Forming Processes in Manufacture of an Access Cover and Control of Position Tolerances

Procesy formovania pri výrobe kontrolného krytu a kontrola tolerancií umiestnenia

Marko Kršulja, Mauro Štefančić, Branimir Barišić

Abstract

In this paper a sheet metal manufacturing process is presented with the combination of several metal forming technologies that are needed for the production of an access cover. Process consists of several stages: modelling (*Solid Works*), NC programing (*ToPs*), punching, bending etc. This article is about the adoption of known engineering solutions CAD, CAM and CAPP into CIM. In order to avoid bending errors the correct calculation of punch penetration and the bend allowance has to be made. Bending parameters have to be right (straight bends, precise shapes, accurate angles) and punching distances calculated in regard to bending. Especially important for parts with holes or notches that have to be aligned perfectly after bending. This study illustrates prediction with acceptable accuracy of the bend allowance, springback angle, and punch stroke to obtain the desired final product dimensions. This is important for high quality production especially of assemblies with narrow tolerances where a perfect fit is required for installation of the part.

Keywords: sheet metal, bending, punching, modelling

Abstrakt

Príspevok prezentuje výrobný proces oceľových plechov v kombinácii s niektorými technológiami tvarovania ocele, ktoré sú pri výrobe kontrolného krytu prítomné. Proces pozostáva s niekoľkých krokov: modelovanie (Solid Works), NC programovanie (ToPs), razenie, ohýbanie atď. Príspevok pojednáva o osvojení si známych inžinierskych riešení CAD, CAM a CAPP z CIM. Za účelom eliminácie chýb pri ohýbaní sa musia previesť presné výpočty prieniku razníka a vôle ohýbania. Parametre ohýbania musia byť správne (priame ohyby, presné tvary, presné uhly) a výpočet vzdialenosť razníka sa vykonáva s ohľadom na ohýbanie. Toto je obzvlášť dôležité pre diely s dierami alebo žľabmi, ktorá musia byť umiestnené po ohýbaní presne. Článok objasňuje predpoveď s akceptovateľnou presnosťou vôle ohýbania, uhlom odpruženia a pohybom razníka pre získanie požadovaných rozmerov výrobku. Toto je dôležité pre vysokokvalitnú výrobu hlavne pri zostavách s nízkymi toleranciami, kde je vyžadujúce pri inštalácii presné lícovanie.

Kľúčové slová: oceľové plechy, ohýbanie, razenie, modelovanie

1 Introduction

The demands on the quality of bends and formed parts have increased considerably in recent years. Today a precision bend is within half or even a quarter of a degree. Making extra parts is not an option for a company that wants to be profitable. Sheet metal is simply metal formed into thin and flat pieces. It can be cut and bent into a variety of different shapes. Countless everyday objects are constructed of this material like control boards [1], computer housings etc. Thicknesses can vary significantly, although extremely thin thicknesses are considered foil or leaf, and pieces thicker than 6 mm are considered plate. Sheet metal is available as flat pieces or as a coiled strip. In presented work flat pieces ware used.

Most products around us are assemblages of components; some are relatively simple consisting of a few different parts joined by a selected means (figure 1).

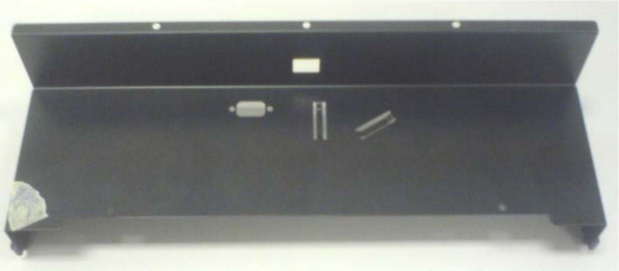


***Fig. 1** Final product access cover for electrical appliances*
***Obr. 1** Koncový produkt - kontrolný kryt pre elektrické zariadenia*

Others are more complex, these products normally use a greater number of parts of different sizes, shapes, and materials. Planning of forming production processes is an important engineering activity in a manufacturing system (figure 2). Wrong procedure can lead to a scrap tool and the production run can be very costly, much more than a test piece. The objective is not only to meet the quality and specification requirements, but also to reduce the production cost. Bottoming and coining allow us to produce high accuracies but with a high tooling cost, therefore hydraulic press brakes evolved to become dominant forming method in precision market.

Process planning is an act of preparing detailed work instructions to produce a part or to assemble a product. It establishes the sequence of the operations to convert a piece part from its initial form to the final form or an intermediate product into the final product. Report generation, storage, and retrieval of plans lead to CAPP (Computer-Aided Process Planning) systems. Previously prepared process plans are stored in a database. When a new component is planned, a process plan for a similar component is retrieved and subsequently modified by a process engineer to satisfy special requirements [2].

The electric cover was produced in *Alpron Company* for manufacture of sheet metal pieces and is a part of a Computer integrated manufacturing factory system. CIM factory system is made up of a part fabrication centre, a component assembly centre, and a product assembly centre. Centres are subdivided into work cells, cells into stations, and stations into processes. In optimal organization the computer hardware and software are essential and for the given product a part fabrication system is explained with modelling, punching and bending cell. These include designing, predicting, controlling, monitoring, planning, scheduling, ordering, changing, communicating and analyzing of the production process.

