Transactions of the Nigerian Association of Mathematical Physics
***Volume 14, (January - March, 2021 Issue*), pp89 –94**
© Trans. of NAMP

INVESTIGATION AND ANALYSIS OF TURNING ELEMENT USING FINITE ELEMENT METHOD

[1]ALIEMEKE B.N.G.,[2] UMUFO O.,[3] OGHENEOVO R.J. and [4]ENODE I.T.

Department of Mechanical Engineering, Auchi Polytechnic, Auchi, Nigeria

Abstract

Finite element method is an effective numerical technique which prove to very effective for analyzing problems of contacting surfaces such as chip morphology and process variables i.e. forces, temperature, stresses and strain .To study the effect of the factors such as spindle speed, depth of cut and feed rate on the material removal rate, for analyzing and prediction of the performance in machining process using ANSYS software. Great attention was targeted at investigating the equivalent stresses, strain and total deformation developed in the titanium work piece when cut by AISI 1045 turning tool. The equivalent stress on the workpiece was determined to be 1023Mpa. The simulation produced an equivalent (Von mises) strain of 2.5662mm. The workpiece sustained a total deformation of 6.108mm and 0.0000012mm as its maximum and minimum values respectively.

Keywords*: Finite element, Machining, equivalent stress and deformation*

1.0 Introduction

Turning is a form of machining process which is used to design, reduce and create cylindrical parts by cutting away unwanted materials [1].It involves the removal of unwanted materials in a workpiece in order to produce a desired product. The turning process requires a turning machine and cutting tool [2].

Turning is reputed to be time consuming that is why improved technology is needed for the machining of hardened steels that will provide high material removal rate and also to increase flexibility in terms of part geometry [3] .As a result recent developments of advance cutting tool materials that engenders time reduction has increased significant.

The finite element method is a widely applied numerical technique for solving complex physical phenomenon governed by the differential equations [4]. Many of the practical engineering problems such as structural, thermal, magnetic and fluid mechanics can be solved by the finite element method. Moreover, the finite element method is an increasing common tool for engineering designs [5].

In the recent past, finite element method has particularly become the principal tool for simulating turning operations. Finite element method (FEM) is of great necessity to the field of engineering, especially in the aspect of predicting the failure of a structure, object or material under stress, because it allows the designer or manufacturer to understand all the theoretical stresses within the structure [6].

A study on the changes of cutting forces depending on cutting parameters (cutting speed, depth of cut and feed rate) was investigated by [7]. The study utilized the machining of DIN 1.2344 tool steel (55±1 HRC) with uncoated ceramic inserts. The stress distribution on cutting tools was analysed based on finite element ANSYS software.

An orthogonal cutting process in shaping operations for AISI 1020 steel was modeled by [8]. The developed model applied a Lagrangian approach in processing the AISI steel work piece. The model was able to predict initial chip formation, chip growth and steady state chip formation and does not need any prior assumption regarding the chip flow[9]. The accuracy of the model was verified by comparing the various process parameters. In addition, tool performance and surface integrity of the work piece was analysed using stress distribution in the work piece and the cutting tool which is a major objective of this current study.

Similarly, a mathematical model for machine turning was developed with the help of DEFORM 3D simulation software by [10].The developed model was applied in determining the effect of cutting parameters such as cutting speed, feed and depth of cut on cutting forces. Explicit dynamics analysis system of ANSYS workbench was used to obtain results for FEM cutting tool and work piece geometry.

