ACTA TECHNICA CORVINIENSIS – Bulletin of Engineering
Tome IV (Year 2011). ISSN 2067-3809

[1.] Pavel KOVAČ, [2.] Borislav SAVKOVIĆ, [3.] Branislav SERDAR, [4] Milenko SEKULIĆ

MODELING MECHANICAL AND THERMAL LOAD OF GUTTING TOOL

ABSTRACT:
Finite element method, as a method of simulation of the cutting phenomenon during machining process allows obtaining information relevant for further computational analysis of tool wear, cutting temperature and cutting forces. These are the most important factors that influence the accuracy of processing and combined with other factors they affect deformation of cutting tools. The paper presents computer modelling of tool deformation and thermal load during turning using finite element method. The modelling was conducted during cutting time from 20 to 180 seconds.

KEYWORDS:
finite element method, mechanic, thermal load control, turning

INTRODUCTION

Machining is one of the most important and most common manufacturing processes in the metal processing industry. Cost of production in the metal processing industry is primarily achieved by optimal choice of all the factors that influence the cutting process.

The application of digital computers has brought revolutionary changes in the domain of various engineering and scientific disciplines, and one of the first was the area of mechanics of solids.

Increase processor enabled computer can significantly improve the relationship between price systems and their performance, resulting in an increased number of users in industry who Finite Element Method (FEM) systems integrated into the process of construction and development.

In recent years, finite element analysis has become a major method in simulation of metal cutting. Recently developed software systems based on the method of finite elements that are exclusively designed for cutting process simulation.

The process of cutting occurs when edge of cutting tools penetrate with cutting speed v in to processing materials. Entering edge of cutting tools, under the influence of external forces (cutting force F); there is a conversion of surplus material and thickness (depth of cut) in the chip thickness as [1, 2].

The appearance of the heat in the cutting zone is a consequence of converting mechanical energy into heat. Heat affects: the chip forming process, chip plastic deformation, chip compression ratio, cutting forces, the buildup edge, the intensity of the development of tool wear process of the structure and thickness of the machining surface defective layer.

More than 99.5% of energy (mechanical work) consumed in the deformation processing of material, and mastering of friction forces on contact surfaces cutting tools surfaces (rake and flank) is transformed into heat, so that the quantity of generated heat during cutting process [1]:

Generated heat in the cutting zone leads to a warming workpiece chip and cutting tool as well as the appearance of the characteristic temperature field and temperature.

FINITE ELEMENT METHOD

By definition, the finite element method is a method for approximately solving the so-called space problem. Finite element method is a method of computer analysis of problems in mechanics, fluid mechanics, thermodynamics, etc... In manufacturing, mostly in the metal forming process, the finite element method has proved to be an irreplaceable tool in research and development as well as in industrial applications. The basic principle of the finite element method is the division of the continuum (space) on a finite number of parts or elements. Thus the initial, very complex problem, we have a finite number of discrete and independent problems [3].

SOFTWARE PACKAGE

For temperature analysis during turning was used ANSYS software package. ANSYS is a general-purpose software package designed for analysis using finite elements. ANSYS Contains equations that governs the *behavior of finite elements, fixes them and gives a comprehensive explanation of the functioning of the system as a whole. These results can be presented in tabular or graphical form [4].*

The steps in solving problems of ANSYS software package:

- *Development of geometric models*
- *Defining the properties of ϖ materials*
- *Generation Network*
- *Defining loads*
- *Troubleshooting*
- *Find results presentation*

The Finite Element Method Analysis Of Force Effects During Turning

Computer analysis of the effects of cutting force in turning was realized using the ANSYS Workbench v11 software system and its module for structural analysis. The computer simulated analysis for the processing time of 1 and 2 seconds.

Building of geometry

In the first phase closed 2D contour is modeled using the Design Modeler and its module Sketcher. Using 3D modeling command Extrude, model tools and workpiece are given a third dimension. 3D model is then introduced into the ANSYS Workbench's module Simulation for further analysis where the meshing of the model is performed (Fig. 1.).

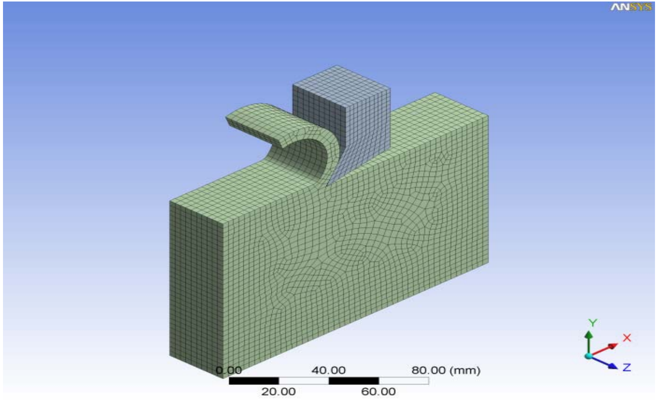

Fig. 1. Mesh of 3D model

Defining material properties

After the phase of generating discrete 3D model, mechanical and thermal properties of tool material [5,6] (Table 1.) are entered in Engineering Data window.

