STATE OF STRESSES WITHIN U BENT PARTS AND ITS EFFECT ON THE SPRINGBACK AMOUNT

Crina AXINTE[1], Monica IORDACHE[2]
[1]University of Bacau, Romania,[2]University of Pitesti, Romania

Abstract: *This paper presents the stresses distribution within U bent parts and its influence on the amount of springback, when different blankholder forces and friction coefficients are used. The analysis was performed with the help of the ABAQUS software. It was found that greater stresses induced inside the part and a uniform distribution of them through the sheet thickness reduces the springback amount, increasing thus the part accuracy.*

Keywords: U bent parts, stresses distribution, springback amount

INTRODUCTION

In recent years, considerable efforts have been made in solving sheet metal forming problems by using finite element simulation. This method allows to check part and tools geometry at early design stage, as well as to optimize the process parameters, in order to obtain a final part without failures such as necking, wrinkling, springback, etc.
Many simulation programs, like LS-DYNA, PAM-STAMP, DYNAFORM, AUTOFORM, MARK, ABAQUS and so on, are used to study the sheet metal forming processes and their afferent phenomena. Generally, an explicit scheme is used for the forming simulations while for the springback simulation an implicit integration method is preferred. The optimal solution is then to have both implicit and explicit methods available in the same program and to be able to switch automatically from one to the other.
In this paper, a dynamic explicit procedure was used to perform a typical three-dimensional U bending process and an implicit scheme was used to simulate the springback. Two process parameters – blankholder force and friction conditions, respectively, were varied. The aim was to analyze the stress distribution within part and its effect on springback.
The analysis was made with ABAQUS software (ABAQUS/Explicit for the loading process and ABAQUS/Standard for the unloading process).

SIMULATION OF U BENDING PROCESS

In order to simulate the U bending process, a 3D model was used (fig. 1). For simplicity, the tools (die, punch and blank holder) were described by using rigid analytical surface and only the blank was considered deformable. The elements used for the mesh were of S4R type and 5 integration points were used through the sheet thickness. The diameter of the punch was 78mm and the punch radius was 10 mm. The outer diameter of the die was 180mm and the inner diameter was 81mm with the entry radius of 5mm.
The Coulomb's friction law was imposed between the contact interfaces of the sheet and the tools surfaces.
The initial blank geometry was a rectangular shape, 350 mm×30 mm, and 0.8 mm in thickness. The material for the blank was A5 STAS 10318-80 steel, whose mechanical characteristics and the methodology of their determination are presented in [1]. The anisotropic behaviour of material was described into ABAQUS software by using the Hill's criterion [eq. 1] and the POTENTIAL function from ABAQUS.

$$f(\sigma)=\sqrt{F(\sigma_{22}-\sigma_{33})^2+G(\sigma_{33}-\sigma_{11})^2+H(\sigma_{11}-\sigma_{22})^2+2L\sigma_{23}^2+2M\sigma_{31}^2+2N\sigma_{12}^2} \quad (1)$$

where F, G, H, L, M and N are constants determined with the help of the Lankford's coefficients obtained by tests of the material in different orientations (0°, 45° and 90°).
In the case of planar anisotropy, the eq. 1 becomes:

$$f(\sigma)=\sqrt{F(\sigma_{22}-\sigma_{33})^2+G(\sigma_{33}-\sigma_{11})^2+H(\sigma_{11}-\sigma_{22})^2+2N\sigma_{12}^2} \tag{2}$$

Knowing the values of the coefficients r_0, r_{45} and r_{90}, experimentally determined in [1], the constants F, G, H, L, M and N could be calculated with the help of the following equation:

$$r_\theta=\frac{H+(2N-F-G-4H)\sin^2\theta\cos^2\theta}{F\sin^2\theta+G\cos^2\theta} \tag{3}$$

where θ is the angle against rolling direction (0°, 45° and 90°).

Different blankholder force values (1.5, 2.5, 5, 10, 15 and 25 kN) were considered in simulation and two different coefficients of friction (0.1 and 0.075) were applied to the interface between the blank and the tools surface.
By using an import procedure, the results of the forming process were passed into ABAQUS/Standard in order to simulate the springback phenomenon.

SIMULATION RESULTS

The dimensional accuracy after springback was evaluated by three geometric parameters, illustrated in figure 2: the angle between the bottom and the wall (θ_1), the angle between the flange and the wall (θ_2), and the curvature (ρ) of the side wall. In the ideal cases of no springback, 90° angles of θ_1 and θ_2 and the flat side wall were expected.

