

UDK 658.52.011:621.771:004.925.8

TEHNOLOGIJA IZRADE METALNE LADICE PRIMJENOM CAD-CAM POVEZIVANJA

TECHNOLOGY OF METAL DRAWER MANUFACTURE BY IMPLEMENTATION OF CAD-CAM CONNECTION

Saša MIJIĆ – Branimir BARIŠIĆ – Jan KUDLAČEK

Sažetak: Rad prikazuje tehnologiju izrade metalne ladice (materijala DC01 prema EN 10130:1991) od konstruiranja uz upotrebu računala i konačno do slanja CNC programa na strojeve za deformiranje i izradu ladice tj. primjenom CAD-CAM povezivanja. Korištenjem prikazanog CAD-CAM pristupa u proizvodnji ladice, proizvodnja se može optimizirati i lako kontrolirati, a sve u cilju brže, kvalitetnije i efikasnije proizvodnje.

Ključne riječi: – oblikovanje lima
– metalna ladica
– modeliranje

Abstract: The paper presents the technology for metal drawer manufacture (material DC01 according to EN 10130:1991) from construction by usage of computer, and finally to sending of the CNC program on machines for metal forming and drawer construction, i.e. by implementation of the CAD – CAM connection. With the use of the presented CAD – CAM approach in the production of a drawer, production can be optimized and easily controlled, with the goal of a faster and more efficient production of a higher quality.

Keywords: – sheet metal forming
– metal drawer
– modeling

1. UVOD

Današnji CNC programi izrađuju se poluautomatski ili automatski, korištenjem softvera za programiranje koji sadrže precizne informacije o stroju i alatima [1]. Programer odabire obradnu strategiju, izabire alate, i optimizira parametre te definira softveru za programiranje u cilju generiranja CNC programa. Softver za programiranje automatski određuje optimalne obradne nizove i proizvodi CNC programe bez pogreške. Tijekom proizvodnje svaki dio mora proći kroz nekoliko stanica, limovi se probijaju, ili se koriste laseri za izrezivanje, grickaju se, oblikuju, plitko vuku i savijaju na strojevima za savijanje. Brza i rentabilna proizvodnja znači izradu konačnog oblika u što je manje koraka, a korištenjem optimiziranog modeliranja povezanog s CNC programiranjem, te pravilnim definiranjem tehnologija oblikovanja moguće je postići traženo [2]. Dakle pri izradi zadane metalne ladice bit će prikazan pristup naprednog modeliranja pomoću *SolidWorks* [3] programa s konačnom svrhom izrade razvijenog oblika limene ladice, da bi se napravio CAD oblik pogodan za stvaranje CNC koda za daljnju proizvodnju. Konačni proizvod koji inženjer izrađuje na zaslonu je trodimenzionalan, dok je lim koji će se koristiti za izradu proizvoda ravan, pa se

1. INTRODUCTION

Today, CNC programs are created automatically or half-automatically, by the use of programming software which contains precise information's about machines and tools [1]. The programmer selects the work strategy, chooses the tools, optimises the parameters and defines the programming software with the goal of CNC code generation. Software for automatic programming determines the optimal working sequence and produces CNC programs without mistakes. During production, every part has to go through several stations, sheet metal is punched, or cutting lasers are used, nibbled, shaped, shallow drawn and bended on the bending machines. Fast and efficient production means the creation of a final shape in the smallest amount of steps, by using optimal modelling combined with CNC programming, and with the correct definition of forming technologies it is possible to obtain the requirements [2]. In the manufacture of the given metal drawer, the approach of advanced modelling with the use of the *SolidWorks* [3] program will be presented with the final purpose of creating the developed shape of the sheet metal drawer, in order to create a CAD code for further production. The final product that the engineer creates on the screen is

stoga mora početi s „razvijanjem“ virtualnoga trodimenzionalnog dijela. Proces razvijanja ravnog dijela iz crteža gotovog proizvoda određuje kako treba biti odrezan ili probijen početni oblik lima. Rupe, oblikovani dijelovi, i crte savijanja također se razrađuju. Podaci se tada prenose u softver za programiranje. Na osnovi razvijenog dijela izrađuje se CNC program za sve strojeve koji se koriste za proizvodnju dijela i CNC program definira stroju koje operacije treba izvršiti. *ToPs* program bit će korišten u ovom radu za izradu CNC koda za obradu na strojevima zbog toga što znatnije skraćuje vrijeme obrade i omogućuje uštedu materijala.

2. MODELIRANJE LADICE U *SOLIDWORKS*U KORIŠTENJEM NAPREDNIH *SHEET METAL* FUNKCIJA

Najbrži i najefikasniji način modeliranja u *SolidWorks*u je korištenjem *Sheet Metal* funkcija, dok se konstruiranje *Solida* te naknadno pretvaranje u *sheet metal* koristi samo u iznimnim slučajevima kao npr. kod koničnog savijanja. *SolidWorks* omogućuje izradu dijelova (*part*), njihovo sklapanje u jedinstveni sklop (*assembly*) te izradu crteža koji obično predstavljaju dijelove sklopa. Sklop ladice bit će napravljen od pet dijelova:

- 1) *Ladica_korito*,
- 2) *Ladica_leđa*,
- 3) *Ladica_pregrada*,
- 4) *Ladica_zadnja* i
- 5) *Ladica_prednja*.

