ISSN: 1231-4005
e-ISSN: 2354-0133
DOI: 10.5604/12314005.1166002

THERMAL-MECHANICAL COUPLED SIMULATION

Hong Phong Nguyen, Agnieszka Derewońko, Tadeusz Niezgoda

Military University of Technology
Department of Mechanics and Applied Computer Science
Kaliskiego Street 2, 00-908 Warsaw, Poland
tel.: +48 261 837906, fax: +48 261 839355
e-mail: hong.phong.nguyen@wat.edu.pl

Abstract

Optimal design of engineering structures and technological processes requires taking into account various factors affecting the state of strain and stresses in the structure. Coupled thermo-mechanical analysis enables, among others, determination of undesirable changes in a body shape resulting from the implementation method of the initially-boundary conditions, for example, time-varying load and physical properties of the material depending on the temperature. They are also used to determine residual stresses remaining after manufacturing to prevent revalued stiffness and rigidity of the designed construction. In this study, coupled thermo-mechanical analysis illustrated by metal machining operation is presented. The commercial code MSC.Marc has been used to develop a coupled thermo-mechanical finite element model of plane-stress orthogonal metal cutting operations. Metal cutting is one of the most important and common manufacturing processes in the car industry. A thermal mechanical transient analysis is performed to convert mechanical work into heat by plastic deformation of the workpiece material and friction during metal machining operation. The finite element mesh distortion, due to large deformations, requires a remesh technique. The influence of parameters of the 2D and 3D finite element mesh adaptation on plastic deformation and temperature generated in the cutting processing is considered.

Keywords: *thermo-mechanical coupled, simulation, plastic deformation*

1. Introduction

The finite element method is one of the methods usually employed to design and simulate technological processes. However, taking into account all of aspects in the cutting process, it requires the use of the available algorithms such as discretization methods, constitutive models of materials or formulation of solutions, for example Lagrange, Eulerian or ALE [6-8].

In this paper, 2D and 3D finite element models of orthogonal metal cutting operation are presented with the use of commercial code MSC.Marc. The dynamic effects, thermo-mechanical coupling between deformation and temperature fields and the contact friction are considered. The yield stress is taken as function of the strain, the strain rate, and the temperature. In the analysis, the updated Lagrange formulation was used with the assumption of the formation of a chip due to plastic flow. The finite element mesh distortion requires, resulting from large deformations, the use of a strategy of a global remeshing procedure in which several adaptive parameters are considered. Finally, a series of sensitivity analyses was performed in order to evaluate an influence of the global edge length of the elements on the results of the analysis.

2. Finite element formulation

Coupled thermo-mechanical numerical analysis of the orthogonal metal cutting operation was carried out with the use of engineering software MSC.Marc in which the matrix equations for the thermal-mechanical problem are as follows [1]:

$$M\,\ddot{u} + D(T)\,\dot{u} + K\,(T,u,t)\,u = F \tag{1}$$

$$C^T(T)\,\dot{T}+K^T(T)T=Q+Q^I+Q^F\,, \tag{2}$$

where: M – system mass matrix, F – force vector, $u, \dot{u}, \ddot{u}$ – nodal displacement, velocity vector and acceleration vector, respectively, D – damping matrix, K – stiffness matrix, C^T, K^T – heat capacity and thermal conductivity matrix, all dependent on system nodal temperature vector T, Q – heat flux vector, Q^I – internal heat generated due to plastic deformation, Q^F – results from friction.

The heat generated due to plastic deformation is calculated by the formula:

$$Q^I=f\,W^p\,, \tag{3}$$

where: W^p – platic work, f – fraction of plastic work converted into heat, which is approximately equal to 0.9 for most metals.

The coupling between the heat transfer problem and the mechanical problem results from the temperature dependent mechanical properties and the internal heat generated. For the evaluation of temperature dependent matrices, the temperatures at two previous steps provide an extrapolated temperature description over the desired interval:

$$T(\tau)=T(t-\Delta t)+\frac{\tau}{\Delta t}[T(t-\Delta t)-T(t-2\Delta t)]. \tag{4}$$

