Design of Machines and Structures, Vol. 10, No. 2 (2020), pp. 11–19.
https://doi.org/10.32972/dms.2020.008

THERMAL AND STRUCTURAL SIMULATIONS OF A CNC TURNING CENTER

SEBASTIAN CABEZAS – SZILÁGYI ATTILA

University of Miskolc, Department of Machine Tools
3515 Miskolc-Egyetemváros
sebascabezas1@gmail.com

Abstract: This article shows the application of Finite Element Analysis FEA using Siemens NX 12 Nastran for thermal and structural simulations of a CNC turning center CTX Alpha-500 Gildemeister. The simulations were performed under steady state conditions, stationary system, the heat generated by the workpiece and the machine is neglected and the control volume remains constant.

Keywords: *FEA, steady state, control volume*

1. INTRODUCTION

When it comes to determine the thermal distribution temperatures, the thermal gradient and the displacements occasioned due to heating in a complete system, it is not suitable to make numerical calculations, because the machine is treated as a three dimensional element. Moreover, the machine has many individual components that has a non-determined geometrical shape that cannot be assumed as one-dimensional element.

The use of FEM software is a powerful tool that provides approximate solutions with high accuracy in less time. By computer simulations, the task which by traditional methods will take some days, with these software's the results can be obtained in a few hours and it is possible to make changes and perform more simulations.

2. PROCEDURE

The procedure has been divided into two main parts, 3D modelling and thermodynamical model in which the main aspects to create the 3-dimensional model and the required characteristics to make thermal and structural analysis have been described.

2.1. 3D modelling of the turning center

This section describes the steps that were followed to create the 3D model of the turning center CTX-Alpha 500 GILDEMEISTER using CAD software SIEMENS NX 12.

Measurements:

The dimensions were obtained with a flexible meter $ELLIX$ 573 522 $ECII$ $2m$ for large distances, and a caliper Mitutoyo for small components.

3D CAD Model:

The machine was modelled with the software Siemens NX 12 Modelling. The CAD model of the machine was described in chapter 4. The parts of the machine were divided in subassemblies named: X Axis, Xs Axis, Y Axis, Chuck N1, Chuck N2 and Main Table (see *Figure 1*).

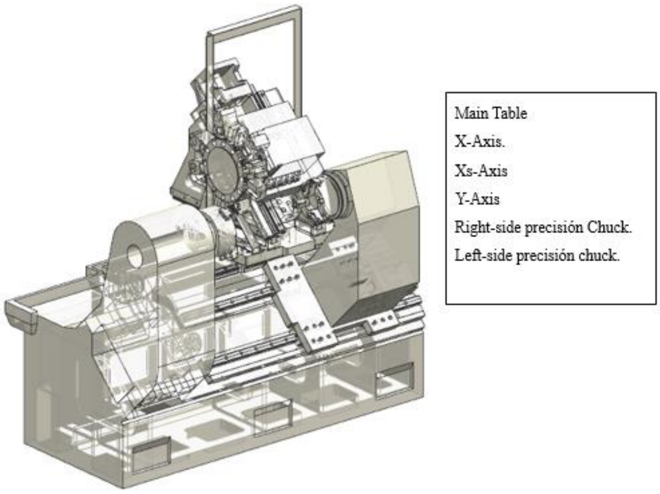

Figure 1
3D Assembly turning machine CTX-Alpha 500

2.2. Thermodynamical model

This section describes the steps followed to create the finite element model for thermodynamical analysis of the turning center in which the fundamentals of heat transfer has to be applied. Moreover, it is needed to stablish thermal conditions and analyze the type of solution, the location of the heat sources in the 3D model, the boundary and initial conditions, restrictions, topology of mesh and the thermal loads.

Finite element method

The finite element method is a numerical tool for determining approximate solutions to a large class of engineering problems. In the finite element method, the actual continuum or body of matter (solid, liquid, or gas), is represented as an arrangement of subdivisions called finite elements.

