Research Progress in Mechanical and Manufacturing Engineering Vol. 2 No. 2 (2021) 104-111

RPMME

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

Numerical Study of Drilling Process Using Finite Element Method

Muhamad Afiq Abdul Aziz[1], **Haslina Abdullah**[1*]

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

*Corresponding Author Designation

DOI: https://doi.org/10.30880/ rpmme.2021.02.02.013
Received 02 Aug. 2021; Accepted 27 Nov. 2021; Available online 25 December 2021

Abstract: Like most machining process, cutting tool deflection in drilling is affected the efficiency of machining process. Nowadays simulation studies have been widely used in predict the cutting tool deflection and at the same time it can reduce the cost of manufacturing. Therefore, in this current research, a simulation procedure has been developed to study the cutting tool deflection in drilling process. Other than that, the objective of this study is to analyses the stress and deflection acting on cutting tool. Finite Element Analysis (FEA) has been used in this study which is Abaqus software has been implemented in order to simulate and analyses the results. A three-dimensional model of cutting tool has been develop and a force loading has been applied at the tip of cutting tool. Based on the simulation, it has been found, the deflection of cutting tool is related to the results of stress distribution and displacement along the path that has been created along cutting tool. In addition, it has been stated that the maximum stress distribute on the cutting tool can be seen from different color contour and the maximum value is 4.707e+03 which is occurred at the tip of cutting tool. While for the value of displacement, the maximum value is 1.640e-08. Lastly, by using finite element method, many parameters of cutting tool and drilling process can be studied to obtain the best cutting parameters and enhance the performance of machining process

Keywords: Tool Deflection, Drilling Process, Stress

1. Introduction

Nowadays in industries, many machining processes are performed in order to produce products such as drilling, milling and turning. Drilling is a process where it uses a drill bit to cut or enlarge a hole of circular cross-section in a solid material such as block or thin metal plates. In this process also which is drilling, known that the bit is pressed against the workpiece where it will force the cutting edge against the workpiece [1]. The formation of low residual stresses around the opening of the hole and a very thin layer on the newly formed surface of highly stressed and disturbed material. Elastic deflections during the machining process are due to cutting forces which is vertical and horizontal forces where the force

*Corresponding author: haslina@uthm.edu.my

will act on the tool and tilt the tool in the opposite direction starting from the tip[2]. Workpieces are essential sources of shape and dimensional mistakes and it will induce burr formation in the exit hole during the drilling process . A large number of researchers have done a lot of research on the gun-drilling borehole quality and found that the key factors causing the gun-drilling axis deviation are the misalignment of the drill shaft[3]. Types of structure modelling used are Finite Element Analysis (FEA) model and study of the displacement of each element using elastic hypotheses. FEA modelling are used to determine the static stiffness and fundamental vibration modes for a parallel kinematics machine[4]. In short, if the drilling process can be manipulated and understand, also on how to make sure that the end products are in good quality[5]. All problems that are related to this process can be overcome or maybe some precautions step can be taken before the process is done[6]. Where, all of this problem can be studied by using some engineering software or by using Finite Element Analysis (FEA) itself. In the drilling process, when the workpiece is subjected to dynamic load which is the high-speed machinery, one of the problems that will occur is cutting tool deflection[7]. Cutting tool deflection is a consequence of cutting force of the machining process. Even when the spindle of the machine tool followed the lead, the tip might not follow the path because of the bending of the tool to the side[8]. It will also affect the quality of the holes, surface roughness and burr formation. In both cases, time and financial loses are observed. The objective of this study is to develop simulation procedure to determine the deflection and stress in cutting tool of drilling process and also to analyse the deflection and stress acts on the cutting tool for drilling process.

2. Methodology

In this study, a model of cutting tool has been developed using Solidwork software. Then analysis on the tool was done using Abaqus software to analyze the force acting on the tool where the model will be export to Abaqus software. In Abaqus software also, the material of cutting tool has been defined. In order to succeed the simulation of the cutting tool, several of parameters need to be assigned such as properties for the cutting tool, boundary condition and the force acting on the cutting tool need to be determine. The simulation data will be validate and compare with previous study. The main purpose of this study is to investigate the stress and displacement acting on the cutting tool. The value of stress and displacement will be analyses to relate with the cutting tool deflection.

