

# TEHNOLOGIJA IZRADE DIJELOVA TRAKASTOG TRANSPORTERA PRIMJENOM CAD-CAM POVEZIVANJA TECHNOLOGY OF ROLLER BED BELT CONVEYOR MANUFACTURE BY IMPLEMENTATION OF CAD-CAM CONNECTION

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**Sažetak:** Rad prikazuje tehnologiju izrade trakastog transportera (materijala AISI 304L) od modeliranja dijelova uz pomoć računala, sklapanja svih dijelova transportera u virtualni sklop i izradu dijelova transportera primjenom CAD-CAM povezivanja. Proizvodnja korištenjem CAD-CAM pristupa se može optimizirati i lako kontrolirati, iz čega proizlazi brža, kvalitetnija i efikasnija proizvodnja.

**Ključne riječi:**

- oblikovanje lima
- trakasti transporter
- modeliranje

**Abstract:** The paper presents the technology for the roller bed belt conveyor (material AISI 304L) from modeling of parts by usage of computer, joining of all conveyor parts in virtual assembly and manufacturing of parts by implementation of the CAD-CAM connection. With the use of the presented CAD-CAM approach, 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
- Roller bed belt conveyor
- Modeling

## 1. UVOD

Današnja moderna proizvodnja nezamisliva je bez upotrebe računala koje predstavlja osnovu za izradu 2D ili 3D konstrukcija, izradu nacrti i direktnog povezivanja računala sa strojem, sve u cilju brže, kvalitetnije i efikasnije proizvodnje [1-5]. U današnje vrijeme konkurencija je sve nemilosrdnija i ako želite provesti značajnija poboljšanja koja će donijeti bitnije promjene unutar kompanije kao što je npr. skraćenje trajanja razvojnog ciklusa proizvoda, upotreba 3D modeliranja postaje nešto neminovno. Sa *SolidWorks* programskim paketom bez puno muke možete napraviti iskorak u svijet 3D modeliranja a da pri tom i dalje u potpunosti koristite sve podatke koje ste godinama stvarali u 2D. Pri izradi dijelova trakastog transportera [1] biti će prikazan pristup naprednog modeliranja pomoću *SolidWorks* programa s konačnom svrhom izrade razvijenog oblika dijela, kako bi se napravio pogodan CAD oblik za stvaranje CNC koda za daljnju proizvodnju.

Dakle sa napravljenim 3D modelom dijela, kreće se sa stvaranjem razvijenog virtualnog dijela, na čijoj osnovi

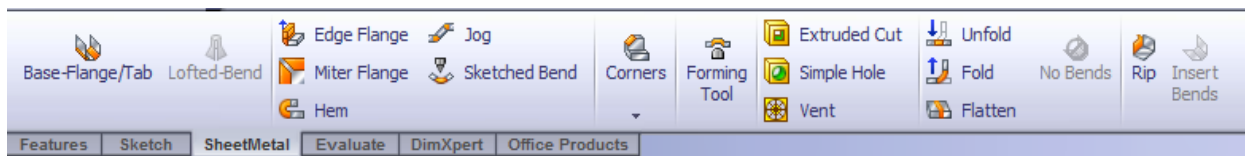
## 1. INTRODUCTION

Today, modern manufacturing is unimaginable without the use of a computer, which represents the fundamental basis for production of 2D or 3D constructions, draftings and direct connection of a computer with machine, all with the goal of faster and more efficient production of a higher quality [1-5]. Nowadays, the competition is getting more merciless and if you want to make some significant improvement which will bring about important changes in the company, as with shorter development production time, use of 3D modeling becomes something that is inevitable. With the *SolidWorks* program packet, you can easily get into the world of 3D modeling, still using all the data that you created for years in 2D. During the production of parts of the roller bed belt conveyor [1], you will see the approach of advanced modeling with the *SolidWorks* program, with the final goal of making a developed body shape, in order to make an appropriate CAD shape for generating CNC code for further production. Furthermore, with a modeled 3D part begins the

programer izrađuje NC program za sve strojeve koji se koriste za proizvodnju dijela. NC program govori stroju koje operacije treba izvršiti [6]. Današnji NC programi izrađuju se poluautomatski ili automatski, korištenjem softvera za programiranje koji sadrže precizne informacije o stroju i alatima. Programer odabire obradnu strategiju, izabire alate, i optimizira parametre. Tada, s jednostavnim klikom na miš, programer kazuje softveru za programiranje, da generira NC program. ToPs program biti će korišten za izradu CNC koda za obradu na strojevima [7].