The temperature $T(x)$ within an element is interpolated from the element nodal values T through the interpolation functions $N(x)$,

$$T(x)=N(x)T\,. \tag{5}$$

An important feature for modelling of the metal cutting operation is the constitutive material model. Defining this model requires consideration of such phenomena as strain hardening, strain softening and thermal softening caused by the strain rate and the temperature. In the finite element simulations, the Johnson-Cook constitutive model is commonly used [5-9], which determines the von Mises tensile flow stress σ_y as a function of strain rate and temperature:

$$\sigma_y=(A+B\,\bar{\varepsilon}_p^n)\left(1+C\ln\left(\frac{\dot{\bar{\varepsilon}}_p}{\dot{\bar{\varepsilon}}_0}\right)\right)\left(1-\left(\frac{T-T_{room}}{T_{melt}-T_{room}}\right)^m\right), \tag{6}$$

where: T – current temperature, T_{melt} – material melt temperature and T_{room} – ambient temperature, $\bar{\varepsilon}_p$ – effective plastic strain, $\dot{\bar{\varepsilon}}$ – effective plastic strain rate and $\dot{\bar{\varepsilon}}_0$ – reference strain rate, A, B, C, n, and m – constants. The expression in the first set of brackets gives a nonlinear hardening law for T_{room} and $\dot{\bar{\varepsilon}}_0$. The expression in the second and third sets of brackets represents the effects of strain rate and temperature, respectively.

In order to avoid penetration of the workpiece material into the tool material, the contact between the workpiece and the tool is applied. The tool is assumed as a rigid body. Once contact is detected, the degrees of freedom are transformed to a local system and the constraints are imposed. If the contact normal stress is in tension and its value exceeds 10% of the maximum compressive stress for all the contacting nodes, then the coincident nodes are separated and a chip is formed.

When the tool gets in contact, there is heat flux across the interface and to environment on the boundary surfaces of the workpiece. In the case of the distance between the contacting bodies equal to 0, the heat flux is written as:

$$q=H_{TC}(T_2-T_1)\,, \tag{7}$$

where: T_1 – surface temperature of the tool, H_{TC} – film coefficient between the two surfaces, T_2 – temperature obtained from interpolation of nodal temperatures of the workpiece.

The stick – slip Coulomb friction model is assumed, in which the contacting bodies are sticked together if:

$$\|\sigma_t\| < \mu \sigma_n , \tag{8}$$

and in relative motion if:

$$\sigma_t = \mu\sigma_n .\tau , \tag{9}$$

where: σ_t, μ, σ_n, τ – friction stress, friction coefficient, normal stress and tangential vector, respectively.

During chip separation and formation, ductile failure or fracture occurs, which requires taking into account a criterion in cutting simulation. In recent years, formation of a continuous chip is assumed to be the result of plastic flow and the workpiece is frequently remeshed to simulate chip separation without failure criterion [3-5, 8, 9]. The mechanism of chip separation by the remeshing procedure is presented in Fig. 1.

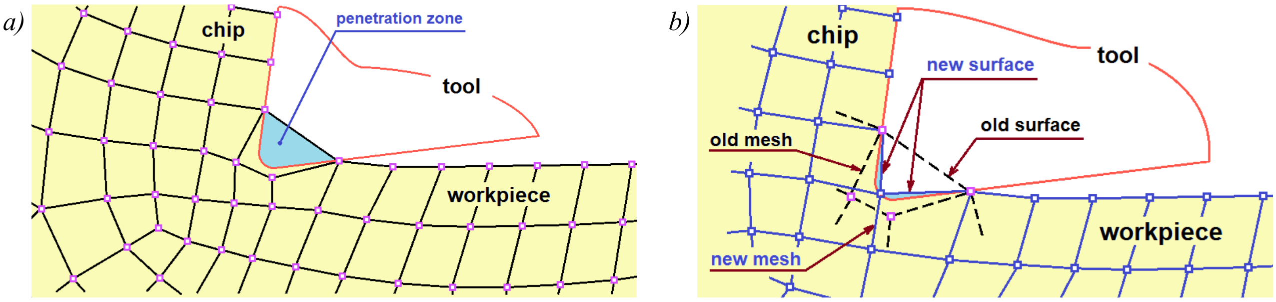

Fig. 1. Mechanism of chip separation by the remeshing procedure [4]: penetration (a) and remeshing (b)

The remeshing criteria are used to initiate the remeshing procedure. When one of them is met, the analysis automatically starts the remeshing procedure by generating a new mesh, transferring history data from the previous mesh and resuming the analysis. The new mesh on the deformed workpiece material is generated based on of the Delaunay Triangulation method. The analysis is also improved by mesh refinement in the cutting zone, where small elements are required due to a contact and geometry change.

