Proceedings of the 17th International Conference on Manufacturing Systems – ICMaS
ISSN 1842-3183
University POLITEHNICA of Bucharest, Machine and Manufacturing Systems Department
Bucharest, Romania

VIRTUAL ANALYSIS OF THE FEED DRIVE SYSTEM

Ramona Cristina VARBAN, Radu Constantin PARPALĂ, Cristina PUPĂZĂ

Abstract: *Using a set of dedicated software's engineers are able to analyze all aspects of real life usage of machine tools elements without spending money on real prototypes. A high-speed machine tools drive system generates high friction at contact areas, into the screw-ball mechanism and bearings, thereby causing thermal expansion of the screw which adversely affects machining accuracy. In the dynamic analysis the position of all bodies are determinate as a result of time dependent forces applied to the mechanism. The finite element method is used to determinate dynamic characteristics of the feed drive system, e.g., natural-frequencies and mode shapes. The structural behaviour under dynamic and thermal loads is evaluated in order to obtain an optimized design of the feed drive.*

Key words: *ball screw, thermal error, finite element method, thermocouple, natural-frequencies.*

1. INTRODUCTION

In a feed drive system, the ball screw plays an important role as a power transmission unit and a linear scale. When a large speed of motion is demanded, the rate of feed of the ball screw transmission is much increased. Much heat is produced at both ends of the bearing support and the nut because of a greater speed of rotation.

The accumulated heat causes the temperature to rise in these areas. Then the ball screw deforms thermally and has a seriously negative effect on machine accuracy. Several types of analysis must be performed for a complete study of machine tools behavior. The structural behavior under thermal and dynamic loads is evaluated in order to obtain an optimized design of the feed drive [2].

Thermally induced error is a time-dependent nonlinear process caused by no uniform temperature variation in the machine structure. The interaction between the heat source location, its intensity, thermal expansion coefficient and the machine system configuration creates complex thermal behavior.

In this paper, for future work of combined analysis of deformation and temperature distribution, the modeling and estimation of the thermal behavior of a ball screw were performed using the finite element method (FEM).

Unfortunately at the moment there are no integrated software platform for the virtual design and analyze of the machines tools so as a result there are many problems that appear when importing models from different software applications.

Because of the impact that feed drives imply over the finite piece quality the design demands are very high so accurate analysis must be performed in order to assure a very good behavior of the whole machine tools.

This paper presents the current state of virtual prototyping of a virtual machine tools and the work focuses on the design of the feed drive system.

The main aspects of virtual design concept covered by this paper are the dynamical and thermal analysis of a machine tools feed drive using the finite element method.

2. COMPUTER AIDED DESIGN

One of the first steps in virtual prototyping is building the CAD model. The 3D model of the feed drive must be designed in order to be accepted as input by various software that have to be used for further analyses [1].

During the design phase simplified simulation models are used to estimate the impact of design parameters over the machine performance. It is well known that a very complex model can generate errors during the FEM analysis; because of this, many features from the 3D model are eliminated. Also some details that are essential, for FEM analyses are not so important for kinematical analysis and must be ignored. A CAD model must be easily redesigned so it is important to use parameters to define all the key elements of the model.

The 3D model of the feed drive was designed using the CATIA V5 CAD software mainly because of his good integration with the ANSYS software which was used for statical and dynamical FEM analyses (Fig. 1).

All 3D parts where fully parameterized in order to optimize needed parameters in FEM analysis.

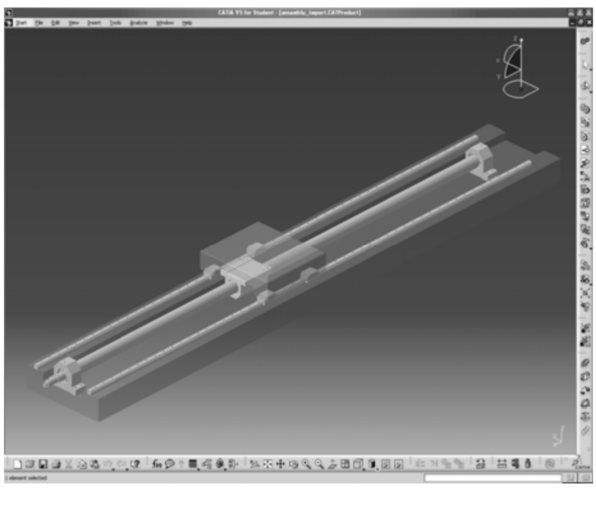


Fig. 1. 3D CAD model of the feed drive.

