

Finite Element Analysis of Von Mises Stresses & Deformation at Tip of Cutting Tool

Maheshwari N Patil

Dr.D.Y.Patil Institute of Engineering & Technology, Ambi
Pune, India
maheshwari23589@gmail.com

Shreepad Sarange

Dr.D.Y.Patil College of Engineering ,Lohegaon
Pune, India
sarangeshreepad@rediffmail.com

Abstract— Finite element analysis based techniques are available to analyse the effect of temperature and cutting forces (Horizontal force, vertical force) on the tip of tool using three dimensional model of an single point cutting tool in ANSYS software and simulate cutting processes and offer several advantages including prediction of temperatures, distribution of von misses stresses and deformation of tip of single point cutting tool using tool forces, estimation of tool wear and residual stresses on machined surfaces, optimization of cutting tool geometry and cutting conditions. However, work material flow stress and friction characteristics at cutting regimes are not always available. This paper consist temperatures measured by experimental setup during machining and calculated forces analytically for the same condition. This paper utilizes a metal cutting forces model and presents an improved methodology to characterize work material flow stress and friction at primary and secondary deformation zones around the cutting edge. In this paper, constitutive work flow stress model is used to characterize work flow stress in deformation zones. The model is based on estimation of the normal stress distribution over the tool. The tool forces over the tool can either directly be entered in FEA software or used in determining distribution of stresses and deformation at the tool tip. The methodology is practical and estimates the unknowns of the work material constitutive model.

Keywords— finite element analysis; ANSYS; temperatures ;metal cutting forces; von misses stresses; deformation at the tool tip.

I. INTRODUCTION

In this paper, an advanced FEM simulation technique is utilized to investigate the physical cutting process for predicting von mises stresses and deformation of tip of single point cutting tool. There has been a great deal of research on single point cutting tool analysis and large body of literature on single point cutting tool has been published. FEM based simulation of machining processes has been providing a better understanding for heat generation in cutting zones, and resulting stress and temperature fields. The premise of the numerical models is to be able to lead predictions in machinability, tool wear, tool failure, and surface integrity on the machined surfaces. Success and reliability of numerical models is basically dependent upon work material flow stress models in function of strain, strain rate and temperatures, also on friction parameters between the tool and work material interfaces. There has been a considerable amount of research devoted to analyse the effect of machining variables such as speed, depth of cut, temperature, and cutting forces on the tool by using modelling and analysis software ANSYS. Predicting the numerical process, temperature and stress fields accurately play a pivotal role for predictive process engineering for high speed machining processes. Tool edge geometry is very important, because its influence on obtaining most important tool life and surface integrity is considered significant.

II. INTRODUCTION OF SINGLE POINT CUTTING TOOL

The single point cutting tool used for turning, boring, shaping and planning operations, that is, tool used on lathes, boring machines, shaper, planer etc. are single point cutting tool.

A solid model of Single point cutting tool as shown in figure 2.1 It is generated using the below parameters and values by ANSYS. Save the file in ANSYD format and used this model for analysis purpose in simulation applications.

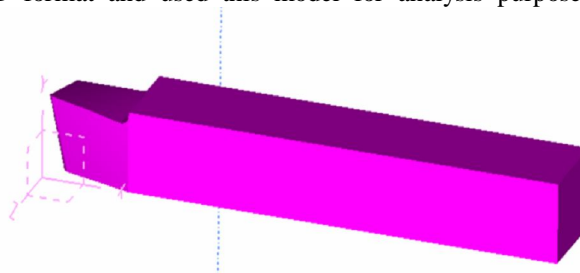


Fig 2.1 Solid Model of Single Point Cutting Tool

TABLE 2.1 SPECIFICATIONS OF SINGLE POINT CUTTING TOOL

Sr.No	Parameters	Value
1	Back rack angle	8 degree
2	Side rake angle	10 degree
3	End relief angle	6-8 degree
4	Side relief angle	7-9
5	Standard shank	B×h = 16×20
6	Tool length depends on cross section	100 to 500 mm
7	Nose radius	1.2 to 1.6

III. BRIEF OVERVIEW OF FINITE ELEMENT ANALYSIS

An analytical solution is a mathematical expression that gives values of the desired unknown quantity at any location in a body (or structure) and as a consequence, it is valid for an infinite number of locations in the body. However, analytical solutions can be obtained only for simple engineering problems. It is extremely difficult and many a times impossible to obtain the exact analytical mathematical solutions for many complex engineering problems. In such cases, the technique known as Finite Element Method (FEM) is used.

