PROGRAMIRANJE OBRADE GLODANJA UPOTREBOM ESPRITA

PROGRAMMING OF MILLING OPERATIONS BY USING ESPRIT

Sanela Hrnjica

Univerzitet u Bihaću
Tehnički fakultet
Mašinski odsjek
77000 Bihać

Stručni rad

REZIME

Brzina obrade je jedna od najbitnijih komponenti u procesu proizvodnje, pa je prema tome potrebno koristiti što kompletniji programski paket odnosno softver. Esprit je idealan izbor, jer je to CAD/CAM softver u kojem je moguće crtati, modelirati, programirati, a na kraju generirati G-kod za bilo koji alatni stroj. On je jako moćan alat visokih performansi sa punim spektrom programskih sistema za glodanje, tokarenje, bušenje itd. Kroz ovaj rad obradit će se proces programiranja obrade glodanjem i prikazati neke od najčešće korištenih opcija koje nudi Esprit. Jedna od bitnijih mogućnosti ovog softvera jeste realna simulacija samog procesa na mašini, što je jako bitno kod procesa obrade, posebno glodanja na petoosnim mašinama.

Professional paper

SUMMARY

Machining speed is one of the most important components in the production process, so it is necessary to use the most complete program package or software. Esprit is the ideal choice, because it is a CAD/CAM software in which it is possible to draw, model, program, and finally generate G-code for any machine tool. It is a very powerful, high-performance tool with a full spectrum of programming systems for milling, turning, drilling, etc. This paper will cover the milling programming process and show some of the most used options offered by Esprit. One of important features of this software is the realistic simulation of the process itself on the machine, what bears significance in the machining process, especially in milling on five-axis machines.

1. UVOD

Esprit je softver koji omogućuje programiranje, simulaciju i automatski generiran G-kod za bilo koju klasu CNC alatnih mašina, uključujući horizontalne, vertikalne i portalne mašine. Izgrađen kroz 38 godina inovacija, danas je jedan od najboljih CAM programa. DP tehnologija Esprit je moćan sistem za CNC programiranje, simulaciju, kao i optimizaciju izrade tehnoloških procesa. Podržava cijeli proces proizvodnje od CAD datoteke do obrađenog dijela. Ovaj program je u potpunosti usklađen s industrijskim standardima i omogućuje simulaciju procesa obrade u stvarnom vremenu. Sve te karakteristike su se kroz dugi niz inovacija razvijale i još uvijek se razvijaju i upotpunjuju [1]. Ovaj program je veoma lako instalirati uz odgovarajuću licencu i određene karakteristike koje računar mora posjedovati da bi instalacija bila moguća, a te karakteristike su radna memorija od 1 GB RAM-a, slobodni prostor na hard disku od 2 GB i procesor Intel Dual Core [2].

1. INTRODUCTION

Esprit is software that enables programming, simulation and automatically generated G-code for any class of CNC machine tools including horizontal, vertical and gantry machines. Software built through 38 years of innovation is one of the best CAM programs today. DP technology Esprit is a powerful system for CNC programming, simulation and optimization of technological processes. It supports the entire production process from the CAD file to the machined part. This program is fully compliant with industry standards and allows simulation of the machining process in real time. All these characteristics have been developed through a long series of innovations and are still being developed and completed [1]. This program is very easy to install with the appropriate license and certain characteristics that the computer must have in order for installation to be possible, and these characteristics are a working memory of 1 GB of RAM, free space on the hard disk of 2 GB, and an Intel Dual Core processor [2].

2. RADNI PROSTOR ESPRITA

Podešavanje radnog prostora Esprita je subjektivan izbor, što se tiče samog izgleda i postavljanja alatnih traka, ali postoje obavezne alatne trake koje se koriste pri izradi tehnološkog procesa odnosno pri programiranju komada koji će biti izrađeni na CNC mašinama. Prije samog definiranja alatnih traka i prečaca, koji će se koristiti tokom programiranja, potrebno je na računaru spremiti odgovarajuće postprocesore koji će služiti za kreiranje NC koda, te *template* mašine, koja se nalazi u proizvodnji, da bi se pri programiranju dobio stvarni izgled mašine i komada koji će se obrađivati na toj mašini. Današnji CAM sistemi za izradu NC programa moraju imati definiran postprocesor koji se izrađuje za svaki alatni stroj zasebno. Postprocesor ima funkciju pretvaranja CL (engl. *Cutter location file*) u NC datoteku u obliku G-koda. Tako prilagođen program se pohranjuje u bazu podataka CAD/CAM sistema i po potrebi se koristi.