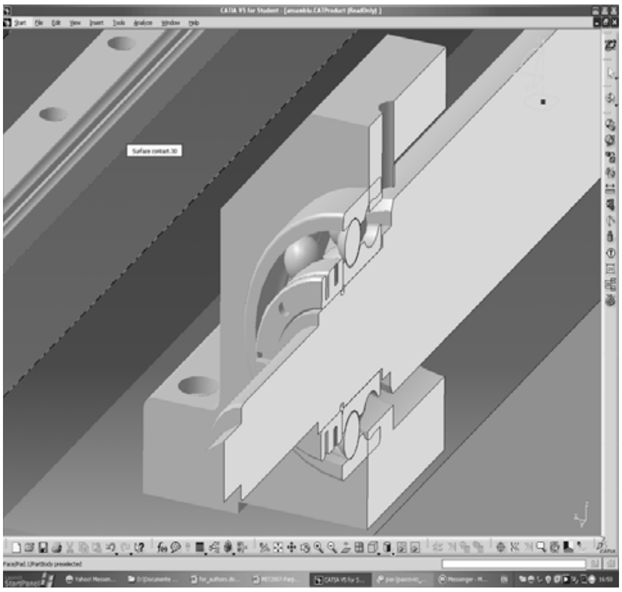


Fig. 2. Section through 3D model.

Fig. 4. CATIA inertia measurement tool.

In order to generate all the contacts between surfaces it is necessary to correctly design the 3D assembly. It is also very usefull to check all the clashes and clearances within the CAD environment.

Using the information provided by the CAD software we can set the correct tolerances for the automatic contact generation. During the simulation we can observe the mecatronical system movement and it is possible to freeze one position which can be latter imported into the FEM software.

CATIA software provides some usefully tools for space analysis. In Fig. 2 it is an example of a misplaced component. In this case the error is detectable by a visual inspection of the model.

Because we need to use the 3D model for further analyses it is a good practice to define also material properties by using a predefined model from CATIA library or by defining new materials (Fig. 3).

By defining material properties we can also check important aspects of the assembly like volume, mass, moment of inertia, etc. (Fig. 4).

Some of those information are computed by the FEM software. This way we can check if all the components are imported correctly (material properties, units, volumes).

3. ANSYS FEM ANALYSIS

3.1. Dynamic analysis for the feed drive system

The ANSYS Workbench platform is an environment that offers an efficient and intuitive user interface, superior CAD integration, automatic meshing, and access to model parameters as well as to the functionality available within the ANSYS Mechanical products.

Because of the good integration between CATIA V5R15 and ANSYS Workbench we decided to use this specialized FEM software for the static and dynamic analysis. The model was not built in ANSYS Workbench native CAD system, so we must first check if all the model's features are imported before we proceed to further analysis (Fig. 5).

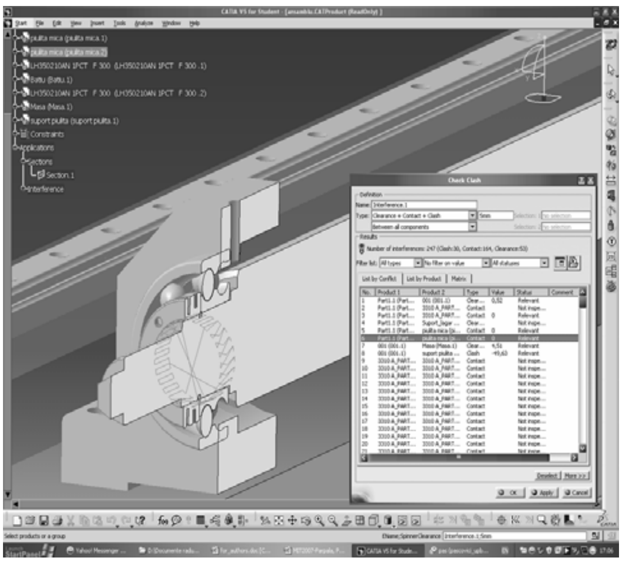


Fig. 3. Space analysis of the 3D model.

