7th International DAAAM Baltic Conference
"INDUSTRIAL ENGINEERING"
22-24 April 2010, Tallinn, Estonia

COMPUTER MODELING APPLICATION IN FORMING PROCESSES OF SHEET METAL PARTS

Taraba, B. & Hajdu, Š.

Abstract: *This article deals with the prediction of bending force quantification for sheet metal part made of low carbon steel. Dimensions of sheet metal part were defined and material properties of steel were obtained from experiments. Geometry of bending device was assigned by the rules for bending technology. Analysis of the bending force value was assigned analytically (mathematical and physical model based on forming theory) and computer modeling in scientific-technical computer aided engineering (CAE) software. The simulation models were generated as contact structural task with large nonlinearity behaviors. Obtained results contain stress-strain states, bending force time dependence and force value results for realization of bending technological process.*

Key words: FEM, CAE, forming, bending, computer modeling

1. INTRODUCTION

Modeling generally means exploring objects using artificially generated objects. The engineering modeling has undergone a long evolution - since traditional practices into modern system solutions. The proposed object (PO) is created for the defined technical object (TO). As a result of creative process is then generation of material object (MO), i. e. model which is similar to TO and consistent with theory of similarity relation [1]. Computer modeling was used for forming technology with orientation on bending processes. The solved technical object (TO) was the technological process of sheet metal bending in a “U” shape. The creation of proposed object (PO) consisted of classical approach based on theory of forming technology and numerical analysis. Computer simulations were processed with software Ansys and Abaqus. Retaliations between various approaches and chosen parameters of bending process were taken from obtained results.

2. THEORETICAL BASE

Theoretical base for bending “proposed object (PO)” is applied for calculation of blank length, bending force, calibration force and bending work. Length of blank equals to the length of neutral line l_p at point of bend and total length of undeformed segment. For the length l_p is a mathematical relationship [2]

$$l_p = \frac{\pi}{180} \alpha R, \qquad (1)$$

where α is acute angle [°] and R is radius of the neutral line, $R = r + xs$, x is a ratio of material thickness to the location of the neutral line [-] ($x = 0.39$) [2], s is material thickness [m].

The equation for maximum bending force uses relationship [2],

$$F_b = \frac{bs^2}{2L} R_m C, \qquad (2)$$

where b is width of sheet metal [m], L is bending arm [m], ($L = 2r + 1.15s$), R_m is failure strength of material and C is a factor taking into account several parameters including friction, $C = 1,6 \div 1,8$. The technological gap for “U” shape bending is setting up to value 1.15*s*.

Calibration bending force is used by equation: $F_{cal}= (2.0 \div 2.5)\, F_b$. The chosen value was 2.5. The bending work W is described in the classical theory with the formula [2]

$$W = F_b h \psi \,, \qquad (3)$$

where h is length of the punch displacement and ψ is coefficient of the diagram fullness, $\psi = 0.5 \div 0.6$.

3. SIMULATION MODEL

The simulation model was created for proposed model of technological bending process in stages: a) geometrical model, b) material model, c) generation of contact pair, d) constraints defining, and solution procedure setting. For the simulation models analysis were applied the interpretation codes Ansys [3] and Abaqus [4].

Geometry of the sheet metal is in Fig. 1. Length of unfolded part was obtained with help of equation (1) and was compared with length of unfolded part in software Catia V5 [5]. Length of unfolded sheet metal was 108.16 mm.

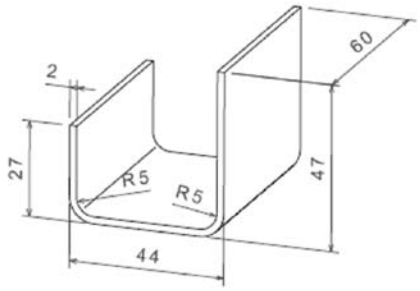

Fig. 1. Geometry of sheet metal

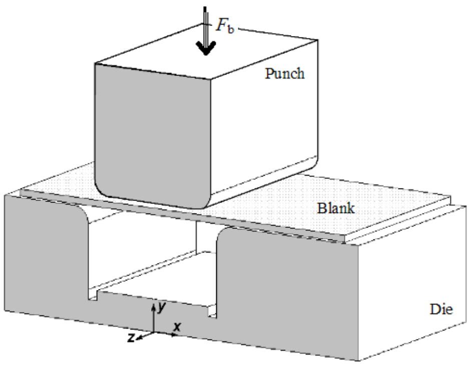

Fig. 2. Design of bending tools

The geometrical model of bending tool was created in two-dimensional plane x-y. On the base of sheet metal dimensions and technological gap was generated mesh of simulation model for Ansys, Fig. 3. It was applied plane element Plane182 with plane-stress behavior.

