ISSN: 2687-6043
e-ISSN: 2687-6035
https://dergipark.org.tr/en/pub/wie

Volume 3, Issue 2
2021
pp. 1-11

Wood Industry and Engineering
International Scientific Journal

EVALUATION OF TOOL PATH STRATEGIES IN CNC WOODWORKING MACHINES AND A CASE STUDY

Özer Özçelik[1], Küçük Hüseyin Koç[2]

Citation

*Özçelik, Ö., Koç, K.H. Evaluation of Tool Path Strategies in CNC Woodworking Machines and A Case Study. **Wood Industry and Engineering**. 2021; 3(2): 1-11.*

Keywords
CAD
CAM
CNC Woodworking
Tool path

Paper Type
Research Article

Article History
Received: 07/12/2020
Accepted: 08/01/2021

Corresponding Author
Özer Özçelik
ozerozcelik@trakya.edu.tr

Phone: +90-284-2240283
Fax: +90-0284-2240288

Abstract

Since computerized numerical controlled (CNC) machines began to be used in the manufacturing industry, accurate determination of tool path strategies has always been an important problem area. In addition to the increasing design expectations especially in products and parts, developments in CNC machines have increased the efficiency expectation in tool path applications. The tool path can also vary according to the nature of the work and the geometric structure needed. Different tool path strategies can be used depending on product structure and machine characteristics in different sectors. The geometry of the workpiece to be machined, especially designed for machining complex shaped parts, plays an important role in determining the cutting tool path and tool to be used. In addition, the preferred tool path strategy raises issues such as reducing manufacturing costs, reducing manufacturing time and improving surface quality. In this context, the parameters such as the form of the workpiece, the quality of the workpiece, the characteristics of the cutting tool, the machine characteristics, the cutting conditions are among the main factors to be considered in the tool path evaluations.

In this study, tool path strategies for a sample piece of product were evaluated in a CNC woodworking machine. A three-axis CNC machine was chosen for the application. For this purpose, first of all, product part is modeled in CAM program and tool path strategies are developed on product design. These strategies were applied on CNC woodworking machine and the results of preferred tool path strategies were evaluated with comparative analysis. As a result of the research, the approaches that can be applied in determining the correct way of team strategies and the benefits that can be obtained are discussed.

[1] Trakya University, Department of Material and Material Processing Technologies, Edirne, Turkey
[2] Istanbul University-Cerrahpaşa, Department of Forest Products Engineering, Istanbul, Turkey

EVALUATION OF TOOL PATH STRATEGIES IN CNC WOODWORKING MACHINES AND A CASE STUDY

Özer Özçelik
ozerozcelik@trakya.edu.tr
ORCID: 0000-0002-2799-387X

Küçük Hüseyin Koç
hkoc@istanbul.edu.tr
ORCID: 0000-0001-6370-2016

1. Introduction

It is true that CNC machines have an extremely complex technology. On the other hand, it is very easy to learn and use the machine, even for those who have not yet met with the CNC machines. A person who has experiences on conventional machine tools, knows the principles of chip removal, four operations in mathematics, basic computer knowledge and the concept of coordinates, learns how to program and operate the machine after an average of 16 hours of training (Amasya University, 2013).

CNC machines, which are vital for the manufacturing industry, contain many complex issues that require long-term work. Some of these topics are cutting tool characteristics and selection, machine cutting parameters, axis movements, material properties and toolpath strategies. An innovation in one of these subjects, which have been studied and developed for years, causes the other areas to be affected in parallel. Toolpath strategies have also been continuously developing for years in parallel with the developments in other fields.

2. Materials and Methods

In the sample study, 18 mm medium density melamine faced fiberboard (MDF) was used as the material. In addition, a 2100x2800 mm flat and vacuum table CNC machine with tool change was used in the experiments. In the example application, a 5 mm diameter flat tip cutting tool is used.

The part to be used for the sample application on the CNC machine was modelled in both ArtCAM and Fusion 360 programs, as toolpath strategies of two different CAD / CAM programs were used. While creating the tool path strategies in both programs the cutting tool properties (t) speed (S) cutting speed (G1), idle movement speed (G0), cutting direction (M03), such as steps and steep side-step cutting parameters were kept the same.

