RESEARCH PROGRESS IN MECHANICAL AND MANUFACTURING ENGINEERING VOL. 1 NO. 1 (2020) 132–138

RPMME

Homepage: http://penerbit.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

Simulation of Tool Path Using Conventional Method for Milling Application

A.M Shafie[1], H. Abdullah[1,*]

[1]Faculty of Mechanical and Manufacturing Engineering, University Tun Hussein Onn Malaysia, 86400 Parit Raja, BatuPahat, Johor, Malaysia

*Corresponding Author

DOI: https://doi.org/10.30880/rpmme.2020.01.01.017
Received; 8 October 2020; Accepted 27 October 2020; Available online 10 November 2020

Abstract: Today our manufacturing industry has become more far more advanced compared to decades before. The development and wide application of Computer-Aided Design and Manufacturing (CAD-CAM) software have led to the use of Computer Numerical Controlled (CNC) machines in the manufacturing industry. This software, which includes tool paths, will generate key machining parameters. This study describes the form of tool paths typically used to manufacture a product and how the 3-axis CNC vertical milling machine is used to refine the tool paths. The objective of this study is to simulate the machining time of pocket milling by using a conventional method by using the software CAD/CAM. Second is to investigate the axial and radial depth on machining time. This study can be done by using CAM software which is MasterCAM. The tool paths can be optimized using MasterCAM software through simulations. This program allowed the tool paths to be simulated, and results were obtained in machining time. To achieve the optimal machining time to be used in the actual machining process, parameters such as tool diameter, federate, spindle speed, and cut depth were evaluated. Testing shows that in terms of shortest machining time, there were three styles of machining strategies that were high speed, parallel spiral and Zig-Zag that were more beneficial than any other commodity machining strategies. Based on this study, the tool diameter will achieve the shortest simulation time by using the maximum tool diameter. By referring to the result of this study, three proven results produce the shorter time which is Zig-Zag, High-Speed or Parallel Spiral which is the time for Zig-Zag was 10 hours and 20 minutes and the other two was the closest time with the Zig-Zag. Same goes to cut depth that produced the shortest machining time by the highest cut depth value used by referring to the result of this study, where it can be seen that the difference of 1.5mm could influence the time by 10 seconds above approximately. Lastly, it can be concluded that the machining method and cutting depth are essential factors to achieve a shorter machining time.

Keywords: Simulation, Tool path, Pocket Milling

**Corresponding author: haslinaa@uthm.edu.my*

1. Introduction

The milling operation is a process of cutting metal using one or more teeth of a rotating cutter. Determination of optimum cutting parameters such as cutting depth, cutting speed and feed for assigned cutting tools, It is one of the vital modules in metal parts process planning, since machining operations have an important role to play in increasing productivity and competitiveness[1]. NC machines can significantly reduce lead times, the machining time is nearly the same as when machining parameters are selected from machining databases or handbooks. Since of the NC machines ' high capital and machining costs compared to conventional machines, there is an economic need to operate NC machines as efficiently as possible to obtain the required pay-back. Since machining costs on NC machines are sensitive to the parameters of machining, optimal values must be determined before a part is produce [2].

Pocket milling is a popular machining operation used for forming a depression in a workpiece. A commonly used NC tool path pattern for hogging away the material inside a pocket is in the form of several horizontal layers of contour lines which are formed by repeatedly offsetting the pocket boundary inwards with a stepover. Offset contours usually comprise of concave and convex corners formed by the intersection of different types of geometric entities such as line and arc [3]. If the area within the limit of a pocket is at all large compared to the end cross section. mills that are suitable for cutting it, it is common Practice making a rough cut to eliminate the bulk of the material making a slightly smaller pocket, leaving a thin (0.0254 cm or 0.01 inch is typical) layer of material that is then removed by Final finishing mill [4].

The basic method by which the three-dimensional design model is carried out by Solidworks, assembly and then converting the graphics data to the MasterCAM software for the NC programming method for contour parts uses the layer, this method can integrate the design into the manufacturing technology in order to improve the efficiency of production and apply it to a broad engineering and technical base[5]. The Solidworks software has more powerful entity parameterized modelling, intelligent driving function, easy to use, has some advantages in product design and parts design. The

MasterCAM is the software that been use widely at the manufacturing industry, this software could reduce the cost for company in term of material and work piece without being wasted the workpiece for the purpose of research. MasterCAM is the software that have an integrated CAD/CAM system which creates geometry, prepares important points and completed drawings, visualizes the image tool path and generates the NC programmer. MasterCAM lets in the person to trade the processing parameters or tool at any time throughout the process, to alter the tool path, and later to visualize the machining again(Prajapati et al., 2013). Before making use of the programmed directly on a machine tool, it is viable for a programmer to test the impact of his or her work on the screen. In addition to creating geometry, the programmer person can analyse it, manage the tool path, and see on the display the machining effect. In other words, the actual appearance of the worked-on element inclusive of the tool tracks shaped in the machining process [7].

