THE VEINING STRUCTURE METHOD, THE FINITE ELEMENT METHOD IN THERMAL DEFORMATION DETERMINATION FOR THE MAIN SPINDLE AT NUMERICAL CONTROL LATHES

by

Adrian BUT and Ioan LUMINOSU

Original scientific paper
UDC: 620.251.2:517.987
BIBLID: 0354-9836, *9* (2005), 1, 99-110

In the determination of the process accuracy on computer numerical control (CNC) lathe [1] it is very important to know the thermal deformations. This article presents a new solution to obtain the thermal deformations at the principal arbor (the most important part of a principal actuation for a CNC lathe) by using the element finite method. The first part of the theoretical aspect presents the steps of this method used especially for the principal arbor. The experimental part presents how this method was used to measure and process the signals. The experimental tests were applied in two different situations: (a) when material was catch only in principal arbor, and (b) when material was catch in principal arbor and chuck. The NASTRAN MSC program was used, which has the possibility to determine the temperature values in the whole mass of the principal arbor and later the deformation temperature of every point of the ensemble was obtained.
In order to analyze the thermal strain influence on the main spindle, the finite element method, the veining structure of a body was used. This is a method through which the "mother" (complex) structure was split in composing parts considered as independent (substructures). The link between different substructures in which the complex structure was split, is made on the common contour of substructures. The substructure becomes this way an independent structure with edge condition, constraint by assembly continuity of initial structure.

Key words: *finite element, thermal deformation, main spindle*

Introduction

The veining structure's problem presented in this article, using the finite element method, specifies that if eliminate the movements as a rigid body, the substructure's behavior is unique determined by the outline's movements, representing the border between substructures, where we can find the so-called common node [2]. Using the displacement method, the substructures behavior is described by rigidity matrix reduced to the border's nodes and as result the substructure can be considered as a complex element.

Methodology

The rigidity matrix assembly is made two-stage: (1) the rigidity matrices of substructure elements are assembled, and reduced to common nodes, and (2) after that the structure assembly in structure rigidity matrix is made [3].

Excepting the substructure's border nodes, all other nodes can be deactivated because for those can be written final equilibrium equations, thus the nodes on the substructure edge are becoming information carriers towards the rest of the structure, and the respective unknowns remains active. Those remaining active nodes are called ”master”.

In conclusion resolving complex structures by splitting them in substructures is made two-stages: (1) the deformation state for every substructure, and (2) the connection of substructures so that in the intersection zones the equilibrium and continuity of displacement conditions to be observed.

In the veining structure the nodes on the common edges of neighbor substructures are defined as contact nodes *c* and the others are called interior nodes *i*. After the structure splitting in substructures, the contact nodes will be blocked using limiting devices in order to prevent the displacements we want to analyze. In this way the substructure can be resolved by drawing out the equation system for the *j* order substructure [4]:

$$[M^{i}_{i,j}]\{\rho^{j}_{i}\} \quad \{V^{j}_{i}\} \tag{1}$$

where

$[M^{i}_{i,j}]$ – structure rigidity matrix (with blocked interior nodes),
$\{\rho^{j}_{i}\}$ – column vector for the displacements of interior nodes, and
$\{V^{j}_{i}\}$ – column vector for external loads applied in interior nodes.

By resolving this system results that the reactions vector in blocked nodes generated by reaction vectors from all substructures *j* interfaces with other substructures.

The equilibrium condition of interfaces $j > k$ will be:

$$\{R^{j,k}_{e}\} \quad \{R^{k,j}_{e}\} \quad \{V^{j,k}_{\Lambda}\} \quad 0 \tag{2}$$

where

$\{V^{j,k}_{\Lambda}\}$ – column vector for loads on interface $j > k$.

After the substructures connection and contact nodes unblocking the displacement continuity condition are written:

$$\{\rho_{c}\} \quad [e][T] \tag{3}$$

where

$\{\rho_{c}\}$ – column vector of all contact nodes displacements for all substructure interfaces, written in the local reference system (X, Y, Z),

$[e]$ – is obtained from the same nodes displacements written in the global reference system (X, Y, Z), and

$[T]$ – the transformation matrix for rotary axis displacements.