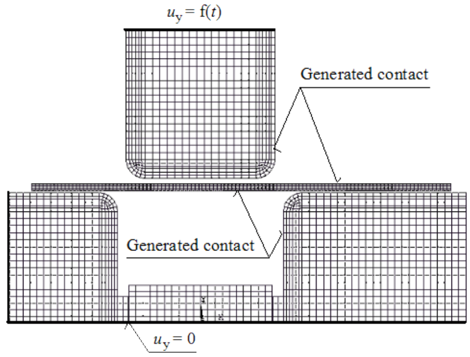

Fig. 3. Generated mesh for Ansys

For numerical analysis by Abaqus was used generated mesh, Fig. 4. Type of the used 2D element was CPS4R with plane-stress behavior.

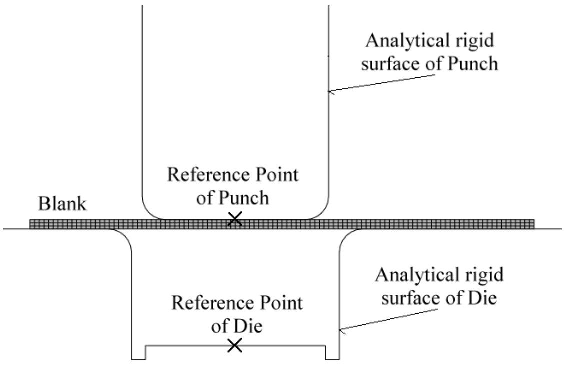

Fig. 4. Generated mesh for Abaqus

Blank is made from low carbon steel STN 411373 (DIN 1.0036). Elasticity modulus of steel is 210 GPa and Poisson's ratio is 0.29. Strain-stress curve for used material was obtained from experiment. The failure strength R_m is for applied material 430.15 MPa. The curve which was used in the calculation is in Fig. 5. It was used nonlinear material model with multi-linear kinematic hardening.

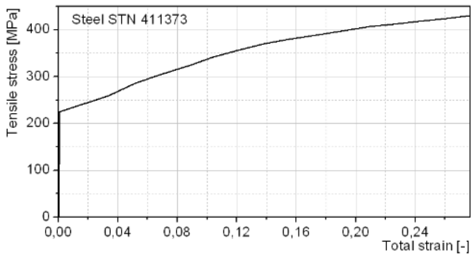

Fig. 5. Strain-stress curve for used material

Generated contact pair between parts Punch-Blank and Blank-Die was of standard type "node to surface". Friction coefficient between blank and other parts of bending tool was taken account 0.3. According to [4] was possible to consider dynamic friction coefficient in interval from 0.12 (slick surface) to 0.52 (dry surface). The constraints of the simulation model were defined with displacement $u_x = 0$ on the nodes of mesh with coordinate $x = 0$ and $u_y = 0$ for nodes on base plate of die, location $y = 0$. Velocity of the punch was not taken in account. The analysis was static. The displacement was defined on the top nodes of punch $u_y = -0.027$ m for Ansys code and -0.025 for Abaqus code. Computational procedure was static with solution control by "large displacement static" [3,4].

4. OBTAINED RESULTS

From results files were chosen data which are representative for solved task. Fig.6 shows bending force F_b dependence on punch displacement. Maximal bending force witch was obtained from numerical analysis is 7271 N at punch displacement 0.013 m (Ansys) and 7120 N at 0.012 m (Abaqus). Fig. 6 is showing forces: maximal bending force by [2] F_b = 7134 N and calibration force F_{cal} = 17835 N and the bending work (area under curve) also.

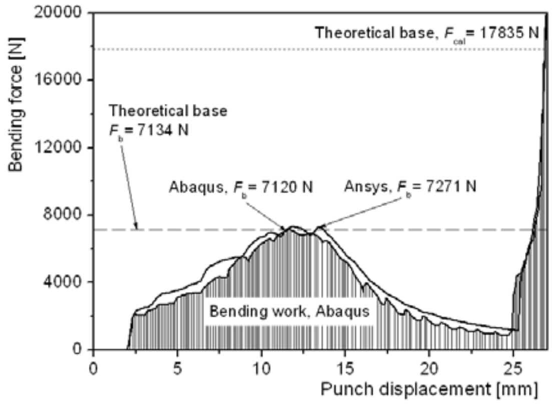

Fig. 6. Bending forces and bending works dependences on punch displacement

Figure 7 shows the shape geometries of the bending process solved by Ansys. The maximum bending force arised at the configuration in detail "D". The deformed shape at the end of bending process is shown in Fig. 7 (left).