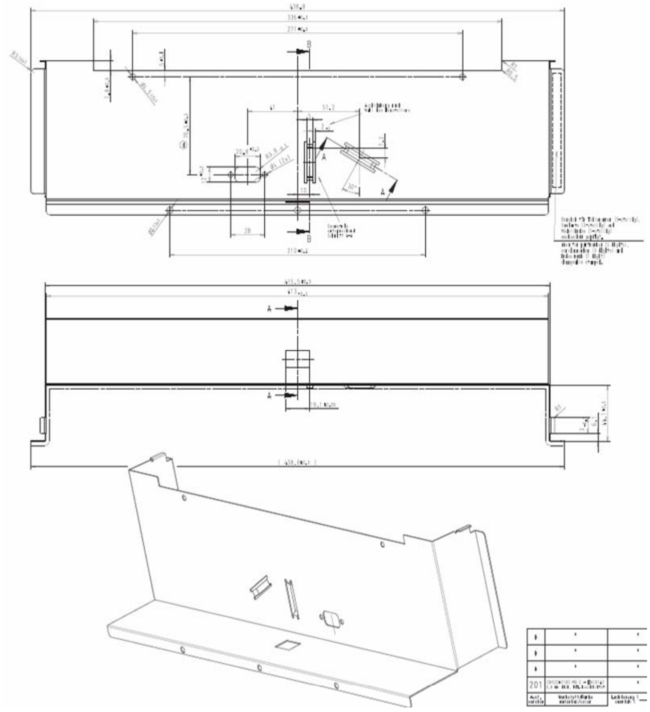


Fig. 2 *Visualization of product planning*
Obr. 2 *Vizualizácia plánovania produktu*

2 Modelling

The drawing of the sheet metal part serves as the basis for the NC program. Simple, flat parts are easy to create using the sheet metal design module of the *SolidWorks* software, figure 3.

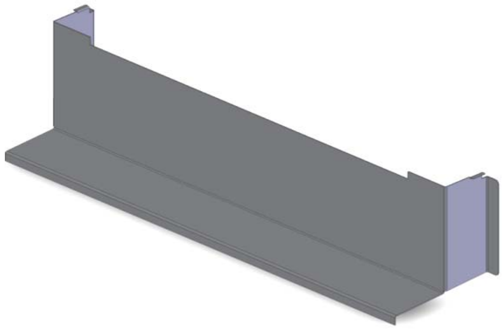


Fig. 3 *Model of the Final product in SolidWorks*
Obr. 3 *Model koncového produktu v SolidWorks*

Complicated sheet metal parts have multiple bends or bends that are at difficult angles. Bends are preformed in different angles and in four main directions up, down, to the right and to the left. *SolidWorks* offers visualization of the finished part and thus helping the operator. When a (3D) sheet metal part modelling is finished, the model is then unfolded in order to gain the initial blank needed for sequent operations. Part is then saved as a sheet metal pattern which produces a 2D drawing with bend lines and degrees. (figure 4) Part is then transferred in *ToPs* software in IGES format for punching operations. After punching operations are finished the bending operations take place. Different sequences are possible for producing the bend lines. The main goal of process planning is to determine an executable bend sequence and to select tools to use for each bend line and the punch displacements. Bend sequence can be researched and displayed with this software. Two-dimensional or three-dimensional views show the operator which bending operation needs to be performed and what happens to the part as it is bent. The bending sequence should ensure that all bending processes can be completed without destroying previous bends or causing collisions between the tool, workpiece, and machine.

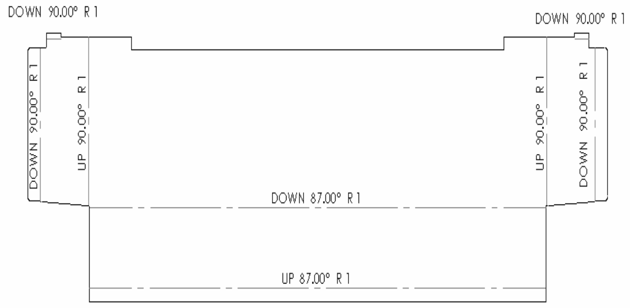
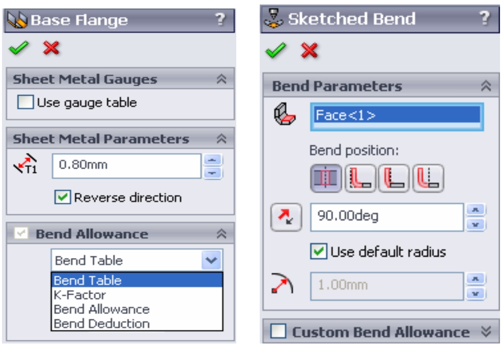


Fig. 4 *2D drawing with bend lines*
Obr. 4 *2D výkres s ohybovými čiarami*

SolidWorks also allows creation of complex products with conection of multiple parts such as closets and drawers [3]. An optimum bending sequence is the one that enables the fastest possible processing of the part. *SolidWorks* offers 3-D mechanical design software that helps engineers speed up product development. Suggested bending sequence depends on the tool shape, especially its maximum nest height, care should be taken for collision detection after each bend. *SolidWorks* allows planning of bend development as in figure 5.