Correspondence Author: Aliemeke B.N.G., Email: aliemeka@yahoo.com, Tel: +2348030648051

A safe prediction of the near surface residual stresses using numerical approach was carried out by [11] in the turning of AISI 52100 alloys steel. Effect of cutting parameters, namely cutting speed and depth of cut on induced residual stresses in machined surface was investigated by using ABAQUS/CAE 14.0 software. Explicit Dynamics time integration with adaptive meshing finite element method was employed in the simulation.
A study on the surface roughness and cutting forces evolution in hard turning of AISI52100 steel conducted by [12] showed that the feed rate and the cutting speed mainly affect the surface roughness while the cutting forces are mainly influenced by the depth of cut. Similarly, an optimization strategy was developed by [13] to determine optimal cutting parameters to improve machining characteristics in radial orthogonal turning of AISI steel. To achieve this, first, a predictive finite element tool is developed to model cutting temperature, cutting force and residual stresses. Orthogonal turning experiment is conducted to measure machining forces, chip thickness and stresses to validate the developed predictive model [14]. The validated model was used to construct response functions for practical and efficient implementation of the optimization problem. This present study is aimed at investigating the equivalent stresses, strain and deformation developed in titanium work piece when cut by AISI 1045 turning tool.

2.0 Material and Methods

In this research we will use Titanium metal as the work piece and AISI 1045 steel as the material for turning tool. The work piece was machined and the Explicit dynamics analysis systems of the ANSYS finite element software was used in the designing of the experiment. The Colchester lathe machine was also used. The input parameters for the finite element software are in Table 1.

Table 1: Explicit dynamics input parameters

Work piece parameters	
Material	**Titanium**
Length	**7mm**
Width	**5mm**
Height	**4mm**
Tool parameters	
Material	**AISI 1045**
Rake angle	**7^o**
Rake face length	**3mm**
Relief angle	**10^o**
Relief face length	**2mm**
Process Parameters	
Length of cut	**2.0mm**
Depth of cut	**0.1mm, 0.2mm and 0.3mm**
Feed	**0.5mm/s**
Friction coefficient	**0.4**
Element size	**1.0mm**

2.1 Design of Experiment using Finite Element Method

In order to improve the performance of the design, it is essential to arrive at the proper level combinations of the input parameters that affect the performance of the design. At the same time, it may not be necessary to consider the design variables which do not contribute significantly. Many of the standard finite element software / packages do not have facilities to check the contribution or significance of a design variable on the performance parameter. One way to solve this problem is to use design of experiments for varying the input design parameter values and predict the significance of each variable by studying the change in the objective function (performance parameter).
The Taguchi method such tool for conducting experiments using a statistical approach to understand the significance of independent factors and levels. The main advantage of Taguchi method is that the number of experiments conducted in most of the cases is lesser than that of any other experiment using a statistical approach.
The workpiece and the tool is made up on Titanium and high manganese steel respectively. the tool geometry was modeled in Solid works CAD system and imported to the ANSYS Workbench. An assembly of the workpiece and turning drill tool shown in the CAD system third angle orthographic projection in Figure 1 was analysed using the Explicit Dynamics Workbench system. Explicit dynamics was chosen for its ability to determine dynamic response for structures under time dependent load [12]. Material properties from the Workbench engineering data was inputted to the imported workpiece and the turning tool. Also, the coordinate setting attributes were added to assembly parts separately.

3.0 Results and discussion

The third angle orthographic projection and isometric drawing of the assembly are shown in Figure 1 and 2 respectively.

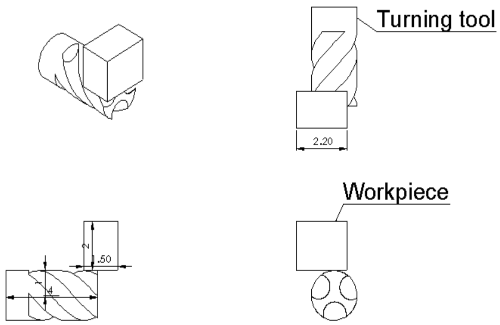


Figure. 1: Orthographic progression of the workpiece and the tool

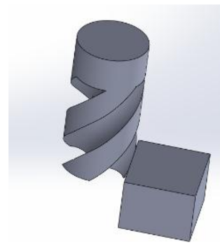


Figure 2: Isometric drawing of the work piece and tool

The turning tool was constrained with rotational movement while the work piece had translational movement towards the work piece. Furthermore, displacement parameters were inputted to the two entities of the assembly. The turning tool was made to rotate in three revolutions (1080º) and the titanium workpiece was made to displace towards the tool by 2.5mm as shown by Figures 3 and 4 respectively.