Table 1. Tools and workpiece material properties

	Module of elasticity [N/m^2]	Specific density [kg/m^3]	Poisoned coefficient	Heat conduction [W/m°C]	Specific heat [J/kg°C]
Tool material	$4,15\cdot 10^{11}$	14300	0,2	55	560
Workpiece material	$2,17\cdot 10^{11}$	7850	0,3	41,5	460,5

Defining the type of analysis and boundary conditions

Flexible Dynamic (dynamic load variable over time) analysis is selected. It allows analysis of deformation caused by the action of forces that occur during cutting. It is necessary to constrain the tool, to take away its degrees of movement freedom in the direction of X, Y and Z-axis (Fixed Support).

Defining loads

For defining of boundary conditions and input data of the cutting forces data from the literature is used [4]. The following experimental data for the force (Load) Fr = 2410 N (X = 880N, Y = -2200 N, Z = -440 N) is used. The value of cutting force was obtained on the universal lathe. During machining the following cutting conditions were used feed = 0.426 [mm/rev], cutting speed = 2 [m/s], rpm = 530 [rpm], depth of cut = 2 [mm] machining diameter = 72 [mm] tool material HM P25 and workpiece material steel Č.1730.

Solving the mathematical model

Since all the necessary elements for a specific analysis were defined, next step in solving the problem was using a command SOLVE.

After solving a problem ANSYS Workbench provides a graphical display of the results obtained in the form of images and/or animation.

Chosen output result parameters were:

- *Directional Deformation X*
- *Directional Deformation Y*
- *Directional Deformation Z*
- *Total Deformation*
- *Equivalent Elastic Strain*
- *Strain Energy*
- *Total Acceleration*
- *Total Velocity*
- *Vector Principal Elastic Strain*

Analysis And Out-Line Of Results

The analysis results are shown in Table 2. which contains data about tool deformation in individual axes and the overall deformation.

Table 2. Results of finite element analysis

	Minimum [mm/mm]	Maximum [mm/mm]
Equivalent Elastic Strain	7,4402e-016	8,879e-004
Directional Deformation X	-3,4122e-003	2,4002e-003
Directional Deformation Y	-8,3198e-003	8,8709e-005
Directional Deformation Z	-2,682e-003	5,9462e-005
Total Deformation	0	9,2854e-003

Figures 2 to 4 shows Deformation of cutting tool in 3 orthogonal direction and Figure 5 shows total deformation of cutting tool.

Analysis of the occurrence of tools deformation in cutting metal, simulating the effects of cutting forces in three-dimensional model of the tool by use of the finite element method, gave satisfactory results. The result of the total deformation, due to the effects of force, after one second is 0.0054 mm, and after two seconds 0.0092 millimeters.

