



MARMARA UNIVERSITY
INSTITUTE FOR GRADUATE STUDIES



SYSTEM IDENTIFICATION OF CNC MACHINE TOOL AND DAMPING OF VIBRATIONS

ENES DEMİR

MASTER THESIS

Department of Mechanical Engineering

Thesis Supervisor

Prof. Dr. Bülent EKİCİ

ISTANBUL, 2023

ACKNOWLEDGEMENT

First of all, I would like to thank my family for supporting me at all times.

I would like to express my deep gratitude to my supervisor Prof. Dr. Bülent Ekici whose guidance and patience made this thesis possible.

I would also like to thank my company Milteksan CNC Teknoloji ve Kontrol Sistemleri San. A.Ş. and General Manager Prof. Dr. Onur Tunçer for creating a fruitful research and development environment. I would like to thank Usel Makine for the machine tool they manufactured. I would like to express my gratitude for the help I received from my colleagues Ahmet İhsan and Beyza Nur during the experiments.

Furthermore, I would like to thank TUBITAK for their support in the project “3210224 – Beş Eksen Yeni Nesil CNC Kontrol Sistemi Ve Dijital İkiz Platformu Geliştirilmesi”.

Also, I would like to thank Prof. Dr. Hasan Erdal who is the dean of Marmara University Faculty of Technology for his support in providing laboratory.

I would like to thank Dr. Selim Hartomacıoğlu for his support in use of a workstation.

Finally, many thanks go to all my friends from this program specially Gurur Furkan Sarılmaz and Nural Yılmaz for their friendship and motivations during my thesis study.

Enes DEMİR

Istanbul, September 2023

CONTENTS

ÖZET	iii
ABSTRACT.....	iv
SYMBOLS.....	v
ABBREVIATIONS.....	vii
FIGURES	viii
LIST OF TABLES	x
1. INTRODUCTION	11
1.1. Vibration Suppression	11
1.2. Modelling of Dynamic System	14
2. MATERIALS AND METHODS.....	19
2.1. Three Axis Machine Tool	19
2.2. Finite Element Analysis of Ball Screw Mechanism	23
2.2.1. Analytical Hybrid Finite Element Model	23
2.2.2. Finite Element Analysis in Commercial Analysis Software.....	27
2.3. Frequency Domain Analysis of The Machine Tool's Structure	30
2.3.1. Frequency Response Analysis in Commercial Analysis Tool.....	30
2.3.2. Power Spectral Analysis with Impulse Tests.....	37
3. RESULTS AND DISCUSSIONS.....	42

3.1. Ball Screw FEA Results of Analytical Hybrid FE Model	42
3.2. Ball Screw FEA Results of Commercial Analysis Software	44
3.3. Frequency Response Analysis of Machine Tool in Commercial Tool	46
3.4. Power Spectrum Analysis of Machine Tool's with Vibration Tests.....	49
3.4.1. Positioning (air cut) test.....	49
3.4.2. Impulse torque tests	51
REFERENCES	62
APPENDIX.....	65
A. Ball Screw Mechanism Finite Element Analysis Codes in MATLAB	65
B. APDL Codes	72
1. COMBIN 14 Torsional Spring for Coupling.....	72
2. MPC184 Screw Element for Ball Screw – Nut Interface	72
C. Mode Shapes of Ball Screw Mechanism by MATLAB.....	73
D. Mode Shapes of Ball Screw Mechanism by ANSYS	75
E. Mode Shapes of Machine Tool Frame by ANSYS	77

ÖZET

CNC TEZGAHI SİSTEM TANIMLAMASI VE TİTREŞİMLERİN SÖNÜMLENMESİ

Günümüzde üretimin büyük bölümünü oluşturan takım tezgahları, gelişmiş bilgisayarlarla daha hassas ve hızlı hale getirilmiştir. Takım tezgahlarındaki hassas işlemenin önündeki en büyük engellerden birisi de titreşimdir. Titreşim makine sağlığını da olumsuz etkilemektedir. Makine üzerinde rezonansa kolayca girebilen bir çok mekanik parça bulunmaktadır. Takım tezgahlarında sıkça kullanılan bilyalı vida mekanizmaları, yüksek hassasiyet, katılık ve düşük maliyete sahiptir. Bu mekanizmalarda özellikle, eksen narinliği arttığında doğal frekans düşmekte ve kolayca rezonansa girebilmektedir. Ayrıca makinede malzeme seçimi montaj stratejisi, yapısal bütünlük gibi konular makine gövdesinin titreşim özelliklerini belirler. Takım tezgahlarının titreşimlerinin analiz edilmesi makinenin sağlığı, hassas ve hızlı kesim açısından önemlidir. Tezgahın modal özellikleri bilindiği takdirde servo kontrolde bu titreşimler sönmülenebilmektedir. Ayrıca makinenin üretim öncesi tasarım sürecinde de fikir vermektedir. Buna göre tasarım tekrar iyileştirilebilmektedir. Bu çalışmada bir üç eksen köprü tipi freze takım tezgahının modal özellikleri tespit edilmiştir. Bu tezgah alüminyum parçalardan oluşmaktadır ve döküm parçaları olmayıp çoğunlukla montajdan oluşmaktadır. Bu tür tezgahlarda titreşim fazla görülmektedir. Bu tezgahın bilyalı vida mekanizması ve makinenin geri kalan gövde yapısı ayrı incelenmiştir. Bilyalı vida mekanizması ve vida somun arayüzünün modal analizi için analitik sonlu eleman modeli geliştirilmiştir. Bu sonuçlar ticari analiz yazılımı ile sadece bilyalı vida mekanizması için karşılaştırılmıştır. Makinenin gövdesi ise yine ticari analiz yazılımında modal ve harmonik cevap yönlerinden incelenmiştir. Bu analizleri kıyaslamak için tezgah üzerinde ivme sensörleri ile titreşim testleri yapılmıştır. Spektrum analizlerinden sonra ticari analiz yazılımı sonuçları karşılaştırılmıştır. Sonuç olarak bilyalı vida mekanizması için geliştirilen analitik model ile ticari analiz yazılımında özel bir eleman ile geliştirilen modelin analiz sonuçları büyük benzerlik göstermektedir. Bilyalı vida mekanizmasının yanı sıra, tezgah gövdesinin modal analizlerinde ise özellikle düşük doğal frekanslarda frekanslarda testler ve analizler uyuma göstermektedir. Bu testler yüksek frekans aralıklarında tatmin edici olmayıp testler iyileştirilecektir.

ABSTRACT

SYSTEM IDENTIFICATION OF CNC MACHINE TOOL AND DAMPING OF VIBRATIONS

Today, machine tools, which constitute a large part of production, have been made more precise and high speed with advanced computers. One of the most significant obstacles against precision machining on machine tools is vibration. Vibration also negatively affects machine health. There are many mechanical parts on the machine that can easily resonate. Ball screw mechanisms, which are frequently used in machine tools, have high precision, rigidity and are low cost. In these mechanisms, especially when the axis length is extended, the natural frequency decreases and can easily resonate. In addition, issues such as material selection, assembly strategy and structural integrity determine the vibration characteristics of the machine frame. Analyzing the vibrations of machine tools is crucial in terms of machine health, precision and high-speed cutting. If the modal characteristics of the machine tool are known, these vibrations can be damped in servo control. It also gives an idea in the pre-production design process of the machine. Accordingly, the design can be improved again. In this study, the modal characteristics of a three-axis bridge type milling machine tool are determined. This machine consists of aluminum parts and there are no casting parts and mostly consists of assembly. Vibration is a critical issue in this type of machine tools. The ball screw mechanism of this machine and the rest of the machine body structure are analyzed separately. An analytical finite element model was developed for the modal analysis of the ball screw mechanism and the screw nut interface. These modal analysis results are compared with commercial analysis software results only for the ball screw mechanism. The body of the machine was also analyzed with commercial analysis software in modal properties and harmonic response. In order to compare these analyses, vibration tests were performed on the machine with two acceleration sensors. After spectrum analyses, commercial software results were compared. As a result, the analysis results of the analytical model developed for the ball screw mechanism and the model developed with a special element in commercial software show great similarity. In addition to the ball screw mechanism, in the modal analyses of the machine body, the tests and analyses show agreement especially at low natural frequencies. These tests are not satisfactory in high frequency ranges and the tests will be improved.

SYMBOLS

Symbols	Detail	Unit
E	: Elastic modulus	Pa
G	: Shear modulus	Pa
J	: Polar moment of area	m^4
<i>l</i>	: Element length	m
ρ	: Density of the element	kg/m^3
p	: Screw pitch	m
r	: Transmission ratio of screw	m/rad
A	: Cross sectional area of the element	m^2
U1	: Deflection of the first node of the element	m
R1	: Rotation of the first node of the element	rad
U2	: Deflection of the second node of the element	m
R2	: Rotational of the second node of the element	rad
U_{screw}	: Deflection of the node on ball screw	m
R_{screw}	: Rotation of the node on ball screw	rad
U_{nut}	: Deflection of the node on ball nut	m
K_{el}	: Stiffness matrix of a finite element	
M_{el}	: Mass matrix of a finite element	
$K_{screw-nut}$	: Stiffness matrix of ball screw-nut interface	
K_{axial}	: Direct axial stiffness of joint along z axis	N/m
$K_{torsional}$	: Direct torsional stiffness of joint along Z axis	$N\ m/rad$
$K_{ax-tors}$	: Cross coupling stiffness due to relation of axial and torsional deflection in Z axis	$N\ m /m$
J_{eqv}	: Equivalent moment of inertia of the axis	$N\ m / \frac{rad}{s^2}$
J_{rotor}	: Moment of inertia of the rotor	$N\ m / \frac{rad}{s^2}$
$J_{ballscrew}$	: Moment of inertia of the ball screw	$N\ m / \frac{rad}{s^2}$

J_{table}	: Axis table moment of inertia	$N\ m / \frac{rad}{s^2}$
m_{table}	: Axis table mass	kg



ABBREVIATONS

CSP	: Cyclic Synchronous Position
FRF	: Frequency Response Function
PID	: Proportional Integral Derivative
FE	: Finite element
LS	: Least Square
FFT	: Fast Fourier Transform
MSUP	: Mode Superposition
PSD	: Power Spectral Density



FIGURES

Figure 1 Vibration reduction types [1].....	12
Figure 2 Different levels of continuity in feed profiles and their spectrums	12
Figure 3 Cascaded P- PI control with feedforward function [5]	13
Figure 4 Control algorithm structure with notch filters [7].....	14
Figure 5 Ball screw drive and screw nut interface.....	15
Figure 6 Dynamic system modelling strategies of machine tools [1]	15
Figure 7 The rigid body model proposed by Erkorkmaz and Altıntaş [10].....	16
Figure 8 Three axis milling test machine tool	19
Figure 9 Axes and motors of the machine tool	20
Figure 10 X axis ball screw feed drive	21
Figure 11 The system configuration	22
Figure 12 Beam element with translational and torsional freedoms	23
Figure 13 Overall model of ball screw feed drive	24
Figure 14 The CAD geometry of Y axis for modal analysis and constraints	27
Figure 15 Local coordinate frames on joint.....	29
Figure 16 Machine tool's entire CAD model.....	31
Figure 17 Linear guides in the machine	35
Figure 18 Modal analysis and harmonic analysis with MSUP	36
Figure 19 The measurement point on the Z table plate	37
Figure 20 ADXL 345 accelerometer with Raspberry Pi placed on the most flexible region	38
Figure 21 Piezo electric sensor placed on X and Y axes respectively.....	39
Figure 22 Data acquisition card.....	39
Figure 23 Mode shape of ball screw #1 – 118 Hz	42
Figure 24 Mode shape of ball screw #2 – 975 Hz	43
Figure 25 Mode shape of ball screw #3 – 2132 Hz	43
Figure 26 Mode shape of ball screw #4 – 2199 Hz	44
Figure 27 Mode shape of ball screw by ANSYS #1 – 238 Hz.....	45
Figure 28 Mode shape of ball screw by ANSYS #2 – 1019 Hz.....	45
Figure 29 Mode shape of ball screw by ANSYS #3 – 2152 Hz.....	45

Figure 30 Mode shapes analyzed in ANSYS.....	47
Figure 31 X axis frequency response of flexible point by ANSYS.....	48
Figure 32 Y axis frequency response of flexible point by ANSYS.....	49
Figure 33 Positions of the axes while air cutting	49
Figure 34 Velocities of the axes while air cutting	50
Figure 35 Motor torques of the axes while air cutting	50
Figure 36 Periodograms of single peaks in velocity and torque readings in positioning test	50
Figure 37 Applied torque impulses repeatedly on the axes.....	51
Figure 38 One of the impulses (zoomed)	52
Figure 39 Acceleration read by capacitive sensor (ADXL345).....	52
Figure 40 Acceleration read by piezo electric sensor (received from a different impulse test).....	53
Figure 41 PSD of acceleration in X axis read by capacitive sensor	54
Figure 42 PSD analysis of acceleration in Y axis read by capacitive sensor	54
Figure 43 Comparison of capacitive sensor and piezo sensor PSDs in X axis	55
Figure 44 Comparison of capacitive sensor and piezo sensor PSDs in Y axis	55
Figure 45 PSDs of input torques	56
Figure 46 Comparison of ANSYS FRF and PSD analyzes of two sensor outputs in X axis	57
Figure 47 Comparison of ANSYS FRF and PSD analyzes of two sensor outputs in Y axis	59
Figure 48 Mode shape of ball screw #5 – 3374 Hz	73
Figure 49 Mode shape of ball screw #6 – 4160 Hz	73
Figure 50 Mode shape of ball screw #7 – 4637 Hz	74
Figure 51 Mode shape of ball screw #8 – 5547 Hz	74
Figure 52 Mode shape of ball screw #9 – 5893 Hz	75
Figure 53 Mode shape of ball screw by ANSYS screw #4 – 3356 Hz.....	75
Figure 54 Mode shape of ball screw by ANSYS #5 – 4352 Hz.....	76
Figure 55 Mode shape of ball screw by ANSYS #6 – 4581 Hz.....	76
Figure 56 More mode shapes of machine frame by ANSYS	77

LIST OF TABLES

<i>Table 1 Mechanical parameters of the rotor</i>	<i>24</i>
<i>Table 2 Mechanical parameters of the ball screw.....</i>	<i>25</i>
<i>Table 3 Motor weights</i>	<i>31</i>
<i>Table 4 Some of the critical material assignments</i>	<i>32</i>
<i>Table 5 Properties of some materials</i>	<i>32</i>
<i>Table 6 Y axis mass properties in CAD model</i>	<i>33</i>
<i>Table 7 Y axis mechanical parameters identified with LS method</i>	<i>33</i>
<i>Table 8 Comparison of natural frequencies of analytical model and commercial software model.....</i>	<i>46</i>
<i>Table 9 Comparison of natural frequencies in X axis</i>	<i>58</i>
Table 10 Matching natural frequencies of Y axis by PSD and ANSYS	60

1. INTRODUCTION

Starting with the universal and NC machine tools, the workpieces could not be machined in series and with high precision. These types of machining centers were not well automatized and need a lot of human interference. With the increasing demand of higher mass production, more automatized, higher precision and more complex workpiece machining, the machining centers became well equipped with advanced hardware and software that forms CNC machine tools. Thanks to CNC, more advanced functions are adapted in order to satisfy these demands. With the help of high-speed communication protocols like, EtherCAT, the cutter tool trajectory interpolation time can go below 1 ms. Also, CNC machine tools can handle all kinematic transformations for various types of machines. The best example of this is 5 axis CNC machine tool since it can machine natural forms like a turbine blade.