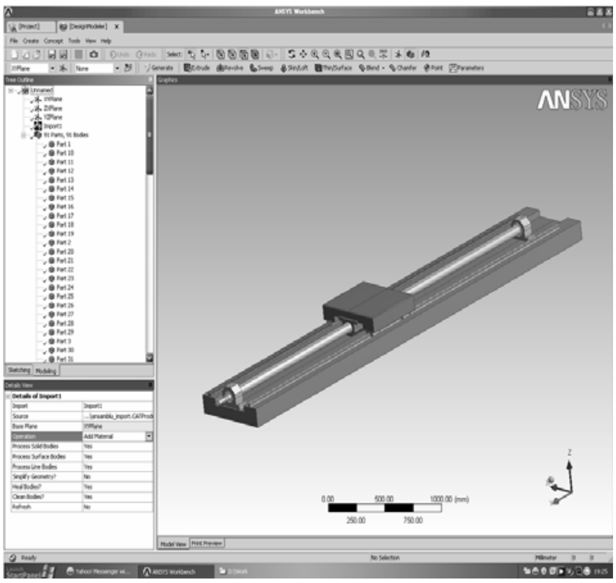


Fig. 5. Imported model.

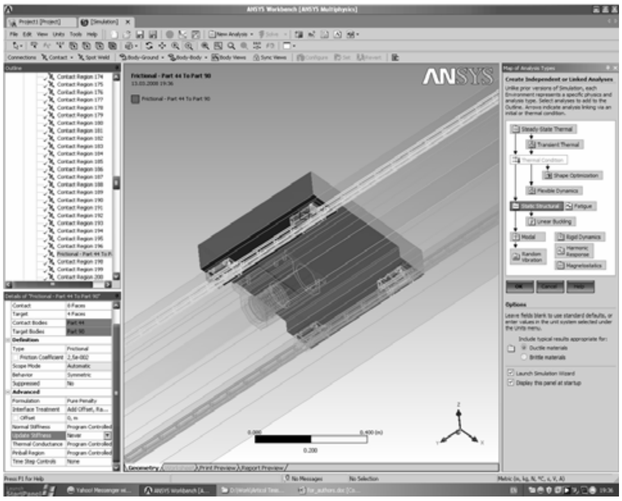


Fig. 6. Contact generation in ANSYS workbench.

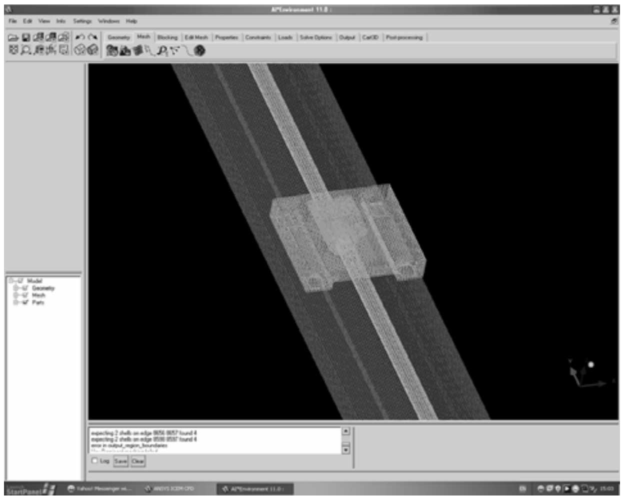


Fig. 7. The hexa dominant mesh of the model.

Getting the geometry Into Design Modeler, simulation or advanced meshing is now much easier than in almost any other FEM software.

The first thing to know about formats is that there are two classes readers and plug-ins. Readers simply translate the CAD format into workbench's internal format. A plug-in actually uses software from the CAD vendor and opens up the geometry in the native format and gives workbench all the information it needs in a native format.

Often the geometry provided by readers is referred as dumb and plug-in geometry as smart because this kind of geometry can be transferred back to the CAD files. It is a good practice to import all the parts into the Design Modeler before proceeding to simulation.