3.1 Applications of FEA

This method was original developed for structural analysis. However the general nature of the theory on which it is based has made problem.

In other field of engineering such as Mechanical design, Mechanical vibration, Heat transfer, Fluid flow, soil and Rock Mechanics, Bioengineering, Aeronautics.

The different types of analysis carried out by Finite element Analysis are:

1. Static linear analysis.
2. Static Non-linear analysis.
3. Dynamics linear analysis.
4. Dynamics Non-linear analysis.
5. Fluid flow analysis.
6. Thermal analysis.

3.2 Steps Involved In the Finite Element Analysis

1. Discretization of the continuum: -

The continuum is the body, structure or solid being analysed. Discretization is the process of dividing the body into a finite number of elements. These elements may have, its shape and refinement are required to be decided before discretization. The success of the Discretization, which is also known as meshing.

2. Formation of element stiffness matrices:-

After the continuum is discretised with the desired type of elements, the element stiffness matrices are formulated for all the elements. The element stiffness matrices in global coordinate system is given by $[K]_e = [L]^T [K1]_e [L]$

3. Formation of global stiffness matrix:-

After the element stiffness matrices in global coordinate system are obtained for all the elements, they are assembled to form the global stiffness matrix $[K]$ for the entire body. $[K] = [K]_e$

4. Formation of global load vector:-

The element force vectors in the global coordinate system $\{f\}_e$ for all the elements are assembled to form the global load vector $\{F\}$ for the entire body $\{F\} = \{f\}_e$

5. Formation of global nodal displacement vector: - The global nodal displacement vector $\{UN\}$ is formed for the entire body.

6. Assembly of global stiffness-nodal displacement-load equations:-

The relation between the global stiffness matrix $[K]$, global nodal displacement vector $\{UN\}$ and global load vector $\{F\}$ is expressed as a set of simultaneous algebraic equations. $[K] \{UN\} = \{F\}$

7. Incorporation of specified boundary condition:-

The specified boundary conditions are incorporated in equilibrium equations by using approaches.

8. Solution of simultaneous equation:-

After including the specified boundary conditions in equilibrium are solved for the unknown nodal displacements by using methods like Gaussian elimination, Gauss seidal and Jacobi iterations.

9. Computation of element strains and stresses:-

Knowing the nodal displacement the element strains and stresses are calculated. The components of strains at any point within the element are given by, $\{\hat{\epsilon}\} = [B] \{UN\}$ Similarly, the components of stresses at any point within the element are given by, $\{S\} = [D] [\hat{\epsilon}]$

Where,

$[B]$ = Element strain-nodal displacement matrix.

$[D]$ = Element stress-strain matrix.

3.3 Modelling of Single Point Cutting Tool in ANSYS

Modeling is the art of representing the object or system. The geometric modeling is defined as the complete representation of a structure with graphical and non-graphical information.

It generates the mathematical description of the geometry and no geometry of a structure in the computer database and an image of an object on the graphics screen.

In geometric modeling, the graphics image of an object is generated a graphics screen of the system by inputting three types of commands to the computer.

- The first type of command generates basic geometric entities such as points, lines and circles
- The second types of command accomplish the transformation of these graphics elements.
- The third type of command causes the various graphics elements to be joined into the desired shape through shape of the objects.

The user construct the input devices, the software then convert such as data into a mathematical representation, which is stored in model database for later use. The model, which is in the database, can be subsequently called for analysis, modification or transfer.

3.4 Selection of CAD Software

There is different software available in market. The classification of software is as follows.

1. CAD software, which performs only CAD work 2D and 3D: -AutoCAD, INVENTER, SOLIDEDGE, and MECHANICAL Desktop etc.
2. CAD/CAM/CAE software: - I-DEAS, PRO-E, CATIA, SOLID EDGE, SOLID WORKS, and UNIGRAPHICS etc.
3. CAD/CAM software, which performs CAD as well as CAM, works both suitable for manufacturing: -DelCAM, Surf CAM, Win Cam, Simitron.
4. Special category of software like analysis: - IDEAS, ANSYS, NISA, NATRAN, and PATRAN etc.