Machine tools are still being developed for more precise machining and longevity of machine. Machine tool's natural frequencies can be triggered by many external factors like, chattering resulting from spindle and workpiece resonance, high bandwidth tool trajectory, frictions, disruptions in surfaces and moving parts. All of these factors affect the machine tool's natural frequencies. Resonance of the machine causes poor quality machined surfaces, noisy machining and malfunctioning of the machine structure. In order to provide smooth machining and increase the life of machine these natural frequencies must be analyzed and damped.

1.1. Vibration Suppression

Hosseiniabadi [1] proposed a tree showing vibration reduction methods in his thesis. The control algorithm designer can either avoid the natural frequencies or suppress them intentionally.

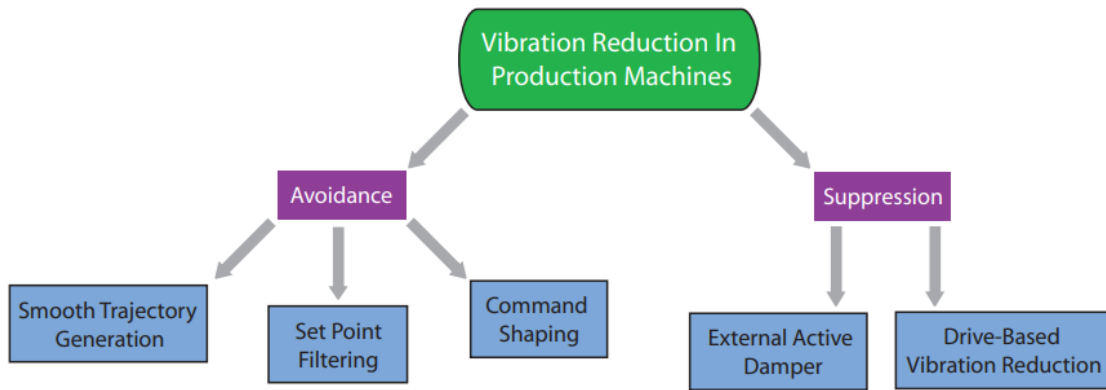


Figure 1 Vibration reduction types [1]

The interpolated tool trajectory is the most crucial factor in smooth operation. In order to create smooth, continuous and trajectory with less frequency content, various algorithms are proposed. Erkorkmaz and Altintas [2] generated trapezoidal acceleration shaped feed profiles for smooth trajectory generation (Figure 1). However, the degree of continuity can be increased as in Figure 1 [3]. According to these profiles, vibration can be avoided by increasing continuity level of feed profiles on trajectory paths. However, this causes increasing computational load. Continuous tool trajectories can also be generated using FIR filters for their real time capability for trajectory generations [4].

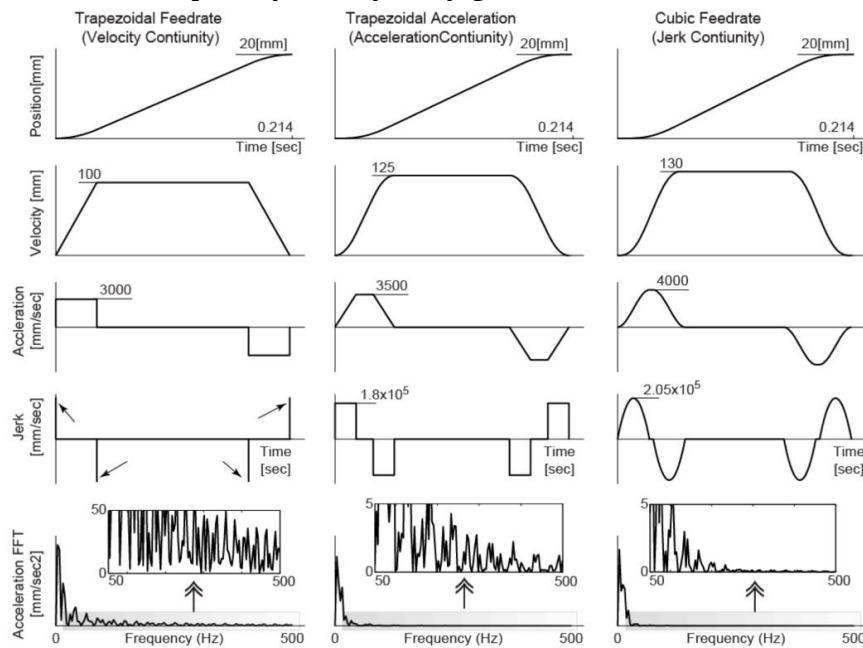


Figure 2 Different levels of continuity in feed profiles and their spectrums

The CNC consist of precise machining functions like feedforward control, friction compensation, notch filtering etc. in servo control layer. In servo control, the drivers usually use cascaded P-PI controller to calculate control torque of motors for perfect tracking of target position desired by CNC unit. Some features were added to this control algorithm for precise machining and vibration damping. An example of control loop is like Figure 3.

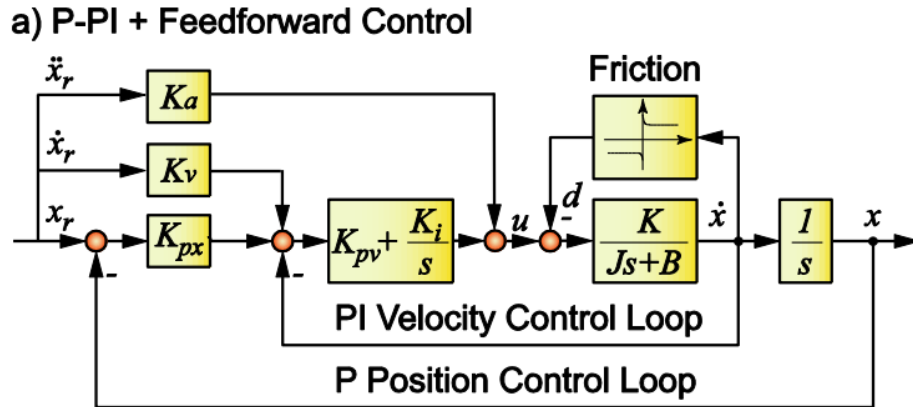


Figure 3 Cascaded P- PI control with feedforward function [5]

Tsai et al. [6] performed an input shaping (set point filtering in Figure 1) on target position to prevent vibrations. They consider the mode of the machine bed, longitudinal and torsional modes. Therefore, the input target trajectory does not contain any natural frequency content within its spectrum not to trigger any resonance of the machine.

Some researcher follow a different path other than input shaping. They shape the internal command in control loop [7] [8]. Liu et al. [7] placed consecutive notch filters. The torque command is internally calculated by the servo driver after the velocity control and the final torque command should not excite the machine natural frequencies, so the final torque command is shaped via notch filters. Their proposed control loop diagram is shown in Figure 4.

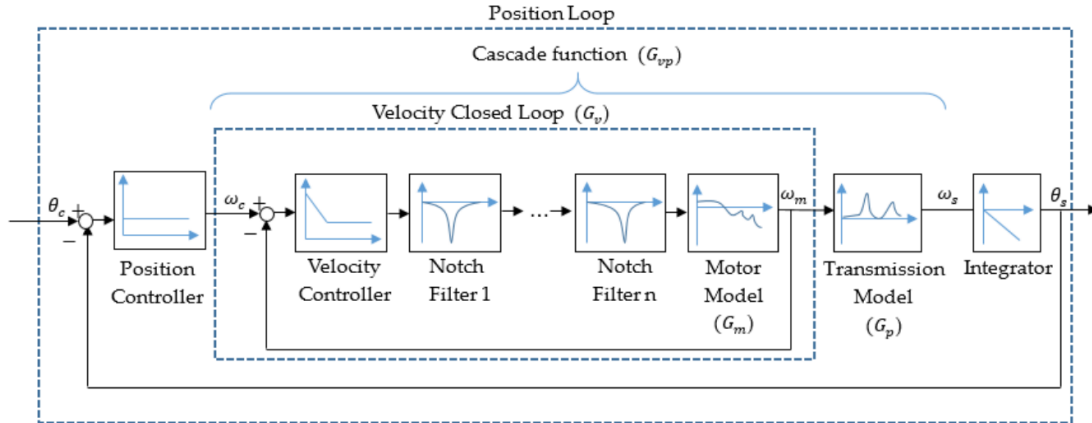


Figure 4 Control algorithm structure with notch filters [7]

1.2. Modelling of Dynamic System

The control algorithm functions that are mentioned above help precise machining. However, control system designer should have prior knowledge about the system dynamics and mechanical parameters before designing the control system for perfect positioning of axes. In addition to this, even the mechanical designer should analyze the structure before manufacturing of the machine tool, so he can fix the design. In this way, the machine can operate long term and properly. Also, the positioning accuracy gets higher.

In the machine tools, the linear axes are generally driven by a rotary motor and a screw type shaft (Lead screw or ball screw) that transmits the rotary motion to linear motion. The reason of using rotary motor rather than using linear servo motor is that rotary motor is more affordable, and, together with leadscrew or ball screw, provides less tolerance to variation in mass [9]. The linear axes can be driven by leadscrew or ball screw with the help of rotary motors. Nowadays the ball screws are widely used in machine tools since they carry heavier loads, do not self-lock and are more efficient. However, ball screw is less affordable than lead screw. The ball screw drive is significant in dynamic system modelling and vibration analysis because it is generally the most flexible part due to its length to radius ratio and load transmitting mechanism in machine tools. Ball screw mechanism in machine tools must be carefully designed to avoid vibrations.

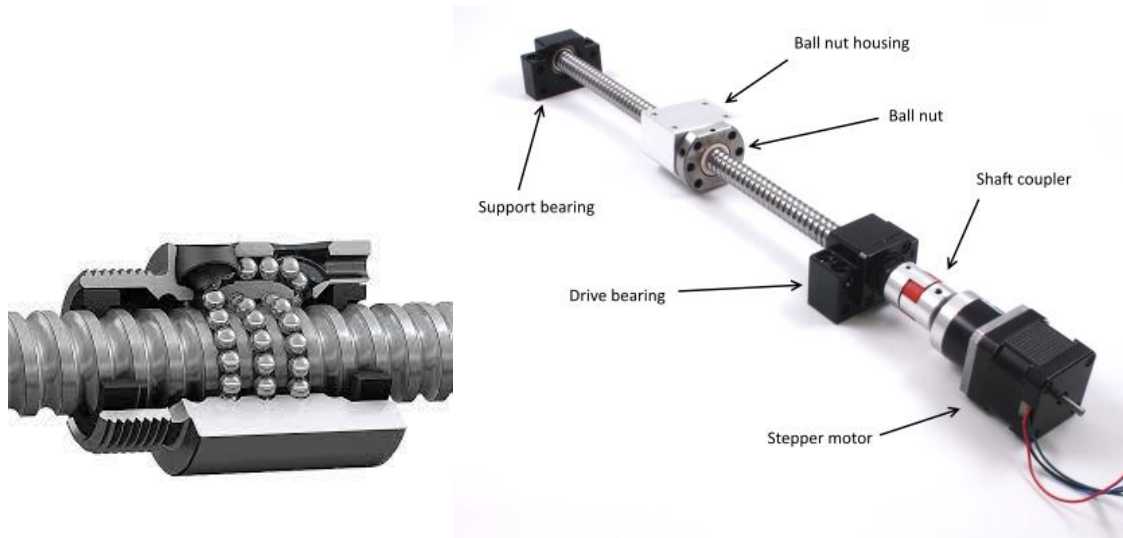


Figure 5 Ball screw drive and screw nut interface

There are several modelling strategies to analyze the dynamics of the machine tools. The simplest and low computation load method is rigid body [1]. In rigid body modelling, the structure itself is considered as non-vibrating and has no flexibilities. Modelling fidelity can be enhanced with full finite element modelling [1]. While rigid body modelling is more beneficial in simulation, the finite element modelling is more beneficial in structural analyzes such as modal analysis. There are various dynamic system modelling methods in literature. Their summary is shown in Figure 6.

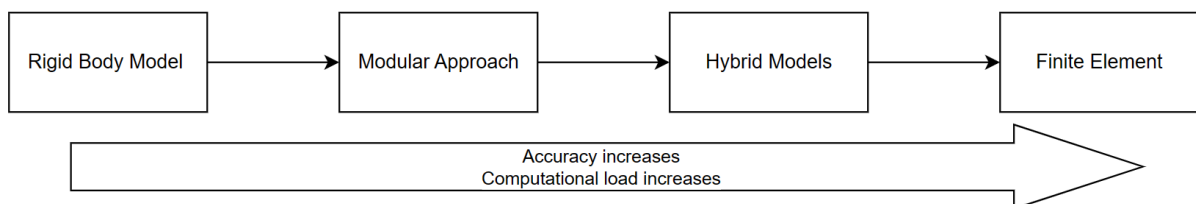


Figure 6 Dynamic system modelling strategies of machine tools [1]

Erkorkmaz and Altıntaş [10] proposed a feed drive dynamic model which is a rigid body model. This model consists of only equivalent moment of inertia, viscous damping and Coulomb friction of each axis and they are identified by least square method. They analyze the model parameters with the help of a special torque input and corresponding velocity output data. This analysis neglects the flexibilities of the system. This analysis can help simulate the system in

time domain. Their [10] proposed model is below.