### 2.1 Kreiranje alatnih traka, steznih naprava i alata

Espritov grafički interfejs dopušta brz i lak pristup komandama s potrebom definiranja maksimalnog područja za rad. Da bi se moglo brzo i efikasno programirati, potrebno je pozvati odnosno kreirati određene prečace koji će ubrzati sâm proces programiranja. Sva podešavanja se odvijaju pomoću glavne alatne trake. Izbor alatnih traka je velik i treba se ograničiti na određeni broj alatnih traka koje će se najčešće koristiti, tako da izgled radnog prostora ostane pregledan i lako upotrebljiv, slika 1. Alatne trake i postavke radnog područja se vrše s pretpostavkom da su sve prethodne postavke odrađene, a koje se tiču prethodno spomenutih postprocesa i uzoraka različitih mašina koje se posjeduju u proizvodnom pogonu. Osnovna podešavanja jesu dijelovi mašine, kao npr. okvir mašine, radni stol, broj osovina mašina, revoler itd. Među prvim podešavanjima je definiranje radnog prostora mašine odnosno mjesta na mašini gdje će se obradak uvijek nalaziti kada se pozove u program. To je bitna postavka, jer se tako dobija stvarna preglednost mašine i obratka, te se lako uoči ako je obradak veći od mašine.

2. ESPRIT WORKSPACE

Adjusting the Esprit workspace is a subjective choice as far as the appearance and placement of toolbars are concerned, but there are mandatory toolbars that are used when creating a technological process, i.e. when programming pieces that will be made on CNC machines. Before defining the toolbars and shortcuts that will be used during programming, it is necessary to save the appropriate post-processors on the computer that will be used to create the NC code, and the machine template, that is in production, in order to get the actual appearance of the machine and the pieces that will be processed on that machine. Today's CAM systems for creating NC programs must have a defined postprocessor that is created for each machine tool separately. The post-processor has the function of converting the CL (Cutter location file) into an NC file in the form of G-code. Such a customized program is stored in the database of the CAD/CAM system and used as needed.

2.1 Creation of toolbars, clamps and tools

Esprit's graphical interface allows quick and easy access to commands, but it is needed to define the maximum area for work. In order to be able to program quickly and efficiently, it is necessary to call or create certain shortcuts that will speed up the programming process itself. All adjustments are made using the main toolbar. The selection of toolbars is large and should be limited to a certain number of toolbars that will be used most often, so that the appearance of the workspace remains clear and easy to use, Figure 1. Toolbars and work area settings are made with the assumption that all previous settings have been done, which concern the previously mentioned post-processes and samples of the different machines that are owned in the production plant. The basic settings are the parts of the machine, such as the machine frame, work table, number of machine axes, turret, etc. Among the first settings, the working area of the machine is defined, i.e. the place on the machine where the workpiece will always be when it is called into the program. This is an important setting because this way you get a real view of the machine and the workpiece, and it is easy to see if the workpiece is bigger than the machine.
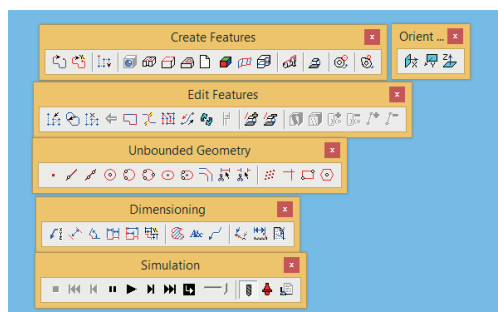
Slika 1. Često korištene alatne trake
Figure 1 Commonly used toolbars