For each tool path determined for the sample study, the workpiece was first modelled in the CAD program and then G code was created in accordance with this tool path. After the generated G codes were transferred to the machine, the sample was processed in the cutting parameters determined on the workpiece. After repeating this process for all the determined toolpaths, the data obtained were discussed in the findings section and their comparisons were examined through the table data.

3. CNC Technology

The general purpose of CNC machines is to give the desired shape to a material in the form of a raw material. Depending on the type of machine, the desired shape can be given as a result of tool movement, part movement, or tool and part moving together. CNC machines perform the desired operations with the programs in which these movements are shown with "G", "M", "S" and "T" codes. Programs consisting of "G", "M", "S" and "T" codes describe the paths the tool. Therefore, tool paths have an important place in CNC machines (Özçelik, 2006).

As with every system, CNC machines and systems have some advantages and disadvantages. These advantages and disadvantages are:

3.1. Advantages of CNC Machine Tools

- The adjustment time of the machine is very short when compared with some clamping molds, gauges etc. used in conventional machines.
- Loss of time caused by adjustment, measurement, control, manual movement, etc. is eliminated.
- Since the human factor is not very effective in manufacturing, serial and precise manufacturing is possible.
- Sometimes or no need for qualified people.
- Machine operations have a high precision.
- The working tempo of the machine is always high and the same.
- All kinds of consumption (electricity, labor, material, etc.) are minimized.

- All kinds of personal errors caused by the operator in manufacturing have been eliminated.
- The system is cheaper since expensive elements such as mold, gauge, template etc. are not used.
- Less space required for storage.
- The transition to new parts manufacturing is faster (Olam, 2012).

3.2. Disadvantages of CNC Machine Tools

- A detailed manufacturing plan is required.
- Requires an expensive investment.
- The hourly price of the machines is high.
- Compared to conventional benches, they require more meticulous use and maintenance.
- High quality cutters with high cutting speeds should be used.
- Periodic maintenance should be done regularly by experts and authorized persons (Olam, 2012).

3.3. Axes on 3 Axis CNC Machines

Cartesian coordinate system is used in the axis definitions of CNC machine tools. In this system, the axes are indicated by the letters X, Y and Z. The intersection points of these 3 axes are the origin (Amasya University, 2013).

The most commonly used method in axis definitions is the Right Hand Rule. In this rule, the thumb refers to X, index finger to Y, and middle finger to the Z axis. The intersection point of these three axes is the palm. The tips of all three fingers show the positive (+) directions of these axes and the opposite (-) directions (Figure 1) (Amasya University, 2013).

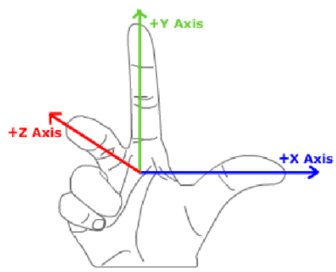

Figure 1: Right hand rule (Amasya University, 2013).

4. Team Path and Creation

4.1. Definition of Toolpath

If it is defined in general, the route that the cutting tool follows by removing chips on the workpiece is called the tool path. Many tool path strategies have been developed over the years according to the nature of the work to be done and the characteristics of the cutting tool to be used. There are certain stages in creating a tool path. First, the workpiece to be processed is modelled in a CAD (Computer Aided Design) program. The workpiece is usually modelled in 2D or 3D. The tool path strategies used in 2D and 3D models differ. The workpiece with completed modelling is converted to G,S,M,T codes in order to be processed on the CNC machine. Different conversion processes are required on machines of different brands. Although the G,S,M,T codes that are required for CNC machines to work have universally the same meanings and functions, CNC machines produced by different brands use different converters (post process) because the software and machine features are different. Some machines automatically recognize drawings with the dxf extension and perform the conversion process themselves. Again, most of CNC machines used in the furniture industry allow minifix holes to be drawn with pre-drawn cover models or some jobs that require only linear cutting in 2D with their own drawing program and create a tool path.

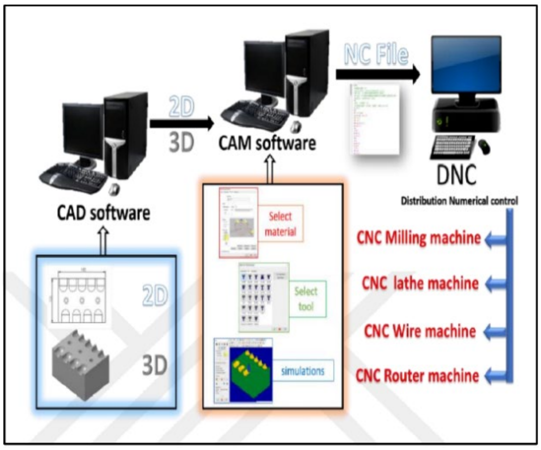

Figure 2: Process stages of CNC machine from design to production stage (Alabayed, 2018).

