Research Progress in Mechanical and Manufacturing Engineering Vol. 2 No. 2 (2021) 130-136
© Universiti Tun Hussein Onn Malaysia Publisher's Office

RPMME

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

Simulation Approach of End Milling Process Based on Finite Element Method

Muhammad Nur Ariff Zulkifli[1], Haslina Abdullah[1]*

[1]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, 86400, Batu Pahat, Johor. MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.02.016
Received 02 Aug. 2021; Accepted 27 Nov. 2021; Available online 25 December 2021

Abstract: During end milling operation, the cutting tool pushes against the material, but the material is immovable. Some of that force cuts the material, but some of it pushes against the cantilevered cutting part. This pushing causes the tool to bend away from the centerline, and deflection of the tool occurred. These deflections also cause dimensions errors or surface errors on the completed component. This study presents a simulation of a three-dimensional cutting tool in end milling process using finite element analysis. This study aims to develop a simulation procedure using Abaqus software to determine the deflection and stress in a cutting tool of the milling process. The deflection value is determined based on the value stress, strain, and displacement obtained from the simulation. The simulation results indicate that the simulation-based on Finite element analysis can predict the stress and tool deflection.

Keywords: Tool Deflection, End Milling, Finite Element Method

1. Introduction

Milling cutting is now one of the most common production techniques due to its versatility and high material removal rate in creating desired dimensions. In the aerospace and automotive sectors, end milling is a critical machining process. Pocketing of aircraft panels and end milling of stamping dies in automobile production are two examples of typical uses [1]. Predictions of numerous parameters such as cutting pressures, stresses, and temperature are crucial in selecting tool material and design prior to production to guarantee an effective machining process. The primary source of machining mistakes in end milling, particularly with tools having a high length-to-diameter ratio, is tool deflection [2]. Among the errors, tool deflection is the most severe impediment to precision machining [3]. This study is to understand the forces acting on the cutting tool to design and estimate power requirements.

With the emergence of powerful computers and numerical algorithms such as the finite difference method (FDM), artificial intelligence (AI), and finite element method (FEM), these are now frequently utilized in the machining sector [4]. The finite element approach has evolved into a potent tool in the simulated cutting process among the numerical method. Thus, simulations of the cutting force are performed using the finite element approach to estimate the tool deflection that occurs.

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

The cutting process is still one of the most common industrial processes nowadays. Vibrations that occur during the cutting process significantly influence the workpiece's accuracy [5]. In general, dimensional surface inaccuracy is caused mainly by tool and workpiece deformation during milling, resulting in the desired material removal. Typically, end milling deflections were measured in proportion to force, resulting in static deflections [1]. Cutting forces during the end milling process causes tool deflection and machined surface form inaccuracy. As a result, this research will model cutting tool deflection to improve cutting parameters, reduce tool deflection and eliminate defects in a final manufactured component. This research aims to develop simulation procedure to determine the deflection and stress in a cutting tool of the milling process. This study also is to analyze the deflection and stress of the cutting tool for the milling process.

2. Methodology

The Finite Element Method (FEM) is the most often used numerical approach in metal machining. The essential idea of the finite element technique is the replacement of a continuum with finite elements forming a mesh. Solidworks and Abaqus FEM simulation tools were utilized in this study. The parameter of the cutting tool managed to identify from some study that has been reviewed. Then, the cutting tool is models in Solidworks.

2.1 Simulation software

After the model's design has been completed, the analysis of the model on tool deflection of the cutting tool can be identified using Abaqus software. At the same time, the simulation conduct in the Abaqus software. Figure 1 shows the overall process using Solidworks and Abaqus software to study the tool deflection of the cutting tool during the milling process.

