COMPARARTIVE STUDY BETWEEN PROGRAMMING IN A DESIGN PROGRAM AND A CNC PROGRAM

Adrian-Marcel MORARIU, Dan Ovidiu GLAVAN, Elena Stela MUNCUȚ, Ioan RADU, Andrei KOMJATY

Abstract: *The contribution of this paper is to present the processing operations of an aluminum work piece, which will be performed on a CNC with 3 axes. The main part is the difference between manually programming a machine and programming it using CAD / CAM software on a computer. To be successful, we need basic knowledge, such as the composition and strength of the tools used and the material to be processed. After acquiring this knowledge, the operations will be programmed to obtain from a raw material, the work piece according to the requirements of the technical drawing.*

Key words: *CNC, Processing, Programming, Advance, Speed, Design.*

1. INTRODUCTION

This paper presents the steps to be followed in a machining process, the tools needed for execution, as well the CNC that is used will be presented, the programming on it and then the programming with the help of a design program, [1].

The paper is organized as follows: Section 1 – The material we use is presented, Section 2 is about the design of the part and the tools used. The programming using a design program is described in Section 3 and the manual programming on CNC is in Section 4. Conclusions are presented in Section 5 and the References in section 6.

1.1 THE CHARACTERISTICS OF THE MATERIAL TO BE PROCESSED

First of all, some of the characteristics of the aluminum material must be mentioned. It will be an F34(7020) alloy, which is composed of AlZn4.5Mg1, this alloy corrodes extremely quickly in the air, covering itself with a thin but strong layer of oxide that protects the metal from further corrosion. Its hardness is 105HB, and the temper of it is T6 according to EN515, which means that after extruding the aluminum from the extruder, it is cooled by water to achieve higher hardness.

The modulus of elasticity is approximately 70GPa, the density is 2770 kg/m^3, the melting range is 600-650ºC and the thermal conductivity is 130-160 W/m.K. It is an alloy suitable for cutting, milling operations and a HSS (High-speed steel) tools will be used to obtain the cuttings. The industry uses it in a wide range, this is the reason why this composition of aluminum was chosen, [2].

1.2 THE EQUIPMENT AND DEVICES USED

The used CNC is a FP 4 NC Deckel semi-automat 3-axis milling machine, which travels 550x420x310mm. The table size is 900x430mm. The milling feed rates are between 2 – 6000 mm/min. The rotation speed range is 0 – 5000 RPM. The operating voltage is 380V – 50-60Hz, the rated current 20/22/24 A, control 42/48V and the maximum supply line is 35Atr. (Average true range). The tool holder is a SK 40 DIN 69871 model. The power of engine is

7.5kW at 100% of usage and the total weight of the machine is 5000 kg.

It contains an electrical hand wheel, with the help of which manual movements can be performed on the cutter machine. The tools are changed manually, [3]. The head of the cutter moves on the Y axis, and on the X and Z axis the table. The lubrication oil used is Maccurat Z Mac D220 and the hydraulic oil Energol HLP N46. During execution, the tools used are cooled with water mixed with emulsion. The G and M codes are used to write programs on the CNC and there is also the option to send the programs via cable connection created on a PC with the help of a computer-aided design (CAD) program, which contains the computer-aided manufacturing (CAM) soft.

The clamping device of the part will be a hydraulic vise. Its dimensions are 570x170x150mm and the weight 65 kg, and the most important thing, the clamping force, which is 6000 Kgf, Figure 1 shows the working mode of the hydraulic vise used on the machine, [4].
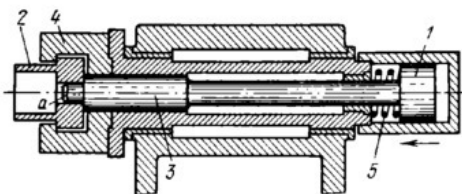

Fig. 1. The hydraulic vise

When the piston 1 moves to the left, by the action of the fluid, the work 2 is blocked. The work 2 is placed on the pin of the piston rod 3.

Here the yoke 4 is the stationary jaw, and the rod 3 the mobile jaw. As the piston 1 is moved to the right to the spring 5, the work 2 is released.

2. DESIGN AND TOOLS

The initial size of the semi-finished product is 294x100x80 and, as can be seen in Fig. 3, the final size will be 289x98.5x75.

Steps to follow when choosing the material: in this work is used a composition of Aluminum F34. There is a material library of the design program, where you can find a wide range of materials. In Fig. 2 you can see a browser, where is selected the composition we want to work with. Also here you can add some sorts of materials if you want to.

