Experimentální Analýza Napětí 2001
Experimental Stress Analysis 2001
39th International Conference
June 4 - 6 , 2001 Tábor, Czech Republic

HEAT TRANSFER IN CUTTING INSERTS

Kabała Andrzej

Abstract: *In the paper an example of heat transfer numerical analysis is presented. The aim of the analysis is a research of temperature distribution in cutting tool during cutting process. The finite element system ABAQUS/Standard is used to numerical solution of the problem. The numerical results of calculations are the base material to study phenomenon and to verify it experimentally. The experimental verification is presented in another paper.*

Key words: heat transfer, cutting process, cutting inserts, finite element method, numerical calculations.

The aim of this study is a research of heat transfer phenomenon in a cutting insert during turning process and its influence on cutting quality. Tool's temperature increase leads to enlargement of tool dimensions. It influences on turning diameter changes as a function of cutting time, intensity of heat generation and cooling conditions. Knowledge of this phenomenon as investigation results can be used as material to design an automatic unit that allows to correct undesirable changes of the cutting edge position.

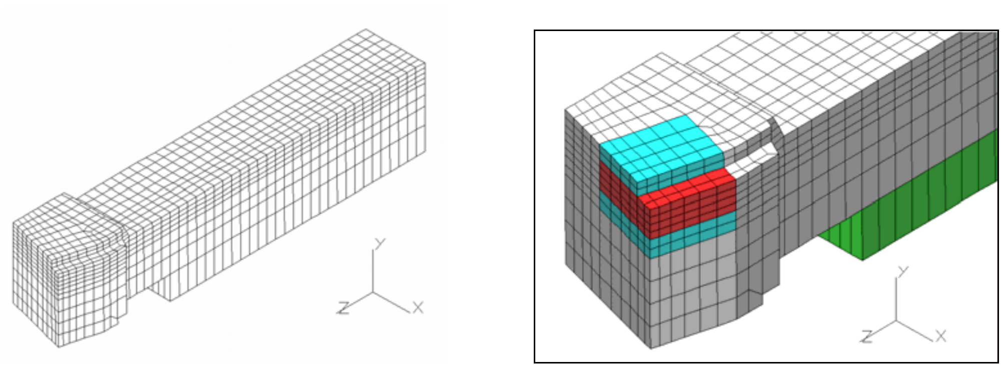

Fig. 1. FEM discrete model of turning tool

Finite element method system ABAQUS/Standard (*HEAT TRANSFER procedure) is used to solve the problem. Discrete FEM model is built on the base of choosen turning tool with MC2 ceramic insert. 8-node solid 3D (brick) elements are used to build the model. The configuration of the model is shown in Fig. 1. Initial temperature condition is 20^{o}C for all elements of the model. The numerical investigation step begins when turning process is started and heat flux generation appears suddenly. Temperature of 800 ^{o}C is an assumption as a thermal shock of cutting edge. Heat is conducted through the cutting insert towards the base

Kabala Andrzej Sc.Dr Eng. ,TUP Instytute of Mechanical Technology , ul.Piotrowo 3 ,61-138 Poznań, Poland
E-mail: kabala@rose.man.poznan.pl

and portion of the heat is dissipated by natural convection from the outside surfaces to the surrounding medium which is at a temperature of 20°C. Radiation is neglected according to low temperature of the majority surfaces. The considered step time is a 150-seconds period when the steady state conditions is approximately fulfiled.

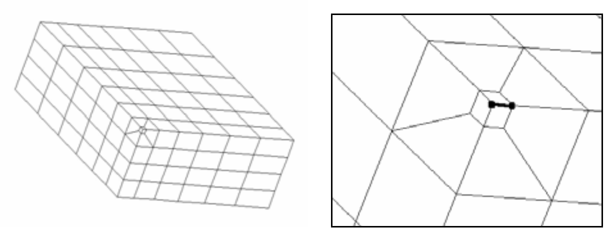

Fig. 2. Discrete model of the considered ceramic insert and its cutting edge

Description of the model

Phisical model consist of the following parts:

- MC2-ceramic cutting insert,
- locking inserts,
- shank of turning tool,
- tool holder.

The dimensions of the parts are assumed on the base of choosen tool. In the model small locking details and wholes are neglected.