Elements are interconnected at specified joints. Since the variation of the field (displacement, stress, temperature, velocity, pressure) inside the continuum is not known, assuming that the variation of the field variable inside a finite element can be approximated by a simple function. These functions known as interpolation functions are defined in terms of the values of the field at the joints. By solving the field equations, the nodal values of the field variable will be determined [1].

To determine the thermal distribution temperatures, the thermal gradient and the displacements due to heating in a complete system, it is not suitable to make numerical calculations. The turning center CTX Alpha 500 is treated as a three dimensional component. Moreover, the machine has many individual components that has a non-determined geometrical shape and cannot be assumed as one-dimensional element.

Type of elements

Engineering systems may be simplified by subdividing them in elements. The elements can be analyzed by first principles. By assembling the elements together, the analysis of a full original system can be reconstructed. These systems are called discrete systems. The turning machine was divided in three dimensional elements called tetrahedrons. This procedure establishes the most accurate approximations compared to one-dimensional or two-dimensional elements.

The temperature distribution in a tetrahedron element is given by the equation:

$$T(x, y, z) = \propto_1 + \propto_2 x + \propto_3 y + \propto_4 z \tag{1}$$

The number of subdivisions of the turning center obtained by the software was 597210 elements.

Type of solution

Here the steady state solution occurs when there is no accumulation of mass and energy within the control volume, and the properties at any point within the system are independent of time. In this project the system was considered as a steady state for the following reasons: the heat loads will be constant and won’t vary with time, the heat sources are independent of time, the temperature distribution is considered as function of time, there is no variation of mass in the control volume, there is no variation of energy in the control volume, the materials of the machine are considered to be isotropic and are the same at any point in the machine.

Heat Sources and location

To perform the thermal simulation is necessary to identify the energy losses produced by the heat sources in the turning center. Heat sources are servomotor loads, linear rail guides and bearings. These heat loads are located in the Axis of rotation and in the linear guides who are used to slide the systems in the machine. Convective heat sources are not considered in this analysis as loads but are considered as initial and boundary conditions. In the simulation the heat dissipated between the workpiece and the machine is not considered.

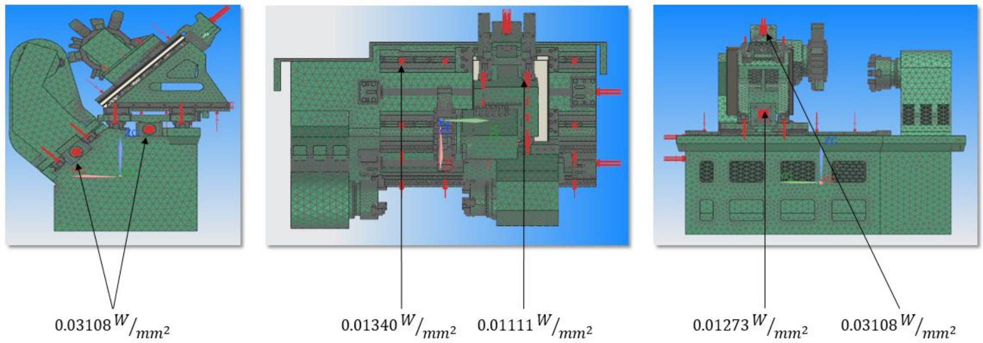

Figure 2
Heat sources and location

Table 1
Heat Sources

Heat sources	$Q[watts]$
Servomotors S-1FK7060-5AF71-1FH0. (x4)	25
Linear Motion blocks HSR-35A (x8)	31.74
Linear Motion blocks HSR-35LA (x8)	38.28

Boundary Conditions

Convective flow of the air is considered in the analysis as a boundary condition. The media where the machine is surrounded is natural air and the natural convective coefficients are considered.