Table 1

Material browser
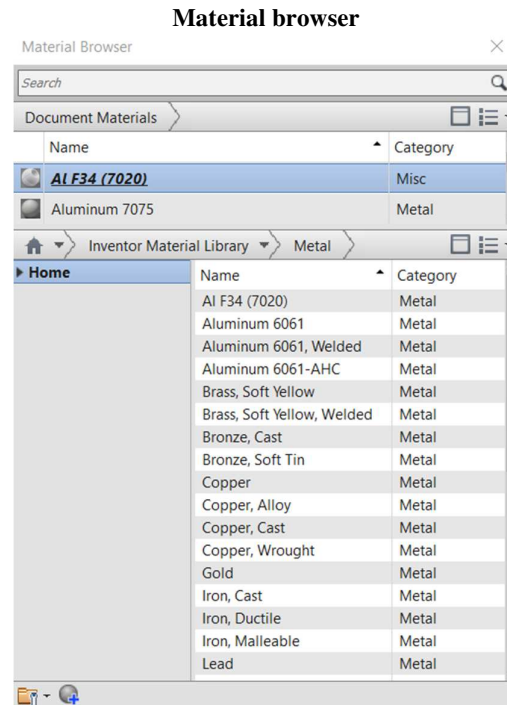
A design program will be chosen, in which the sketches of the piece are created. These sketches are extruded, so we will get the part in 3D. Fig. 4 shows the designed piece which was chosen to do a comparative study between the two programming methods.

This piece of aluminum is designed in a way that it gives the chance to present on it a face milling, contour milling, pocket milling, centering, drilling, boring and threading operations.
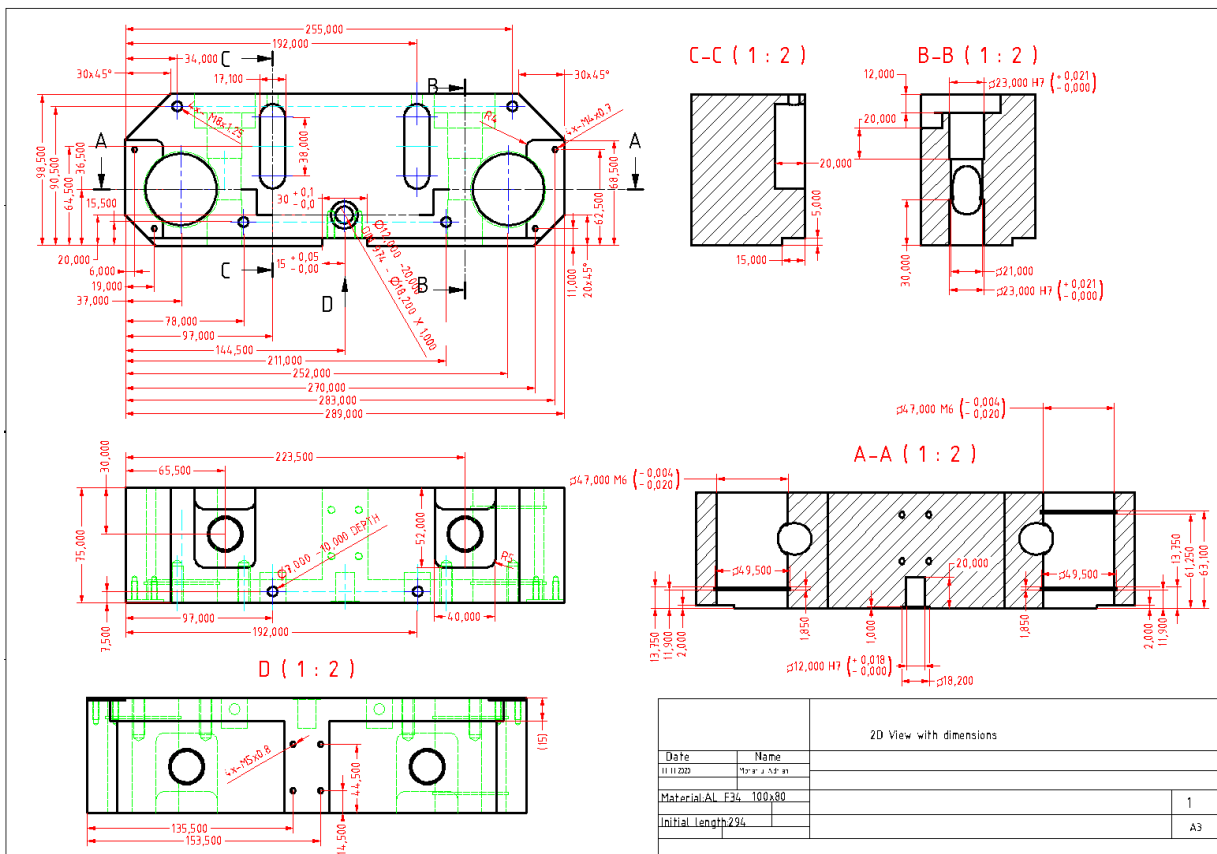
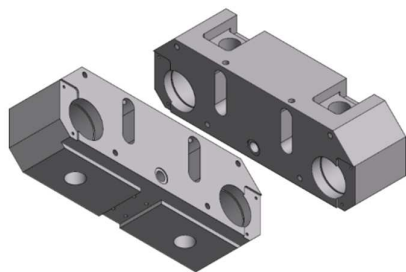**Fig. 2.** Execution drawing of the part

