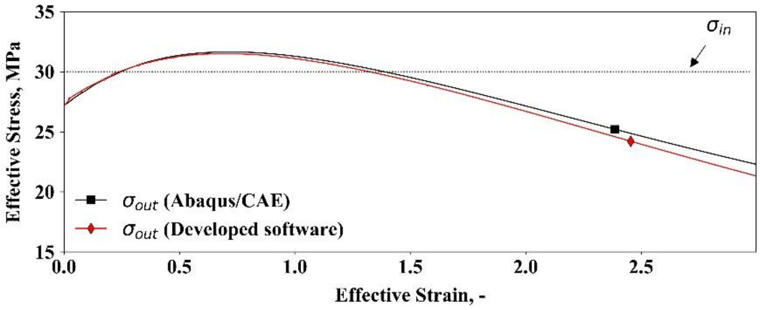

**Figure 5.** Modeling the slow shape change of a rigid body.

Input stress value is based on the input parameters and output stress-strain curves are calculated by equations (6) and (4), respectively. The Backofen model does not have any deformation effects, so the input stress-strain curve is illustrated as a straight horizontal dashed line. Average difference between output stress-strain curves from Abaqus\CAE and from developed software is less than 1.5%. The difference between input and output stress-strain curves both in developed software product and Abaqus/CAE is significant and happens due to the inhomogeneity of the distribution of strain rates in the volume of the specimen (Figure 4). Thus, the differences between input and output stresses, caused by the nature of the tensile test itself, are substantially greater than the differences between output stress-strain curves received from Abaqus\CAE and developed software.

### *3.3. The effect of specimen thickness and grip section*

The effects of such geometrical parameters as specimen thickness and grip section is studied using Backofen model from Equation (6) ($K = 300, n = 0.0, m = 0.5$ and constant strain rate $\dot{\varepsilon} = 0.01\ s^{-1}$). The effect of the specimen thickness was studied for the values from 1 mm to 6 mm. Figure 6 illustrates that the effect of the specimen thickness on the tensile test results is negligible. These results are consistent with the findings of the following research [11] based on full three-dimensional modeling in Abaqus/CAE. The effect of grip section is illustrated in Figure 7. Smaller grip section provides larger errors in the effective stress evaluation.

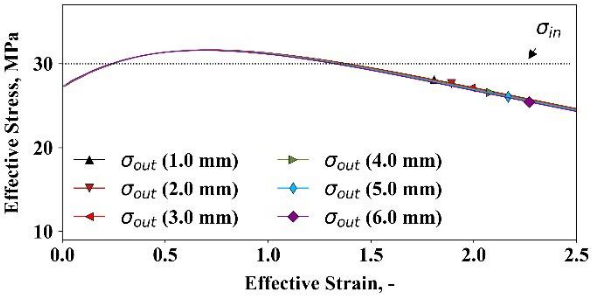

**Figure 6.** The effect of the thickness

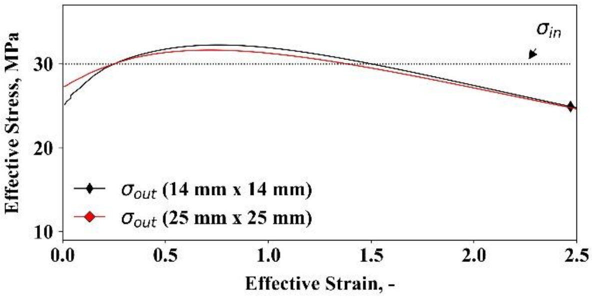

**Figure 7.** The effect of the grip section

### *3.4. The effect of the gauge length*

All simulations were made with constant strain rate $\dot{\varepsilon} = 0.01\ s^{-1}$. Figure 8 demonstrates the effect of the gauge length in tensile test results using the Backofen model from Equation (6) with parameters $K = 300, n = 0.0, m = 0.5$. Figure 9 demonstrates the effect of the gauge length obtained in the simulations utilising the model taking the strain hardening effect into account from Equation (6) with parameters $K = 300$, $m = 0.5$ and $n = 0.5$.

