PEAT

Machining Analysis on a Pin Chuck by using Mastercam Software

Mohamad Izzat Azahri[1], Haffidzudin Hehsan[1], Muharam Rafidi Jaafar[1]*

[1]Department of Mechanical Engineering Technology, Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, Pagoh, Johor, 84600 MALAYSIA

*Corresponding Author Designation

Abstract: Machining is a vital manufacturing process that is used in a variety of applications. It is the process of removing material from a workpiece using power-driven machine tools to shape it into the desired design. A cylindrical shape is a product that needs to be transformed into a pin chuck design. The significance of this research is to study more about the Mastercam software and the machining analysis of the pin chuck. The Mastercam software can design and develop a simulation for the product. This study was conducted to design the pin chuck, simulate the toolpath process using Mastercam software, and determine the relationship between spindle speed and machining time. Carbide is a cutting tool to be used to operate the cylindrical material. Furthermore, three parameters that influence machining time are spindle speed, feed rate, and plunge rate. In the simulation Mastercam software, these parameters were observed to see how they may affect the machining time. Moreover, the relationship between these parameters have not been yet established based on the simulation analysis. Mastercam software is used to complete the pin chuck process. There were numerous steps to create the product, including design, lathe machine operation, toolpath processes, and more. The analysis aims to investigate the relationship between spindle speed and machining time. The spindle speed has the greatest influence on the differences in machining time that occur. As a result, the spindle speed is inversely proportional to the machining time. It is because the higher the spindle speed, the machining time will be shorter. Last but not least, in terms of toolpath types processes, the finishing process has the shortest machining time followed by center drill, drill, facing, grooving and roughing. Lastly, Mastercam software was able to analyze the machining time of the product and successful simulate the toolpath process.

Keywords: Machining, Parameter, Toolpath, Mastercam, Machining Time

1. Introduction

Machining is a manufacturing term that refers to a variety of technologies and techniques. It is the process of removing material from a workpiece using power-driven machine tools to shape it into the desired design. The primary types of machining tools are as follows which are cutting tools, drilling tools, turning tools, milling tools, and others [1]. Each of the machining tools has its functions. Besides, CNC machining (computer numerical control machining) is a computer-aided technique that can be used in conjunction with a wide range of equipment. CNC Machining is subtractive manufacturing which producing a custom-designed part by removing layers of material from a stock piece known as the blank or workpiece using computerized controls and machine tools [2]. The terms of CNC stand for Computer Numerical Control while CNC Machining is subtractive manufacturing which producing a custom-designed part by removing layers of material from a stock piece known as the blank or workpiece using computerized controls and machine tools [3]. In addition, CNC machines such as mills, routers, lathes, and wire EDMs are all controlled by CAD/CAM software.

Moreover, Mastercam is a piece of software that combines CAD and CAM capabilities to efficiently drive CNC machines for maximum productivity [4]. Parts, fixtures, and assemblies are designed using CAD software. It can be used to design new parts, modify existing ones, or prepare parts for machining. CAM software is used to program toolpaths and generate G-code which is used to control CNC machines such as mills, lathes, and others. Although CAD and CAM software can be used together in the manufacturing process, they are two distinct types of software. CAD is used to create or modify an object, whereas CAM is used to control a machine that creates an object. Computer- aided design (CAD) is a technology that uses computer systems to aid in the creation, modification, analysis, and optimization of a design [5].

In particular, a pin chuck is a tool that is used to hold very small diameter drill bits or other materials that a standard drill chuck cannot hold [6]. A pin chuck is used with high-speed rotary tools. The pin chuck head is detachable, screwing on and off to allow for the insertion of the appropriate collet. The pin chuck is made from mild steel. Mild steel is carbon steel with low carbon content. It is also referred to as low-carbon steel. Mild steel is not alloy steel, so it does not contain a lot of other elements besides iron. Because its carbon and alloying element content are relatively low, it has several properties that set it apart from higher carbon and alloy steels [7].

Lastly, this study was conducted to design the pin chuck, simulate the toolpath process using Mastercam software and find the relationship between spindle speed and machining time. Carbide is a cutting tool to be used to operate the cylindrical material. Three parameters that influence machining time are spindle speed, feed rate, and plunge rate. In the simulation Mastercam software, these parameters will be observed to see how they may affect the machining time.

2. Materials and Methods

Mastercam provides CAD/CAM software tools for a wide range of CNC programming requirements which is from simple to complex. There are many kinds of the process that can be used in Mastercam software such as lathe, mill, drill, and others. This software can design 2-dimensional and 3-dimensional wireframe models and able to create a simulation for the product which can do facing, roughing, finishing, and so on. This software also can generate a G-code and M-code program to apply on CNC machines. CAD/CAM software eliminates errors that can occur when manually programming a CNC machine.