The features of the cutting tool are of great importance in determining the tool path. Features such as the quality of the cutting tool to be preferred, diameter, cutting direction, insert depth, tool length, heating and cooling properties, cutting speed characteristics, rotation speed feature and form are all factors that should be taken into consideration in tool path creation.

4.2. The Most Frequently Used Commands on A CNC Machine

Many commands are used in CNC machines. As the commands are determined according to the nature of the work done, the commands can be diversified and customized according to the sector where the work is performed. For example, since the material processed in the machinery industry is mostly metal, a certain heat occurs when it comes into contact with the cutting tool and this heat must be minimized for a qualified job. Therefore, CNCs with water, oil and air cooling are generally used in the machines used in the machinery industry. However, since wood is a soft material in the furniture industry, there is not much heat. Therefore, the cooling feature of the machines used in the furniture industry is either with air or not at all. This situation requires using additional commands on machines used in the machinery industry. The same is necessary in the furniture industry. The machine is designed differently for the drilling processes on the lateral surfaces required for minifix and similar fasteners in the furniture industry, and different commands must be used from other sectors to use the hole units on these lateral surfaces.

The most frequently used commands in the furniture industry are as follows;

G0: When the machine is not cutting, G0 command is used to move from one position to a different position. Under normal conditions, high speed motion is performed to avoid loss of time during cutting.

G1: It is the command used for the machine to make linear cutting by removing chips. After the command is entered, the machine continues by making a linear cut up to the X and Y coordinates.

G2: It is the command used for the machine to cut curvilinear clockwise. With the command, values such as X, Y, Z values and the diameter or radius of the arc to be cut, the center point, the X and Y distance from the end point of the arc to the center point of the arc are requested. It is necessary to write one or more of these values, which are specified according to the broadcast characteristics to be cut, into the program.

G3: This command works in the same way as the G2 command. Used for counter clockwise cutting only.

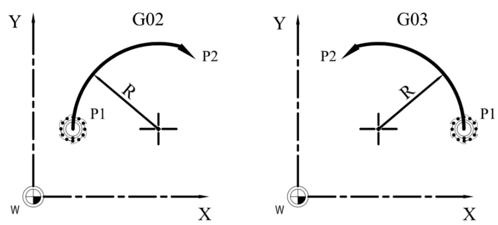

Figure 3: G2 and G3 circular interpolation direction determination (Programming in CNC Milling, 2006).

G17: It is used for the selection of the surface to be treated. The G17 command specifies to work on the top (XY) surface.

G18: It is necessary to work on the lateral surface in boring and orifice processes required for fasteners such as Minifix. Surface selection should be made in CNC machines that allow working on the side surface. The G18 command states that it will work on the ZX lateral surface.

G19: G19 command indicates that it will work on YZ lateral surface.

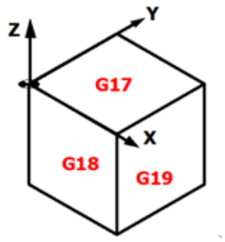

Figure 4: Planes in CNC machines

G90: It indicates that the G codes created for the workpiece to be machined are created in the absolute coordinate system and will be processed in this system.

G91: It indicates that the G codes generated for the workpiece to be machined are created in the relative (incremental) coordinate system and is processed in this system.

M03: Indicates that the spindle on which the cutting tool is mounted is rotated clockwise.

M04: Indicates that the spindle with the cutting tool rotates counter clockwise.

M30: It is used for the end of the program. When the work done by the CNC machine is finished, the program is written at the end. This indicates that the work is done.

T: Used for Tool Number. In some CNC programs, "A" command is used for the tool.

S: It expresses the number of revolutions of the spindle per minute.