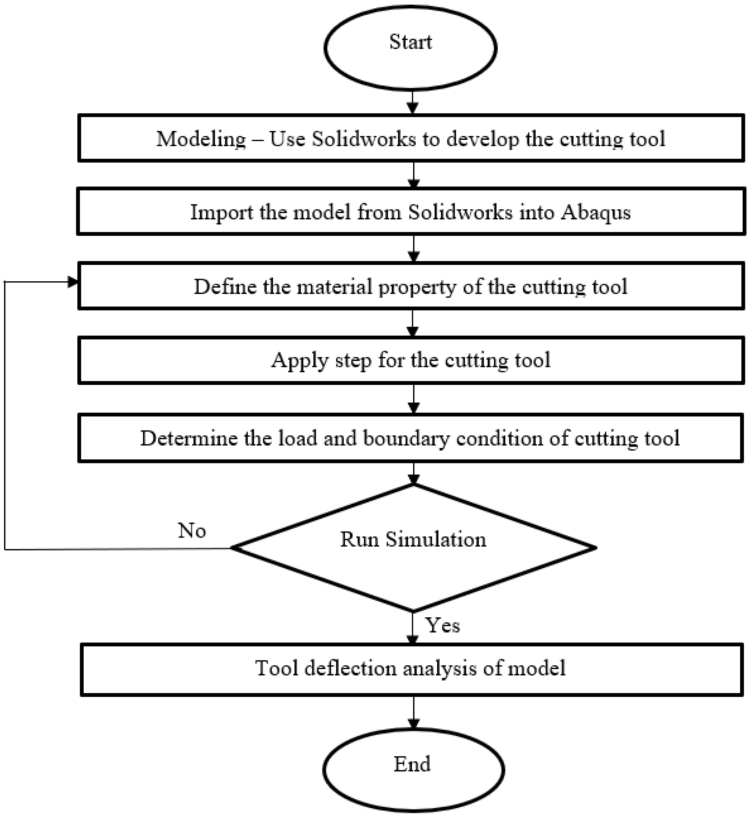

Figure 1: Flowchart for simulation process

2.1.1 Create Design Model

After the model is completed, the model is exported to the Abaqus software to run a simulation cutting tool. Figure 2 shows the model of the cutting tool with two flutes in 3 dimensional and two dimensional. All the dimension lengths are in millimeters (mm).

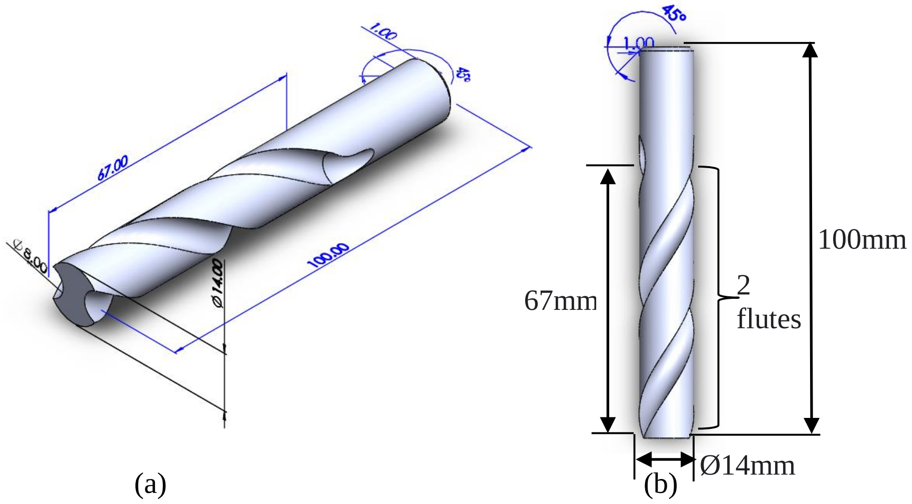

Figure 2: Two flutes cutter (a) in 3 dimensional (b) in 2 dimensional

2.1.2 Properties of Cutting Tool and Workpiece

The Jonson-Cook model is one of the most commonly used constitutive models in analytical modeling and finite element simulation of machining because it is reasonably practical, straightforward, and effective [6]. Table 1 summarizes the mechanical and physical characteristics of Aluminum 6061, which was utilized by Suraidah and is used in this work.

Table 1: Mechanical and physical properties for cutting tool and workpiece[7]

	Cutting Tool
Modulus of Elasticity E(GPa)	200
Poisson's ratio, v	0.33
Desity, ρ (Kg/cm^3)	7870
Specific Heat capacity Cp (J/Kg.°c)	0.477
Thermal Conductivity, K (W/m°c)	44.5
Melting temperature, T (°c)	1520
Linear thermal coefficient of expansion, ƺ (µm/m°c)	12.3
Displacement at failure	0.1

2.1.3 Import the model into Abaqus

In this study, the three-dimensional model of the cutting tool of milling has been developed using Solidwork software. The models have been imported to Abaqus software using STEP widely used because it contained more 3D data and part geometry than IGES. The STEP format is versatile, functional and should remain a trusted 3D CAD option for years to come

2.1.4 Define the Material Property

For the next module after the Part module, the material property for the cutting tool should be defined under the Property module. The material for the cutting tool is defined, which is the Young's Modulus, E = 200GPa, and the Poisson's Ratio, v = 0.33. Both values are used in this study are based on the study by Suraidah that also used a high-speed steel cutting tool.

