Impact Factor:

ISRA (India)	= **6.317**	**SIS** (USA)	= **0.912**	**ICV** (Poland)	= **6.630**
ISI (Dubai, UAE)	= **1.582**	**РИНЦ** (Russia)	= **3.939**	**PIF** (India)	= **1.940**
GIF (Australia)	= **0.564**	**ESJI** (KZ)	= **8.771**	**IBI** (India)	= **4.260**
JIF	= **1.500**	**SJIF** (Morocco)	= **7.184**	**OAJI** (USA)	= **0.350**

SOI: 1.1/TAS DOI: 10.15863/TAS

International Scientific Journal
Theoretical & Applied Science

p-ISSN: 2308-4944 (print) **e-ISSN:** 2409-0085 (online)

Year: 2023 **Issue:** 10 **Volume:** 126

Published: 10.10.2023 **http://T-Science.org**

Issue **Article**

Makhliyo Sayfidinova
Navoi State University of Mining and Technologies
Corresponding author
sayfidinova.mk@gmail.com

Okhunjon Sayfidinov
Navoi State University of Mining and Technologies
Corresponding author

Mukhlisa Aslonova
Navoi State University of Mining and Technologies
Corresponding author

INVESTIGATING THE THERMAL IMPACT ON LATHE MACHINE TOOL INSERTS

Abstract: *This study employs Finite Element Method (FEM) simulations to analyze the lathe machine tool insert, employing a division into small 3D tetrahedra with multiple nodes, optimizing tetrahedron sizes for enhanced interaction time. A nominal ambient temperature of 20°C is specified at the center of the insert due to its connection to the holder, affecting its temperature. The FEM simulations scrutinize the insert, revealing the influence of various parameters on material temperatures. Materials with lower heat transfer properties and lower density exhibit lower temperatures.*

The study indicates that assembling the lathe machine tool for simulation yields more precise results. The temperature effect, though minimal for the insert, reveals advantages when using carbon and titanium. The accuracy of results for simulating the lathe machine tool assembly relies on detecting heat sources accurately, optimizing mesh sizing, and characterizing the material properties of the machine tool components.

Key words: *Finite Element Method, FEM, Simulation, Lathe Machine Tool, Temperature Effect, Material Properties, Mesh Sizing, Heat Transfer.*

Language: *English*

Citation: Sayfidinova, M., Sayfidinov, O., & Aslonova, M. (2023). Investigating the thermal impact on lathe machine tool inserts. *ISJ Theoretical & Applied Science, 10 (126)*, 250-261.

Soi: http://s-o-i.org/1.1/TAS-10-126-17 ***Doi***: crossref https://dx.doi.org/10.15863/TAS.2023.10.126.17

Scopus ASCC: *2200.*

Introduction

Errors may manifest during machining across various domains, encompassing tool wear, mechanical deflection, and thermal influences. Achieving the desired thermal error of less than ±10 µn, which the manufacturing sector seeks throughout the entire operational range of the machine tool [1], remains a challenge. Despite strenuous efforts by machine tool manufacturers, residual thermal displacements persist due to the complexity of designing for the full spectrum of operating conditions and the presence of uncontrollable external factors. Consequently, the adoption of a feedback control system emerges as a logical and pragmatic solution [2].

A significant hurdle in thermal error control lies in the real-time measurement of thermal displacement with respect to the tool and workpiece during machining [2, 3]. As a response to this challenge, process models that establish correlations between thermal deformation and temperature elevation at specific locations on the structure have been developed [4-7].

To regulate the temperature field within the structure, Sata et al. [8, 9] employed a finite element model to furnish feedback information to a control system. This approach employed a coarsely idealized mesh to expedite computations and control cycles, albeit at the cost of accuracy. Moriwaki [10], on the other hand, employed a finite difference thermal model to estimate the axial thermal displacement of a hydrostatically supported precision spindle. The predictions were subsequently utilized to trigger a feedforward open-loop controller. In order to achieve control cycles of the order of 10 seconds, this method assumed two-dimensionality in the thermal deformation process and restricted the analysis to the spindle and its housing exclusively. It becomes apparent that numerical methods, in their current forms, oscillate between being overly intricate and slow or overly simplistic and imprecise for effective feedback control applications. Furthermore, these methods relied on off-line calibration during idle running conditions to ascertain heat input to the structure, without real-time adjustments during actual machining operations.

The focal point of this research entails an in-depth exploration of errors inherent in machine tools, which can manifest in various forms, including but not limited to motion-related, geometric, dynamic, and thermal aberrations, all of which impact both machine performance and the quality of the final manufactured product. This study, in particular, centers on the investigation of temperature-induced errors in machine tools.

The investigation leverages the Finite Element Method (FEM) for precise analysis and employs 3D simulations as its primary methodology. The research focuses on the cutting tool insert of a lathe machine, utilizing a variety of materials with their respective parameters to achieve the targeted research outcomes. A detailed 3D model of the tool insert is meticulously crafted using a laptop computer (ASUS, Taiwan) running NX Siemens software. The selected insert belongs to the SNMG type and is securely fitted within its holder.