Za svaki dio sklopa izradit će se i odgovarajući crtež, kao i za konačni sklop koji će se sastojati od navedenih dijelova. Glavni koraci u procesu modeliranja ladice dani su u sljedećem popisu:

- **Modeliranje alatima *Sheet Metal***

Koriste se alati koji su specifični za modeliranje dijelova od lima i to tipovi: *Flange*, *Tabs* i *Added Bends* te alati za razvijanje;

- **Korištenje alata za oblikovanje (*Forming Tools*)**

Koriste se alati za oblikovanje kako bi se oblikovao dio od lima. Alati mogu biti korišteni onakvi kakvi jesu, izmijenjeni, ili prepravljani od postojećih;

- **Crteži dijelova od lima**

Izrada crteža korištenjem dijelova od lima ima s nizom mogućnosti;

- **Korištenje simetrije**

Simetrični modeli mogu se izraditi kao djelomični modeli i zrcaljeni korištenjem opcije zrcaljenja modela (*Mirror Bodies*) itd.

Funkcija *Closed Corners* ili funkcija za približavanje rubova lima može produžiti jednu ili dvije rubne površine dovoljno da se naprave tri moguća načina izgleda rubova:

1. Približavanje unutarnjih rubova

three-dimensional, while the sheet metal that will be used for product creation is straight, so it has to begin with the “development” of a virtual three-dimensional part. The process of the development of a straight part from the drawing of a final product determines how the initial shape has to be cut or punched. Holes, shaped parts, and lines of bending are also elaborated. Information is then transferred into the programming software. On the basis of a developed part, a CNC program is created for all machines that are used in part production and the CNC defines to the machine the operations that it needs to execute. The *ToPs* program will be used in this work for the creation of a CNC code for manufacture on machines because of its feature of considerably shortening production time and allowing for material savings.

2. DRAWER MODELLING IN *SOLIDWORKS* BY THE USE OF ADVANCED *SHEET METAL* FUNCTIONS

The fastest and the most efficient method of modelling in *SolidWorks* is with the use of the *Sheet Metal* function, while construction of *Solid* and then afterwards transformation into sheet metal is used only in exceptional cases like, e.g., in cone bending.

SolidWorks enables part creation, joining of parts into one unique assembly and the creation of parts that usually represent parts of the assembly. Assembly of the drawer will be created from five parts:

- 1) *Drawer_bed*,
- 2) *Drawer_back*,
- 3) *Drawer_compartment*,
- 4) *Drawer_last* and
- 5) *Drawer_front*.

For every part of the assembly the correspondent drawing will also be created, and also for the final assembly that will be made from the mentioned parts. The main steps in the modelling of a drawer are given in the next list:

- **Modelling with *Sheet Metal* tools**

Tools are used that are specific for modelling of sheet metal parts and they are: *Flange*, *Tabs* and *Added Bends*, and tools for development;

- **Use of *Forming Tools***

Tools are used in order to form a piece of sheet metal. Tools can be used in the way they are, changed, or reconstructed from the existing;

- **Drawings of sheet metal parts**

Creation of sheet metal parts has a series of possibilities;

- **Use of symmetry**

Symmetric models can be constructed as a partial model that can be mirrored with the use of the option *Mirror Bodies* etc.

The function *Closed Corners*, or the function for the approach of sheet metal edges, can elongate one or two edge surfaces enough to create three types of edges appearance:

Unutarnji rub do unutarnjeg ruba (slika 1.a.). U tom slučaju obje su stranice produžene da bi se zbližile u zajedničkom rubu. To može biti vrlo korisno kada se koristi rubno zavarivanje na vanjskoj stranici i potrebno je minimalno omatanje.

2. Preklapanje ruba

U tom slučaju (slika 1.b.) jedna je stranica produžena na vanjsku površinu druge stranice i predstavlja potpuno preklapanje na vanjsku površinu.

3. Približavanje plus preklapanje

Predstavlja produžavanje unutarnjih površina s unutarnjim preklapanjem (slika 1.c.). U tom slučaju jedna je površina produžena prema unutrašnjosti druge stranice, a vanjska je površina produžena do vanjske površine druge stranice. Općenito, taj se tip rubova teško postiže standardnom funkcijom *Closed Corner*, ali može biti jednostavno modeliran. Za što efikasnije modeliranje bitno je na nacrtu uočiti profil koji je zajednički svim daljnjim savijanjima, polazišnu bazu s kojom se može riješiti najveći broj savijanja. U slučaju dijela *Ladica_prednja* to je okomita poprečna ravnina (slika 2).

1. Approaching of the inner edges

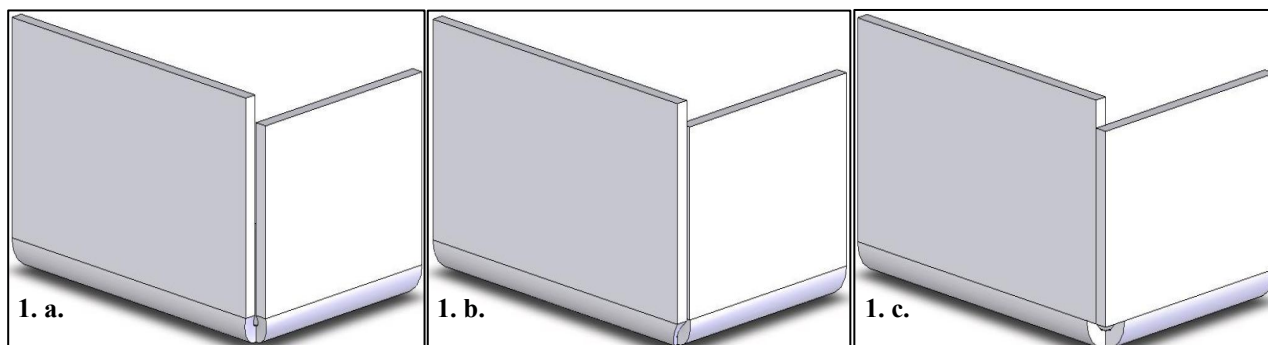
Inner edge to inner edge (Figure 1.a.). In this case, both sides are elongated in order to close in a joint edge. This can be useful when the edge welding is used on the outer side and a minimal envelopment is needed.

