Influence of DC05 deep drawing steel real material properties on numerical simulation of incremental forming process

D Nasulea* and G Oancea1

1Manufacturing Engineering Department, Transilvania University of Braşov, Eroilor nr. 29, 500036 Brasov, Romania

*E-mail: nasulea.marius.daniel@unitbv.ro

Abstract. The finite element method is one of the most useful virtual tools in industrial engineering, which allows quick and cheap predictions and verifications. As it is expected, the FEM simulations are also integrated for incremental forming processes to anticipate different aspects as forming forces, the material stress and strain, dimensional accuracy of parts, or simply to check if the desired part can be manufactured without material failure, before to start the effective manufacturing. The FEM predictions validity is influenced by many aspects of which the most significant are the sheet metal blank material properties. Each sheet blank material has its own mechanical properties, according with material standards, considered to be the theoretical properties. In most of the situations, those theoretical material properties are not matched with the real material properties. In this paper, it is presented the experimental method of determination of the real material properties for DC05 deep drawing steel, based on the tensile strength of the sheet specimens, and the true stress-strain curve which, in fact, represents the real material characteristic curve. A comparison is made between the FEM results, obtained using both, the theoretical and the determined real material properties, and the experimental trials for parts with frustum of a cone shapes.

1. Introduction

In modern industries, the efficiency and minimizing the manufacturing costs, keeping and even improving the products quality, are the successful keys for industrial companies or research organisations. For this purpose, one of the most useful tool is the finite elements method (FEM), being very cheap and easy to be implemented in products lifecycle development or as a preliminary check in the research stages, avoiding unwanted costs. FEM is a numerical simulation method that supposes, in the first stage, the product shape discretization, because, for complex shapes it is difficult to be analysed different aspects of the entire part. For a simplified calculation, the FEM method allows to divide the products into multiple simple sub-divisions, usually called finite elements, which are then individually analysed. The final result is a sum of the sub-divisions results. In the last years the FEM simulation methods are more and more performant and has gain a great confidence and good correlations of the provided results with the real word [1]. Taking into consideration FEM advantages, the researchers implemented this method also for incremental forming process simulations. There were used the most usual dedicated software such as ABAQUS, LS-DYNA or ANSYS, to analyse different process aspects as stress and strain, forming forces, thickness reduction, material behaviour, process parameters influence and tool path corrections, or even only as a preliminary check of the forming conditions or for material fracture identification [1-4].
However, the results provided by FEM simulations are not always in agreement with the reality. In order to ensure the confidence of the provided results, the right input parameters should be introduced in FEM model preparation stage. This paper presents how the material properties of DC05 deep drawing steel, used in a FEM simulation, influence the provided results, being the key element to obtain a great rate of confidence on real predictions on numerical simulation of incremental deformed parts. Two parts dimensional configurations will be simulated and validated by experiments.

2. Research description
One of the most important input parameters in FEM simulations, are the part material properties. The main topic of this research is to study the influence of the material properties in numerical simulation on single point incremental forming process (SPIF) for frustum of a cone shapes. The material used for parts simulation and manufacturing are sheet metal blanks of deep drawing steel DC05 material, with 1mm thickness. For SPIF process implementation, a technological system is required which suppose using a CNC milling machine, a blank fixing device and a forming tool. The setup configuration for the current research is composed from a CNC Victor Vcentre-55 milling machine, a fixing device which is able to clamp cone parts with the upper base diameter up to 100mm [5], and a head rounded tool, all these are presented in figure 1 [6]. A 12mm tool diameter were chosen because it is one of the most used values in the literature [7, 8]. In figure 1 there are presented also the frustum of a cone part dimensional parameters.

![Technological system](image)

**Figure 1.** Technological system: (a) machine tool and blank fixing device, (b) forming tool dimensions, (c) parts parameters [6].