Fig. 3. The final look of the part; illustrated by two isometric views

2.1 THE TOOLS USED IN PROCESSING

The tools used are all HSS (High Speed Steel) compositions, although carbide tools last longer and have a better high-heat tolerance, the HSS tools are easier to sharpen and are cheaper. For aluminum machining this is a good choice. A typical composition contains 4% chromium, approx. 6% tungsten, approx. 2% vanadium, 1% carbon, cobalt up to 9% and molybdenum up to 10%. Chromium prevents scaling and improves

hardening, molybdenum improves hardness and cutting efficiency, as well as hardening resistance. Carbon is responsible for the basic hardness and increases the wear resistance, about 62-65 HRC. The tensile strength is up to 800 MPa.

The steps for choosing tools in the program: just like when choosing the material to be processed, there is a library here as well. The tool library is shown in figure 5. There is the possibility to select them from the program, where are a wide range of models with different diameters, also here you can find standard cones. You can also create new tools, where you can enter the name, type, length, thickness, number of edges, feed, rotation, etc. and also new cones can be created, where introducing certain dimensions will obtain its geometric shape.

The first tool is used to obtain the thickness and width of the part. The Sandvik Coromant milling tool is designed with a unique axial and radial positioning of the inserts, which allows

each insert to instantly cut chips efficiently without any insert adjustments.

In the present study, a model that has an entry angle of 45⁰ was used. It has five screwed cutting teeth and even allows the removal of 4-5 mm of material in a single pass.
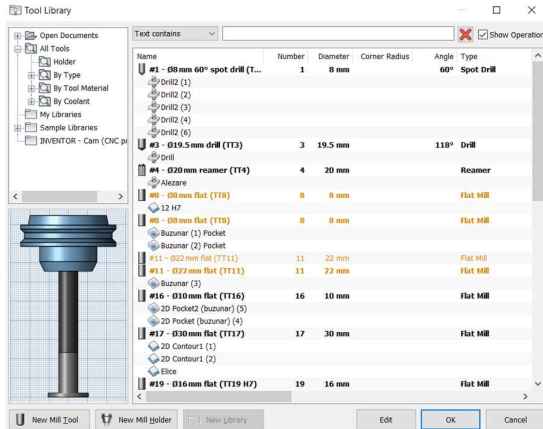

Fig. 4. Tool library

The flat end mills used for operations are Ø8, Ø10, Ø16, Ø22 and Ø30. Each of them is up-cut cutters, meaning the chipping is carried out upwards. Each of them has three cutting edges, as well as three flutes, which are the spiral grooves in the tool that allow the formation and evacuation of chips.

Fig. 5. The Sandvik Coromant milling tool

The smaller the number of flutes, the higher the flute wave, i.e. the empty space between the cuts is. This affects the strength of the tool, but allows larger chips, ideal for soft materials such as aluminum.

The Ø16 cutter will be used to create a tolerance of H7. It is a tool that is used strictly for finishing, to keep the diameter of the tool as long as possible at the nominal value and to obtain clean and balanced surfaces from all points of view.

The centerer is a tool with two cutting edges and is used to mark the holes according to the technical drawing, after which these markings will be drilled by the drills. The main purpose is to locate the hole accurately, the drills with a stiffer tip tend to slip.

The drills: The cutting angle of the tool is measured between the two main cutting edges. The smaller it is, the easier it is to center the material, this reduces the risk of slipping. Small angle means long main cutting edges, and in the case of large angles, there are short main cutting edges.