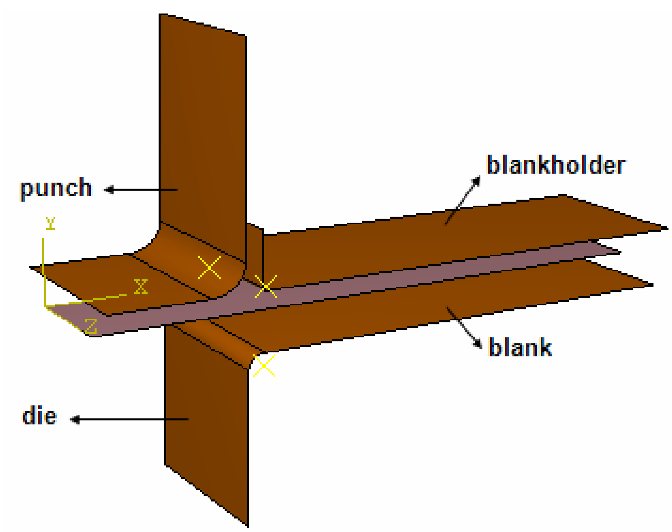

Fig. 1 Model used in simulation

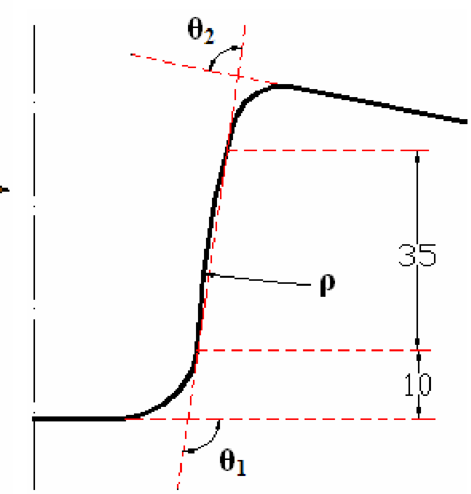

Fig. 2 Parameters of springback

Distribution of stresses and its effect on springback when different BHFs and a friction coefficient μ = 0.1 were used

Distribution of stresses within part at the end of the forming process is presented in figure 3. This state of stresses determined the springback occurrence. After springback, a redistribution of stresses could be observed.
The variation of the three parameters that quantify the springback amount is presented in figure 4 and figure 5.