Figure 1: Mastercam software

The lathe operation process is used in methodology. When performing machining operations on the lathe, there are numerous types of toolpaths. It would begin with the processes of facing, roughing, and finishing. Other toolpath types include a groove, drill, and so on. The lathe face toolpath prepares the face of the part for further machining. Once the face of the part is clean, use it to set tools or determine tool offsets. The rough toolpath on the lathe allows for the rapid removal of a large amount of stock in preparation for the finish pass. Besides, a groove toolpath is a cut which able to forms a narrow cavity of a certain depth on external and internal surface. A drill toolpath allows the centers of selected to be drilled to a specified depth. Lastly, a finish toolpath is typically followed by a roughing toolpath.

2.1 Procedure

The flowchart in Figure 1 below shows the process of the product by using the Mastercam software from start until the end.
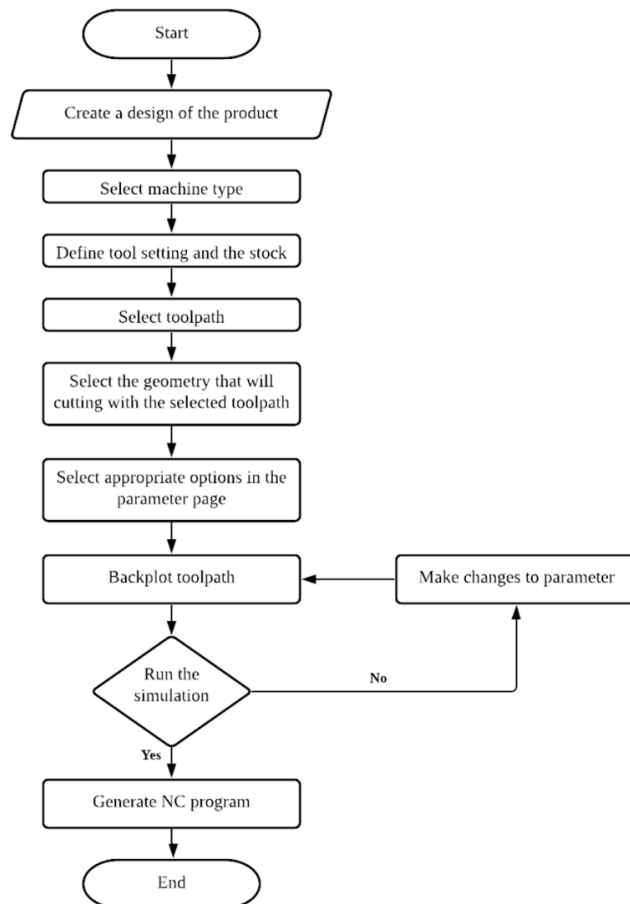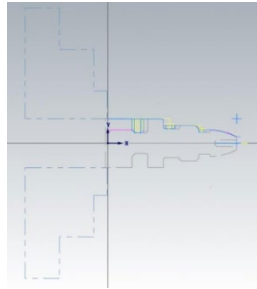

Figure 2 (a): The process of the product by using the Mastercam Software

A design of the product was created, which is a pin chuck, in Mastercam software. This is critical because if the design is not completed, the process cannot proceed to the next step. As a result, ensure that the design has already been prepared with a nice and proper sketch.

Figure 2 (b): Pin chuck design

Then, the machine operation that will be used was chosen. The operation that should be performed is a lathe machine based on the product, which is a pin chuck. It is due to the shape of a pin chuck in a cylinder. The product can be operated using a lathe machine.