The fixing device presented in figure 1, was designed for clamping square sheet metal blanks. Because the FEM simulation for SPIF processes requires a considerable CPU time [7], necessary to analyse the forming process for one part, it was decided that, for FEM implementation, to be used round sheet metal blank. In this way the number of finite elements is decreased and the unnecessary runs for the blank corners are avoided, without influence upon the final provided results. The goal of the FEM simulations from the current research is to check if a certain part dimensional configuration can be deform without material fracture, the results being preceded by experimental validation.
3. Preliminary simulations

Two parts with different dimensional configurations were chosen for numerical simulation and experimental validation of the FEM results. The configurations, named C1 and C2, have the following dimensional parameters:

- **C1**: upper base diameter, D=85mm, part height, H=25mm and wall draw angle, α=55º;
- **C2**: upper base diameter, D=95mm, part height, H=50mm and wall draw angle, α=60º.

The first configuration is a small one and it is deformed using an incremental step depth ∆z=1.5mm. The second one is bigger and with more difficult deformation conditions, being the reason of choosing an incremental step depth ∆z=0.5mm. Both are simulated using for FEM model preparation the ANSYS parametric design language (APDL), and for process analyse, LS-Dyna explicit solver [6]. The material properties introduced in FEM model preparation stage were according with DC05 standard SR EN 10130:2007, and there are presented in table 1 [6].

| DC05 deep drawing steel | Density [Kg/m³] | Young modulus [MPa] | Poisson’s ratio | Yield strength [MPa] | Tensile strength [MPa] |
|-------------------------|----------------|---------------------|----------------|---------------------|----------------------|
|                         | 7850           | 210000              | 0.3            | 180                 | 270...330            |

Following all necessary steps for FEM implementation [1, 6], both part configurations were simulated and the post-processed results are presented in figure 2.

**Figure 2.** Simulation results for SPIF processing parts, FEM model and axial force plots: (a) C1, (b) C2.
According to figure 2, at a close look upon the FEM deformed models, it can be seen a series of shape undulation on the parts walls that can note the material fracture in those areas. This aspect is strongly visible also on both axial force plots, where, marked with blue colour, it can be seen that suddenly decrease of the forces on axial direction, even decreasing down to zero newton, which clearly suggests the deformation process failure because of material fracture. Therefore, it was decided to improve the FEM predictions by replacing the standard material properties with the real ones which are experimental determined for the specific used batch of DC05 material.

4. Experimental determination of the real mechanical properties for DC05 material
The real material properties are determined by applying the uniaxial tensile strength on 15 sheet metal specimens of which shape and dimensions are presented in figure 3.

![Figure 3. Parts manufactured without material][6]

Because the DC05 deep drawing steel sheet is a laminated blank, the material properties could be different depending on the directions in which they are measured. To avoid errors due to material anisotropy, it was decided to test five specimens cropped/machined along three direction, as follow (figure 4a) [6, 9].
- S1.1, S1.2,....., S1.5 - five specimen cropped along the material rolling direction;
- S2.1, S2.2,....., S2.5 - five specimen cropped on a 90° related to material rolling direction;
- S3.1, S3.2,....., S3.5 - five specimens cropped on a 45° related to material rolling direction.

![Figure 4. Tensile test setup: (a) 15 specimens, (b) tensile test machine, (c) tested specimens][6]
The specimens were manufactured using abrasive water jet cutting process on MAXIEM 1530 machine. The abrasive water jet technology was chosen because, compared to laser or plasma cutting, it does not affect material properties by heating the specimens, avoiding in this way obtaining erroneous results. Further, the specimens were tested using the uniaxial tensile test machine LLOYD LS100 Plus, together with the machine data management software NEXYGEN Plus (figure 4b) [6]. In figure 4c are presented all 15 specimens on which was applied the uniaxial tensile strength.

The result of the uniaxial tensile test is a set of values representing stress-strain values \((\sigma, \varepsilon)\), based on which the material characteristics curve were represented for each tested specimen. Those curves represent the engineering stress – engineering strain graphics and they are presented in figure 5. It was selected for a further processing, the specimen S1.3, being the closest to the average values.