Također, definira se i prozor koji sadrži glavni XYZ koordinatni sistem s pripadajućim operacijama, karticama pripadajućih alata te redoslijedom operacija. U tom prozoru postoje i detaljne informacije o svakoj operaciji koja se odabere te vrijednosti koordinata i vremena trajanja operacija. Pored osnovnih prozora kreiraju se i pomoćni prozori sa slojevima (engl. *layers)* koji se mogu kreirati, uključivati i isključivati prema potrebi i broju operacija i geometriji koju posjeduje komad. Oni olakšavaju programiranje, jer program postaje pregledniji i jasno organiziran. Jedan od pomoćnih prozora jeste i prozor maske koji omogućuje prikazivanje i sakrivanje nacrtne geometrije, linija, putanje alata, 3D modela, krivulja, ploha, koordinatnih sistema itd. Za bolju preglednost u toku programiranja Esprit nudi i podešavanje boja radnog prostora i svega onoga što se selektira tokom rada. Tako je moguće odabrati različite boje komada, kao i boju za određene površine komada da se lakše prepoznaju kad se selektiraju. Dodatno, mogu se definirati boje prema alatima koji rade tako da su različite boje pri procesu grubog glodanja i finog glodanja. Kreiranje steznih naprava je izrazito bitan postupak kada se komadi rade na steznim napravama, jer se tako pri simulaciji programa može uočiti eventualni sudar stezne naprave i alata odnosno sudar sa bilo kojim dijelom u mašini. To je izuzetno bitno kod procesa glodanja na petoosnim mašinama, gdje su u pitanju simultana kretanja stola i alata. Stezne naprave je moguće kreirati u SOLID-u ili drugom CAD programu, te ih kao takve uvesti u Esprit. Kreiranje alata je jako olakšano u Espritu, jer program pruža jasan izgled alata i svih dimenzija alata, koje se unose prema kataloškim karakteristikama i mogućnostima alata, ili ako su u pitanju specijalni alati, onda prema kreiranim dimenzijama alata.
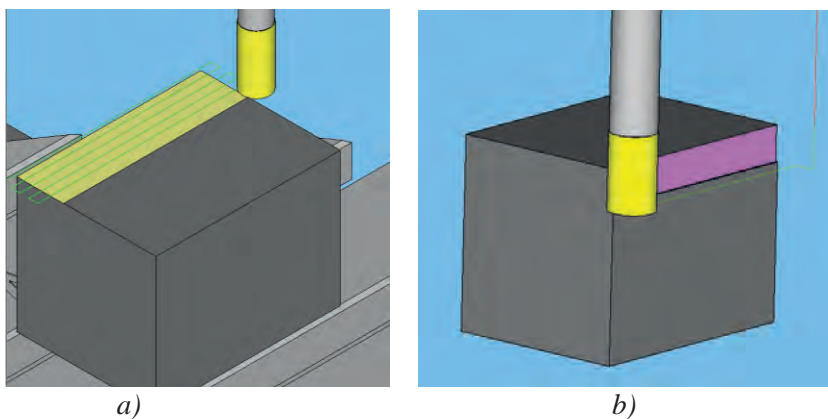
A window is also defined that contains the main XYZ coordinate system with associated operations, tabs of associated tools, and the order of operations. In that window, there is also detailed information about each operation that is selected, as well as the coordinate values and duration of the operations. In addition to the basic windows, auxiliary windows with layers are created, which can be created, turned on and off according to the need and the number of operations and the geometry of the piece. They make programming easier because the program becomes more transparent and clearly organized. One of the auxiliary windows is the mask window, which allows the operator to display and hide the design geometry, lines, tool paths, 3D models, curves, surfaces, coordinate systems, etc. For better visibility during programming, Esprit also offers adjustment of the colors of the workspace and everything else needed during operation. So it is possible to choose different colors for the piece, and also the color of a certain surface of the piece can be selected to make it easier to recognize it when selected. The operator can also define colors according to the tools that work, so tools can be of one color during the process of rough milling and of other during fine milling. The creation of clamping devices is an extremely important procedure when pieces are worked on clamping devices, because during the simulation of the program it is possible to observe a possible collision between the clamping device and the tool, i.e. a collision with any part of the machine. This is especially important in the milling process on five-axis machines where simultaneous movements of the table and tool are involved. Clamping devices can be created in SOLID or any other CAD program, and imported as such into Esprit. Tool creation is greatly facilitated in Esprit because this program provides a clear view of the tool and all tool dimensions that

