7th INTERNATIONAL MULTIDISCIPLINARY CONFERENCE
Baia Mare, Romania, May 17-18, 2007
ISSN-1224-3264

SPRINGBACK PREDICTION OF THE V BENDING PROCESS USING FINITE ELEMENT SIMULATION

Florica Mioara Groze[1]; Gheorghe Achimaş[2] Lucian Lăzărescu[3] , Vasile Adrian Ceclan[4]
[1]Ph.D. Student; [2]Ph.D., Prof.,;[3]P.h.D. Assist. Lect.; [4]Ph.D., Student.,
Department of Manufacturing Engineering, Faculty of Machine Building
B-dul Muncii 103-105, RO-400641 Cluj-Napoca, Romania

Abstract:
This paper presents a study of springback in the V-bending metal forming process using the finite elements method. In order to elaborate the simulation model of the V bending process of sheet metals we established a set of parameters which have a significant influence on the springback in a V bending process: dimensional parameters of the raw part (rectangular strip), geometry of the tools and the elastic-plastic behavior of the material. The analysis is carried out using the commercial code ABAQUS. The forming process is simulated in two steps with ABAQUS/Explicit. The results are analyzed using Abaqus/CAE.
Keywords: Metal forming, Springback, Finite elements, V-bending

1. INTRODUCTION

In the automobile industry bending process is widely used. One of the most important parameter which influences the quality of the bended pieces is springback, that's why the accurate estimation of springback in this industry is very important.

The springback simulation is quite difficult and complicated: in order to obtain useful results, a precise description of the material behavior and of the tool configuration and motion is necessary. The simulation has as main purpose to increase of accuracy when determining the springback angle, to simplify the technology and to reduce the computation time. In the past, sheet metal bending process was depending on the designer's experience and involved trials and errors to obtain the desired results (Thanki, 2001). Trial and error can involve adjustments made to the machine tools and process control to compensate for variation in material and unexpected parameters (Yang,1996). Nowadays, with the advent of computation technology, sheet metal bending processes can be analyzed prior to experiments using the finite element method. Lee and Yang (Lee&Yang, 1998)[use the finite element method to evaluate large spring-back in a U-draw bending process. Esat, Darendeliler and Goker (Esat, 2002) apply the finite elements simulation to analyse the springback, the total equivalent strain and the equivalent von Misses stress in the sheet bending process of various aluminum alloys with different thicknesses. The simulations have been done using MARC/MENTAT.

Oh and Kobayashi (Oh & Kobayashi, 1980) used FEM to simulate the bending process, comparing the predictions of rigid-plastic and elastic-plastic material models.

Li and others (Li, 2002) studied the springback of V bent sheets, using a 2D elastic-plastic model. With the help of a linear constitutive model, they kept in mind the fact that Young's modulus is variable during the plastic forming. They proved that the material hardening has a direct influence on the calculus accuracy of the springback obtained from FE simulation.

2. FINITE ELEMENT SIMULATION

Numerical simulation for V bending process of sheet metals was carried out by using a software package ABAQUS/Standard. Because V bending is a symmetric process all the definitions were performed only for the right part. In order to elaborate the simulation model of the V bending process of sheet metals, the following aspects have been taken into account. Establishing a set of parameters which have a significant influence on the springback in a V bending process:

a) dimensional parameters of the raw part (rectangular strip): thickness, length, width;
b) geometry of the tools (Fig. 1): α_s, l, r_p , $r_{pl;}$
c) elastic-plastic behavior of the material:
- elastic properties: E, ν
- coefficients of the Swift hardening law: k, ε_0, n ;

We have used the same material: a deep-drawing quality (DDQ) steel. All the numerical have been performed keeping constant the length (L) and the width of the raw part.

The variable parameters of the bending process are:

- thickness of the part g,
- punch and die angle α_s ;
- bending distance l;
- fillet radius at the punch nose $r_{p,}$
- fillet radius of the die $r_{pl,}$

We have developed a specialized preprocessing program (abaqus_inp), which generates the ABAQUS command file corresponding to a parametric model of the V bending process. The scheme of the process is presented in Figure 1. The parameters defining the shape and the relative positions of the tools are highlighted on this picture.