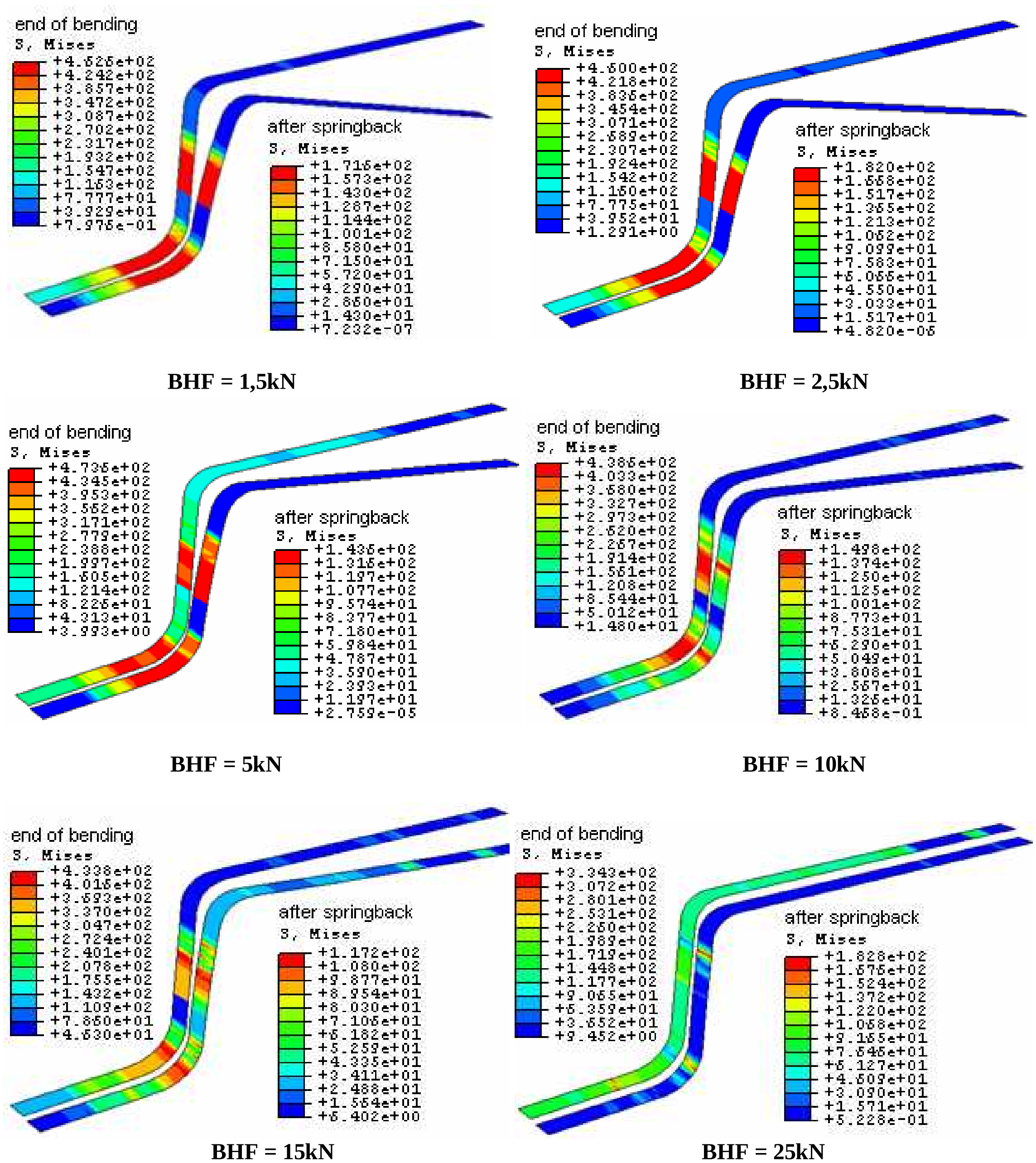

Fig. 3 Distribution of stresses before and after springback

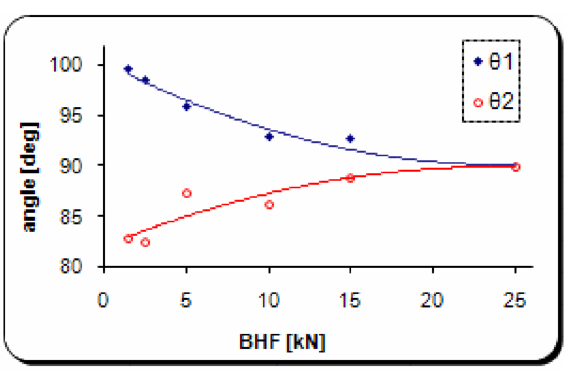

Fig. 4 Variation of the springback angles

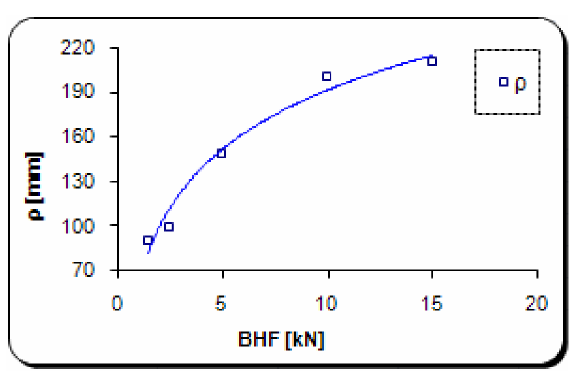

Fig. 5 Variation of the side wall curvature

Distribution of stresses and its effect on springback when different BHFs and a friction coefficient μ = 0.075 were used

Distribution of stresses within parts at the end of the forming process as well as their distribution after the springback occurrence is presented in figure 6.

The variation of the three parameters that quantify the springback amount is presented in figure 7 and figure 8.