2. Edge overlapping

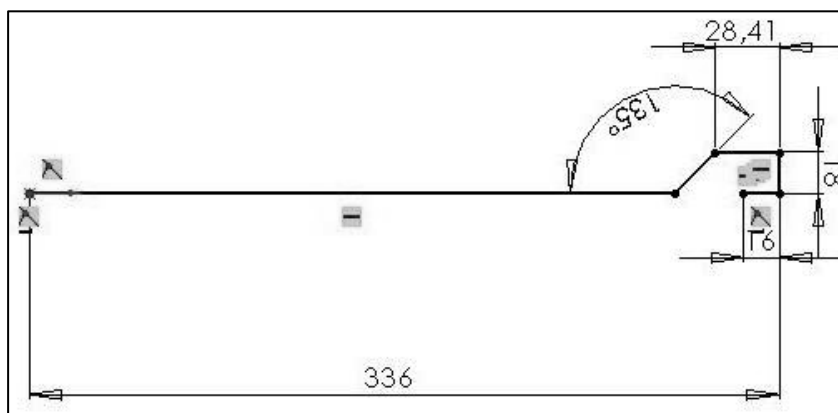
In this case (Figure 1.b.), one side is elongated on an outer surface of the second side and represents the total overlapping on the outer side.

3. Approach plus overlapping

Represents the elongation of the inner surfaces with the inner overlapping (Figure 1.c.). In this case, one surface is elongated towards the inner part of the second side, and the outer surface is elongated towards the outer surface of second side. Generally this type of edge is hard to achieve with standard the function *Closed Corner*, but it can be simply modelled. For more efficient modelling, it is important to notice on the drawing the profile that is mutual to all further bendings, the starting base with which the greatest number of bendings can be solved. In the case of the part *Drawer_front*, it is the vertical transverse plane (Figure 2).



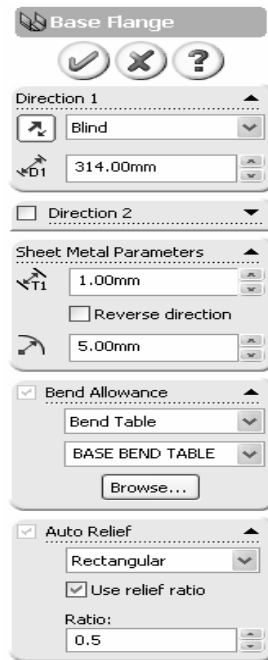
Slika 1. Tri moguća načina izgleda rubova
Figure 1. Three possible types of edges



Slika 2. Profil dijela *Ladica_prednja*
Figure 2. Profile of the part *Drawer_front*

Odabirom funkcije *Base Flange* stvorit će se taj profil u ravnini *Front*, jednostavnim alatima za skiciranje te će se kotirati zadanim dimenzijama. U *Base Flange* funkciji potrebno je definirati i 3D komponentu, kao i radijuse savijanja i ubaciti tablicu iz koje će *Solidworks* izvlačiti korektorne faktore (slika 3). Sva će savijanja biti automatski izvedena sa zadanim radijusima te je to i dokaz brzine pristupa. (Napomena: ovdje je rađena polovica izratka te je kasnije korištena *Mirror* funkcija, jer se time skraćuje vrijeme).

With the selection of the function *Base Flange*, this profile will be created in the plane *Front*, with simple tools for sketching, and it will be dimensioned with the given dimensions. In the *Base Flange* function, it is also necessary to define the 3D component and radius of bending and import tables from which *SolidWorks* will draw the corrective factors (Figure 3). All bendings will be automatically executed with a given radius, and this is also proof of the access speed. (Annotation: here half of the part was done, and later the *Mirror* function was used, because it shortens time).



Slika 3. Tablica korekturnih faktora
Figure 3. Table of corrective factors

Na slici 3 prikazane su opcije funkcije *Base Flange*. Pod poljem *Direction 1* odabiru se smjer izvlačenja profila i duljina izvlačenja. Pod poljem *Direction 2* je opcionalni smjer izvlačenja. Pod poljem *Sheet Metal Parameters* unosi se vrijednost debljine lima i radijus savijanja. Za polje *Bend Allowance* (dodanost savijanju) moguće je odabrati između K-faktora i tablice savijanja. Kada je lim savijen, da bi se razradilo koja je prava duljina tog dijela potrebna, treba izračunati dodanost savijanju (*bend allowance*) ili umanjenost savijanju (*bend deduction*) koje govore koliko treba dodati ili oduzeti dužine da bi se dobila tražena vrijednost. Jedini pravi djelotvorni način da se radi s točnom dodanosti savijanju jest reverzibilnim inženjeringom tako da se uzme uska traka materijala, savije se i tada se izmjeri da se izračuna dodanost savijanju. Dodanosti savijanju mogu biti izmjerene za mnoštvo materijala i scenarija i tada tabelirane, a tablice se zatim mogu koristiti u programu za izradu preciznog modela. Mnogi CAD programi ipak rade dodanosti savijanja automatski korištenjem proračuna K-faktora (ili