3.5 ANSYS Master Series Application

3.5.1 Design

The Modeler is where initial design concepts are developed. The part that's created here can be used for mass properties calculations, interference checking, assembly modeling, finite element stress analysis, system dynamics, manufacturing, and drafting.

3.5.2 Drafting

Program files can now be created that span tasks. In other words, you can write a macro that begins in the Modeler task and ends in the Drafting task. Because of this new functionality, some reserved words that could not be used in program files affecting solid models are now also applicable to macros affecting 2D drawings. The Statement and Command Descriptions article contains a list of those reserved words.

3.5.3 Simulation

ANYIS Simulation tools give you access to powerful geometry construction and editing capabilities, extensive capabilities for building models, and a wide selection of solutions to simulate real world conditions.

3.5.4 Assembly

You use Assembly to create complex assemblies from the parts created in the Modeler. Assemblies can include separate parts or multiple "instances" of individual parts. Assemblies can have multiple configurations. In each configuration, the system can be analyzed for mass properties and checked for interferences.

3.5.6 Manufacturing

Generative Machining automates numerical control (NC) programming by providing assembly modeling tools - allowing you to construct complete machining setups, associating tool path data with the setup geometry, and generating CL data to direct efficient machining operations.

Sr.No.	Application	Task
1	Design	Modeler Assembly Drafting setup Mechanism Design
2	Drafting	Drawing
3	Simulation	Modeling, Boundary conditions, Meshing, Model solution, Post processing, Optimization
4	Manufacturing	Modeler, Generative machining, Assemble Setup

TABLE 3.1 ANSYS SERIES APPLICATION

3.6 Finite Element Analysis of Single Point Cutting Tool by Using ANSYS

1) At this the model is designed in Master Modeler. First, half the framework is created and then it is completed using Revolve command. Then the tool piece is made on the work piece. Then the cross section command is used followed by Build section command. Then both the parts are extruded using the Extrude command, then we put away the parts to bin using Filter and Fictroixity section command. Then only frame work is then kept and surface by boundary command is used. After that the Join command is used to join the parts perfectly.

2) Now we come to Boundary conditions and use Create FEA Model command to name the existing part as a new part.

3) Now we go to meshing and apply the command Define Shell Mesh, then we use the Mesh Preview icon and use the keep mesh icon to form mesh. Now we return to boundary Condition and select the end of the work piece. Then we specify the Restraint and keep the x-direction as free. After then we create sets end apply force in x, y, & z direction respectively in a combined manner. Then we also apply temperature at three different areas namely Chip-Tool interface, Tip of tool and on the work piece. Then we go to boundary condition and turn on Restraint set command and temperature set-1 which we have created earlier.

4) Now we go to model solution for creating a solution set of the model and then it is solved using the Solve icon.

5) Lastly we entered in Post Processing and use the Display command and select the Stress condition to get required Result. Then it can be utilized and animated to get desired combinations. Also we can use the select Result icons to obtain various combinations of this model. Thus the above step gives us the whole procedure of the working of Model using FEA.

MODELING:-

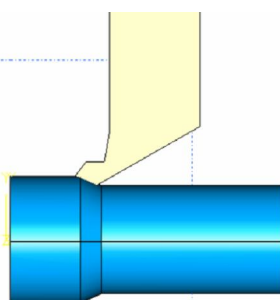


Fig. 3.1 Solid Modelling of Single Point Cutting Tool Mechanism

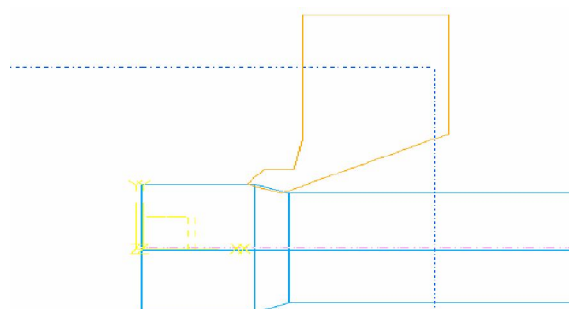


Fig. 3.2 Wire frame Model of Single Point Cutting Tool

3.7 Mesh Generation/ Discretization

During mesh generation changing the master modeler tasks into meshing under the simulation application work of finite element analysis of Single point cutting tool. Mesh is generated by using solid mesh command and entering input values such as element length (2mm length) size and select element shapes (Tetradral element) and keep the mesh by using free meshing option and then it is redefined again for setting accurate results of stresses and deformation.