2. Methodology

In this study, a three-dimensional model of has been develop using Solidwork software as shown in Figure 1. The detail dimension and the type of process involve in the model has been shown in Figure 2. The design is based from sugar cane crusher that was inspired from the research of tool selection for rough and finish CNC milling operations based on tool path generation and machining optimisation [8]. The design has been saved as STEP format in Solidworks software and then has been export to the MasterCAM software for simulation process and generating the tool path.

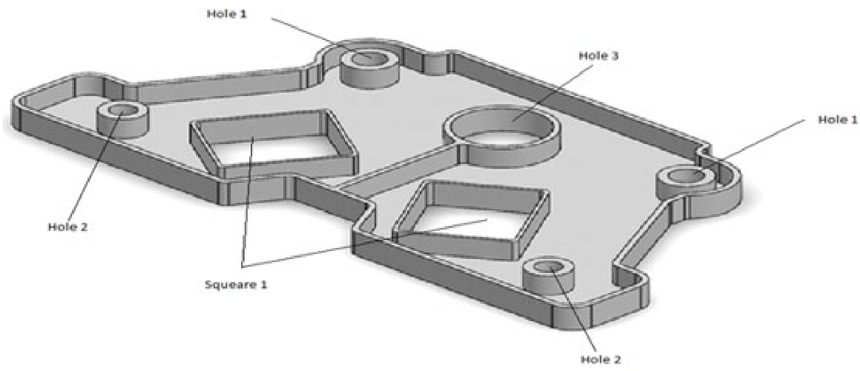

Figure 1: Three dimensional of model

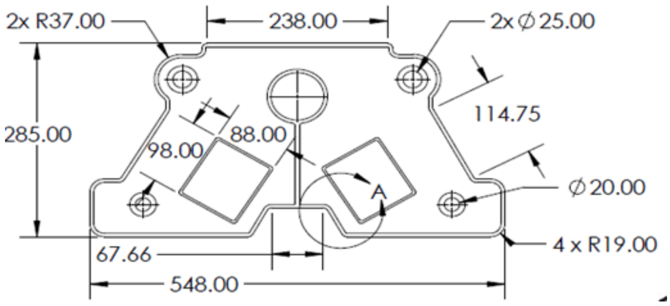

Figure 2: Detail dimension of three dimensional of model

2.1 Tool path generation in Mastercam

To generate the tool path for each machining process in sugar cane crusher model, a conventional method which is Mastercam software has been employed. Figure 3 described the process of generating and simulation of tool path in CAD/CAM software. Once the model has been constructed in CAD software, it will be exported to the CAM software based on specific format file. In the CAM software, the type of machining and parameters of machining has been defined. Then, the tool path of each process of machining can be generated. The results of the machining process such as tool path length and machining time can be explored by using backplot part.

2.2 Selection of cutting parameters

The selection of cutting parameters have been made by used the data from the published journal [9]. The parameters involved in this simulation is cutting speed, spindle speed, and stepdown. The cutting tools diameter are based on the size of the workpiece that will be used to remove which is for Sugar Cane Body operation, the tool diameters are 16mm. The tool diameter for Drill/Counterbore 1,2, and 3 is 25mm,20mm,30mm. Lastly, for the Pocket operation, the tool diameter for pocket 1 and 2 is 16mm and for pocket 3 is 10mm. The tool diameter for pocket 3 operation is smallest because it required to do machining at the tight space between the Drill/Counterbore 1 and 2. Mastercam software generates the machining time in term of hours. In order to convert a data into the graph too, the time was changed in terms of second Table 41 indicates the cutting parameter that will be obtained from the journal and used in this analysis [9].

