

PAPER • OPEN ACCESS

Usage of true stress-strain curve for FE simulation and the influencing parameters

To cite this article: A Petřík and R Ároch 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **566** 012025

View the [article online](#) for updates and enhancements.

You may also like

- [Measuring flow curve and failure conditions for a MEMS-scale electrodeposited nickel alloy](#)
D T Read, L A Liew, R M White et al.
- [Porosity evolution in a creeping single crystal](#)
A Srivastava and A Needleman
- [Mechanical properties and strengthening mechanism of V-containing weathering steel](#)
Wang Bo, Zhang Hua, Zhang Bo et al.



ECS
The
Electrochemical
Society
Advancing solid state &
electrochemical science & technology

DISCOVER
how sustainability
intersects with
electrochemistry & solid
state science research

Usage of true stress-strain curve for FE simulation and the influencing parameters

A Petřík¹, and R Ároch¹

¹ Slovak University of Technology in Bratislava, Faculty of Civil Engineering, Department of Steel and Timber Structures, Slovakia

Abstract. Computer simulations require appropriate input data to match the relevant calculated results with those of experiments. The true stress-strain curve suitable for finite element analysis is describing the non-linear behaviour of the steel material with a proper pre-peak as well as necking stress-strain relationship. The pre-peak part of true stress-strain curve can be easily covered by an analytical method (i.e. logarithmic equations), but the necking part of the curve cannot be described by such a relationship between the normalized and the true stress-strain curves. Simple tensile test of steel specimen was performed to establish the true stress-strain curve in ANSYS software. Different sizes of finite elements were applied for the non-linear FE analysis to find an appropriate size function while creating the necking part of true stress-strain curve using an iterative method. Stress and strain from the computer analyses were compared with the results of the tensile test. The dependence of size of finite elements and/or element type was analysed to achieve the most precise results.

1. Introduction

Nowadays, increasingly complicated and complex simulations require appropriate input to create and verify computer models with experiments. Especially the steel plastic behaviour be considered by various methods and material characteristics. For simple simulations, the usage of bilinear stress-strain curve with or without hardening is sufficient to obtain ordinary usable results. However, the most precise results are obtained by considering the engineering stress-strain curve with its elastic and plastic parts. The engineering stress-strain curve can be easily gained from the tensile coupon test.

In ANSYS software, as in other programs using the finite element method, the stress-strain data should be input in a correct way, i.e. to use the true stress-strain characteristics of the steel. The conversion between engineering and true stress-strain curves can be simply made by the logarithmical equations. This procedure can be used in elastic region and partially in plastic region from the yield point to ultimate strength point, where the strain of test specimen is uniform within the gauge length. The problem can be reached in the necking region (from ultimate strength point to fracture point) of the stress-strain curve where the conversion of curves cannot be done by those equations and other methods must be used. One of the methods is an iterative method based on continuous determining the necking part of the true stress-strain curve until the fracture point of the curve.

2. Theoretical background

In general, the result of the uniaxial tensile test is a curve similar to the engineering stress-strain curve, where the stress and the elongation is measured in real time. However, the elongation needs to be recalculated into strain by the gauge length. This parameter depends on the type and size of cross-sectional area of the test specimen. Subsequently, the engineering stress-strain curve can be designed.



It is worth mentioning that stress value at each point of the engineering stress-strain curve is determined in the undeformed initial state and this is the main reason, why it is not correct to use the engineering stress-strain curve as a material property for computer simulations based on finite elements method.

Because of the fact that the shape of the test specimen in the finite element analysis can be deformed, it is necessary to transform the engineering stress-strain curve into a true stress-strain curve, in which the current deformed state of specimen is in correlation with the history of previously states. Without the conversion of the stress-strain curve from the engineering type to the true one, a substantial miscalculation can occur not only in the ultimate strength point of the curve, but in the whole plastic region of the curve.