As with flat end mills, they have a system of channels that help remove the chip is done in a spiral during cutting. The wider the groove profile is, the better the absorption and removal of the chip will be.

The reamer is a conical or cylindrical rotary cutting tool that is used to finish / enlarge holes with exact dimensions. They are provided with straight, helical or longitudinal grooves. Cutting occurs on the sides of the tool or on the beveled edges at the tip of the reamer. The channels allow the lubricating and cooling fluids to reach the cutting edges and contribute to the passage of the chips. The manual ones are slightly conical to help entering the hole in a straight position.

The boring operation for finishing the holes of Ø47M6 will be performed with a tool which is actually a bar on which is screwed a turning insert with an entry angle of 80⁰. Its diameter is adjustable between Ø35 and Ø50 with a wrench. An addition on the wall from the previous tool is left and finally the hole is finished using the boring method.

The tap is categorized according to certain main parts, such as the type of cutting edge, the shape of the head, the cutting point, the profile of the blade, the flutes and the tail. With this tool the threads are created, in this work the metric system is used. As an example, at a hole of M4x0.7 a hole with a drill of 3.3 will be made, after which the thread of 4 with the step of 0.7 will be created. The working feed rate of this tool is calculated using the formula: rotation/minute x thread pitch.

The last tool presented will be similar to the boring bar, only the turning insert screwed on the

bar will be different, it is not used to finish the holes, but to create inner grooves at greater depths, these grooves are called Seeger channels, in which are inserted elastic springs for spring steel shaft.
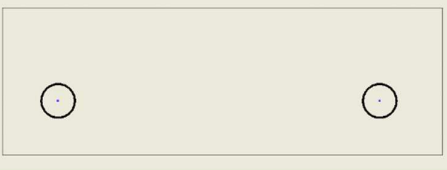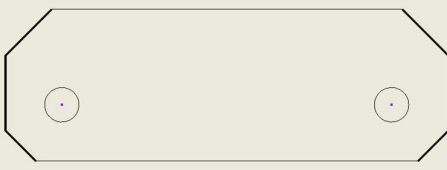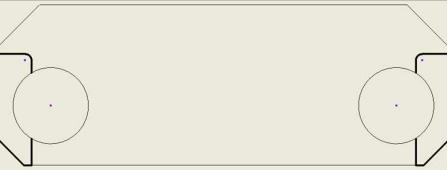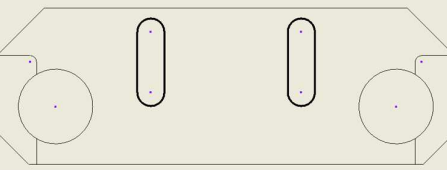
For a better understanding of the use of tools and processing technologies, respectively of programming methods, the table below shows the technological itinerary of the first setup. Here you will find information about milling and drilling methods, tools used and cutting regimes.
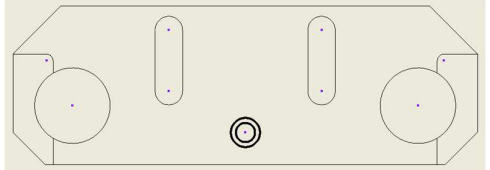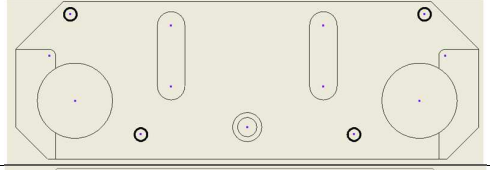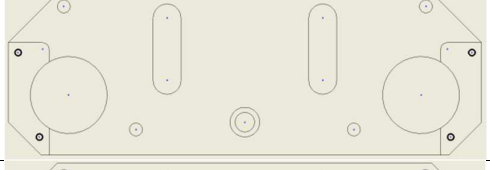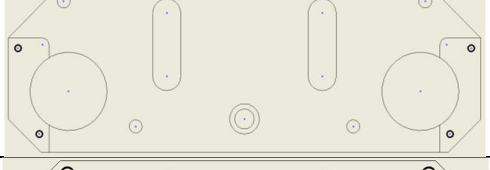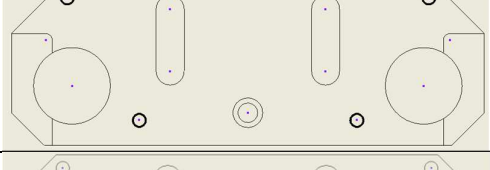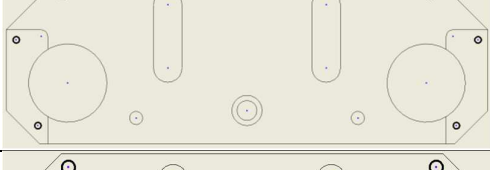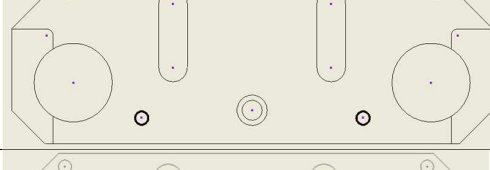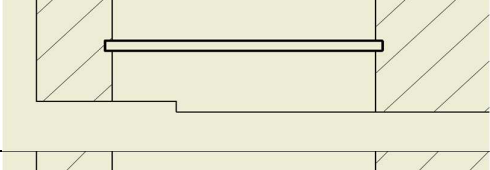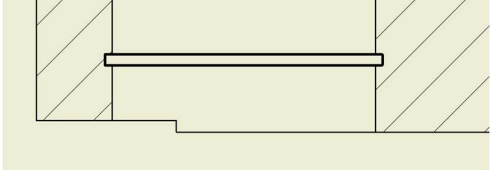
3. CREATING THE PROGRAM IN A GRAPHICS PROGRAM