In Figure 3, the options of the *Base Flange* are shown. Under the field *Direction 1*, a direction of profile drawing is selected as well as the length of the draw. Under the field *Direction 2* is an optional direction of drawing. Under the field *Sheet Metal Parameters* is imported the value of the thickness of sheet metal and radius of bending. For the field *Bend Allowance*, it is possible to choose between the K-factor and the table of bending. When the sheet metal is bent, in order to elaborate which length is the real length of the part that is needed, it is necessary to calculate bend allowance or bend deduction, because they reveal how much it is necessary to give or take from the length in order to gain the required value. The only right and effective method of working with the exact bend allowance in bending is reverse engineering, by taking a narrow strip of material, bending it, and then calculating the bend allowance. These allowances for bending can be measured for multiple materials and sceneries and then put into tables for use of such tables in a program for the creation of a precise model. Many CAD

Y-faktora u slučaju Pro-E gdje je Y-faktor = K-faktor x $\pi/2$). Dodanost savijanju izračunava se korištenjem K-faktora kao što slijedi [1]:

$$BA = \pi \cdot (R + KT) \cdot A / 180 \quad (1)$$

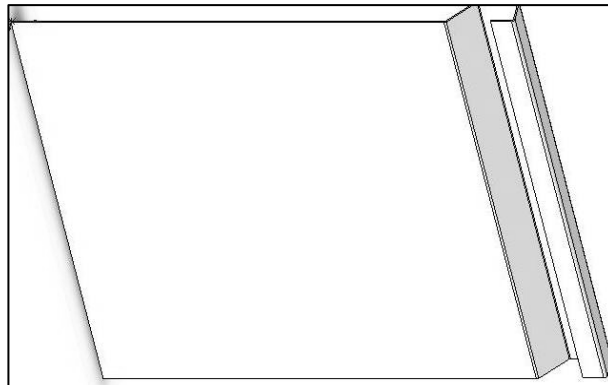
R – unutarnji radijus savijanja,
 K – K-faktor, koji je t/T ,
 T – debljina materijala,
 t – udaljenost unutrašnje strane do neutralne plohe,
 A – kut savijanja u stupnjevima (kut kroz koji je materijal savinut).

programs, however, create bend allowance automatically with the usage of K-factor calculus (or Y-factor in case of Pro-E where Y-factor = K-factor x $\pi/2$). Bend allowance is calculated with the usage of K-factor as follows [1]:

R – inner radius of bending,
 K – K-factor, which is t/T ,
 T – thickness of material,
 t – distance of the inner side to the neutral plane,
 A – Bend angle in degrees (angle through which the material is bended).

Slika 4 prikazuje izvučeni profil sa slike 2 i vrijednosti sa slike 3 s razlikom što je pod radijusom savijanja korištena vrijednost 1 mm.

Figure 4 shows the drawn profile from Figure 2, and values from Figure 3, with the difference that under a radius of bending, the value of 1 mm was used.



Slika 4. Gotov profil
 Figure 4. Finished profile

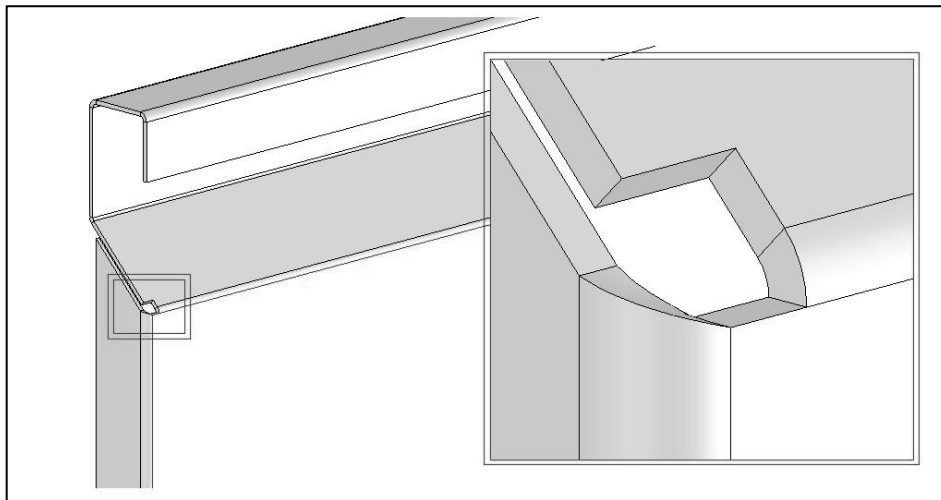
Tablica savijanja (*Bend Table*) specifična je tablica materijala koja sadrži proračune savijanja temeljene na debljini lima i radijusu savijanja, a formira se kao *Excelova* datoteka i ima *.xls datotečnu ekstenziju. Ta se tablica lako unosi u *SolidWorks* (odabire se *Insert -> Sheet Metal -> Bend Table -> From File*). Također se može odabrati iz *Use Bend Table* padajućeg izbornika u okviru *Sheet-Metal*. Tablica savijanja također se može napraviti korištenjem datoteka za primjer i *Excelsa*. Primjeri se mogu naći pod instalacijskim direktorijom *i: lang\<language name>\Sheetmetal Bend Tables*. Slika 5 prikazuje jednu takvu tablicu savijanja za vrijednost smanjenja savijanja gdje se vide vrijednosti umanjnosti savijanja pri različitim vrijednostima radijusa i debljine lima.