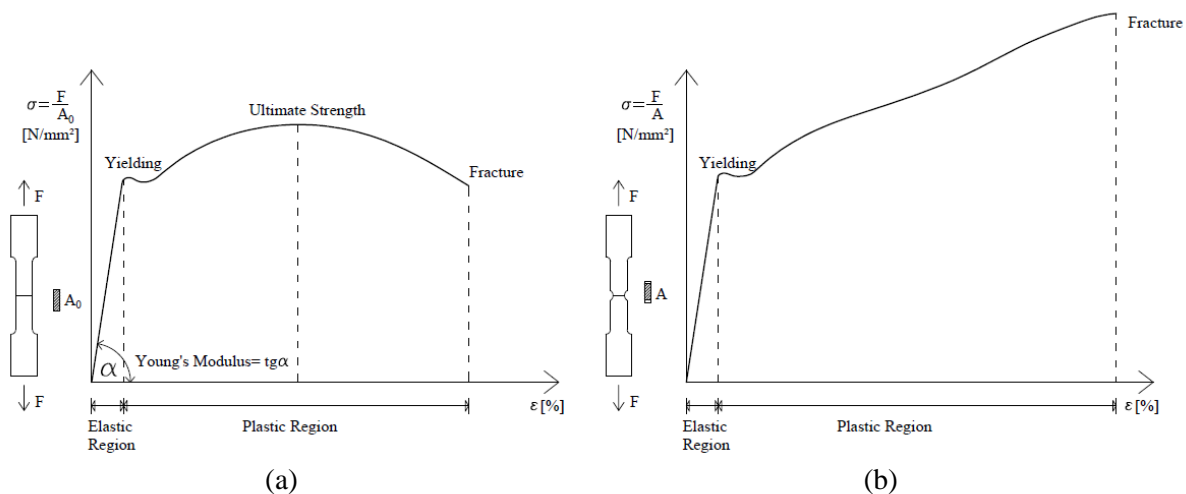


Figure 1. Stress-strain curve: engineering stress-strain curve (a), true stress-strain curve (b).

As already mentioned, engineering stress-strain curve can be converted to a true stress-strain curve using logarithmic equations. These can be applied in most cases of problems, because they cover a large region of the engineering stress-strain curves – from beginning of the test until the moment when the ultimate strength of the specimen is reached. Afterwards it is obligatory to use another method to gain the remaining part of the true stress-strain curve, where the necking of the test specimen is localized. The true strain is characterized by the division of the increment of the strain and the actual length (1). Because of the uniformity of the strain along the test specimen, the true stress can be calculated as a division of the axial force and the actual cross-sectional area of the test specimen (2)[1].

$$\varepsilon_t = \int_{L_0}^L \frac{1}{L} dl = \ln(1 + \varepsilon_e) \quad (1)$$

$$\sigma_t = \frac{P}{A} = \frac{PL}{A_0 L_0} = \sigma_e \frac{L}{L_0} = \sigma_e (\varepsilon_e + 1) \quad (2)$$

One of the possibilities is to use an iterative method in finite element software, e.g. ANSYS, where the test specimen can be modelled. The main idea of the process is to continuously determine the necking part of the true stress-strain curve until the fracture point of the curve. The iterative procedure is as follows: increase the displacement of the test specimen and exceed the strain of the last known point; afterwards, adjustment of the slope of the true stress-strain curve to obtain the appropriate stress value at the engineering stress-strain curve at the corresponding strain value. The adjustment of the slope should be repeated until the value of the stress with the appropriate precision has been reached. Then, the displacement should be increased until the strain at the fracture point is obtained. Finally, the true stress-strain curve is completed and can be used in finite element analysis [1].

3. Model

In order to generate the true stress-strain curve, a rectangular cross-sectional test specimen was prepared of steel X60 (according to API 5L). The geometry of the specimen is specified in figure 2. In the hatched areas, the model was constrained in z-direction and the left hatched area was constrained in x-direction. The displacement was defined on the right hatched area.

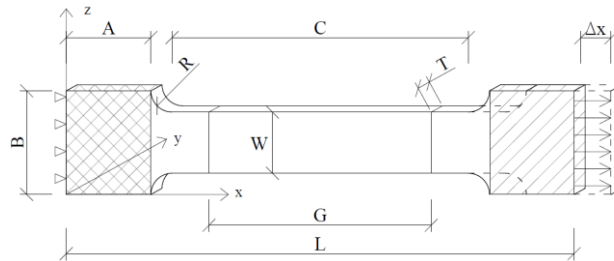


Figure 2. Geometry of the test specimen.

Table 1. Dimensions of the test specimen.