2.1.5 Apply Step of the Cutting Tool

Under the Step module, the user needs to create a new step after the initial step to obtain the result. In this study, the deflection and stress are needed to analyze. The output for field output and history output needed to define by the user. In this study, the field output variables consist of stress, strain, and displacement, while for history output consists of contact only. All the variables that have been chosen are to analyze the tool deflection of the cutting tool.

2.1.6 Apply Load and Boundary Conditions

In the Load module, the user needs to apply loads and boundary conditions of the component. Figure 3 shows the load is applied to the cutting tool that consists of concentrated force in 3 directions: Fx, Fy, and Fz. The value for Fx, Fy, and Fz are 167.34N, 161.05N, and 77.35N, respectively [8]. All the forces were obtained from a previous study by Jiang, which experimental study of the milling force. The boundary condition is also needed to determine by the user in the Load module. Figure 4 shows the boundary condition on top of the cutting tool. The encastre boundary condition is applied on top of the cutting tool to constrains all active structural degrees of freedom in the region specified. The boundary condition acts as a spindle to hold the cutting tool at the top surface.

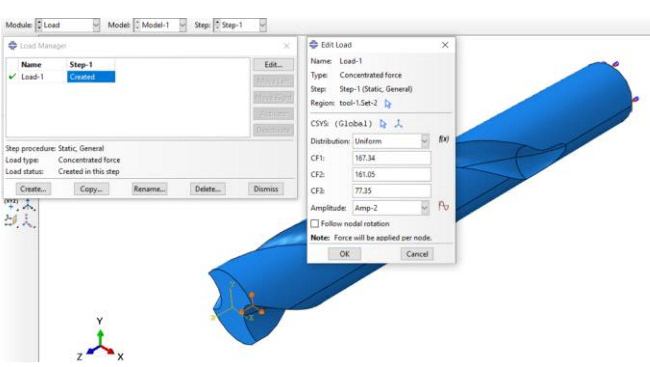

Figure 3: Create a load of cutting tool

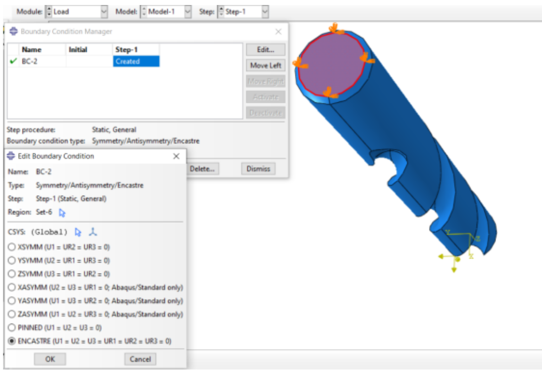

Figure 4: Boundary condition of cutting tool

2.2 Finite Element Analysis

The presented model in this paper is built in Abaqus. It offers more flexibility in defining the material strength and fracture properties. In general, cutting forces in machining play a significant influence on the operation's quality and efficiency. In order to investigate the cutting forces, methods such as Lagrangian Formulation utilizing the finite element approach can be used. The motion of the mesh is associated with material locations in the Lagrangian mesh description. In Abaqus, the mesh used to discretize the tool is a non-uniform mesh.

3. Results and Discussion

Analysis of the tool deflection in the milling process has been done to achieve the objectives, analyzing deflection and stress of the cutting tool during the milling process. This study also focuses on three-dimensional cutting tool and also include three forces in three directions. The force that assumes act on the cutting tool is when the cutting tool operates on a workpiece. The output of this study consists of 3 variables: maximum principal, magnitude, and stress acting on the cutting tool.

In order to obtain the result after the parameter and material property are determined, the user can obtain the result by simulating the Job module. The function of the monitor job is to show the status either the status is completed or aborted. Furthermore, the Visualization module offers a graphical representation of the finite element model and its findings. Changing output requests in the Step module receives model and result information, which is then stored in the output database. Based on the contours, all variable output might see a distinct outcome. A contour plot shows the values of an analysis variable at a particular phase and frame of the analysis. For X-Y data, a two-dimensional graph of one variable against a variable set by the user. Before the graph is plot, the path to analyze the cutting tool needed to determine first to create a graph of one variable against the true distance of the path.