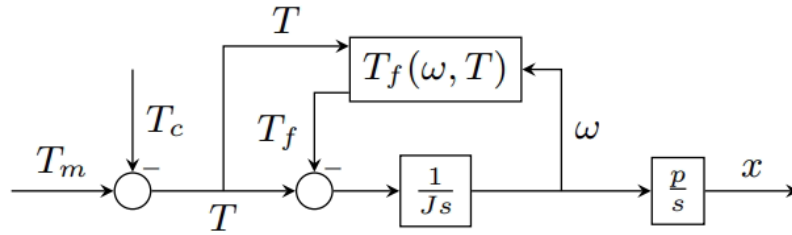


Figure 7 The rigid body model proposed by Erkorkmaz and Altıntaş [10]

The model is nonlinear because of the Coulomb friction (T_f block in Figure 7). The Coulomb friction model is built on the Stribeck curve model [11].

In order to include flexibilities and increase accuracy of the model, some researchers increased the model complexity. Varanasi and Nayfeh [12] developed a feed drive model, which is based on distributed inertia and compliance of ball screw and thrust bearings and coupling. Then, they reduced the distributed parameter model to a low order model using Galerkin procedure. They verified the model by experiments performed on a ball screw system. Pislariu et al. [13] also created a hybrid model with distributed parameter modeling of ballscrew and lumped parameter modeling of other components. They implemented this model in Simulink with block representation. They explained the reasons of difference and similarities between simulations and tests (performed with sinusoids). Okwudire and Altintas [14] rather than considering whole system as lumped mass, they split the structure into finite elements. They modeled the ball screw by Timoshenko beam finite elements. The balls inside the ball screw-nut interface and the joints, fasteners, bearings are modelled as simple springs, the rigid components are modelled as lumped masses. In this method, they examined the coupling between lateral, torsional and axial modes in ball screw-nut and created a stiffness matrix with coupled terms between ball screw and nut. In this way they could analyze the frequency response of the drives.

Full finite element analysis can be performed for the most accurate analyzes like modal analysis, frequency response analysis. However, dividing an assembly into many finite elements increases computation load. Therefore, the real time simulations are too challenging

in FEA. This strategy is more useful at design process and analyzing the structural dynamics offline. Hong et al. [15] analyzed the ball screw drive in ANSYS structural analysis, modal analysis and harmonic analysis for its deflections, natural frequencies mode shapes and frequency response functions. Zhou and Li [16] analyzed the axial stiffness of ball screw drive by considering the shearing on balls in nut by ANSYS. The axial stiffness is important for the measurement of axial natural frequency of the drive. Erkorkmaz and Kamalzadeh [8] desired to improve the positioning accuracy. Therefore, they compensated the torsional vibration of ball screw drive with notch filter and active cancellation techniques. They observed improvement in stability margins and bandwidth. In order to design control algorithm, they needed to identify the system dynamics. With this purpose, they performed analyzes and tests on the drive. Firstly, they identified the rigid body model with Least Square method [10]. After these tests, the moment of inertia, viscous damping and Coulomb friction are analyzed. Next, they performed finite element analysis on the drive for torsional modes and frequency response functions of torsional dynamics. However, due to the complexity of modeling the ball screw-nut interface, they received help from experiments to determine equivalent inertia reflected from linear motion of table and nut. Finally, they performed frequency response tests by applying sinusoidal torque signals to amplifiers and measuring rotational displacements. They concluded that the experimentally measured FRF's are in close agreement with the torsional mode shapes obtained from finite element analysis. Zaeh et al. [17] have created a more realistic FE model by using universal FEM programs. They detailed the model of ball screw-nut interface with a stiffness matrix. This matrix is 12x12 and includes torsional, axial and bending modes. Finally, they applied a special spring element with the help of proposed stiffness matrix on a universal FE program to analyze the system. However, this study might be insufficient in cross coupling of bending, axial and torsional directions. Based on this, Okwudire and Altıntaş [14] proposed an advanced method to consider the coupling terms between axial, torsional and lateral dynamics. Smith [18] desired to design a wide bandwidth control algorithm. Therefore, he needed a high order dynamic model. He implies that a coupling occurs between axial and torsional dynamics due to ball screw-nut assembly and the coupling causes coaxial moment on ball screw with force on ball nut. This coupling may be neglected in many cases since the transmission ratio of the ball screw is usually small [18]. So, he developed a beam type finite element model including coupling of axial and torsional

dynamics. He modeled the beam finite elements with two nodes that each has axial and torsional degree of freedom. The bearings and couplings were considered as pure springs. The axis table was considered as lumped mass. He linked the table (with ball nut) to ball screw node with a stiffness matrix that consists of axial and torsional dynamics. He performed modal analyzes on this model using MATLAB.

In this study, frequency domain analyzes on a 3-axis machine tool are performed. The machine tool that is analyzed has 3 axes, it is gantry type and placed in Milteksan R&D laboratory in Marmara University. Firstly, a finite element model is developed that only scopes the ball screw mechanism of the machine tool in MATLAB. Modal analyzes are performed on this model and compared to modal analysis realized in ANSYS Modal Analysis with a special element MPC184 for ball screw-nut interface. This model helps to identify the axial and torsional properties of the ball screw. Hence, the control system can be designed with this prior knowledge. The mechanical designer of ball screw may forecast the structural instabilities and revise the design for longer life of machine. Also, this model is appropriate for simulation purposes. Secondly, the whole machine including the ball screw (not as detailed as in the previous model) is analyzed in ANSYS Modal Analysis and Harmonic Analysis in order to observe the natural frequencies of a region that is foreseen before as the most flexible region. This flexibility of the region directly affects the positioning accuracy since it is close to cutting tool. After that, vibration tests on the flexible region are performed and power spectrum is gathered from 2 axes of the machine tool. The test is performed with two piezo electric and capacitive type accelerometers to compare with ANSYS analyzes. The importance of this test is proving the ANSYS model of machine tool and analyzing the frequency response. This function gives information of natural frequencies that affects the positioning accuracy and can give information for controller design. These determined natural frequencies are analyzed to ensure that the machine is not excited for long life and positioning accuracy.

2. MATERIALS AND METHODS

2.1. Three Axis Machine Tool

The analyzes and tests are run on a 3-axis gantry type milling machine that is manufactured for the development of the Local CNC system. Since the machine tool is not commercial, the purpose of creating such a structure is to test the developed CNC system.

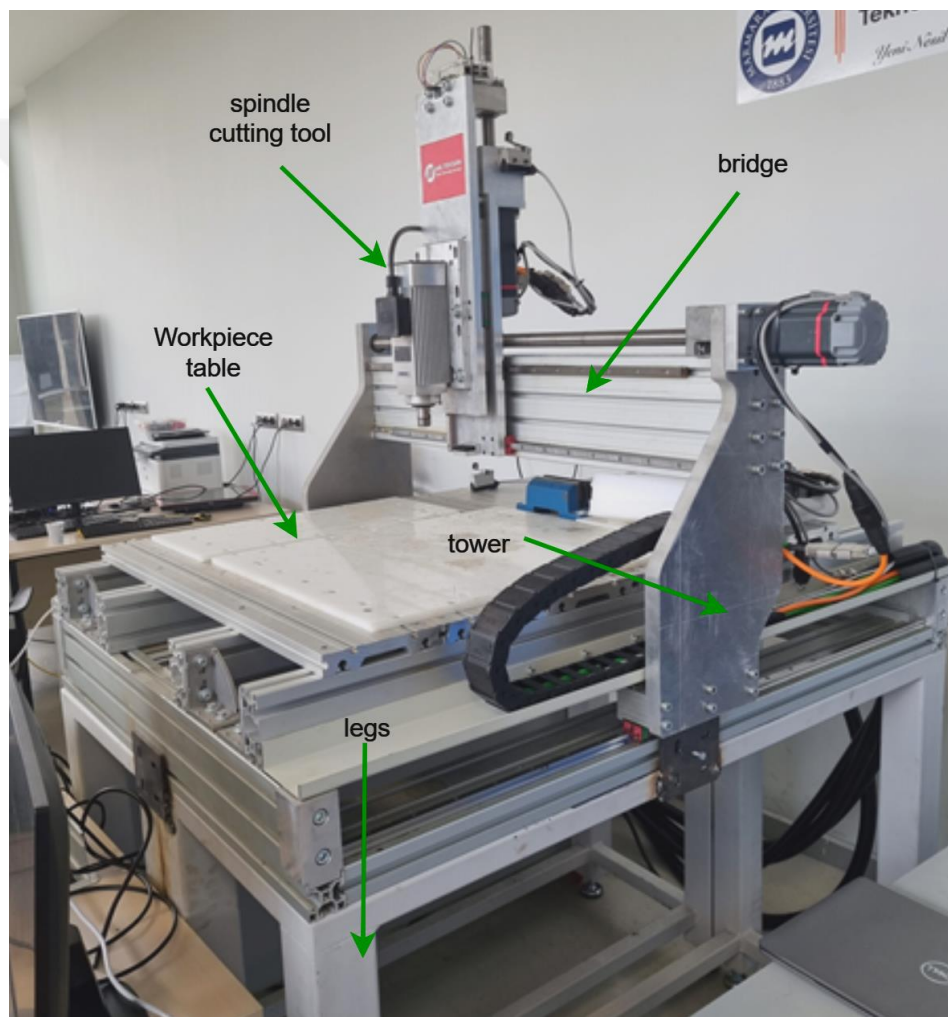


Figure 8 Three axis milling test machine tool

The Y axis (Figure 9) is the first axis and carried towers and bridge and other X and Z axes. X axis drives the Z axis on the bridge. Also, the Z axis drives the cutting tool up and down. The machine is mostly constructed by aluminum parts. The towers are aluminum plates that are

manufactured in water jet. The bridge that is a critical part and workpiece bed is constructed by aluminum sigma profiles. This machine tool manufacturing strategy makes the process cheaper and machine lighter. However, the major drawback of this strategy is that the machine becomes more flexible. Because, instead of a cast machine body, the machine consist of assembly of many parts and each assembly brings more fitting, flexibility and difficulty in modeling the dynamics. Also, the machine is not fixed to the ground. It has several aluminum legs. That's why, the machine sways on its legs.

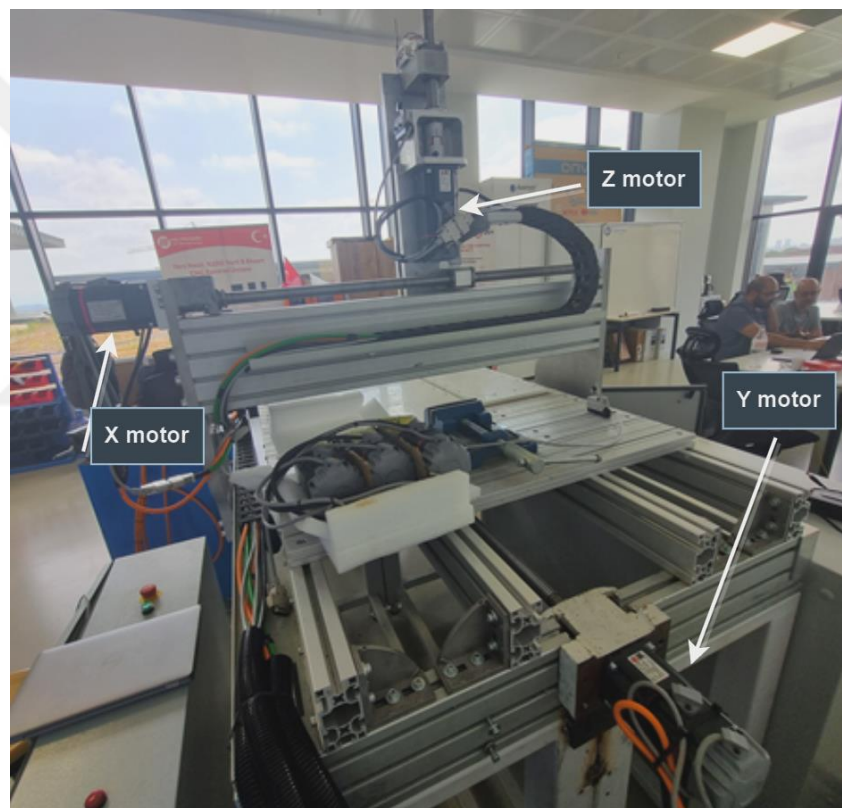


Figure 9 Axes and motors of the machine tool

The axes are driven by ball screw mechanisms. For example, X axis ball screw is shown in Figure 10. Ball screw mechanisms are insensitive to changes in axis inertia and axis disturbance forces. Each ball screw feed drive consists of several components like, motor, coupling, ball screw, ball nut, table, rails and bearings.