Base Flange
Sheet Metal Gauges
☐ Use gauge table
Sheet Metal Parameters
0.80mm
☑ Reverse direction
Bend Allowance
Bend Table
Bend Table
K-Factor
Bend Allowance
Bend Deduction

Sketched Bend
Bend Parameters
Face<1>
Bend position:
90.00deg
☑ Use default radius
1.00mm
☐ Custom Bend Allowance

Fig. 5 *Selection of bend development in SolidWorks*
Obr. 5 *Výber parametrov ohýbania v SolidWorks*

3 Programming

The programming software has to make a lot of decisions when defining the machining operation. The first step in the manufacture of sheet metal parts is always flat processing, during which one or more parts are made from a sheet. The model was made in *SolidWorks* than transferred in IGES format into *ToPs* software. The programmer uses the programming software to specify the location of each part on the sheet. This process is known as “nesting” and for given product *ToPs* software was used. Then the programmer indicates the required quantity for each part. The nesting program then calculates the optimum sheet layout, taking into account manufacturing specifications such as the distance between two parts. *ToPs* 300 [4] was used for creation of the NC program, this is a programming system for punching and combination of laser/punching machining. Import of 2D or 3D data is possible for IGES, DXF of MI standard interfaces. Program performs on automatic analyses of the drawing, it closes open contours, deletes superimposed drawing elements and smoothens contours transition, clarifies inner and outer contours, defines single hole processing, it also defines special tools. Suction cups and stacking of parts is also done with *ToPs* as the loading operations and sheet skeleton removal. Optimal nesting selection of sheet parts was also done with the *ToPs* optimal sheet utilisation. The choice is to divide parts on sheet into sectors of optimal size in rows, columns etc. in order to nest a maximum number of parts with minimum material waste.

For Bending important factors for calculation of tool selection are the tensile strength of the material, the bend length, the bent angle, and the tool dimensions. With knowledge of the tonnage, the programmer or operator can determine whether the press capacity of the press brake will be sufficient and which tools will be able to withstand the loads produced by the bending operation. The die width, however, affects the bend radius. So it is not only the question of tonnage. The programmer specifies the type of material, dimensions (of the tool, the die and the workpiece), and bending specifications. The programmer then retrieves the remaining data on the material and tool properties from a database, suggests tools that can be used, and calculates the press force or tonnage. The parameters obtained in this way are optimally configured to the machine and tools. The programmer can use various software's in order to calculate the bending sequence. A bending plan and simulation allows the programmer to check that everything works smoothly.

4 Used technologies

Stamping includes a variety of operations, [5] such as punching, blanking, embossing, bending, flanging, and coining; simple or complex shapes formed at high production rates. Punching is performed by moving the sheet of metal between the top tool (punch) and bottom tool (die) they mate and a simple shape is cut from the sheet. An area can be cut out by making several hundred small (e.g. a square, circle, or hexagon) cuts around the perimeter. A punch is less flexible than a laser for cutting compound shapes, but faster for repetitive shapes. A typical CNC punch (figure 6) has a choice of up to 60 tools in a "turret" that can be rotated to bring any tool to the active punching position. A modern CNC punch can take 600 blows per minute. In figure 7 are shown punching specifications for given example. Notch and hole (punch) are templates used to cut and relive sheet metal walls. Industry standards place notches on edges and punches in the middle of the sheet metal wall. Hole position sometimes cannot be held to tolerance when a workpiece is pierced before bending. Holes made before bending are likely to be displaced during bending. Whether or not this displacement can be tolerated must be carefully considered in planning of the operation sequence.

Fig. 6 *Amada air bending and Trumpf punching machine*
Obr. 6 *Pneumatická ohýbačka Amada a raziaci stroj Trumpf*

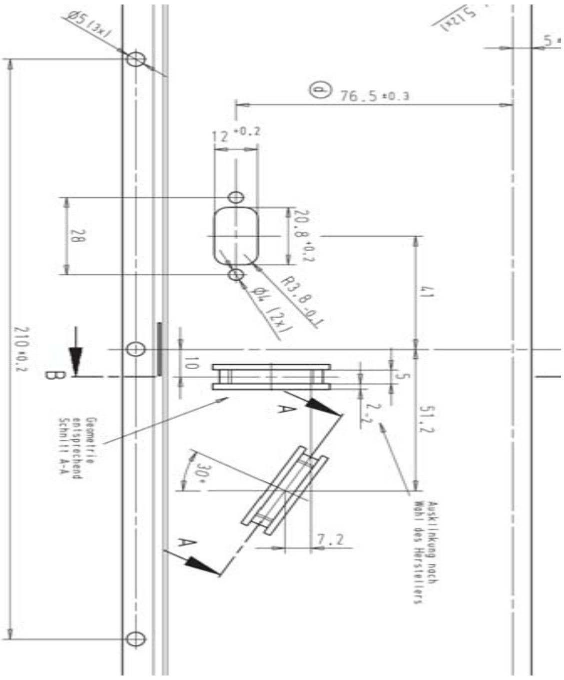


Fig. 7 *Visualization of punching positions*
Obr. 7 *Vizualizácia razných pozícií*