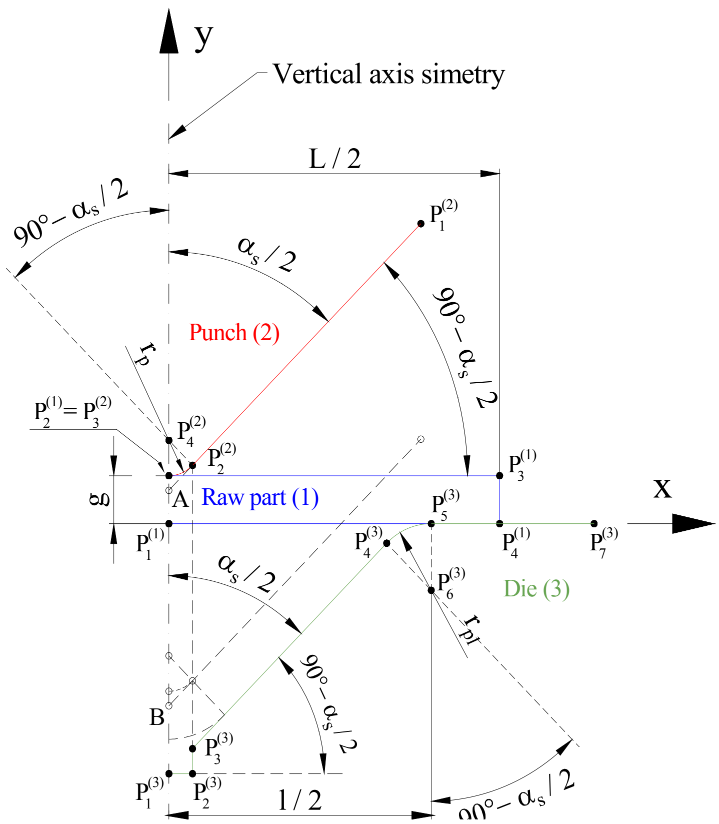

Fig.1 *Parametric model of the V-bending process*

Legend

i=1 index used to identify the points belonging to the outline of the raw part

i=2 index used to identify the points belonging to the punch outline

i=3 index used to identify the points belonging to the die outline

The geometric coordinates of the points defining half of the punch outline (Fig. 1).

Note: In order to have a contact between the part and the punch along its entire length, we establish the following convention:

$$d\left(P_1^{(2)}, P_2^{(2)}\right) = L/2 \tag{1}$$

The points $P_3^{(2)}$ and $P_2^{(1)}$ are coincident:

$$x_3^{(2)} = 0, \qquad y_3^{(2)} = g\,. \tag{2}$$

Using $P_3^{(2)}$'s coordinates, we can determine the $P_4^{(2)}$ point (figure .1):

$$x_4^{(2)} = 0 \qquad y_4^{(2)} = y_3^{(2)} + r_p = g + r_p \tag{3}$$

Next we calculate $P_2^{(2)}$'s coordinates (see fig. 1 and eqns (3)):

$$\begin{cases} x_2^{(2)} = r_p \sin\left(90^\circ - \dfrac{\alpha_s}{2}\right) = r_p \cos\dfrac{\alpha_s}{2}, \\ y_2^{(2)} = y_4^{(2)} - r_p \cos\left(90^\circ - \dfrac{\alpha_s}{2}\right) = g + r_p\left(1 - \sin\dfrac{\alpha_s}{2}\right) \end{cases} \tag{4}$$

Using $P_2^{(2)}$'s point and convention (1), we can find out $P_1^{(2)}$'s position:

$$\begin{cases} x_1^{(2)} = x_2^{(2)} + \dfrac{L}{2}\cos\left(90^\circ - \dfrac{\alpha_s}{2}\right) = r_p \cos\dfrac{\alpha_s}{2} + \dfrac{L}{2}\sin\dfrac{\alpha_s}{2}, \\ y_1^{(2)} = y_2^{(2)} + \dfrac{L}{2}\sin\left(90^\circ - \dfrac{\alpha_s}{2}\right) = g + r_p\left(1 - \sin\dfrac{\alpha_s}{2}\right) + \dfrac{L}{2}\cos\dfrac{\alpha_s}{2} \end{cases} \tag{5}$$

The geometric coordinates of the points defining half of the die outline (figure 1)

The formulas used to find out the coordinates of the points $P_1^{(3)},...,P_7^{(3)}$ are gathered in table 1.

Establishing the length of the punch stroke (fig. 1)

The theoretical stroke of the punch is equal to the distance between points A and B (figure 1), where:

A- intersection of the line $P_1^{(2)}P_2^{(2)}$ with the vertical axis of the bending process;

B- intersection of the vertical axes of the bending process with a line being parallel to line $P_1^{(3)}P_4^3$ and distanced with g value from $P_1^{(3)}P_4^3$.

$$H_{teoretic} = y_B - y_A < 0 \tag{6}$$

Theoretical stroke of the punch is defined by:

$$H_{teoretic} = \frac{r_{pl}\left(1-\sin\frac{\alpha_s}{2}\right)-\frac{l}{2}\cos\frac{\alpha_s}{2}+g}{\sin\frac{\alpha_s}{2}} - g + r_p\frac{1-\sin\frac{\alpha_s}{2}}{\sin\frac{\alpha_s}{2}} = \tag{7}$$
$$= \frac{(r_p + r_{pl} + g)\left(1-\sin\frac{\alpha_s}{2}\right)-\frac{l}{2}\cos\frac{\alpha_s}{2}}{\sin\frac{\alpha_s}{2}}$$