3. Numerical modelling

The finite element model is composed of a deformable workpiece and a rigid tool. The model of orthogonal metal cutting operation is presented in Fig. 2 with the cutting parameters given in Tab. 1. These are real values corresponding to the physical process. The length of workpiece is 12 mm, the height is 5 mm, and the thickness is 1 mm. The rigid cutting tool has a rake angle α equal to 8°, the flank angle is 7°, and the radius of the cutting edge is equal to 0.02 mm.

Tab. 1. Parameters for orthogonal metal cutting operation

Cutting speed, v_c [mm/s]	847.8
Deep of cut, f [mm]	0.2
Width of cut, a_p [mm]	1.0

For the 2D model, the workpiece was discretized by the Marc elements 201. This is a three-node isoparametric triangular element written for plane stress applications. This element uses bilinear interpolation functions. The stresses are constant throughout the element. The stiffness of this element is formed using one point integration at its centroid.

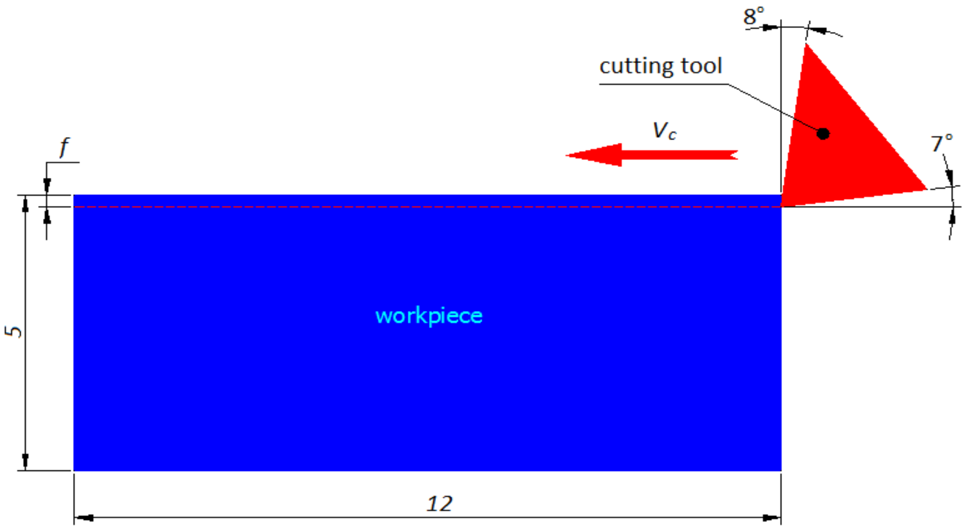

Fig. 2. Model of orthogonal metal cutting operation

For the 3D model, the workpiece was discretized by the Marc elements 134. This element is a linear isoparametric three-dimensional tetrahedron. As this element uses linear interpolation functions, the strains are constant throughout the element. The element is integrated numerically using one point at the centroid of this element.

In both models, the left and bottom boundary nodes of the workpiece were fixed in all the directions (Fig. 3).