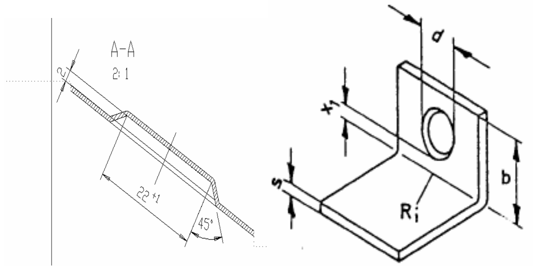


Fig. 8 *Specific punch requirement*
Obr. 8 Špecifické požiadavky na razenie

Minimum distance and length must be specified from the bending line if the hole or notch is to be maintained [6]. Distance is calculated (figure 8) with following formulas [7]:

$$x_1 = \sqrt{d \cdot s} + 0.8R_i \cdot \sqrt{\frac{b}{d}}\,, \tag{1}$$

$$x_2 = 1.1 \cdot \sqrt{d \cdot s} + 0.8R_i \cdot \sqrt{\frac{b}{d}}\,. \tag{2}$$

Where x_1 is minimum distance from the bend line for hole (figure 8) (1) and x_2 is for notch [mm] (2), d – hole diameter or width of the notch [mm], s – sheet thickness [mm], Ri – inner radius of the bend [mm], b - side length [mm]. Important is to control raw material utilization trough *ToPs* software when punching and to create an integrated data processing system. Setup plan for punching is then created and conducted on the *Trumpf* punching *Trumatic 200* machine. In figure 9 the final product with holes and bend is shown. Further optimisation can be reached by the optimisation of punching paths of a given setup plan [11].

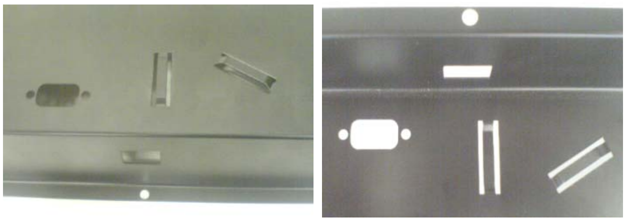


Fig. 9 *Realized punching back and front*
Obr. 9 *Realizované razenie predok a zadok*

Bend Allowance

Bend Table

SCHIAVI

☑ Auto Relief

Tear

***Fig. 10** Bend parameters, SCHIAVI AMADA bend table*
***Obr. 10** Parametre ohýbania, ohýbacia tabuľa SCHIAVI AMADA*

Bending is conduced on a CNC press of Amada Company (figure 6) where worker holds and turns the part. In the beginning of the modelling process bending table of Amada machine is inserted into the Sheet metal modul of *Solid Works* software, figure 10. Bending procedure is executed by the CNC that governs the boundary conditions.

Maximum bending force is:

$$F = \frac{LT^2(\sigma_M)}{W}, \tag{3}$$

where

L – length of sheet metal, mm.
T – sheet metal thickness, mm.
σ_M – boundary tensile stress of material, N/mm^2.
W – opening width, mm.
F – force, N.

The variable W is opening width (4) of a V-die or Wiping die.

$$W=(6 \text{ to } 10)\cdot s\,. \tag{4}$$

In presented example sheet thicknes of 0.8 mm and therefore W= 6·0.8= 4.8 mm.

$$F = \frac{1{,}33 \cdot l \cdot R_m \cdot s^2}{W - (2 \cdot \cos 45° \cdot r_{OW})}\,. \tag{5}$$

Bending force (F=52 kN/m) was calculated with Trumpf standard formula (5) shown above [7]. Where 1.33 - is friction between material and lower tool, l - bend lenght, R_m material thensile strenght, s – sheet thickness, W – width die, r_{OW} - radius of the upper tool.The width, angle, height, and working radius of the die; the radius, angle, and height of the punch; and the bend's inside and outside radii all determine which tools can be used. Press force can be also calculated with Trumpf bending slide or with press force table.

Air bending is a standard technique for producing angles from 30 to 179 degrees, (figure 11).

