Journal of Trends in the Development of Machinery and Associated Technology
Vol. 20, No. 1, 2016, ISSN 2303-4009 (online), p.p. 17-20

NUMERICAL SIMULATION OF THE FIRST OPERATION OF A DEEP DRAWING PROCESS

Sabahudin Ekinović, Edin Begović, Ibrahim Plančić
University of Zenica
Faculty of Mechanical Engineering
Fakultetska 1., 72000 Zenica
Bosnia and Herzegovina

Adnan Softić
MAN+HUMMEL
Bukva, 74260 Tešanj
Bosnia and Herzegovina

ABSTRACT
In this paper a numerical simulation of the first operation in the process of deep drawing a circular steel blank conducted in software Abaqus 6.13, which is based on FEM, is presented. Wherever possible simplifications of the process are made in order to obtain a quick and economical simulation. An accurate simulation of this kind of forming process can be very useful in predicting stresses and strains and the optimisation of the various process parameters, bypassing the necessity of very often complex and time-consuming experimental procedure.
Keywords: Numerical simulation, deep drawing

1. INTRODUCTION
The simulation in processes aims to manufacture products economically. Therefore, the application of process simulation must be always more economical than the application of the real process.[2] A simulation of the first operation of a deep drawing process of a circular blank with a 200 mm diameter and thickness of 1 mm, made of deep drawing class steel DC04 (1.0338), in software Abaqus 6.13, which is based on the finite element method, was attempted. The height of drawing in the first operation is 25 mm and the inner diameter of the deep-drawn part after first operation is 120 mm. Following the recommendations of the software manufacturer some simplifications of the process were made in order to avoid unnecessary computational time. Process was considered as quasi-static so the inertial forces can be neglected and the simulation was conducted in Abaqus Standard. Material of the part is considered to be isotropic and the deformation process was observed as planar. Since the part is axisymmetric analysis of stresses and strains was considered only on one half of the part. Generally, every simulation in the mentioned software consists of three main phases:

1. Pre-processing: defining the model for the analysis, adding material characteristics, defining boundary conditions, defining loads,...,
2. Analysis or processing: conducting the simulation,
3. Post-processing: analysis of simulation results and visual interpretation of results.

2. PRE-PROCESSING
As a first part of the process all necessary elements for the process of deep drawing were modelled in the software itself. Blank was modelled as a deformable 2D shell part and due to the fact that all the rest parts, die, punch and holder are made of much stronger materials they are approximated as rigid bodies for the analysis.

One of the most important parts of the process is defining the material properties of the blank. In our case, for a DC04 (1.0338) steel, Young's modulus was defined as 210 GPa and Poisson coefficient as 0,3. Elasticity characteristics were considered from diagram 1., since for the simulation, relation between true strains and true stresses is necessary.
An assembly of the parts was made, considering all the necessary constraints of movement and after that steps of the simulation were created. Two steps for the simulation are considered. In the first step holder force, which is later defined as a load of 500 kN, is applied and in the second step punch is moved to its final position, leaving the blank formed in its final shape. Additionally, for each step, frequency of recording the resultant stresses, strains displacements and forces is determined for a whole assembly and also for discrete reference points defined on the parts. Later on, boundary conditions are defined for the purpose of limiting certain displacements and rotations of parts during each step.

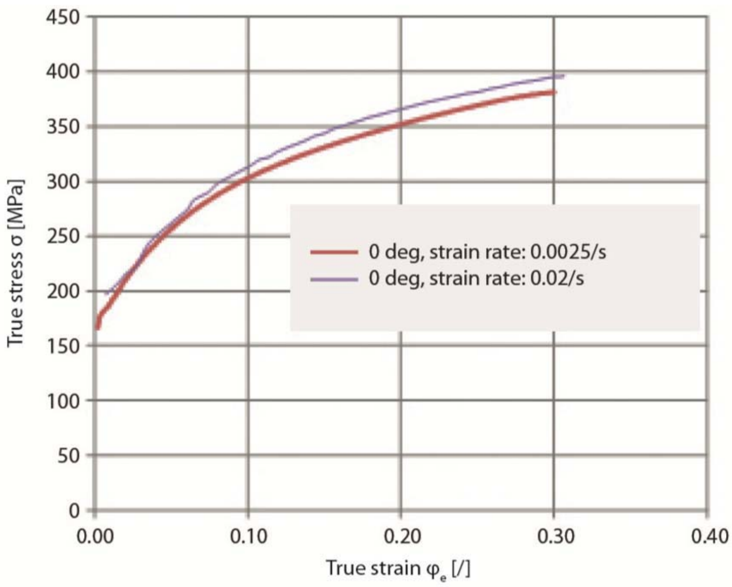

Diagram 1. True stress and true strain for DC04 steel [3]

Contact interactions of the parts are defined together with the assumption of the occurrence of friction through defining friction coefficient as 0,1 for each contact area.

Finally, at the end of the first step, mesh is created. Considering the type of the process and the fact that a considerable bending of the elements is expected CPE4R elements are used for the mesh of the blank. These are reduced integration elements used for plane strain. 100 elements are defined on the horizontal edge, with four elements through the thickness of the blank. For all other parts mesh is not necessary as they are defined as rigid bodies.

3. PROCESSING

In this phase of the analysis new simulation job is created and adjusted. Simulation defined in the previous step took 169 increments to finish.