	A	B	C	L	G	W	R	T
	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]
TS	50.0	61.0	175.0	300.0	131.3	36.0	12.5	15.0

Subsequently, it was inserted in the tensile test machine in order to obtain the dependency curve of the change in elongation depending on the applied load. Using the gauge length [2], the elongation of the test specimen was recalculated to the strain and due to local inaccuracy, the shape of the curve was corrected.

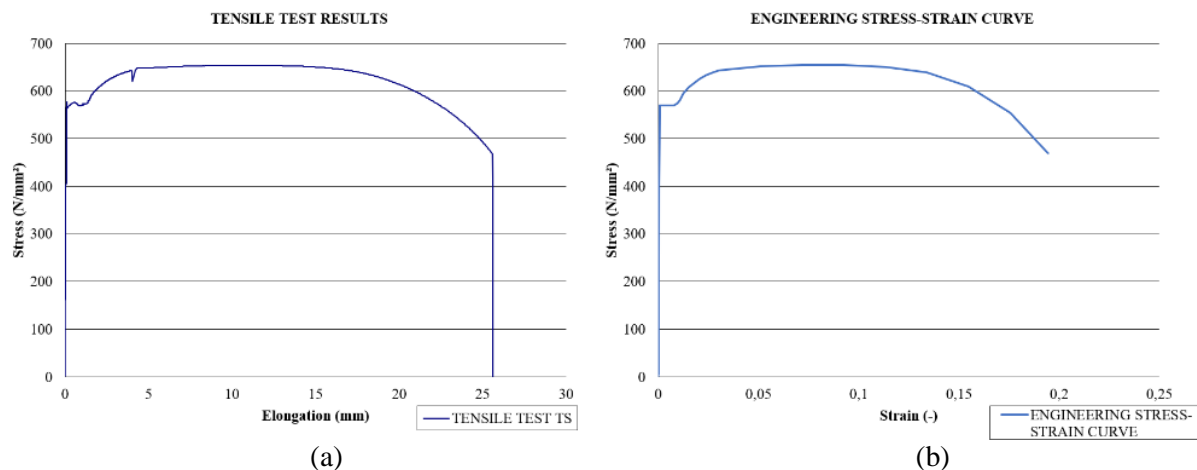


Figure 3. Original tensile test results (a), corrected and recalculated results into engineering stress-strain curve (b).

The work has also focused on the impact of the size of the finite elements and of the type of the hexahedral elements. Lower size of the elements can increase the accuracy but can significantly raise the duration of the analysis. Also, the element order is an important parameter that can affect the output data of the simulation. Linear elements (without the mid-side nodes) usually provide accurate solution in reasonable time and it is recommended to use this type of elements in nonlinear problems. Usage of the quadratic elements (with the mid-side nodes) considerably extended the duration of the simulations

and in comparison with the equivalent mesh consisting of linear elements, they provide less accurate solution in more time.

The test specimen was modelled in ANSYS software in accordance with the abovementioned geometry as a 3D model. Four models of the test specimen were prepared: three with the linear elements order and one with the quadratic elements order. Also, the mesh size varied in the analysis – in the first group of the elements, three different size of finite elements were used – 1 millimetre, 1,5 millimetre and 2 millimetres. For comparison 1.5-millimetre elements were used for the model with quadratic elements order.

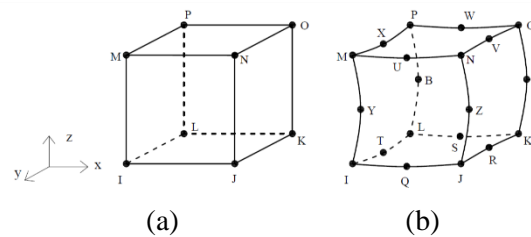


Figure 4. Linear element (a), quadratic element (b).

Because of the requirement to achieve not only the ultimate strength point, but also the fracture point, it was needed to run the analysis as displacement controlled. Additionally, the Newton-Raphson algorithm does not accept a decreasing slope of the stress-strain curve.

Firstly, the usage of the true stress-strain curve as the correct material input was needed to be proved. The engineering stress-strain curve prepared from the results of the tensile test was inserted as the material property in the first analysis, while the true stress-strain curve of the steel recalculated from the engineering stress-strain curve by the logarithmic equations was used in the second analysis. In both simulations, the intention was to continuously increase the value of the displacement until the ultimate strength point is reached and to monitor the development of the plastic strain.

