sciendo
Mechanics and Mechanical Engineering 2019; 23:287–290

Research Article

Prashant Elango* and K. Prakash Marimuthu

Numerical Validation of Drilling of Al6061-T6 with Experimental Data

https://doi.org/10.2478/mme-2019-0038
Received May 28, 2018; revised Sep 10, 2018; accepted Nov 20, 2018

Abstract: Drilling is a cutting process that uses a drill bit to cut a circular profile in workpiece. Forces acting on the drill bit reduce its life expectance. Analysis of forces acting on the drill bit during drilling prevents the tool from failing prematurely because of wear and excess feed rate. Excess feed rate can induce excessive internal stress on both the tool and workpiece. This paper aims to study the effects of reaction force acting on a drill bit during drilling of Al6061-T6. A numerical finite element simulation study is performed with commercially available software called Abaqus. Simulation results depend on the right choice of material property such as Johnson–Cook material property and Johnson–Cook damage property. Validation of material property is achieved by comparison of experimental results with simulative results. Reaction force acting against the drill bit during drilling is compared.

Keywords: Abaqus, Finite element, Numerical simulation, Reaction forces, Johnson–Cook

1 Introduction

Drilling is predominately common in manufacturing sector. Simulation of machining process contributes to an efficient design of product. Evaluation of cutting forces during drilling helps in reducing residual stress [5] induced in workpiece and premature failure of drill bit. This paper aims at evaluating reaction forces on the drill bit during drilling of Al6061-T6 through numerical simulation. Drill bit has two main areas which come in contact with the workpiece during drilling; they are main cutting lips and chisel edge. The reaction forces [6] acting in these two areas are experimentally recorded with the help of thrust force dynamometer. The reaction forces at the tip of drill bit are recorded in simulation. This tip gives direct reaction force as the surface area at the tip is very minimal. Heat generated during drilling is assumed to be completely removed with the use of coolant. No heat models are considered in drilling simulation.

2 Model

A 3D model is developed using the commercial finite element software Abaqus. The finite element model is based on Lagrangian formulation [4] with explicit integration method [1]. It is assumed that heat generated during drilling because of friction is removed actively with coolants. So, a no heat generation model is considered. The drill bit is modelled as a rigid body with initial point mass inertia of 0.5 tones. As this model follows millimetre, tones and second as primary base units, this approach prevents error because of float variable limitation in the central processing unit. The workpiece is modelled as a deformable part of a rectangular slab with dimensions of 36×36×11 mm. The drill bit and workpiece are assembled according to the drilling model. Translation and rotational components along the three axes system are arrested in the workpiece. In the drill bit, translation and rotational components along the respective axis are permitted with 0.2 mm/rev and 170 rpm. Interaction model is formed with surface-to-surface contact to initiate material removal. A tangential behaviour is defined under contact property options with penalty friction formulation with friction coefficient of 0.45. To have accelerated explicit simulation, penalty contact method is defined instead of kinematic contact model. Further acceleration is achieved by use of mass scaling option in step module. Mass scaling is defined for the whole model with target-type time increment of 0.001 seconds and it is applied throughout the explicit step. A rigid body constraint is defined for the drill bit to mark as rigid with appropriate selection of body and reference point.

K. Prakash Marimuthu: Department of Mechanical Engineering, Amrita School of Engineering Amrita Vishwa Vidyapeetham, Coimbatore, Tamilnadu, India; Email: prakashkmuthu@gmail.com
***Corresponding Author: Prashant Elango:** Department of Mechanical Engineering, Amrita School of Engineering Amrita Vishwa Vidyapeetham, Coimbatore, Tamilnadu, India

2.1 Simulation

The von Mises stress is depicted to show the process of drilling. General assembly of the drilling model is depicted in Figure 1. A medium of mesh is used in both the drill bit and workpiece. This mesh is used to overcome the computational challenges presented during a fine mesh. Chip formation is observed from entry to exit. The chips formed are very small and cannot be seen. The chips can be visualized clearly, only when a fine mesh is utilized with very low stable increment time, approximately around $1{\times}10^{-6}$ or even lesser. The after-effects of stress is shown before, after and during drilling on Al6061-T6 (Figures 2–5).

2.2 Computational challenges

The simulation that ran on an Intel second-generation mobile processor took approximately 2 days to complete. Subsequent models were created with only change in drill bit diameter. The simulation in whole with consideration of subsequent models took approximately around 8 to 16 days. The stable time increment is manually controlled to reduce overall simulation time by trial-and-error method.

3 Material property

Many researchers have simulated metal cutting operation of Al6061-T6 [1]. Johnson–Cook plasticity model is chosen to avoid entering stress–strain data curves. Johnson–Cook plasticity model goes well with Johnson–Cook damage criteria. The Abaqus input model file is attached in reference section. All required values are provided in Table 1.