Figure 2 (c): Machining Operating

Tool setting and stock was defined. This stock setup allows determining the size of the stock, the size and location of the chuck, the size of the tailstock, and the steady rest. After that, the appropriate toolpath was selected. A pin chuck product was created by using six processes. It would begin with the processes of facing followed by roughing, grooving, center drill, drill and finishing.
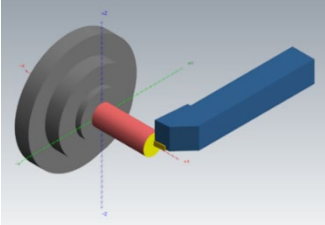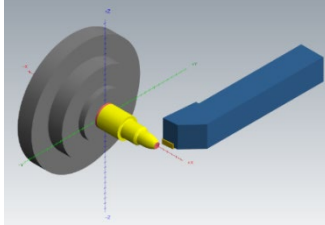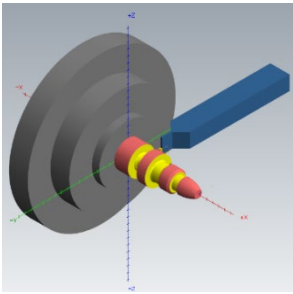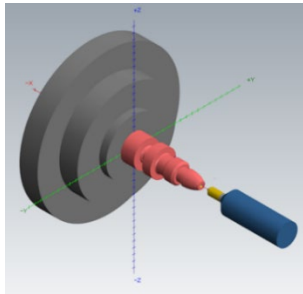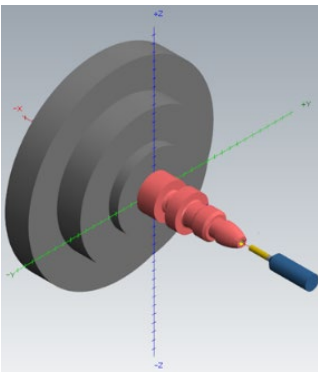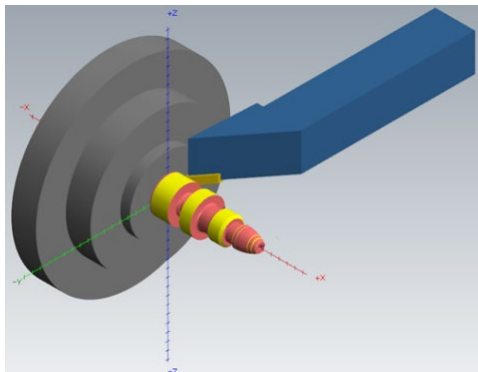

Figure 2 (d): Types of toolpath

Then, the toolpath was backplotted. Backplotting shows the path the tools take to cut the part. This display allows for the detection of program errors before the part is machined. It also displays any collisions between the workpiece and the tool. Next, the simulation on the pin chuck product was run. If the simulation does not run, something went wrong during the procedure. The parameters were changed. Then, the simulation was run again until it succeeds. Lastly, the post-processor was run to generate the NC program.

3. Results and Discussion

Machining is a collection of processes that involve the cutting of raw materials into a specific shape or the removal of certain materials. A simulation on a pin chuck was run to determine the objective of study. Therefore, the machining time of the pin chuck was analyzed and recorded on how much time is taken to produce the material by the process of toolpath.

3.1 The process operation of the toolpath on a pin chuck

Figure 3 (a): Lathe Face

- Prepares the face of the part for further machining.

Figure 3 (b): Lathe Rough

- Allows for the rapid removal of a large amount of stock in preparation for the finish pass.

Figure 3 (c): Lathe Groove

- A cut which able to forms a narrow cavity of a certain depth on external and internal surface.

Figure 3 (d): Lathe Center Drill

- Only drill the center of the product surface.

Figure 3 (e): Lathe Drill

- Allows the centers of selected to be drilled to a specified depth.

Figure 3 (f): Lathe Finish

- Produces a smooth surface finish.

There are three different processes that operate the pin chuck. The processes are general turning by using outer diameter rough right and outer diameter finish right, grooving tool by using outer diameter groove center (narrow), and for drilling tool by using spot tool 6 diameter and drill 3 diameter. All of the tools used in the operation are made of carbide. Besides, the yellow colour represents material that has already been cut and finished.
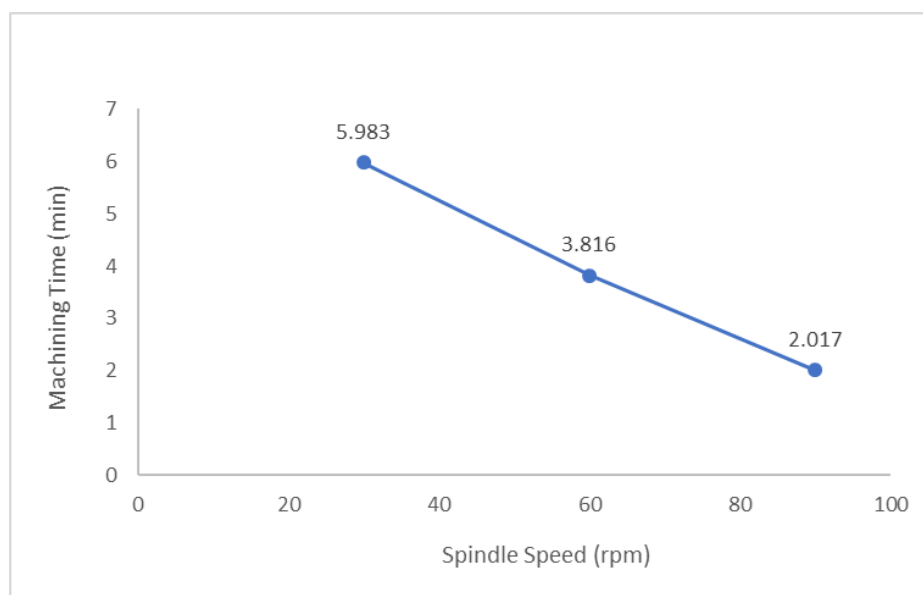
3.2 Spindle Speed Against Machining Time