This method is simpler than the manual method and more often used in today's industry. After acquiring knowledge of the operation of the machine tool, the proper use of tools and milling methods that can be performed depending on the processed material, you can start creating programs for parts in a design program.

Table 2

Technological Itinerary

Operation number	Operation type	Sketch	CNC type	Tool type High speed steel	RPM S (rotation/minute) - Feed rate F (mm/min)
1	Drill – Chip breaking			Φ22.5 Drill	S400 F80
2	2D Contour1 Length rough			Φ30 Flat end mill	S450 F175
3	2D Contour2 Length finish		FP 4NC Deckel	Φ30 Flat end mill	S450 F175
4	Helicoidal Φ46.7 rough			Φ30 Flat end mill	S450 F175
5	Pocket 1			Φ8 Flat end mill	S1800 F250
6	Pocket 2			Φ8 Flat end mill	S1800 F250

7	2D Contour3			Φ8 Flat end mill	S1800 F250
8	Drill – Rapid out			Spot drill	S1600 F175
9	Drill – Rapid out			Spot drill	S1600 F175
10	Drill – Chip breaking			Φ3.3 Drill	S1300 F130
11	Drill – Chip breaking			Φ6.7 Drill	S1000 F100
12	Tap			M4x0.7 Tap	S300 F = Step x RPM = 210
13	Tap			M8x1.25 Tap	S420 F = Step x RPM = 525
14	Boring Φ47 finish			Fly cutter	S1000 F100
15	2D Contour4			Fly cutter for seeger channel	S1350 F175
16	2D Contour5			Fly cutter for seeger channel	S1350 F175

First of all, it will be explained what Post Process means, an option which is present in every design program. It makes the connection between creating the G and M coded programming language, and creating this program in Inventor. Basically, the programmer creates the tool paths that will be selected on the 3D part, after which Post Process option will be done to these operations and the design program will generate the program that can be transferred to the CNC machine. There are many programming languages, in this case the Deckel Dialog 11 will be selected.

Each operator must receive a sheet with the operations for the part, containing the tools, with their advances and speed of rotation, the reference point marked, respectively the estimated time for creating the piece.

This worksheet contains all the information needed to adjust the part in the vise and to introduce the tools used.

Each tool appears on the execution sheet, where the name of the tool is presented in the program, what model of cutter is used, their diameter, the entire length of the tool, the number of cutting edges. After that, there is the maximum lowering of the tool on the Z axis, the maximum feedrate and the maximum rotation speed recommended by the programmer, [5].

At the beginning of any program, the first step is to create the Setup of the workpiece. Here in general the reference points will be selected, the additions on the piece will be noted.

Then we are going to create the operations. In this phase, we must first select the milling model we want to make, which can be contour, pocket, drilling, reaming and so on. We will select the tools from the tool library, which was mentioned in section 2.1, here we can find details about the tool, such as its name, the coolant used, whether the change is manual or automatic, diameter, length, tool type, number of blades, rotation, feeds used and so on. Also, depending on the selected operation, the next step will be to select on the part the areas we want to process, after which the tool heights will be entered, such as retract clearance height, retract height, feed height, top height and bottom height .