Bočne stranice izvedene su korištenjem *Edge Flange* funkcije koja je u mnogočemu slična *Base Flange* funkciji. Nacrta se skica savijenog ruba, odaberu se radijusi i tablice savijanja i rub je gotov. Potrebna popuštanja na mjestima gdje bi dolazilo do ulaska jednog dijela u drugi, rade se klasičnim *extrude/remove material* funkcijama (slika 6).

The bend table is a specific table of material that contains calculations of bending based on the thickness of sheet metal and the radius of bending, and it is formed as an Excel file and it has the *.xls file extension. This table is easily imported into *SolidWorks* (the sequence to select is *Insert-> Sheet metal-> Bend Table -> From File*). Also it can be selected from *Use Bend Table* from the falling menu in the frame of *Sheet-Metal*. The Bend Table can also be created with the use of files of examples and Excel. Examples can be found under the installation directory *i: lang\<language name>\Sheetmetal Bend Tables*. Figure 5 shows one such bend table for the value of the bend deduction, where the values of bend deduction can be seen with different values of radius and thickness of sheet metal. Flange sides are created with the use of the *Edge Flange* function that is in many ways similar to the *Base Flange* function. A drawing of a bended edge is sketched, the radii are selected, the bend tables and the edge are finished. The necessary releases in places where one part enters into another, are done with the classic *extrude/remove material* functions (Figure 6).

Microsoft Excel - BEND_TABLE.xls													
File Edit View Insert Format Tools Data Window Help													
Unit: Millimeters													
1	Unit:	Millimeters											
2	Type:	Bend Deduction											
3	Material:	Steel											
4	Comment:	Values specified are for 90-degree bends											
5	Radius	Thickness											
6		0,6	0,8	1,0	1,2	1,5	2,0	2,5	3,0	4,0			
7	1,0	1,3	1,6	1,9	2,3								
8	1,3	1,4	1,7	2	2,3	2,8							
9	1,6		1,8	2,1	2,4	2,9	3,7						
10	2,0			2,2	2,5	3	3,8	4,7					
11	2,6				2,7	3,2	4	4,8	5,7				
12	3,3					3,4	4,2	5	5,8	7,5			
13	4,0						4,5	5,2	6	7,7			
14	5,0							5,6	6,3	7,9			
15	6,5								6,8	8,4			
16	8,0									8,9			
17	10,0												
18	Comment:												
19	Comment:												
20													

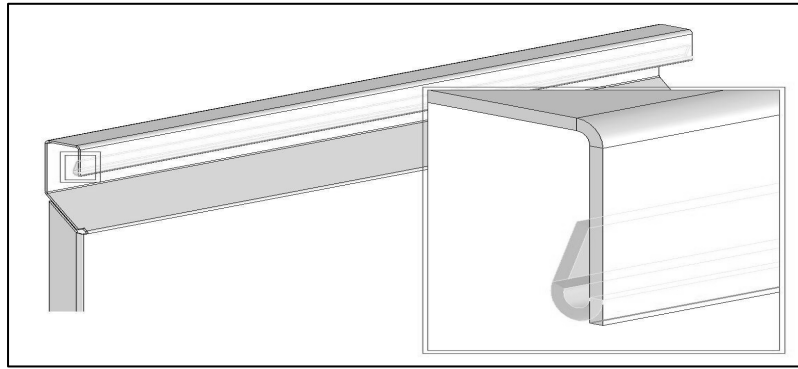
Slika 5. Tablica savijanja
Figure 5. Bend table



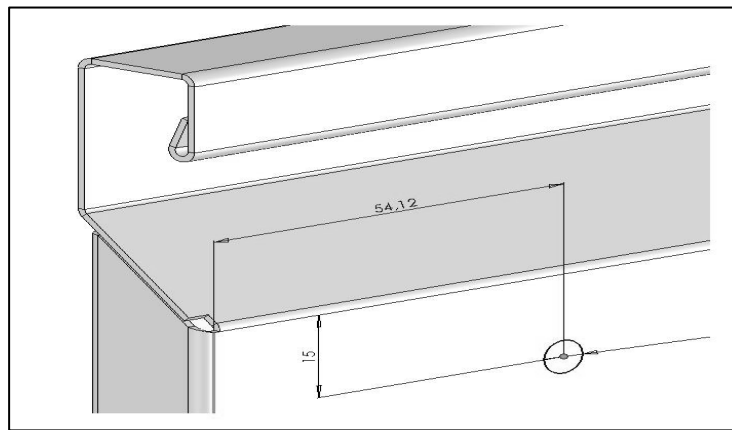
Slika 6. Primjer izreza
Figure 6. Representation of a cut-out

Savijanja je moguće izvesti korištenjem napredne *Miter Flange* funkcije, ali se sporije definira svako savijanje nakon *Base Flange Elementa*. Još jedna korištena funkcija kod izrade ladice je *Hem* (slika 7), kojom se definiraju završeci na limu, u različitim oblicima, a ovdje je korišten *Tear Drop* oblik. Na slici 8 prikazano je bušenje rupe na dijelu *Ladica_prednja*. Pozicija rupe lako se određuje korištenjem funkcije *Smart Dimension*.

