DOI: 10.2478/aucts-2025-0012

ACTA UNIVERSITATIS CIBINIENSIS – TECHNICAL SERIES
Vol. 77 2025

FINITE ELEMENT ANALYSIS OF THIN SHEET METAL BENDING PROCESS

POPP Mihai-Octavian
Faculty of Engineering, Department of Industrial Machines and Equipment, "Lucian Blaga" University, Sibiu, Romania, mihai.popp@ulbsibiu.ro

POPP Gabriela-Petruța
Faculty of Engineering, Department of Industrial Machines and Equipment, "Lucian Blaga" University, Sibiu, Romania, gabriela.popp@ulbsibiu.ro

RACZ Gabriel-Sever
Faculty of Engineering, Department of Industrial Machines and Equipment, "Lucian Blaga" University, Sibiu, Romania, gabriel.racz@ulbsibiu.ro

POPP Ilie-Octavian
Faculty of Engineering, Department of Industrial Machines and Equipment, "Lucian Blaga" University, Sibiu, Romania, ilie.popp@ulbsibiu.ro

Abstract: *The paper presents the numerical analysis of the bending process of thin DC01 steel sheets using the finite element method implemented in the ABAQUS/CAE software. The developed three-dimensional model included the definition of the material's elastic and plastic properties, frictional contact conditions, and geometric nonlinearity effects. The distributions of principal, secondary, and directional strains (ε_x, ε_y) were analyzed, highlighting the critical regions in the contact area with the punch. The maximum strain values, in the range of 1.3–1.6 mm/mm, indicate severe plastic deformation regimes typical of intensive bending processes. The results confirm the ability of the finite element method to reproduce the real material behavior and provide a solid basis for optimizing technological parameters. The study demonstrates the usefulness of numerical simulation as an essential stage in computer-aided design of sheet metal forming processes.*

Key words: bending, finite element method, DC01, numerical simulation, plastic deformation.

1. Introduction

The deformation processes of thin sheet metals represent an essential category of operations in industrial manufacturing, with wide applications in the automotive industry, industrial equipment, lightweight structures, household appliances, and many other sectors. Due to their favorable strength-to-weight ratio, sheet metals enable the production of complex components with low costs and high efficiency, while being compatible with a wide range of forming technologies [1].

Sheet bending is one of the most commonly used plastic deformation operations, applied to obtain prismatic or curved shapes, generally without altering the material thickness. Through careful control of tool geometry and process parameters, precise bending angles can be achieved, which are essential for the subsequent assembly of components [2]. However, the process is influenced by numerous factors: the mechanical properties of the material, elastic springback effects, contact conditions, friction, and fixture stiffness. These factors can lead to geometric deviations from the desired shape, affecting dimensional accuracy and the quality of the final product [3].

In this context, numerical analysis using the Finite Element Method (FEM) becomes an indispensable tool for engineers. Numerical simulation makes it possible to evaluate the distributions of strain and stress, identify critical regions, and estimate the overall behavior of the workpiece before physical prototypes are produced [4]. Furthermore, it allows rapid testing of multiple geometric configurations or process conditions, significantly reducing experimental costs and development time. The ABAQUS software provides advanced capabilities for modeling forming processes, including nonlinear static analysis, frictional contact, complex plastic behavior, and precise geometry discretization [5].

This study presents a numerical simulation of the bending process of a thin plate, using a three-dimensional model developed in ABAQUS. The main objective is to analyze the distribution of specific

DOI: 10.2478/aucts-2025-0012

strains in different directions to better understand the local behavior of the material during bending. The obtained results allow the identification of critical regions and provide useful information for improving geometric accuracy and optimizing the technological process [6].

2. Methodology

The numerical simulation of the bending process was carried out using the ABAQUS/CAE software, following the classical structure of a finite element method (FEM) analysis, divided into three main stages: preprocessing, solving, and postprocessing. The stages and the main activities specific to each phase are summarized in Figure 1. This workflow ensures a systematic approach, from the definition of the geometric and material model to the interpretation of the results obtained from the numerical solution.

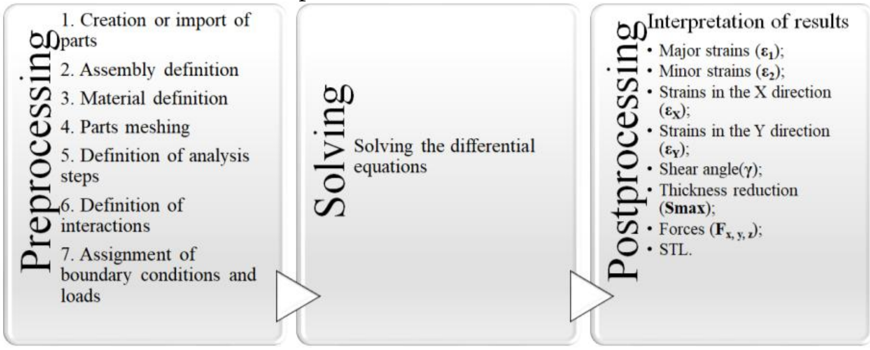

Figure 1: Stages of the finite element analysis

The analyzed model consisted of three main components (Figure 2): the sheet, the punch, and the die plate. The sheet was modeled as a deformable body, while the punch and die plate were defined as discrete rigid parts, since the main focus of the study is the behavior of the workpiece during the bending process. The material used for the sheet is DC01 steel with a thickness of 0.8 mm, characterized by experimentally determined mechanical properties [7]. Table 1 presents the measured values obtained from six specimens, as well as the average and median values used in the simulation.