Table 1 (a) displays three parameters and one output which are feed rate, plunge rate, and spindle speed, and machining time. A feed rate of 0.2 mm/rev was reduced for the lathe face procedure. Then it proceeded to cut at a 0.1 mm/rev plunging rate. Besides, three different spindle speeds were used 30, 60, and 90 RPM. The optimum speed for lathe face operating in Mastercam software is 90 RPM. The spindle speeds of 30 RPM and 60 RPM were only for comparison in order to monitor the time required for turning. The spindle speed (RPM) was used in the lathe face process because it is already automatic selected by the program of Mastercam software.

Table 1 (a): Cutting Parameter of Lathe Face

Feed Rate, (mm/rev)	Plunge Rate, (mm/rev)	Spindle Speed, (RPM)	Machining Time, (min)
0.2	0.1	30	5.983
		60	3.816
		90	2.017

The relationship between spindle speed and machining time is illustrated in Figure 4 (a) below. The spindle speed has the greatest influence on the differences in machining time that occur. When using 30 RPM spindle speed in simulation Mastercam software, the machining time is 5.983 minutes, while 60 RPM spindle speed takes about 3.816 minutes. The time required for machining is 2.017 minutes at 90 RPM spindle speed. The graph is arranged in descending order. As a result, the spindle speed is inversely proportional to the machining time. It is because the higher the spindle speed, the shorter the machining time. In the case of high-speed machining leads to a shorter production time and a higher productivity of the machining operation [8].
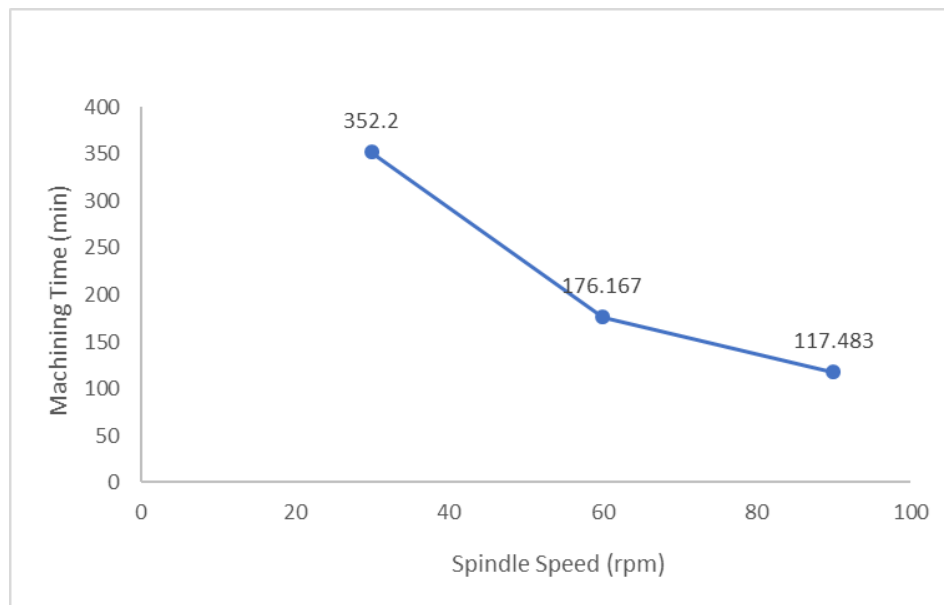

Figure 4 (a): The Relationship Between Spindle Speed and Machining Time

The cutting parameters of lathe rough are shown in Table 1 (b) below. For the lathe rough procedure, a feed rate of 0.2 mm/rev was used. It then proceeded to plunge at a rate of 0.1 mm/rev. Furthermore, three different spindle speeds (30, 60, and 90 RPM) were used. The spindle speeds of 30 RPM and 60 RPM were only used for comparison purposes in order to track the time required for turning. In Mastercam software, the optimum speed for rough lathe operation is 90 RPM.

Table 1 (b): Cutting Parameter of Lathe Rough

Feed Rate, (mm/rev)	Plunge Rate, (mm/rev)	Spindle Speed, (RPM)	Machining Time, (min)
0.2	0.1	30	352.200
		60	176.167
		90	117.483

The Figure 4 (b) below represents the relationship between spindle speed and machining time. The machining time is 352.2 minutes when using 30 RPM spindle speed, and 176.167 minutes when using 60 RPM spindle speed. At 90 RPM spindle speed, the machining time is 117.483 minutes. The graph is ordered descending. As a result, the shorter the machining time, the faster the spindle speed.