After a successful import of the model it is very easy to automatically generate all the contacts between components, setting up the correct parameters for contact creation (Fig. 6). It is essential to determine which behavior formulation, to use (Table 1).

There are three basic motions that are described by various behaviors. The contact pair can be open, closed or slide relative to each other. Depending on the behavior type, there are different ways that ANSYS enforces these relationships (Table 2).

Table 1

Contact behavior

Name	Gap Open/Close?	Sliding allowed
Bonded	NO	NO
Rough	YES	NO, infinite μ
No Separation	NO	YES, μ = 0
Frictionless	YES	YES, μ = 0
Frictional	YES	YES, if $F_{sliding}<F_{friction}$

Table 2

Contact behavior

Name	Equation Solved	Default Contact Detection
MPC	Generate GE's	Nodes
Pure Penalty	$F = kx$	Gauss
Augmented Lagrange	$F = kx + \lambda$	Gauss
Normal Lagrange	Nodal Pressure DOF	Node

The MPC method simply writes constraint equations between the contacting bodies. Pure penalty enforce contact stiffness for a given penetration. The Augmented Lagrange is similar to Pure Penalty, except that it includes a factor for contact pressure to eliminate some chattering effect. The Normal Lagrange adds a Contact Pressure DOF to the model. In terms of runtime, the formulations shown are listed in increasing run time.

There is a strong connection between contact generation and mesh generation. For a proper meshed structure it is necessary to repair the geometry before any further operations. The main purpose of the CAD repair tool is to detect and close gaps between neighboring surfaces.

The quality of the mesh needs to be checked before applying loads and constraints. Quality can be measured by the various criteria. For Hexa dominant meshes the quality is calculated as the determinant (Fig. 7). The Determinant, more properly defined as the relative determinant, is the ratio of the smallest determinant of the Jacobian matrix divided by the largest determinant of the Jacobian matrix, where each determinant is computed at each node of the element. The Determinant can be found for all linear hexahedral, quadrahedral, and pyramidal elements.

The dynamic response of a machine, structure, or system can be determined by superposing its natural modes of vibration when the amplitudes of motion are small.

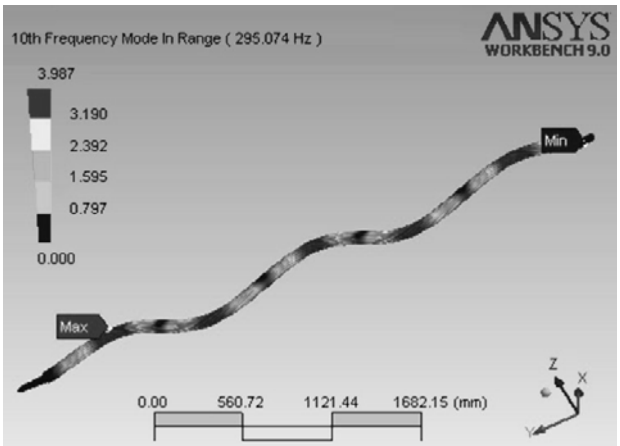


Fig. 8. 10^{th} Frequency mode for the screw-drive.

Table 3

Natural frequency

Mode	Frequency, [Hz]
1	22.29
2	22.3
3	61.41
4	61.43
5	120.18
6	120.23
7	198.15
8	198.23
9	294.94
10	295.07

Thus, a complete dynamic description of the machine requires the determination of the modal frequencies, mode shapes, and the system parameters-equivalent mass, stiffness, and damping ratio. The procedure determining this information of a system is called Modal Analysis [5].

The Finite Element Method (FEM) is widely used to perform a Modal Analysis. FEM is extremely useful for complicated devices and structures with unusual geometric shapes. The frequencies that are calculated by the program (Table 3) can be further used for other verifications (Fig. 8).

3.2. Thermal analysis for the feed drive system

The thermo-elastic machine behavior, i.e. the load dependent temperature distribution and the resulting deformation of the machine tool, is influenced by a variety of constructional and thermo technical parameters (Fig 9). To closer look, the energy transmission system first subdivided into a electromechanical and a purely mechanical system. In electromechanical transfers system the electrical energy into mechanical energy of rotation changed.