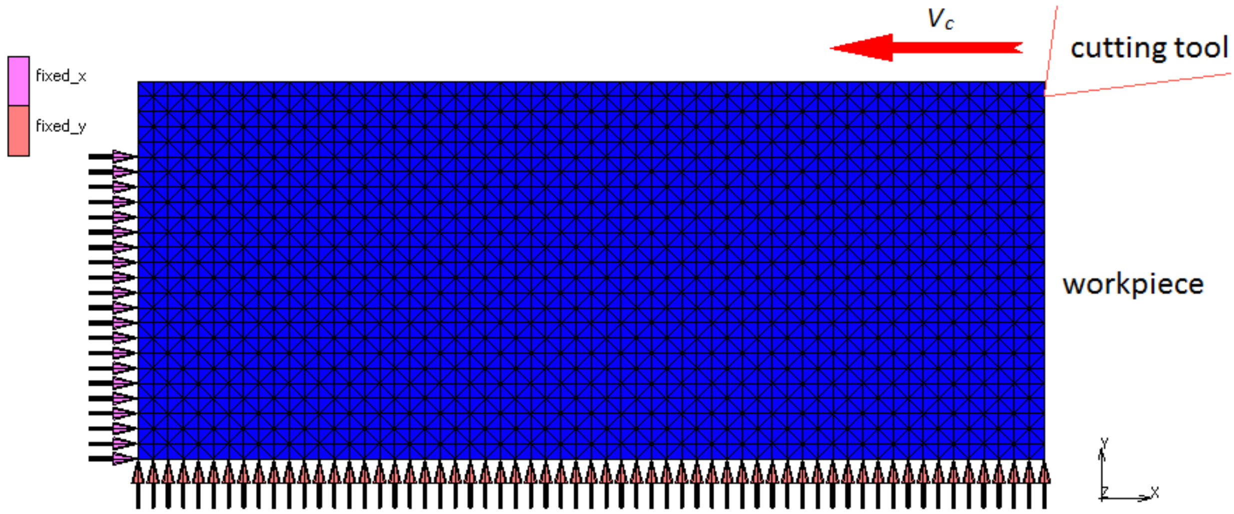

Fig. 3. Finite element model of cutting

The values of coefficients *A*, *B*, *C*, *m*, *n* of the constitutive material model Johnson-Cook for steel C15 are assumed according to the literature [2] and presented in Tab. 2.

Tab. 2. Mechanical properties of steel C15

A [MPa]	*B* [MPa]	*n*	*C*	*m*	T_{ot} [°C]	T_{op} [°C]
350	275	0.36	0.022	1.0	20	1538

The temperature dependent material properties such as Young's modulus, thermal coefficient of expansion, thermal conductivity and specific heat are taken from the material database of the MSC.Marc software. Fig. 4 presents these dependences in a graph form for material C15.