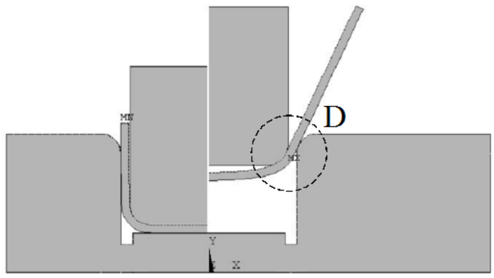

Fig. 7. Shape geometries of the bending - deformed edge for max. F_b (right), bending process finish (left).

The contour plots of equivalent von Mises stresses and von Mises total strains are in Fig. 8.

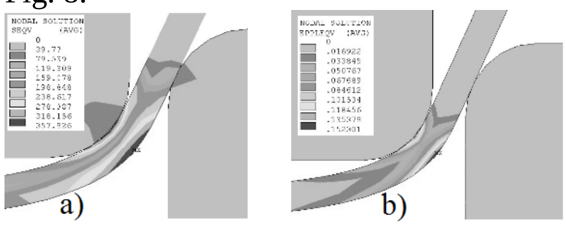

Fig.8. Detail "D": a) Contour plots of equivalent von Mises stress [MPa], b) von Mises total strain [-]

Comparison of results from theoretical base and numerical simulations is in Table 1.

Analysis type		F_b [N]	W [J]	ψ [-]
Theoretical base [2]		7134	98.1	0.55
Nume-rical	Ansys	7271	108.2	0.61
	Abaqus	7120	101.7	0.57

Table 1. Comparison of the obtained results

5. CONCLUSION

System approach of the modeling of chosen problem from forming technology deepens knowledge of solved process.

Bending process data of sheet metal were calculated with help classical approach and were compared with results obtained from software Ansys and Abaqus.

Comparison of Ansys and Abaqus results with obtained results of the theoretical base gives a very good agreement at bending force quantification. The following relative errors were obtained: by comparison of bending forces: 1.9 % for Ansys and 0.2 % for Abaqus, by comparison of bending works: 10.3 % for Ansys and 3.7 % for Abaqus, by comparison of coefficients of fullness: 10.1 % for Ansys and 3.6 % for Abaqus. Effective exploitation of actual software must be supported with wide range of theoretical knowledge. The purpose was to point out that numerical analysis is very useful for solution forming tasks.

6. REFERENCES

1. Janíček P., Ondráček E.: *Řešení problémů modelováním*. *Modeling problem solving*. Brno, VUT Brno, FS 2001.

2. Bílik, J., Kapustová, M., Ulík, A.: *Forming Technology: Instructions for exercises.* Bratislava: STU Bratislava, 2004. ISBN 80-227-2099-2. Available on the Internet: (https://sweb.mtf.stuba.sk)

3. *Ansys Theoretical Manual*, Release 10.0. (2005). Available on the Internet: http://www.tsne.co.kr/intra/data_center/ansys/theory.pdf Accessed 2010-02-20

4. *Abaqus Theoretical Manual*, Release 6.6. Available on the Internet http://129.25.16.135:2080/v6.5/books/stm/default.htm Accessed 2010-02-20

5. *CATIA V5*: *Konstruktionsprozesse in der Praxis*. Munchen: Hanser Verlag, ISBN 3-446-22970-1, 2005.

6. *LS-DYNA theory manual*, *LS-DYNA Support*, 2005. Available on the Internet: http://www.dynasupport.com/manuals/additional/ls-dyna-theory-manual-2005 beta/view

7. Panthi, S. K., Ramakrishnan, N., Pathak, K. K., Chouhan, J.S.: *An analysis of springback in sheet metal bending using finite element method (FEM).* (2007). Journal of Materials Processing Technology, Vol. 186, Issues 1-3, 7 May 2007, Pages 120-124. Available on the Internet: 10.1016/j.jmatprotec.2006.12.026

ACKNOWLEDGEMENT

This article was realised with the support of grants: VEGA 1/0721/08 and VEGA 1/0256/09.