3. OBRADA GLODANJEM UPOTREBOM ESPRITA

Glodanje je nakon tokarenja najvažniji postupak obrade metala skidanjem strugotine. Značajan broj radova ukazuje na aktuelnost istraživanja i optimizacije tehnologije, sistema, alata, softverskih aplikacija, upravljanja podacima itd. [3-5]. To je postupak obrade skidanjem čestica kod kojeg alat obavlja glavno kretanje. Obavlja se alatima s više jednakih oštrica ili sa sastavljenim alatima. Prema tome, glodanje je složenija operacija od tokarenja ili bušenja. Glodanje se može podijeliti prema različitim kriterijima, slika 2.:
- prema proizvedenom kvalitetu obrađene površine (grubo glodanje, završno glodanje, fino glodanje) i
- prema položaju reznih oštrica na glodalu (obodno i čeono) [6].

Programiranje obrade glodanjem u EPSRIT-u bit će predstavljano na jednom komadu, materijala En Aw 5083 (legura aluminija sa magnezijom i tragovima mangana i kroma). Komad će se raditi na petoosnoj Haasovoj mašini UMC 750 i bit će stegnut u pakne LANG, jer je komad manjih dimenzija, stoga će se raditi u steznom uređaju.

3. MILLING OPERATION USING ESPRIT

After turning, milling is the most important metal processing procedure by removing chips. A significant number of papers indicate the relevance of research and optimization of technology, systems, tools, software applications, data management, etc. [3-5]. It is a machining process by removing particles where the tool performs the main movement. It is performed with tools with several equal blades or with assembled tools. Therefore, milling is a more complex operation than turning or drilling. Milling can be divided according to different criteria, Figure 2:
- according to the produced quality of the processed surface: rough milling, final milling, fine milling,
- according to the position of the cutting blades on the milling cutter: contoure and face milling [6].

are entered according to the catalog characteristics and capabilities of the tool or, if it is a question of special tools, then according to the created dimensions of the tool.

Slika 2. Vrste glodanja: a) čeono b) obodno
Figure 2 Milling types: a) face milling b) contouring

Da bi se uspješno programirao komad u Espritu, potrebno je najprije definirati sve alate odnosno glodala koja će se koristiti. To su u većini slučajeva standardni alati koje mašine posjeduju. U nastavku će biti objašnjene mogućnosti Esprita pri obradi glodanjem.

Milling programming in Esprit will be presented on a single piece, material En Aw 5083 (an alloy of aluminum with magnesium and traces of manganese and chromium), and the piece will be worked on a five-axis Haas machine UMC 750, and the piece will be clamped in LANG pads, because the piece is of small dimensions and will, therefore, be worked on in a clamping device. In order to successfully program a piece in Esprit, it is necessary to define all the tools or milling cutters that will be used, in most cases they are