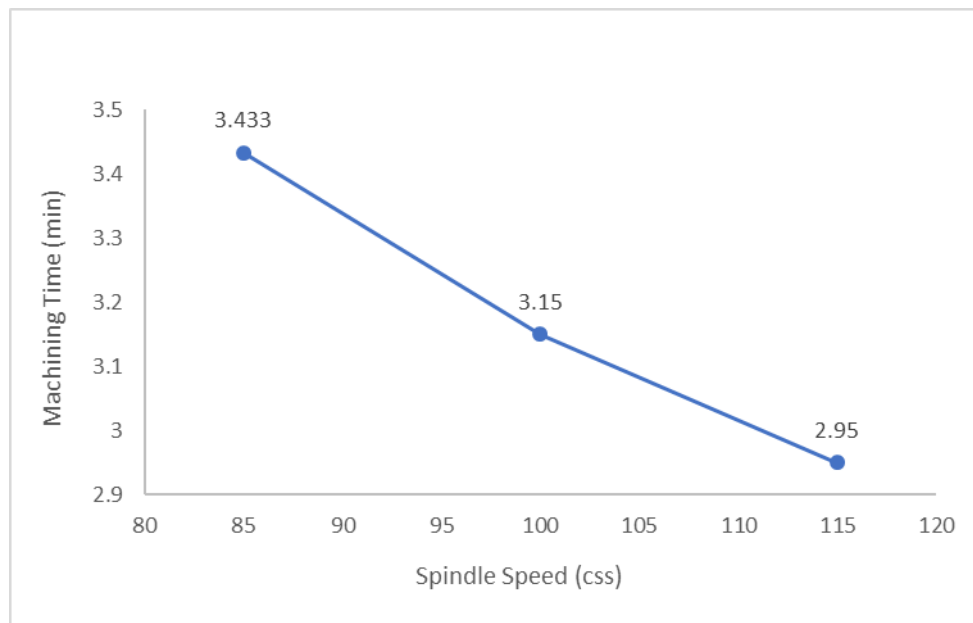**Figure 4 (b): The Relationship Between Spindle Speed and Machining Time**

Table 1 (c) below shows the cutting parameters of a lathe groove. For the lathe groove procedure, a feed rate and plunge rate of 0.1 mm/rev are used. Furthermore, three spindle speeds (85, 100, and 115 CSS) were used. The spindle speed (CSS) was used in the lathe face process because it is already automatic selected by the program of Mastercam software. This lathe groove process was not suitable to use spindle speed (RPM) because the operation will be caused crash in the Mastercam software. Then, the optimum speed for lathe groove operation in Mastercam software is 115 CSS. The spindle speeds of 85 CSS and 115 CSS were only used as a comparison to track the required time for turning.

Table 1 (c): Cutting Parameter of Lathe Groove

Feed Rate, (mm/rev)	Plunge Rate, (mm/rev)	Spindle Speed, (CSS)	Machining Time, (min)

0.1	0.1	85	3.433
		100	3.150
		115	2.950

The relationship between spindle speed and machining time is illustrated in Figure 4 (c) below. The machining time is 3.433 minutes with a spindle speed of 85 CSS and 3.15 minutes with a spindle speed of 100 CSS. The machining time is 2.95 minutes at 115 CSS spindle speed. The graph is arranged in descending order. As a result, the shorter the machining time, the faster the spindle speed.