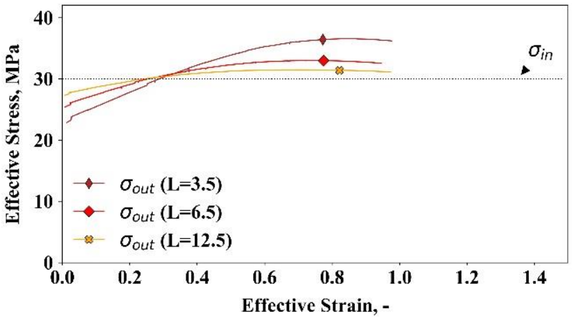

**Figure 8.** Backofen model.

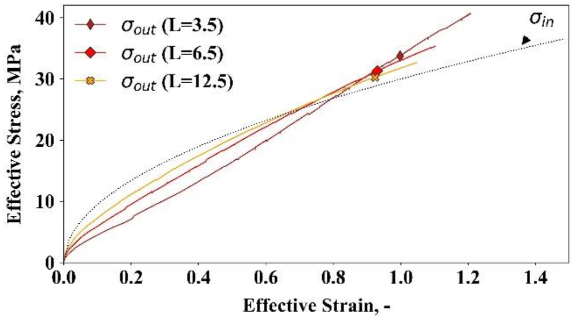

**Figure 9.** The model with strain hardening effect.

In these tests the value of effective strain was used as a fracture criterion so the simulations were performed until the maximum effective strain is less than 1.0. As it can be seen from Figure 8, where the model is based on the Backofen law, the fracture occurs at about the same strain value, but the difference between stress-strain output curves is significant. On the contrary, the stress-strain curves on

Figure 9, where the model is based on strain hardening effect, are closer to each other, but the specimen breaks at different strain values. The shorter gauge section provides larger tensile deformations. This result is consistent with the experimental observations provided in [12].

## 4. Summary

The computer software for simulation of superplastic tensile test was developed and implemented for investigation of flow inhomogeneity in the specimen volume and its effect on the results of testing. The size of the grip section and the thickness of the specimen affect tensile test result negligibly, while the effect of gauge length is much more noticeable and depends on the material properties. That happens because of the inhomogeneity of strain rate distribution in the specimen which leads to drawing-in of the material from the grid section to the gauge section. The amount of the draw-in material varies slightly for the specimens with different gauge length, but it has much greater effect in the specimens with short gauge section. This effect should be taken into account while the data obtained from tensile tests is analyzing. Otherwise, it may lead to bad accuracy of stress-strain output data. The developed software allows to evaluate those effects. An inverse analysis of the experimental results using the developed program can be applied to increase the accuracy of provided stress-strain relations.

## 5. References

[1] Langdon T G 1982 The mechanical properties of superplastic materials *Metall. Trans. A*

[2] Sherby O D and Wadsworth J 1989 Superplasticity-Recent advances and future directions *Prog. Mater. Sci.*

[3] Enikeev F U and Kruglov A A 1995 An analysis of the superplastic forming of a thin circular diaphragm *Int. J. Mech. Sci.*

[4] Giuliano G and Franchitti S 2007 On the evaluation of superplastic characteristics using the finite element method *Int. J. Mach. Tools Manuf.*

[5] Aksenov S A, Chumachenko E N, Kolesnikov A V. and Osipov S A 2014 Determination of optimal conditions for gas forming of aluminum Sheets *Procedia Engineering*

[6] Guglielmi P, Sorgente D, Lombardi A and Palumbo G 2020 A new experimental approach for modelling the constitutive behaviour of sheet metals at elevated temperature through interrupted bulge tests *Int. J. Mech. Sci.*

[7] E2448-11 2011 Standard Test Method for Determining the Superplastic Properties of Metallic Sheet Materials *ASTM B. Stand.*

[8] British Standard Institution 2007 Method for evaluation of tensile properties of metallic superplastic materials **2013**

[9] Khaleel M A, Johnson K I, Lavender C A, Smith M T and Hamilton C H 1996 Specimen geometry effect on the accuracy of constitutive relations in a superplastic 5083 aluminum alloy *Scr. Mater.*

[10] Sorgente D and Tricarico L 2007 Analysis of different specimen geometries for tensile tests in superplastic conditions for an aluminium alloy *Materials Science Forum*

[11] Aksenov S and Mikolaenko V 2020 The effect of material properties on the accuracy of superplastic tensile test *Metals (Basel).*

[12] Bate P S, Ridley N and Sotoudeh K 2008 Effect of gauge length in superplastic tensile tests *Mater. Sci. Technol.*

[13] Abu-Farha F, Nazzal M and Curtis R 2011 Optimum Specimen Geometry for Accurate Tensile Testing of Superplastic Metallic Materials *Exp. Mech.*

[14] Kruglov A A, Ganieva V R and Enikeev F U 2017 Determination of superplastic properties from the results of technological experiments *Adv. Eng. Softw.*

[15] Backofen, W.A., Turner, I.R. and Avery D H 1964 Superplasticity in an Al-Zn Alloy *ASM Trans. Quart.* **57** 980–990