2.1 Development of cutting tool

Solidworks has been used to design the cutting tool drawing for three dimensional views. This drawing is important to be used as a guideline because it will be used in the software simulation, ABAQUS. The dimension of the cutting tool used is 14mm diameter and 100mm long and it is made up from material solid carbide. While the material properties for the cutting tool are shown in table 1.

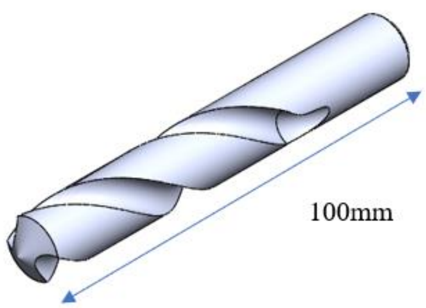

Figure 1: 3-Dimension view

Table 1: Material parameters for drill [9].

Parameters	Value
Modulus of elasticity (Gpa)	180
Poisson's ratio	0.29

2.2 Simulation Procedure in Abaqus

In order to run the simulation, the cutting tool was designed in Solidwork software and the file was saved as .STEP format in order to import the cutting tool into the Abaqus software. Where .STEP file is the most versatile that are easy to use. First of all, after the cutting tool were successfully imported into the Abaqus software. After the cutting tool were import into Abaqus, the material properties for the cutting tool were assigned as shown in table 1. For this study, it is a static study of a cutting tool where there is no movement in any axis. Step will be applied to the cutting tool. When determining the step of the cutting tool, it can be seen that new step need to be created after the initial step where in this study stress and deflection will be analyzed. Other than that, field output and history output also needed to be determine. All the variables that have been assigned and determine will be used to study the tool deflection of cutting tool.

Figure 2 shows the boundary condition that are assigned to the cutting tool on top of the cutting tool where it shows that the cutting tool are being hold without any rotation acting on any axis and the cutting tool also are static which means it does not have any torque value which means we are just studying the cutting tool without any movement or rotation. The boundary condition set for the cutting tool are ENCASTRE (U1=U2=U3=UR1=UR2=UR3) which is all the value are set as 0 respectively. Other than that, the field output request for this study was focusing on stresses, strains and also displacement. The loads for the cutting tool also were determined. While Figure 3 shows the output request in step module. After that the load will be assigned at the tip of the cutting tool by the value of 553.8N.