This way, in the continuity equation is unknown only the contact nodes displacement, and makes necessary the rewriting of substructures rigidity matrix by eliminating interior nodes displacements.

The method consists in splitting the matrix by the degree of freedom of interior contact nodes:

$$[M^j] \quad \begin{matrix} [M^j_{i,i}][k^j_{i,c}] \\ [k^j_{c,i}][k^j_{c,c}] \end{matrix} \tag{4}$$

and

$$[\rho^j] \quad \begin{matrix} \{\rho^j_i\} \\ \{\rho^j_c\} \end{matrix} \tag{5}$$

After unblocking, in the contact nodes are appearing final reactions and external loads applied directly in these nodes $\{F^j_\Lambda\}$; the equation system can be written as:

$$\begin{matrix} [M^i_{i,j}]\{\rho^j_i\} \quad [M^i_{i,c}]\{\rho^j_c\} \quad \{F^j_i\} \\ [M^i_{c,j}]\{\rho^j_i\} \quad [M^i_{c,c}]\{\rho^j_c\} \quad \{F^j_c\} \quad \{R^j\} \end{matrix} \tag{6}$$

From here results the displacement matrix of interior nodes:

$$\{\rho^j_i\} \quad [M^i_{i,j}]^{\ 1}\{F^j_i\} \quad [H^i_{i,j}]\{\rho^j_c\} \tag{7}$$

where

$\{\rho^j_i\}$ – the column vector of interior nodes displacements when the contact nodes are blocked:

$$[H^j_{i,j}] \quad [M^j_{i,i}]^{\ 1}[M^j_{i,c}] \tag{8}$$

It results:

$$[\rho^j_i] \quad \{\rho^j_i\} \quad [H^j_{i,c}]\{\rho^j_c\} \tag{9}$$

Obtaining:

$$([M^j_{c,c}] \quad [M^j_{c,i}][H^j_{i,c}])\{\rho^j_c\} \quad \{F^j_c\} \quad \{R^j\} \quad [M^j_{c,i}][M^j_{i,i}]^{\ 1}\{F^j_i\} \tag{10}$$

For the development of this process we have to use the coordinates transformation of rotary axis.

The algorithm of this method is presented below.

(1) – division of the mother structure in substructures,
(2) – defining the structure topology, defining structure and substructure nodes,
(3) – defining interfaces and marking out the contact nodes, and
(4) – calculating transfer matrix from the local axis system to global axis system $[\rho_i^j]$ and $[k_c^j]$,
where $[\rho_i^j]$ – selects from the displacements vectors the one who matches with Boolean type j substructures nodes which reflects the topological proprieties of the structure that is the rotation mode of nodes in the structures, and substructures.
Noting with $[\gamma]$ the directors cosine matrix of axes according to the global axis X, Y, Z:

$$[\gamma] \quad \begin{matrix} \gamma_{XX} \gamma_{XY} \gamma_{XZ} \\ \gamma_{YX} \gamma_{YY} \gamma_{YZ} \\ \gamma_{ZX} \gamma_{ZY} \gamma_{ZZ} \end{matrix} \tag{11}$$

Or, for nodes with tree degrees of freedom with planar axis rotation:

$$[\gamma] \quad \begin{matrix} \cos\theta & \sin\theta & 0 \\ \sin\theta & \cos\theta & 0 \\ 0 & 0 & 1 \end{matrix} \tag{12}$$

where θ is the rotation angle in the XY plane.
The transforming matrix for a node is:

$$[S] \quad \begin{matrix} [\gamma][0] \\ [0][\gamma] \end{matrix} \tag{13}$$

From which results the structure transfer matrix referring to a number of n contact nodes:

$$T_c \quad \begin{matrix} [S]^T & 0 & 0... & 0 \\ 0 & [S]^T & 0... & 0 \\ 0 & 0 & [S]^T ... & 0 \\ 0 & 0 & 0... & [S]^T \end{matrix} \quad \text{type } (6n \quad 6n) \tag{14}$$