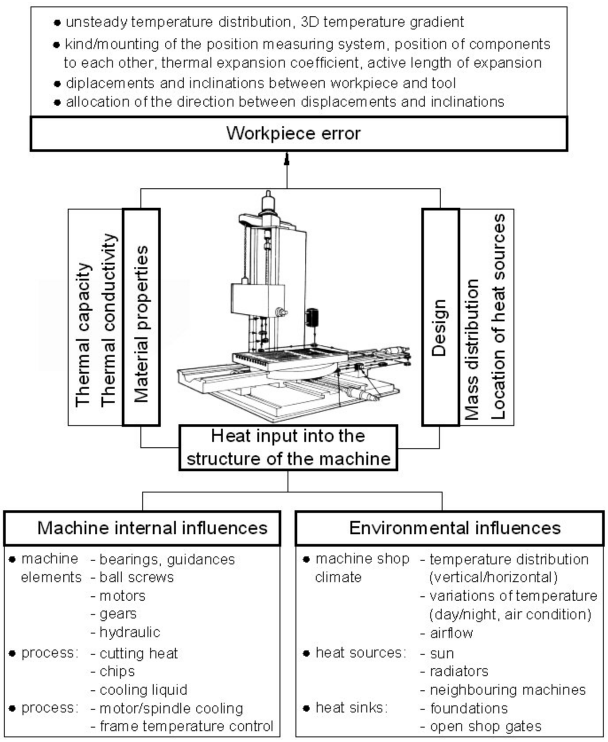
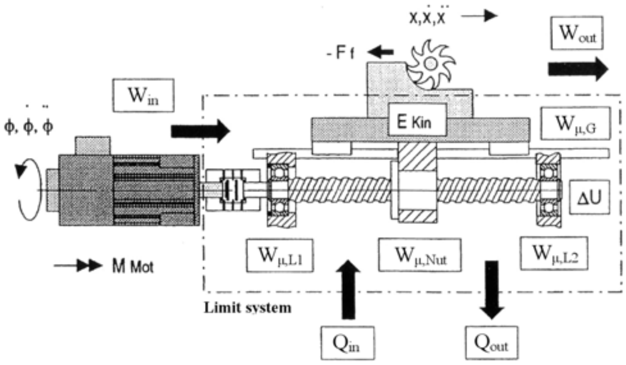


Fig. 9. Reasons for thermo-elastic deformations.



M_{Mot} – torques moment; F_f – friction force; W_{in}, W_{out} – work feed in and out; Q_{in}, Q_{out} – warm volume in and out; E_{kin} – kinematic energy; ΔU – difference of internal energy; $W_{\mu,G}$ – mechanical work guide; $W_{\mu,L1}$ – mechanical work in warehouses of the screw ($L1$); $W_{\mu,Nut}$ – mechanical work in nut; $W_{\mu,L2}$ – mechanical work in warehouses of the screw ($L2$).

Fig. 10. Thermodynamic model for the mechanical energy transfer of the feed drive system.

The thermodynamic processes in this transformation are not studied in details. The rotational energy is transmitted to the transfer system from the servomotor in a period of time Δt as external work W_{in} (Fig. 10).

In this energy transformation, the outer heat Q_{in} goes into the bearings, guideways and spindle-nut and influences the transformation of the internal energy ΔU. Due to this heat transfer processes, in the mechanical structure an unsteady temperature distribution arises, which are the target of the analytical models. The mechanical structure of the feed axis therefore embodies both a mechanical and thermal energy transmission system. For the purposes of thermodynamics, this is considered a closed moving system.

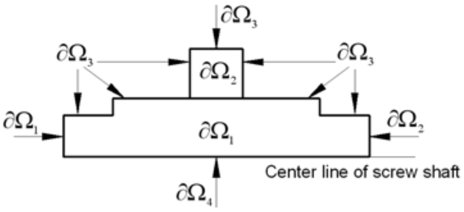
Thermally induced error is a time-dependent nonlinear process caused by no uniform temperature variation in the machine structure. The interaction between the heat source location, its intensity, thermal expansion coefficient and the machine system configuration creates complex thermal behavior.