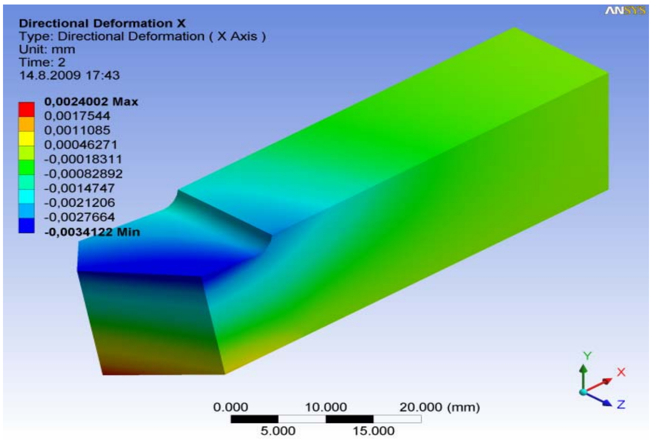

Fig. 2. Directional deformation X-axis

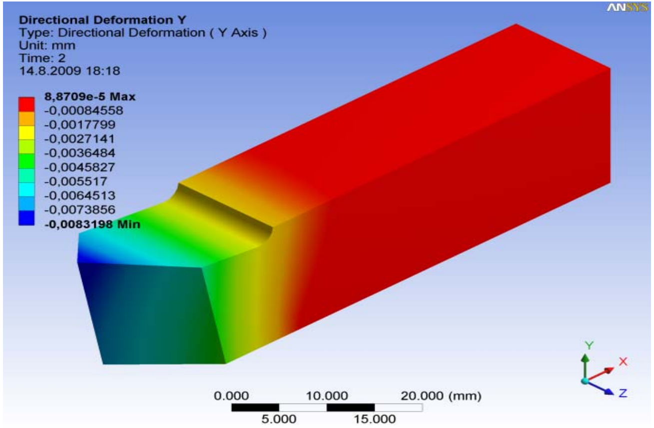

Fig. 3. Directional deformation Y-axis

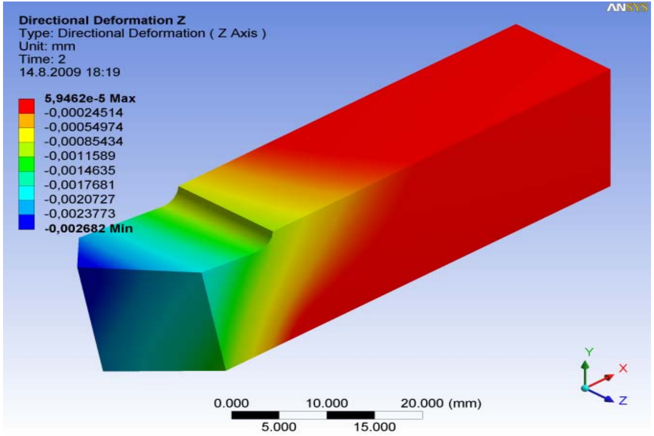

Fig. 4. Directional deformation Z-axis

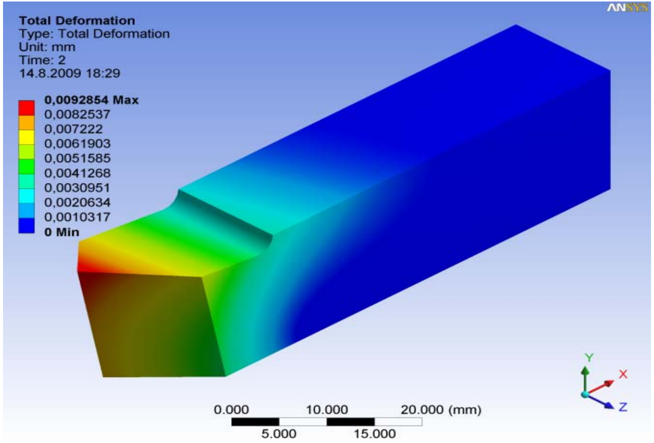

Fig. 5. Total Deformation

ANALYSIS OF TEMPERATURE IN TURNING HEAT BY FINITE ELEMENT METHOD

The amount of heat flux is approximated by a computer model. Heat source of variable intensity simulation in four phases was generated by tool movement. In the first phase of setting of heat load modeled was tool in the admission procedure, and cutting process is modeled after 20 [s] of machining. Based on the given feed movement, which was 0.428 [mm / rev] and speed v = 112 [m / min], after 20 [s] machined was 72 [mm] of workpiece length. At this phase the heat flux taken from the literature [5] and [6] based on the maximum temperature of operation, was about 0.0018 x 106 [J/m^2s] because relations for the determination of heat flux that goes into the chips were not available in the literature. At the end of the first phase, after 20 [s] calculate heat flux was 0.0029 x10^6 [J/m^2s].