(5) – calculus for every substructure, referring to local reference system of rigidity matrix:

$$[M_{i,i}]; \quad [M_{i,c}]; \quad [M_{c,c}],$$

(6) – calculus for every substructure, referring to global reference system of rigidity matrix,

(7) – defining every substructure of reactions in blocked contact nodes, at every interface, $j \quad k, [R^{j,k}]$ and additional loads $\{F_A\}$,

(8) – calculus of load matrix according to global reference system:

$$\{F_A^j\}_{\mathrm{XYZ}} \quad [T_c]\{F_A^j\}_{\mathrm{XYZ}} \tag{15}$$

(9) – calculus of condensed matrix,

(10) – assembly of structure rigidity matrix and equivalent load vector [5]:

$$[M_{cc}] \quad {}_{1}^{n} [W_c^j][M_{cc}^j][W_c^j]^T \tag{16}$$

and

$$F_c \quad {}_{1}^{n} [W_c^j][M_{ci}^j] \quad \{F_{c,A}\} \tag{17}$$

(11) – solving the equation system and determine nodes displacement in every substructure.

Thermal strain test on the main spindle

Regarding the classical procedure which lies in displacement method (presented above) we will present a methodology based on the following considerations:

$$[\rho_0^j] \quad [e_i^j][e] \tag{18}$$

and

$$[M_{cc}]\{e\} \quad [F_c] \tag{19}$$

Supposing it can be made a digitization of the whole structure limited by the maximum memory of a computer, from which we can obtain a displacement field in the substructures splitting zone, we obtain a prototype on which we can make test measurements. Measurements will be focused on the temperature measurements in the desired zones; results with which we will correct the values obtained using the finite element method.

The main spindle can be considered as a special case of space structure formed by plates so that every plate can be considered as a finite element. In order to obtain displacements results were used a calculus program assisted by finite elements NASTRAN.

Such a program contains the following sequences:

– data preprocessing:

bi-dimensional or three-dimensional design drawing,
simulation (digitization, material choosing, border conditions, calculus analyze),
drawing import/export [6] – model finite element, and
special calculus as: signals processing, fatigue calculus, dynamic and contact analysis, temperatures, *etc*.;

– data processing:

effective interactive calculus or directly by post processing data,
the evaluation of analysis results,
the results processing (graphical, tabular), and
displaying results or printing results.

Next will be presented the steps followed in using NASTRAN program in order to obtain the calculus model with the logical scheme for solving the problem as specified in fig. 1 [2].