The first operation selected will be the drilling, the piece will be drilled with a Ø22.5

drill in the center of the Ø47M6 holes. The top height will be selected as the model top, so the tool will descend with a fast advance to 2 mm above the piece, after which it will start working with working feed to the bottom height, where the bottom model is selected -10, so the drill will exceed the piece by 10 mm.

There is one more step to take at the end, namely selecting the drilling method, chip breaking - partial retract or deep drilling - full retract. The drill is Ø22.5 diameter and has grooves approximately 90 mm from the edge, which means that it will allow the chip to be evacuated, it exceeds the thickness of the piece, so the chip breaking method has been used.

These two holes are created to allow the evacuation of the chip that will accumulate when they are widened by the Ø30 milling cutter, thus reducing the risk of overheating or the risk of tool breakage. A 2D Contour operation with a Ø30 milling cutter follows, in order to mill the ends, the finish will also be made with this tool. The selection of milling areas - certain contour lines. In this operation, the 30x45º chamfer will also be created.

Compared to the drilling operation, in this case we have two more setting methods, Passes and Linking. The Passes will introduce the tolerances, the type of compensation left (climb milling) or right (conventional milling), smoothing tolerance, stock to leave if necessary, multiple depths, finishing passes, feed optimization and roughing passes.

The tolerance entered will be 0.01 mm, it is used to obtain the highest possible accuracy. Using the conventional method, the chip will be thicker and became thinner at the cutting edge, which means that it will heat up more the chip and less the work piece.

The Smoothing tolerance is used to reduce the size of the code in the program without damaging the milling accuracy. As an example, if we have collinear lines, they will be changed into a single line and the tangent arcs will be replaced with multiple lines at the curve areas.

The Feed Optimization specifies that the feed should be reduced to corners. This can be an advantage to increase the life of the tool as long as possible. Roughing passes is used, if we have a milling contour, and we have a larger amount

of add-on per part, this method is activated, the maximum stepover of the tool will be introduced, so it will gradually eliminate the amount of material introduced at Maximum stepover, until it reaches to the contour selected by the programmer.

In the next part, I will present some examples of 2D Milling which are used in my program:

• 2D pocket - is a cutting operation with options for creating a finishing pass. The processing path will resemble the shape of the processed space. The processing area can be selected as in 2D adaptive compensation. Includes options for tapered walls and inlet positions before drilling.
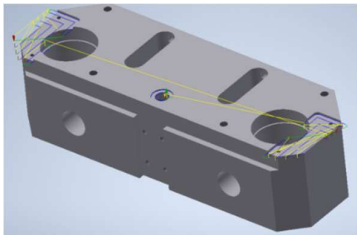

Fig. 6. 2D Pocket

• 2D contour - the 2D contour allows the processing of profiles. The processing area can be selected exactly as mentioned above. It is usually a finishing operation, but can also be used to make several splinters.
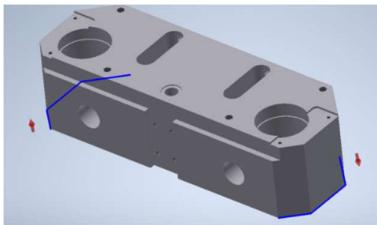

Fig. 7. 2D Contour

• Thread - threading is used for milling threads in a hole with straight or tapered walls. Select any internal or external circular face to create single or multiple threads.
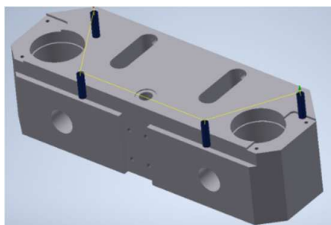

Fig. 8. Thread

• Circular - is used for milling cylindrical holes and drilling. Only circular faces can be selected for this route.
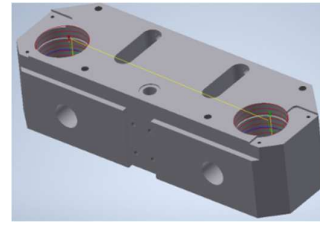

Fig. 9. Circular

4. CNC PROGRAMMING

Every programmer first usually learns manual programming on the mill, then moves on to programming on a PC.

Each CNC operator / programmer is familiar with the G and M codes, which are used to write programs on the machine, which are the basis of these programs, with which you can execute the work piece, accompanied by numbers, each with a certain meaning. These codes can also be found on the internet, for those interested. It would be more important to present some models of their use. The examples are from a FP 4 NC Deckel machine - Dialog 11, and were tested and used on the machine.
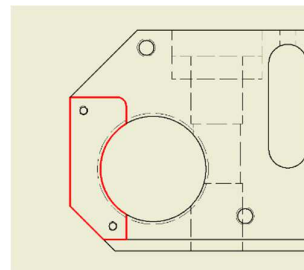
Fig. 10. Pocket milling marked with bold lines

G87 * 9 OA0 TA-2 MI1.9 AM7.5 SA2

-after which we are going to send the tool with the help of a correction to the dimensions given on the drawing.