![Figure 5](image5.png)

**Figure 5.** Engineering stress – engineering strain graphics for all 15 specimens [6].

The purpose of this further data processing of the engineering stress-strain values is to obtain the real material properties and, certainly, the true stress – true strain characteristic curve \((\sigma_{\text{real}}, \varepsilon_{\text{real}})\). The stress and strain values resulted direct from the machine tensile test are generated taking into consideration constant values for specimen gauge section area, \(S_0\), and for gauge length \(L_0\), and they are equal with the initial specimen dimensions. In fact, those two values are continuously changing during the tensile test, the real values for this two parameters being notated \(S_i\) and \(L_i\), the instantaneous values being different for each time increment. The difference between the initial specimen dimensions and the instantaneous values are presented in figure 6.

![Figure 6](image6.png)

**Figure 6.** Difference between \(S_0 / L_0\) and \(S_i / L_i\) during tensile test [6].
Taking into consideration those facts, it is necessary to follow the below presented steps, to obtain the true stress – true strain characteristic curve and also the real material properties [6]:

- From the law of constant volumes in material forming, results the following equation [6, 10]:
  \[ V_0 = V_l \Rightarrow S_0L_0 = S_lL_l \] (1)

- The instantaneous value for \( S_i \) is [6]
  \[ S_i = \frac{S_0}{1 + \frac{\Delta l}{L_0}} \] (2)

- The instantaneous value for \( L_i \) is [6, 11]:
  \[ L_i = L_0 + \Delta l \] (3)

where \( \Delta l \) is the specimen elongation,

- From the engineering stress relation, results the true stress relation is as follow [11]:
  \[ \sigma = \frac{F}{S_0} \Rightarrow \sigma_{real} = \frac{F}{S_i} \] (4)

where \( F \) is the traction force, also given by the tensile test machine;

- The real strain can be calculated using relation (5) [6]:
  \[ \varepsilon_{real} = \frac{\Delta l}{L_0} \] (5)

- The true stress relation becomes [6]:
  \[ \sigma_{real} = \frac{F}{S_0} \left(1 + \varepsilon_{real}\right) \] (6)

Using the mathematical relations above presented (1-6), the initial set of values given by the tensile test machine was processed in an Excel worksheet, and the real values for true stress – true strain were obtained. Based on those processed values, the real characteristic curve was represented for specimen S1.3.

The Young modulus was also calculated using the relation (7), where \( \alpha \) is the angle between the strain axis and the proportionality limit line from the real characteristic curve [6].

\[ E = \tan(\alpha). \] (7)

Based on those processed data and on true stress – true strain characteristic curve \( (\sigma_{real}, \varepsilon_{real}) \), it was identified also the values for the real tensile strength and the real yield strength. In table 2 there are presented for comparison, the mechanical properties according to material standard and according to the real experimental determination.

| Table 2. Material properties comparison – standard vs real experimental determination [6]. |
|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|
| According to:   | Density [Kg/m³]| Young modulus [MPa] | Poisson’s ratio | Yield strength [MPa] | Tensile strength [MPa] |
| SR EN 10130:2007: | 7850            | 210000           | 0.3            | 180              | 270...330       |
| Real processed data: | 7850            | 87952            | 0.3            | 190.59           | 388.67          |
Those new data are the true values of the material properties and they are specific for the used batch of DC05 material. Those are used further as input data for FEM numerical simulations, in order to obtain predictions as accurate as possible.

5. Second run of FEM simulations
Using those experimental determined values for DC05 material, the FEM simulations for C1 and C2 part configuration was solved once again. In addition to these real properties inputs, ANSYS APDL allows to specify also as input data, the real characteristics curve. The real characteristic curve is described in ANSYS as sets of points true stress – true strain ($\sigma_{\text{real}}$, $\varepsilon_{\text{real}}$), and this implementation can conduct to even more precise results. Thus, the simulations was repeated, obtaining much better results in comparison with the first trial, results which are presented in figure 7.