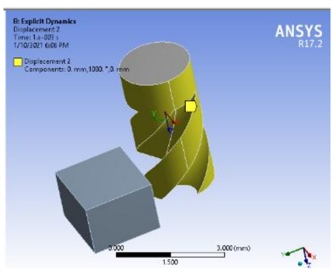


Figure 3: Cylindrical (Rotational) movement in ANSYS Co-ordinate system

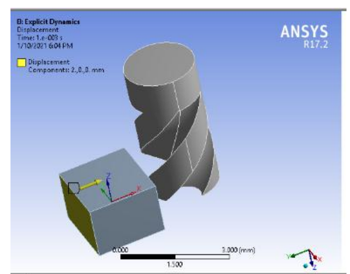


Figure 4: Translational movement in ANSYS Co-ordinate system

The workpiece and the tool were meshed as shown in Figure 5. The number of nodes and elements applied in this study are 3328 and 13527 respectively. The size of element used is 1mm.

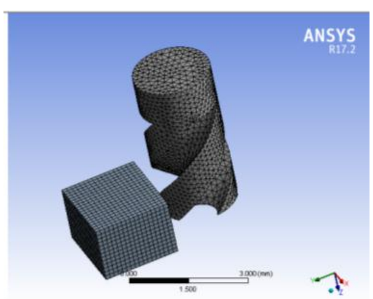


Figure 5: Meshed structure in ANSYS

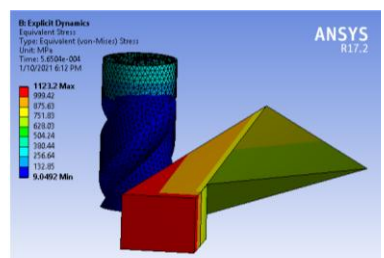


Figure 6: Equivalent Von mises stress distribution

3.0 Result and Discussions

The total equivalent (Von Mises) stress impacted on the workpiece by the turning tool has a maximum value of 1023Mpa and a minimum value of 132.85Mpa as shown in Figure 6.The result obtained is similar to that of [15].

Table 2: Equivalent stress values from Explicit Dynamics

Time	Minimum(Mpa)	Maximum(Mpa)
1.1755e-038	0	0
5.000e-005	1.1683	1020.7
1.0e-0.04	1.5782	1140.9
1.5e-0.04	1.4584	1169.2
2.0e0.04	3.0434	1183.3
2.5e-0.04	5.1742	1184.98
3.0e-0.04	6.404	825.98
3.5e-0.04	6.3134	1204.4
4.0e-0.04	8.7471	989.87
4.5e-0.04	7.3471	1192.5
5.0e-0.04	9.5452	1154.8
5.5e-0.04	9.3745	1123.1
5.65e-0.04	9.0492	1123.2

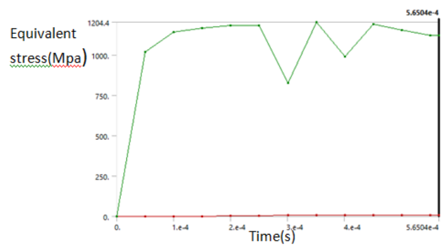


Figure 7: A plot of Maximum stress values with time

The total deformation experienced by the workpiece was determined to be 6.108mm and 0.0000012mm as its maximum and minimum values respectively as shown in Figure 8.The theoretical values obtained for total deformation are shown in Table 3. The Figure 9 showed The plot of the maximum deformation values against the time taken for deformation of the workpiece to occur while Table 3 showed the obtained values. It showed that the deformation of the workpiece increases progressively with time.

