INTRODUCTION

In several practical cases, the ultimate ductile fracture strain determined with tensile test is accepted as a material plasticity measure [1]. In this case, the plasticity has to be defined as an ability of a material to accommodate high permanent strains until fracture appears where this strain reaches certain value called ultimate fracture strain $\varepsilon_p$. The strain value until fracture depends not only on the material type, but also on other several factors, as: strain speed, strain history, material starting structure, temperature, specimen geometry, etc. It is impossible to account for all factors in a single mathematical description, due to a complexity of phenomena and an insufficient state of the art, mainly for phenomena present during a plastic strain. Several experiments [2-5] have demonstrated that the material fracture process strongly depends on the hydrostatic stress. This conclusion has been independently induced based on experiments [6-8].

Recently, several different fracture criteria, including the state of hydrostatic stress, have been developed [8-10]. However, the practical application of above criteria to forecast the fracture during the metal forming process has been feasible thanks the numerical computing methods, which enable to determine the material’s state of stress during the plastic forming process. Currently, the ductile fracture criteria are commonly used when simulating various plastic processing processes [10-13]. However, the practical application of the criteria requires the experimental determination of the ductile fracture strain $\varepsilon_p$ value for a given material. This strain is usually determined based on the tensile test, but the determination method indeed is not so obvious, and in several cases even doubtful.

In most cases, the tensile test is performed against circular or rectangular cross-section specimens. Considering that the ductile fracture strain $\varepsilon_p$ around the fracture zone equals the equivalent strain $\varepsilon_e$ in this zone, it can be calculated using the equation:

$$\varepsilon_p = \varepsilon_e = \frac{\sqrt{2}}{3} \sqrt{\varepsilon_1^2 + \varepsilon_2^2 + \varepsilon_3^2}$$

(1)

For circular cross-section specimen (Figure 1a), the strain components in direction 1 and 2 are calculated using the equation:

$$\varepsilon_1 = \varepsilon_2 = \varepsilon = \ln \frac{d_i}{d_o}$$

(2)
The strain component in direction 3 is calculated using the constant volume condition:

\[ \varepsilon_1 + \varepsilon_2 + \varepsilon_3 = 0 \Rightarrow \varepsilon_3 = -2\varepsilon_1 \]  
(3)

If (2) and (3) are substituted to (1) and transformed, the equation to determine the ductile fracture strain \( \varepsilon_p \) for circular cross-section specimen is achieved.

\[ \varepsilon_p = 2 \ln \left( \frac{d_1}{d_0} \right) \]  
(4)

If a tensile tested specimen is plane (Figure 1b), the strain components in directions 1 and 2 are different and may be calculated using the equation:

\[ \varepsilon_1 = \ln \left( \frac{b_1}{b_0} \right) \]  
(5)

\[ \varepsilon_2 = \ln \left( \frac{g_1}{g_0} \right) \]  
(6)

The strain component in direction 3 is calculated using the equation:

\[ \varepsilon_1 + \varepsilon_2 + \varepsilon_3 = 0 \Rightarrow \varepsilon_3 = -(\varepsilon_1 + \varepsilon_2) \]  
(7)

If (5), (6) and (7) are substituted to (1), the equation to determine the ductile fracture strain \( \varepsilon_p \) for rectangular cross-section specimen is achieved:

\[ \varepsilon_p = \frac{2}{\sqrt{3}} \ln \sqrt{\varepsilon_1^2 + \varepsilon_2^2 + \left[-(\varepsilon_1 + \varepsilon_2)\right]^2} \]  
(8)

The equation (8) has been derived provided that the specimen’s cross-section shape is not changed after the strain. Actually, the specimen’s cross-section shape after the tensile failure differs significantly from the starting shape (Figure 2a). After the tensile failure, the plane specimen’s cross-section has a shape of a saddle (Figure 2b), and it means that the ductile fracture strain value is not identical within the cross-section, but varies significantly. That’s why the calculation of the ductile fracture strain \( \varepsilon_p \) for plane specimen is not so obvious, as for the circular cross-section specimen.

The lack of reference data how to proceed in this case has been a basis to perform the experiments, in order to develop the method to specify the ductile fracture strain \( \varepsilon_p \) for rectangular cross-section specimens.