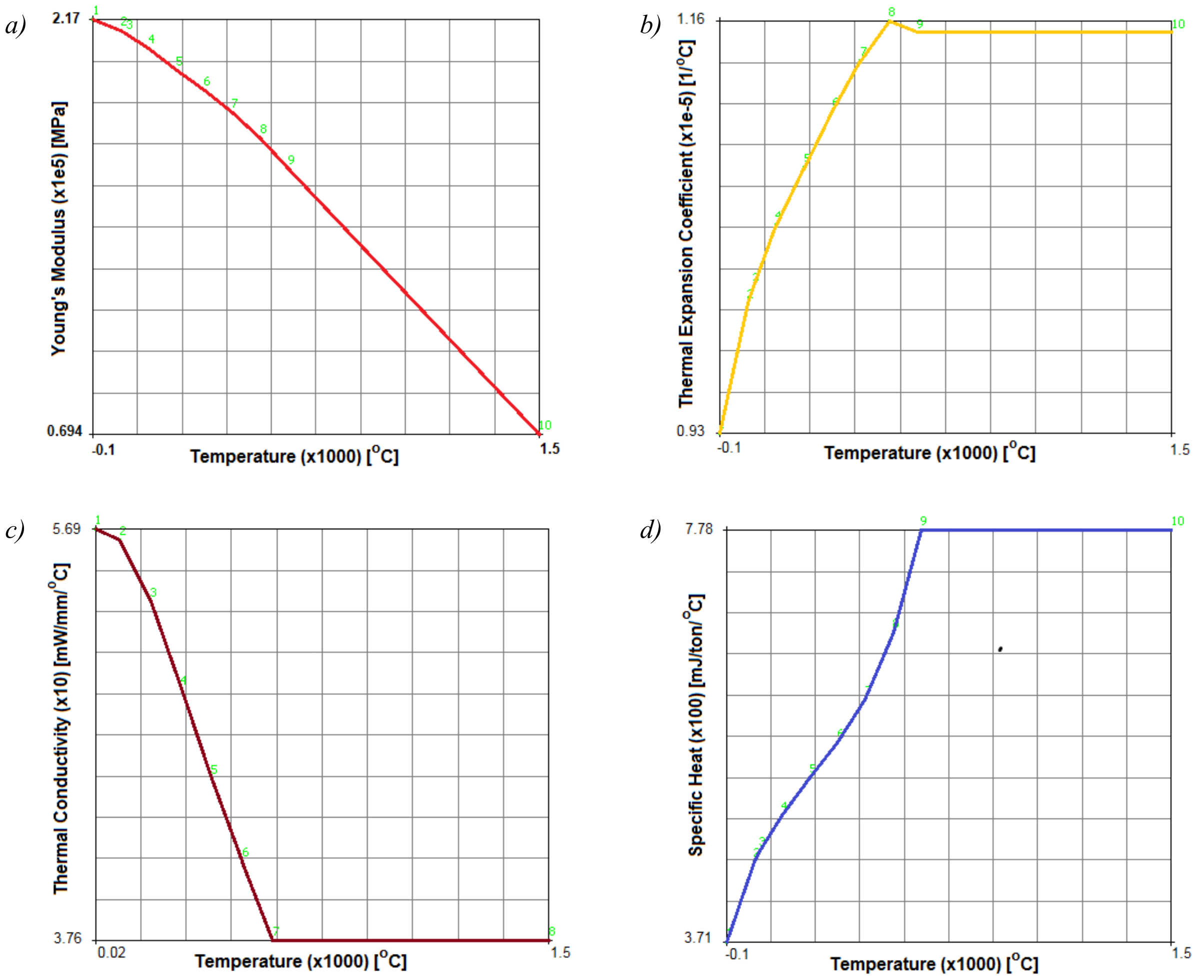

Fig. 4. Temperature dependent material properties: Young's modulus (a), thermal coefficient of expansion (b), thermal conductivity (c), specific heat (d)

The friction coefficient $\mu = 0.3$ was regarded as an average value. The fraction of friction work converted into heat is equal to 0.9. The rigid tool and the environment are considered to have a constant temperature of 20°C. The contact heat transfer coefficient between the workpiece and the tool is 40 mW/mm/°C, whereas the heat transfer coefficient of the workpiece to the environment is equal to 0.04 mW/mm/°C.

The remeshing procedure is carried out when one of the following criteria is satisfied:

– the equivalent strain measure is greater than 0.4,
– the penetration of the workpiece into the tool exceeds two times as much as the contact tolerance, which is 0.05 of the smallest element length;
– for 2D model, the element distortion based on the inner angle change is greater than 40°, whereas for the 3D model this check based on the volume ratio is smaller 0.1.

The control parameters of the global remeshing were assumed with:

– the coarsening factor of 1.2 is used,
– the feature edge angle is 60°,
– the feature vertex angle equals to 100.

In the 2D model, 3 levels of the mesh refinement in the cutting zone are considered with the curvature control equalling to 100 and the smallest edge length of the elements as 0.04 mm. However, in the 3D model, these parameters are 2, 60 and 0.07 mm, respectively.

In order to evaluate the influence of the global edge length of the elements on the results of the analysis, a series of sensitivity analyses were performed with the values in order of 0.32, 0.4, 0.5, 0.64 and 0.74 mm.

4. Results

The temperature, total equivalent plastic strain of the 2D and 3D model for edge length equal to 0.4 mm are presented in Fig. 5-8.