3.2.1. Description of prediction of thermal errors

The mathematical foundation of the proposed approach is multiple regression analysis. In this statistic approach the values of a dependent variable are described or predicted in terms of at least two independent variables. Thus a model that represents the behavior of the variation of thermal errors is written in the form:

$$Y = b_0 + b_1 \cdot X_1 + b_2 \cdot X_2 + \ldots + b_n \cdot X_n + e. \quad (1)$$

The dependent variable Y (thermal error) is expressed as a function of N independent variables X_1, X_2, ... , X_n (temperature raises). The random error term is e that is normally distributed with mean zero and constant variance, added to allow for deviation between the deterministic part of the model, $b_0 + b_1 \cdot X_1 + \ldots + b_n \cdot X_n$, and the value of the dependent variable Y. The random component makes the model probabilistic rather than deterministic. The coefficients b_0, b_1, ..., b_n, estimated from sampling data, are called regression coefficients.



$\partial\Omega_1$, $\partial\Omega_2$ - constant temperature boundary; $\partial\Omega_3$ - convective heat transfer boundary; $\partial\Omega_4$ - adiabatic boundary;
Ω_1 - screw shaft domain; Ω_2 - nut domain

Fig. 11. Simplified model of the ball screw for FEM.

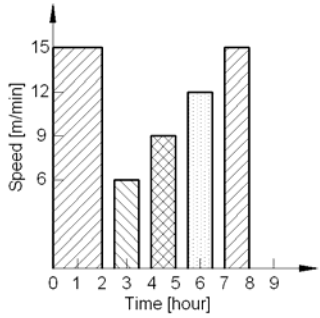


Fig. 13. Speed history of the table motion.

Two-dimensional temperature distributions of a ball-screw system preloaded in the axial direction are estimated at various moving velocities and stop times by the finite element method (FEM).

The simplified model for heat transfer calculation is described in Fig. 11 along with the boundary conditions. Since the temperature of a ball screw ranges between several scores, the radiation term can be neglected. The problem is therefore defined as transient heat conduction in non-deforming media during a given time-interval without radiation.

3.2.2. Experimental set-up and procedure

The objectives of the experiments were to establish a system to measure the ball screw transmission system thermal errors, to determine a suitable model for the thermal error data, and to predict (and to compensate) thermal error according to the determined model.

The experimental set-up comprises a ball screw, a driving unit, a data acquisition unit and a controller.

The specifications of the system follow:

- travel: 1000 mm,
- ball screw: diameter 80 mm, pitch 10 mm, double-nut, preloaded,
- max. feed rates: 15 m/min,
- mounting method: fixed supported.

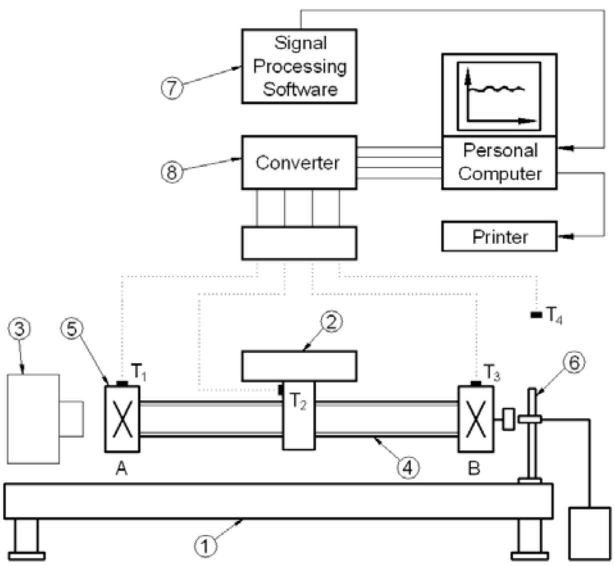


Fig. 12. Schematic diagram of the experimental set-up:
1 – uniaxial feed drive system; 2 – table; 3 – servomotor; 4 – ballscrew; 5 – thermocouple; 6 – displacement gauge; 7 – data processing software; 8 – data collection system.

In order to estimate the heat transfer and deformation of the ball-screw system under long-time movement of the nut, experiments were performed with the arrangement shown in Fig. 12.