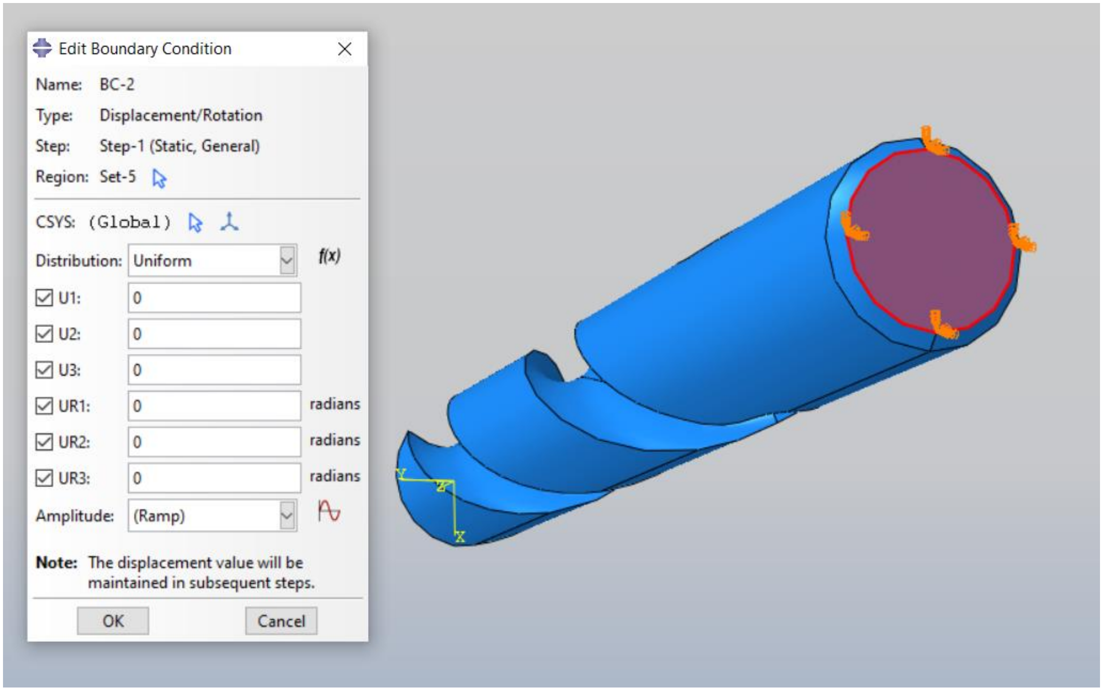

Figure 2: Boundary condition

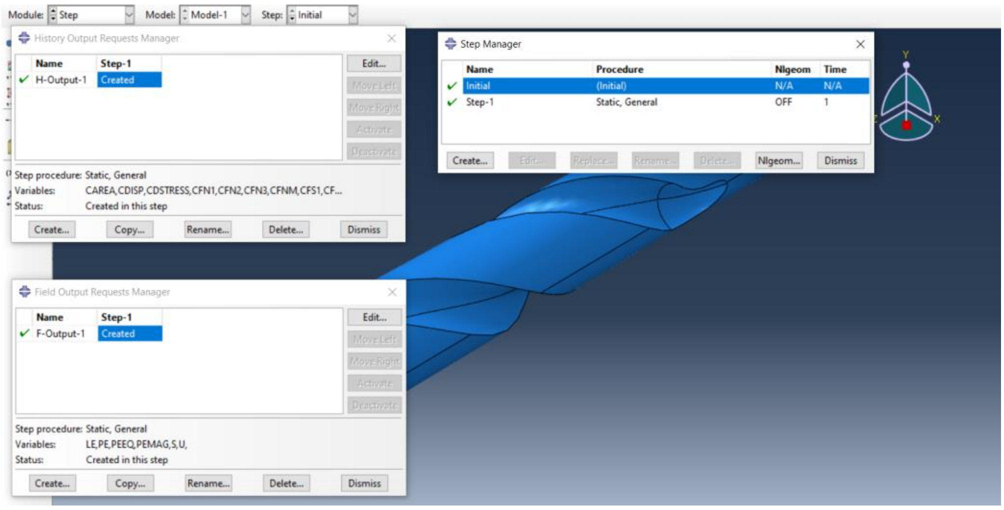

Figure 3: Output request in step module

Next is the meshing of the cutting tool where we can see figure 4 shows the cutting tool that has been meshed. Load 553.8 N were referred from the previous study and the force were put on CF3 because the force is acting on the z-axis since it is a drilling process there would be only force acting on the z-axis. Since this study is a drilling process, the load was set at the tip of the cutting tool. Under job module, job file was created in order to ensure that there was no error so that data can be obtain and collected.

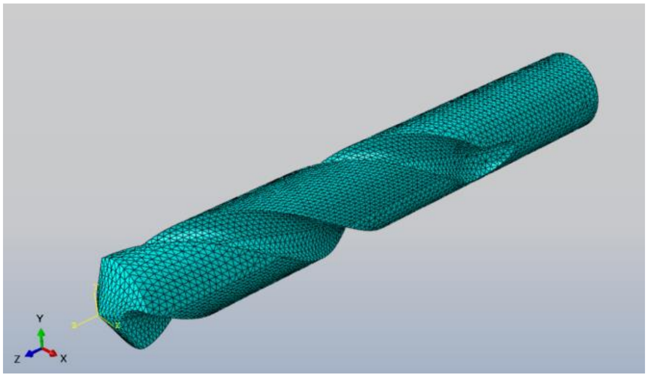

Figure 4: Meshed cutting tool

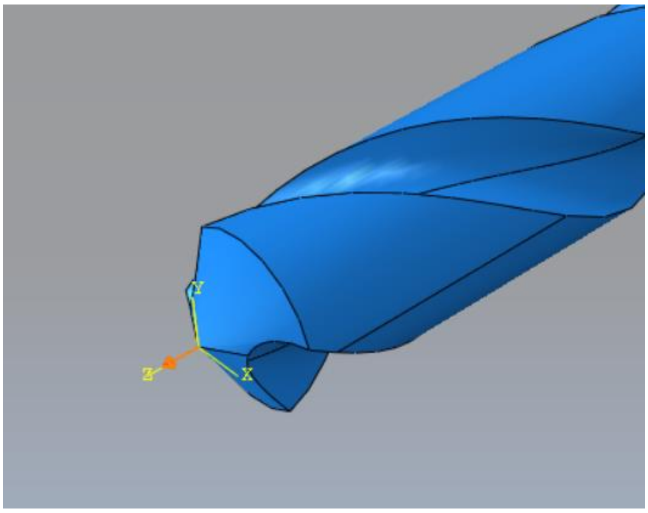

Figure 5: Load applied at the cutting tool

2.3 Calculation of cutting tool deflection

In order to study about the tool deflection of the cutting tool, an equation will be used to calculate the value of deflection where the equation can be seen as follows:

$$f(z) = C\frac{F_z}{E}\left[\frac{L1^3}{D1^4} + \frac{(L2^3 - L1^3)}{D2^4}\right]^N \qquad \text{Eq 1}$$

3. Results and Discussion

Analysis of the simulation results which includes the stress, strain and magnitude acting on the cutting tool. Results are obtained from the finite element method using Abaqus software.

The purpose of this study is to identify the relationship of amount of stress acting on the cutting tool towards the deflection of the cutting tool where stress is defined as the force applied force across a small boundary per unit area of that boundary that causes it to alter shape. In other words, the resisting force or the internal resistance generated within the material is called as stress. There are two types of stresses which is tensile and compressive. Depending on the application, structural materials are chosen for their capacity to resist tensile or compressive forces. As for this study, the stress acting on the tool were generated using Abaqus software when all the parameters were done setup on the software

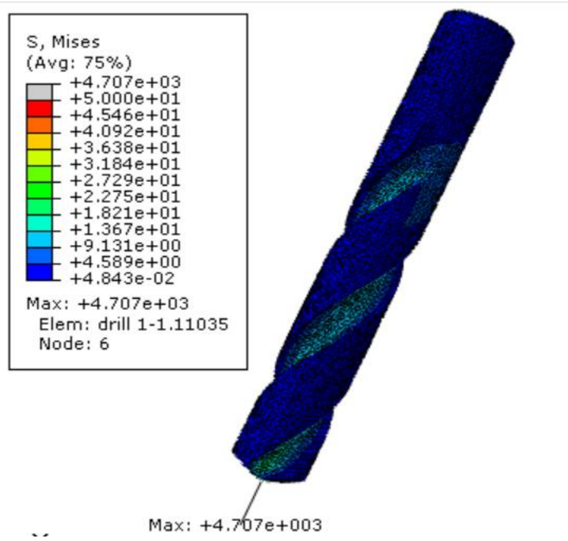

Figure 6; S, misses stress acting on tool

Based on the results generates from Abaqus, the maximum value of the stresses acting on the cutting tool is 4.707e+03 where the maximum stress are acting on the tip of the cutting tool because the loads are assigned acting on the tip of the cutting tool since this study is about drilling process the load will be acting on the tip of the cutting tool. While the minimum value is 4.843e-02 where the minimum stress acting on the body of cutting tool not nearby the tip. A path also was created on the cutting tool in order to create a graph of stress vs distance along the path. The graph can be seen in Figure 7.