3.8 Boundary Condition

In metal cutting process maximum temperature is acts on the nose radius and forces acts on the tips of the tool. During the machining tool is clamp at the top surface of the tool. In this way applying boundary conditions such as temperature, forces and clamp constraint apply on the tool. Figure 3.3 shows the 'Finite Element model with boundary condition' applying on the tool.

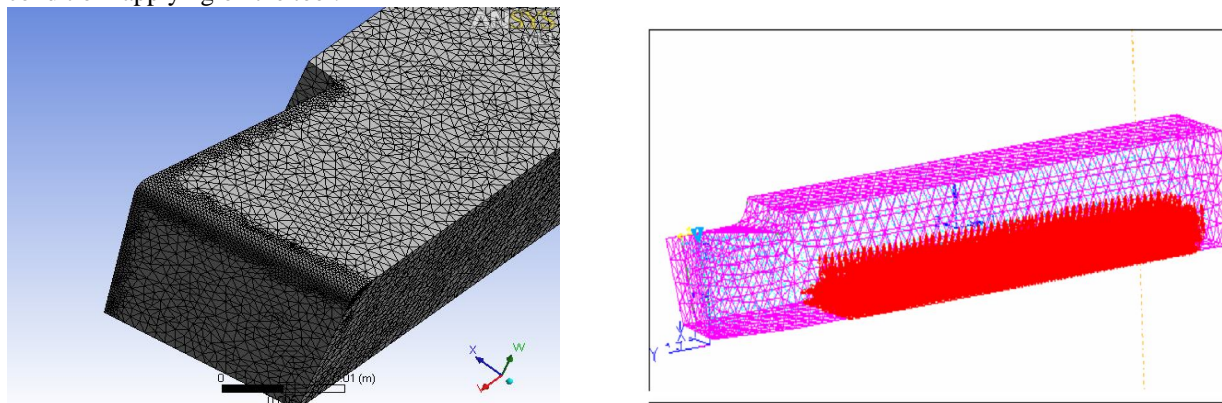


Figure 3.3 Shows the Finite Element Model with Meshing and Boundary Condition of the Tool

3.9 Model Solution

After boundary condition create the new solution set. Then solve the solution set with Specified boundary condition by using solves command. After 5-7 minute software solve the solution and detect the errors in the solution.

3.10 Post Processing

In post processing results is selected such as stress, displacement and use display command and select the model and show selected results in the form of graphical representation and finally use visualize command to animate the model.

V. RESULT ANALYSIS

4.1 Result Obtain From the Experimental Set Up

TABLE 4.1:-TEMPERATURES AT VARIOUS DEPTH OF CUT

Depth of Cut (mm)	Maximum Temperature of tool tip (°c)
0.2	119
0.5	165
1.0	200
2.0	448
2.5	548

GRAPH 4.1:-TEMPERATURE VS. DEPTH OF CUT

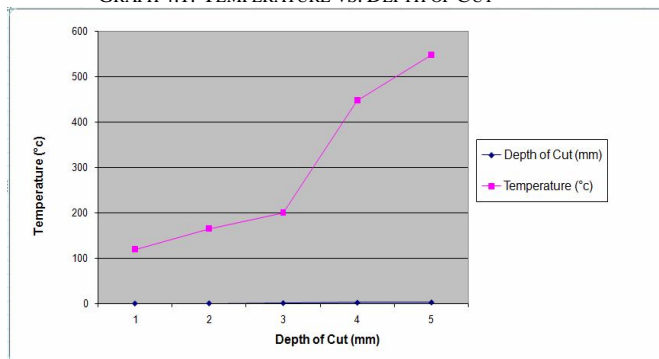
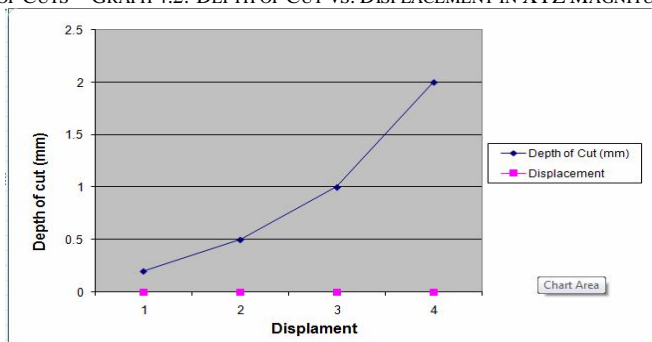