5. Most Preferred Team Road Strategies

There are 2 types of toolpath tracking strategies in a 3 axis CNC machine. These strategies are 2D toolpath and 3D toolpath strategies. In fact, the toolpath strategies used in both drawing types basically work on the same logic. The difference of 3D toolpaths from 2D toolpaths is that curved surfaces cannot be machined in a 2D drawing. In a 3D drawing, on the other hand, as the drawing is three-dimensional due to the structure of the drawing, depending on the form of the drawing, the cutting tool moves by removing chips on the workpiece while moving up and down to different cutting depths (Z) on flat, angled and inclined surfaces. This makes toolpath creation for a three-dimensional model more complex and difficult. CNC machine manufacturers and CAD / CAM program developers generally use the same toolpath strategies. However, in recent years, especially CAD / CAM programs have developed new toolpath strategies according to the field of activity, the nature of the work, workpiece and cutting tool characteristics, cutting parameters. It was possible to see a different toolpath option in each CAD / CAM program. This study focused on 2D toolpaths.

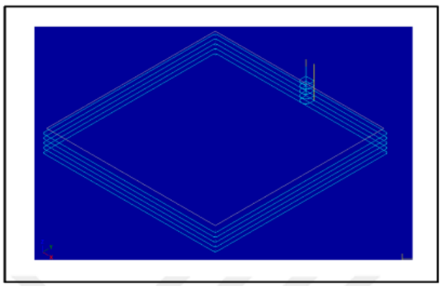

Figure 5: 2D Toolpath example (Alabayed, 2018).

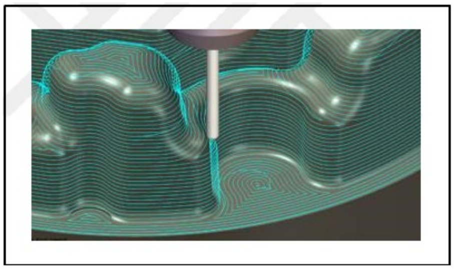

Figure 6: 3D toolpath example (Alabayed, 2018).

As mentioned above, basic toolpath strategies have been used for years. The basic logic of these strategies has changed little. The basic logic of the toolpaths is based on cutting or scanning the workpiece. The most preferred 2D toolpath strategies are as follows.

5.1 2D Toolpath Strategies

a. Zigzag Parallel Toolpath Strategy

In the zigzag strategy, the cutting tool continues in a zigzag from the starting point to the end point where it dives into the workpiece. Thus, it scans the area where it cut. If the contour of the scanned area is in a different geometric structure, the cutting tool can draw zigzags in accordance with this geometric shape. It can cut or scan clockwise or counter clockwise according to the cutting direction of the cutter blade.

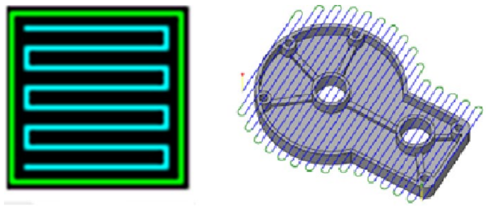

Figure 7: Zigzag parallel toolpath strategy

b. One-Way Cut Toolpath Strategy

In the one-way cutting strategy, the cutting tool cuts only in one direction. For example, in the Zigzag strategy, the cutting tool moves in the + X –X or + Y –Y directions, while in this strategy the cutting tool does not cut in the X direction if it cuts in the + X direction. The same is true for the Y axis.

Picture 8: One way cutting strategy

c. Parallel Spiral Toolpath Strategy

In the Parallel Spiral Tool Path Strategy, the cutting tool is cut by following a route in the form of nested squares from the outside to the inside or from the inside to the outside. The side step size used in the tool path varies depending on the tip form of the cutting tool. If cutting is done with a flat tip cutting tool, the side step is usually given at 50% of the tool diameter. If the cutting tool has an oval insert, the side step rate varies between 1% and 10%.

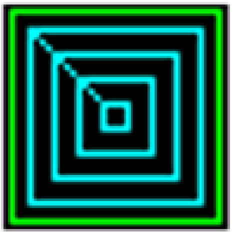

Figure 9: Parallel spiral toolpath strategy

d. Spiral Toolpath Strategy

The spiral toolpath strategy is the tool path that takes the center point of the workpiece as the center and it is in a spiral form around this center, from the outside to the inside or from the inside to the outside. Very effective results are obtained in the processing of workpieces with mostly circular forms.