## 2. MODELIRANJE DIJELOVA TRANSPORTERA U SOLIDWORKSU KORIŠTENJEM NAPREDNIH SHEET METAL FUNKCIJA

U tehnologiji oblikovanja deformiranja lima najvažnija alatna traka *SolidWorksa* je Sheet Metal (slika 1). Ona omogućava oblikovanje limova. Limove možemo savijati, dodavati i oduzimati dijelove, oblikovati provrte, zaobliti rubove, razviti plašt modeliranog dijela iz sheet metala itd.



Slika 1. Sheet Metal alatna traka  
Figure 1. Sheet Metal tool

Glavni koraci u procesu modeliranja dijelova trakastog transportera koji će biti izrađeni iz lima su:

- Modeliranje sheet metal alatima,
- korištenje alata za oblikovanje,
- razvijanje modela,
- izrada crteža iz 3D modela.

Na početku modeliranja 3D dijelova, prva polazišna točka je uvijek odabir ravnine u kojoj se počinje crtati te odabir naredbe za izradu Sheet Metal dijela. Nakon izrađenog sketcha dijela, dodajemo 3D parametre. Za što efikasnije modeliranje bitno na nacrtu uočiti profil koji je zajednički svim daljnjim savijanjima, odnosno polazišnu bazu s kojom se može riješiti najveći broj savijanja. [8]. U slučaju dijela TD103-001-01-202\_D2 to je okomita poprečna ravnina [1]. Dakle odabirom funkcije Base Flange stvorit će se taj profil u ravnini Front, jednostavnim alatima za skiciranje te će se izvršiti kotiranje zadanim dimenzijama [8]. Na slici 2 prikazan je profil dijela TD 103-001-01-202\_D2 tijekom skiciranja.

construction of the developed virtual part, on the basis of which the programmer builds the NC program for all of the machines which are necessary for production of the part. The NC program gives instructions to the machine on which operations need to be done [6]. Today, CNC programs are created automatically or half-automatically, by use of programming software which contains precise information about machines and tools. The programmer selects the work strategy, chooses the tools, and optimises the parameters. Then, with a simple click of a mouse, the programmer shows the software for programming, in order to generate an NC program. The ToPs program will be used for the production of the CNC code for the analysis [7].

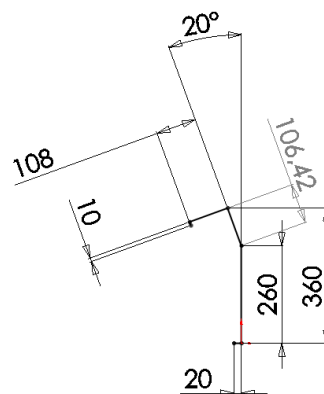
## 2. MODELING OF THE PARTS OF THE TRANSPORTER IN SOLIDWORKS USING ADVANCED SHEET METAL FUNCTIONS

In the technology of sheet metal forming, the most important *SolidWorks* tool is sheet metal (Figure 1). This tool enables forming of sheet metal. We can bend sheets, add or cut parts, form holes, round off edges, unfold a sheet of modeled sheet metal part etc.

The main steps in the modeling process of the roller bed belt conveyor parts which will be build in sheet metal are:

- Modeling with the *Sheet Metal* tools,
- Using forming tools,
- Unfolding the model,
- Making the drawings of 3D parts.

At the beginning of the 3D part modelling, the first step is always choosing the plane in which you start sketching and choosing the command for the modelling of sheet metal parts. After the sketching part, 3D parameters are added. For more efficient modelling, it is important to see if the design has a profile that is common to all the bendings in the model, with regard to the starting base with which we can solve the most of the bendings. [8]. In the case of the part *TD 103-001-01-202\_D2*, this is the front plane [1]. Choosing the Base Flange function, the profile of the front plane will be created with simple sketch tools and it will be properly dimensioned [8]. In the Figure 2 is shown the profile of the part TD 103-001-01-202\_D2 during the sketching.