standard tools that are available on machines. Therefore, only Esprit's options for milling processing will be explained further.
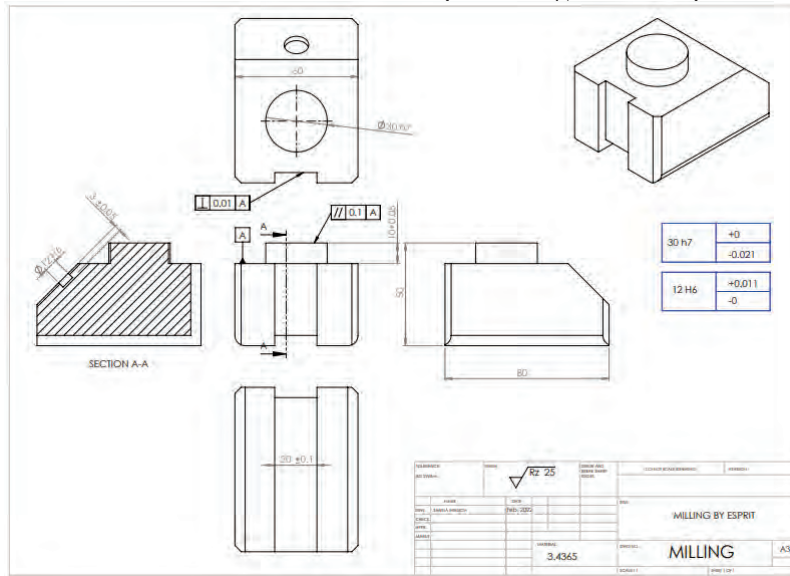
Slika 3. Komad za glodanje
Figure 3 Milling peace

Komad se radi na steznom uređaju, pa će prema tome komad biti urađen u dvije obrade. U prvoj obradi će biti odrađeno čeono glodanje, te glodanje kontura, ivica i kanala na komadu, slike 3. i 4.

The piece is positioned in clamping device and, therefore, two operations will be applied. During the first one, face milling will be performed, as well as contour, edge and channel milling on the piece, Figures 3 and 4.
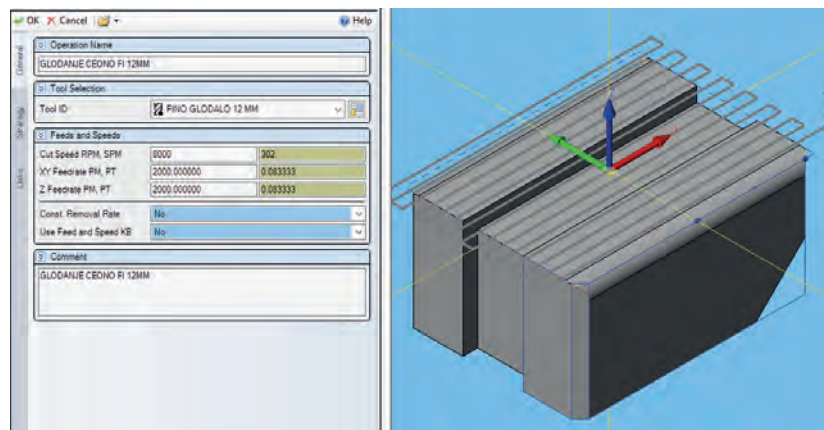
Slika 4. Čeono glodanje
Figure 4 Face milling

Ovaj program daje jasan prikaz operacija, identičan onome kako će se vršiti na mašini. U programu se podešavaju režimi obrade prema standardnim izrazima za izračunavanje tih vrijednosti ili iskustveno prema onome kako se odvija u proizvodnji. Kod obodnog glodanja i glodanja kanala na komadu, ovaj program nudi dosta mogućnosti kako bi se postigli najbolji rezultati i optimiziralo vrijeme izrade, a alat najbolje iskoristio.

This program provides a clear representation of operations identical to what will be performed on the machine. In the program, processing regimes are adjusted according to standard expressions for calculating these values or empirically according to what happens in production. For contour milling and channel milling on a piece, this program offers a lot of options to achieve the best results and optimize the production time, and make the best use of the tool.
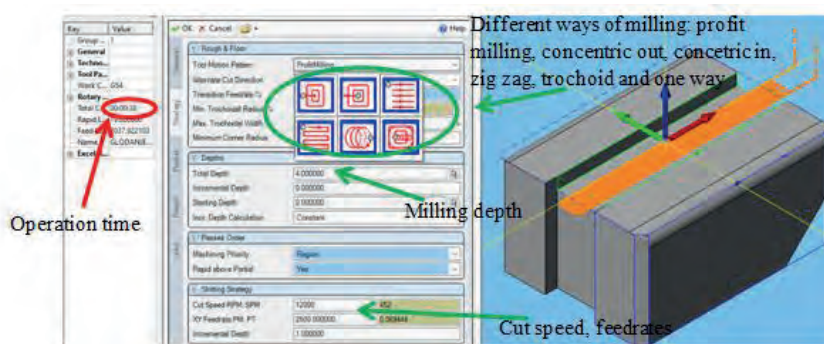
Slika 5. Žlijebljenje u softveru Esprit
Figure 5 Pocketing in Esprit software