It is possible to execute bending with the use of the advanced *Miter Flange* function, but every bending after the *Base Flange Element* is then defined with a slower speed. One more function used in the manufacture of the drawer is *Hem* (figure 7), with which the endings on the sheet metal are defined, in different shapes, and here the *Tear Drop* shape was used. In Figure 8, drilling of a hole on the part of the *Drawer_front* is shown. The position of the hole is easily selected with the function *Smart Dimension*.



Slika 7. Prikaz završetka korištenjem funkcije Hem
Figure 7. Representation of ending with the use of Hem function



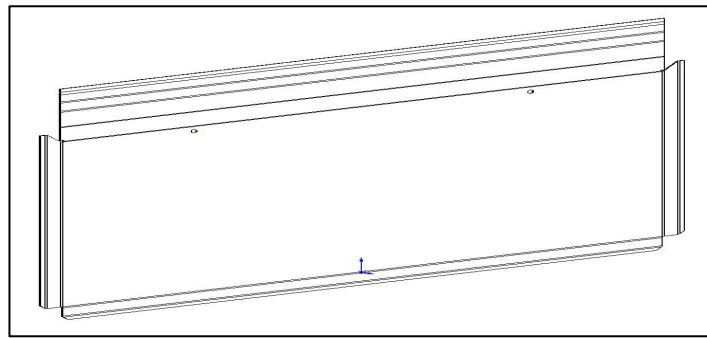
Slika 8. Prikaz završetka korištenjem funkcije Hem
Figure 8 Representation of ending with the Hem function

Pri modeliranju korištena je i funkcija *Miter Flange* kao što je to prikazano na slici 9. Funkcija *Miter Flange* koristi se kako bi se dodali rubovi koji spajaju i automatski kreiraju potrebne rubne rezove. Nakon zrcaljenja polovice ladice korištenjem funkcije *Mirror* napravljen je model do čijeg je razvijenog oblika vrlo jednostavno doći naredbom *Unsupress Flat Pattern* (slika 10) te je kao takav spremljen u IGES format i spreman za prijenos u ToPs program [4, 5].

In modelling, the function *Miter Flange* was used as it is shown in Figure 9. The *Miter Flange* function is used in order to add edges that connect and automatically create the necessary cuts. After mirroring half of the drawer with the use of *Mirror* function a model is created with which is very easy to arrive at the developed shape, with the order *Unsupress Flat Pattern* (Figure 10), and as such it is saved in IGES format and ready for transfer into the ToPs program [4, 5].



Slika 9. Prikaz korištenja funkcije Miter Flange
Figure 9. Representation of the use of Miter Flange function



Slika 10. Razvijeni oblik dijela Ladica_prednja
Figure 10. Unfolded shape of the part Drawer_front

Zadnja je faza u konstrukciji ladice stvaranje sklopa. To se radi tako da se odabere funkcija *Assembly* i umetnu se svi dijelovi ladice na način da se postave međusobne relacije. Sparivanje dijelova vrši se korištenjem funkcije *Mate* tako da se odaberu plohe, rubovi, ravnine i sl. koje se želi spariti. Standardne opcije sparivanja su:

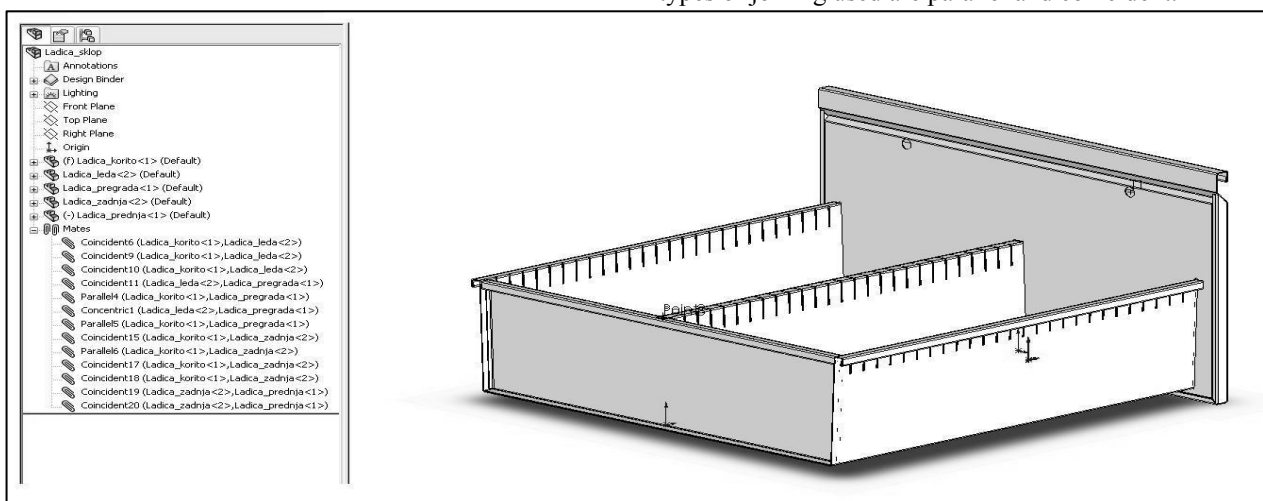
- Kut;
- Podudarnost;
- Koncentričnost;
- Udaljenost;
- Paralelnost;
- Pravokutnost i
- Tangencijalnost.