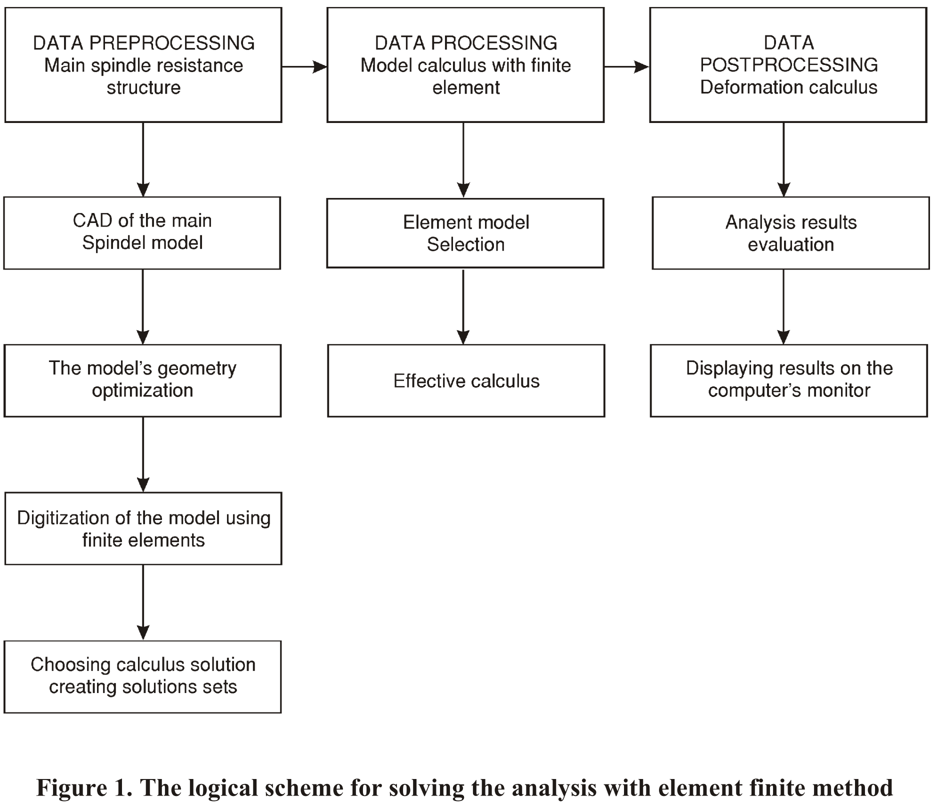

Figure 1. The logical scheme for solving the analysis with element finite method

Data preprocessing

(1) Starting from the main spindle shop drawings for the SP 630 NC lathe designed in AUTOCAD, after it was made data import from AUTOCAD into the NASTRAN design program. It was created the source project with the suitable folder. This project and this folder will be found during the experiment.

(2) The dates are imported in 2D – information that through cutting and recombining led to 3D structure. Every part of the structure has been separately designed and introduced into the program part catalog. When all the parts are designed the final assembly is made, obtaining the 3D structure [7].

(3) In the defined structure were introduced the real material's characteristics (OLC 45).

(4) The finite element model was designed, using as support the structure realized in 3D.

Using the program's facilities in digitizing, by choosing some sections through the spindle which were first multiplied, and after that joined, so that at the end the finite element model was obtained. The structure presents 4813 nodes.

(5) The digitizing quality verification was made, for large structures. It is absolutely necessary nodes overwriting verification, and free edges verification, resulting after the digitization which means undesired network discontinuities.

After the verifications is made a renumbering of nodes and an optimization of the matrix elements.

Data processing

After creating the finite element model, defining the deformation and the analysis we will pass to data processing in interactive direct or indirect system.

Data postprocessing

A solution analysis is made. Those results can be represented with their value in nodes, on elements or on the whole structure. The model deformation evolution, or the model deformation evolution in successive, steps can be selected.

The purpose of the research is the thermal deformations test of the product fixed between the chuck and the tailstock spindle sleeve vertically and horizontally. The measurements have been made at the spindle head stock, spindle sleeve and tailstock. There were installed four dial gauges, two in a vertical plane (1 and 2) and other two in horizontal plane (3 and 4), which indicates the deformations along the two planes (fig.2). The spindle drive speed n = 2000 rot./min., during 180 minutes (fig. 3), period in which were measured the temperatures and deformations every 30 minutes (fig. 4).