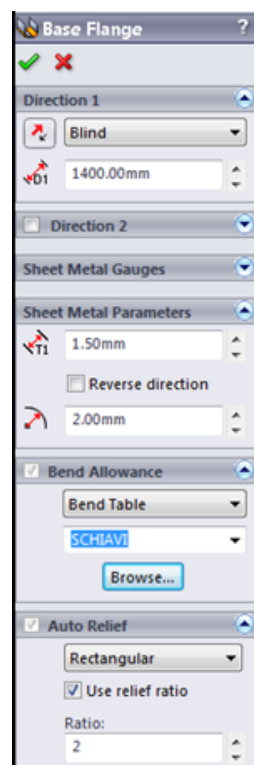


Slika 2. Profil dijela TD 103-001-01-202\_D2 [1]

Figure 2. Profile of the part TD 103-001-01-202\_D2 [1]

Nakon što je skiciranje završeno, slijedeći korak je definiranje 3D komponente, radijusa savijanja i definiranje korekturnog faktora  $K$ . Po izlasku iz sketch naredbe, otvara nam se meni Base Flange function u kojem definiramo 3D komponentu (slika 3).

When the sketching is done, the next step is defining the 3D components, the bend radius and the  $K$  factor. When the sketch function is closed, the Base Flange function menu appears, in which the 3D parameters are defined. (Figure 3).



Slika 3. Base Flange funkcija [1]

Figure 3. Base Flange function [1]

Na slici 3 prikazane su opcije funkcije Base Flange. Pod poljem Direction 1, odabire se smjer izvlačenja profila i duljina izvlačenja. Polje Direction 2, omogućava nam drugi tj. opcionalni smjer izvlačenja. U polju Sheet Metal Parameters, unose se vrijednosti debljine lima i radijus

Figure 3 shows the options of the Base Flange function. Under the field of Direction 1, the directions of extruding the profile and the length of extruding have been chosen. The Direction 2 field allows another - the optional direction of extruding. In the Sheet Metal

savijanja. Za polje Bend Allowance (dodanost-umanjenost savijanju) moguće je odabrati između K-faktora i tablice savijanja. To znači, da kada je modeliran dio savijen, da bi se dobila koja je prava duljina tog dijela potrebna za izradu, treba izračunati dodanost savijanju (bend allowance) ili umanjenost savijanju (bend deduction). To nam govori koliko treba dodati ili oduzeti dužine da bi se dobila tražena vrijednost [8].

### 2.1. K-faktor

Dodanost savijanju izračunava se korištenjem  $K$ -faktora po formuli:

$$BA = \pi (R + KT) A / 180.$$

$R$  – unutarnji radijus savijanja [mm].

$K$  – faktor, koji je  $t/T$ .

$T$  – debljina materijala [mm].

$t$  – udaljenost unutrašnje strane do neutralne plohe [mm].

$A$  – kut savijanja u stupnjevima (kut kroz koji je materijal savinut) [°].

the sheet metal thickness and the bending radius have been defined. For the Bend Allowance field, it is possible to choose between the  $K$  factor and the bend table. This means that, when the modelled part is bended, for getting the real length of the part used in the production, it is necessary to calculate the bend allowance or the bend deduction. This gives information about how much length is needed to be added or removed to obtain the right value [8].

### 2.1. K-factor

The bend allowance is calculated with using the  $K$  factor such that:

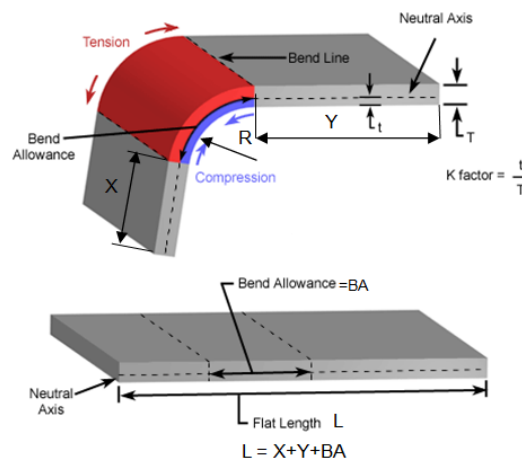
$R$  – Inner bending radius [mm].

$K$  – factor, which is  $t/T$ .

$T$  – thickness [mm].

$t$  – the distance between the inner part and the neutral line [mm].

$A$  – the bending angle in degrees [°].