Sve su te vrste sparivanja uvijek prikazane pod *PropertyMenagerom*, ali samo one vrste sparivanja koje su primjenjive na trenutni odabir. Na slici 11 prikazan je sklop koji sadrži sve dijelove ladice. Na lijevoj strani prikazana su korištena sparivanja. Budući da se radi o dijelovima ladice koji imaju većinom ravne površine, to su uglavnom oslanjajuće površine, te su stoga korištene vrste sparivanja paralelnost i podudarnost.

The last phase in drawer construction is assembly creation. The method in which this is done is that the *Assembly* function is selected and all parts of the drawer are inserted in such a way that mutual relations are configured. The joining of parts is executed with the use of the function *Mate* by selecting the surfaces, planes, edges and etc. that we wish to join together. Standard options of joining are:

- Angle;
- Coincident;
- Concentric;
- Distance;
- Parallel;
- Perpendicular and
- Tangent.

All these types of joining are always shown under the *Property Manager*, but only those types of joinings that are applicative to the current selection. In Figure 11, an assembly is shown that contains all parts of a drawer. On the left part, the joinings that are used are shown. Because this concerns the parts of drawer whose surfaces are mostly straight, these are mostly the support surfaces; the types of joining used are parallel and coincident.



Slika 11. Prikaz sklopa ladice i pripadajućih sparivanja
Figure 11. Representation of drawer assembly and accompanying joinings

3. ZAVRŠNA IZRADA U TOPSU

Kada je modeliranje svih dijelova u *SolidWorksu* završeno i nakon što je kreiranjem sklopa potvrđena dimenzijska ispravnost svih dijelova, može se pristupiti izradi CNC programa. Izrada CNC programa predstavlja prije početka proizvodnje zadnji korak u izradi ladice. Izrada CNC programa vrši se za priliku ovog rada upotrebom sustava programiranja za probijanje i kombiniranu obradu ToPs 300 [6]. Dijelove ladice za tu je priliku potrebno spremati u .iges format kako bi ToPs 300 mogao koristiti modele. Nakon unosa crteža bilo je potrebno obaviti optimizaciju procesa obrade za stroj *Minimatic 100*, napraviti gniježđenje i simulirati obradu. Kada su postignute željena tehnika i strategija procesa, sve je spremno za izradu CNC programa koje softver ToPs 300 automatski generira. Na slici 12 prikazan je dio radnog naloga koji ToPs automatski generira pri izradi programa. Na radnom nalogu prikazuju se svi bitni podaci potrebni za proizvodnju. Nakon izrade svih dijelova ladice potrebno je saviti dijelove strojem *Schiavi Hfb 50-20* kako bi se dobili konačni oblici. Kada su dobiveni konačni oblici prema lancu procesiranja limova, potrebno je napraviti završnu obradu koju predstavljaju čišćenje dijelova te sklapanje, kako bi se dobio konačni oblik ladice prikazan na slici 13.

3. FINAL CREATION IN TOPS

When the modelling of all parts in *SolidWork* is finished, and after the dimensional accuracy of all parts with the creation of assembly has been verified, the creation of the CNC program can begin. Creation of the CNC program represents the last step in the creation of a metal drawer before production. The creation of a CNC program in the case of this paper, is done with the use of the ToPs 300 [6] system of programming for punching and combined production. It is necessary to save parts of the drawer for this occasion in the .iges format, in order that ToPs 300 can use the models. After the import of a drawing, it is necessary to do an optimisation of the production process for the machine *Minimatic 100*, to create a nesting and to simulate the production. When the desired technique and strategy of the process has been obtained, everything is ready for the creation of a CNC program, which the software ToPs 300 automatically generates. In Figure 12, a piece of work order is shown that ToPs automatically generates while creating the program. In the work order, all important data necessary for production are shown. After creation of all parts of the drawer, it is necessary to bend all parts on a *Schiavi Hfb 50-20* machine in order to gain the final shapes. When the final shapes have been obtained according to the sequence of sheet metal processing, it is necessary to create the final production that is represented by the cleaning of parts and the assembly of parts, in order to gain the final shape of the drawer shown in Figure 13.

STROJ:	TC100 - TYPE: 1, Sin 3N VARIANT: 1															
NAZIV PROGRAMA:	O:\ALPRON\11-CADCAMSOLIDWORKS\1-PROJEKT\ALPRONKARTO\TECNITOPSLADICA_KORITO.LST															
PROGRAM NAME:	1501 (LADICA_KORITO)															
TIP	S137-10	SIFRA.MAT:	FORMAT:924 x 585 x 1mm,(4.27028 kg)	SKART: 1.83 %												
MATERIJALA:																
TEORETSKO STROJNO VRIJEME:	0 : 00 : 00 (h:min:s)	REALNO VR:	BROJ PONAVLJANJA: 1													
	NO TRUMPF		TIMES AVAILABLE													
POSTAVKE STROJA																
POZICIJA KLJESTIA:	3 9															
PIN POZICIJA:	1 FROM THE LEFT															
POCETNA TOCKA:	X = 439.000 mm; Y = 280.000 mm															
NAPOMENA:																
LISTA ALATA																
SIFRA ALATA	TIP	DIM.1	DIM.2	DIM.3	KUT	OPIS	WT	MT	LT	PTT	OPIS	TDC	BDC	ISTROSENOT ALATA	STROKES	
1042000	1	4.200	0.000	0.000	0.000	Round 4.2	0	1	0						mm	54
1605000	16	5.000	5.000	0.000	0.000	Radius 5 width betw.radii 5	0	1	0						mm	4
4200020	4	20.000	2.000	0.000	0.000	Rectangle 20x2	0	1	0						mm	104
3120000	3	12.000	0.000	0.000	0.000	Square 12	0	1	0						mm	73
BROJ DRZACA ALATA: 4 / 0																