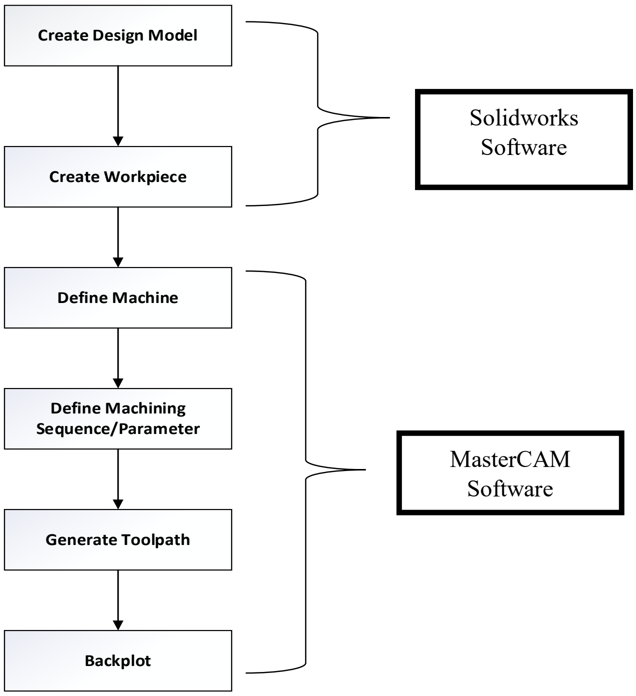

Figure 3: Overall process using MasterCAM software

Table 1: Parameters for machining [9]

Cutting Parameter / **Operation**	**Spindle Speed**	**Cutting Feed**	**Step Down(mm)**
Sugar Cane Body	5000	950	2.0
Drill Counterbore 1	5000	950	2.5
Drill Counterbore 2	5000	950	2.5
Drill Counterbore 3	5000	950	2.5
`Pocket 1	5000	950	2.5
`Pocket 2	5000	950	2.5
`Pocket 3	9000	1800	1.0

3. Results and Discussion

3.1 Results

The parameter of this study was taken from the study Intelligent tools selection for roughing and finishing in machining [9]. The parameter was chosen from his study because based on his research. It mainly focuses on the condition on a workpiece, which means a consideration about tool wear, surface roughness and force on a workpiece. Based on Figure 4, the result below shows all the total time of each type of operation where the parameters of each operation based on Escamillar Salazar research. The parameters involved in this study are spindle speed, cutting rate and step down in order to produce the machining time.

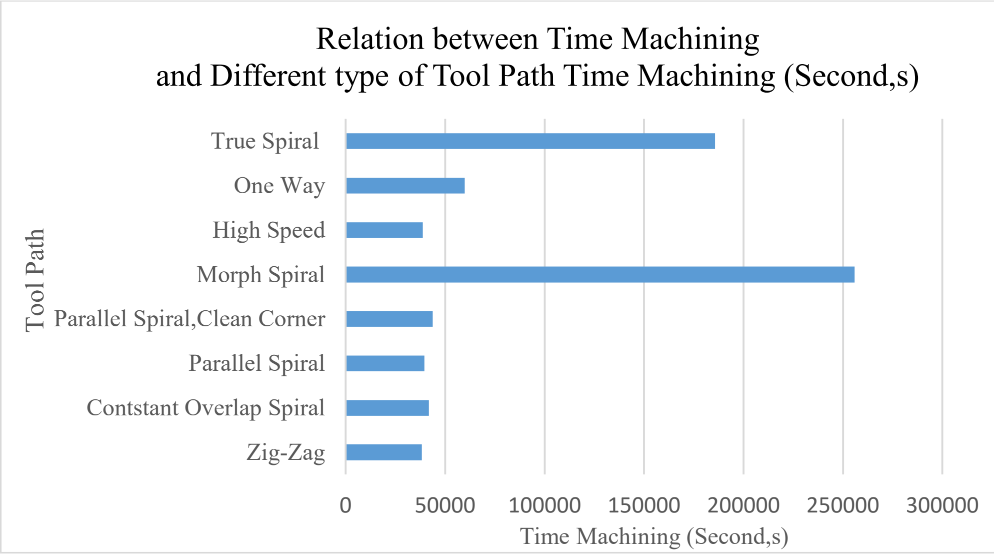

Figure 4: Graph of comparison result

Table 2: The relation between time machining and different type of tool path

Method of machining	Time Machining (s)
Zig-Zag	38214.94
Constant Overlap Spiral	41860.69
Parallel Spiral	39542.41
Parallel Spiral, Clean Corner	43662.16
Morph Spiral	255853.32
High Speed	38687.28
One Way	59749.75
True Spiral	185628.78

By referring to the graph and all the data obtained, the most suitable tool path for the sugar crane crasher design is Zig-Zag because it shows the shortest simulation time which is 10 hours 20 minutes compared to the other tool path. The second suggestion tool path can be used either High Speed or parallel spiral were both of this tool path have the nearest value. The longest simulation time is morph spiral where it takes 71 hours. A most extended hour of machining is not recommended because it will increase the cost of production.

3.2 Effect of depth of cut

The cutting rate and feed rate are combined with the cutting depth to determine the material removal rate, which is the volume of material in the workpiece that can be removed per unit time. Axial and radial depth of machining are dominant factor which is required proper entrance where the axial is the distance a tool enters the centreline of a workpiece and radial is the distance a tool enters a workpiece. Based on MasterCAM, the depth is the distance to finish from the surface to end of removing material. The step down is the value to determine the optimum cutting rate for a depth of cut to remove the material efficiently.