Slika 4. Prikaz izračuna dodanosti savijanja [1]

Figure 4. The calculation of the bend allowance [1]

### 2.2. Tablica savijanja

Tablica savijanja (Bend Table) specifična je tablica materijala koja sadrži proračune savijanja temeljene na debljini lima i radijusu savijanja, a formira se kao Excel datoteka. Do podataka koji su uneseni u tablici savijanja dolazi se reverzibilnim inženjeringom, na način da se uzme uska traka materijala, savije se i tada se izmjeri da se izračuna dodanost savijanju. Ta se tablica lako unosi u SolidWorks na način da se odabere Insert→Sheet Metal→Bend Table→From File, ili na način da se iz Use Bend Table padajućeg izbornika odabere u okviru Sheet

### 2.2. Bend table

The bend table is a specific table of materials which contains the bending calculations based on the thickness and the bending radius. It can be made as an Excel file. Also, reverse engineering gives the data that can be imported into the bend table. It is easy to import this table into *SolidWorks*, choosing Bend Table Insert→Sheet Metal→Bend Table→From File, or by means of the Use Bend Table falling menu in the Sheet Metal command [8].

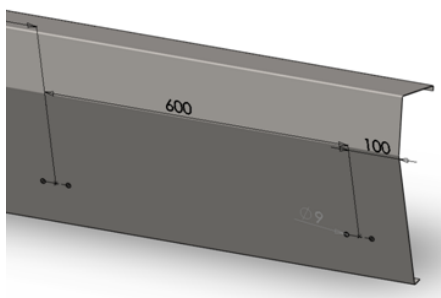
Metal naredbe [8]. Na ovom projektu izrade svih dijelova koristi se tablica savijanja (slika 5). Slika 6 prikazuje savijen profil dijela TD 103-001-01-202-D2.

Radius	0.6	0.8	1.0	1.2	1.5	2.0	2.5	3.0	4.0	5.0	6.0
1.0	1.3	1.6	1.9	2.3							
1.5	1.4	1.7	2.0	2.3	2.8						
1.8		1.8	2.1	2.4	2.9	3.7					
2.0			2.2	2.5	3.0	3.8	4.7				
2.5				2.7	3.2	4.0	4.8	5.7			
3.0					3.4	4.2	5.0	5.8	7.5		
4.0						4.5	5.2	6.0	7.7	9.4	
5.0							5.6	6.3	7.9	9.6	11
6.0								6.8	8.4	10	12
8.0									8.9	10	12
10.0										11	13

Comment: Dodani parametri za debljinu 5.0 i 6.0 mm  
 Comment: REV 28/09/2007.

Slika 5. Tablica savijanja  
 Figure 5. Bend Table

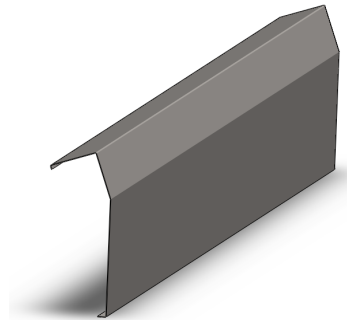
Nakon što je izrađen osnovni profil, slijedi izrada provrta te rezanje gornjeg dijela profila. Sve ovo vrlo jednostavno se napravi korištenjem funkcije Extruded Cut. Potrebno je odabrati funkciju, skicirati dio koji želimo odstraniti, te odabrati duljinu na koju želimo izraditi rez. Na slici 7 je prikazan profil tijekom skiciranja provrta (koji u sklopu služe za prihvat gornjih valjaka) nakon čega se oni modeliraju, te gotov dio TD 103-001-01-200\_D nakon korištenja funkcije Extruded Cut (slika 8).



Slika 7. Prikaz dijela TD 103-001-01-202\_D2 tijekom skiciranja provrta [1]  
 Figure 7. TD 103-001-01-202\_D2 during the sketching [1]

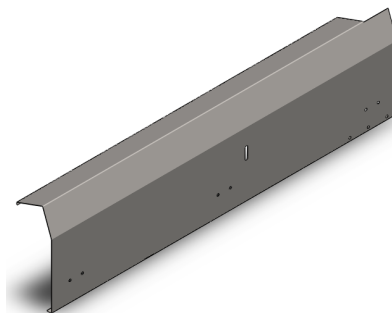
Slijedeći korak pri modeliranju dijela TD 103-001-01-202\_D2 je dodavanje bočnih stranica. Bočne stranice izvedene su korištenjem funkcije Edge Flange, koja je

On this project the Bend Table is used for producing all parts (Figure 5). Figure 6 shows the bended profile of the part TD 103-001-01-202-D2.