3.1 Stress Acting on Cutting Tool

Stress is the second variable of the output in this study. Stress also occurred when the force was acting on the cutting tool. The value of stress obtain from this study is 3.500×10^2 Pa. The stress is high when the force is applied at the cutting tool's edge and can affect deflection due to high elasticity. Figure 5 shows the path of the cutting tool and the graph of stress against true distance along the path. The highest value of stress is located at the edge cutting too, and the value is 2.7×10^3 Pa then decreases until no value of stress obtains from the stress graph against true distance along the path. The value of stress is bigger compared to the value of strain. The cutting tool tends to deflect when the value of stress is higher.

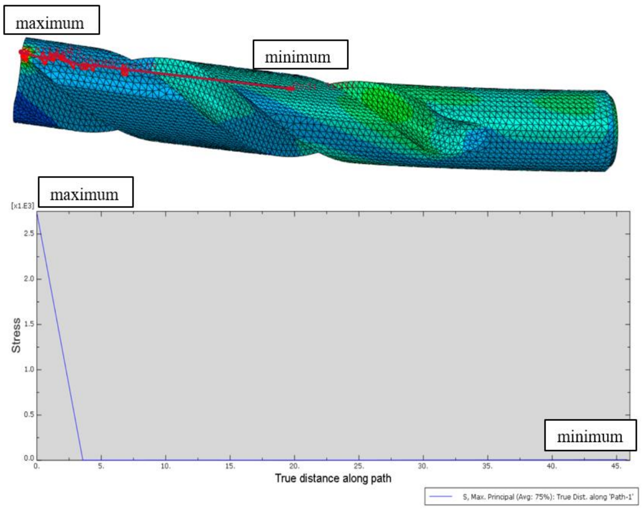

Figure 5: The path of the cutting tool and graph of Stress vs. True distance along the path

3.2 Displacement Acting on Cutting Tool

The next output variable is displacement, which displays the combined magnitude of both the real and imaginary portions of the result value. The forces acting on the cutting tool are the force in 3 directions: Fx, Fy, and Fz. From this variable, the force that most affects the cutting tool can be determined. The magnitude of displacement obtained from this study is 3.546×10^{-7}. Figure 6 shows the path of the cutting tool and displacement vs. true distance graph along the. The highest value for displacement at located at the cutting tool edge, which is 0.35×10^{-6} along the path. For X-axis, the displacement is 0.26×10^{-6} which is the highest compared to Y-axis, 0.24×10^{-6} and Z-axis, -40×10^{-9}. For the Z-axis, the displacement value obtained is negative, and the graph increases with true distance along the path. This is due to the force applied on the cutting tool in the Z-axis toward the opposite direction. The tool deflection occurs at all axes, but the X-axis has the highest deflection of the cutting tool.

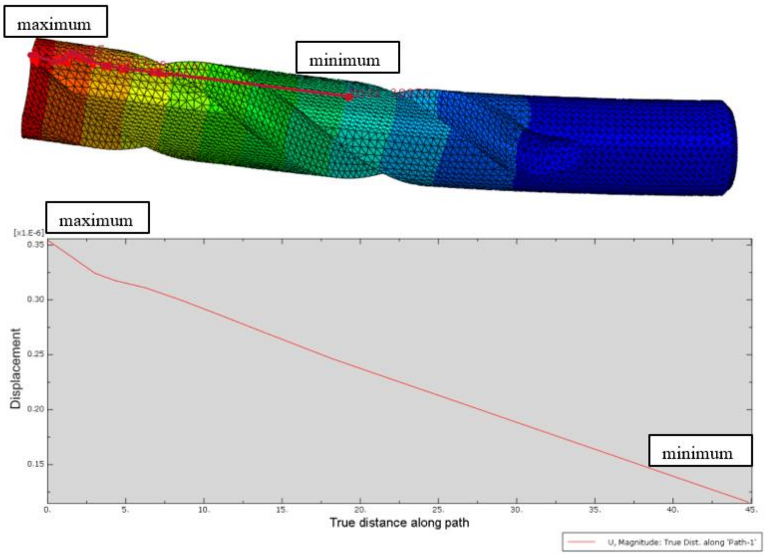

Figure 6: The path of the cutting tool and graph of Displacement vs. True distance along the path

3.3 Comparison of Simulation Result

This study is to analyzes the tool deflection of end milling. The property of the cutting tool is based on the study by Suraidah. The modulus of elasticity, E is 200Gpa, and Poisson ratio, v is 0.33 for this cutting tool. From Figure 7, we can see that the stress obtained from the study by Suraidah is 4.255×10^{8} Pa while this study is 3.114×10^{3} Pa. This value stress is different because all the properties used by Suraidah are not included in this study, such as density, specific heat capacity, thermal conductivity, and melting temperature.