<u>Basic dimensions of the model [mm]:</u>

Ceramic cutting insert	-	12.7 x 12.7 x 4.7
Shank of turning tool	-	126 x 24 x 24
Cutting tool stand out (distance tool holder-cutting edge)	-	50 mm

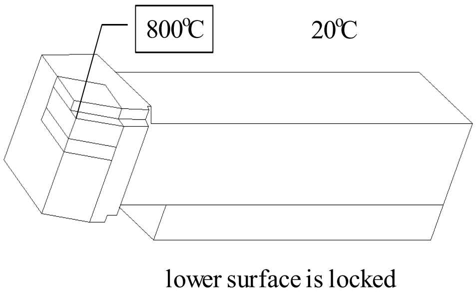

Fig. 3. Boundary conditions and temperature load of discrete model

<u>Properties of ceramic cutting insert MC2</u>

Density	-	4250	kg/m^3
Young's modulus	-	3.6 E11	N/m^2
Poisson's ratio	-	0.24	
Conductivity	-	38.0	W/(m °K)
Thermal coefficient of expansion	-	7.7 E-61/ °K	

<u>Properties of the other details</u>

Density	-	7850	kg/m^3
Young's modulus	-	2.09 E11	N/m^2
Poisson's ratio	-	0.3	
Conductivity	-	50.0	W/(m °K)
Thermal coefficient of expansion	-	12 E-6 1/ °K	

<u>Temperature loading</u>

- initial temperature of the model	-	20° C
- temperature of cutting edge	-	800°C
- temperature of surrounding medium	-	20° C
- temperature of base	-	20° C

The discrete model that is shown in Fig. 1. consists of: 4301 nodes of mesh and 3343 D3D8R solid elements. Refinement of the mesh is done nearby the cutting edge – see Fig. 2.

The assumption of cutting edge temperature value 800^{o}C is an approximate value because density of heat flux depends on many components and parameters. The most important of them are: cutting edge quality and its condition, cutting parameters, material properties, conditions of heat exchange between tool and surrounding medium. It means that only approximate value of computer simulation results is possible to obtain and experimental verification of numerical results ought to be done. The investgation has wide field of tolerance.

Results of numerical analysis

Considered transient heat transfer step is performed to simulate the 150-seconds cutting process of turning. The integration procedure used in ABAQUS/Standard FEM system for transient heat transfer uses limitation of minimum usable time increment. The needed value of minimum increment depends on elements size and material properties. In presented investigation value of 0.0002 second is choosen as minimum time increment in the begining of the time step.