Table 1 The coordinates of the points $P_1^{(3)},...,P_7^{(3)}$

Point	**Coordinates**	
$P_j^{(3)}$	$x_j^{(3)}$	$y_j^{(3)}$
$P_1^{(3)}$	0	$\frac{r_{pl}\left(1-\sin\frac{\alpha_s}{2}\right)-\frac{l}{2}\cos\frac{\alpha_s}{2}}{\sin\frac{\alpha_s}{2}}$
$P_2^{(3)}$	$r_p\cos\frac{\alpha_s}{2}$	$\frac{r_{pl}\left(1-\sin\frac{\alpha_s}{2}\right)-\frac{l}{2}\cos\frac{\alpha_s}{2}}{\sin\frac{\alpha_s}{2}}$
$P_3^{(3)}$	$r_p\cos\frac{\alpha_s}{2}$	$\frac{r_{pl}\left(1-\sin\frac{\alpha_s}{2}\right)-\frac{l}{2}\cos\frac{\alpha_s}{2}}{\sin\frac{\alpha_s}{2}}+r_p\cos\frac{\alpha_s}{2}ctg\frac{\alpha_s}{2}$
$P_4^{(3)}$	$\frac{l}{2}-r_{pl}\cos\frac{\alpha_s}{2}$	$-r_{pl}\left(1-\sin\frac{\alpha_s}{2}\right)$
$P_5^{(3)}$	$\frac{l}{2}$	0
$P_6^{(3)}$	$\frac{l}{2}$	$-r_{pl}$
$P_7^{(3)}$	$1.1\cdot\frac{L}{2}$	0

3. RESULTS AND ANALYSIS

The ABAQUS model (fig. 2) of the V bending process consists in two steps:

- first step: descending motion of the punch;
- second step: simulation of the springback process following the extraction of the bent part from the bending die.

The simulation program is performed for different thickness of the part g, punch and die angle α_s ;bending distance l; fillet radius at the punch nose $r_{p,}$ fillet radius of the die $r_{pl,}$

- thickness of the part (g=0.6mm; 1mm; 1.6mm);
- punch and die angle (αs=600; 900; 1200);
- bending distance (l=60mm; 72mm; 84mm);
- fillet radius at the punch nose (rp=4mm; 5mm; 6mm);
- fillet radius of the die (rpl=4mm; 6mm; 8mm).

Thus we have performed a number of 243 tests using ABAQUS**.**

The ABAQUS software offers a wide range of elements for different types of simulations. The CPE4R element was chosen for our simulation.

As stated in the basic assumptions, the V-bending process is considered a plane strain situation because the width of the work piece is considered to be much larger than the thickness. Hence, a plane strain element was chosen.

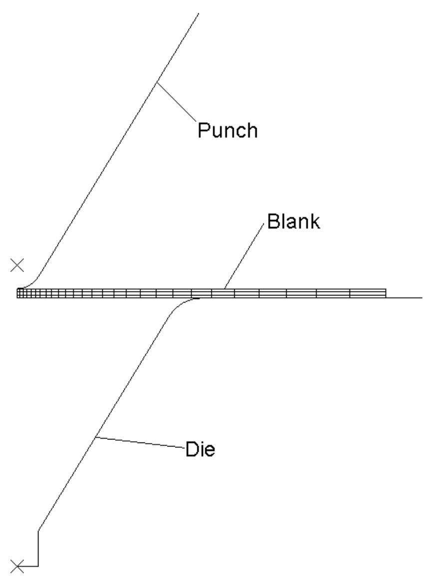

Fig.2 *ABAQUS finite element model of the V-bending process*

Figures 3 – 4 show some numerical results obtained by numerical simulation of a V bending process.

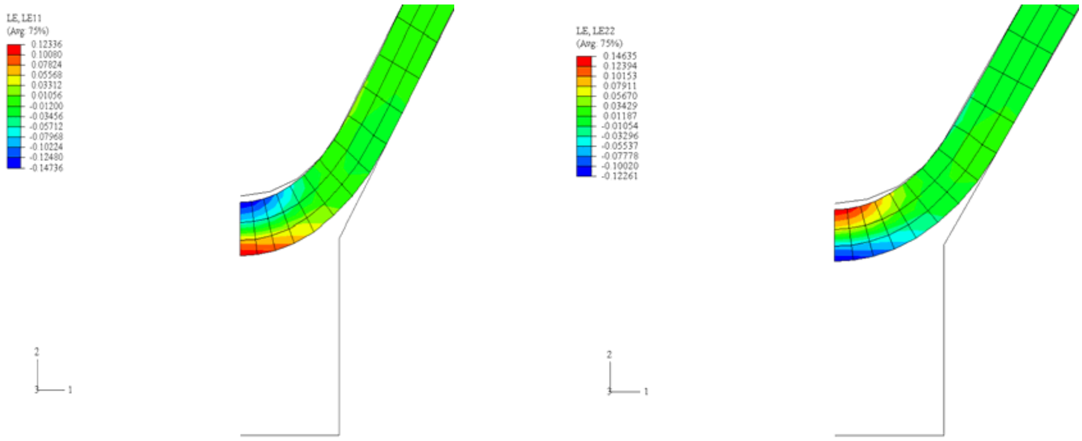

Fig.3 *Logarithmic strain components in the bending area before springback (first step of the simulation)*