The study delves into the intricacies of temperature propagation within the machine tool, drawing upon fundamental principles such as Fourier's law for conduction and Newton's law of cooling for convection. Material properties and characteristics are meticulously referenced from authoritative sources and guides.

A comprehensive exploration of the Finite Element Method is undertaken, and its principles are exhaustively examined throughout the simulation process. Mathematical and numerical computations are conducted within a one-dimensional quadratic element framework, specifically tailored to the insert of the turning machine.

Finite Element Method (FEM) simulations are carried out for both thermal and structural analyses, particularly in the context of steady-state conditions. Siemens NX12 Nastran software is employed for this purpose, yielding invaluable insights into temperature increases, thermal gradients, and the extremal values of temperature within the system.

The resulting data offer a comprehensive understanding of how temperature influences the behavior of the insert during steady-state heat transfer operations. This knowledge serves as a foundation for devising strategies and methods aimed at enhancing the overall performance and operational efficiency of machine tools.

1. Finite element analysis of heat transition

Finite Element Analysis (FEM) constitutes a numerical methodology employed to address problems associated with differential or integral equations. This computational approach involves the transformation of such equations in various ways, either by their complete removal or their conversion into algebraic equations. Additionally, partial differential equations (PDEs) can be reformulated to approximate ordinary differential equations (ODEs). Consequently, this facilitates the utilization of well-established numerical methods to derive solutions.

Finite elements, a fundamental component of FEM, can be categorized into distinct concepts, one of which involves triangular or rectangular elements.

- Nodes: Nodes are endpoints, vertices or specific points of an element. Physical changes in elements are represented by nodes. For example, in the elastic problem, the quality of linear elements with two nodes is obtained at their two nodes. Forces are also applied to these nodes. Deformation is represented as nodal displacement.
- Degrees of Freedom: A node's degrees of freedom is the number of changes within the node. For example, if the translation method is used to solve a structural problem, the nodal degrees of freedom are 3. This indicates translation in his three coordinate directions at one node.

Another example is a thermal analysis with 1 nodal degree of freedom. This will give you the temperature value for a particular node.

Mesh: A mesh is a network of elements made up of nodes used by multiple elements and is used to represent a problem domain waiting to be solved. Due to the complex spindle boundary conditions and geometry used in this experiment and the multiple types of heat sources, the modeling time was too long, and the model was inaccurate. Therefore, the model can only predict part of the error.

Therefore, finite elements are used in design to predict the thermal deformation tendency or dynamic analysis of thermal behavior and indicate the improvement direction of thermal deformation error [11].

Heat generation results of different material inserts

Here, consideration is on simulating different materials for current chosen insert. Fig. 1 presents insert parameters. As, the work intention is to find out heat generation on the lathe machine insert and its effect to the machining surface, initially work starts with choosing different materials and simulated them to find out their heat effects.

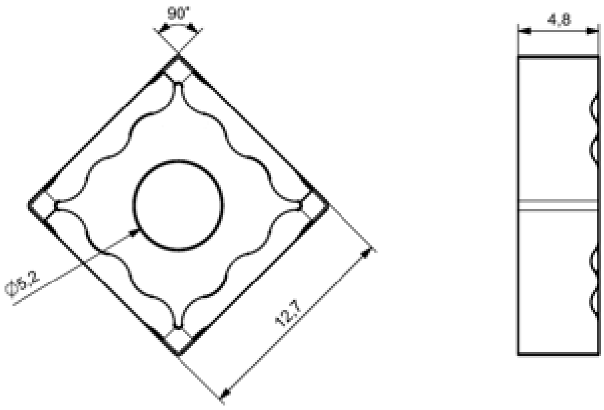


Figure 1. Drawing of lathe machine tool insert [12].

The simulation starts with following materials: molybdenum, cobalt, vanadium, tungsten, carbon, chromium, titanium, carbide. As they have different heat transfer coefficient, the following table presents different values of materials.

Table 1. Different materials heat transfer coefficients [10].

	Material	Heat transfer coefficient $k\left[\frac{W}{m^2K}\right]$
1	Molybdenum	138.00
2	Cobalt	69.21
3	Vanadium	31.00
4	Tungsten	164.00
5	Carbon	0.01663
6	Titanium alloy	7.50
7	Carbide	110