Often used when angle variation of one or two degrees is satisfactory. In precision air bending the punch is positioned to within ±0.01 mm or less than a die. No constant bending radius is formed with air bending, but rather a curvature line. Air bending is programed by y axis position; it is a travel-dependent bending method. In air bending the punch is not lowered completely but to a pre-calculated position (punch penetration). In this way it is possible to produce bends with different angles with one set of tools. Springback is then addressed by selection of different values of the punch penetration. The amount of spring back depends on the material, thickness, grain and temper. The spring back usually ranges from 5 to 10 degrees. Usually same angle is used in both the punch and the die to minimize setup time. The inner radius of the bend is the same as the radius on the punch; the smallest curvature is in the bending apex. Metal can bend only to a certain angle before the appearance of cracks and, finally, the broken part. In order to predict the minimum reachable radius the following equation (6) is used:

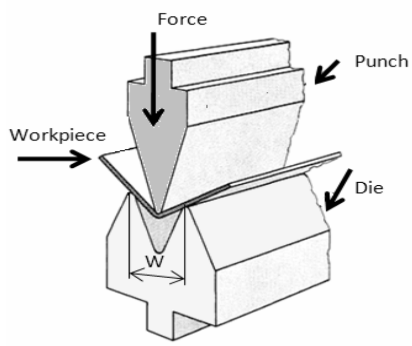


***Fig. 11** Air bending punch and die*
***Obr. 11** Pneumatické ohýbanie- razník a raznica*

$$R_{\min} = t \cdot \left(\frac{50}{r} - 1\right), \tag{6}$$

t - thickness of sheet,
$R_{\min}$ - minimum bending radius,
r - reduction in area in tensile test for a given material (%).

When bends must be precisely defined for critical thin sheet metal inside or outside radii it is better to use bottom bending / coining techniques.

In air bending the punch radius can also be smaller than the sheet metal radius, this is called the ''3 - point bending''. The sheet can be wrapped around the tool nose but a large sheet section remains free and has no contact with the tool or die. The bent angle goes into free position with air bending. Variations in material and sheet thickness can lead to errors in bend angle or incorrect calculation of bend allowance. In simple model where sheet [8] wraps around the punch nose and die shoulders remain strait (figure 12), relation is written as:

$$Z = \frac{W \tan(\frac{1}{2}\beta)}{2} + \left[1 - \cos(\frac{1}{2}\beta) - \tan(\frac{1}{2}\beta)\sin(\frac{1}{2}\beta)\right](R_p + R_d + s) \tag{7}$$

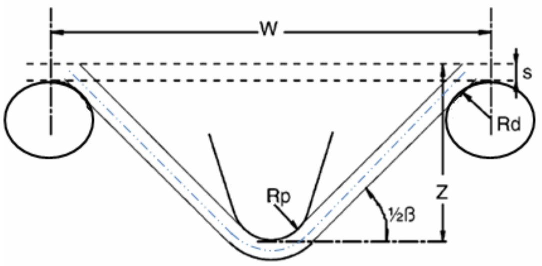


***Fig. 12** Sheet wraps around the punch nose* [4]
***Obr. 12 Z**vinutie plechu okolo nosa razníka*

One problem with this is that a plastic deformation happens between the punch nose and die shoulder, the second is the need to assume the wrap-around. There are three stages of sheet curvatures [8]: 1. Angle obtained in initial stage, 2. Wraparound occurs, 3. Curving has stopped and wraparound hasn't started yet. Simple models can result in overbending (especially with no wrap-around). Wrap-around starts with the inner sheet curvature and punch nose curvature are equalized. Small die opening wrap-around is more likely to occur and thus the tendency to select small die openings. The thumb rules are accurate but for a limited number of angles usually only 90˚ which is a serious limitation when high quality is required.

Another way for manufacture of precision sheet metal is bottom bending or coining. In bottoming with penetration or coining the punch presses the workpiece completely into the lower die, so that the punch, workpiece, and die are sandwiched together. This also means that a different tool set is required for each angle and shape. The nose of the punch is pushed into the die have and they have to fit together exactly, the punch is unable to move down any farther. However tonnage pressure continues to increase until the specified value is attained. Tonnage can be 10 times greater than with air bending and the pressure can damage the tools. With nose penetration plastic flow occurs and material solidifies negating the springback effect stabilizing the required angle.

When creating multiple bends in a part (figure 13, 14), the effect of springback is ordinarily cumulative and may necessitate closer control of the operation than would be needed for just one bend. This is especially noticeable in assemblies where wrong bend parameters can cause an error up to 3 mm and more in the assembly process.

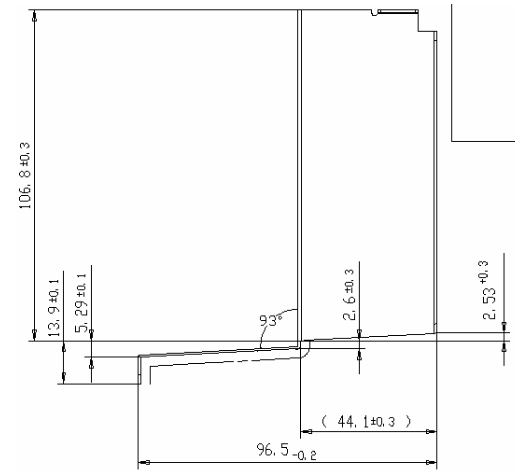
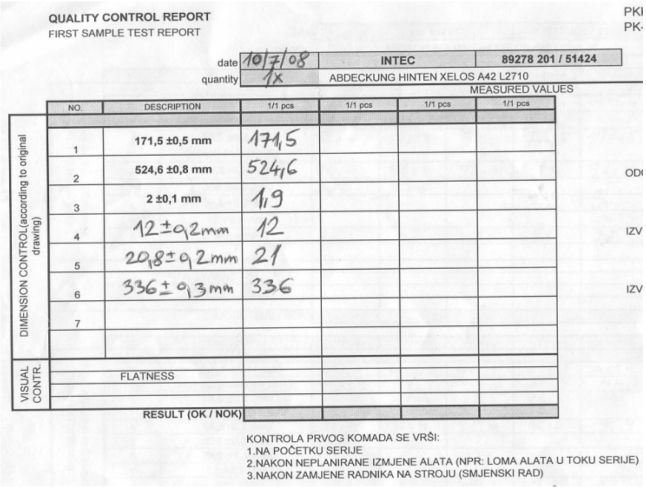