From the simulation results it can be stated, that by using the true stress-strain parameters it is possible to achieve the ultimate strength point with high precision (figure 7). On the other hand, using the engineering stress-strain parameters, the ultimate strength point cannot be reached. Furthermore, the strain is not uniform along the gauge length, the necking of the test specimen starts to appear (figure 5), and the value of the plastic strain is more than 6 times larger than in the simulation with the true stress-strain parameters (figure 7).

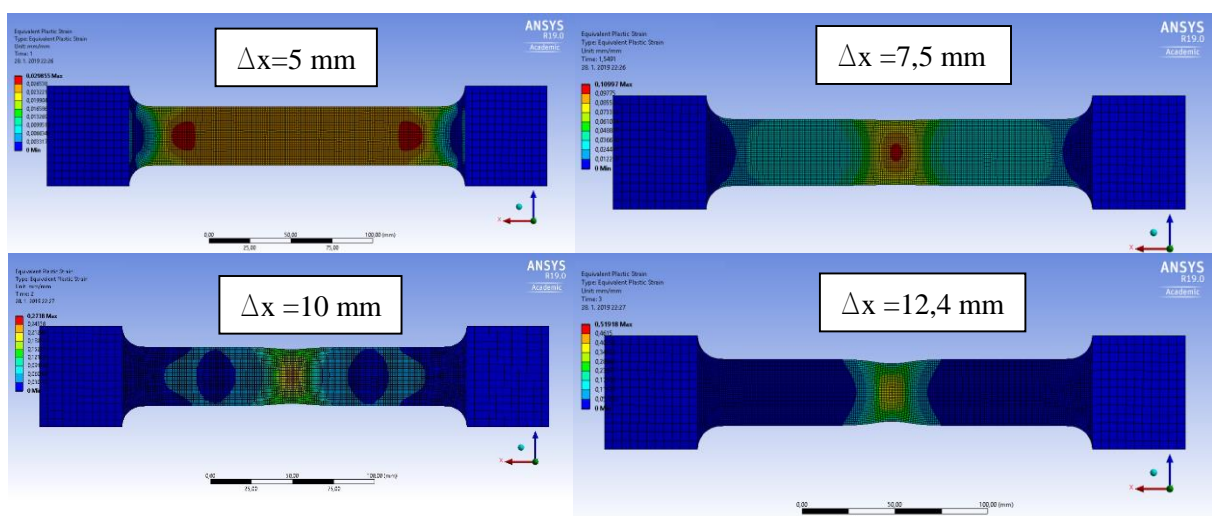


Figure 5. Time development of the plastic strain using engineering stress-strain material properties.

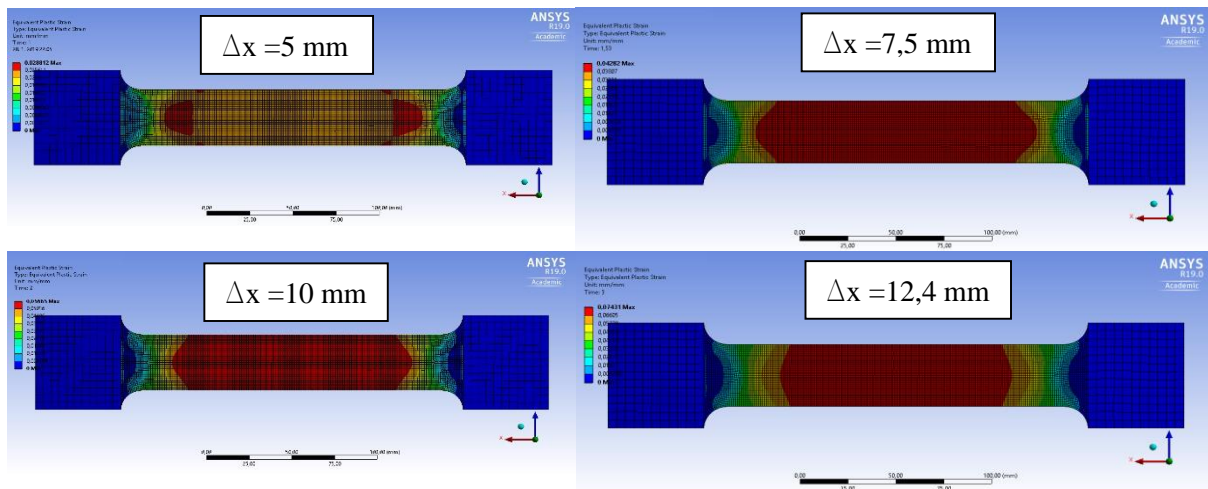


Figure 6. Time development of the plastic strain using true stress-strain material properties.