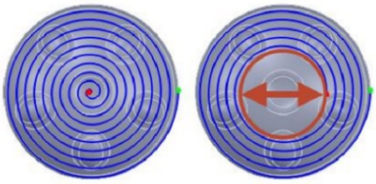

Figure 10: 2D spiral toolpath strategy (CAM Milling on 2½ Axis, 2019).

e. Outline Spiral Toolpath Strategy

This toolpath strategy is mostly used in workpieces with different contour forms. It is a tool path in a spiral form from outside to inside or from inside to outside according to the form structure of the contour shape. Since there is a seamless connection between the tool paths, the cutting tool only makes one dive and one exit into the material.

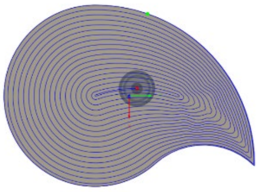

Figure 11: Outline spiral toolpath strategy

f. Drilling Toolpath Strategy

In the furniture industry, fasteners, shelf pins and holes that need to be drilled into the workpiece for different purposes can be drilled with this strategy. The hole diameter is drawn as it is in the design program and the CAM program automatically defines the hole diameter. In this strategy, both vertical and horizontal holes can be drilled. In order to be able to drill horizontal holes, the CNC table must be suitable for horizontal drilling and at the same time the CNC machine must have horizontal hole attachments.

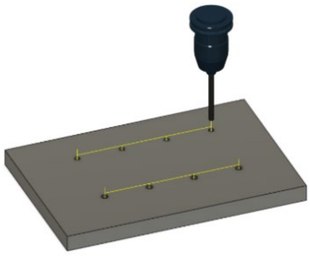

Figure 12: Drilling strategy

g. Writing Toolpath Strategy

With this strategy, text can be written in both 2D and 3D form workpieces. In this strategy, generally V-tip cutting tools are used.

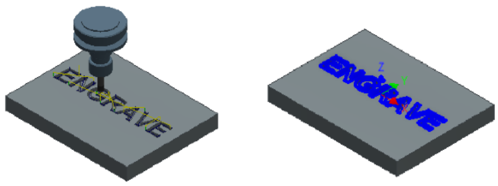

Figure 13: Writing strategy

6. Findings

The CNC machine used for example applications and the cutting tool used are shown in Figure 14.

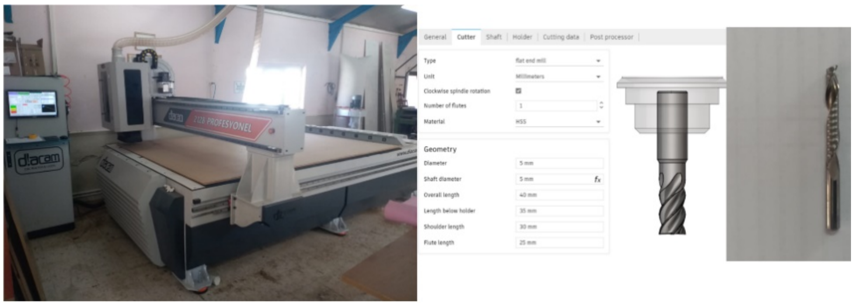

Figure 14: CNC machine and cutting tool features used in sample applications

The sample workpiece is designed as a part with different geometric forms in order to use different toolpath strategies. A total of five tool path strategies have been determined, and the CAM images of these tool paths and the results obtained after the workpiece is processed are as follows.