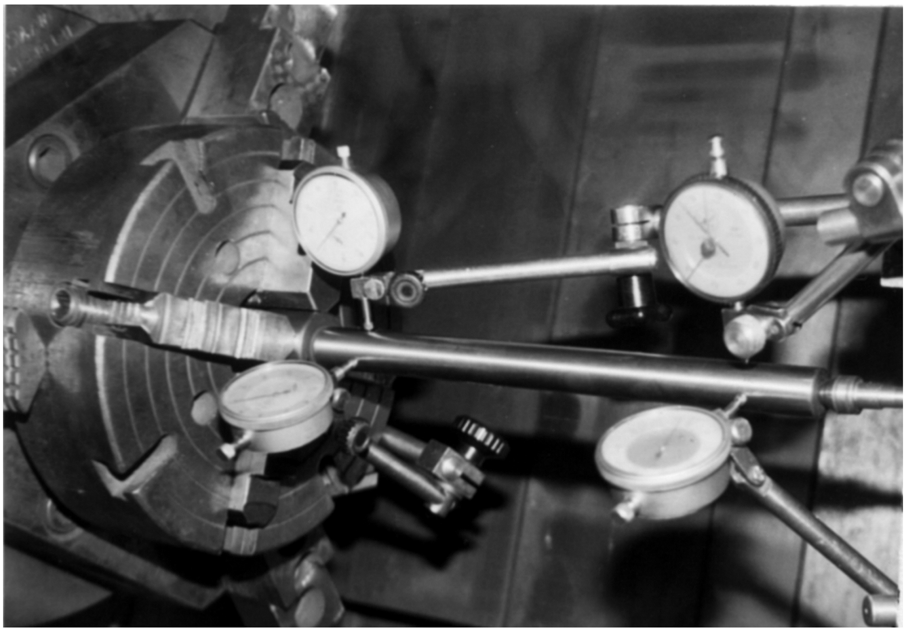

Figure 2. Measurement modes for the product fixed between the chuck and the tailstock spindle sleeve