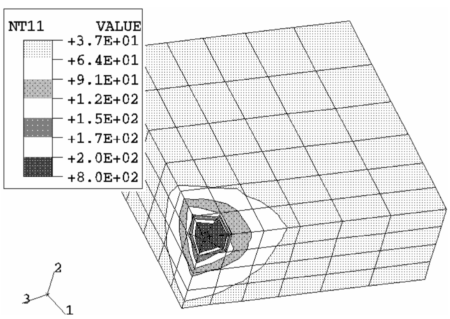

Fig. 4. Temperature distribution at cutting insert

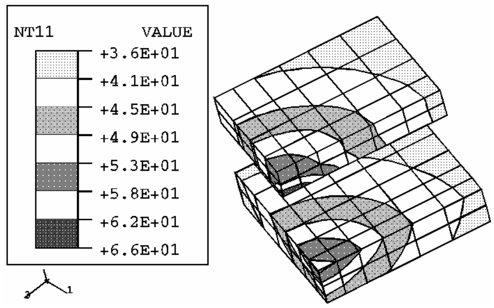

Fig. 5. Temperature distribution at locking inserts

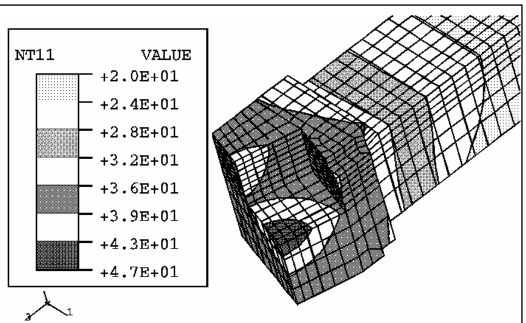

Fig. 6. Temperature distribution at shank of turning tool

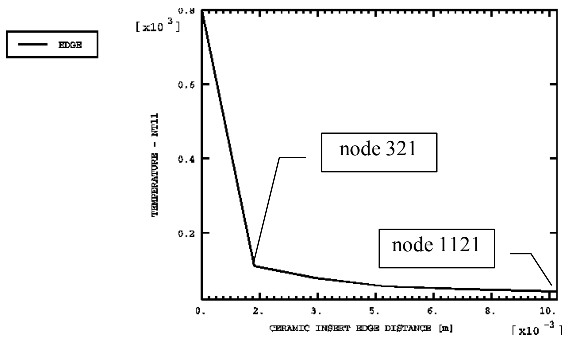

Fig. 7. Temperature distribution at edge of ceramic insert nearby cutting edge

Results of numerical calculation in 146 time increment (150-second summary time) is shown in Fig. 4. to Fig. 8. Temperature distribution at ceramic insert that is presented in Fig.4. shows high gradient of temperature nearby cutting edge. Temperature range at the insert is 800^{o}C to 37^{o}C. Temperature distribution at locking plates that is shown in Fig. 5. has range of 66^{o}C to 36^{o}C. Range of temperature distribution for shank of turning tool (Fig. 6.) is 47^{o}C to 20^{o}C. Temperature of 20^{o}C is established by boundary condition formulation.

Fig. 7. ilustrates temperature at ceramic edge nodes along cutting edge in final 146 time increment of analysied time step. Temperature time history plot of choosen insert edge nodes,

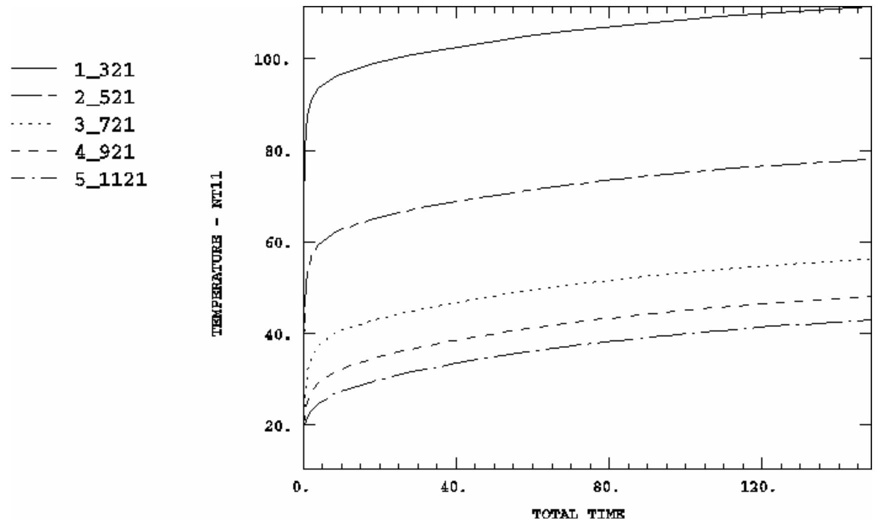

Fig. 8. Temperature history of the insert edge nodes

that ars situated along cutting edge line, is shown in Fig. 8. Time axis is denoted in seconds and temperature axis in ^{o}C.

Conclusions

- The highest temperature gradient is nearby the cutting edge at the ceramic insert.
- Influence of cutting edge quality and its condition, cutting parameters, material properties, conditions of heat exchange between tool and surrounding medium on heat flux generation leads to conclusion that only approximate value of computer simulation results is possible to obtain. An experimental verification of numerical rsults ought to be done.
- Comparison of temperature history at edge nodes of the cutting insert (Fig. 6.) to experimental observations gives good results.
- Time of 5 to 10 seconds is a period when approximate steady state conditions are done.
- FEM method allows to study the problem inside the model and to know the phenomenon better.

Bibliography

1. Kabala, A.,Morzynski, M., Thiele, F., Zielnica, J., „Static Analysis of the Container Under Pressure and Thermal Load”, Proceedings of the 1996 Engineering Systems Design and Analysis Conference, vol.9, pp.287-292, Montpellier, July 1996.
2. Algor, Inc.,1994, ALGOR Finite Element Analysis System „Processor Reference Manual”, Pitsburgh.
3. Hibbit, Karlsson & Sorensen, Inc., 2000, ABAQUS/Standard v.5.8, User’s Manual USA.
4. Zebala. W., „The Temperature Field Distribution in the Edge-Workpiece Contact Zone”, Proceedings of the Second International Symposium on Thermal Stresses and Related Topics – Thermal Stresses ‘97, pp.403-406, New York, June 1997.