**EXPERIMENTAL WORK**

The static tensile test has been performed using UTS 100 tensile testing machine. The plane sheet metal specimens have been tested made of the following material: steel, copper, and aluminum 5 251. The mechanical characteristics and the strain hardening curve parameters for tested materials, achieved based on the tensile test, have been presented in Table 1. In order to determine material constants \( K \) and \( n \), the specimen elongation has been measured using the extensometer along the section \( l_0 = 80 \, \text{mm} \). Then the strain hardening curve \( \sigma_p = f(\varepsilon) \) has been plotted for the points below maximum tension force. The stress \( \sigma_p \) for individual strain hardening curve points has been calculated as the ratio of the force to the variable specimen cross-section, calculated based on the constant volume condition. The logarithmic strain for individual strain hardening curve points has been calculated from the equation \( \varepsilon = \ln(l/l_0) \), where: \( l_0 = 80 \, \text{mm} \), \( l \) – the length of section after specimen elongation. The strain hardening curve points calculated this way have been approximated with an equation \( \sigma_p = K\varepsilon^n \). The measuring bases to indicate the measuring zone have been marked on the specimen surface. L, P – Lateral, S – Middle (Figure 3). The zone width and specimen thickness have been measured in these locations before and after the specimen tensile failure. The geometrical values have been measured using the toolmaker’s microscope with an accuracy of 0,01 mm. The average specimen thickness \( g_l \) after the tensile failure within individual areas (L, P, and S, C) has been determined as follows:

1) the specimen thickness has been measured after the tensile failure in examined areas, with an interval of approx. 0,5 mm on specimen width,

2) thickness \( g_l \) has been calculated as an arithmetical mean of measured thickness values within individual areas.

The measured values of geometrical parameters in individual locations before and after the specimen tensile

<table>
<thead>
<tr>
<th>Material</th>
<th>Yield stress ( R_y ) MPa</th>
<th>Ultimate strength ( R_m ) MPa</th>
<th>Strain hardening coefficient</th>
<th>Strain hardening exponent</th>
</tr>
</thead>
<tbody>
<tr>
<td>steel</td>
<td>399</td>
<td>447</td>
<td>591</td>
<td>0,072</td>
</tr>
<tr>
<td>copper</td>
<td>97</td>
<td>217</td>
<td>389</td>
<td>0,262</td>
</tr>
<tr>
<td>aluminium</td>
<td>65</td>
<td>175</td>
<td>378</td>
<td>0,31</td>
</tr>
</tbody>
</table>

---

**Figure 1** Typical tensile specimens: a) a round specimen, b) a flat specimen

**Figure 2** Cross-sectional area in the neck at fracture: a) before fracture, b) after fracture
The analysis of the fracture strain value $e_f$ for individual materials demonstrates that, for all cases, the highest strain appears in the middle part of specimen, and the lowest in the lateral part. The experiments performed show that the more plastic material is (in this case - copper), the higher fracture strain value is achieved as the specimen width is increased. The mean strain value measured for an entire specimen is inaccurate and depends mainly on its geometry. The following question arises: where the fracture strain for the plane specimen should be measured?

### Table 2  
Specimen’s geometry and average strain values in analyzed regions

<table>
<thead>
<tr>
<th>Material</th>
<th>Designation of parameter</th>
<th>Measuring zones</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>L P</td>
</tr>
<tr>
<td><strong>Steel</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$q_0$</td>
<td>3.55</td>
</tr>
<tr>
<td></td>
<td>$b_1$</td>
<td>2.1</td>
</tr>
<tr>
<td></td>
<td>$g_1$</td>
<td>2.34</td>
</tr>
<tr>
<td></td>
<td>$b_1$</td>
<td>1.47</td>
</tr>
<tr>
<td></td>
<td>$e_1$</td>
<td>-0.357</td>
</tr>
<tr>
<td></td>
<td>$e_2$</td>
<td>-0.417</td>
</tr>
<tr>
<td></td>
<td>$e_f$</td>
<td>0.774</td>
</tr>
<tr>
<td><strong>Copper</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$q_0$</td>
<td>3.55</td>
</tr>
<tr>
<td></td>
<td>$b_1$</td>
<td>2.1</td>
</tr>
<tr>
<td></td>
<td>$g_1$</td>
<td>1.27</td>
</tr>
<tr>
<td></td>
<td>$b_1$</td>
<td>1.51</td>
</tr>
<tr>
<td></td>
<td>$e_1$</td>
<td>-0.32</td>
</tr>
<tr>
<td></td>
<td>$e_2$</td>
<td>-1.028</td>
</tr>
<tr>
<td></td>
<td>$e_f$</td>
<td>1.416</td>
</tr>
<tr>
<td><strong>Aluminum</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$q_0$</td>
<td>3.55</td>
</tr>
<tr>
<td></td>
<td>$b_1$</td>
<td>2.1</td>
</tr>
<tr>
<td></td>
<td>$g_1$</td>
<td>2.1</td>
</tr>
<tr>
<td></td>
<td>$b_1$</td>
<td>1.81</td>
</tr>
<tr>
<td></td>
<td>$e_1$</td>
<td>-0.148</td>
</tr>
<tr>
<td></td>
<td>$e_2$</td>
<td>-0.525</td>
</tr>
<tr>
<td></td>
<td>$e_f$</td>
<td>0.707</td>
</tr>
</tbody>
</table>