![Figure 7](image)

Figure 7. FEM simulation results using as input, the real material properties: (a) C1, (b) C2 [6].

For the new simulations, as can be seen in figure 7, the FEM deformed models has no shape undulation on its walls, and also on graphical representation of the axial loads, there are no sudden tendencies of force decreasing down to zero newton. Thus, it can be concluded that, according to FEM simulations using the real material properties, both parts can be manufacture without material fracture.

6. Experimental validation
In order to ensure that the FEM simulation provide confidence results, both parts configuration were manufactured using the same conditions implemented in the virtual analysis. For both, simulations and manufacturing, the used process parameters was a tool spindle speed of 200rpm, and a feed rate of 1500mm/min, following a spiral forming toolpath movement. In contrast with the first FEM results, both parts were deformed in good conditions, without material fracture, and they are presented in figure 8. Thus, there are no agreements between the manufacturing process and the FEM simulation results provided using, as mechanical properties, the theoretical ones from the material standard SR EN 10130:2007.
In comparison with the first simulations, for the second FEM trials, it can be said that the provided results are in a good agreement with the forming process. It can be noticed that, for real predictions, the right input data should be introduced in the FEM model preparation stage, with increased attention especially when refer on material properties. Therefore, because the material mechanical properties used as input data, has one of the major influence upon the numerical simulation results, defining the material behaviour during forming process, it is recommended to be determined the real material properties by experimental uniaxial tensile tests.

7. Conclusions
FEM simulation is a very advantageous engineering tool which can be used in product development and also in research. It is able to provide confidence results only if the proper input data are available.

The paper presented a research on how the material properties, as input data, can influence the FEM results and how the results are in connection with the practical experiments. It is presented the case of single point incremental forming, implemented for two frustum of a cone parts manufactured from DC05 deep drawing steel with 1mm thickness. These two parts were simulated, in a first stage, using the mechanical properties given by the material standard SR EN 10130:2007. Those results were erroneous because they were not in good agreements with the reality. It was also presented the method of real material properties determinations, based on uniaxial tensile tests and a series of relations available in the literature. Those real material properties, used in FEM simulation, conducted to improved results that match with the real forming process. It can be said that, if confidence results from FEM simulation are expected, the right input data should be provided, one of the most significant influence belonging to material properties. The material properties given by standards or suppliers are not always so accurate. The properties could be different from batch to batch and can be influenced by the date of material obtaining, storage conditions and other aspects.

References
[1] Nasulea D and Oancea G 2018 Stroj. Vestn-J. Mech. E 64 643
[2] Blaga A and Oleksik V 2013 Adv. Mater. Sci. Eng. 2013 2
[3] Gatea S, Ou H and McCartney G 2016 Int. J. Adv. Manuf. Tech. 87 479
[4] Fu Z, Mo J, Han F and Gong P 2013 Int. J. Adv. Manuf. Tech. 64 1239
[5] Nasulea D and Oancea G 2018 MATEC Web Conf. 178 02004
[6] Nasulea D 2019 Research Regarding the Incremental Forming of DC05 Steel Sheet Parts (PhD thesis,Transilvania University of Brasov, Romania)
[7] Nimbalkar D H and Nandedkar V M 2013 Int. J. Eng. Res. Appl. 3 39
[8] Behera A K, De Sousa R A, Ingarao G and Oleksik V 2017 J. Manuf. Process. 27 37
[9] Zein H and Al-Mufadi 2018 Minia J. Eng. Technol. 37 152
[10] Bologa O 2014 Cold plastic deformation processing (“Lucian Blaga” University Press, Sibiu, Romania)
[11] Oleksik V S 2005 Theoretical and Experimental Research on Incremental Forming of Indigenous Metal Sheets (PhD thesis, Lucian Blaga University of Sibiu, Romania)