Secondly, the influence of the size function of the elements on the generation of the necking region of the true stress-strain curve was observed. It is important to say that for every different size of the element new true stress-strain curve needs to be calculated. The increasing number of nodes extends the time of solving the model; it can be reached by two processes – decreasing the size of the elements or the usage of higher order elements with the mid-side nodes. Four different test models were analysed with the abovementioned mesh variations and the variation of the elements order. The true stress-strain curves developed by the analysis are similar, but it can be claimed that with the decreasing element size, the plastic strain and the true stress raise. Also, the slope of the curves increased with the finer elements mesh (figure 8).

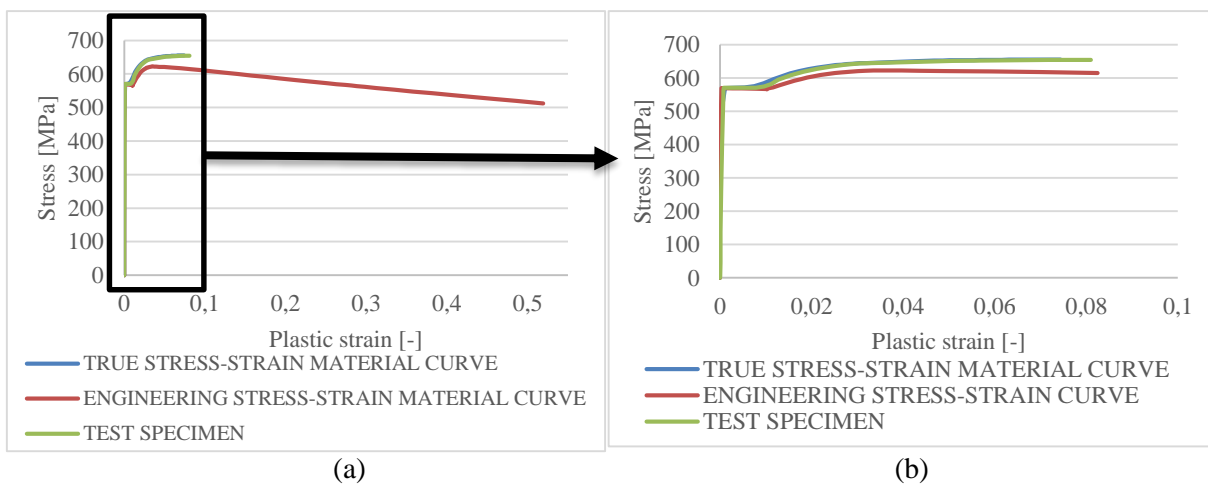


Figure 7. Comparison of material inputs in solution (a), detail of comparison (b).

Despite the fact that the quadratic elements order is not recommended for the nonlinear simulations, they were used in the last simulation. With the highest number of the nodes, the duration of this analysis was the longest among the whole research and the development of the true stress-strain curve was the most time consuming. The value of the true stress and the plastic strain at the fracture point is higher with the quadratic elements order than with the linear elements order with the same size of the elements. However, the usage of the higher density of linear elements makes the solution faster and the curves are

almost similar, there is no reason why not to follow the recommendation to use rather smaller linear elements order, than larger quadratic elements order.