Fig. 13 *Critical bending tolerances*
Obr. 13 *Kritické tolerancie ohybu*

Fig. 14 *Finished part a view of important tolerances*
Obr. 14 *Finálny produkt – pohľad na dôležité tolerancie*

Variation in springback in a part with more than one bend can cause dimensions to reduce or to increase. Problems happen when all parts have to be finished parts, starting with the very first one and the openings have to be aligned perfectly after multiple bends are made. When several edges meet and will be joined together at a later stage. When several single parts are to be joined together in a component and it is no longer possible to compensate for deviations manually.

Problem was to create a 93˚ angle for the part that will be joined in assembly in a latter stage. During production critical bending tolerances ware monitored (figure 13). There ware 6 critical bending tolerances that greatly affected the finished part. Specifications requested minimum tolerances of ±0.1 mm for certain dimensions, (figure 15) and no error margin for the 93˚ angle. One of the things to consider was also painting of the final product, which increased the total thickness of the part and its dimensions. For control purposes during the modelling of the part in SolidWork, a layer of paint was added. This showed that final dimensions after bending will be wrong, therefore an error margin was introduced into modelling. The sheet coil was selected according to this analysis result. After process of punching and bending the final dimensions ware in accordance to the specified tolerances. Quality control method that was used was FMEA (Failure Mode and Effect Analysis) for noticing, removing and monitoring of errors in the process. Quality control results ware then sent to the main CIM assembly centre.



QUALITY CONTROL REPORT
FIRST SAMPLE TEST REPORT

PKI
PK-

date 10/7/08 | INTEC | 89278 201 / 51424
quantity 1x | ABDECKUNG HINTEN XELOS A42 L2710

MEASURED VALUES

	NO.	DESCRIPTION	1/1 pcs	1/1 pcs	1/1 pcs	1/1 pcs
DIMENSION CONTROL(according to original drawing)	1	171,5 ±0,5 mm	171,5			
	2	524,6 ±0,8 mm	524,6			
	3	2 ±0,1 mm	1,9			
	4	12±0,2mm	12			
	5	20,8±0,2mm	21			
	6	336±0,3mm	336			
	7					
VISUAL CONTR.		FLATNESS				
		RESULT (OK / NOK)				

KONTROLA PRVOG KOMADA SE VRŠI:
1.NA POČETKU SERIJE
2.NAKON NEPLANIRANE IZMJENE ALATA (NPR: LOMA ALATA U TOKU SERIJE)
3.NAKON ZAMJENE RADNIKA NA STROJU (SMJENSKI RAD)

Fig. 15 *Important bending tolerances quality report*
Obr. 15 *Kvalitatívna správa o dôležitých toleranciách ohybu*

When solving the bend development we have to consider that a sheet is bent in a press brake and the region of the sheet close to and in touch with the punch elongates. The solution is to predict the initial flat length. Bend stretching and compressing creates changes on the workpiece. To ensure that the finished part has the right dimensions, the design engineer has to compensate for the change in size. While most bend developments can be predicted and will develop correctly, there is no perfectly scientific method for predicting bend allowance. Total allowance is represented with ''Bend Development'' where the sum of all allowances is represented as in figure x. Bend allowance describes how much material is needed between two panels to accommodate a given bend. Many companies will develop their bend allowances based on standard formulas, standard forming practices and historical trial and error.

There are several methods for calculation of bend development (figure 16):

1. Bend table [2],
2. Bend allowance,
3. K- Factor,
4. Bend deduction,
5. Gauge tables.

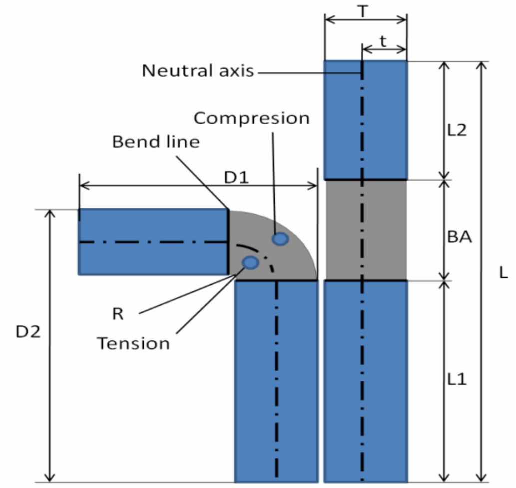


Fig. 16 *Bend allowance and bend deduction*
Obr. 16 *Vôľa ohybu a záverečný ohyb*

L = L1+L2+BA, (8)
L= D1+D2-BD, (9)
BA= 2P ·A(R+KT)/360, (10)
where:
K= Neutral axis offset t/T (K factor),
T= Thickness of material,
BA= Length of Bend allowance,
BD= Length of Bend deduction,
P= 3.14,
A= Angle (degrees).