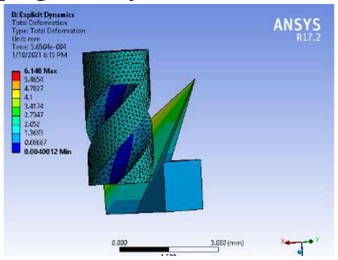


Fig. 8: Total deformation

Time [s]	Minimum [mm]	Maximum [mm]
1.1755e-038	0.	3.5108e-016
5.0001e-005	2.2397e-003	0.92165
1.e-004	3.9626e-003	1.6424
1.5e-004	4.8179e-003	2.0051
2.e-004	4.6647e-003	1.9307
2.5e-004	3.465e-003	1.4442
3.e-004	1.5181e-003	1.386
3.5e-004	7.7776e-004	2.2331
4.e-004	2.8658e-003	2.2421
4.5e-004	4.3647e-003	3.359
5.e-004	4.9007e-003	3.8107
5.5e-004	4.3619e-003	4.9802
5.6504e-004	4.0012e-003	6.148

Table 3: Total deformation values in Explicit dynamics Analysis

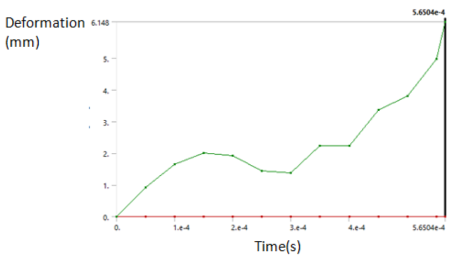
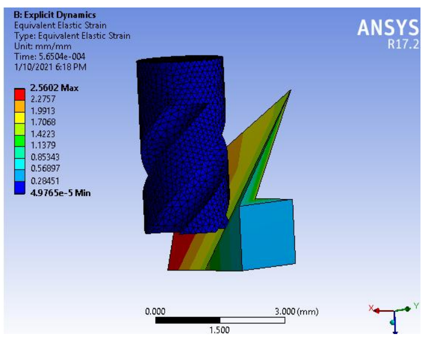


Figure 9: A plot of Maximum deformation values with time

The simulation produced an equivalent (Von mises) strain of 2.5662mm and 0.00048mm as maximum and minimum values respectively as depicted in Figure 10. The obtained values are shown in Table 3. Equivalent strain values ploted against time to accomplish strain reveals that the strain increases with time as shown in Figure 11.



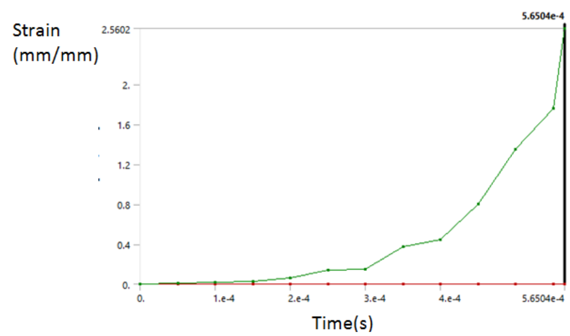


Figure 10: Equivalent strain distribution

Figure 11: Plot of maximum strain values time

Table 3: Equivalent strain values

Time [s]	Minimum [mm/mm]	Maximum [mm/mm]
1.1755e-038	0.	0.
5.0001e-005	9.3797e-006	8.7315e-003
1.e-004	1.3876e-005	1.3194e-002
1.5e-004	1.8912e-005	2.5604e-002
2.e-004	2.2765e-005	6.0423e-002
2.5e-004	3.1949e-005	0.14052
3.e-004	3.6144e-005	0.14874
3.5e-004	3.5481e-005	0.37628
4.e-004	5.1505e-005	0.44204
4.5e-004	4.243e-005	0.80205
5.e-004	4.8888e-005	1.3539
5.5e-004	4.9377e-005	1.7631
5.6504e-004	4.9765e-005	2.5602

4.0 Conclusion

An investigation of the effect of AISI 1045 steel turning tool on a titanium work piece to ascertain the effect of stresses, strain and total deformation was effectively carried out. The study showed that an increase in cutting force leads to increase in equivalent stress and deformation. The equivalent stress of 1023Mpa imparted on the titanium workpiece was noticed to be very large and it resulted in a deformation of 6.108mm which is considerably high for a 7mm long work piece.