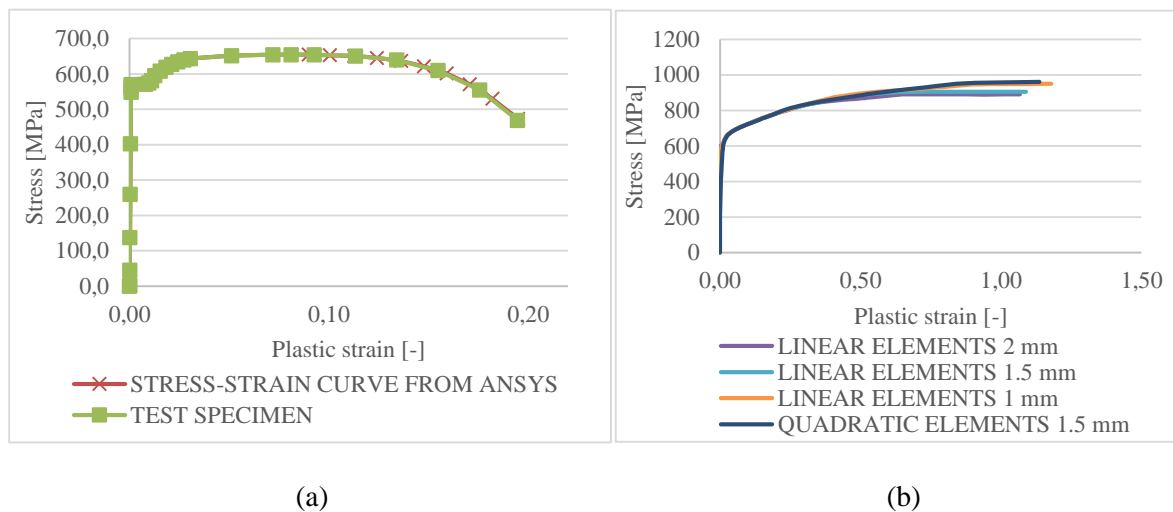


Figure 8. Comparison of stress-strain curves obtained by the tensile test and the iterative method in ANSYS software (a), comparison of true stress-strain curves by the size of the elements (b).

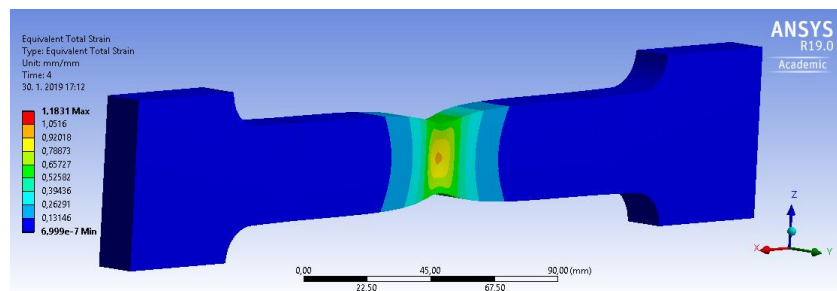


Figure 9. Strain of the specimen after reaching the fracture point.

4. Conclusion

The main aim of this research was to prove the usage of the true stress-strain curve as the correct material property for nonlinear simulations of steel components. After the analyses and their comparison, usage of the engineering stress-strain curve leads to strongly incorrect results. Also, the Newton-Raphson method for solution cannot cooperate with the decreasing slope of the stress-strain so the engineering stress-strain curve needs to be converted into the true stress-strain curve by analytic (logarithmic equations) and iterative methods (FE software).

Size function of the elements and elements order are the principle parameters that influence the precision of the solution. Increasing the mesh density of the elements also increases the duration of the solution, as well as the use of quadratic elements instead of the linear ones. For practical reasons, for every size function of the elements, a unique true stress-strain curve exists, and it needs to be generated before every simulation.

5. References

- [1] Gannon L 2011 *Mesh Dependence of True Stress-Strain Curves in Finite Element Analysis of Steel Structures* (DRDC Atlantic TM 2001-283; Defence R&D Canada – Atlantic) pp 1-20
- [2] International Organization for Standardization 2016 *Metallic materials – Tensile testing – Part 1: Method of test at room temperature* (ISO Standard No. 6892-1) retrieved from <https://www.iso.org/standard/61856.html>

- [3] Ehlers S 2010 Strain and stress relation until fracture for finite element simulations of a thin circular plate *Thin-Walled Structures* **48** pp 1-10

Acknowledgements

This paper has been supported by the Scientific Grant Agency of the Ministry of Education, science, research and sport of the Slovak Republic and the Slovak Academy of Sciences – grant VEGA 1/0773/18.