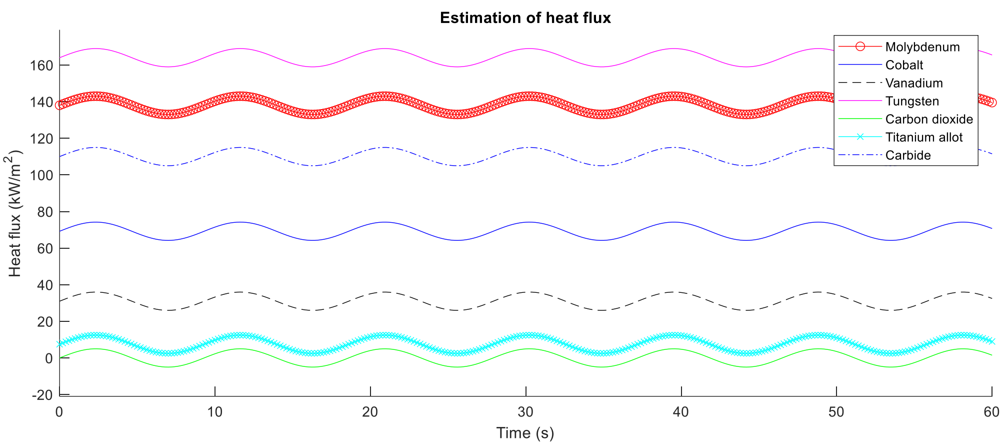


Figure 2. Simulation of heat flux equation using different materials for the same insert. Simulation was done under the following parameters: final time t = 60, time step ∆t=0.2

FEM software

For the simulations to measure the heat distribution, we used a laptop computer (ASUS, Taiwan) with a 2.6 GHz Intel I CITM i7-10750H CPU, 8.0 GB RAM with NX Nastran software (Siemens Digital Industries Software, MSC Software, NEi Software, Munich, Germany).

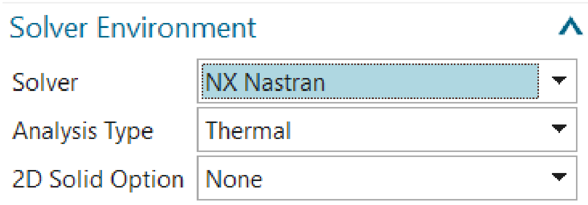


Figure 3. Solve Environment of NX.

Assigning or choosing materials

As our work is to find out thermal property and its effect to the surface, we considered to insert thermal properties of different materials on the software that is shown in Fig. 4. All the properties and data are taken from above mentioned tables.

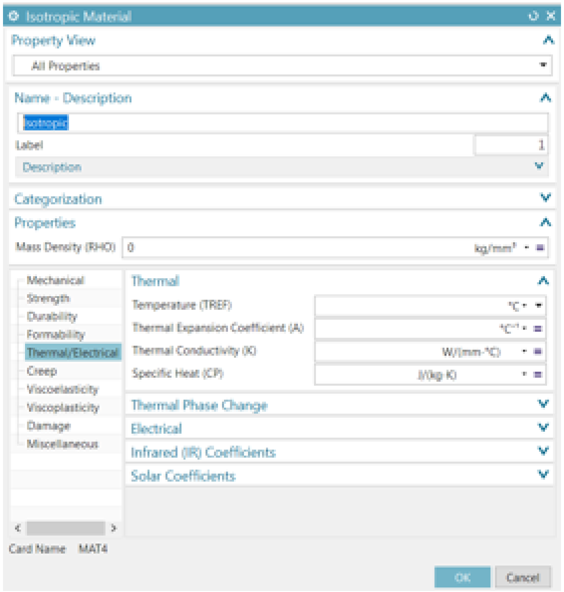


Figure 4. Assigning material panel from NX Nastran.

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. Within the structured and unstructured mesh classifications, the methods may further be subdivided into uniform and non-uniform categories (see Figure 5).

The mesh topology used and performed by NX Nastran for this experiment is non-uniform structured mesh.

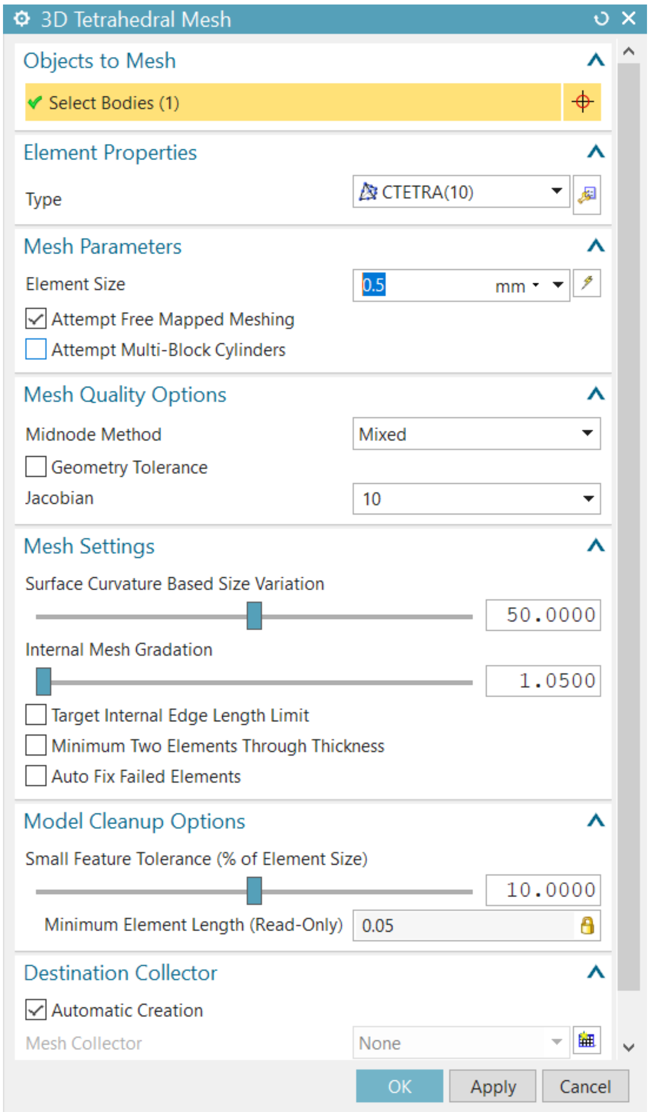
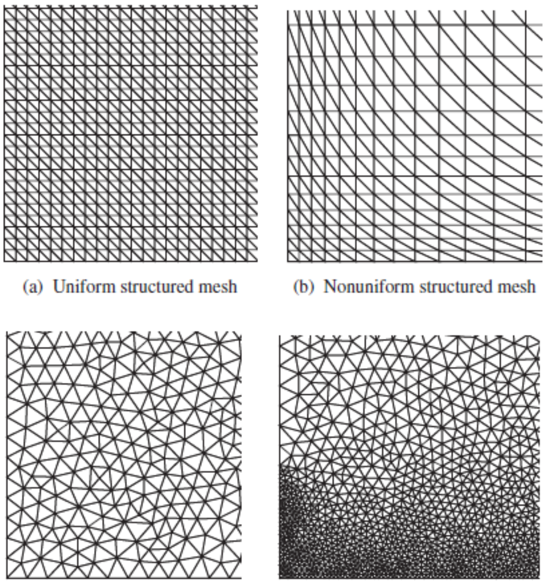