Initial Conditions

The temperature of the air in the workshop is considered as constant and it is used as the initial condition at any time. $\mathrm{T}(\mathrm{t}=0)=\mathrm{T}(\mathrm{t})$

Restrictions

The restrictions considered for this analysis are:

- Seasons of the year are not taken into account. The values considered for simulation are said to be the same in spring, summer, fall and winter.
- The operational speeds of the motors are taken from catalogues and considered as nominal speeds.
- The heat generated by the workpiece and the machine is neglected.

- The time operation of the machine is not considered because the system will be treated as steady state.

Thermal Loads

The available thermal loads in NX Nastran environment are:

- QVECT Directional heat flux from a distant source.
- QVOL Volumetric internal heat generation.
- QHBDY Heat flux applied to an area defined by grid points.
- QBDY1 Heat flux applied to surface elements.
- QBDY2 Heat flux applied to grid points associated with a surface element.
- QBDY3 Heat flux applied to surface elements with control node capacity.
- SLOAD Power into a grid or scalar point.
- NOLIN1 Nonlinear transient load as a tabular function.

The heat dissipation produced by the electric motors connected to the shafts, the linear motion blocks rolling on the surfaces of the rail guides and bearings is treated as heat conduction. For the simulation the required loads dealing with heat transfer by conduction are QBDY1 and SLOAD.

Geometric surface element mode

Surface element geometries are associated with surface types. NX Nastran classifies in three forms:

- CHBDYE deals with the geometry type implicitly by reference to the underlying conduction element.
- CHBDYG identifies the radiation surface geometry and material.
- CHBDYP this specification is identified on the CONVM entry as the element identification number.

In this analysis the geometric surface element mode was CHBDYE dealing with heat transfer by conduction.

Material components

The material for simulations were obtained from the catalogues and manuals of the turning center and from the manufacturers of different components of the machine e.g. bearings, linear rail guides, linear motion blocks.

Mesh generation

The domain discretization is classified under different categories such as topology, method of generation, element type, conformity, body alignment. Structured and unstructured meshes are widely classified by topology. The mesh topology used and performed by NX Nastran for this experiment is non-uniform structured mesh.

3. THERMAL SIMULATIONS

After defining, the thermal loads, the mesh generation, boundary and initial conditions into the software is possible to run the thermal simulation of the turning center obtaining the results shown in *Table 2*.

- CHBDY Geometry types 180598.
- CONV Constraint points 179092.
- CTETRA Number of elements 597210.

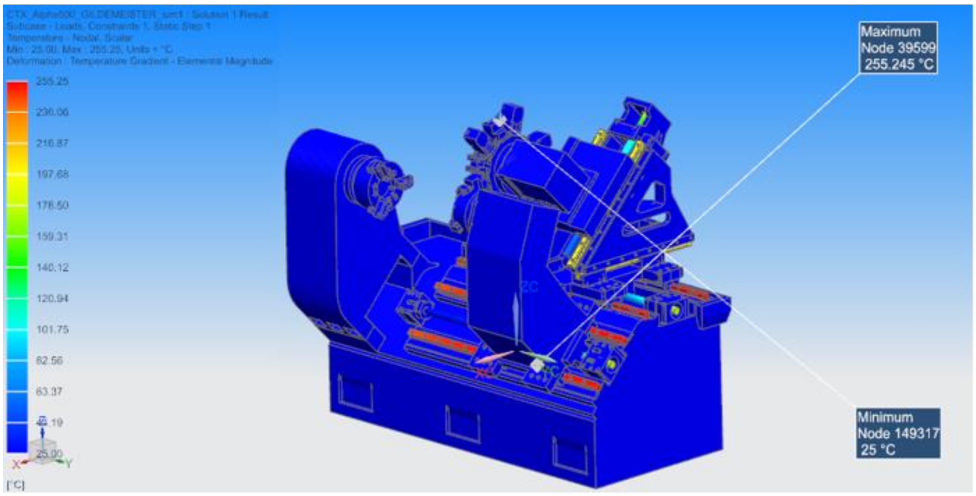

Figure 3
Temperature distribution

Table 2
Temperature distribution results

	Node	Value [°C]
Minimum scalar temperature	149,317	25
Maximum scalar temperature	39,599	255.245
MTAxis1	95,567	175.952
MTAxis2	102,085	175.95
XsAxis	88,075	114.564
Xaxis	149,820	177.371
Linear rail guides Main table	36,478	250.984
Linear rail guides XsAxis	34,667	203.685
Linear rail guides XAxis	33,762	201.887

4. STRUCTURAL SIMULATIONS

Applying the results obtained during the thermal simulation as temperature loads, to find the displacements which will cause inaccuracies in the precision of the turning center.