TABLE 4.2: - DISPLACEMENT IN XYZ MAGNITUDE AT VARIOUS DEPTH OF CUTS

Depth of Cut (mm)	Displacement in XYZ magnitude (mm)
0.2	4.3817x10 ⁻⁴
0.5	1.095x10 ⁻³
1.0	2.19x10 ⁻³
2.0	4.38x10 ⁻³

GRAPH 4.2:-DEPTH OF CUT VS. DISPLACEMENT IN XYZ MAGNITUDE



4.2 Result from Analysis

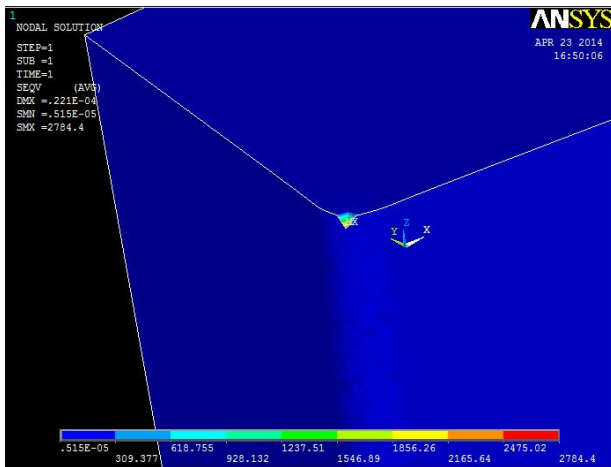


Fig 4-1 Von-Misses Stress at Depth of Cut 0.2mm

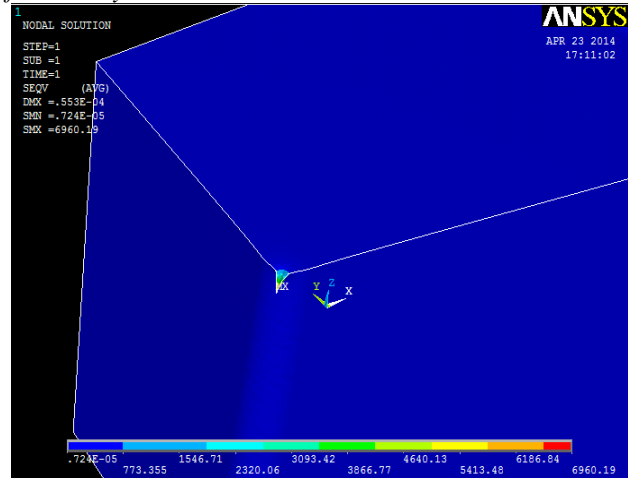


Fig 4-2 Von-Misses Stress at Depth of Cut 0.5mm

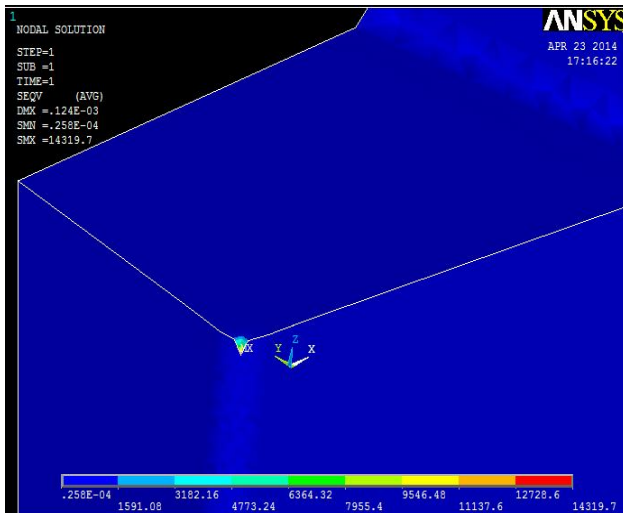


Fig 4-3 Von-Misses Stress at Depth of Cut 1mm

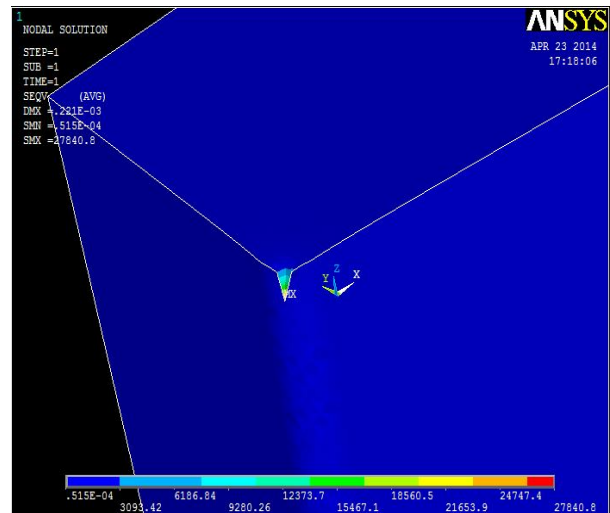


Fig 4-4 Von-Misses Stress at Depth of Cut 2mm

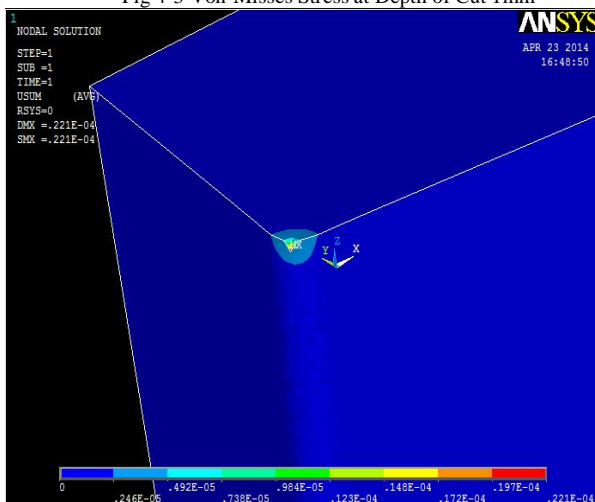


Fig 4-5 Displacement in XYZ Direction at Depth of Cut 0.2 mm

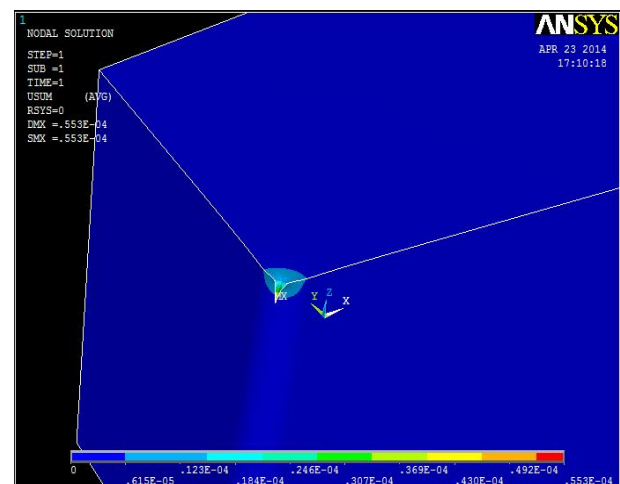


Fig 4-6 Displacement in XYZ Direction at Depth of Cut 0.5 mm

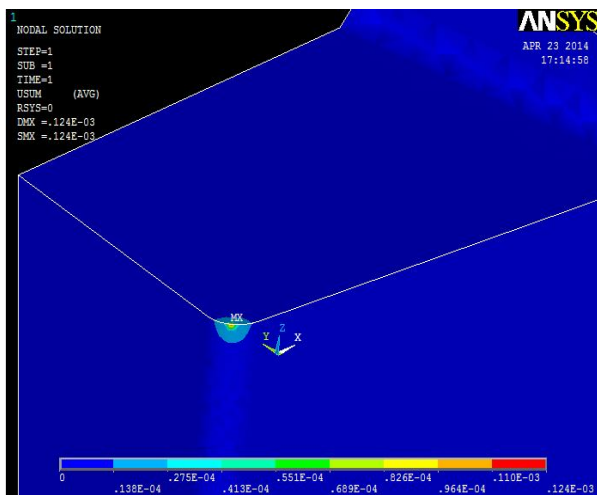


Fig 4-7 Displacement in XYZ Direction at Depth of Cut 1 mm

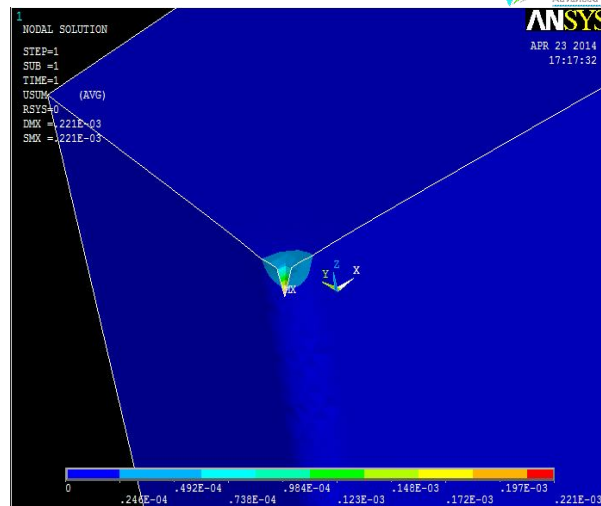


Fig 4-8 Displacement in XYZ Direction at Depth of Cut 2.0mm

v. CONCLUSION AND FUTURE WORK

From the experimental result and software result following conclusion is made

- 1) It is observed that as depth of cut increases, the von-mises stresses developed in the tool increases which are the main reason of tool failure.