Figure 5. Tetrahedral mesh and CTETRA (10) type.



(a) Uniform structured mesh (b) Nonuniform structured mesh

(c) Uniform unstructured mesh (d) Nonuniformun structured mesh

Figure 6. Types of mesh. Topology categories [13].

1.1. Heat source

In order to initiate the simulation, there has been entered thermal constrains in the hole of the insert that is defined as ambient temperature $T_{amb} = 20$ °C degree. It is shown in the Fig. 7 right side of the insert. The heat flux parameter is taken from the table 3 and shown in the |Fig. 7 on the left side.

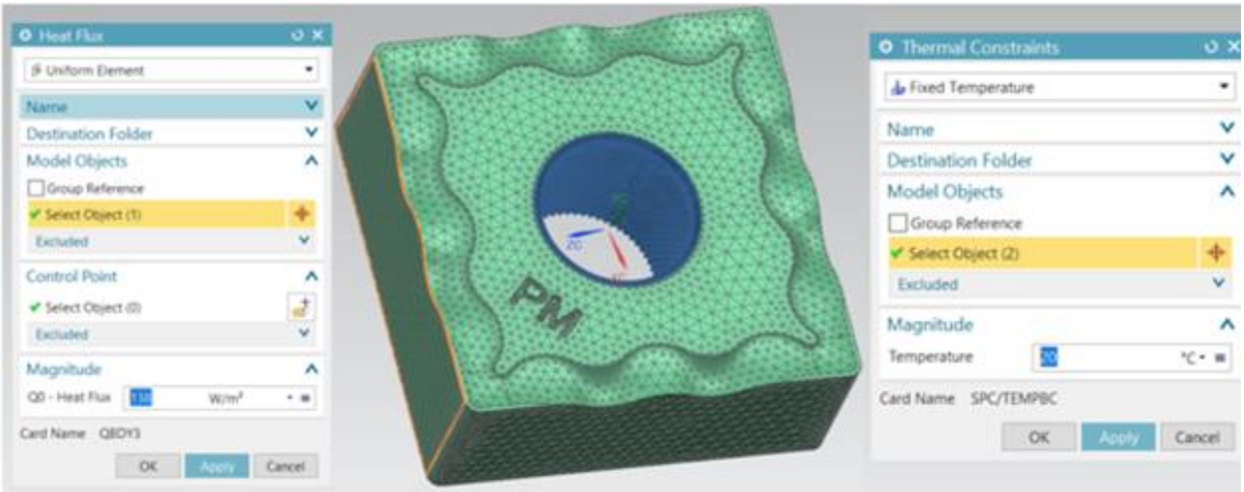


Figure 7. Load and boundary conditions.

1.2. Thermal simulation and results.

In Fig 8 shows the post processing navigation tab. In order to get the result all processes that mentioned above should be followed and can be navigated and checked here. After setting all the required parameters, the simulation initiated, and the result appears in the end of the navigation tab. Using this results tab, it is possible to see the process or simulation final result.