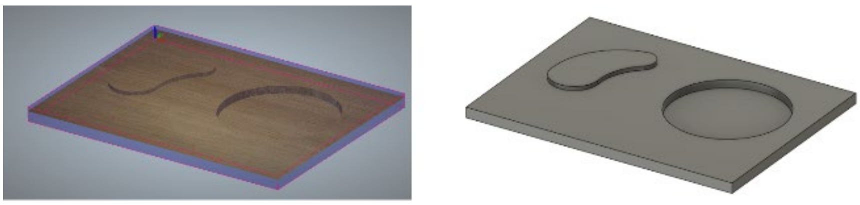

Figure 15: Sample part drawing in ArtCAM and Fusion 360

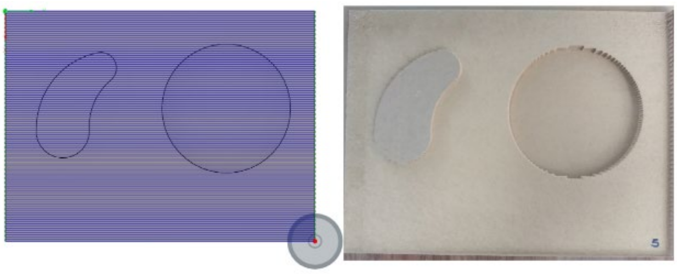

Figure 16: Zigzag parallel toolpath strategy

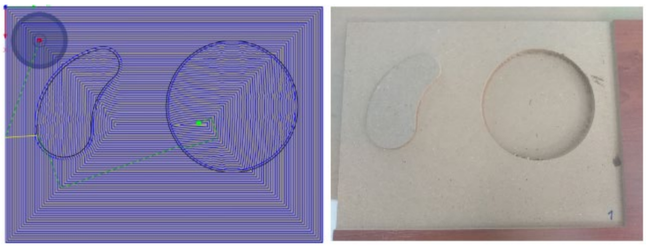

Figure 17: Parallel spiral toolpath strategy

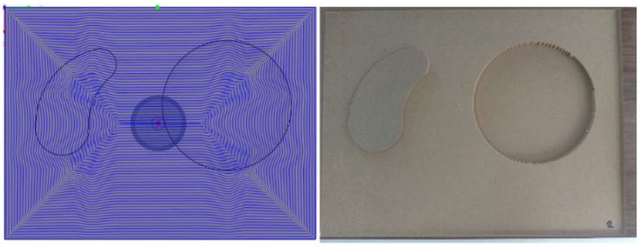

Figure 18: Outline spiral toolpath strategy

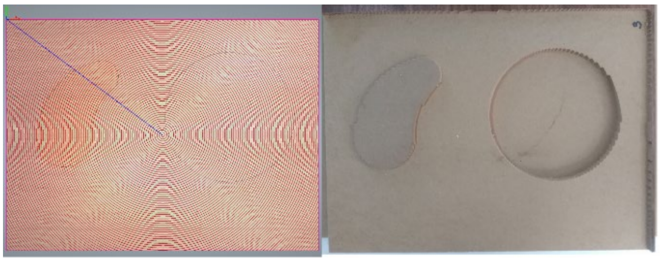

Figure 19. Spiral (classic) toolpath strategy

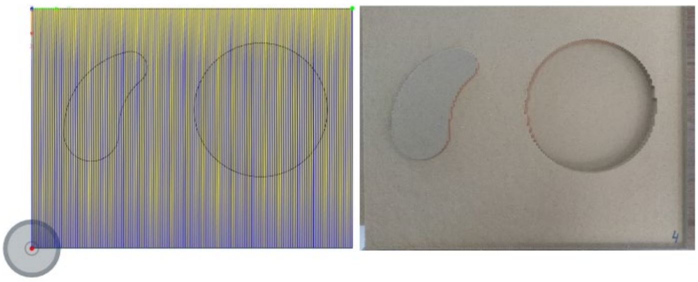

Figure 20. One way parallel toolpath strategy

A total of 5 toolpath strategies have been determined for the sample workpiece. In tool paths, machining time, cutting length, cutting speed, diving and exit speed of the cutting tool to the workpiece, steep step and side step distance were taken into consideration. The cutting speed, speed, blade diameter, diving and output speeds of the cutting tool to the workpiece and the side step and vertical step distance of the blade were kept constant. Below are the data obtained from the tool paths given in tabular form.

Table 1: Obtained Data of Tool Path Strategies

No	Toolpath Strategy	Processing Time	Tool Path Length	Cutting Tool Diameter (mm)	Turnover (S)	Cutting Speed (F) (mm / min)	Diving Speed (mm / min)	Output Speed (mm / min)	Upright Step (mm)	Side Step (mm)
1	Zigzag	11 dk 30 sn	38,8 mt	5	15000	4000	1000	1000	max.	2,5
2	Parallel Spiral	11 dk 31 sn	39,8 mt	5	15000	4000	1000	1000	max.	2,5
3	Outer line Spiral	16 dk 20 sn	57,9 mt	5	15000	4000	1000	1000	max.	2,5
4	Spiral (Classic)	15 dk 22 sn	60,6 mt	5	15000	4000	1000	1000	max.	2,5
5	One Way Parallel	14 dk 25 sn	77,6 mt	5	15000	4000	1000	1000	max.	2,5

7. Conclusion and Recommendations

In the studies, no surface roughness device and method were used to determine the surface roughness, and an opinion about the surface roughness was reached according to the eye examination. In addition, a 5 mm flat tip cutting tool was preferred for all sample applications and 18 mm medium density melamine coated fiberboard for the workpiece.