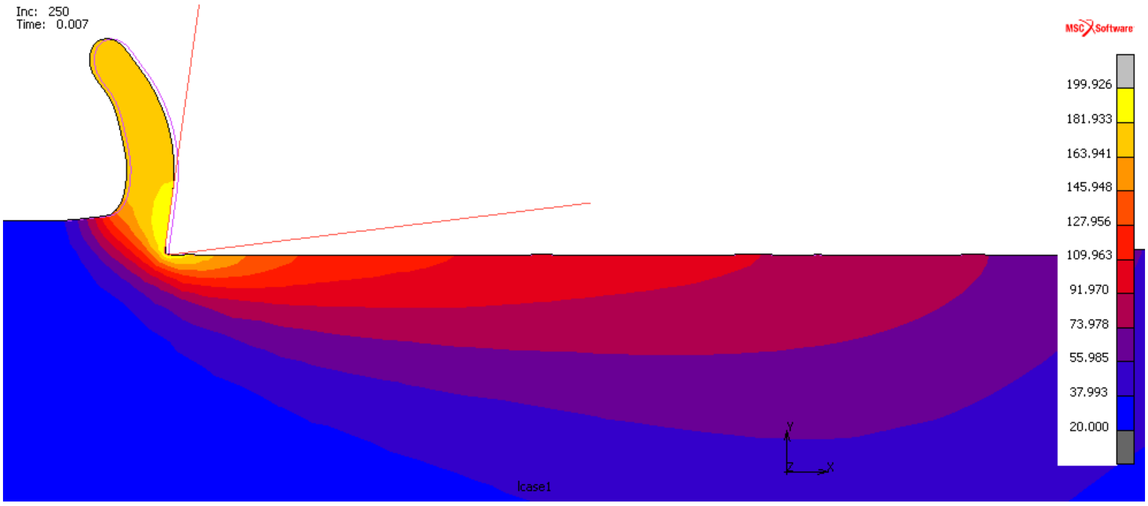

Fig. 5. Temperature field in the cutting zone – 2D model

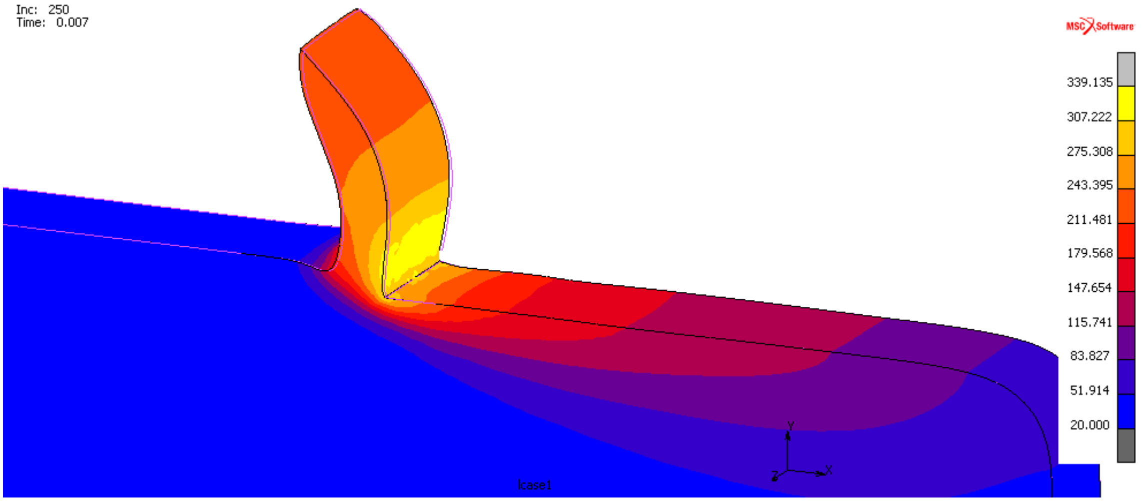

Fig. 6. Temperature field in the cutting zone – 3D model

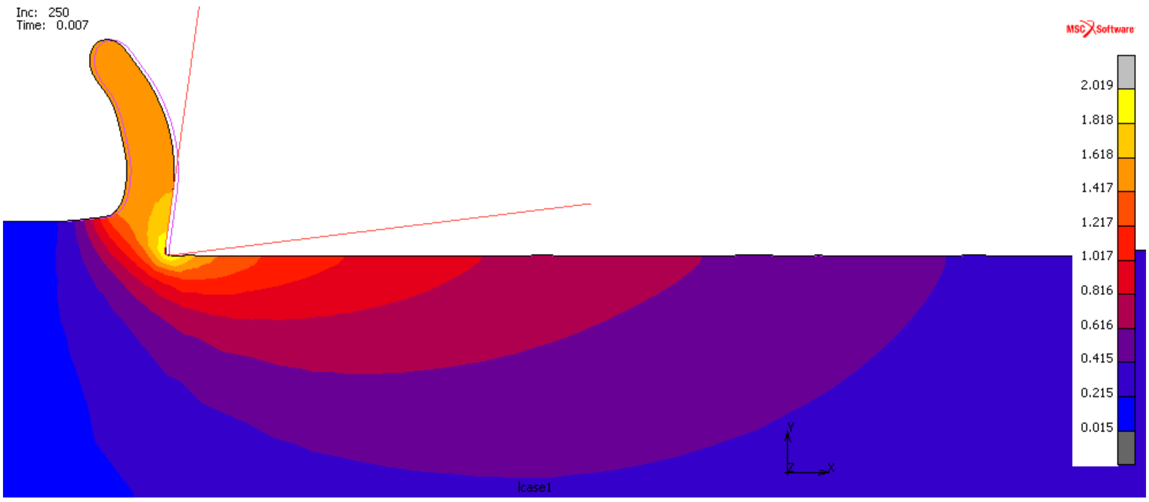

Fig. 7. Total equivalent plastic strain field in the cutting zone – 2D model