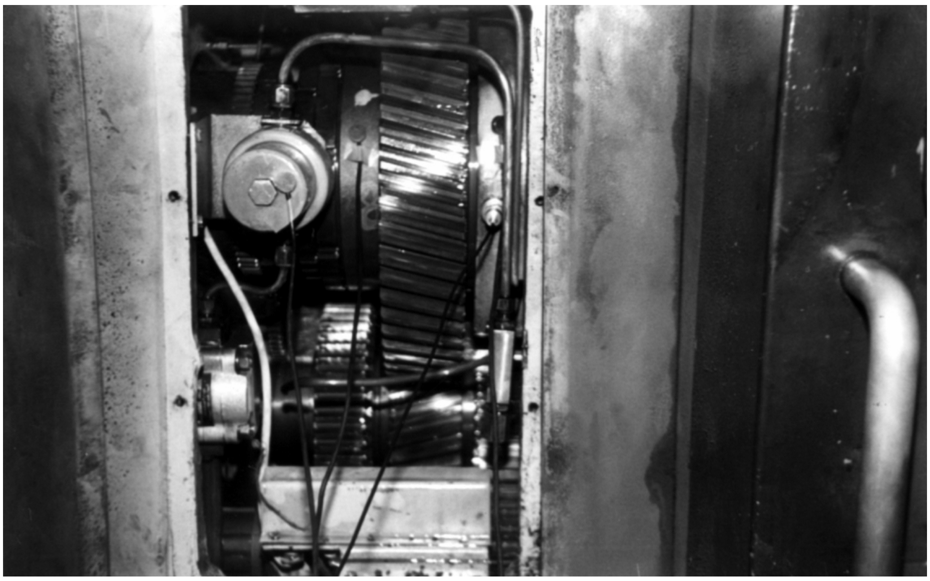

Figure 3. Gear box for a CNC lathe

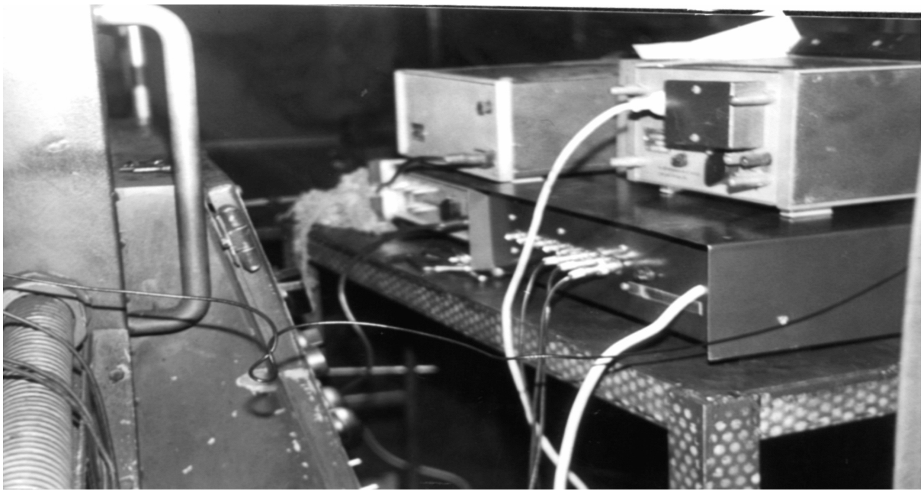

Figure 4. Measure mode

Conclusions

After the thermal deformation test for the product fixed between the chuck and the tailstock spindle sleeve it was found that after the deformations stabilization the product moved vertically for –0.014 mm at the end towards the spindle head stock and for –0.011 mm at the end towards spindle sleeve. Horizontally the product was deformed for 0.025 mm at the end towards the spindle head stock and for 0.03 mm at the end towards spindle sleeve (fig. 5 and 6).

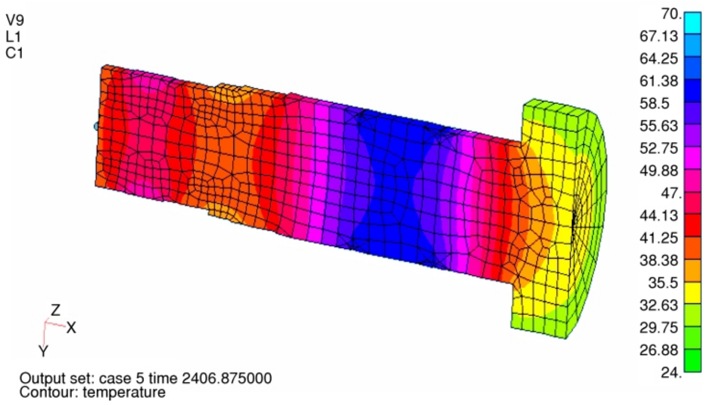

Figure 5. Temperature values in all nodes of the principal arbor at 2406 seconds from measuring start

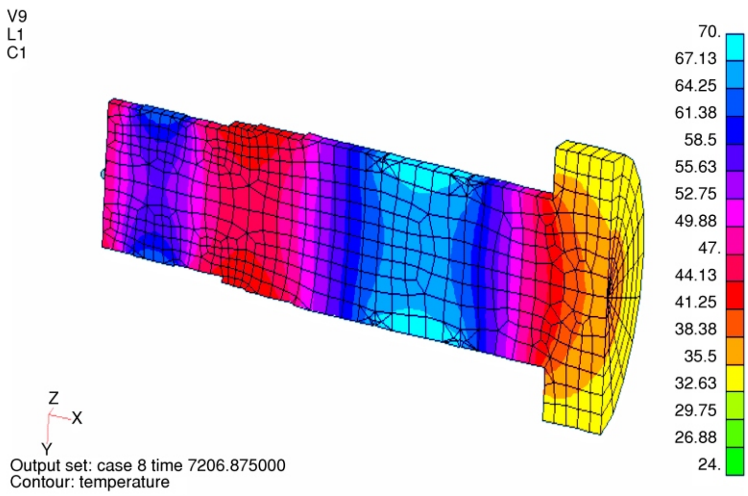

Figure 6. Temperature values in all nodes of the principal arbor at 7206 seconds from measuring start

The temperatures were stabilized after 180 minutes of functioning: at the main spindle at 48 °C, at the fix top at 44 °C, at the spindle head stock at 44 °C, and at the spindle sleeve at 50 °C (fig. 7 and 8).