Prostor glodanja se kreira ručno ili automatski, zavisno od toga koristi li se 3D ili 2D model, te ovisi o tome koliko je otvor na komadu pristupačan. Kod ručnog kreiranja *Chain*-a, potrebno je crtanje graničnih linija odnosno prostora koji će alat glodati. Potrebno je jasno definirati smjer glodanja, a to se može korigovati i jasno definirati u traci *Edit Features*. Ova traka nudi opcije promjene smjera, promjenu početka kretanja alata, a pored toga na tom *Chain*-u se može definirati je li on otvoren ili zatvoren. To automatski olakšava kretanje alata, jer alat ima otvoren prostor za početak i ne mora započeti svoje kretanje u punom materijalu. Kod korištenja 3D modela kreiranje otvora je uvelike olakšano, jer je na taj način moguće automatski prepoznati dijelove komada koje je potrebno glodati i nije potrebno ručno crtanje linija. Za takav način glodanja koristi se opcija *Pocket*, koja omogućuje automatsko kreiranje *Feature* džepa, slika 5. Kreira se tako što se označi površina gdje se taj džep nalazi i ta opcija odmah prepoznaje radi li se o otvorenom ili zatvorenom džepu. Kod odabira načina kretanja glodala mogu se odabrati *Profit Milling*, *Concentric In*, *Concentric Out*, *Zig Zag*, *One Way*. Opcija *Concentric In* znači da će alat glodati džep od ivica džepa prema centru. Ako u odabranom džepu postoji neko ostrvo koje ne bi trebalo biti odglodano, tada se to definira kao zatvoreni *Island Features* i glodalo će ga pri obradi zaobilaziti. Ako ne postoji nikakav oblik prepreke, glodalo će obraditi cijeli označeni džep, bilo da je označen kao *Pocket* ili kao *Chain*. *Concentric Out* je operacija kod koje će se glodalo kretati od centra prema ivici džepa. *One Way* znači da se alat kreće samo u jednom smjeru, dok opcija *Zig Zag* znači da će se kretati naizmjenično u oba smjera. Opcija *Profit Milling* se koristi kod otvorenih i zatvorenih džepova, i to kada je potrebna maksimalna iskorištenost alata. U ovom slučaju alat gloda sa svojom ukupnom dužinom u jednom prolazu zbog manjeg opterećenja na alat.

The milling space is created manually or automatically depending on whether a 3D or 2D model is used, and it depends on how accessible the opening on the piece is. When manually creating a Chain, it is necessary to draw boundary lines, that is, the space that the tool will mill. It is necessary to clearly define the milling direction, and this can be corrected and clearly defined in the Edit Features bar. This bar offers options for changing the direction, changing the start of the tool's movement, and in addition, it can be defined on that Chain whether it is open or closed. This automatically makes tool movement easier, because the tool has an open space to start and does not have to start its movement in full material. When using a 3D model, the creation of openings is greatly facilitated, because in this way it is possible to automatically recognize the parts of the piece that need to be milled, and manual drawing of lines is not required. For this type of milling, the Pocket option is used, which enables the automatic creation of a Feature pocket, Figure 5. It is created by marking the surface where that pocket is located and this option immediately recognizes whether it is an open or closed pocket. When choosing the way of movement of the milling cutter, the operator can choose Profit Milling, Concentric In, Concentric Out, Zig Zag, One way. The Concentric In option means that the tool will mill the pocket from the edges of the pocket towards the center, if there is an 'island' in the selected pocket that should not be milled, then it is defined as closed Island Features and the router will bypass it during processing. If there is no obstruction of any kind, the router will machine the entire marked pocket whether it is marked as Pocket or as Chain. Concentric Out is an operation where the milling cutter will move from the center towards the edge of the pocket.