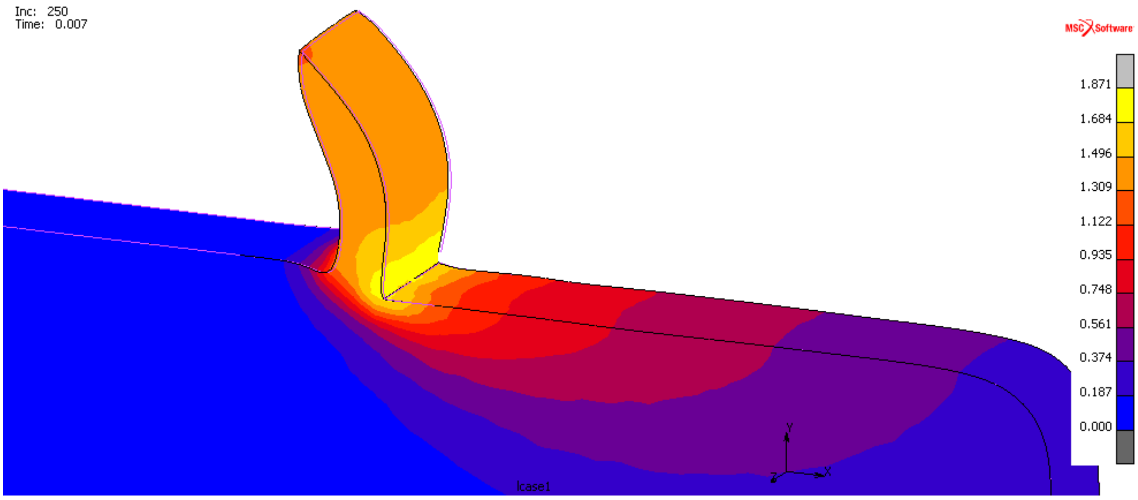

Fig. 8. Total equivalent plastic strain field in the cutting zone – 3D model

Figure 9 and 10 presents the temperature – time curves which were generated to compare the computed results for different adaptation edge length of the finite elements.