When modelling an assembly it is important to use uniform system of defining the angles, especially if different engineers are involved in the process. And to take in consideration the production capabilities of used machines.

When developing a flat blank length, there is a length of the part that does not change. Looking at the cross section of the bend, the neutral axis is the theoretical location at which the material is neither compressed nor stretched. Material on the inside of the neutral axis will compress, while material on the outside will stretch. Where the neutral axis is situated in a bend is commonly called the "K-Factor". It is measured as the distance from the inside of the material to the neutral axis divided by the material thickness. Since the inside compression can not exceed the outside tension, the k-factor can never exceed 50 % in practical use. This means that the neutral axis cannot migrate past the midpoint of the material (i.e. towards the outside). A reasonable assumption is that the k-factor cannot be less than 25 %. Resulting neutral axis is based on the material thickness, form radius and forming methods, the ratio of compression to tension in the part. Sheet metal thickness gages [9] are based on a weight of 41.82 pounds (18.97 kg) per square foot (0.3048 m) per inch (0.0254 m) of thickness. This is known as the Manufacturers' Standard Gage for Sheet Steel, and is primarily used for sheet steel.

When a piece of metal is being bent the outside of the bend stretches so that it is under tension. It creates an imbalance of stresses that causes the bend to "spring back" slightly when the punch is raised again, thereby changing the angle (figure 17). The inside of the bend is compressed. Because all materials have a finite modulus of elasticity, plastic deformation is followed by elastic recovery upon removal of the load. In bending, the recovery is known as springback. Influenced not only by the tensile and yield strengths, but also by thickness, bend radius and bend angle. The obvious way to counter springback is simply to over bend the piece to a smaller angle and let it open up to the desired angle.

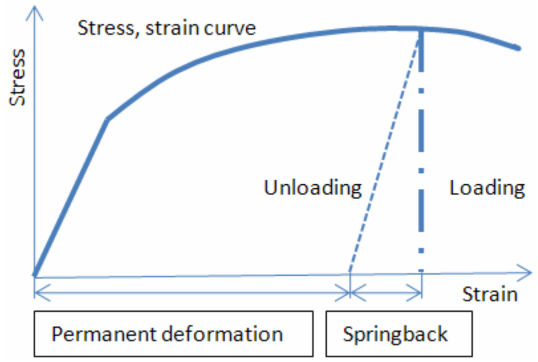


Fig. 17 *Springback forming diagram*
Obr. 17 *Formovací diagram odpruženia*

Calculating springback:

After a part is bent, it will springback to some degree, depending on its geometry and material properties. The V-bend springback prediction equation is:

$$\frac{R_i}{R_f} = 4\cdot\left(\frac{R_i S_y}{ET}\right)^3 - 3\cdot\left(\frac{R_i S_y}{ET}\right)+1, \qquad (11)$$

where:
Rf: bend radius after springback,
Ri: bend radius before springback,
Sy: yield strength of the material,
E: modulus of material elasticity,
T: thickness of the material, strip thickness.

One of encountered problems is the thickness of the product and that is 0.8 mm that makes handling of the product sensitive. This can lead to errors that are out of the planned process.

5 Conclusions

Computer-aided design (CAD) was used for converting the initial idea of a product into a detailed engineering design in order to communicate design information. The electro-galvanized zinc sheet metal of thickness of 0.8 mm was treated to desired shape. Then the electrostatic painting procedure was conducted with a 30 microns deposit of polyester paint for inner conditions. Computer-aided engineering (CAE) was used in order to conduct optimisation analyses on the model such as bend allowance, material behaviour, spring-back etc. in order to select the optimal values for design parameters. Prior to the manufacturing process was simulated with *SolidWorks* and *ToPs* to visualize the manufacturing process. Computer-aided manufacturing (CAM) was used for converting engineering designs into finished products, creation of a process plans and elimination of possible bend errors. CAD, CAE and CAM capable software's *SolidWorks* and *ToPs* ware used to eliminate redundant design, increase the efficiency of equipment, reduce waste and scrap, decrease the time required to design and make a product, and improve the ability of the factory to pro-duce high quality products.

Marko Kršulja, Eng., MsC
Mauro Štefančić, Eng., MsC
Branimir Barišić, Eng., PhD
Technical University of Rijeka, Technical Faculty,
Vukovarska 58, 51000 Rijeka, Croatia,
tel.: +385-51-651472, fax: +385-51-651468
e-mail: mkrsulja@riteh.hr, barisic@riteh.hr,
mauro.stefancic@alpron.hr

References

[1] Pauletić N., Barišić B., Kršulja M.: Controll board for Trumph presses manufacturing by metal forming application, *Engineering Review*, Volume 27, Issue 2, page 103-111, 2007.