REFERENCES

[1] Lin. W.S. (2008). The study of high speed fine turning of austenitic stainless steel. *Journal of Achievements in Materials and Manufacturing*, Vol. 27, No. 2, pp. 10-12

[2] Kryzhanivskyy, V. Bushlya, O. Gutnichenko, I., Petrusha, A. and Stahl, J.(2015). Modelling and Experimental Investigation of Cutting Temperature when Rough Turning Hardened Tool Steel with PCBN Tools, Journal for Material Process Technology, Vol. 214, No. 4, pp. 489–495.

[3] Al-Ahmari, A.M.A.(2007). Predictive machinability models for a selected hard material in turning operations. *Journal of Material Processing Technology*, Vol. 190, No. 3, pp. 305-311.

[4] Sadeghifar, M., Sedaghati, R. Jomaa, W. and Songmene, V.(2018). Finite element analysis and response surface method for robust multi-performance optimization of radial turning of hard 300M steel, International Journal for Advanced Manufacturing, Vol. 94, No. 14, pp. 2457–2474.

[5] Borsos, B., Csorgo, A., Hidas, A. and Kotnyek, B.(2017). Two-Dimensional Finite Element Analysis of Turning Processes, *Periodica Journal for Polytechnics*, Vol. 61, No. 1, pp. 44-54.

[6] Chen, G., Kong, L. and Liu, X. (2014). Finite element analysis and experiments of saw chip formation for bearing steel GCr15 in hard cutting process, Journal for Vibroengineering, Vol. 6, No. 3, pp. 2834–2842**.**

[7] Gunay, M., Korkmaz, E. and Yaşar, N. (2017). Finite element modeling of tool stresses on ceramic tools in hard turning, Journal Mechanika, Vol. 23, No. 3, pp. 432–440.

[8] Bashistakumar, M. and B. Pushkal,(2018) "Finite Element Analysis of Orthogonal Cutting Forces in Machining AISI 1020 Steel, Journal for Engineering Science, Vol. 5, No. 2, pp. 8-15.

[9] Kumar, R. and Srinivasu, G. (2017). Finite Element Analysis of Cutting Forces Generated in Turning Process using Deform 3D Software, Journal for Materials, Vol. 4, No. 8, pp. 8432–8438.

[10] Duttraj, P. and Vinayagam, B. K., (2013). Finite Element Method-Based Machining Simulation for Analyzing Surface Roughness During Turning Operation with HSS and Carbide Insert Tool, Arab Journal for Science Engineering, Vol. 23, No.38, pp.1615-1623.

[11] Basil, K.M. & Tina, R. (2014). Optimization of Cutting parameters in Hard Turning of AISI 4340 Steel" , *International Journal of Innovative Research in Advanced Engineering* Vol.1, No. 8, pp. 24-30.

[12] Beno, J., (2011). Outline of FEM Simulation and Modelling of Hard Turning Process, Journal for Material Processing Technology, Vol. 86, No. 2, pp. 14–21.

[13] Saravanan, R.P. & Vijayakumar, K.(2008), Machining Parameter Optimization for Turning Cylindrical Stock into a Continuous Finished Profile Using Genetic Algorithm and Simulation, *International journal of advanced manufacturing technology* Vol. 21, No.1, pp. 1-9.

[14] Outeiro J. C., Umbrello D., and Saoubi R.,(2006). Experimental and numerical modelling of the residual stresses induced in orthogonal cutting of AISI 316L steel, International Journal of Machine Tools & ManufactureVol. 46, No.147, pp. 1786 – 1794

[15] Rajeshi, M., Prasad, V. S. , and Rao, N.G.(2016) Investigation and Analysis Of TurningTool (High Manganese Steel) using Finite Element Method, Internaional Journal of Engineering and Management Research, Vol. 6, No. 6, pp. 178-184