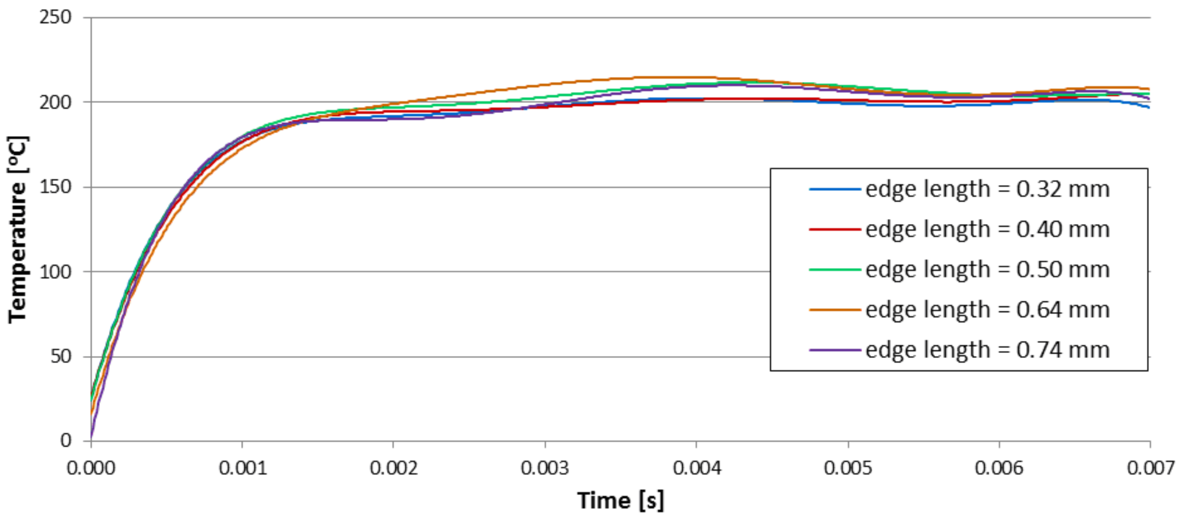

Fig. 9. Comparison of maximum temperature in the 2D model

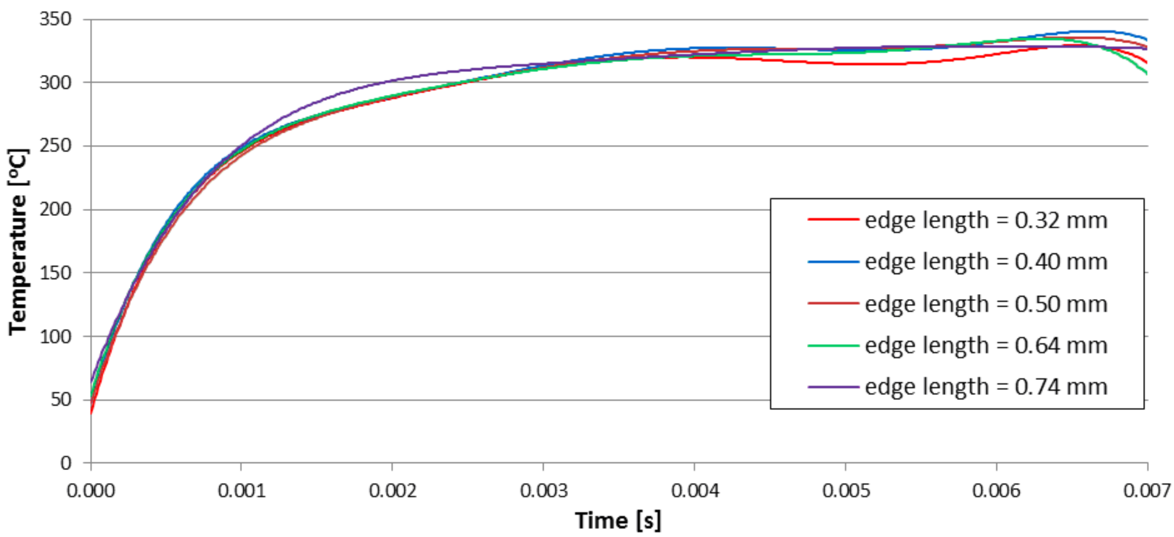

Fig. 10. Comparison of maximum temperature in the 3D model

5. Conclusions

In the cutting process, the heat is generated from the plastic and friction work, which causes the temperature field in the cutting zone. In order to simulate this field, a coupled thermo-mechanical analysis of the orthogonal cutting operation is developed. The simulation is performed with the use of a 2D plane strain model and a 3D model. A number of numerical problems are solved, particularly for the need for reliable remeshing and contact algorithms.

In order to avoid the premature termination of the analysis, the 2D model requires smaller edge length of the elements in the cutting zone than the 3D model with respect to the depth of cut f of 0.2 mm. This is defined by the level of refinement in the cutting zone and the minimum edge length of the elements.

The maximum temperature in the cutting zone in the 2D model is around 200°C, whereas in the 3D model it is around 325°C. This is caused by plane stress assumption in the 2D model.

There is an influence of the global element size on the results of the analyses, in which there are pick values if the edge length is greater than 0.5 mm in the 2D model and smaller than 0.5 mm in the 3D model. This may cause inaccuracy in the results of the analyses.

References

[1] *Marc® 2011, Volume A: Theory and User Information*, Copyright MSC.Software Corp., 2011.

[2] Johnson, G. R., Cook,W. H., *A constitutive model and data for metals subjected to large strains, high strain rates and high temperatures*, Proceedings of the Seventh Symposium on Ballistics, pp. 1-7, The Hague, The Netherlands 1983.

[3] Fang, G., Zeng, P., *Three-dimensional thermo-elastic-plastic coupled FEM simulations for metal oblique cutting processes*, Journal of Materials Processing Technology, Vol. 168, pp. 42-48, 2005.

[4] Bil, H., Kilic, S. E., Tekkaya, A. E., *A comparison of orthogonal cutting data from experiments with three different finite element models*, International Journal of Machine Tool & Manufacture, Vol. 44, pp. 933-944, 2004.

[5] Svoboda, A., Wedberg, D., Lindgren, L. E., *Simulation of metal cutting using a physically based plasticity model*, Modelling and Simulation in Materials Sciense and Engineering, Vol. 18, 2010.

[6] Zębala, W., *Modelowanie procesu skrawania*, Politechnika Krakowska, Krakow 2011.

[7] Niesłony, P., Grzesik, W., *Modelowanie procesu i operacji skrawania metodą elementów skończonych, cz. I i II*, MES, Mechanik, 10/2013 and 11/2013.

[8] Arrazola, P. J., Ozel, T., Umbrello, D., Davies, M., Jawahir, I. S., *Recent advances in modelling of metal machining proces*, CIRP – Manufacturing Technology, 62, pp. 695-718, 2013.

[9] Diaz-Alvarez, J., Cantero, J. L., Miguelez, H., Soldani, X., *Numerical analysis of thermo-mechanical phenomena influencing tool wear in finishing turning of Inconel 718*, International Journal of Mechanical Sciences, Vol. 82, pp. 161-169, 2014.