Displacement in X direction:

Table 3
X-Direction deformation

X- Direction	Node	Deformation [mm]	Deformation [µm]
Minimum	87,681	0.00558361	5.58361
Maximum	39,162	–0.0125772	–12.5772

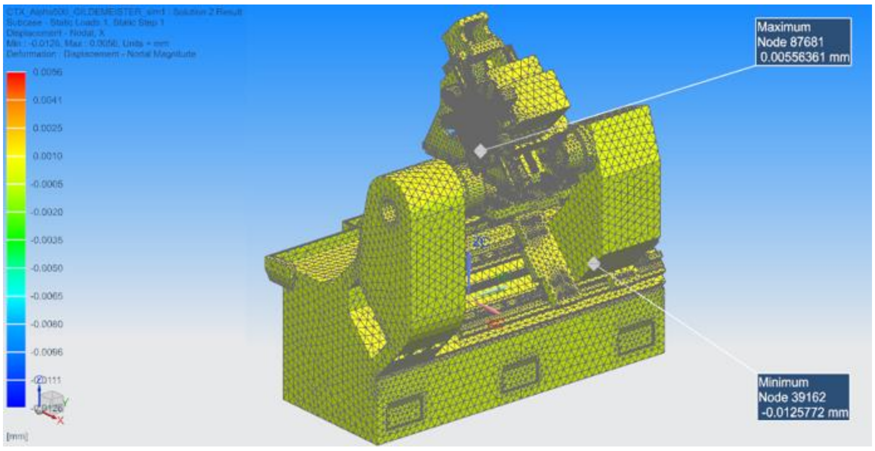

Figure 4
Displacement in X-direction

Displacement in Y direction:

Table 4
Y-Direction deformation

Y- Direction	Node	Deformation [mm]	Deformation [µm]
Minimum	33,239	0.00397778	3.97778
Maximum	89,884	–0.00621152	–6.21152

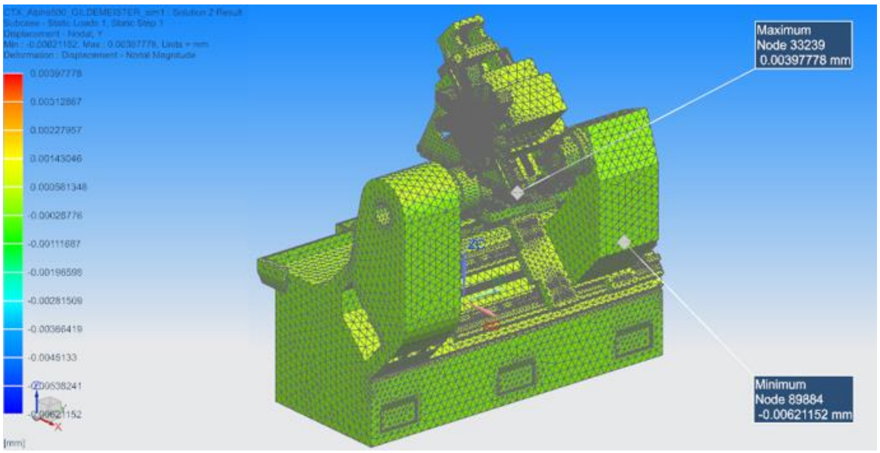

Figure 5
Displacement in Y-direction

Displacement in Z direction:

Table 5
Z-Direction deformation

Z- Direction	Node	Deformation [mm]	Deformation [µm]
Minimum	35,395	0.0102213	10.2213
Maximum	36,178	–0.0125486	–12.5486

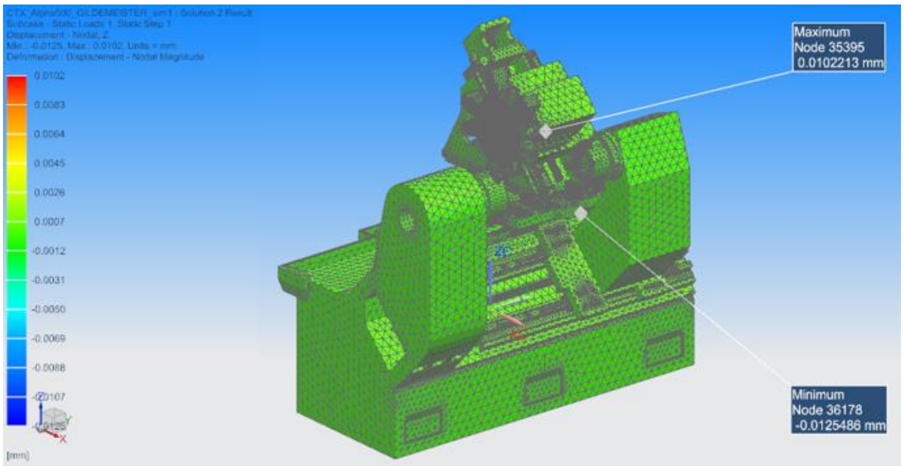

Figure 6
Displacement in Z-direction

5. SUMMARY

The turning machine was analyzed by FEM simulations dividing into small 3D tetrahedrons with four nodes each one of them, the size of the tetrahedrons were defined in order to improve the time of the interactions in the simulations that was around 20 minutes for thermal analysis and 23 minutes for structural analysis.

The workshop environment where the turning machine is located can be of significant importance for accuracy of manufacturing. Temperature controlled environments require a big amount of investment, which in the majority of the cases are undesirable and impractical. For the simulation analysis of the turning machine, the temperature environment established was 25 °C, due to the fact that the workshop where the machine is located possesses air conditioning system which can be regulated independently of the seasons of the year.

The maximum temperature estimated in steady state with the established parameters of the heat sources was $T_{max} = 255.205\ °C$, node 39,599, located in the rail guides of the main table and Chuck N2. No cooling and lubricant factors were considered during the simulation. The increase of the temperature in the rail guides occur due to the friction if the system is not properly aligned, lubrication is not appropriate or the bearings are failing. To avoid this problem is needed to take preventive maintenance to check the state of the linear rail guides and the bearing components.

ACKNOWLEDGMENT

The described article/presentation/study was carried out as part of the EFOP-3.6.1-16-2016-00011 *Younger and renewing University – Innovative Knowledge City – institutional development of the University of Miskolc aiming at intelligent specialization* project implemented in the framework of the Szechenyi 2020 program. The realization of this project is supported by the European Union, co-financed by the European Social Fund.

REFERENCES

[1] Nithiarasu, P., Lewis, R. W. Seetharamu, K. N. (2016). *Fundamentals of the finite element method for heat and mass transfer*. Chichester, UK: John Wiley & Sons.

[2] Mekid, S. (2009). *Introduction to precision machine design and error assessment*. Boca Raton, FL, Taylor & Francis Group.

[3] Zhao, H., Yang, J., Shen, J. (2006). *Simulation of thermal behavior of a CNC machine tool spindle*. Sanghai, Elsevier.

[4] Mian, N. S., Fletcher, S., Longstaff, A. P., Myers, A. (2012). *Efficient estimation by FEA of machine tool distortion due to environmental temperature perturbations*. Huddersfielsd, UK, Elsevier.

[5] SIEMENS (2014). *Thermal Analysis User's Guide*.