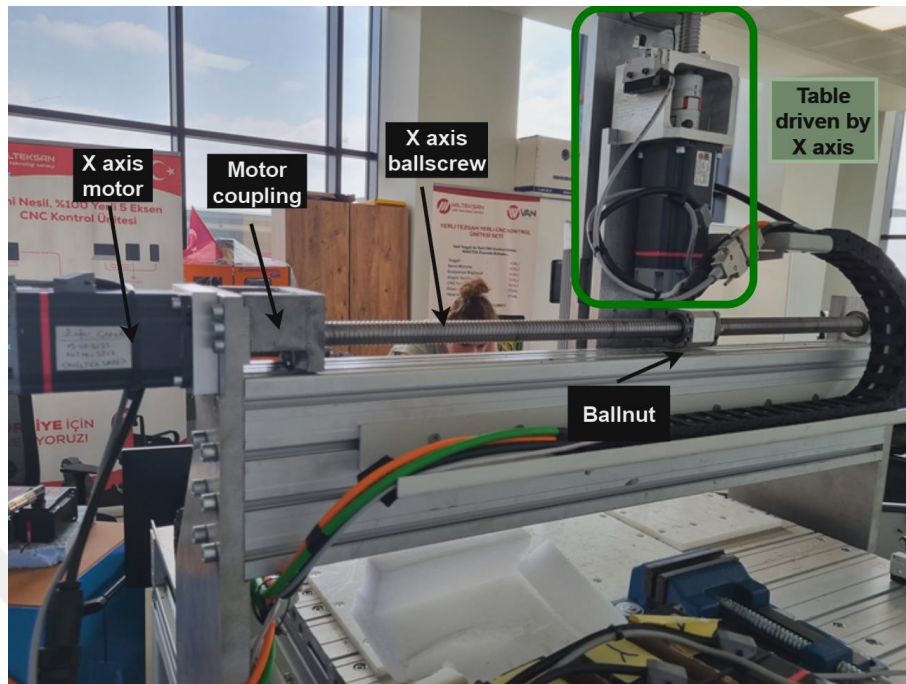


Figure 10 X axis ball screw feed drive

The axes are actuated with three servo systems. The servo system consists of a servo driver and servo motor. Servo driver has three mode that are, cyclic synchronous position, cyclic synchronous velocity and cyclic synchronous torque. Servo driver has closed loop position control function and it is currently used in industry for precise position control. Servo system utilizes a modern communication protocol that is EtherCAT. This protocol is based on ethernet but, faster and has no jitter. With this protocol it is possible to communicate with the servo system in 1ms precisely. For example, in CSP mode, target position that is interpolated in 1ms is fed to the servo system with EtherCAT frames. Because the servo systems are EtherCAT slaves, a master is needed to create the trajectory and feeding them into the servo systems for positioning. The overall CNC communication diagram is like below.

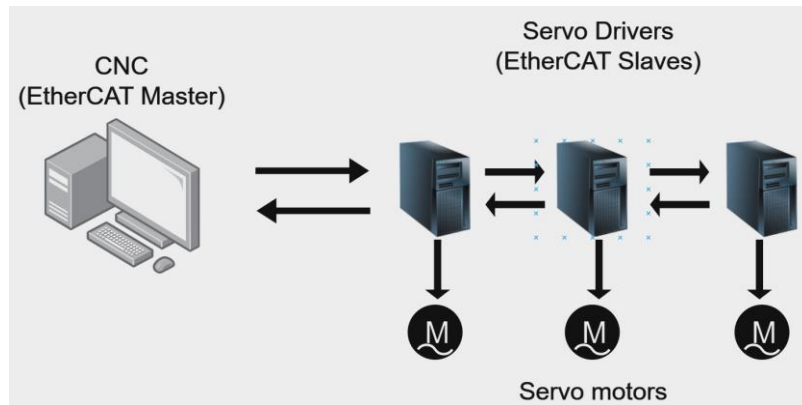


Figure 11 The system configuration

A servo system performs closed loop position control by feeding measured position back to control loop. In this setup, the measurements are taken from motor encoders with the sampling rate of 1000 Hz thanks to EtherCAT. This sampling rate can even be higher in certain situations. At each 1 ms, motor encoder position, velocity and current (related to torque) can be read real time. So, these measurements play a significant role in the tests. The only sensor read by servo system is motor encoder and guide rails don't have linear encoder. Hence the table may be positioned in somewhere that cannot be controlled by CNC because of the flexibilities.

In this study, two types of analysis are performed. The first analysis only focuses on the ball screw mechanism. Regardless of the remaining structure of the machine tool, the aim of developing a focused analysis for the ball screw mechanism is to create a universal model for it. Because the machine structure can show infinite variety. The ball screw mechanism, on the other hand, is mostly similar among machine tools. The second analysis focuses on remaining structure instead of ball screw mechanism. The whole structure is analyzed in frequency domain with ANSYS, and the results are compared to test results realized with two types accelerometers.

2.2. Finite Element Analysis of Ball Screw Mechanism

2.2.1. Analytical Hybrid Finite Element Model

In this section, the ball screw mechanism is analyzed in detail. Firstly, a model that does not include too much complex relationships, is developed to gain insight about the behavior of the ball screw in vibratory situations. This finite element model is mostly based on Smith's [18] model and it is adapted to the 3-axis test machine tool in this project. The finite element model is developed in MATLAB and the related codes can found in Appendix A. In this model, the ball screw and rotor are divided into finite elements with two degrees of freedom at each node. Each element has two nodes. Each node (Figure 12) has a translational (U) and torsional (R) freedoms so, the lateral dynamics are ignored in this model.

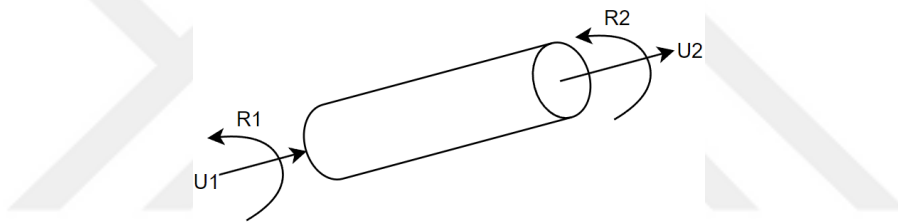


Figure 12 Beam element with translational and torsional freedoms

Each element has a stiffness and mass matrix (Equation 1, 2) that are later connected to create dynamic system matrix representation.

$$K_{el} = \frac{1}{l} \begin{bmatrix} EA & 0 & -EA & 0 \\ 0 & GJ & 0 & -GJ \\ -EA & 0 & EA & 0 \\ 0 & -GJ & 0 & GJ \end{bmatrix} \begin{matrix} U1 \\ R1 \\ U2 \\ R2 \end{matrix} \quad (1)$$

$$M_{el} = \frac{\rho l}{6} \begin{bmatrix} 2A & 0 & 2A & 0 \\ 0 & 2J & 0 & 2J \\ 2A & 0 & 2A & 0 \\ 0 & 2J & 0 & 2J \end{bmatrix} \quad (2)$$

The overall connections between parts are shown in Figure 13.

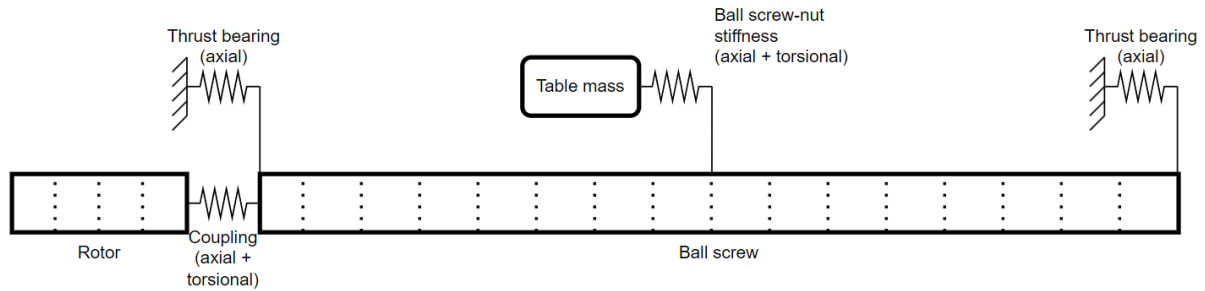


Figure 13 Overall model of ball screw feed drive

The model parameters were chosen to simulate the Y-axis ball screw mechanism of the real 3-axis test machine tool. Since the Y axis ball screw is very long and carries the highest load, the most critical axis was determined to be the Y axis, and the FE model was created for the Y axis only.

The dimensions of the rotor are adjusted to meet the inertia value in the catalogue since the technical drawing of rotor is not present. The moment of inertia of the rotor is $0.000109 \text{ kg} - \text{m}^2$ according to the catalogue [19]. Also, the material is not exactly known so, the elastic modulus and shear modulus are estimated as general steel's properties. The parameters are shown in below table.

Table 1 Mechanical parameters of the rotor

Length	0.186 m
Diameter	0.0295 m
Elastic modulus (E)	210 GPa
Shear modulus (G)	80 GPa
Density	7850 kg/ m ³

The coupling between the rotor and ball screw is a type of jaw coupling. This coupling consists of a plastic insert between two steel clamp and it helps damping the vibrations and compensates misalignments. The coupling in this model is considered as both an axial spring and torsional spring. The spring stiffness values are accepted as in the study by Smith [18] as the manufacturer of the 3-axis machine tool did not have access to the coupling catalogue. The axial stiffness is $1 * 10^{12} \text{ N/m}$ and the torsional stiffness is $5 * 10^{12} \text{ N/rad}$. It is estimated

that the coupling has no mass and consists of pure springs.

The ball screw in the model reflects the Y axis ball screw of the 3-axis test machine tool. Its model is GTEN 2505-4. According to catalogue [20], the material is 50CrMo4 alloy steel.

Table 2 Mechanical parameters of the ball screw

Length	1.2 m
Diameter	0.0225 m
Elastic modulus (E)	210 <i>GPa</i>
Shear modulus (G)	80 <i>GPa</i>
Density	7850 kg/ m ³

Since the threads on ball screw affect the stiffness even if it is small, the diameter is averaged between minor and major diameter of screw. The ball screw is mounted on the base with thrust bearings on each end with axial stiffnesses. In the test machine tool, the Y-axis is mounted with two tapered rollers bearing and they have both axial, radial stiffnesses. However, radial forces are ignored in the finite element model, so the bearings should only have axial stiffnesses in the developed model. The axial stiffness of the bearings is $4 * 10^9 \text{ N/m}$ (also gathered from Smith's model [18]). The ball nut has a total mass of 64.151 kg (Table 6) representing the whole mass of the table carried by Y axis. This mass is guaranteed by machine's entire CAD model and previous system identification studies on the test machine tool [21] (see Section 2.3.1.2).

The interface between the ball nut and ball screw is the most critical section in the model. Because, the geometry is complex and mechanism exhibits hysteresis type of motion. The coordinate transformations must be performed carefully. Also, it contains many nonlinearities like backlash and Coulomb friction. In this study, the model is developed in a simple way to get a starting insight about the mode shapes. The ball screw-nut interface is modeled with 3 degrees of freedom that are axial and rotational displacement of ball screw node and axial displacement of the ball nut (including entire Y axis table mass). These freedoms are related to each other with transmission ratio which is 0.005 m/rev. The stiffness matrix between ball screw node and ball nut is shown in Equation 3.

$$K_{screw-nut} = K_{axial} \begin{bmatrix} 1 & -r & -1 \\ -r & r^2 & r \\ -1 & r & 1 \end{bmatrix} \begin{bmatrix} U_{screw} \\ R_{screw} \\ U_{nut} \end{bmatrix} \quad (3)$$

The eigen vectors and eigen values are analyzed by solving below equations. The equation of motion is shown in Equation 4 [22]. The model is not exposed to any external forces because, a modal analysis is performed and structure free oscillates. This model does not include any damping since it requires accurate knowledge of damping properties of materials and fittings. Also, the resulting mass and stiffness matrices are constrained, reduced and they are supposed to be nonsingular matrices.

$$M\ddot{U} + KU = 0 \quad (4)$$

M: Global mass matrix

K: Global stiffness matrix

U: Displacement vector

Because the system oscillates, the displacements can be written as follow,

$$U = q e^{i\omega t} \quad (5)$$

q: displacement amplitude vector

ω : frequency of oscillations

t: time

The Equation 4 can be rewritten as follows,

$$(\omega^2 M + K)q = 0 \quad (6)$$

Solving this eigen problem in MATLAB yields eigen values (natural frequencies) related to eigen vectors (mode shapes).

2.2.2. Finite Element Analysis in Commercial Analysis Software

In order to measure the accuracy of the developed analytical, a parallel finite element model is developed in ANSYS from three axis test machine tool. Then, the mode shapes are compared to each other.

2.2.2.1. Geometry

The geometry to be analyzed in the software should reflect the Y axis of test machine as well as possible. This is one of the most critical steps. Each material should be defined to get realistic mechanical properties like Young's modulus, density and weight. Since in Section 2.2, the ball screw is observed in detail, the geometry of Y axis of the test machine tool is simplified according to Figure 13. The geometry to be analyzed in the software is shown in Figure 14.

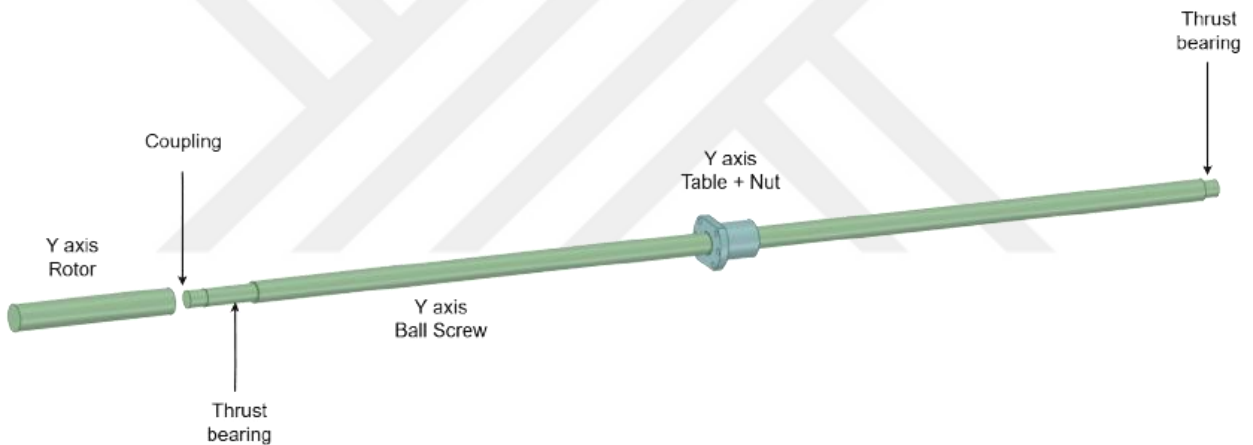


Figure 14 The CAD geometry of Y axis for modal analysis and constraints

The rotor geometry was created to satisfy moment of inertia in catalogue [19]. The diameter of the ball screw was taken as 22.5 mm (averaged from minor and major dia.). From the whole CAD model (Section 2.3.1.1) the Y axis table mass is lumped to a single 64.171 kg mass which is represented as the nut (with table mass) in Figure 14. The mechanical parameters are parallel with the analytical hybrid FE model (Section 2.2.1). The masses in machine's entire CAD model were previously proved with tests run by Enes et al. [21]. In these tests, the equivalent moment of inertia of each axis was found. Then, with the help of transmission ratio, total mass of each axis table was calculated. The CAD model satisfies these masses. The density, elastic and shear moduli are same as the FE model developed in previous Section 2.2.1.