Kod svih ovih opcija dodatno je potrebno definirati ulaz glodala, što je jako bitno, jer ako je nepravilan ulaz glodala, najčešći ishod jeste pucanje alata. Pri odabiru džepa (*Pocket*) postoji prozor naziva *Entry/Exit*, što zapravo označava mjesto u kojem se namješta ulaz i izlaz alata, pa je ponuđeno nekoliko opcija: *Plunge, Plunge at Point, Ramp along pass, Ramp contained, Helical contained* i *Helical at Point*. Opcija *Plunge* znači da alat ulazi u materijal po z osi, a *Plunge at Point* znači da će alat ulaziti u materijal u jasno definiranoj tački. To je najčešće rupa koja je većeg promjera od promjera glodala, tako da glodalo može nesmetano ući u tu rupu, a ne u materijal, dok ostale opcije dopuštaju ulaz glodalu spiralno pod uglom. Operaciju glodanja konture moguće je pozvati automatski, ako je uvezen 3D model, i to tako što se mišem označi odgovarajuća površina i pozove se opcija *Wall* u alatnoj traci *Create Features*. Kada se pozove opcija *Wall,* program kreira u stablu operacija *Profile*, na koji se zatim poziva grubo i fino glodalo koje će obrađivati označenu površinu. Ako nije stvorena baza operacija obrada, konture se mogu kreirati koristeći opciju *Machining* i *Contouring*. Za obrade za koje se ne posjeduje odgovarajuće glodalo, najčešće su to u pitanju prevelika skošenja ivica ili zaobljenja, bilo konkavna ili konveksna, kroz Esprit se može odabrati opcija 3D glodanja, koja dopušta samo da se obrađuje taj specifični dio komada. Za bilo kakav oblik 3D glodanja poziva se traka *SolidMill Mold*, koja nudi mnogo opcija glodanja i kretanja alata, a najčešće korištena jeste *Between Curves Finishing* - završna obrada između krivulja. Da bi takav način obrade funkcinirao, definiraju se *Chain*-ovi, gornji i donji, koji zapravo određuje kretanje alata. Nakon njihovog kreiranja, poziva se navedena opcija i u prvom dijelu prozora, koji se naziva *Part*, označi se onaj dio koji će se obrađivati, dok se u drugom dijelu, naziva *Check*, označi onaj dio koji alat ne smije obrađivati. Pored navedenih mogućnosti Esprit nudi još mnogo opcija koje se koriste u zavisnosti od toga kakav komad se obrađuje. Nakon definiranja prve i druge obrade generira se 'kod', koji se na jednostavan način iz programa prebaci na mašinu i kao takav je spreman za upotrebu.

One way means that the tool moves in one direction only, while the Zig Zag option means that it will move alternately in both directions. The Profit Milling option is used for open and closed pockets, and when maximum utilization of the tool is required. In this case, the tool mills with its total length in one pass due to less load on the tool.

With all these options, it is additionally necessary to define the entry of the milling cutter, which is very important, because if the entry of the milling cutter is incorrect, the most common outcome is tool breakage. When selecting Pocket, there is a window called Entry/Exit, which actually indicates the place where the entry and exit of the tool is set, so several options are offered. These options are: Plunge, Plunge at Point, Ramp along pass, Ramp contained, Helical contained and Helical at Point. The Plunge option means that the tool enters the material along the z axis, and the Plunge at Point means that the tool will enter the material at a clearly defined point. This is most often a hole with a larger diameter than the diameter of the router, so that the router can enter the hole (without hindrance), and not into the material, while the other options allow the router to enter spirally at an angle. The contour milling operation can be called automatically if a 3D model has been imported by selecting the appropriate surface, with the mouse and calling the Wall option in the Create Features toolbar. When the Wall option is called, a program creates a Profile in the operation tree, upon which is then called a rough and fine milling tool to process the marked surface. If the operation base has not been created, contour processing can be created using the Machining and Contouring options. For all pieces for which a suitable milling machine is not available, most often the issues are excessive chamfering of edges or rounding, whether concave or convex, through Esprit can be selected the 3D milling option that allows only that specific part of the piece to be processed. For any form of 3D milling, the SolidMill Mold bar is called, which offers many milling and tool movement options, and the most commonly used is Between Curves Finishing. In order for this method of processing to work, Chain's top and bottom are defined, which actually determines the movement of the tool. After creating the chains, the specified option is called, where in the first part of the window, called Part, the part that will be processed is marked, while in the second part,