Figure 4 (c): The Relationship Between Spindle Speed and Machining Time
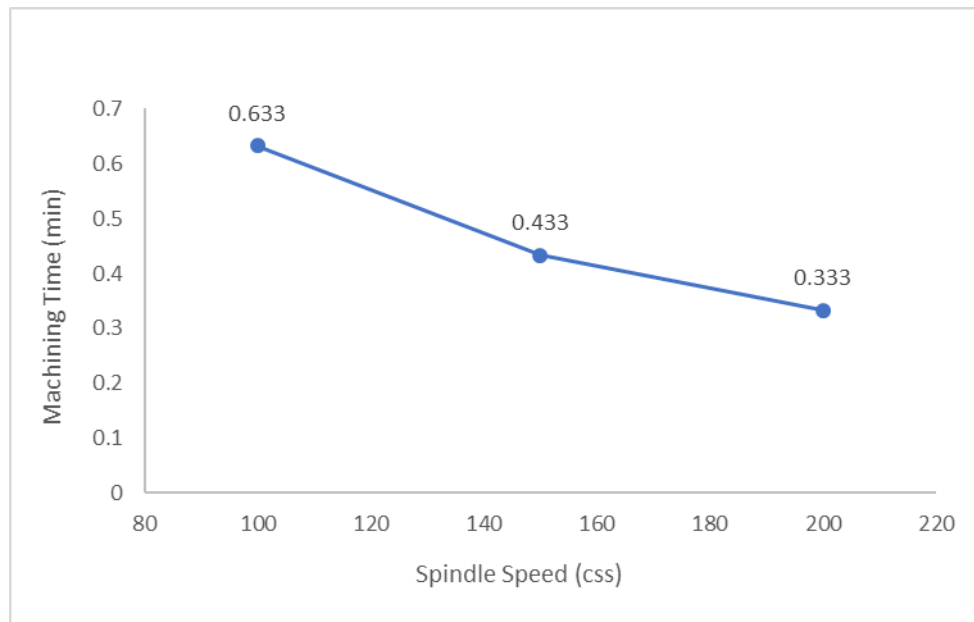
Table 1(d) shows the cutting parameters of a lathe center drill. A feed rate of 0.05 mm/rev was used for the lathe center drill procedure. The plunge rate was not used in this process. In addition, three different spindle speeds (100, 150, and 200 RPM) were utilized. The optimum speed for lathe center drill operation in simulation Mastercam software is 200 RPM. The spindle speeds of 100 and 150 RPM were only used for comparison to track the time required for turning.

Table 1 (d): Cutting Parameter of Lathe Center Drill

Feed Rate, (mm/rev)	Plunge Rate, (mm/rev)	Spindle Speed, (RPM)	Machining Time, (min)
0.05	-	100	0.633
		150	0.433
		200	0.333

The Figure 4 (d) below shows the relationship between spindle speed and machining time. The machining time is 0.633 minute with 100 RPM spindle and 0.433 minute with a 150 RPM spindle. At 200 RPM spindle speed, the machining time is 0.333 minute. The graph is arranged in descending order. As a result, the faster the spindle speed, the shorter the machining time.
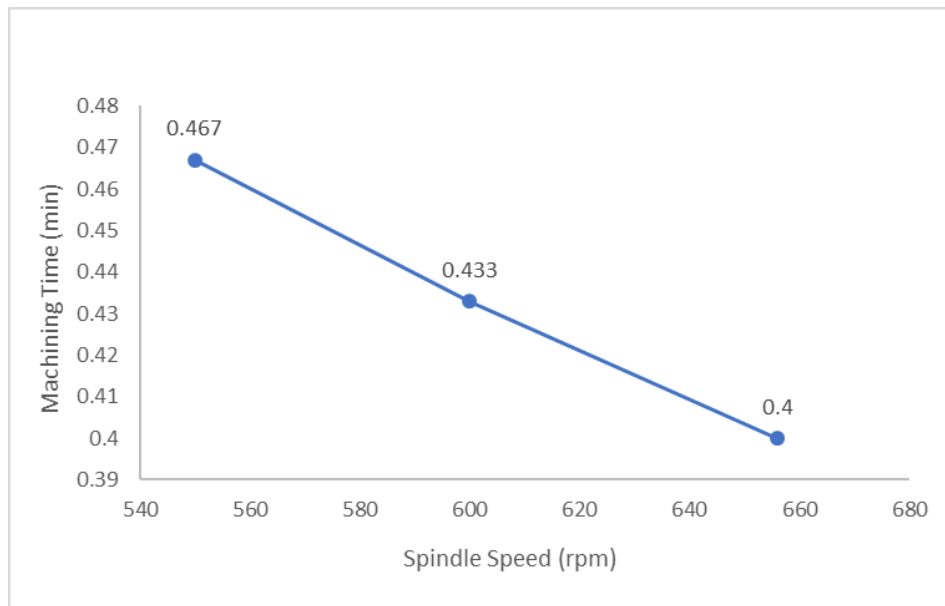
Figure 4 (d): The Relationship Between Spindle Speed and Machining Time

The cutting parameters of lathe drill are shown in Table 1 (e) below. For the lathe drill procedure, a feed rate of 0.2 mm/rev was used. It then proceeded to plunge at a rate of 0.1 mm/rev. Furthermore, three different spindle speeds (550, 600, and 656 RPM) were used. In Mastercam software, the optimum speed for lathe drill operation is 656 RPM. The spindle speeds of 550 RPM and 600 RPM were only used for comparison purposes in order to track the time required for turning.