2.2.2.2. Connections and constraints

The model should be constraint to simulate real machine's behaviors. Firstly, the rotor is constraint with a revolute joint so, it can just rotate on its axis and cannot axially move. The rotor can be thought as mounted with infinite stiffness bearings. The rotor and ball screw are connected with axial and torsional springs (represents coupling). In modal analysis, ANSYS does not allow to create torsional spring. Therefore, a COMBIN14 element is manually created [23]. Its APDL code can be found in Appendix B.1. The axial spring is directly created with no difficulty. Also, the ball screw is mounted to the ground with axial springs on its each end (along the screw axis). These springs represent thrust bearings that constrain the screw from moving axially.

The most challenging part is modeling the ball screw-nut interface. Many approaches can be realized in this section. The most realistic interface can be modeled with defining contacts. In this strategy, the analysis gets more complicated, more available to divergence of solution and causes more computation load. It requires high knowledge of contact theory. Also, the geometry must be realistic with the presence of balls inside the nut. This strategy was excluded due to the inability to create a complete CAD model of the screw nut. The other strategy is basically creating an axial and torsional spring between screw and nut. This strategy is close to the model developed in this study. However, the only difference is the axial displacement is not coupled to rotational displacement. The advantage of this model is its simplicity. The third strategy which is utilized in this study, is creating a MPC184 screw joint with the help of APDL commands (Appendix B.2). MPC184 is a group of constraint elements and it defines constraints in motion. In MPC184 screw element, the axial displacement is related with the rotational displacement by a transmission ratio. The most important property of MPC 184 is the ability to define stiffness matrix of constraint element between its nodes. According to ANSYS Mechanical APDL Element Reference [23] the stiffness matrix of MPC 184 screw joint between screw node and nut node is,

$$K_{screw-nut} = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & K_{axial} & 0 & 0 & K_{tors-ax} \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & K_{ax-tors} & 0 & 0 & K_{torsional} \end{bmatrix} \begin{matrix} UX \\ UY \\ UZ \\ RX \\ RY \\ RZ \end{matrix} \quad (7)$$

The deflections in this model are UX, UY, UZ, RX, RY and RZ . These deflections are according to the local coordinate frames (Figure 15). The frame “I” is on the screw and frame “J” is on the nut. The translation of frame “J” along $e3$ vector is related to rotation of frame “I” about the $e3$ vector. UX, UY, UZ, RX, RY and RZ deflections are differences between the desired location and actual location according to the motion so in this way recovery force and torque is created which cause vibration. For example, axial flexibility caused by UZ causes a torque which is related to $K_{ax-tors}$, in Z axis on both nodes. If the nut is not in the desired rotational location (can be converted from axial location) the nodes are exposed to a recovery torque calculated with the $K_{torsional}$ direct stiffness. As it is clearly seen that, the dynamics are very coupled so, the motions cannot be separated from each other.

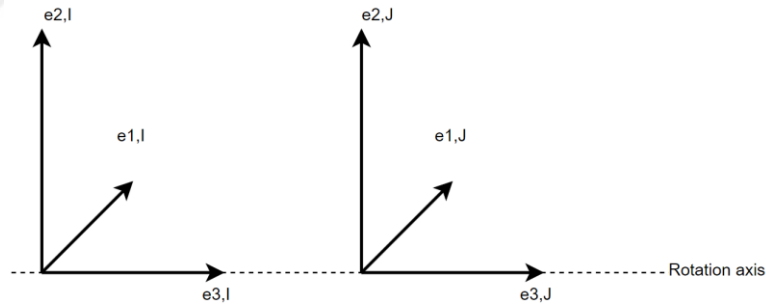


Figure 15 Local coordinate frames on joint

The three stiffness terms in Equation 7 can be calculated with K_{axial} as it is in Equation 3.

$$K_{torsional} = K_{axial} * r * p = 143.239 \text{ Nm/rad} \quad (8)$$

$$K_{ax-tors} = K_{axial} * r = 180000 \text{ Nm/m}$$

To fully constraint the model, a translational joint is applied to the nut along the screw axis. Because the table translates along the rails. Finally, the modal analysis is performed with this fully constraint model to observe the mode shapes and natural frequencies of only ball screw

mechanism.

2.3. Frequency Domain Analysis of The Machine Tool's Structure

2.3.1. Frequency Response Analysis in Commercial Analysis Tool

In this section, entire structure of the machine tool is analyzed in frequency domain. In the previous section, the ball screw mechanism was focused. In this section, the ball screw joint is defined with MPC 184 screw joint but, without stiffness values. Therefore, the ball screw – nut interface does not have any flexibility. Additionally, the coupling between rotor and ball screw is considered as fixed to relax the solution. Contrary to previous section, in this analysis, two axes that are X and Y are analyzed together.

2.3.1.1. Geometry

Original and unedited CAD geometry is received from the manufacturer of the machine tool. The grounded parts like table and legs are removed from the geometry and the only movable parts are left. So, the vibration of whole machine frame on its legs is ignored. Within the scope of this project, the geometry is simplified by flattening small faces, combining small and similar parts together in order to relax the solution. Although the CAD model is taken from the manufacturer, the materials are carefully assigned to each part. For combined parts, the total mass is maintained by tuning related material's density. Similarity between the real machine and the geometry is tried to maintain.

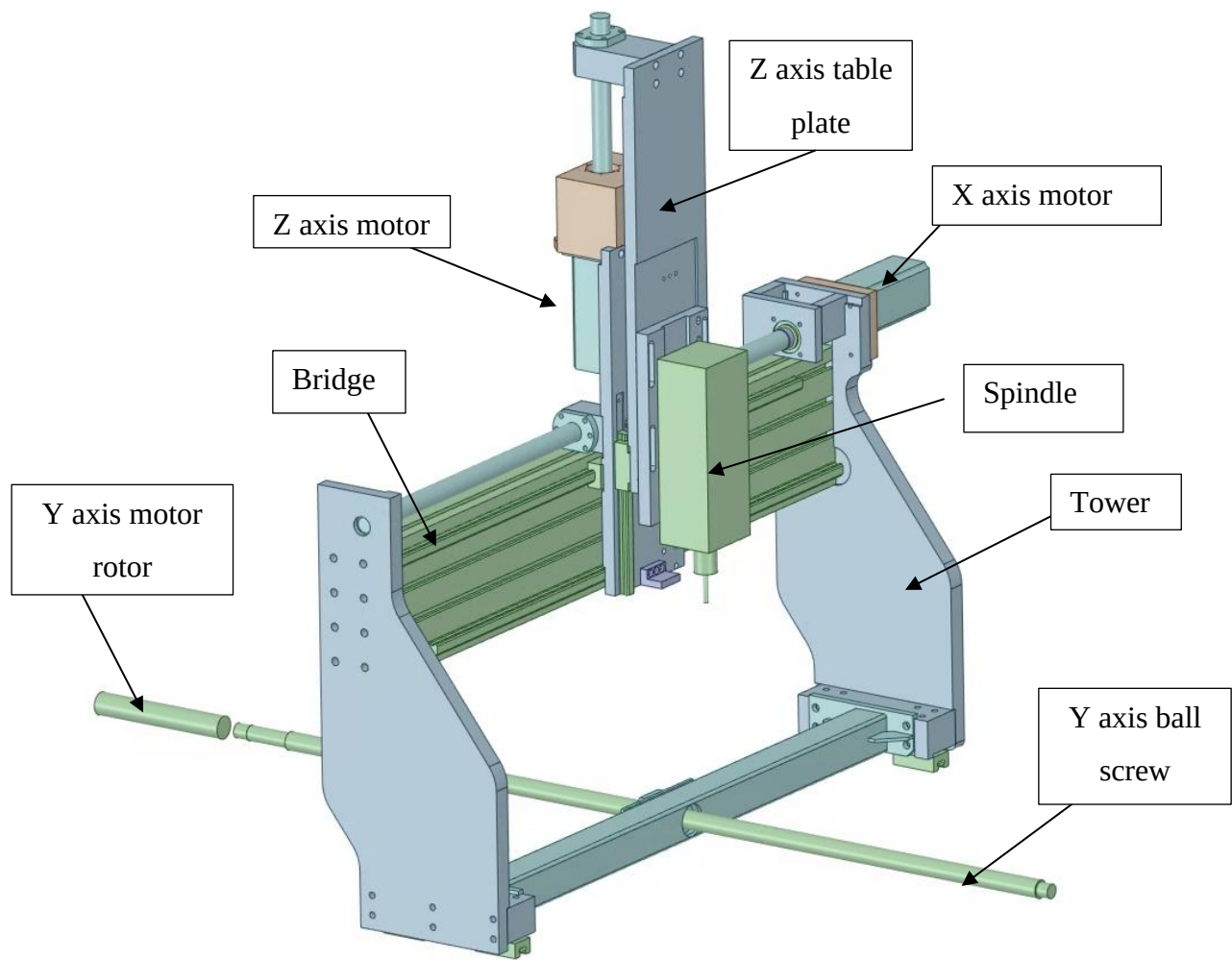


Figure 16 Machine tool's entire CAD model

The motor CAD models are received from their manufacturers, and they are simplified for analysis. Materials of the motors are assigned in order to maintain their catalogue weights.

Table 3 Motor weights

Motor name	Weight
X axis servo	2.8 kg
Z axis servo	3.6 kg
Spindle	4.95 kg

Since the Y axis motor is not carried by any of axes and it is driving Y axis table, only the Y axis rotor shaft is modeled and assigned a material in order to maintain its moment of inertia. Material assignments of some of the most critical parts are below.

Table 4 Some of the critical material assignments

Part Name	Material Names
Y axis rotor	General Steel
Ball screws	50CrMo4
Tower	Aluminum 5083
Bridge	Aluminum 6063-T6
Z axis table plate	Aluminum 5083

The properties of these crucial materials are below.

Table 5 Properties of some materials

General Steel	Density:	7.8 g/cm ³
	Elastic modulus:	200 GPa
	Shear modulus:	60 GPa
50CrMo4	Density:	7.85 g/cm ³
	Elastic modulus:	210 GPa
	Shear modulus:	80 GPa
Aluminum 5083	Density:	2.66 g/cm ³
	Elastic modulus:	70 GPa
	Shear modulus:	26.4 GPa
Aluminum 6063-T6	Density:	2.7 g/cm ³
	Elastic modulus:	68.9 GPa
	Shear modulus:	25.8 GPa

2.3.1.2. Proof of CAD model mass properties

To maintain reality in CAD model, the mechanical parameters must be satisfied. In a previous study carried out by Demir et al. [21], the equivalent moment of inertia, viscous damping and Coulomb friction are estimated for each axis. They applied a special wave torque inputs to X and Y servos separately, then they received angular velocity data from motor encoders. With the torque input and angular velocity output, they performed Least Squares method to find parameters. In this section, the Y axis CAD model mechanical parameters are compared with equivalent moment of inertia with suitable conversions from linear to rotary domains.

Y axis mechanical parameters in CAD model are below.

Table 6 Y axis mass properties in CAD model

Lumped mass	Moment of inertia (kg-m ²)	Mass (kg)
Y axis Motor Rotor	0.000109031	-
Y axis Ball Screw	0.000302604	-
Y axis Table	-	64.151 kg

According to the Least Square algorithm [21] the estimated parameters are below.

Table 7 Y axis mechanical parameters identified with LS method

Parameter	Value
J, equivalent inertia, $Nm / (\frac{rad}{s^2})$	0.000453
B, equivalent viscous damping, $Nm / (\frac{rad}{s})$	0.0064
T_{coul}^+ , Coulomb friction in + direction, Nm	0.5048
T_{coul}^- , Coulomb friction in - direction, Nm	-0.4883

Firstly, the Y axis table mass must be extracted from equivalent inertia found by LS. Then this table mass will be compared to Y axis table mass in CAD model. The equivalent inertia consists of 3 parts and equalized to LS method finding.

$$J_{eqv} = J_{rotor} + J_{ballscrew} + J_{table} = 0.000453 \frac{Nm}{\frac{rad}{s^2}} \quad (9)$$

Y axis rotor inertia (J_{rotor}) is known from the catalogue of the motor.

$$J_{rotor} = 0.000109 \frac{Nm}{\frac{rad}{s^2}}$$

J_{screw} value is received from CAD model because there is no catalogue value. The diameter of the screw was averaged. With the knowledge of rotor and ball screw moment of inertia, J_{table} is obtained.

$$J_{screw} = 0.000302604 \frac{Nm}{\frac{rad}{s^2}}$$

$$J_{eqv} = 0.000109 + 0.000302604 + J_{table} = 0.000453$$

$$J_{table} = 0.000453 - 0.000302604 - 0.000109 = 0.000041396 \frac{Nm}{\frac{rad}{s^2}} \quad (10)$$

Then, the mass of Y axis table (m_{table}) is obtained from the Y axis table moment of inertia, J_{table} (the table translates but can be transformed from rotation). According to Altintas et al. [24], mass of the table from the moment of inertia by using screw pitch is,

$$J_{table} = m_{table} \left(\frac{p}{2\pi} \right)^2$$

$$m_{table} = \left(\frac{2\pi}{0.005 \text{ m}} \right)^2 0.000041396 \frac{Nm}{\frac{rad}{s^2}} \quad (11)$$

$$m_{table} = 65.37 \text{ kg}$$

The result is very close to the Y axis table mass from CAD model (Table 6) which is **64.151 kg**. The difference can be due to the unconsidered masses like screws and cables, and slightly

inaccurate design of ball screw due to diameter averaging.

2.3.1.3. Connections and constraints

In this CAD model in ANSYS, the X and Y axes can move so, their ball screw-nut connections are modeled like Section 2.2.2.2 with MPC184 screw element but with infinite stiffnesses. The Z axis movement is fixed in the analysis because of its high rigidity and short moment arm. Also, the Z axis stroke is very short, so, it is not safe to test the Z axis. Generally, the parts that are moving together, are fixed with bonded contact to each other. For example, tower - bridge, X axis motor – flange – tower, spindle and Z axis table. Some contacts involve bolted connections however, to relax the solution they are simplified to face to face bonded contact. One of the most critical connections is translation of table cars along the rails. If the Z axis rail-car connection is defined as bonded in car-rail contact, the rigidity of machine increases because of constraining the movement of axis. That's why, the car and rail connection in Z axis is defined as translation joint. Although, the Z axis does not move, the translation joint in Z axis car and rail is important so as not to restrict the flexibility of the Z-axis table.