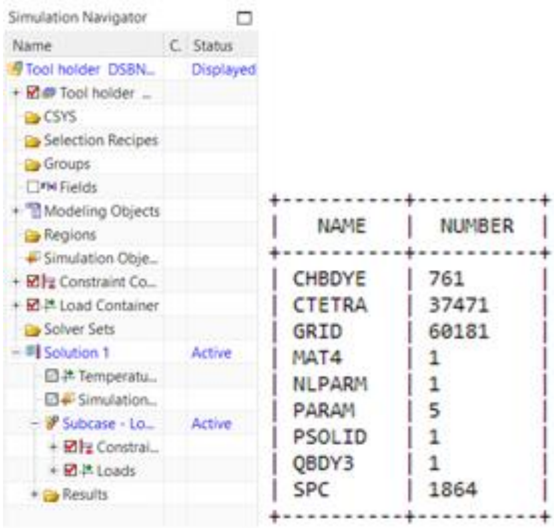


Figure 8. NX Nastran navigation tab and thermal results nodes.

As an insert with various materials is checking or simulating, in below each material simulation results are presented.

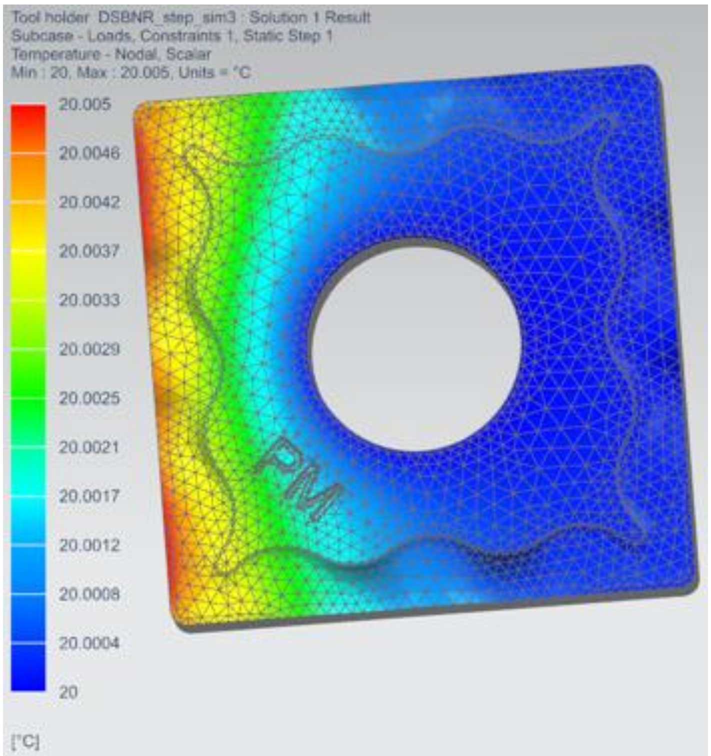


Figure 9. Thermal simulation result of molybdenum inserts

The Fig. 9 presents thermal simulation result of molybdenum insert result that the maximum heat temperature t is $Max = 20.005$.

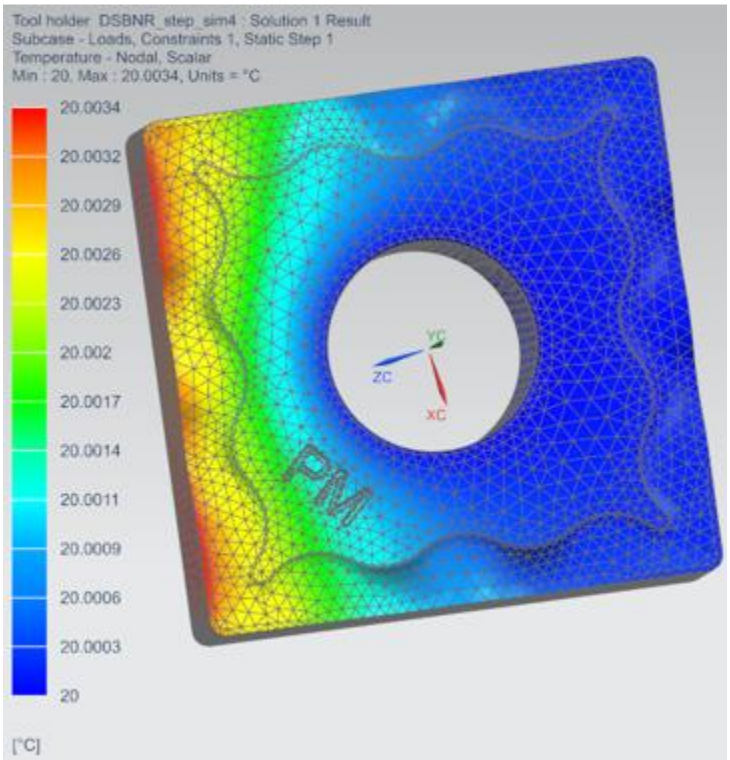


Figure 10. Thermal simulation result of cobalt insert

The Fig. 10 presents thermal simulation result of cobalt insert result that the maximum heat temperature t is $Max = 20.0034$.