Table 1: Material Property of Al6061-T6

S. No.	Property	Value
1.	Density	2700 kg/m^3
2.	Young's modulus	70 GPa
3.	Poisson's ratio	0.33
4.	Yield stress	260 MPs
5.	Johnson–Cook plasticity model	
	A	324.1 MPa
	B	113.8 MPa
	N	0.002
	M	1.34
	C	0.011
6.	Johnson–Cook failure model	
	*D*1	−0.77
	*D*2	1.45
	*D*3	−0.47
	*D*4	0
	*D*5	1.6

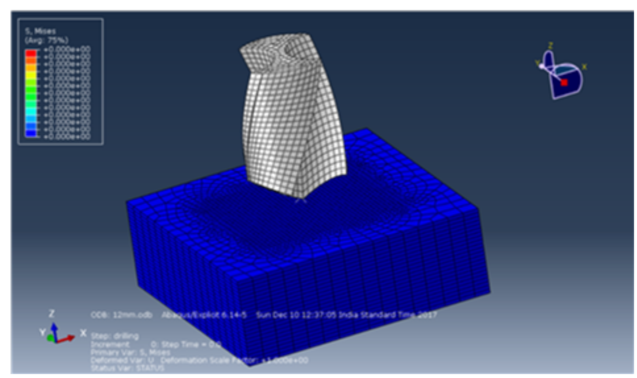

Figure 1: The assembly of drilling model.

4 Visual simulation walkthrough

Figures 1–5 show the procedure by which simulation is completed.

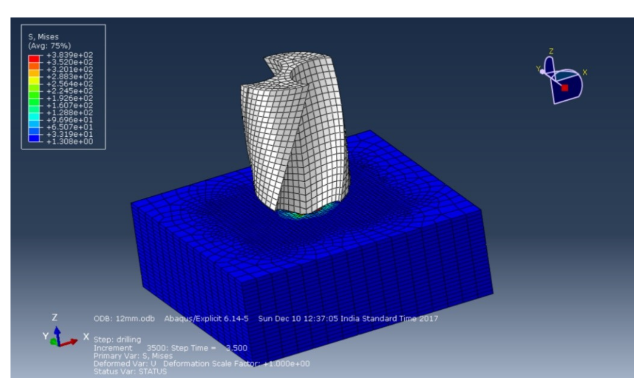

Figure 2: von Mises stress at entry of drill bit.

5 Mesh

The drill bit is meshed with R3D4 element which is a 4-node 3D bilinear quadrilateral with global mesh size of 1 mm. The workpiece is meshed with C3D8R element which is an 8-node linear brick element with reduced integration and default hourglass control with global mesh size of 2

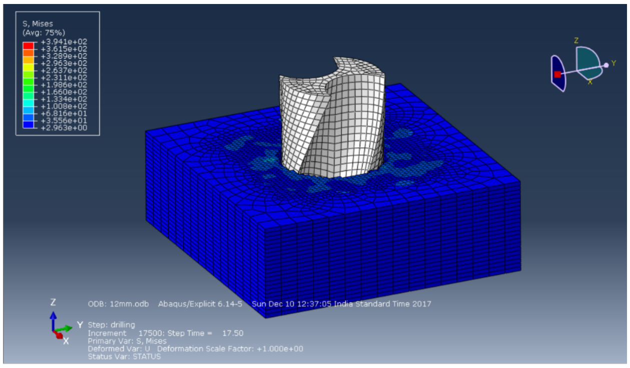

Figure 3: von Mises stress at mid-section of drill bit.

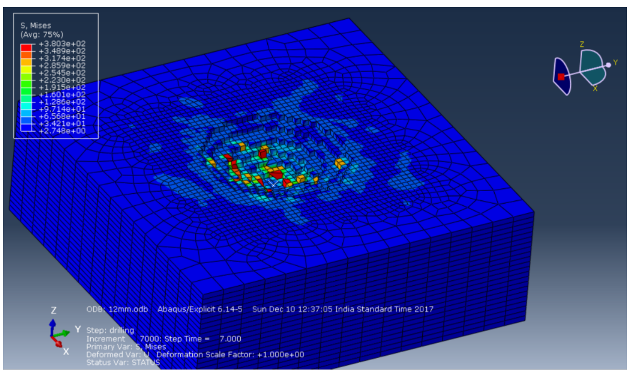

Figure 4: von Mises stress at entry without drill bit.