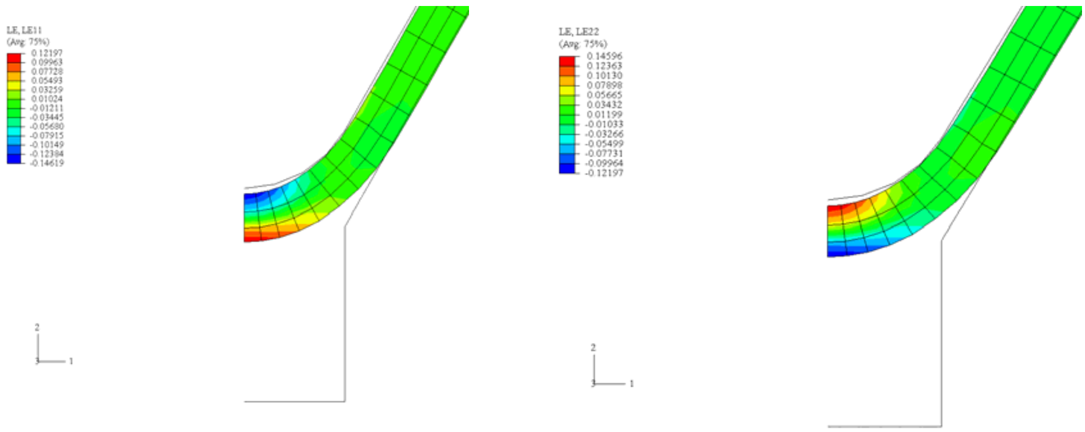

Fig.4 *Logarithmic strain components in the bending area after springback (second step of the simulation)*

In the V-bending process, the material may exhibit negative and positive spring-back caused by deformation as the punch completes the bending operation.

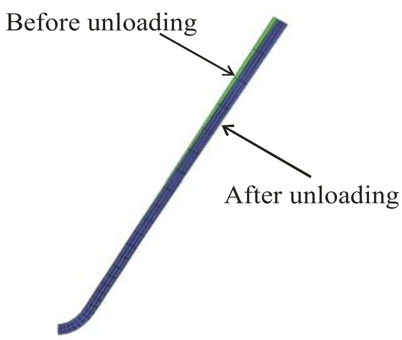

Fig.5 *Springback*

4. CONCLUDING REMARKS

The springback is one of the factors that has a very important influence on the quality of bent parts. Knowing the correct value of the springback angle load removal is important in order to make the correction for the punch geometry before bending. Introducing all data in the program a theoretical load for bending in different conditions can be obtained. The finite element method has been used for evaluating, as correctly as possible, the springback, as well as the stress and strain state in the part before and after the springback. The simulation of the bending process has been performed using ABAQUS Standard.

5. REFERENCES

1) Volkan Esat, Haluk Darendelilr, Mustafa Ihnan Gokler: Finite element analysis in bending of aluminium sheets, Material and Design 23 (2002), pp. 223-229
2) S.W. Lee, D.Y. Yang, An assessment of numerical parameters influencing springback in explicit finite element analysis of sheet metal forming process, J. Mater. Process. Technol. 80/81 (1998) 60–67.
3) Xuechun Li, Yuying Yang, Yongzhi Wang, Jun Bao and Shunping Li: Effect of the material-hardening mode on the springback simulation accuracy of V-free bending, Journal of Materials Processing Technology, Volume 123, Issue 2, 30 April 2002, Pages 209-211
4) S. I. Oh, Shiro Kobayashi: Finite element analysis of plane-strain sheet bending. International Journal of Mechanical Sciences, Volume 22, Issue 9, 1980, Pages 583-594
5) S.J. Thanki, H.K. Raval, A.K. Dave, Prediction of the punch reversal position under V-plate bending using real material (power-law) behavior, J. Mater. Process. Technol. 114 (2001) 227– 232.
6) M. Yang, K.-I. Manabe, H. Nishimura, Development of real-time process control system for precision and flexible V-bending with an on-line database, J. Mater. Process. Technol. 60 (1996) 249–254.
7) *** ABAQUS User's Guide (release 6.4). Electronic documentation