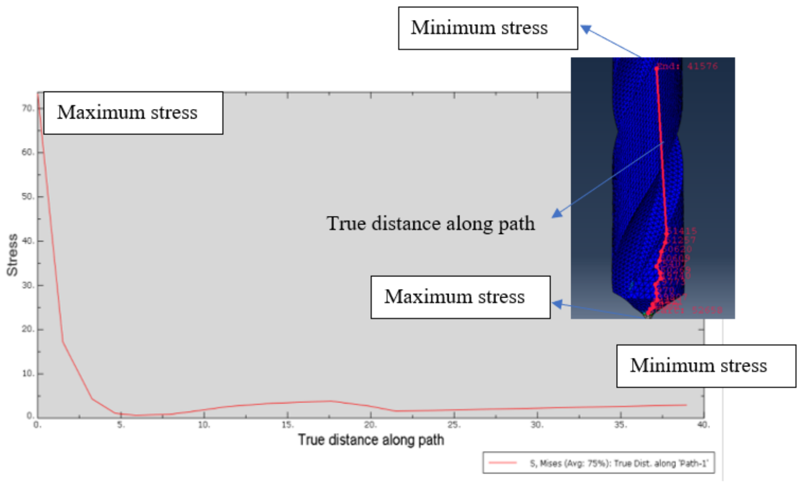

Figure 7 : Graph of stress vs distance along the path

All physical displacement components, including rotations at nodes with rotational degrees of freedom (for output to the output database, only field-type output includes the rotations). Other than stresses, Magnitude also is one of the main things that are being observed within this study. Where magnitude or displacement that are being observed in this study also shows the maximum value of deflection that can been held by the cutting tool with the exerted force assigned to the cutting tool with a value of 553.8N.

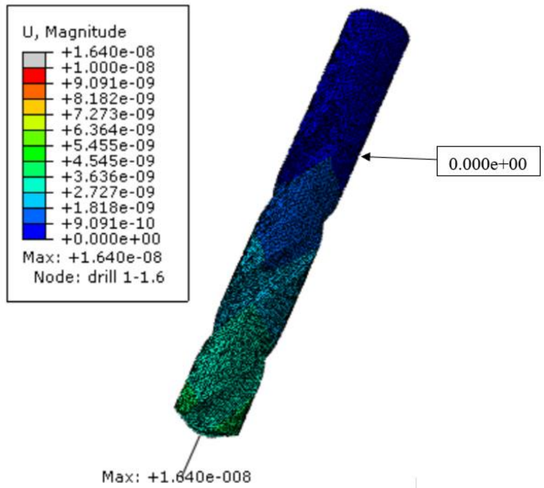

Figure 8: Magnitude result

It shows the part that have a load acting on it are the part that have the tendency to deflect the most. Where the maximum value magnitude on the cutting tool is 1.640e-08 and the minimum value of magnitude is 0.0000e+00 where it does not have any magnitude value acting on the cutting tool. Based on the results obtain from the simulation, the maximum displacement value is 1.640e-08. If the maximum deflection is calculated by formula,

$$F_z = 553.8, L1 = 67, L2 = 100, C = 7.93, D1 = 14, D2 = 14, N = 0.974$$

$$Deflection_{max} = 7.93\frac{553.8}{180 \times 10^9}\left[\frac{67^3}{14^4} + \frac{(100^3 - 67^3)}{14^4}\right]^{0.974}$$

$$Deflection_{max} = 5.835 \times 10^{-7}$$

$$C = \left|\frac{x_2 - x_1}{x_1}\right|$$

$$C = \left|\frac{(1.640 \times 10^{-8}) - (5.835 \times 10^{-7})}{(5.835 \times 10^{-7})}\right| = 97.2\%$$