The thermocouples were attached to positions T_1, T_2 and T_3 as shown in Fig. 12 to measure temperature increases of the front bearing, nut and rear bearing as key heat sources, respectively.

The fourth thermocouple was used to measure the environmental temperature.

Displacement gauge was located at in the end edge of the ball screw to measure its thermal error. The speed history of the table motion is shown in Fig. 13.

The data (temperature increases and thermal error) were obtained every 4 min. The variation of temperature rises of positions T_1, T_2 and T_3 and the thermal error is shown in Figs. 14 – 16.

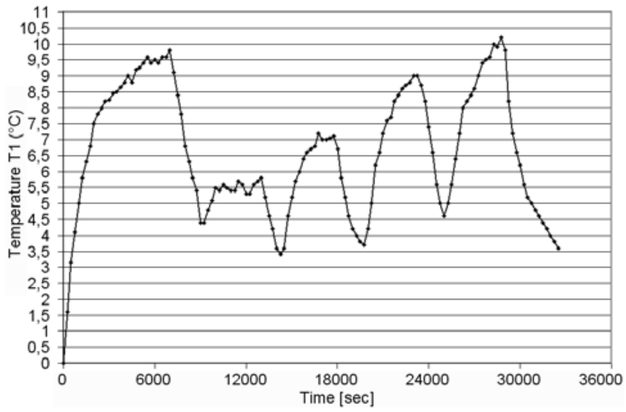


Fig. 14. Temperature profile of the front bearing (T_1) during the thermal displacement measurement.

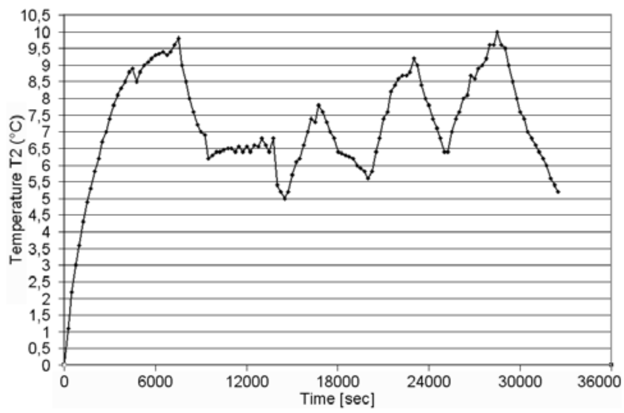


Fig. 15. Temperature profile of the nut (T_2) during the thermal displacement measurement.

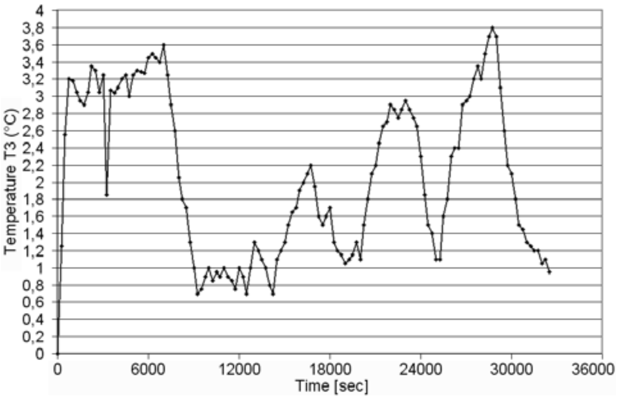


Fig. 16. Temperature profile of the rear bearing (T_3) during the thermal displacement measurement.

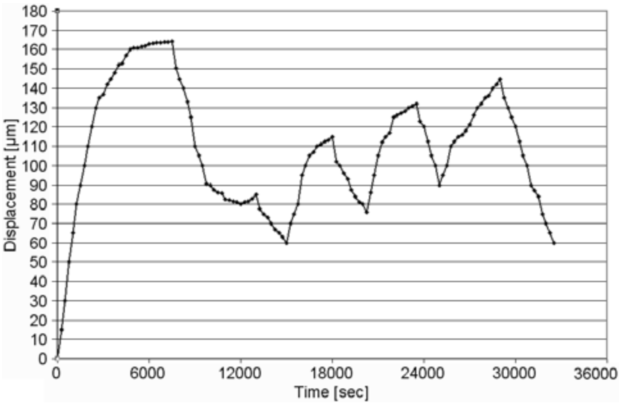


Fig. 17. Thermal displacement profile of the end of the ball screw.