- G87 * 9 is a code that is used to make pockets of this kind, as seen in the drawing, there is also * 1, * 2, for channel or circle type pockets.

- OA0 is the dimension from where TA is calculated, i.e. the descent in the piece from the zero point taken with the taster above the piece.

- TA-2 marks the total entry in the part on the Z axis.

- MI1.9 marks the maximum entry at a depth in the part. In this case, TA-2 MI1.9 was used because this way the first milling cutter will enter the piece only 1.9 mm, after which the pocket is repeated at a depth of 2 mm, thus obtaining a finishing on the bottom.

- AM7.5 is usually the tool diameter -0.5 tenths, in our case the milling is done with an Ø8 flat end mill. This AM indicates how much of the tool diameter is used for a milling pass. Processing aluminum, in this case, subtracting 5 tenths of the diameter, we work with 7.5 mm from the diameter of the tool
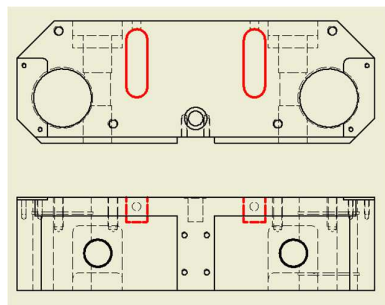

Fig. 11. Pocket milling, model 2

G87*1 OA0 SA2 TA-20 MI9.9 AM9.5 G3 R8.55 LI17.1 BI55.1

- G87 * 1 is used in pockets for channels that, for example, do not penetrate the entire part and the diameter of the cutter is not large enough to use just a simple contour milling.

- G3 is the counterclockwise model, it is most often used in the case of internal milling on the part.

- R8.55, radius at both ends of channels, the width divided by 2

- LI17.1 is the width of the channel.

- BI55.1 is the total length of the channel.

Using these codes, the path of the mill will be generated, it will start from the middle of the channel, will make a smaller channel, after which it will approach its contour up to the dimensions given in the program. This model is used, because the Ø8 milling cutter does not include the width of 17.1 even in two passes. These channels will be milled from two depths, as seen in the code above, the tool enters the piece 20 mm in total, 9.9 mm at a time. Thus, the second depth will be 19.8 and the last one will finish on the bottom as well on the wall.
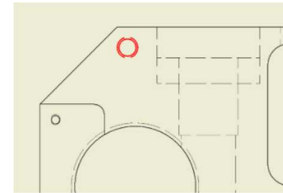

Fig. 12. Drilling

G82 OA2 TA-28 MI1 HI0.1 EA4

- G82 and G83 are used for drilling. G82 is the "chip breaking" method, after every millimeter penetrated into the part the drill does not rise above the part, but rises only as much as it is mentioned in section HI. It is used for holes where its depth does not exceed the length of the grooves in the drill. G83 is deep drilling, where the drill after each millimeter penetrated will rise above the piece with rapid advance, also with rapid advance will descend to HI, i.e. a few tenths compared to the previous penetration, after which it enters again into the piece MI mm.

- EA4 is the first quota to which the tool advances fast.

G84 OA2 TA-24 ST1.25

-G84 is used for tapping, here already appears ST (step) in the thread pitch, it is introduced because by entering the rotation speed of the tap the CNC will calculate the advance based on this step, as I mentioned this in the tool table at Chapter 2:

$$F(Z) = ST x RPM \qquad (1)$$
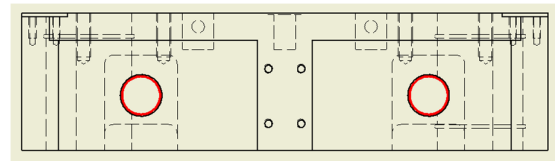
Fig.13 Reaming

G86 OA2 TA-100.5 EA4

5. CONCLUSION

Table 3

Results comparison

Programming		
	Manual	**Design program**
Time	For all 3 operations:	
(qualified programmer)	≈50 min	≈ 20 min
Personnel	It is recommended that each CNC has a person who knows	Only one qualified programmer and several

	programming and adjustment.	operators are required (in a smaller company).
Productivity	Lower productivity	Increased productivity
	Comparing the programming time and the number of employees, respectively training time.	
Financially	The design program method is more advantageous, due to the reasons mentioned above.	
Roughness	It is the same, the programming method does not affect the roughness of the part. Feeds and speeds count.	
Efficiency	It can be deduced that manual programming is more inefficient than programming in a design program.	