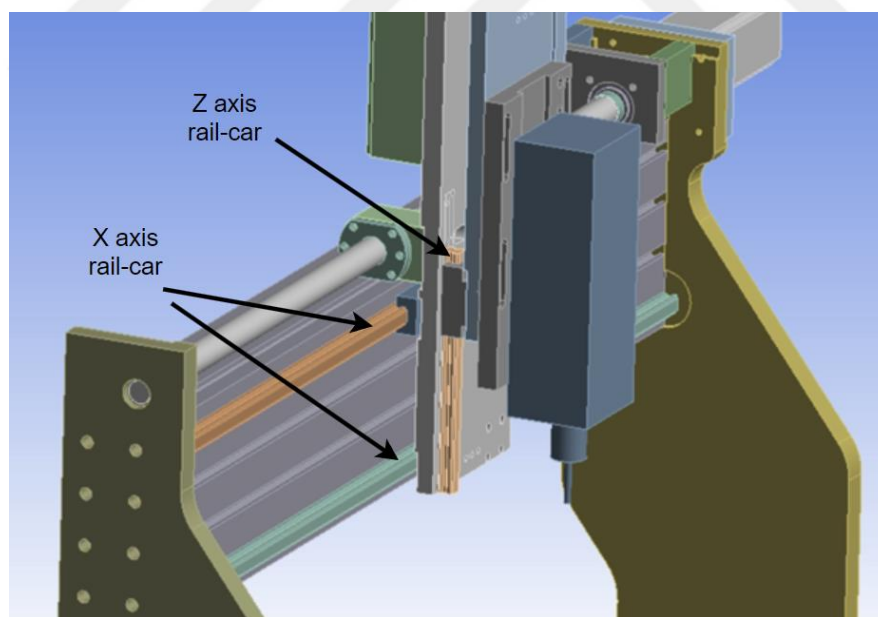


Figure 17 Linear guides in the machine

2.3.1.4. Solution

The model is ready for analyzing in ANSYS. Since natural frequencies, mode shapes and also

frequency response is required in this section, the modal analysis and harmonic response analysis are performed. The modal analysis computes the natural frequencies and mode shapes. The harmonic analysis is utilized to get input and output relation in frequency domain. This analysis yields a FRF plot that shows how the output will react with respect to a specific frequency of input. This method is performed over a range of frequency of input. The modal analysis is set to give results compatible with Mode Superposition Method for harmonic analysis. This method relaxes the solution of harmonic analysis by using modes received from modal analysis. On the other hand, without MSUP, the harmonic analysis is called full harmonic analysis which solves the governing equation directly using the static solver. ANSYS project structure of modal analysis and harmonic analysis with MSUP is shown in Figure 18.

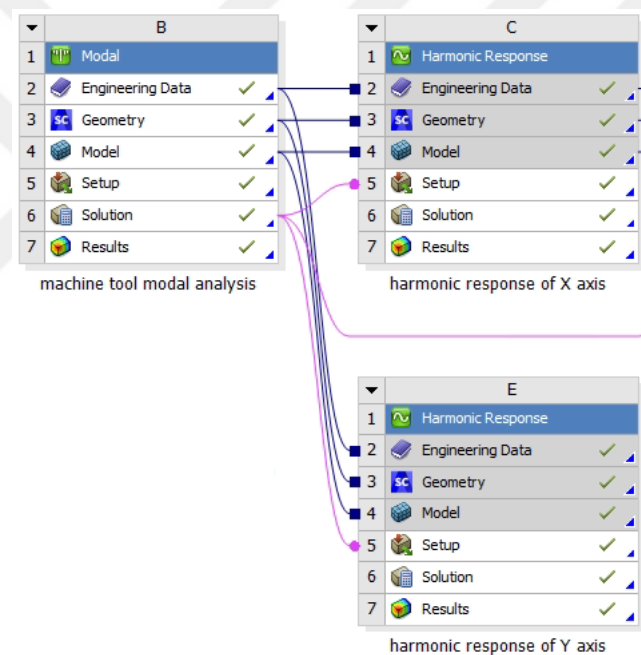


Figure 18 Modal analysis and harmonic analysis with MSUP

In harmonic analysis, the input load for X and Y motors are set to 2.37 N.m which is the nominal torque of motors. The output acceleration is read from the top of the Z table plate which is the most vibrating region as shown in Figure 19. As a result, two frequency response plots that are from X axis motor torque to X axis acceleration of the vibrating point and Y axis motor torque to Y axis acceleration of the vibrating point, are obtained.

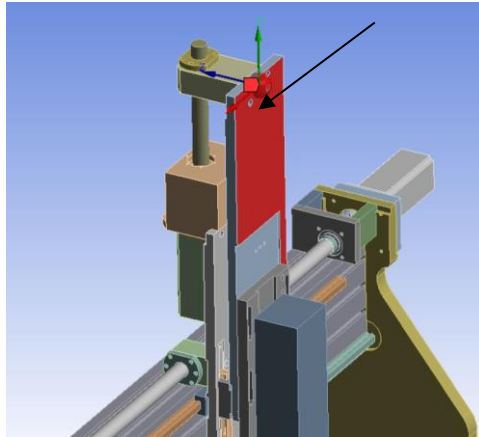


Figure 19 The measurement point on the Z table plate

2.3.2. Power Spectral Analysis with Impulse Tests

The purpose of ANSYS analyzes were observing the vibration properties like natural frequencies, mode shapes. Analysis results must be proved by performing real tests and signal processing. The axis motors are subjected to a couple of different torque inputs and acceleration output data is gathered from the most flexible region with two acceleration sensors. The aim of the test is exciting the natural frequencies of axes and gathering acceleration data to analyze the frequency power which gives natural frequency information.

Servo system is equipped with EtherCAT communication protocol that allows a sampling period of 1 ms. The measurements of the servo system are taken from motor encoders. Servo system can give position, velocity and torque data by utilizing motor encoders. So, while the position and velocity of motor rotors are known, data of any point on the machine frame is not known.

While the servo system is used to excite the axes dynamics, accelerometers are used to gather vibration data of any point on the structure. There are two accelerometers utilized in this study. The purpose of using multiple sensors is to determine the most reliable method among ANSYS and real vibration tests. One of the sensors is a capacitive type sensor. Its model is ADXL345. It is used in many studies since it is affordable and useful at many situations. ADXL345 accelerometer is used with SPI interfacing to Raspberry Pi. After the acceleration reading, the

output data is saved on the Raspberry Pi as text file. The resolution of ADXL345 is set to 13 bit, and sampling frequency is set to 1000 Hz. So, the Nyquist frequency is 500 Hz for both the servo system and ADXL 345. The sensor can read acceleration in three axes simultaneously. The matching axes of sensor with machine's X and Y axes are taken into account. The sensor is placed on the top of the Z axis plate (Figure 20) which is the most flexible region as it is in Figure 19.

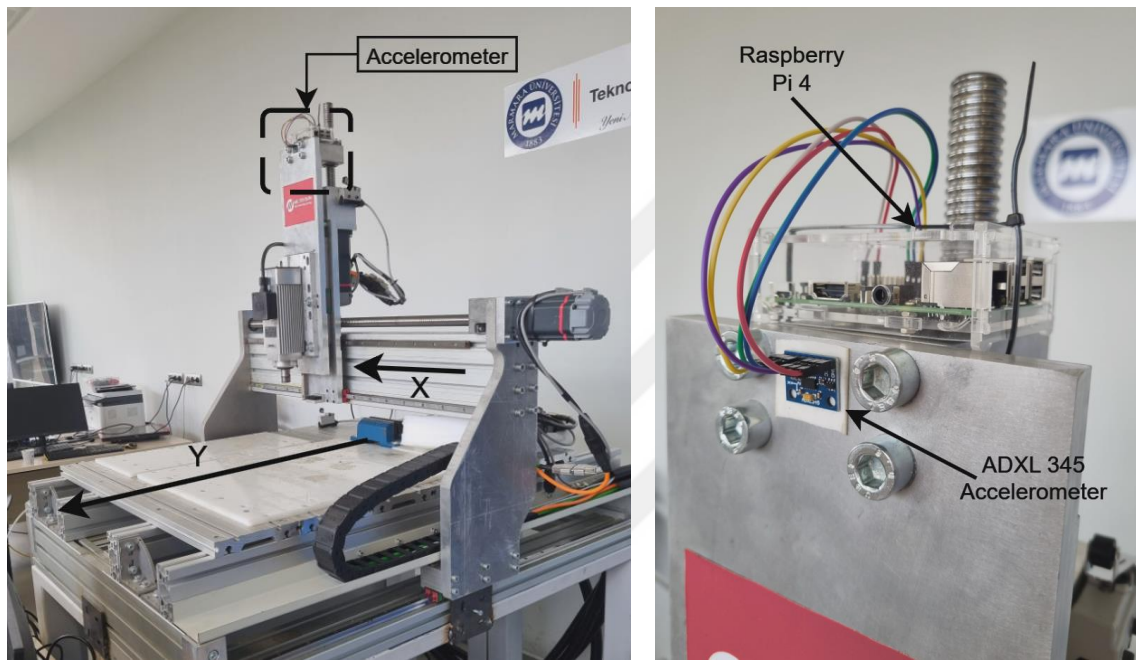


Figure 20 ADXL 345 accelerometer with Raspberry Pi placed on the most flexible region

Another accelerometer used in this study is a piezo electric type sensor. Its model is PCB Piezotronics 393B04. This sensor's usage is more professional and less affordable than ADXL345. Also, thanks to nature of piezo materials, the sensitivity is much higher than capacitive sensors. The mounting is more comfortable and the measurements are more reliable with the piezo electric sensor. This sensor can only measure one direction and the sensor is placed according to the test axis (see Figure 21). The acceleration output from piezo electric sensor is streamed through data acquisition card (see Figure 22) and the results are saved in a text file again.



Figure 21 Piezo electric sensor placed on X and Y axes respectively.



Figure 22 Data acquisition card

Size of the piezo electric sensor is decided while performing tests. Because the signal to noise ratio should be high and sensor sensitivity is important in this case. The heavier sensor provides higher sensitivity. So, the sensor size is increased step by step until getting a reasonable

acceleration data in time domain.

The sensitivity of the 393B04 model piezo electric sensor is 102 mV/(m/s²). The maximum frequency can be read is up to between 450 and 750 Hz according to the test. However, the sampling rate is approximately 5000 Hz.

In the test, both axes are subjected to various input torques. These inputs are expected to include a wide range of frequency content. At the same time acceleration data is received from sensors. The tests were repeated for each sensor. The applied inputs are below,

1. In position control mode, drive X and Y axes simultaneously 200 mm forward and backward with 150 mm/s (maximum possible velocity) and non-stop cornering. This test is realized in position control mode so, the torque is auto calculated inside the servo system for perfect position tracking. This test represents a real cutting operation with non-stop cornering (vibrates the machine)
2. In torque control mode, apply peak torque impulse for 5ms. Impulse test is perfect for revealing the modes of machine. The same test consists of many impulses.

The input data consists of each X and Y axis rotor position, velocity and torque from servo system and output data is the acceleration from the accelerometers on the machine.

2.3.2.1. Signal processing of test results

The applied torque and read acceleration data must be processed in order to observe the frequency content. Frequency content of the torque and acceleration data is revealed with power spectrum analyzes. By observing the spectrum of torque, it can be seen the power of torque corresponding to discrete frequencies in the axis. Furthermore, by investigating the power spectrum of acceleration, it can be seen how the amplitude of acceleration changes over a frequency range. The frequency range of acceleration spectrum is dependent to the frequency range of torque spectrum. The basis of these spectral analysis methods is the Fast Fourier Transform (FFT). However, applying simple FFT on the signals can lead to inaccuracies. Because the FFT assumes the signal is infinite and periodic. Applying FFT can cause spectral leakage due to this property. The solution to this problem is applying a window to the signal. So that, the signal can be extended to an infinite periodic signal without discontinuities in time

series. This leads to smoother spectrum of the signal.

The other problem with the raw FFT (periodogram) is that the noise in the signal is reflected to spectrum graphs. Noise is random and can have a wide range of frequency content. The amplitude peak in spectrum caused by noise can be confused with the machine's natural frequency. The noise can be suppressed as much as possible with averaging segments over the entire signal time series. Under the light of these problems, some methods are developed like Bartlett's method, Welch's method and Blackman-Tukey method. Bartlett divided a time series into non-overlapping segments and calculated periodogram of each segment. At the last, the periodograms are averaged to eliminate noise in the exchange of decreased frequency resolution [25]. Welch advanced this method with some modifications. He also divided time series into segments, but the segments are overlapped. The overlapped segments are exposed to a window function to prevent spectral leakage. Finally, he calculated modified periodograms from overlapped and windowed segments, then averaged the periodograms. In the Blackman-Tukey method, instead of calculating FFT of time domain data, they calculated FFT from auto correlation of time series. Auto correlation shows the relation of a time series data with itself and reveals repeating patterns. This method is based on Wiener-Khinchin theorem.

In this study, Welch's and Blackman-Tukey's methods are utilized and compared to ensure results. For the impulse test, many impulses are applied during single test run. So, the received data involves multiple excitations. The single test data is divided into many tests like they are different tests. In Welch's method, this single excitation data is divided into overlapping segments and windowed. The modified and overlapped periodograms of each excitation signal are averaged. So, the entire data which include many excitations are not analyzed at once. Like before, Blackman Tukey method is also performed on each single excitation and averaged with other excitation data to get reasonable result. It should be said that averaging in Blackman Tukey method is not for reducing vibration affect and it is for decreasing the variance of actual peaks of the spectrum. After all, FRF graph from ANSYS harmonic analysis and, power spectrum graphs from two sensor with different power spectrum analysis methods are compared.