Table 1: Mechanical properties of DC01 material (thickness 0.8 mm)

Property	Average value
Young's Modulus [MPa]	270000
Yield Strength [MPa]	169.62
Ultimate Tensile Strength [MPa]	169.62
Elongation at Break [mm/mm]	0.325

The elastic properties (Young's modulus and Poisson's ratio) were defined in the Property module of ABAQUS. The plastic behavior was specified using the stress–strain curve derived from experimental data, by converting the characteristic points (yield strength, ultimate tensile strength, and strain hardening coefficient) into a true stress–true strain plasticity table.

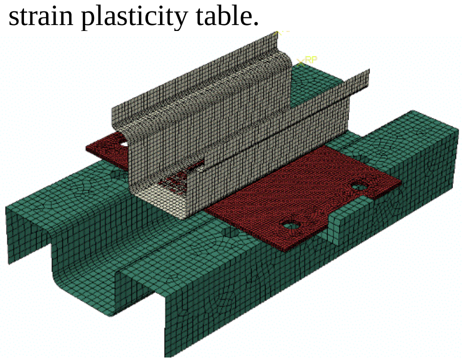

Figure 2: Discretized assembly

The geometry of the parts was discretized using structured hexahedral solid elements, with a global element size of 2 mm, to achieve an optimal compromise between result accuracy and computational cost.

DOI: 10.2478/aucts-2025-0012

The assembly was configured by positioning the sheet between the die plate and the punch. Surface interactions were defined using a surface-to-surface contact formulation, with a friction coefficient of 0.15 and the finite sliding option, suitable for large displacements.

For the boundary conditions:

- the die plate was fully fixed;
- the punch was assigned a controlled vertical displacement corresponding to the deformation phase, followed by a return to its initial position during the unloading step.

The analysis was performed as a nonlinear static simulation, with geometric nonlinearity (NLGEOM) activated. After completing the model definition, an analysis job was created and executed, and the results were evaluated in the postprocessing module of ABAQUS.

3. Results

The numerical analysis enabled the evaluation of strain distributions in the sheet for different directions and components, as well as the identification of mechanically critical regions. The results were extracted from the Visualization module of the ABAQUS software by displaying strain contour plots on the deformed shape of the model.

The distributions of the major (ε_1) and minor (ε_2) strains provide valuable insight into how the material deforms during the bending process. These are shown in Figures 3 and 4.

- The major strains (ε_1) reach their maximum values in the lower central area of the sheet, corresponding to the direct contact with the punch. These valuea indicate the most highly stressed plastic region, where the material undergoes maximum stretching.
- The minor strains (ε_2) show complementary distributions associated with the lateral contraction of the material during plastic flow.

The presence of high strain values in these regions reflects the local behavior of the material during the bending process and allows the identification of points susceptible to the occurrence of instabilities or loss of geometric accuracy.

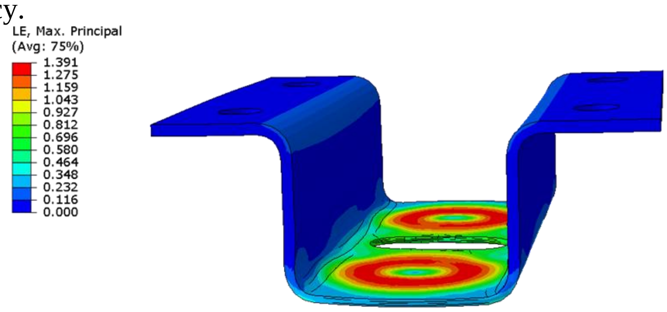

Figure 3: Major strains (ε_1)

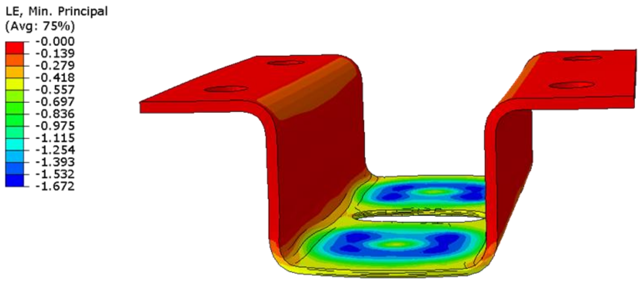

Figure 4: Minor strains (ε_2)

For a detailed evaluation of the sheet behavior, the specific strains along the two main directions of the coordinate system were also analyzed (Figures 5 and 6):

- ε_x – the strain in the X direction, associated with longitudinal stretching;;
- ε_γ – the strain in the Y direction, associated with lateral flow.