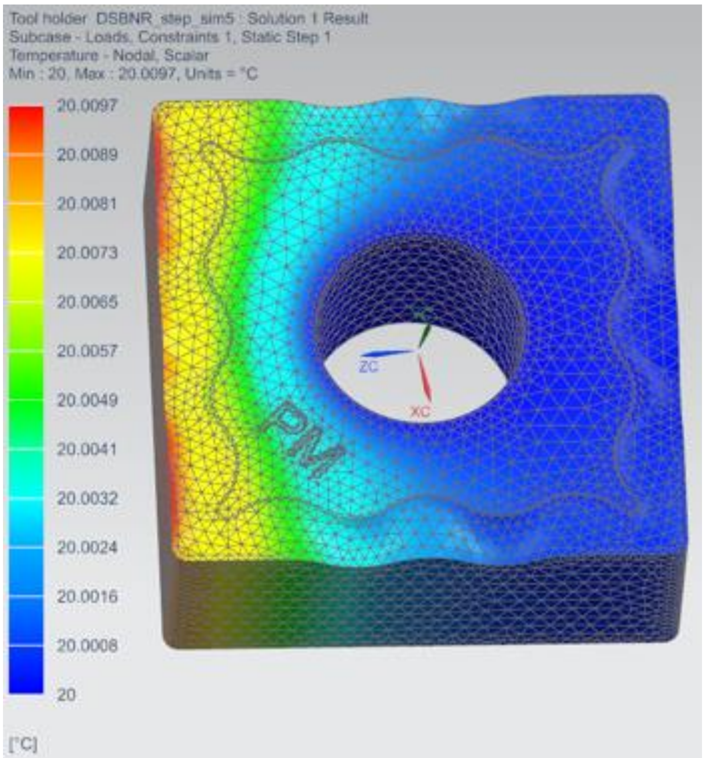


Figure 11. Thermal simulation result of vanadium insert

The Fig. 11 presents thermal simulation result of vanadium insert result that the maximum heat temperature t is $Max = 20.0097$.

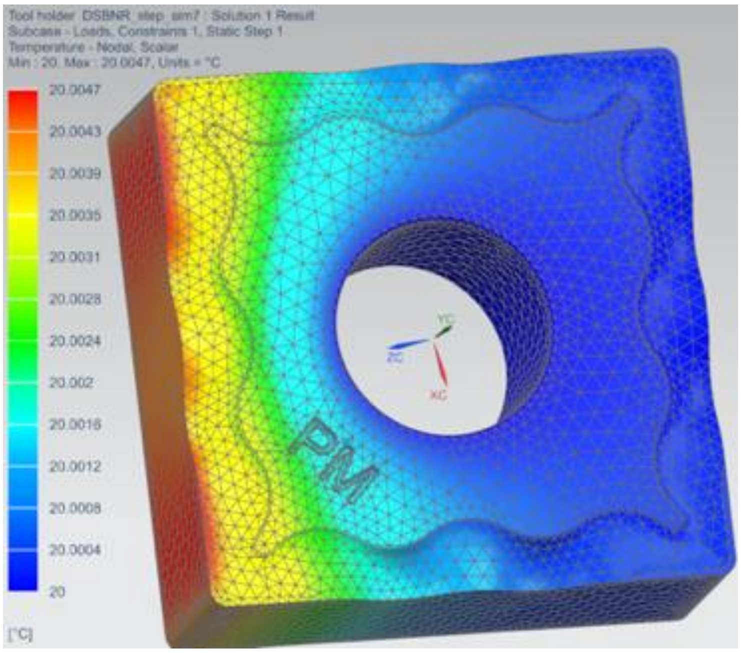


Figure 12. Thermal simulation result of tungsten insert

The Fig. 12 presents thermal simulation result of molybdenum insert result that the maximum temperature t is $Max = 20.0047$.