6. REFERENCES

[1] Instructions for using the CNC. Programming, adjustment, execution
[2] Glavan, DO; Babanatsas, T; Glavan, AI; Babanatis-Merce, RM; Radu, I; Gaspar, MC, *Study of the manufactoring precision on turning machine with inclined bed frame in real time of processing*, ACTA TECHNICA NAPOCENSIS SERIES-APPLIED MATHEMATICS MECHANICS AND ENGINEERING, Volume: 62, Issue:1, Pages:87-90, Published: MAR 2019, Cluj Napoca.
[3] Glavan, DO; Babanatsas, T; Ciupan, C; Babanatis-Merce, RM; Radu, I; Gaspar, MC *Processing precision on turning machine with inclined bed frame*, ACTA TECHNICA NAPOCENSIS SERIES-APPLIED MATHEMATICS MECHANICS AND ENGINEERING,Volume:61, Issue:4, Pages:561-566, Published: NOV 2018
[4] Gaspar, MC; Pikkarainen, A; Celorrio-Barrague, L; Glavan, DO, *online learning environment tools in higher education: contributions to their integration in engineering courses,* 12TH INTERNATIONAL TECHNOLOGY, EDUCATION AND DEVELOPMENT CONFERENCE (INTED), Edited by: Chova, LG; Martinez, AL; Torres, IC, Book Series: INTED Proceedings, Pages: 7331-7338, Published: 2018, ISBN:978-84-697-9480-7**,** Valencia, SPAIN
[5] Liu, GJ; Fu, CH; Wang, C; Liao, YH, *Software Tool Used for Transformer Design in AutoCAD Environment***,** MECHATRONICS AND INTELLIGENT MATERIALS III, PTS 1-3, Edited by: Chen, R; Sung, WP; Kao, JCM, 3rd International Conference on Mechatronics and Intelligent Materials (MIM 2013), XiShuangBanNa, PEOPLES R CHINA, MAY 18-19, 2013, Book Series: Advanced Materials Research, Volume: 706-708, Pages:1963-+, DOI: 10.4028/www.scientific.net/AMR.706-708.196

Studiu comparativ între programarea într-un program de proiectare și pe CNC

Rezumat: Contribuția acestei lucrări este de a prezenta operațiunile de prelucrare a unui material din aluminiu, care vor fi realizate pe un CNC cu 3 axe. Partea esențială este legată de diferența dintre programarea manuală a unei mașini și programarea acesteia utilizând software-ul CAD / CAM de pe un computer. Pentru a reuși, avem nevoie de cunoștințe de bază, cum ar fi compoziția și rezistența sculelor utilizate și despre materialul care va fi prelucrat. După dobândirea acestor cunoștințe, operațiunile vor fi programate pentru a obține dintr-o materie primă, piesa conform cerințelor desenului tehnic.

Adrian-Marcel MORARIU, Industrial Engineer, PAB Production, Email: adrianmorariu84@gmail.com, Phone: 0753012255, Home Address: str. 10, nr. 46, Romania, Sânpaul, Home Phone: 0257416447.
Dan Ovidiu GLAVAN, Conf. Dr. Eng., "Aurel Vlaicu" University Arad, Faculty of Engineering, Department of Automation, Industrial engineering, Textile production and Transport, E-mail: glavan@fortuna.com.ro, Phone: 0257-283010.
Elena MUNCUT (WISZNOVSZKY), lecturer, "Aurel Vlaicu" University Arad, Fac Eng., Dept Automat Ind Eng. Text Prod & Transport, Romania, Arad, Email: muncutstela@yahoo.com, Phone: 0746491247, Home Address: str. Dorului, Bl. B8, Ap.10, Romania, Arad.
Ioan RADU, Conf. Dr. Eng., "Aurel Vlaicu" University Arad, Faculty of Engineering, Department of Automation, Industrial engineering, Textile production and Transport, E-mail: raduioanuav@gmail.com .
Andrei KOMJATY, lecturer, "Aurel Vlaicu" University Arad, Fac Eng., Dept Automat Ind Eng. Text Prod & Transport, Romania, Arad, Email: komjatya@gmail.com, Home Address: str. Stefan Luchian, nr. 23, Bl B-17, Romania, Arad, v, Home Phone: 0357803486.