4. ZAKLJUČAK

Savremene tehnologije obrade na CNC obradnim sistemima zahtijevaju profesionalno upravljanje cjelokupnim procesom, jer u suprotnom može doći do značajnih materijalnih oštećenja, kao i vremenskih gubitaka i zastoja u proizvodnji, što dovodi do probijanja rokova isporuke. Upravljanje savremenim zahtjevima CNC obrade korištenjem CAD/CAM sistema učinjen je značajan napredak, jer se može izvršiti kompletna simulacija obrade. Također, upravljačke jedinice CNC mašina daju mogućnost simulacije putanje alata. Ako su definirani svi parametri simulacija u CAD/CAM sistemu, to je dovoljno, a u toku same proizvodnje potrebno je osigurati odgovarajući broj kontrola obradaka i nadzor nad CNC proizvodnim sistema.

Upravo Esprit predstavlja jedan od odličnih primjera softverskog CAD/CAM sistemski aplikativnog rješenja, jer nudi potpunu simulaciju onoga što se radi na mašini, te u procesu obrade glodanja ima veliki spektar definiranja parametara alata i načina obrade.

REFERENCES

[1] D. T. Panayotov, V. P Dimitrov, K. G. Kalev (2016) *Comparative Analysis of the Processes for Machining of Mold Element with Using TOPSLOID'S CAM and ESPRIT*, International Journal of Mechanical Engineering and Technology, 7, 2, 01-10, IJMET 07 02 001

[2] https://www.espritcam.com/info/why-esprit?pageSource=Home on date 27.12.2021.

[3] Selvaraj, Vignesh & Xu, Zhicheng & Min, Sangkee. (2022). *Intelligent Operation Monitoring of an Ultra-Precision CNC Machine Tool Using Energy Data*. International Journal of Precision Engineering and Manufacturing-Green Technology. 10. 1-11. 10.1007/s40684-022-00449-5.

[4] Y. Guo, Y. Sun, K. Wu (2020) *Research and development of monitoring system and data monitoring system and data acquisition of CNC machine tool in intelligent manufacturing,* International Journal of Advanced Robotic Systems March–April 2020: 1–12, DOI: 10.1177/1729881419898017

[5] J. Wan (2018) *A Study of Comparing CAD/CAM Simulated Machining Time with Practical CNC Machining Time and the Influence of the Parameter Setting in CNC*, Master Degree Thesis,

[6] Slade I.: *Obrada materijala II*, Tehnička škola Zagreb, 2018. [1]

called Check, the part that the tool must not process is marked. In addition to the above options, Esprit offers many more options that are used depending on the type of piece being processed. After defining the first and second operation, a 'code' is generated which is easily transferred from the program to the machine and as such is ready for use.

4. CONCLUSION

Modern processing technologies on CNC processing systems require professional management of the entire process, because otherwise there can be significant material damage, as well as time losses and stoppages in production, which leads to breaking delivery deadlines. Management of modern requirements of CNC processing using the CAD/CAM system has made significant progress, because a complete processing simulation can be performed. Also, control units of CNC machines provide the possibility of simulating the path of the tool. If all the simulation parameters are defined in the CAD/CAM system, that is enough, and during the actual production, it is necessary to ensure the appropriate number of workpiece controls and supervision of the CNC production systems.

Therefore, Esprit represents one of the excellent examples of a software CAD/CAM system application solution, because it offers a complete simulation of what is done on the machine, and in the milling process it has a wide range of defining tool parameters and processing methods.

Corresponding author:
Sanela Hrnjica
University of Bihac
Faculty of Mechanical Engineering
Email: hrnjicasanela393@gmail.com
Phone: +387 63 505 795