4. POST-PROCESSING

In the end, results of the simulation were analysed. The software records various parameters, based on what is defined in the first step. In our case points of interest are stresses and strains which occur during the process. On the figure 1. Von Mises stresses are presented on the finished part and on the figure 2. the relevant strains on the part are presented.

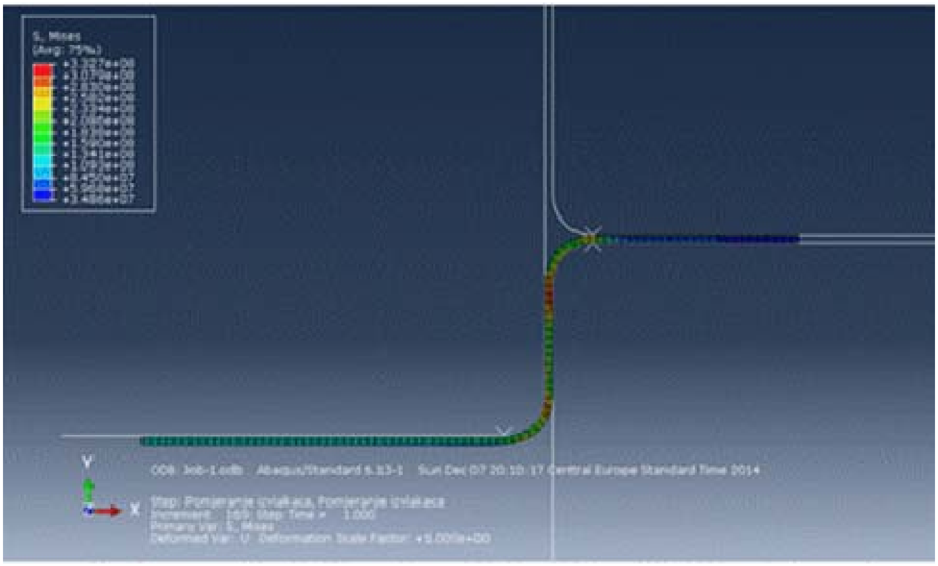

Figure 1. Von Mises stresses

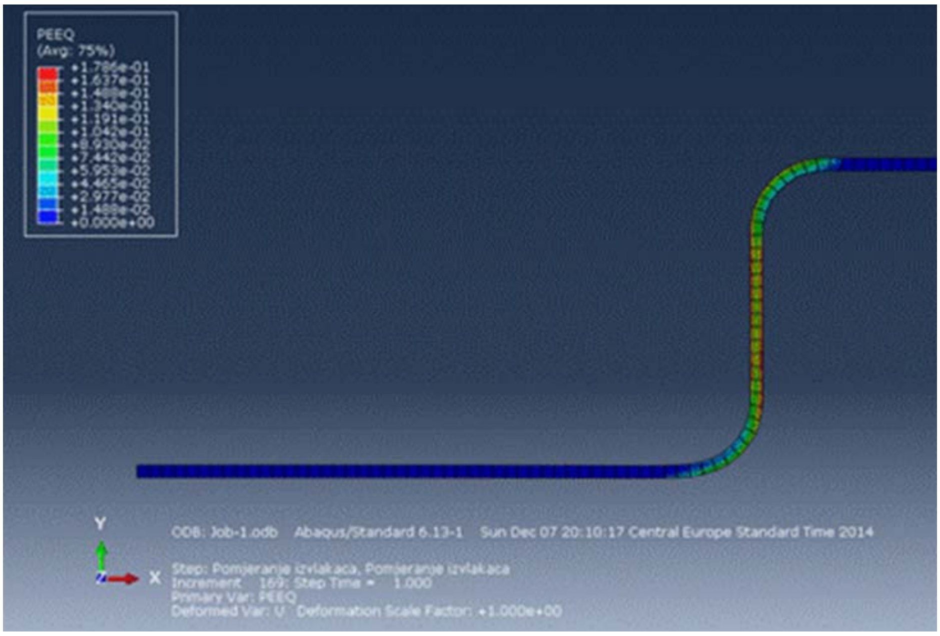

Figure 1. Plastic strain

As expected largest stresses and strains on the part are in the areas of curvature.
Largest recorded strain on the part is 332,7 MPa so we can conclude that the part has gone through significant plastic deformation.
As can be seen from the figure 2. the largest integrated measure of plastic strain is around 17% for this part. If compared to the results on the diagram 1. one can conclude that in this process failure of the part is unlikely.

5. CONCLUSION

Numerical simulation can be of significant use in solving engineering problems which are encountered every day in various processes. Use of these kind of simulation software can be very beneficial in the design stage of the process but also later on for solving manufacturing issues, as shown in the presented example. With correct definition of the problem, correct definition of the parameters of simulation and with approximations which are made in a way to replicate the real state closely enough, decent predictions of stresses and strains for a process which is characterised by high non-linearity of material properties and loads, such as a deep drawing process, can be made.

6. REFERENCES

[1] Documentation: Getting started with Abaqus

[2] Banabic, D., Bunge, H.-J., Pöhlandt, K., Tekkaya, A.E..: Formability of Metallic Materials, Springer-Verlag Berlin Heidelberg, Germany, 2000.

[3] Smoljanič, S.; Pepelnjak, T.: The impact of strain rate on sheet metal formability at room temperature, RMZ - Materials and Geoenvironment, Vol 60., 2013.

[4] Technical data sheet - Steel DC04 (1.0338) - Robert Laminage S. A.