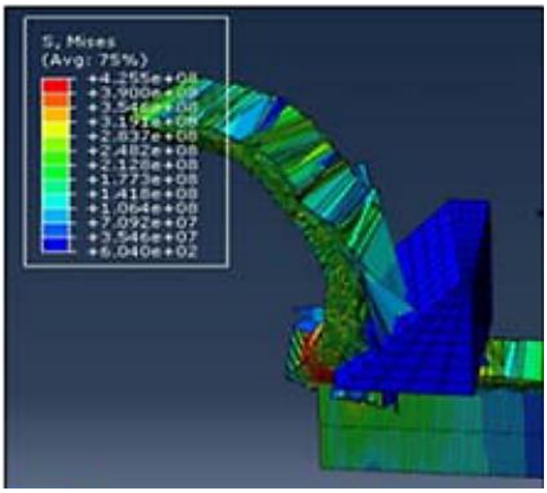

Figure 7: Chip formation formed in the simulation[7]

The percent error between the value of calculating tool deflection using the formula with the magnitude obtained from Abaqus software is 55.27%. The percent error occurred because the simulation is based on three-dimensional while the formula of the tool deflection does not specify all the force acting on the cutting tool. Furthermore, the meshing of the cutting tool is not tiny that cause the result obtained are not optimum. It can cause the difference of value obtained from the Abaqus software with calculation using formula.

4. Conclusion

This study developed a simulation procedure to determine deflection and stress based on the Finite Element Method. This study has analyzed the cutting tool's stress by applying force, and boundary conditions acting on the cutting tool as the next objective of the study has achieved. Based on simulation, it has been found that the percentage error between simulation and calculation of deflection due to the parameter that is not inserted in Abaqus software. In addition, the software simulates a three-dimensional cutting tool to obtain the output while the deflection formula is based on a two-dimensional cutting tool. Other than stress acting on the cutting tool, displacement and strain were also obtained to determine the cutting tool of the end milling process.

Acknowledgment

The authors would like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for giving the opportunity to conduct this study.

References

[1] L. Liu, M. A. Elbestawi, and N. K. Sinha, "Implementation of Self-Tuning Control to Surface Accuracy Control in End Milling," pp. 1172–1177, 1998.

[2] P. Dépincé and J. Y. Hascoët, "Active integration of tool deflection effects in end milling. Part 2. Compensation of tool deflection," Int. *J. Mach. Tools Manuf.*, vol. 46, no. 9, pp. 945–956, 2006

[3] M. Y. Yang and J. G. Choi, "A tool deflection compensation system for end milling accuracy improvement," *J. Manuf. Sci. Eng. Trans. ASME*, vol. 120, no. 2, pp. 222–229, 1998,

[4] R. J. Saffar and M. R. Razfar, "Simulation of end milling operation for predicting cutting forces to minimize tool deflection by genetic algorithm," *Mach. Sci. Technol.*, vol. 14, no. 1, pp. 81–101, 2010,

[5] T. Insperger and G. Stépán, "Stability of the milling process," *Period. Polytech. Mech. Eng.*, vol. 44, no. 1, pp. 47–57, 2000.

[6] J. Camilo, O. S. Abolghasem, and J. P. Casas-Rodriguez, "Predicting the Johnson-Cook constitutive model constants using temperature rise distribution in plane strain machining," 2019.

[7] S. Suraidah, M. Ridzuwan, M. Asmelash, A. Azhar, and F. Mulubrhan, "End milling finite element method for cutting force prediction and material removal analysis," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 788, no. 1, 2020

[8] Z. H. Jiang, M. F. Jia, and P. H. Liu, "Experimental study on milling force in processing Ti6Al4V thin-walled part," vol. 154, no. Icmia, pp. 486–515, 2017.