Slika 6. Savijen profil dijela TD 103-001-01-202\_D2 [1]  
 Figure 6. Bended profile of the TD 103-001-01-202\_D2 part [1]

After the basic profile was made, the making of holes and the cutting of the profile follows. All of this is simply made by using the Extruded Cut function. It is necessary to choose the function, sketch the parts that we want to remove, and choose the length to which we want to cut the part. In the Figure 7 is shown the profile during the sketching of the holes (which are used in assembly to accept the upper rollers support) after which they are modeled, and the finished part TD 103-001-01-200\_D after using the Extruded Cut function (Figure 8).



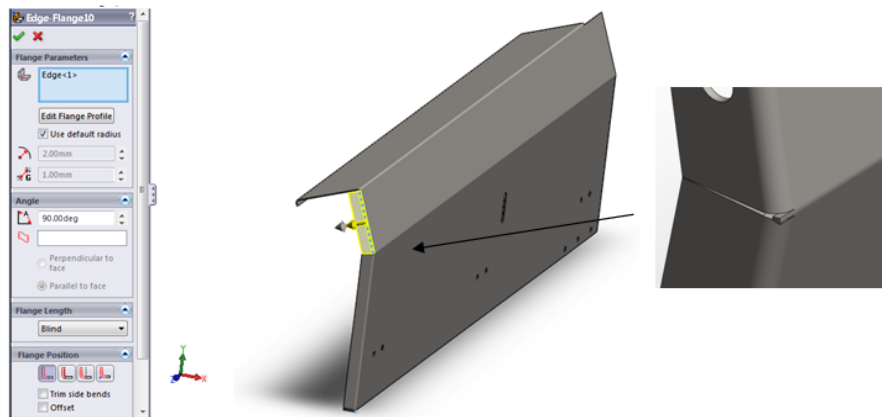
Slika 8. Prikaz dijela TD 103-001-01-202\_D2 nakon izvršenja naredbe Extruded Cut  
 Figure 8. TD 103-001-01-202\_D2 after the using of Extruded Cut function

The next step in modeling is adding the flank sides on part TD 103-001-01-202\_D2. The side bends are modeled by using the Edge Flange function which is very

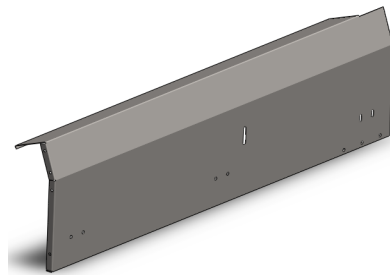
vrlo slična Base Flange funkciji. Potrebno je odrediti rubove koje želimo saviti, tj. rubove na koje želimo dodati.

Odaberu se radijusi i tablice savijanja i rub je gotov. Potreba popuštanja na mjestima gdje bi dolazilo do ulaska jednog materijala u drugi, rade se klasičnim extrude/remove material funkcijama, prikazano na slici 9 u detalju.

similar to the Base Flange function. It is necessary to define the edges we want to bend, i.e., the edges on which we want to add material, material, and to choose the radius and the Bend Table. The necessary releases in places where one part enters in to another are done with the classic Extrude-Remove material functions, as shown in the details of the Figure 9.



Slika 9. Izrada rubova naredbom Edge Flange  
Figure 9. Bending using Edge Flange function



Slika 10. Prikaz gotovog dijela TD 103-001-01-202\_D2 [1]  
Figure 10. Modeled part TD 103-001-01-202\_D2 [1]

Jedna od mogućnosti SolidWorksa je da za modelirani dio, može se napraviti njegov simetrični dio. U sklopu finalnog transportera nalazi se i dio koji je simetričan dijelu TD 103-001-01-202\_D2. Mirror dijela napravi se tako da se otvori nov assembly file u koji unosimo dio koji je modeliran. Nakon toga odabiremo funkciju Linear Component Pattern→Mirror Components. U toj funkciji potrebno je odabrati part koji želimo zrcaliti, te ravninu oko koje želimo zrcaliti predmet. U ova dva kratka koraka, vrlo brzo se napravi simetričan dio. Nakon što je model dijela napravljen pristupa se izradi tehničke dokumentacije. Kote koje se unose iz modela, vođene mjere, trebale bi biti uredno namještene u samom modelu. To dovodi do jednostavnijeg prijenosa i manje posla oko sređivanja crteža. Izmjene izgleda vrijednosti kao što su preciznost i tolerancije, mogu se načiniti na razini modela. Te izmjene se prenose i kada je model već prije unesen u crtež.