Table 1 (e): Cutting Parameter of Lathe Drill

Feed Rate, (mm/rev)	Plunge Rate, (mm/rev)	Spindle Speed, (RPM)	Machining Time, (min)
0.2	0.1	550	0.467
		600	0.433
		656	0.400

The relationship between spindle speed and machining time is illustrated in Figure 4 (e) below. The spindle speed has the greatest influence on the differences in machining time that occur. When using 550 RPM spindle speed in simulation Mastercam software, the machining time is 0.467 minute, while 600 RPM spindle speed takes about 0.433 minute. The time required for machining is 0.4 minute at 656 RPM spindle speed. The graph is arranged in descending order. As a result, the faster the spindle speed, the shorter the machining time.
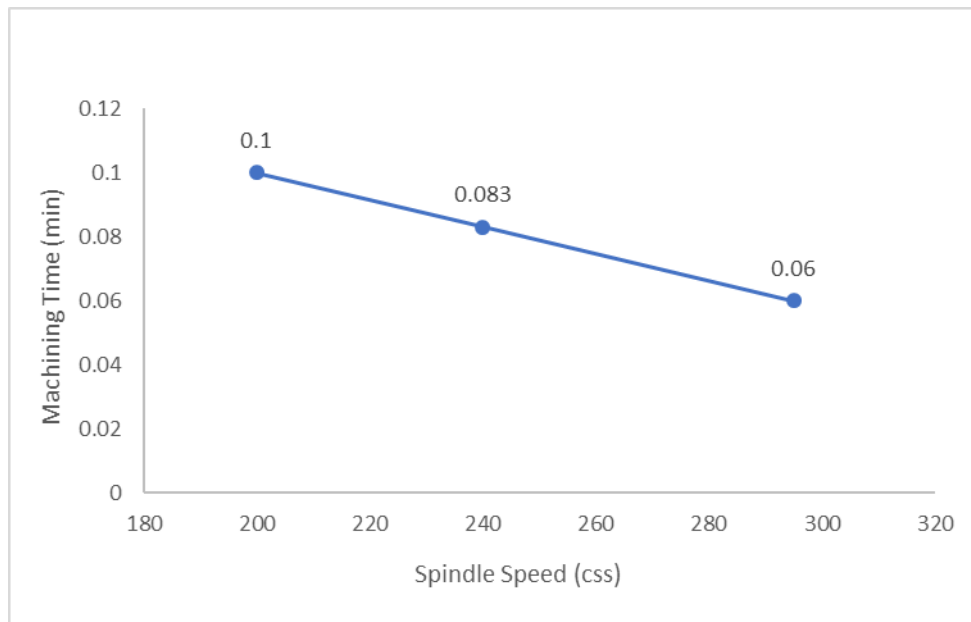
Figure 4 (e): The Relationship Between Spindle Speed and Machining Time

Table 1(f) shows the cutting parameters for lathe finish. A feed rate of 0.2 mm/rev was used for the lathe finishing procedure. It then began to descend at a rate of 0.1 mm/rev. In addition, three different spindle speeds (200, 240, and 295 CSS) were utilized. The optimum speed for lathe finish operation in Mastercam software is 295 CSS. The spindle speeds of 200 CSS and 240 CSS were only used to compare the time required for turning.

Table 1 (f): Cutting Parameter of Lathe Finish

Feed Rate, (mm/rev)	Plunge Rate, (mm/rev)	Spindle Speed, (CSS)	Machining Time, (min)
0.2	0.1	200	0.100
		240	0.083
		295	0.060

The Figure 4 (f) below shows the relationship between spindle speed and machining time. The spindle speed has the largest effect on the machining time differences that occur. Machining time is 0.1 minute when using 200 CSS spindle speed in simulation Mastercam software, and 0.083 minute when using 240 RPM spindle speed. At a spindle speed of 295 RPM, the machining time is 0.06 minute. The pattern of the graph is in descending order. As a result, the spindle speed is inversely proportional to the machining time. It is because the shorter the machining time, the faster the spindle speed.

Figure 4 (f): The Relationship Between Spindle Speed and Machining Time

3.3 The Type of Toolpath Process Against Machining Time

Table 2: Time Machining of The Toolpath Process

Type of Toolpath Process	Time Machining, (min)
Facing	2.017
Roughing	117.483
Grooving	2.95
Center drill	0.333
Drill	0.4
Finishing	0.06