[2] C. Leondes: *The Design of Manufacturing Systems*, CAD, CAE, CAM Encyc. - Vol 5 (ed) (CRC, 2001) WW.

[3] Mijić S., Barišić B., Kudlaček J.: Technology of metal drawer manufacture by implemention of CAD-CAM connection, *Engineering Review*, Volume 28, Issue 1, page 73-83, 2008.

[4] *ToPs 300 - Programming system*, TRUMPF Werkzeugmachinen, Ditzingen, 2005.

[5] *Punching,* Publication date: 10/2006 Publisher and Editor: TRUMPF Werkzeugmaschinen GmbH + Co. KG www.trumpf.com/3.library_start_v1.html.

[6] Joseph R. Davis, S. L. Semiatin, American Society for Metals: *ASM Metals Handbook, Vol. 14: Forming and Forging*, by ASM International; 9 edition (April 1989)

[7] *Bending Technologie,* (2006) Publisher and Editor: TRUMPF GmbH + Co. KG, www.trumpf.com/3.library_start_v1.html.

[8] Leo J. De Vin: Expecting the unexpected, a must for accurate brakeforming, *Journal of Materials Processing Technology*, 117(2001) 244-248.

[9] Engineering fundamentals, (2009) www.efunda.com/designstandards/gages/sheet_forward.cfm?search_string=gauge.

[10] M. Pan, Y. Rao: *An integrated knowledge based system for sheet metal cutting–punching combination processing*, Knowledge-Based Systems, 2009.

Pokračovanie príspevku zo str. 32/
Continuance Papers from Page 32

špičky. Poznamenejme, že vliv poloměru špičky na složky řezné síly by si zasloužil větší pozornost. To by mělo být zohledněno v budoucích výzkumných pracích orientovaných na monitorizaci dokončovacích operací.

Pro obrábění při hloubce řezu a_p = 0,8 mm lze opět pozorovat pokles hodnot F_c/F_f při změně z ostrého stavu VBD na dobrý stav VBD. Za povšimnutí stojí skutečnost, že v případě obou k obrábění vybraných materiálů řezná síla F_c poklesla při zhoršení stavu ostří VBD. Poměr složek řezných sil F_c/F_f přesto poklesl, což potvrzuje původní domněnku, že posuvová složka řezné síly F_f se zvětšujícím se opotřebením roste rychleji. V případě změny z dobrého stavu VBD do stavu špatného pro nástrojovou ocel ČSN 19 810 následuje další pokles poměru řezných sil F_c/F_f, kdežto pro případ konstrukční oceli ČSN 11 600 poměr F_c/F_f vzrostl. Dané chování nelze jednoznačně objasnit a zasluhovalo by další výzkum. Snad by se v daném případě mohlo jednat o vliv nárůstku.

Pro případ obrábění při hloubce řezu a_p = 1,2 mm následuje pokles hodnot F_c/F_f pro přechod ze stavu ostré VBD do stavu dobré VBD a dále pokles pokračuje i při přechodu do stavu špatné VBD. Pokles je výraznější pro nástrojovou ocel ČSN 19 810.

6 Závěr

Na základě provedených experimentálních měření bylo zjištěno a ve většině případů potvrzeno, že posuvová složka výsledné řezné síly je k opotřebení řezného nástroje pro případ podélného soustružení citlivější než hlavní složka, tj. řezná síla.

Zjištěné chování poměru složek řezných sil F_c/F_f přivádí k optimistickému závěru. Lze předpokládat, že pro případ monitorizace podélného soustružení by bylo možné tuto hodnotu použít jako informační parametr.

Přes optimistické výsledky této dílčí etapy řešeného projektu bude potřeba provést další výzkumné práce s cílem upřesnit a lépe pochopit chování řezných sil v závislosti na opotřebení břitu pro další širokou paletu obráběných materiálů a řezných podmínek.

Poznámka:
Tento článek souvisí s řešením projektu MSM 4674788501, který je finančně podporován MŠMT ČR.

Prof. Ing. Jan Žižka, CSc.,
Ing. Tomáš Linhart,
Katedra obrábění a montáže, FS
Technická univerzita v Liberci
Studentská 2, 461 17 Liberec, ČR
Tel: ++420 485 353 367
E-mail: jan.zizka@tul.cz

Literatura

[1] KLAUS, V.: *Politici v době krize jen zvyšují paniku.* Mladá fronta DNES. 5. března 2009, s. A9.

[2] MORIWAKI, T.: Intelligent Machine Tool: Perspective and Thesis for Future Development. In: *Manufacturing Science and Engineering.* New York: ASME, 1994, Vol. 2, p. 841-849.

[3] ŽIŽKA, J.: *Použití akustické emise ke sledování procesu obrábění.* Liberec: TU v Liberci, 2003. 106 s. ISBN 80-7083-687-1.

[4] CHILD, T.H.C., MAEKAWA, K., OBIKAWA, T., and YAMANE Y.: *Metal Machining - Theory and Applications.* London: Arnold, 2000. 408 p. ISBN 0-340-69159-X .