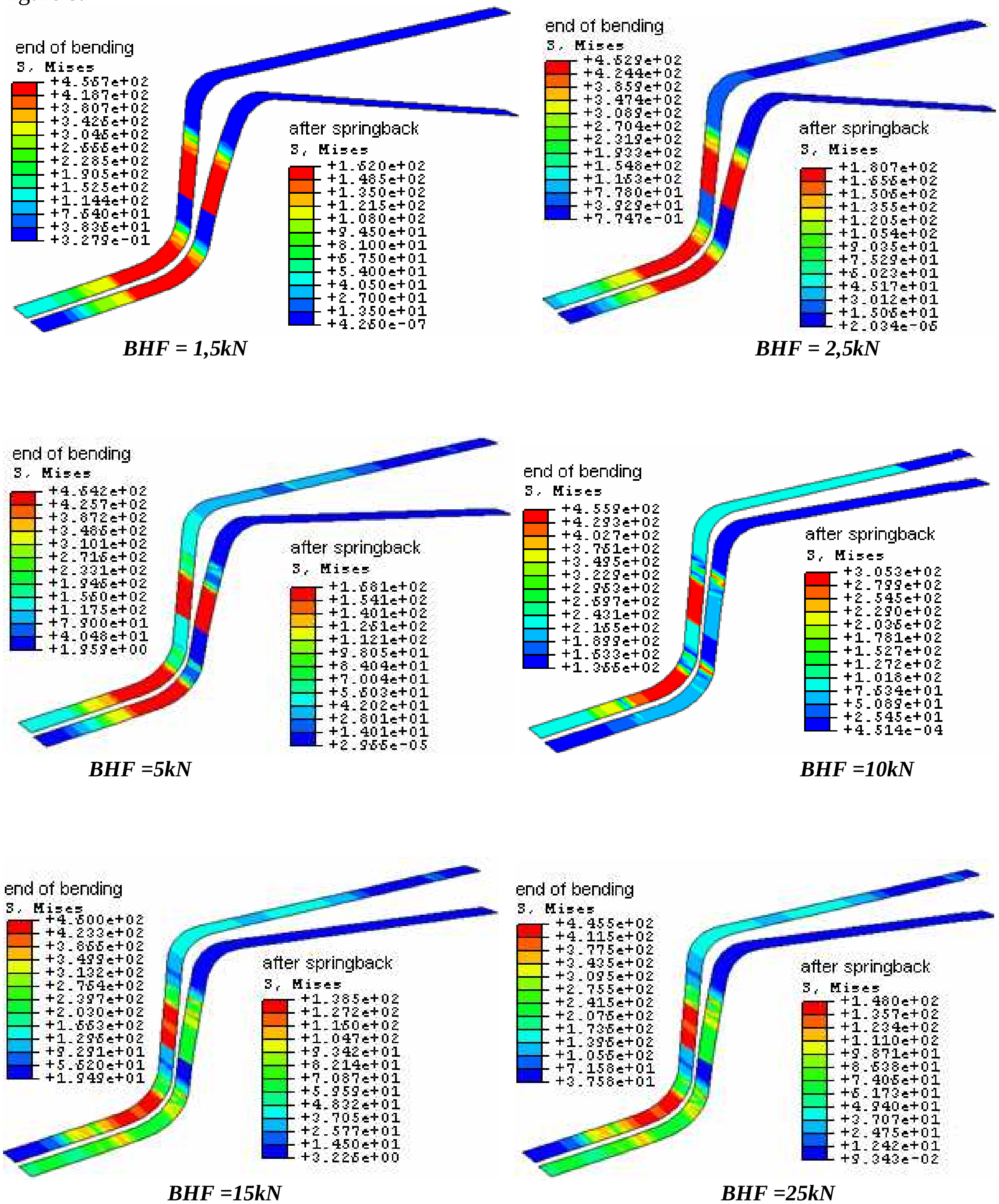

Fig. 6 Distribution of stresses before and after springback

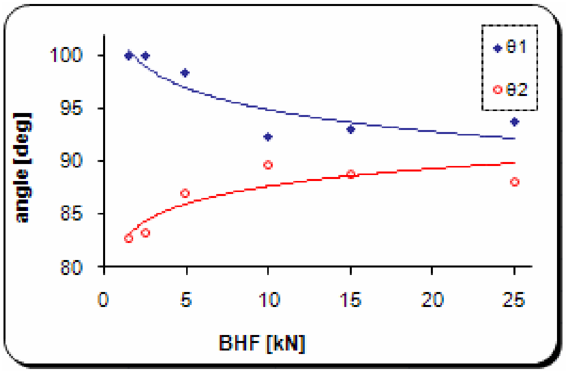

Fig. 7 Variation of the springback angles

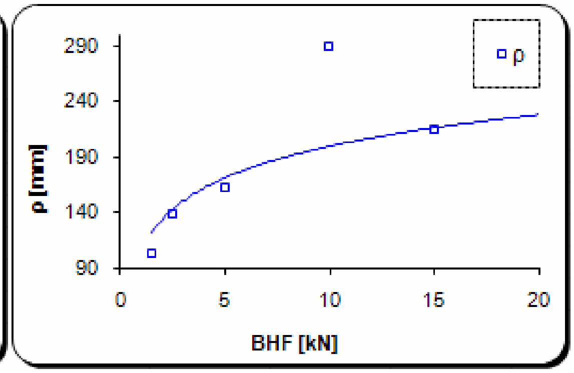

Fig. 8 Variation of the side wall curvature

CONCLUSIONS

As shown in figures above, it was found that different values of the BHF determine different distributions of stresses within parts that, in turn, have different effect on the springback amount.
It was observed that all the three parameters that quantify springback (θ_1, θ_2, ρ) were significantly affected by the BHF value: the angles tended to 90^o while the side wall curvature was getting bigger (it became flat for BHF = 25kN) as BHF increased.
The same variation of the springback parameters was observed when a different coefficient of friction was used (μ = 0.075). However, the values of these parameters were found bigger than in the case when μ was set to 0.1 (figure 9). Thus, a higher friction coefficient between the blank and tools leads to some improvement of the part shape after unloading.
This trend of the three springback parameters could be explained by the fact that bigger BHFs and higher friction conditions induces greater stresses inside the part and uniforms the stresses distribution through the sheet thickness (an example could be seen in figure 10 and figure 11).

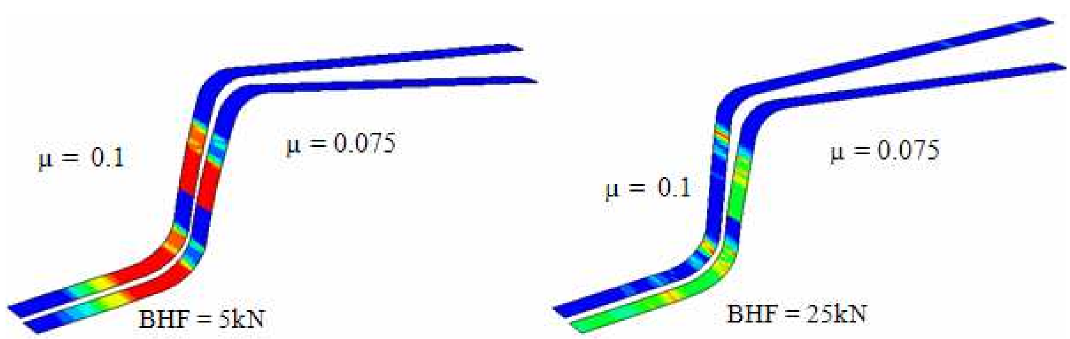

Fig. 9 Influence of the friction conditions on springback

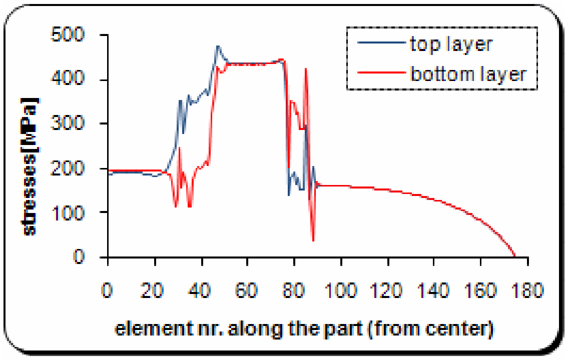

Fig. 10 Stresses distribution, BHF=5kN, µ=0.1

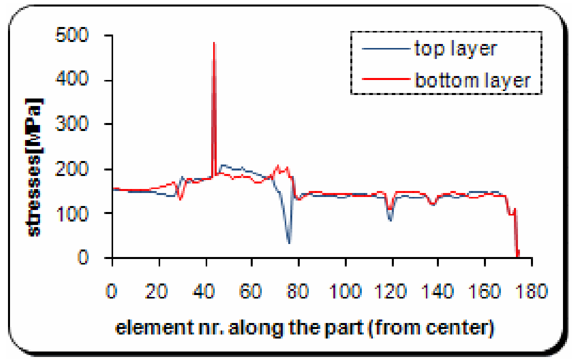

Fig.11 Stresses distribution, BHF=25kN, µ=0.1

REFERENCES

[1] Iordache M, *"Contributii cu privire la simularea proceselor de deformare plastica la rece a tablelor"*, *Phd. thesis (University of Pitesti & University of Metz), 2007*

[2] ABAQUS, version 6.5 Documentation.