Slika 12. Dio radnog naloga u TOPS-u
Figure 12. Part of a work order in TOPS



Slika 13. Izrađena ladica
Figure 13. Created drawer

4. ZAKLJUČAK

Upotreba softvera CAD-CAM omogućila je mnogo manje potrebnog vremena kako bi se traženi proizvod mogao konstruirati, izraditi potrebni crteži, optimizirati strojeve, izraditi CNC programe te ih jednostavno poslati na stroj za početak proizvodnje. Ovaj rad predstavio je takvu CAD-CAM proizvodnju. Prikazano je konstruiranje ladice od lima upotrebom softverskog paketa *SolidWorks*, izrada crteža upotrebom računala, odabir potrebnih strojeva, te izrada CNC programa korištenjem softvera ToPs 300. U nastavku ovog istraživanja primijenit će se pristup utvrđivanja oblikovanja deformiranjem zasnovan na procjeni deformacijske energije i optimiziranog modeliranja povezanog s CNC programiranjem i definiranjem tehnologija oblikovanja. Pristup deformacijske energije bit će izveden pomoću *Abaqus* [7] programa radi procjene ovisnosti deformacijske energije o varijaciji ulaznih oblika dijelova ladice prije razrade njihova razvijenog oblika u *SolidWorksu* i prije njihove proizvodnje na strojevima.

ZAHVALA

Autori zahvaljuju na pomoći pruženoj od Nacionalnog CEEPUS ureda Češke Republike (voditelj: Jitka Nováková) i Nacionalnog CEEPUS ureda Hrvatske (voditelj: Davor Šovagović) koji su pomogli ovo istraživanje kroz tromjesečnu mobilnost i stipendiju diplomanta Saše Mijića u okviru CEEPUS II HR 0108 projekta (voditelj: Branimir Barišić, PhD).

5. POPIS OZNAKA

radijus savijanja	R ,	mm
K-faktor	K ,	
debljina materijala	T ,	mm
udaljenost do neutralne plohe	t ,	mm
kut savijanja	A ,	°

LITERATURA REFERENCES

- [1] Mijić, S., *Izrada ladice tehnologijom oblikovanja deformiranjem*, Diplomski rad, Tehnički fakultet Rijeka, 2007.
- [2] Barišić, B., *Predavanja iz kolegija Obrada deformiranjem*, Tehnički fakultet Rijeka, 2006.
- [3] *SolidWorks® – SolidWorks Essentials*, Parts and Assemblies, Volume 1 and Volume 2, 2005.
- [4] Barišić, B., Sladić, S., Soković, M.: *Numerical Approach For DC 01 Sheet Metal Drawer Forming*, Int. Conf. AMPT 07, Deajeon-Seoul, 2007.

4. CONCLUSION

The use of CAD-CAM software has ensured less time required in the construction of the required product, creation of the necessary drawings, optimisation of the machines, and creation of CNC programs and also the simple act of sending them to a machine for the beginning of production. This paper presents one such type of CAD-CAM production. The construction of a sheet metal drawer has been shown by the use of the software package *SolidWorks*, the creation of drawings by the use of a computer, the selection of necessary machines, and the creation of CNC programs by use of the software ToPs 300. In the following investigation, the approach used in establishing the forming process based on strain energy evaluation and optimized modeling connected with CNC programming and forming technology definition will be applied. The strain energy approach, by means of *Abaqus* [7] software, will be used to assess the dependency of the strain energy on the variation in input shapes of parts of the drawer before the elaboration of their developed shape in *SolidWorks* and before their production on machines.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the support provided by the National CEEPUS Office Czech Republic (head: Jitka Nováková) and National CEEPUS Office Croatia (head: Davor Šovagović) which helped in the research over three months of mobility and scholarship of the diploma thesis student Saša Mijić in the scope of the CEEPUS II HR 0108 project (head: Branimir Barisic, PhD).

5. LIST OF SYMBOLS

inner radius of bending
K-factor
thickness of material
distance to the neutral plane
bend angle

- [5] *ToPs 300 – Programming system*, TRUMPF Werkzeugmaschinen, Ditzingen, 2005.
- [6] Barišić, B., Car, Z., Nastran, M., *Numerical Strategy for USt 13 Sheet Metal DVD Holder Production*, Int. Conf. ICIT&MPT 2007, Bled, Slovenia, 2007.
- [7] *Theory Manual Abaqus /Explicit, Ver. 6.4.*, Karlsson & Sorensen Inc. Editions, Boston, 2003

Primljeno / Received: 30.2.2008

Prihvaćeno / Accepted: 18.6.2008

Prethodno priopćenje

Preliminary note

Adrese autora / Authors' address:

Saša Mijić

Branimir Barišić

Sveučilište u Rijeci, Tehnički fakultet

Vukovarska 58

51 000 Rijeka

HRVATSKA

barisic@riteh.hr

Jan Kudlaček

Czech Technical University Prague, Faculty of Mechanical Engineering

Zikova 4

166 36, Prague 6

CZECH REPUBLIC

jan.kudlacek@fs.cvut.cz