- 2) From the experimental set up, it is clear observed that as depth of cut increases, the temperature generated in the tool at the tool tip also increases.
- 3) It is also observed that, as the depth of cut increases, deformation of the tool tip is also increases. It is main reason of tool failure.
- 4) It is also observed that tool start vibrating at the depth of cut 2.5 mm. At this condition more heat is decapitated at the tool, due to which tool blunt.
- 5) Tool forces can be measured during cutting using dynamometer and analyze the effect on the tool.
- 6) NASTRAN software can be used for analysis of single point cutting tool.
- 7) Analyze the residual stresses developed in the tool.

VI. REFERENCES

- [1] B.L.Juneja, G.S.Sekhon, Nitin Seth, "Fundamental of Metal Cutting and Machine Tools", IInd edition, New age international Publishers.
- [2] D Abdul Budan, "Influence of fibre proportion and position on the machinability of GFRP composite-An FEA model", Indian Journal of Engineering and material science, vol.12, october2005, pp.443-450.
- [3] Geoffrey Booth Royd, Winston. A Night, "Fundamental of Machine Tools." Dekkor and Hall Inc, New York, pp.110-120.
- [4] G.R.Nagpal, "Tool Design", Khanna publishers.
- [5] Handbook of Metalworker, vol.1-5, Mashgiz, 1960.