From the result obtained, it can be seen that the percentage of error for the simulation are quite high. This is due to some lacks of parameters are assigned to the cutting tool and also for the simulation. Accompanying discussions that further explain observations of the results are usually placed immediately below the results paragraph. Only elastic properties were assigned to the cutting tool and fixed boundary condition which means the cutting tool are not rotating on any axis and it is a static study towards the cutting tool. there is no contact surface between the cutting tool and workpiece. Where no central damage can be confirmed if there is no workpiece touched with the cutting tool. Other than that size of the mesh of cutting tool also effects the accuracy of the results. Mesh size is a key issue in finite element analysis. The mesh size is determined by the amount and precision of meshes required for element meshing. The structure and failure theories of the material may be affected by both the statistical and analytical aspects of scaling sizes. The mesh size used in this study are not small enough because in order to do a small mesh the it will slow down the device used to run the simulation. It is effective to use a small mesh size in order to acquire an accurate result but it needs a high-performance laptop or device. various of parameters needed to be assigned and tested in order to get the accurate data for the simulation. Where suitable parameters for cutting tool would be obtained in order to prevent or minimise the tool deflection occurred and path of the cutting tool is correct. Lastly, the factors that might affect the high percentage of error is because study that have been done on the cutting tool is a 3D simulation but the formula used to find the percentage error is a 2D formula where 2D study is well known because of its simplicity.

4. Conclusion

The first objective of this study is to develop simulation procedure to determine the stress and deflection of the cutting tool. Therefore, in order to achieve the objective can be seen in this study's methodology where the dimensions are drawn correctly after that the material property for the cutting tool will be assigned based on the previous study followed and also boundary condition for the cutting tool. All steps are being followed correctly in order to ensure there will be no error when the simulation is running. For the second objective is to analyse the deflection and stresses acting on cutting tool. From this study the maximum value of stress acting on the cutting tool is 4.707e+03 while for the maximum value of displacement is 1.640e-08. Various of cutting parameters needed to be simulate so that the most suitable cutting parameters can be obtain in order the deflection and the stress acting on the cutting tool can be minimised. Where for this study no cutting parameters are being included. Therefore, the percentage error for this study is high with a value of 97.2%. Such as the optimum cutting speed and feed rate where the cutting speed cannot be too fast and too slow. Lastly is the formula used is need to be in 3-dimensional formula.

Acknowledgement

The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its support in this research.

References

[1] M. Anbarasan, N. Senthilkumar, and T. Tamizharasan, "Modelling and Simulation of Conventional Drilling Process using Deform 3D," vol. 5, no. 2, pp. 9–16, 2019.

[2] L. Li, H. Xue, and P. Wu, "Experimental Study on the Axis Line Deflection of Ti6A14V Titanium Alloy in Gun-Drilling Process," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 301, no. 1, 2018,

[3] K. S. Woon, A. Chaudhari, M. Rahman, S. Wan, and A. S. Kumar, "The effects of tool edge radius on drill deflection and hole misalignment in deep hole gundrilling of Inconel-718," *CIRP Ann. - Manuf. Technol.*, vol. 63, no. 1, pp. 125–128, 2014,

[4] H. Chanal, E. Duc, and P. Ray, "A study of the impact of machine tool structure on machining processes," *Int. J. Mach. Tools Manuf.*, vol. 46, no. 2, pp. 98–106, 2006,

[5] R. Teimouri and S. Amini, "Analytical and experimental approaches to study elastic deflection of thin strip in ultrasonic-assisted drilling process," *Proc. Inst. Mech. Eng. Part E J. Process Mech. Eng.*, vol. 233, no. 1, pp. 21–34, 2019,

[6] T. R. Kramer, "Automatic generation of NC-code for hole cutting with in-process metrology," pp. 45–52, 1989,

[7] D. Biermann and I. Iovkov, "Investigations on the thermal workpiece distortion in MQL deep hole drilling of an aluminium cast alloy," *CIRP Ann. - Manuf. Technol.*, vol. 64, no. 1, pp. 85–88, 2015,

[8] E. Bahçe and B. Özdemir, "Determination of tool deflection in drilling by image processing," *IET Image Process.*, vol. 13, no. 13, pp. 2487–2494, 2019,

[9] A. Yıldız, A. Kurt, and S. Yağmur, "Finite element simulation of drilling operation and theoretical analysis of drill stresses with the deform-3D," *Simul. Model. Pract. Theory*, vol. 104, no. February, p. 102153, 2020,