3. RESULTS AND DISCUSSIONS

3.1. Ball Screw FEA Results of Analytical Hybrid FE Model

The output of the analytical model developed in Section 2.2.1 is mode shape plots and natural frequencies of only Y axis ball screw mechanism. The mode shape plots are shown with a ball screw model at the bottom of graph for better visualizing. The red dot visualizes the actual axial displacement of the nut. The axial and torsional mode shape plots show respectively the axial displacement and torsional displacement of each node on the ball screw. The axial mode shape is scaled to transmission ratio to be able to compare to torsional mode shape. The pink, green and purple elements represent ball screw, table (nut) and rotor respectively. The coupling is visualized between ball screw and rotor as a spring. The small purple elements on the ball screw represent thrust bearings.

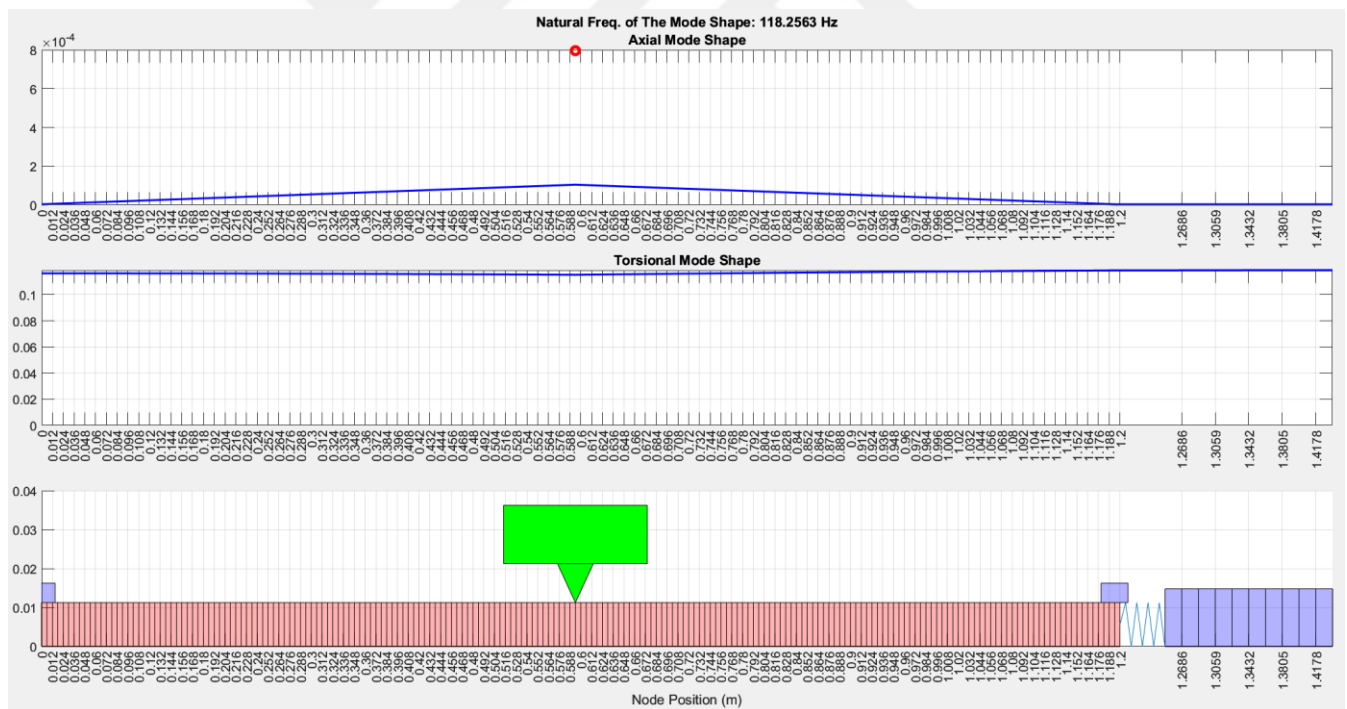


Figure 23 Mode shape of ball screw #1 – 118 Hz

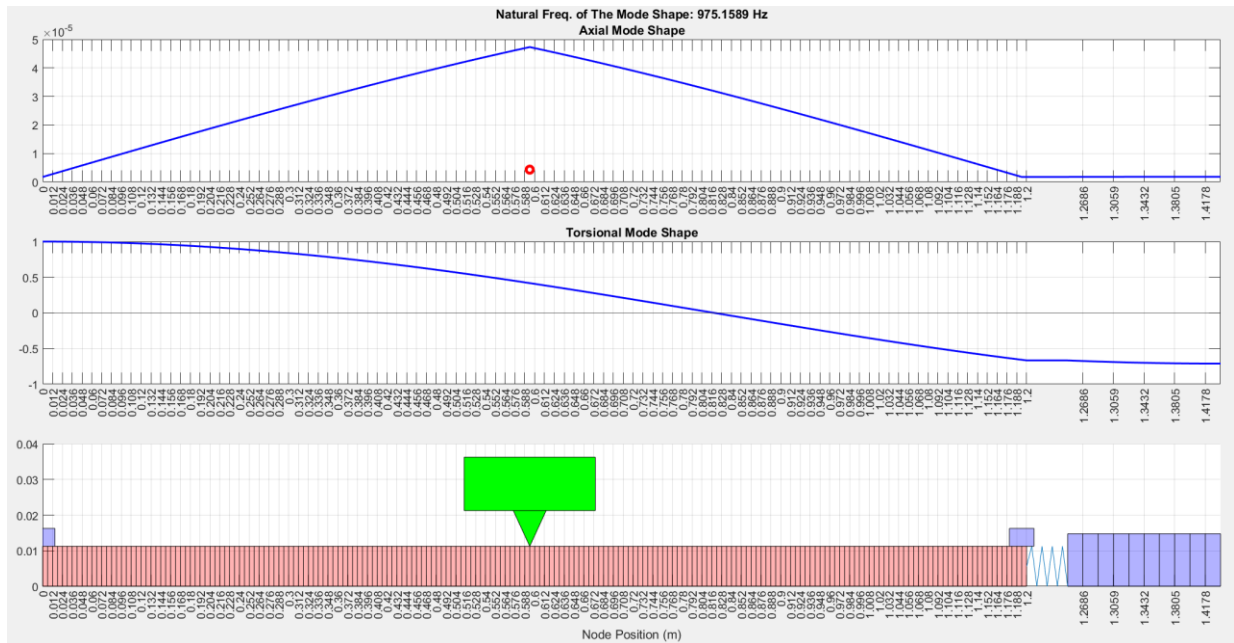


Figure 24 Mode shape of ball screw #2 – 975 Hz

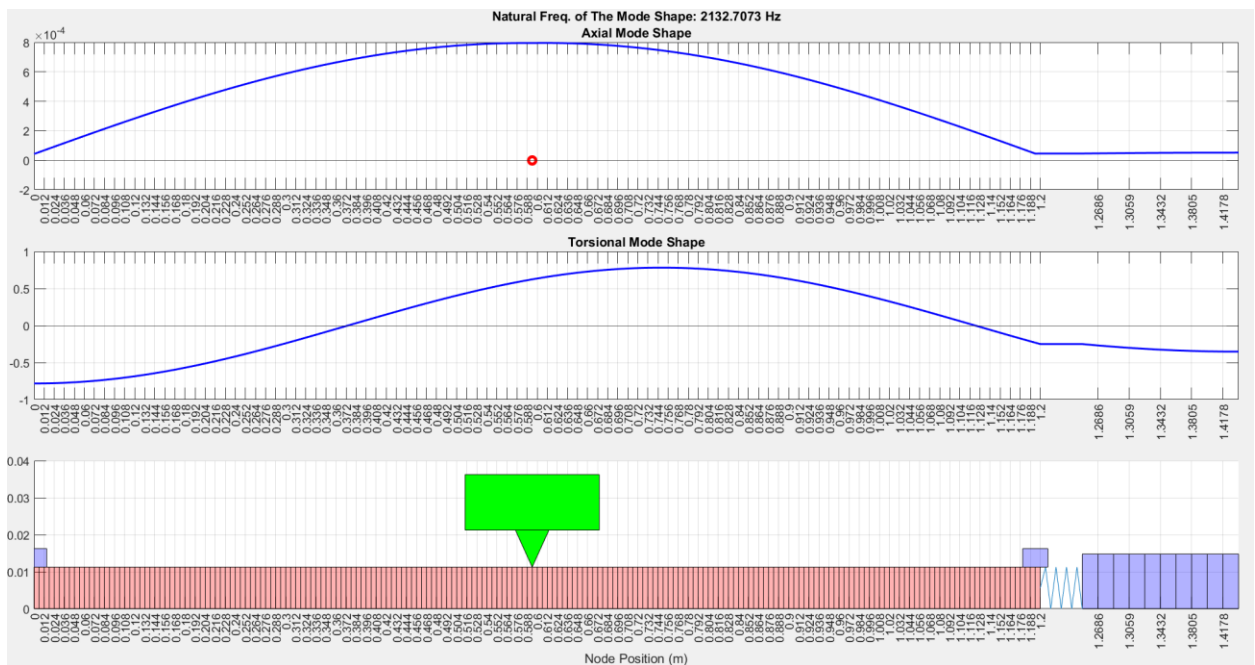


Figure 25 Mode shape of ball screw #3 – 2132 Hz

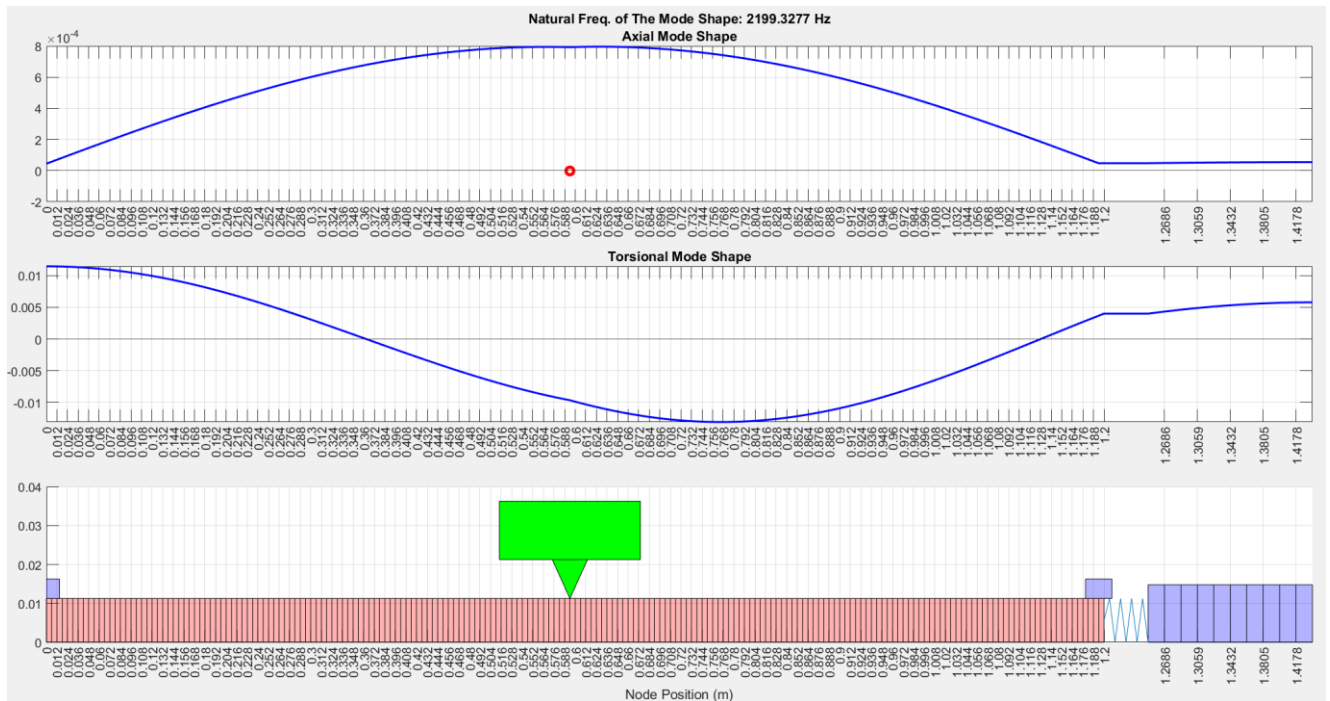


Figure 26 Mode shape of ball screw #4 – 2199 Hz

The other mode shape plots are included in Appendix C. According to the graphs above, the ball screw has an axial mode in 118 Hz. There is a difference between actual nut position and related screw node axial position. Additionally, there are many torsional mode shapes in 975, 2132, 2199 and 3374 Hz (see Appendix C - Figure 48). Furthermore, there are some mode shapes that are dominated with both axial and torsional together in natural frequencies of 4160 and 5547 Hz (see Appendix C - Figure 49 and Figure 51).

3.2. Ball Screw FEA Results of Commercial Analysis Software

In commercial tool, in contrary to analytical model, also the lateral modes are analyzed. These lateral modes cannot be compared with the analytical model so, they are not given in this study. The matching axial and torsional mode shapes are given below.