- [6] Strenkowski,J.S. and Carroll,J.T.,1985, "A finite element model of orthogonal metal cutting,"ASME Journal of Engineering for Industry,107,346-354.
- [7] Tugrul Ozel,Erol Zeren,2007, "Finite Element Analysis of the influence of edge roundness on the stress and temperature fields induced by high speed machining," International Journal Adv Manufacturing Technology,35:255-267.
- [8] Tugrul Ozel,Erol Zeren,2004, "Determination of work material flow stress and friction for FEA of machining using orthogonal cutting tests, "Journal of Material Processing Technology,153-154,1019-1025.
- [9] Mr. Luke H. Huang, Dr. Joseph C. Chen & Dr. Tao Chang, "EFFECT OF tool/Chip CONTACT LENGTH on Orthogonal Turning Performance" , Journal of Industrial Technology, volume15, Number 2-February 1999 to April 1999.
- [10]M.R.Madrigal, J.B.de Aguiar, G.F.Batalha, 2001, "Thermo mechanical model for Orthogonal Metal Cutting,"Swets@Zeitlinger,lisse ISBN 9026518226.
- [11] P.C.Sharma, "Production Engineering", S.Chand publisher, New Delhi, pp 149-155.
- [12] P.N.Rao, "Manufacturing technology-Metal Cutting and Manufacturing Tools", Tata McGraw-Hill Publishing Company Limited, New Delhi.
- [13] P.J.Arrazola, D.Ugarte,J.Montoya,A.Villar,S.Marya,2005, "Finite Element Modeling of chip formation process with ABAQUS/EXPLIIT 6.3",VIII International Conference on Computational plasticity,COMPLAS VIII.