By comparing of the results in Figs. 12 − 15, we find out that there is a relation between the thermal error and the temperature increase at positions T_1, T_2 and T_3.

A multiple regression analysis was conducted to determine the effects temperature rises of key heat sources on thermal error.

For the variation of temperature rises of positions T_2 and the thermal error was used ANSYS Analysis that shown in Fig. 18.

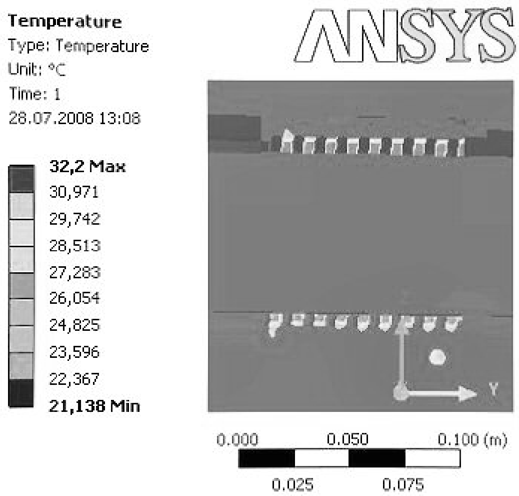


Fig. 18. Temperature profile of the nut (T_2).

4. CONCLUSIONS

By using ANSYS Workbench we can optimize the design process by changing one or more of the initial parameters; those parameters are automatically updated into the CATIA 3D CAD model.

By analyzing the calculation result in the post-processing program the designer can evaluate the machine properties during the design stage.

Today the main problem in checking structures consists in importing and preprocessing the CAD model. It is well known that the geometry of the model can dramatically change FEM results. The ballscrew preload raises the temperature increases of both support bearings, especially the bearing on the driven side.

The surface temperature of the ballscrew decreases because the thermal effects relax the preload, thereby decreasing the friction between the nut and the ballscrew.

The multiple regression analysis is adequate to predict the ball screw thermal errors with variation of table speed and temperature history. The predicted thermal error data can be used to correct the error with a suitable numerical control route.

REFERENCES

[1] Altintas, Y., Brecher C., Weck M. & Witt S. (2005). *Virtual machine tool*, Annals of the CIRP, **54**/2, pp. 651-669.

[2] Kim, S.K., Cho, D.W., (1997). *Real-time estimation of temperature distribution in a ball-screw system*, International Journal of Machine Tools and manufacture, Vol. 37, No. 4, pp. 451-464.

[3] Predincea, N., Pupăză, C., Toma, O. (1998). *Analiza deformațiilor termice ale mecanismului şurub-piuliță* (Analysis of thermal deformations of ball-screw system), The VIII-th Conference of Managerial and Technological Engineering, pp. 499-506, Timişoara, România, 28-30 May.

[4] Schmitt, T., (1996), *Modell der Wärmeübertragungsvorgänge in der mechanischen Struktur von CNC-gesteuerten Vorschubsystemen* (Model of the heat transfer in the mechanical structure of a CNC feed drive system), ISBN 3-8265-1476-9, PhD Thesis, TU Darmstadt, Shaker Verlag, Aachen.

[5] Zaeh, M. & Oertli, Th. (2004). *Finite Element Modeling of Ball Screw Feed Drive Systems*, Annals of the CIRP, **53**/1, pp. 289-292.

Authors:

Eng., Ramona Cristina VARBAN, PhD Student, University "Politehnica" of Bucharest, Faculty of Engineering and Management of Technological Systems, Machines and Production Systems Department,
E-mail: crissram@yahoo.com

Eng., Radu Constantin PARPALĂ, PhD Student, University "Politehnica" of Bucharest, Faculty of Engineering and Management of Technological Systems, Machines and Production Systems Department,
E-mail: pmc@mix.mmi.pub.ro

PhD. Eng., Cristina PUPĂZĂ, Associate Professor, University "Politehnica" of Bucharest, Faculty of Engineering and Management of Technological Systems, Machines and Production Systems Department,
E-mail: cristinapupaza@yahoo.co.uk