The distribution of ε_x highlights the regions where the material is most intensely stretched along the main bending direction, represented by blue shades in the contour map. The ε_γ strains show concentrations mainly around the edges and support areas, suggesting local shear effects and the redistribution of the material volume during the process.

DOI: 10.2478/aucts-2025-0012
ACTA UNIVERSITATIS CIBINIENSIS – TECHNICAL SERIES
Vol. 77 2025

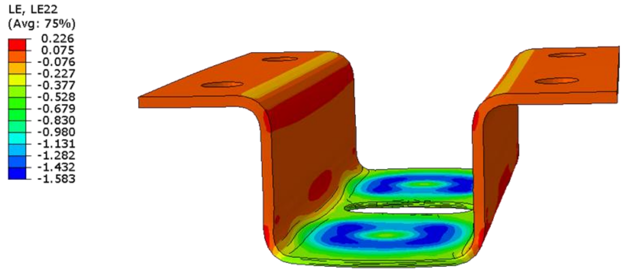

Figure 5: Strains in the X direction (ε_x)

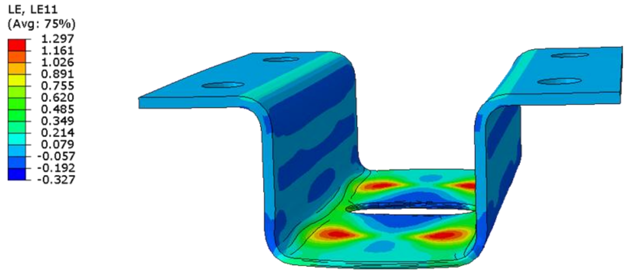

Figure 4: Strains in the Y direction (ε_y)

Table 2 summarizes the maximum strain values obtained from the analysis for the main components: principal and secondary strains, as well as strains along the X and Y directions. The values were extracted directly from the numerical postprocessing in ABAQUS, and the percentages displayed in the legend were converted into relative values [mm/mm].

Table 2: Maximum strain values resulting from the simulation of the bending process

Type of strain	Maximum value [%]	Maximum value [mm/mm]
Major strains (ε_1)	139.1	1.391
Minor strains (ε2)	167.2	1.672
Strains in the X direction (ε_x)	158.3	1.583
Strains in the Y direction (ε_y)	129.7	1.297

4. Conclusions

The numerical simulation performed in ABAQUS enabled a detailed analysis of the behavior of a DC01 steel sheet during the bending process. The developed three-dimensional model, which included frictional contact, plastic behavior, and geometric nonlinearity, provided valuable information regarding strain distributions and the localization of critical regions.

The main conclusions are as follows:

- The distributions of principal and directional strains showed stress concentration in the central area, at the contact between the punch and the sheet, where the material is subjected to significant plastic deformation ($\varepsilon_1 \approx 1.39$ mm/mm).
- The secondary and Y-direction strains revealed the mechanisms of lateral contraction and volumetric redistribution, essential for a complete understanding of the plane-strain flow phenomenon.
- The high strain values obtained numerically are characteristic of intense plastic regimes, confirming the model's effectiveness in capturing local material behavior.
- The use of experimentally determined mechanical properties for DC01 steel contributes to improving the accuracy of the simulation and to correlating the numerical results with the physical behavior of the material.

By applying this methodology, process parameters can be predictively evaluated and potential critical zones identified, providing engineers with valuable support for optimizing tool geometry and process conditions without the need for extensive experimental testing.

5. References

DOI: 10.2478/aucts-2025-0012
ACTA UNIVERSITATIS CIBINIENSIS – TECHNICAL SERIES
Vol. 77 2025

1. Marciniak, Z., Duncan, J.L., Hu, S.J., Mechanics of Sheet Metal Forming, 2nd edition, Butterworth-Heinemann eds., Oxford, UK, (2002).
2. Groover, M.P., Fundamentals of Modern Manufacturing: Materials, Processes, and Systems, 4th edition, John Wiley & Sons eds., Hoboken, NJ, (2010).
3. Banabic, D., Sheet Metal Forming Processes: Constitutive Modelling and Numerical Simulation, Springer-Verlag eds., Berlin, Germany, (2010).
4. Barlat, F., Chung, K., "Plastic behavior and constitutive modeling of sheet metals," International Journal of Plasticity, Vol. 23, No. 4, pp. 481–509, (2007).
5. Picas, I., Cser, L., "Finite element simulation of sheet metal bending process using ABAQUS," Procedia Engineering, Vol. 132, No. 1, pp. 502–509, (2015).
6. ABAQUS Analysis User's Manual, Dassault Systèmes Simulia Corp., Providence, RI, (2023).
7. DIN EN 10130, Cold rolled low carbon steel flat products for cold forming – Technical delivery conditions (DC01–DC06), European Committee for Standardization, Brussels, Belgium, (2006).