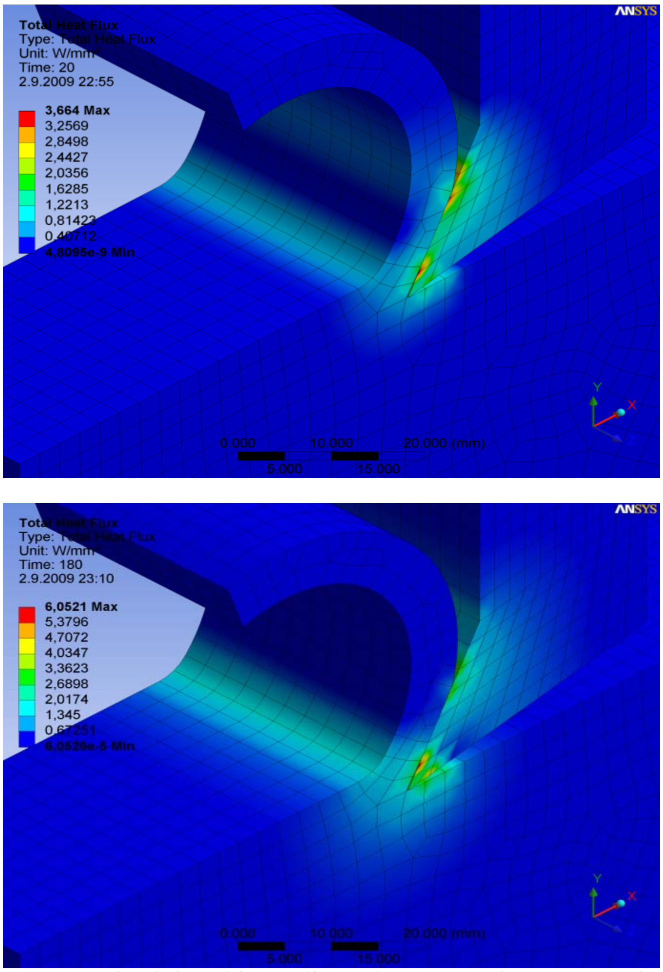

Fig.6. Schedule of heat flux after 20 and 180 seconds

The second stage continued a definition of loading on the first phase. After 80 [s] at the same speed and feed were processed the next 290.4 [mm] of workpiece.

In the third and fourth phase of processing is modeled after 140 [s] or 180 [s], provided that the always contain the next phase as input outputs from previous phases.

Heat flux (Fig. 6.) in the cutting zone gradually changes until equilibrium is established between the created and taken the heat.

From the point of increase in temperature, a computer model has given satisfactory results in some points almost coincide with the experimental (minimum deviation 0.61%) [7, 8]. Primary opportunity for improving the thermal model used is a modification of the heat flux process; its calculation is based on finite difference or finite volume. According to available information from the literature to analyze the process of turning (determining the cutting heat flux) is the most suitable method of finite differences.

Parameterization of the characteristic geometric size, such as dimensions of workpiece, contact length of chip and so on considerably would simplified the iterative modeling of process variation of some characteristic like size, thereby enabling the optimization of machining processes on the basis of selected parameters. In this way the computer model could be used not only to predict the maximum temperature and temperature field distribution, but could find application in one comprehensive system for planning and managing the process of cutting.

CONCLUSIONS

Finite element method as a method for modelling of the cutting phenomena allows obtaining of information relevant for further computational analysis, tool life assessment as one of the most important factors for testing the accuracy of processing in which, among other factors certainly affect deformation cutting tools and heat load.

In the paper shedule in time of heat flux was determined.

Defermation of tool under the load of experimentalz determined cutting forces was determined.

REFERENCES

[1.] *KOVAČ, P., MANKOVA, I., GOSTIMIROVIĆ, M., SEKULIĆ, M., SAVKOVIĆ; A revew of the experimental tehnoloques for the measurement of temperature generated in material removal processes; Journal of Production Engineering, 2010, Volume 13, pp. 1-6.*

[2.] *ARSECULARATNE, J., A. (2003), On prediction of tool life and tool deformation conditions in machining with restricted contact tools, International Journal of Machine Tools & Manufacture Vol. 43.*

[3.] *MESSNER, G.: Modeling metal cutting process under consideration of elastic material properites, Dissertation aus dem Werkzeugmaschinenlabor, Düsseldorf, 2007.*

[4.] *About ANSYS, Department of Mechanical Engineering at Carnegie Mellon. http://www.me.cmu.edu/academics/courses/NSF_Edu_Proj/ThFlEngr_ANSYS/about_ansys.htm.*

[5.] *KOVAČ, P., MILIKIĆ, D. (1998), Rezanje metala, Univerzitet u Novom Sadu, FTN, Novi Sad.*

[6.] *OZEL, T., ZEREN, E. (2004), Determination of work material flow stress and friction for FEA of machining using orthogonal cutting tests, Journal of Materials Processing Technology.*

[7.] *GOSTIMIROVIC, M., KOVAC, P.: The thermal state of the workpiece surface layer during productivity grinding, Manufacturing Technology ISSN: 1213-2489, Praha, 2008, Volume 8, pp 55-61*

[8.] *KOVAČ, P., GOSTIMIROVIĆ, M., SEKULIĆ, M., SAVKOVIĆ.: Primena savremenih metoda u istraživanjima procesa obrade rezanjem, Tehnika - Mašinstvo ISSN: 0461-2531, Beograd, 2010, Volume 65, No 3, pp 45-52*

AUTHORS & AFFILIATION

[1.] *Pavel KOVAČ,*
[2.] *Borislav SAVKOVIĆ,*
[3.] *Branislav SERDAR,*
[4] *Milenko SEKULIĆ*

[1-4.] *UNIVERSITY OF NOVI SAD, FACULTY OF TECHNICAL SCIENCE, DEPARTMENT FOR PRODUCTION ENGINEERING, 21000 NOVI SAD, TRG D OBRADOVICA 6, SERBIA*

ACTA TECHNICA CORVINIENSIS
– BULLETIN of ENGINEERING
ISSN: 2067-3809 [CD-Rom, online]
copyright © University Politehnica Timisoara,
Faculty of Engineering Hunedoara,
5, Revolutiei,
331128, Hunedoara,
ROMANIA
http://acta.fih.upt.ro