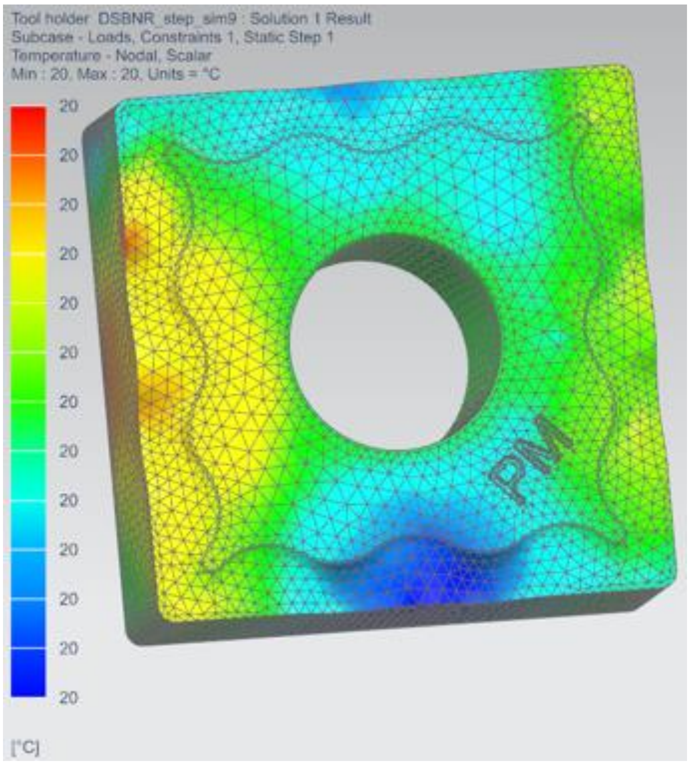


Figure 13. Thermal simulation result of carbon insert

Clarivate Analytics indexed

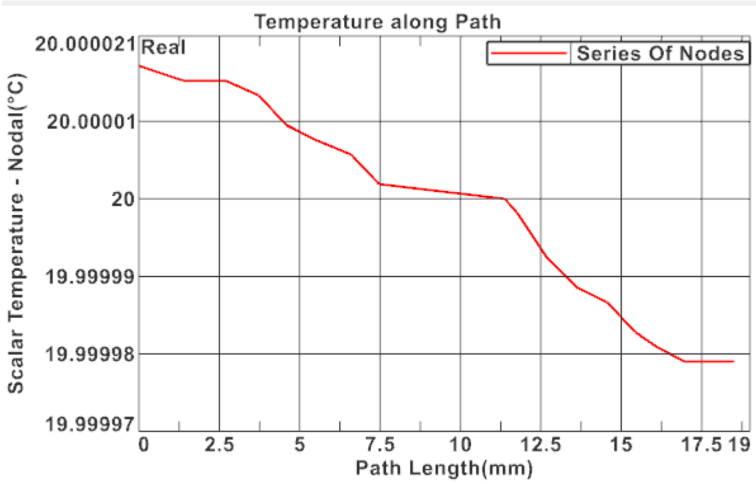


Figure 14. The graph of scaled temperature in different length

The Fig.13 and Fig 14 presents thermal simulation result of carbon insert result that the maximum temperature is $Max = 20.000021$. Here, we presented scalar temperature results due to very small heat effect on the surface [14], [15].

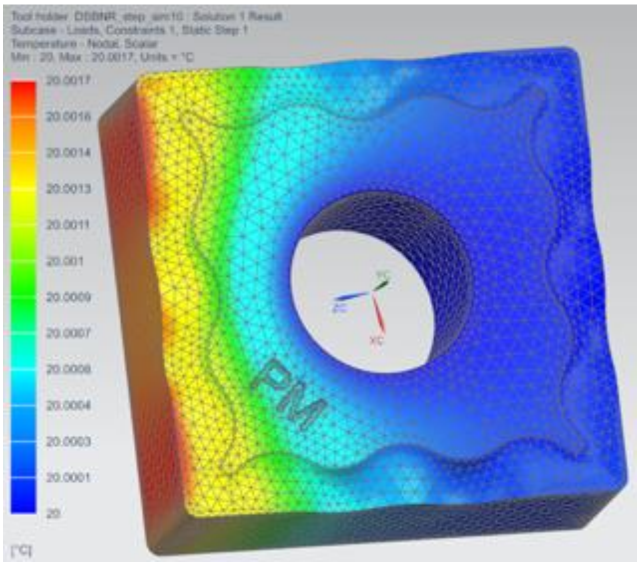


Figure 15. Thermal simulation result of titanium insert

The Fig. 15 presents thermal simulation result of titanium insert result that the maximum heat temperature t is $Max = 20.0017$. We have simulated 6 various material and have found that some materials heat transfers are almost the same like molybdenum and vanadium.

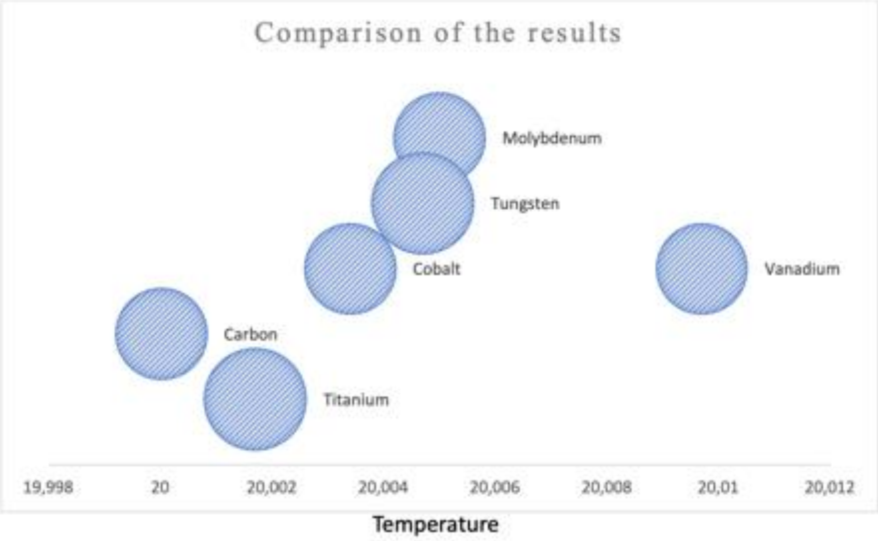


Figure 16. Comparison of different materials thermal effect results

Conclusions.

The lathe machine tool insert was analyzed by FEM simulation by dividing it into small 3D tetrahedra with 10 nodes each. The tetrahedron sizes were defined to improve the interaction time in the simulation. An ambient temperature of 20 °C was specified for the lathe simulation analysis. This was chosen in the center of the insert because this part is fixed to the holder using fixing elements that effects to the temperature of the insert.