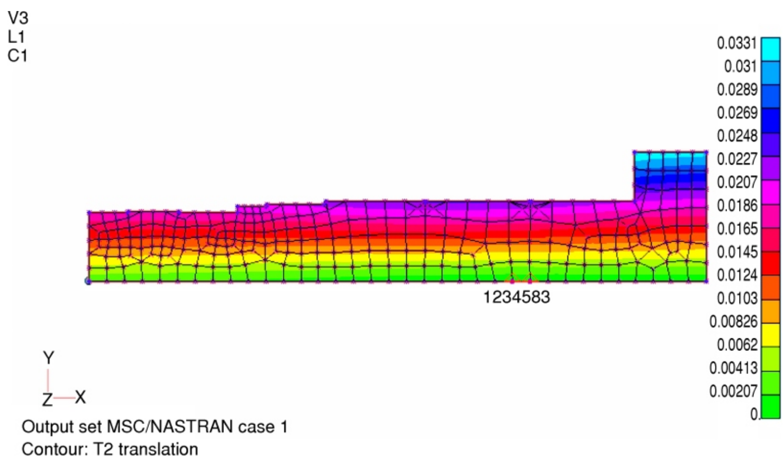

Figure 7. Deformations values in horizontal plane of the principal arbor after 180 minutes

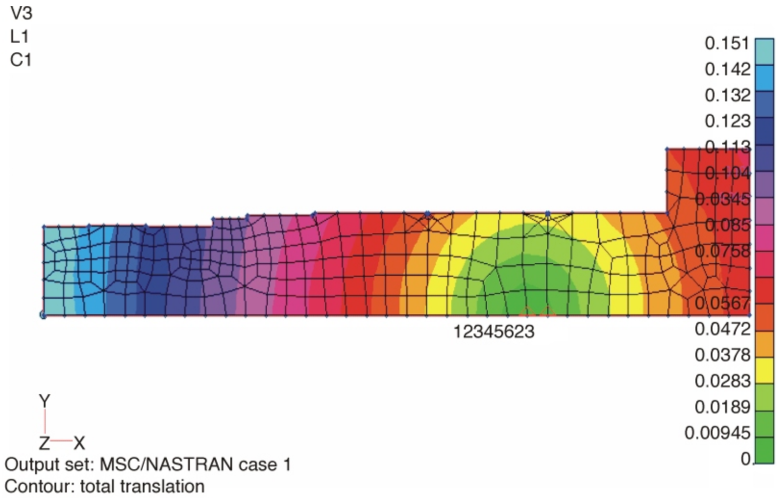

Figure 8. Temperature deformations values in all the mass of the principal arbor after 180 minutes

The maximum temperature emitted in main spindle gear is 70 °C much more like the temperature indicates in technical literature [7] .

The results that was obtained experimental and the methodology who used the element finite, to obtain the thermal deformations, helped to determined the total deformations of the axis of principal arbor at the CNC lathe [2].

References

[1] Bastiurea, G., Numerical Control, “Technical” Publishing, Bucharest, 1976
[2] Cuteanu, E., Manciov, R., Elements Finite Method in Lathe Projecting, “Facla” Publishing, Timisoara, Romania, 1980
[3] Ciclov, D., Resistance and Fiability at Variables Straings, “Facla” Publishing,Timisoara, Romania, 1975
[4] But, A., The Influence of CNC Lathe Construction on Manufacturing Accurancy, “Politehnica” Publishing, Timisoara, Romania, 2003
[5] Acerkan, M., Machine-Tools – Manufacturing and Calculation, “Technical” Publishing, Bucharest, 1978
[6] Rusu, E., Stuparu, A., Chiliman, A., Computer Numerical Control for Machine-Tools, in: AMC, Vol. 20A., “Technical” Publishing, Bucharest, 1975, pp. 28-36
[7] Militaru, C., Fiability and Accurancy in Building Technology of Machine-Tools, “Technical” Publishing, Bucharest, 1964

Authors' addresses:

A. But
Department of Mechanics, “Politehnica” University,
M. Viteazul Bld., No.1
300004 Timisoara, Romania

I. Luminosu
Department of Physics, “Politehnica” University of Timisoara,
Queen Maria Square No.1
300004 Timisoara, Romania

Corresponding author (A. But):
E-mail: adi.but@qmail.com

Paper submitted: September 1, 2004
Paper revised: March, 15, 2005
Paper accepted: April 6, 2005