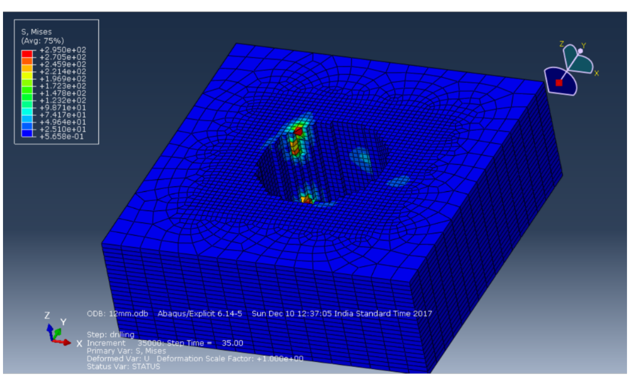

Figure 5: von Mises stress at completion of drilling.

mm. A focused mesh is created near the drilling region with mesh seed of 0.5 mm.

6 Results

Drill experimental values for reaction force were gathered from a published journal [3]. Numerical method results of reaction force were obtained at the tip of the drill bit. The average reaction force starting from entry of drill bit on workpiece till complete penetration on the other side of parallel surface is mentioned. The experimental and simulative reaction forces were obtained with variation in drill diameter and fixed spindle speed. Detailed analysis is reported in Table 2. The table contains variation of drill bit diameter ranging from 6 to 12 mm with corresponding experimental and simulative reaction forces.

Table 2: Variation of Reaction Force with Drill Bit Diameter

Drill Bit Diameter *D* (mm)	Reaction Force (N)	
	Experimental	Simulation
6	317.19	280.95
8	412.89	393.78
10	549.36	585.58
12	686.39	612.15

7 Conclusion

The experimental and simulative values of reaction force are in close agreement. From Figure 6, the numerical method values have gross error of approximately around 5% to 10%. The mentioned model can lead to prediction of reaction forces with different drill bit diameters through simulation, with minimal gross error, and thus to reducing the overall cost of experimental analysis for varying drill bit diameter. Experimental analysis is not easy, as it requires greater effort in recording reaction force via thrust force dynamometer. Simulative analysis reduces the effort of experimenter and thus yields precise numerical values.

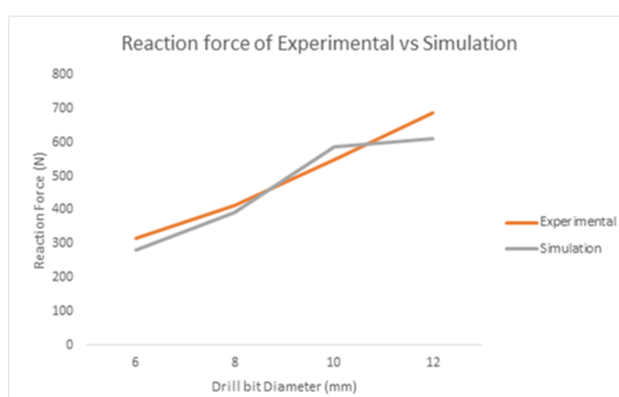

Figure 6: Visual trend of reaction force with drill bit diameter.

8 Future work

It has been observed that coarse mesh leads to approximate simulative reaction force. Further, increase in mesh size results in inaccurate values. This error because of fine mesh has to be investigated in further work.

References

[1] **Boldyrev, L.S., Shchurov, L.A., Nikonov, A.V.**: Numerical Simulation of the Aluminum 6061-T6 Cutting and the Effect of the Constitutive Material Model and Failure Criteria on Cutting Forces' Prediction, *Procedia Engineering*, **150**, 866 – 870, **2016**.

[2] **Isbilir, O., Ghassemieh, E.**: Finite Element Analysis of Drilling of Titanium Alloy, *Procedia Engineering*, **10**, 1877–1882, **2011**.

[3] **Kumar, K.P., Kishore, P., Laxminarayana, P.**: Prediction of Thrust Force and Torque in Drilling on Aluminum 6061-T6 Alloy, *International Journal of Engineering Research & Technology*, **2**(3), **2013**.

[4] **Prakash Marimuthu K., Prasada, H.P., Kumar C.S.**: 3D Finite Element Model To Predict Machining Induced Residual Stresses Using Arbitrary Lagrangian Eulerian Approach, *Journal of Engineering Science and Technology*, **13**(2), **2018**.

[5] **Krishnakumar, K.P., Prakash Marimuthu, P., Rameshkumar, K. and Ramachandran, K.I.**: Finite Element Simulation of Effect of Residual Stresses During Orthogonal Machining Using Ale Approach, *Int. J. Machining And Machinability of Materials*, **14**(3), **2013**.

[6] **Prakash Marimuthu P.K., Thirtha Prasada, H.P., Kumar, C.S.**: Force, Stress Prediction In Drilling Of AISI 1045 Steel Using Finite Element Modelling, IOP Conf. Series: Materials Science and Engineering, **225**, 012030, **2017**.

[7] **Prakash Marimuthu, P.K., Chethan Kumar C S, Thirtha Prasada, H.P.:** Mathematical Modelling To Predict The Residual Stresses Induced In Milling Process, *International Journal of Mechanical and Production Engineering Research and Development*, **8**(1), 423-428, **2018**.