The insert of turning machine tool was analyzed by FEM simulations dividing into small 3D tetrahedrons with four nodes each one of them. The simulation results showed that there are various parameters that effect to the materials temperature. However, when there was small heat transfer property and lower density of material, the temperature also was lower than others.

The maximum temperature transfer was estimated in steady-state form in the vanadium insert $t_{max} = 20.0097$. In contrast the minimum temperature transfer was in the carbon with value of $t_{min} = 20.000021$. The other materials like: titanium, cobalt, tungsten and molybdenum showed a little higher temperature effect in a row, respectively.

More precise and detailed result on temperature effect on the tool can be obtained when assembled lathe machine tool is simulated. Due to the size of insert there was very small temperature effect that showed disadvantage of our simulation. However, we found out that temperature effect was less than others when carbon and titanium was used. The Fig 16 presented crucial part of my findings and simulation results. This information can be used and implemented for further analysis and FEM systems.

Assemble of the lathe machine tool can be analyzed by FEM, if it is needed to focus on how the thermal effect occurs in the whole tool itself. The accuracy in the results depends on the correct detection of heat sources, the correct sizing of the mesh in simulation and the material properties of the components of the machine tool.

References:

1. Chiappulini, R., Giannotti, L., & Galbersanini (1991). *"On-Line Correction via Software of Thermal Errors in Numerically Controlled Machine Tools,"* Scientific Committee M of 41th CIRP General Assembly, August, 1991.
2. Attia, M. H. (1990). *'Modelling of Thermal Deformation of Machine Tool Structures: Design and Control Issues,"* Proc. Int. Symposium on Manufacturing and Materials Processing, Int. Centre of Heat and Mass Transfer, Dubrovnik, Yugoslavia, August, 1990.
3. Tonshoff, H. K., & Wulfsberg, J. P. (1989). *"Developments in Diagnosis of Thermal Induced Displacements in Machine Tools,"* Symposium

on Grinding Fundamentals and Applications, ASME-WAM, San Francisco, Dec. 1989, pp. 281-295.

4. Spur, G., & Heisel, U. (1977). *"Automatic Compensation of Thermal Disturbances in Machine Tools,"* Proceedings of the 3rd Int. Conf. on Production Engineering, Kyoto.
5. Jedrzejewski, J. (1985). "Kompansation thermischer Verlagerungen einer Drehmaschine," *Werkstatt und Betrieb*, Vol. 118, No. 2, pp. 85-87.
6. Ichimiya, R. (1980). *"Compensation for Thermal Deformation of Milling Machine,"* Proc. Conf. Numerical Methods in Machine Tool Design, Wroclaw, Poland, pp. 39-49.
7. Week, M., Schuze, O., Michels, F., & Bonse, R. (1994). *"Optimization of Machine Tools Performance and Accuracy,"* ASME Symposium on Intelligent Machine Tool Systems, M. H. Attia and M. Elbestawi, eds,, Int Mech Engrg. Cong. Expo., November, 1994, pp. 895-908.
8. Hatamura, Y., Nagao, T., Mitsuishi, M., Kato, K., Taguchi, S., Okumura, T., Nakagawa, G., & Sugishita, H. (1993). "Development of an Intelligent Machining Center Incorporating Active Compensation for Thermal Distortion," *Annals of the CIRP*, Vol. 42, No. 1, pp. 549-552.
9. Ni, J., & Wu, S. M. (1993). "An On-line Measurement Technique for Machine Volumetric Error Compensation," *Trans ASME JOURNAL OF ENGINEERING FOR INDUSTRY*, Vol. 1993, Feb., 1993, pp. 85-92.
10. Sata, T., Takeuchi, Y., Sato, N., & Okubo, N. (1973). *"Analysis of Thermal Deformation of Machine Tool Structure and its Application,"* Proc. 14th Int. MTDR Conf, 1973, pp. 275-280.
11. Ramesh, R., et al. (2000). Error Compensation in Machine Tools - a Review: Part II: Thermal Errors, *Int. J. Mach. Tool Manu.*, 40 (2000), 9, pp. 1257-1284.
12. (n.d.). Retrieved from https://www.summaryplanet.com/engineering/lathe-cutting-tools.html
13. Nithiarasu, R., Lewis, W., & Seetharamu, K. N. (2016). *Fundamentals of the finite element method for heat and mass transfer*, Chichester U.K.: John Wiley & Sons.
14. Sayfidinov, O., Klazly, M., & Bognár, G. (2023, September). *The impact of noise terms on solutions of the Kardar-Parisi-Zhang equation*. In AIP Conference Proceedings (Vol. 2849, No. 1). AIP Publishing.
15. Klazly, M., Sayfidinov, O., Hriczó, K., & Bognár, G. (2023, September). *Heat transfer enhancement for laminar nanofluids flow: A numerical study using two phases.* In AIP Conference Proceedings (Vol. 2849, No. 1). AIP Publishing.