When creating a pin chuck, six processes must be followed. Facing, roughing, grooving, center drill, drill, and finishing are the processes. According to the Figure 5, the roughing process has the longest machining time, while the finishing process has the shortest. It is because roughing process is an important part that allows the material to be transformed from a cylindrical shape to a pin chuck shape. The roughing process takes approximately 1 hour 57 minutes and 28 seconds to complete. For finishing processes, it only takes about 4 seconds to complete. From the result, the shortest machining time is finishing process followed by center drill, drill, facing, grooving and roughing.
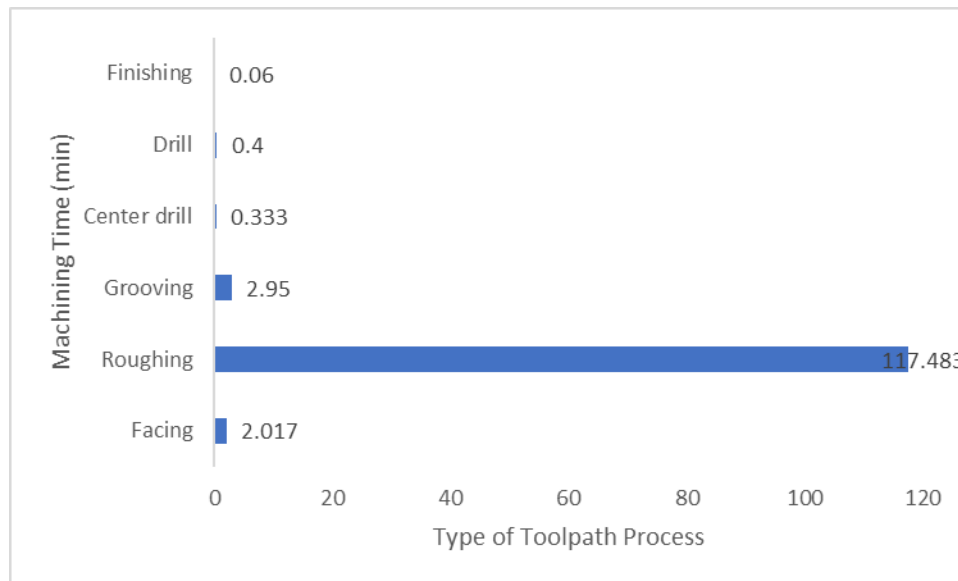
Figure 5: The Analysis between Type of Toolpath Process and Machining Time

4. Conclusion

In conclusion, the objective of studying the machining analysis of the pin chuck by using Mastercam software has been achieved. When simulating the pin chuck with Mastercam software, the first step should be to design the pin chuck component. Furthermore, six processes must go along with when operating the pin chuck. The steps are facing, roughing, grooving, drilling, and finishing. Throughout the operation, a carbide material cutting tool was used. The toolpath process must be followed according to the steps. Otherwise, the process will be unable to simulate. In addition, three parameters affect the machining time when do simulate the pin chuck. Parameters include spindle speed, feed rate, and plunge rate. In those cases, the faster the spindle speed used, the shorter the recorded machining time. Moreover, each toolpath process had its own optimum speed to ensure that the operation ran smoothly and without any crashes or collisions. Finally, when it comes to toolpath types processes, the finishing part has the shortest machining time while the roughing part is the longest machining time when compared to the others.

Acknowledgement

References

[1]   Person. (n.d.). (2020). What is Machining? A Guide to the Different Types of Machining Operations.https://www.thomasnet.com/articles/custom-   manufacturing   fabricating/types machining/.

[2]   Herr, D. K. and T. (n.d.). (2021). G-Code and M-Code: What Do They Mean In CNC Machining? Stecker Machine Co. https://www.steckermachine.com/blog/g- code-m-code.

[3]   Writer, S. (n.d.). (2020) What is CNC Machining? Definition, Processes, Components & More. Thomasnet® - Product Sourcing and Supplier Discovery Platform - Find North American   Manufacturers,   Suppliers   and   Industrial   Companies. https://www.thomasnet.com/articles/custom-   manufacturing-fabricating/understanding-cnc-machining/.

[4]   CAD/CAM Software Solutions for CNC Machining by Mastercam. Mastercam. (2020, November 11). https://www.mastercam.com/solutions/.

[5]   Groover, M. P., & Zimmers, E. W. (1984). Cad / computer-aided design and manufacturing. Prentice-Hall.

[6]   C. 1st J. (2020, March 3). What is a Pin Chuck? Wonkee Donkee Tools. https://www.wonkeedonkeetools.co.uk/pin-chucks-vices/what-is-a- pin-chuck.

[7]   Media, S. by R. (2019, March 14). What Is Mild Steel? Metal Supermarkets - Steel, Aluminum,Stainless, Hot-Rolled, Cold-Rolled, Alloy, Carbon, Galvanized, Brass, Bronze, Copper. https://www.metalsupermarkets.com/what-is-mild-steel/.

[8]   G.Zhang and S.C.-Y. Lu (1992). – Economic Evaluation of Machining Operating Planning.