According to the data obtained as a result of the studies, the tool path strategy with the longest processing time is the Outer Line Spiral. This tool path is followed by Classic Spiral and One Way Parallel. The Zigzag Parallel and Parallel Spiral methods have almost the same processing times. Considering the margin of error of machining time, it can be said that these two toolpath strategies have the same machining time.

When the data is examined on the tool path length value, the toolpath strategy with the longest cut length is the One Way Parallel strategy. Since this tool path only cuts in one direction, every time it reaches the end of the cutting direction, it moves idle and returns to the beginning at maximum speed and starts cutting again. Because of the excess of these idle movements, the cutting length is the longest tool path.

The other longest toolpath in terms of toolpath length values is the Classic Spiral Method. The reason why the classical spiral method is one of the longest tool paths in terms of both processing time and processing length is due to the mismatch of the cutting strategy and the workpiece. Since the spiral toolpath strategy is based on a continuous spiral logic, it continues to spirally cut until the last metal removal command, regardless of the contour form of the workpiece. Since the workpiece used in practice has a rectangular form, after a point the cutting tool has to move freely on the short side of the rectangle. This causes unnecessary loss of distance and time. Therefore, while other toolpaths move only in areas where they remove chips, this tool path moves a significant proportion of the cut length in idle mode.

As a result of all evaluations, it has been observed that toolpaths with parallel machining strategy work in a shorter time and at a shorter machining distance than spiral strategies. It has been observed that the desired surface quality cannot be achieved in all tool paths, especially in the contours of circular shaped surfaces. A second cutting strategy has to be determined in order to smoothen these surfaces.

Although a single type and single-form cutting tool was used in the studies, more than one cutting tool and cutting tools with different end forms should be used according to the form of the workpiece in order to produce more qualified works. Apart from this, cutting strategies should be divided into roughing and finishing operations in order to prevent rapid wear of the cutting tool. However, this process does not only increase the surface quality but also increase the cutting time and cutting length.

While cutting the workpiece, minimizing unnecessary starts and dives of the cutting tool will positively affect the cutting time. While some CAM programs provide the user with diving and take off options, some of them automatically determine this feature.

The sharpness of the preferred cutting tool positively affects the surface roughness. In addition, the cutting tool should be preferred according to the cutting strategy to be made. For example, for a process that only be scanned, the choice of a large diameter cutting tool instead of a narrow diameter cutting tool would increase the side step distance, so there would be a positive improvement in terms of cutting time and cutting distance.

The revolution and speed properties of the cutting tool used should be well known and values should be given according to these features.

It is very important that the workpiece to be machined is stably attached to the CNC machine. It is very important for the quality of the work that the workpiece does not move while the CNC machine is cutting.

Many CAD / CAM programs are used in the industry. Some programs contain both programs, while some programs contain only one. It is very important to use the correct CAD / CAM program to determine the correct tool path according to the nature of the work to be done.

As a result of all these evaluations and literature researches, it has been observed that the toolpath studies specially developed for the furniture industry are not at a sufficient level. In this context, the necessity of scientific studies to develop toolpath strategies for the works that are used extensively in the furniture industry has emerged.

8. Acknowledgments

This study was presented as a full text at the VI. International Furniture Congress held by Karadeniz Technical University, Trabzon.

References

Amasya University (2013). Electronics and Automation Department, Mechatronics Program, Lect. Berkan Zöhra Unpublished Lecture Note.
Özçelik H., (2006). Tool Path Creation Algorithms, Yıldız Technical University, Institute of Science, Mechanical Engineering USA, Master Thesis.
Olam M., (2012). Design of Computer Aided Milling Machine to be Used in Furniture and Decoration Industry, Fırat University, Institute of Science, Mechanical Engineering Department, Master's Thesis.
Alabayed M.A., (2018).‘‘Investigation Of Optımal Tool Path Strategies For Milling Process’’, The Graduate School Of Natural And Applied Sciences Of Karabuk University, M.Sc. Thesis.
Programming in CNC Milling, (2006). MEB Machinery Technology, Megep Lecture Notes. CAM Milling on 2½ Axis, (2019). MEB Machinery Technology, Megep Lecture Note.