One of *SolidWorks* possibilities is that for the modeled part, you can make its symmetrical part. In the final assembly of the transporter, there is a part which is symmetrical to the TD 103-001-01-202\_D2 part. The mirror of the part generally is made by opening a new Assembly file, in which we insert a modelled part. After that, we choose the function Linear Components Pattern→Mirror Components. In that function we have to choose the part that we want to mirror, and the plane about which we want to mirror it. In these two short steps you can easily make a mirror part. After the part is modeled, we begin to compile the technical documentation. Measures from the model, loaded measures, should be properly fixed in the model. This leads to easier transportation and less work in adjusting the documentation. Modifications of the value of measures, such as precision and tolerance, can be done at the level of the model. These changes are transported even when the model is included earlier in the documentation.

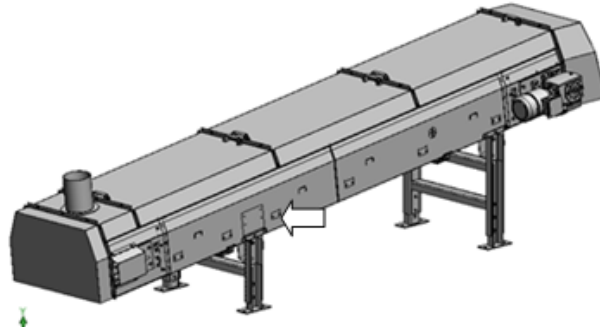


Slijedeća slika prikazuje trakasti transporter TD 103, te strelica miša ukazuje na kojem se mjestu u sklopu nalazi dio TD 103-001-01-202\_D2.

Sklop TD 103 sastoji se od oko 100 različitih dijelova.

The next figure shows the Roller Bed Belt Conveyor TD 103, and the mouse arrow shows the location of the part TD 103-001-01-202\_D2.

The assembly TD 103 contains around 100 different parts.



Slika 12. Prikaz gotovog sklopa trakastog transportera TD 103 [1]

Figure 12. The final assembly of the TD 103 [1]

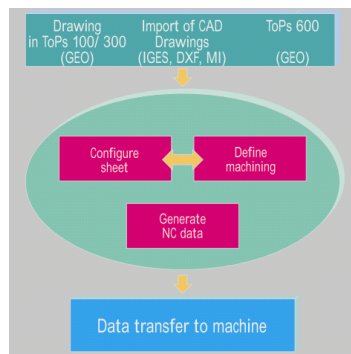
### 3. ZAVRŠNA IZRADA U TOPSU

Nakon što je završeno modeliranje svih dijelova trakastog transportera u SolidWorks-u, te nakon spajanja svih dijelova transportera u virtualni sklop čime je provjerena i potvrđena ispravnost svih dijelova, pristupa se izradi NC programa. Izrada NC programa predstavlja posljednji korak prije početka izrade dijelova transportera.

Izrada NC programa za dijelove transportera vrši se automatski upotrebom ToPs 300 [7] sustava programiranja za probijanje i kombiniranu (laser/probijanje) obradu.

### 3. THE FINAL PROGRAMMING IN TOPS

After the modeling of all parts in SolidWorks is done, and after the mating of all parts in virtual assembly, which allows us to check and confirm the validity of all parts, starts the building of the NC program. The NC program represents the last step before the beginning of the production of all parts of the conveyor. The production of the NC program is made automatically by using the ToPs 300 [7] system for programming, and for punching and combined (laser/punching) treatment.



Slika 13. 3 Koraka do izrade CNC programa [7]

Figure 13. 3 Steps towards the CNC program [7]

3 Koraka do izrade CNC programa:

#### 1. Korak

Izrada nacrtu u ToPs-u ili importiranje nacrtu iz 2D ili 3D CAD sustava putem IGES, DXF ili MI sučelja. Nacrti podesni za korištenje u proizvodnji, osnova su brzog i procesno sigurnog NC programa. Moguće je i izraditi nacrt u ToPs 300 sustavu ili izvršiti unos iz ToPs 600 sustava. ToPs automatski analizira svaki nacrt uzimajući u obzir obradu koja slijedi.

3 Steps towards the CNC program:

#### 1. Step One

Making a drawing in ToPs or importing the drawings from a 2D or 3D CAD system by IGES, DXF or MI interface. The drawings used in the production are the basis of a fast and process safe NC program. It is possible to make a drawing in the ToPs 300 system, or to import from the ToPs 600 system. ToPs automatically analyses every drawing, considering the treatment that follows.