**FEM NUMERICAL SIMULATION**

To answer the question as referred to the above, the steel specimen tension process numerical simulation has been performed using MSC Marc Mentat software, which enables solving non-linear and contact problems. FEM simulation’s geometrical model has been created based on the experimental model. The purpose of the numerical simulation in this case is neither detailed analysis of stresses and strains nor determining their values. The purpose of the simulation is to indicate the area, where the state of stress on the tensioned specimen is the closest to uniaxial tension, within an entire strain range up to specimen tensile failure. Therefore the specimen tension process has been analyzed in the plane stress condition. The elastic-plastic material model with non-linear strain hardening has been adopted, described by the following equation [14]:

$$
\sigma = \begin{cases} 
E \cdot e & (\sigma \leq \sigma_0) \\
K \cdot e^* & (\sigma > \sigma_b)
\end{cases}
$$

(9)

The material parameters for elastic strain have been as follows: $E = 210000$ MPa, $\sigma_0 = 0.3$. The strain hardening parameters $K, n$ are presented in Table 1. In order to create FEM grid of deformable sheet metal, Class 4 Type 3 elements has been used – plane-stress quadrilateral [15]. The start point of necking has been determined based on Hill’s equation in form of [16]:

$$
e^* = \frac{n}{(1 + \rho)}
$$

(10)

where: $e^*$ - critical strain for the onset of local necking, $\rho$ - $e_1, e_1, n$ - strain hardening exponent.

The tension simulations have been performed for an entire specimen, placed in the measuring area of an extensometer (II) holding the gripping area of the specimen, right at the tensile testing machine grips. Such a purposeful placement of the extensometer (II) enabled the introduction of the movement boundary condition for the specimen modeled as in the experiment. This also enabled eliminating the machine structure susceptibility errors. The boundary condition has been also introduced for nodes placed at the ends of modeled specimen in the measuring area of an extensometer (II). The node movements towards the specimen axis have been forced in the boundary condition. The node movement perpendicularly towards the specimen has been disallowed.
Due to such an assumed boundary condition, the local necking appears exactly halfway the length of tensioned specimen.

The tension simulation has been performed until specimen tensile failure, and it corresponds to extensometer (II) displacement, which was 15.33 mm for the steel specimen. The tensile force curves have been prepared and compared (Figure 4) in order to validate the FEM simulation. The limit value of ductile fracture strain depends on the present state of stress. In the mechanical & mathematical modeling approach, non-dimensional stress triaxiality \( k = \sigma_{\text{eq}} / \sigma_{\text{e}} \), where \( \sigma_{\text{eq}} \) is a mean normal stress, \( \sigma_{\text{e}} \) is an equivalent stress, is the very important parameter, which unequivocally specifies the plane state of stress (Figure 5). If this factor is known, it is possible to determine the state of stress in any point of strained object, e.g.: if \( k = 0 \) – this is a simple shear (Figure 5.c), \( k = 0.66 \) – it is a biaxial regular tension (Figure 5.e), \( k = 0.33 \) – it is an uniaxial compression (Figure 5.b), etc.

In considered case we determine the strain for the tensile test, so that \( k \) factor value is 0.33. As seen in FEM calculations, the uniaxial state of stress is present in an initial tension phase and lasts until the neck is created, and then once \( R_m \) limit is exceeded, the states of stress in individual zones differ significantly (Figure 6).

The state of stress in the lateral zone \( L \) changes slightly in the biaxial tension direction, reaching \( k = 0.36 \) in its final phase. Whereas the state of stress in the middle zone \( S \) changes significantly in the simple shear direction, reaching \( k = 0.106 \) in its final phase.

The state of stress factor \( k \) distribution in the initial and final phase of the specimen tensile test has been presented on Figure 7. The \( k \) factor value is explicitly different in the middle and lateral part of the specimen under test.

**CONCLUSIONS**

1. The experiments performed show that the fracture strain in the tensile test for plane specimen
must be determined in L or P zone, as the state of stress in these zones is the closest to the uniaxial tension for all tensile test.
2. The calculation of the ductile fracture strain for an entire cross-section C is highly inaccurate and the error mostly depends on the specimen dimensions.
3. The presented method of the ductile fracture strain determination is simple and can be performed during the conventional tensile test, once the base line is marked on the specimen surface.

REFERENCES


Note: The responsible translator English language is G. Rébisz, Rzeszów, Poland.