Table 3: Step down measure

Operation	Step Down(mm) Journal Parameter	Step Down(mm) MasterCAM parameter	Time Machining Journal Parameter	Time Machining MasterCAM Parameter
Drill/Counterbore (25mm)	2.5	1.0	11.48	30.65
Drill/Counterbore (20mm)	2.5	1.0	11.98	25.8
Drill/Counterbore (30mm)	2.5	2.0	1990.57	2247.6

By referring to the Table 3, the difference of time take to machining does not take a much difference. Which is even the difference parameter value of step down being 1.5, the difference of time taken for machining only for about 20 seconds. Even the Journal Parameter have higher feed rate than MasterCAM parameter, the increment of one second could make the time MasterCAM parameter closer about nine or six-second difference. The closes example can be found Drill/Counterbore(30mm) where the value of time machining does not have much different even to the Journal parameter have a higher feed rate compare to MasterCAM parameter.

4. Conclusion

Producing a product using a CNC machine is quite complicated, especially for a complex product. However, the use of CAD / CAM software simplifies the machining process. First, the exact dimensions of the design should be created to facilitate the simulation of the parameters. In process simulation of selected tool paths must be carried out considering machining time and suitability of the workpiece. Research on CNC machining operations in relation to tool paths in CAM software is a practical way to achieve the goal faster, reduce costs and be more efficient in machining. Incorrect selection of the tool will result in a longer production time cost than finished. The machining method and cutting depth are important factor to achieve a shorter machining time. In this research, among the efficient machining method are Zig-Zag, High Speed and Parallel Spiral. For cutting depth, the shortest machining time can be achieved at the depth of 2.5mm with the suitable cutting speed and tool paths. The suitable cutting speed with the suitable measured value could make the machining efficiently done because its also can extend the longevity of cutting tool to be used frequently in future.

References

[1] P. Palanisamy, I. Rajendran, and S. Shanmugasundaram, "Optimization of machining parameters using genetic algorithm and experimental validation for end-milling operations," *Int. J. Adv. Manuf. Technol.*, vol. 32, no. 7–8, pp. 644–655, 2007, doi: 10.1007/s00170-005-0384-3.

[2] M. Tolouei-Rad and I. M. Bidhendi, "On the optimization of machining parameters for milling operations," *Int. J. Mach. Tools Manuf.*, vol. 37, no. 1, pp. 1–16, 1997, doi: 10.1016/S0890-6955(96)00044-2.

[3] H. S. Choy and K. W. Chan, "A corner-looping based tool path for pocket milling," *CAD Comput. Aided Des.*, vol. 35, no. 2, pp. 155–166, 2003, doi: 10.1016/S0010-4485(02)00049-0.

[4] T. R. Kramer, "Pocket milling with tool engagement detection," *J. Manuf. Syst.*, vol. 11, no. 2, pp. 114–123, 1992, doi: 10.1016/0278-6125(92)90042-E.

[5] X. L. Ye, "Based on Solidworks and Mastercam Combined for Rub a Drum of Four-Axis NC Machining Programming Research," *Appl. Mech. Mater.*, vol. 599–601, pp. 225–229, 2014, doi: 10.4028/www.scientific.net/amm.599-601.225.

[6] R. Prajapati, A. Rajurkar, and V. Chaudhary, "Tool Path Optimization of Contouring Operation and Machining Strategies for Turbo Machinery Blades," *Int. J. Eng. Trends Technol.*, vol. 4, no. May, pp. 1731–1737, 2013, [Online]. Available: http://www.ijettjournal.org.

[7] A. Werner, "Determining Errors in Complex Surfaces," vol. 26, no. 2, 2006.

[8] T. E, M. I, C. A. D. Cam, and C. A. D. Cam, "Tool selection for rough and finish CNC milling operations based on tool - path generation and machining optimisation Izbira orodij za grobe in fine CNC - obdelave pri rezkanju na osnovi generiranja poti orodij in optimizacije obdelave," *Tool Sel. rough Finish CNC milling Oper. based tool-path Gener. Mach. Optim.*, vol. Volume 10, no. Number 1, pp. 18–27, 2015.

[9] I. G. Escamilla-Salazar, L. Torres-Treviño, B. González-Ortiz, R. Praga-Alejo, and D. González-González, "Intelligent tools selection for roughing and finishing in machining of Inconel 718," *Int. J. Adv. Manuf. Technol.*, vol. 88, no. 9–12, pp. 3031–3039, 2017, doi: 10.1007/s00170-016-9005-6.