Otvorene konture postaju zatvorene, konturni prijelazi izglađeni, nadodani elementi obrisani.

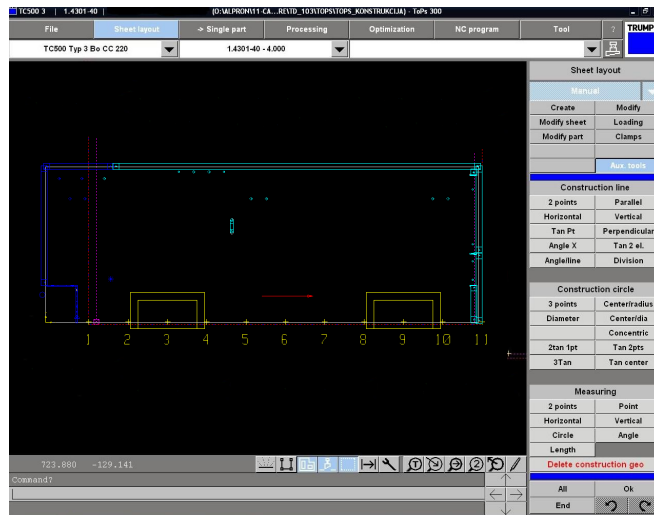
Opened contours becomes closed, contour crossings are shaped, and added elements are erased.

2. Korak

Nakon unosa crteža slijedi konfiguriranje lima radi maksimalnog iskorištenja, paralelno sa definiranjem samog procesa obrade, nakon čega slijedi generiranje NC koda (slika 14).

2. Step Two

After importing the drawings follows configuration of sheet metal because of maximum efficiency, parallel with the defining of the process treatment, and the generation of the NC code (Figure 14).



Slika 14. Konfiguriranje lima, jedan dio [1]  
Figure 14. Sheet metal configuration, single part [1]

3. Korak

Prijenos programa na upravljačku jedinicu stroja. Program ToPs podržava i opciju izrade radnog naloga sa šifrom i popisom alata, te pozicijama i dimenzijama. Jednostavnim klikom dobije se radni nalog sa navedenim podacima. Radni nalog se može vidjeti na slici 15. Na slici 16 prikazan je probijen dio TD 103-001-01-202-D2.

3. Step Three

Transfer of program to control unit of machine. The program ToPs supports the option of making a work order with a code and a list of tools, including positions and dimensions. With a simple click you get a work order with the mentioned data. A work order is shown in Figure 15. Figure 16 shows the punched part TD 103-001-01-202-D2.

TD103\_001\_01\_202\_D2

<b>TRUMPF</b>		<b>OPERACIJA PROBIJANJA</b>		Izradilo:											
		<b>RADNI NALOG br.</b>		Datum: 01.07.2009											
STROJ:	TC2000 - TYPE: 1, 1Bo Typ3, VARIANT: 1														
NAZIV PROGRAMA:	C:\ALPROM\11-CAD\CAM\SOLIDWORKS\11-PROJEKTI\IBD\EBEL ENGENIERIE\SPINNEKOPTRANSPORTER\088L0121-4_LEH\WRE\TD103_280609\TOPS\TOPS_KONSTRUKCIJA\TD103_001-01-202_D2_X1.LST														
PROGRAM NAME:	TD103_001_01_202_D2 (TD103-001-01-202_D2_X1)														
TP:	1.4301-15	SIFRA.MAT:	FORMAT:1430 x 510 x 1.5mm,(8.64221 kg)	SKART: 4.67 %											
MATERIJALA:		TEORETSKO STROJNO VRIJEME:	0 : 02 : 17 (j:mm:d)	REALNO VR:	BRUJ.PONAVLJANJA: 1										
POSTAVKE STROJA															
POZICIJA KLJUŠTA:	3 9														
PN POZICIJA:	1 FROM THE LEFT														
POCETNA TOČKA:	X = 215.000 mm, Y = 140.010 mm														
NAPOMENA:															
LISTA ALATA															
SIFRA ALATA	TIP	DIM.1	DIM.2	DIM.3	KUT	OPIS	WT	MT	LT	PTT	OPIS	TDC	BDC	ISTROJENOST ALATA	STROKES
01090000	1	0.000	0.000	0.000	0.000	Round 9	0	1	0	PTT-1	Round 9	0.000	0.000		11
03200000	3	20.000	0.000	0.000	0.000	Square 20	0	1	0	PTT-2	Square 20	0.000	0.000		141
04370050	4	37.000	5.000	0.000	0.000	Rectangle 37x5	0	1	0	PTT-3	Rectangle 37x5	0.000	0.000		7
01070000	11	7.000	0.000	0.000	0.000	Round 7	0	1	0	PTT-4	Round 7	0.000	0.000		8
01110000	11	11.000	0.000	0.000	0.000	Round 11	0	1	0	PTT-5	Round 11	0.000	0.000		2
04200020	4	20.000	2.000	0.000	0.000	Rectangle 20x2	0	1	0	PTT-6	Rectangle 20x2	0.000	0.000		3
17150000	17	15.000	20.600	7.000	0.000	Triangle 15	0	1	0	PTT-7	Triangle 15	0.000	0.000		2
03100000	3	10.000	0.000	0.000	0.000	Square 10	0	1	0	PTT-8	Square 10	0.000	0.000		10
INFORMACIJE O IZRATKU															
POZICIJA BR.	1	BRUJ NACRTA:		NAZIV:	TD103-001-01-202_D2	KUPAC:	IBD - EBEL								
BRUJ IZRADAKA:	1	DIMENZIJE IZRATKA:	1417.00 x 502.07 mm	POVRŠINA IZRATKA:	695201 mm <sup>2</sup>	MASA:	8.238 kg								

Slika 15. Radni nalog [1]  
Figure 15. Work order [1]



Slika 16. Probijen dio [1]  
Figure 16. Punched part [1]

#### 4. ZAKLJUČAK

Computer Aided Drafting ili Design (CAD) i Computer Aided Manufacturing (CAM) je softver koji pomaže pri konstruiranju, razvoju i optimizaciji dijelova. Oni su zamjena za tradicionalno inženjerstvo. CAD omogućuje konstruktoru konstruiranje i razvoj rada na ekranu, ispis dokumentacije i omogućava da se rad sačuva za buduće korištenje, eventualno editiranje tj. prilagođavanje postojećeg modela kod modeliranja novog. Ovime se skraćuje vrijeme potrebno za modeliranje novih dijelova. Može se reći da CAD stvara beneficije kao što su manji troškovi razvoja proizvoda i skraćeno vrijeme konstruiranja. CAM je programski alat koji omogućuje proizvodnju fizičkih modela koristeći CAD programe.

Ovaj rad predstavio je jednu takvu CAD – CAM proizvodnju. Prikazano je konstruiranje dijela transportera upotrebom softverskog paketa SolidWorks, izrada crteža upotrebom računala, te izrada NC programa koristeći softver ToPs 300. U ovom koraku kod sheet metal proizvoda, nakon što se modelira 3D dio, kreće se sa razvijanjem plašta dijela, koji naknadno koristimo kod izrade CNC programa.

Upotreba CAD – CAM softvera omogućila je mnogo manje potrebnog vremena kako bi se traženi proizvod mogao konstruirati, izraditi potrebni crteži, optimizirati strojeve, izraditi NC programe te ih jednostavno poslati na stroj za početak proizvodnje. Današnje lansiranje proizvodnje postalo je mnogo lakše i omogućene su brze izmjene postojećih rješenja.

#### 5. POPIS OZNAKA

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

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#### 4. CONCLUSION

Computer Aided Drafting or Design (CAD) and Computer Aided Manufacturing (CAM) is a software program which helps in the modeling, development and optimization of parts. They represent the replacement for traditional engineering. CAD allows the constructor to construct and develop the work on a screen, printing the documents, and it also allows the work to be saved for later use, eventual editing, and modification of the present model by modeling a new one. It helps to make time for the modeling of the new parts shorter. We can say that CAD offers such benefits as lower prices of the product and a shorter construction time.

CAM is a program tool which helps in the production of physical models by using CAD programs.

This work shows one such CAD – CAM production. The modeling of part of the transporter is shown, using the *SolidWorks* software packet, making a drawing using a computer, and the building of NC program using ToPs 300 software. In this step by sheet metal products, after the 3D part is modeled begins the unfolding of the 3D model, which we use later during the building of the CNC program.

The use of the CAD – CAM software enabled less time needed for the construction of the product, making necessary drawings, optimization of the machines, making NC programs, and easily sending them on the machine for the beginning of production. Nowadays, the launching of production became much easier and faster changes in present solutions are enabled.

#### 5. LIST OF SYMBOLS

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

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