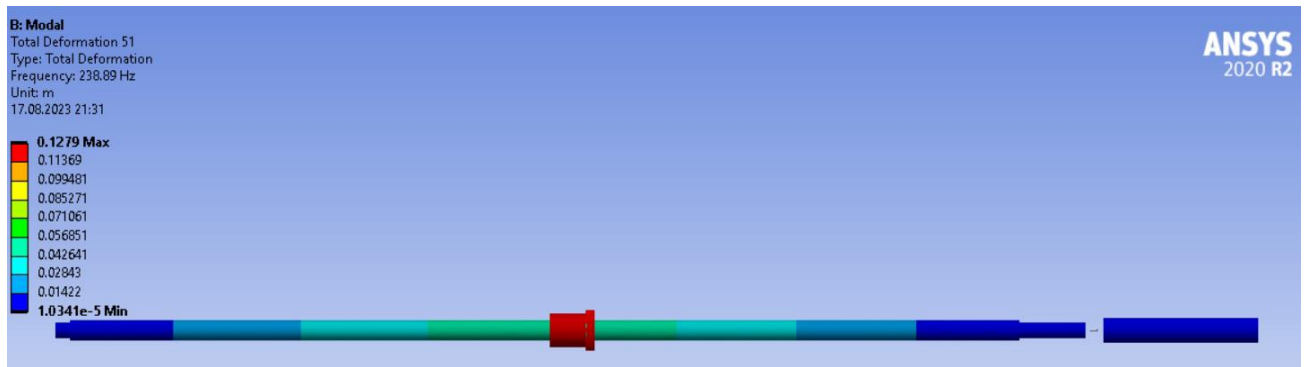


Figure 27 Mode shape of ball screw by ANSYS #1 – 238 Hz

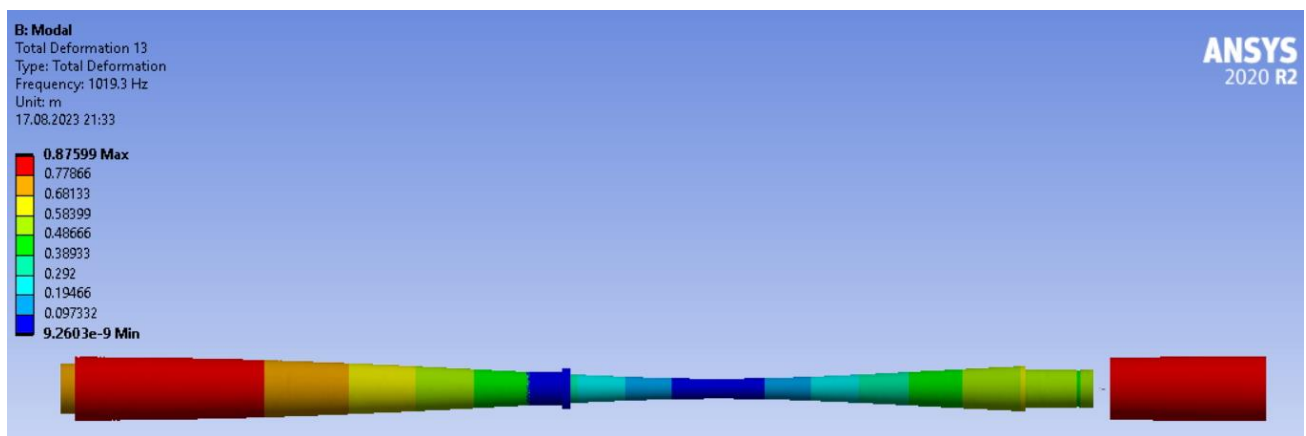


Figure 28 Mode shape of ball screw by ANSYS #2 – 1019 Hz

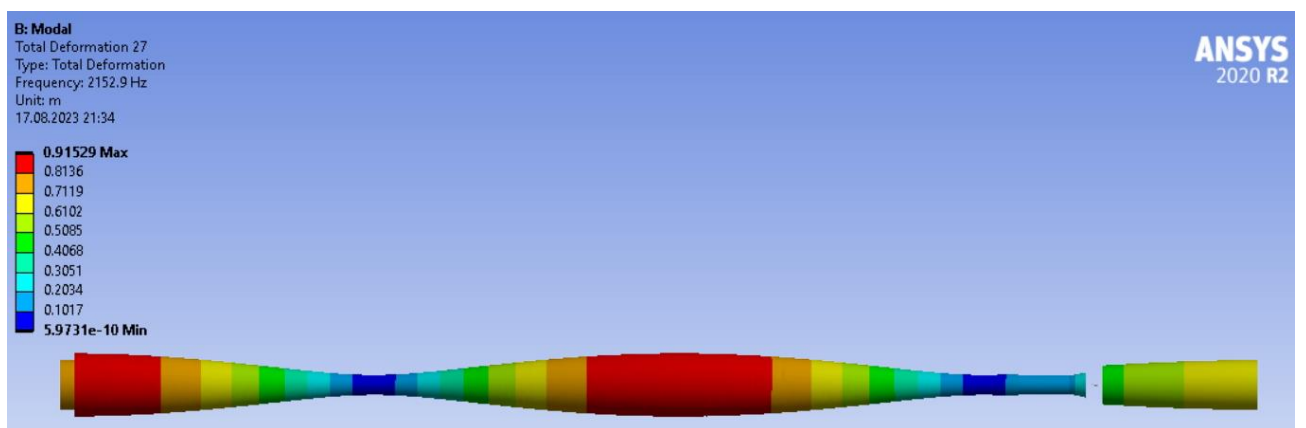


Figure 29 Mode shape of ball screw by ANSYS #3 – 2152 Hz

According to the results, they show a close behavior to analytical hybrid model analysis results. The matching natural frequencies are shown in Table 8.

Table 8 Comparison of natural frequencies of analytical model and commercial software model

Mode no	Analytical model	Commercial Software
#1 natural frequency (axial)	118 Hz	238 Hz
#2 natural frequency (torsional)	975 Hz	1019 Hz
#3 natural frequency (torsional)	2132 - 2199 Hz	2152 Hz
#4 natural frequency (torsional)	3374 Hz	3356 Hz
#5 natural frequency (axial)	4160 Hz	4352 Hz
#6 natural frequency (torsional)	4637 Hz	4581 Hz

As it is clearly seen that, the error is mostly small in the analyzed modes. The errors in axial modes are a bit higher however, the axial modes follow a similar trend and they are in close regions. So, it can be said that commercial software FE model and originally developed analytical hybrid FE model results in close natural frequencies and even mod shapes. Especially the ball screw – nut modelling with MPC184 element in ANSYS and analytically developed ball screw – nut stiffness matrix [18] gives similar results in modes. However, the analysis can be adjusted for more accurate results, such as optimizing the mesh and optimizing the geometries to be more similar to each other. Additionally, the geometry designed in analytical model is much simpler than commercial software model by its nature. This causes some differences. As a result, it can be stated that both models are satisfied and presented in detail for other further researches.

3.3. Frequency Response Analysis of Machine Tool in Commercial Tool

In this section, rather than ball screw mechanism, the frame of the machine tool is analyzed with modal analysis and harmonic analysis in ANSYS. The modal analysis of the machine tool yields dominant natural frequency and mode shapes. The most important modes analyzed are below.

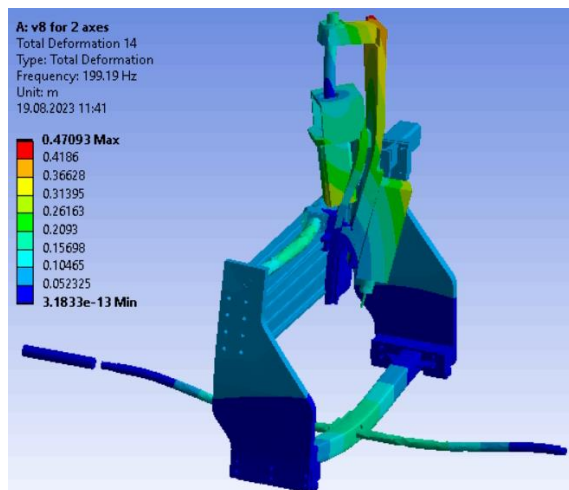
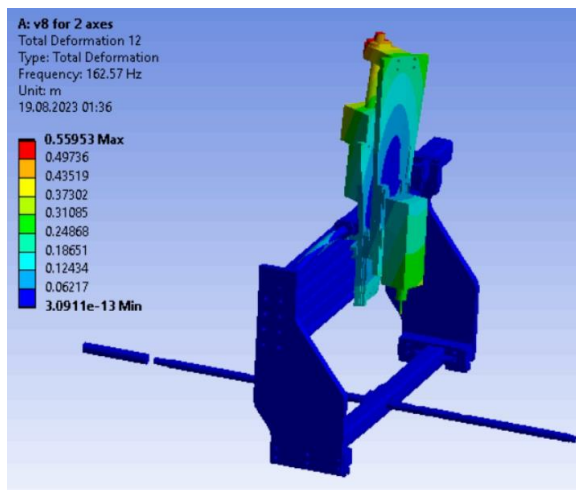
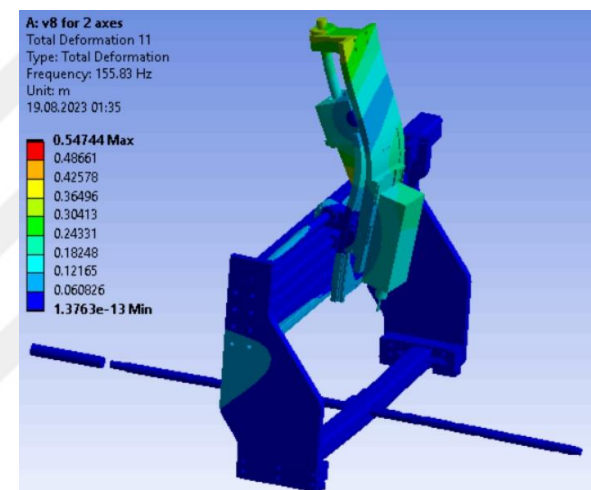
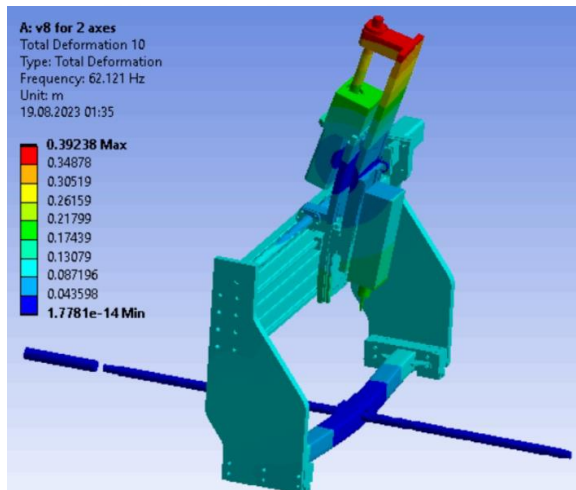
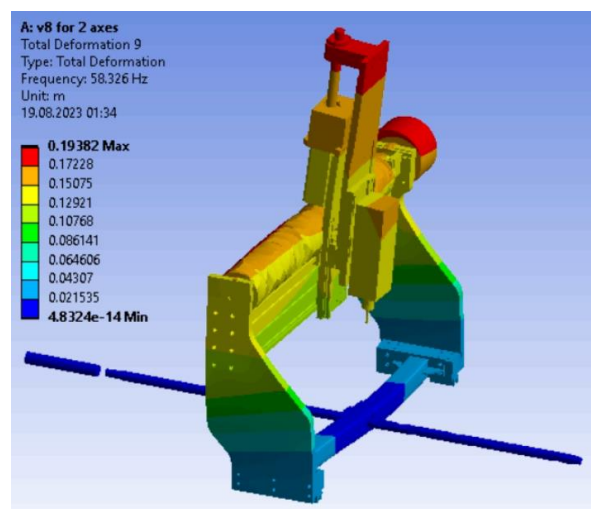
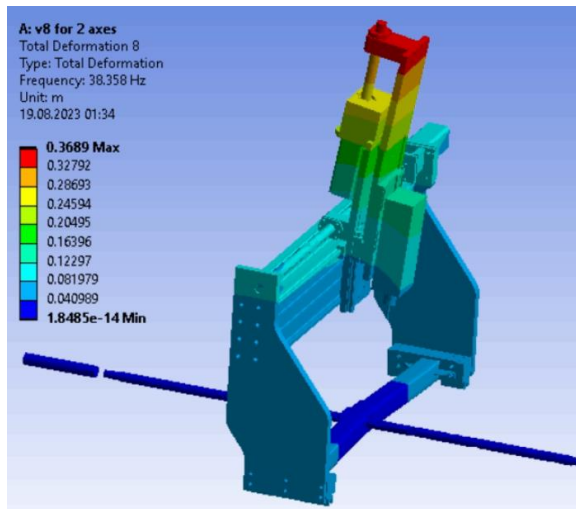


Figure 30 Mode shapes analyzed in ANSYS

By investigating the results, the natural frequencies are dense in some regions like, 38-62 Hz, 155-162 Hz, 200 Hz and 300 Hz. The most dominant modes in Y axis are 38 Hz, 62, 155 Hz and in X axis it is 58 Hz and 162 Hz. It is clear to see that the most vibrating region is the highest point, which is at the top of the Z-table plate. In order to analyze the FRF more clearly and to see the frequency response over a wide frequency range, a harmonic analysis was performed using the modal analysis results with the MSUP method.

The two FRF plots from separately X and Y axis motor torque to acceleration of the point are obtained as below.

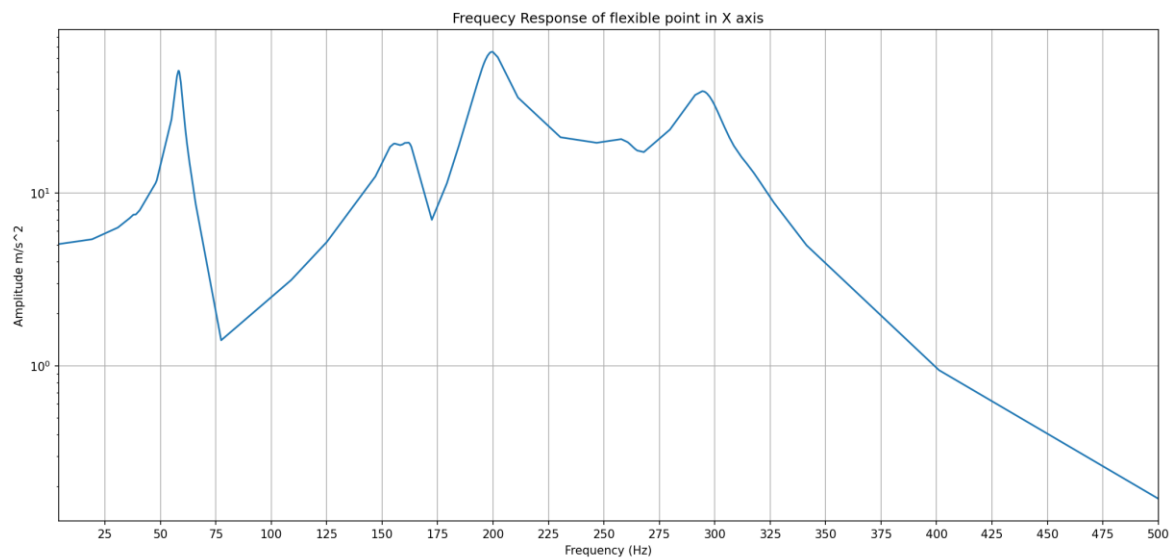


Figure 31 X axis frequency response of flexible point by ANSYS

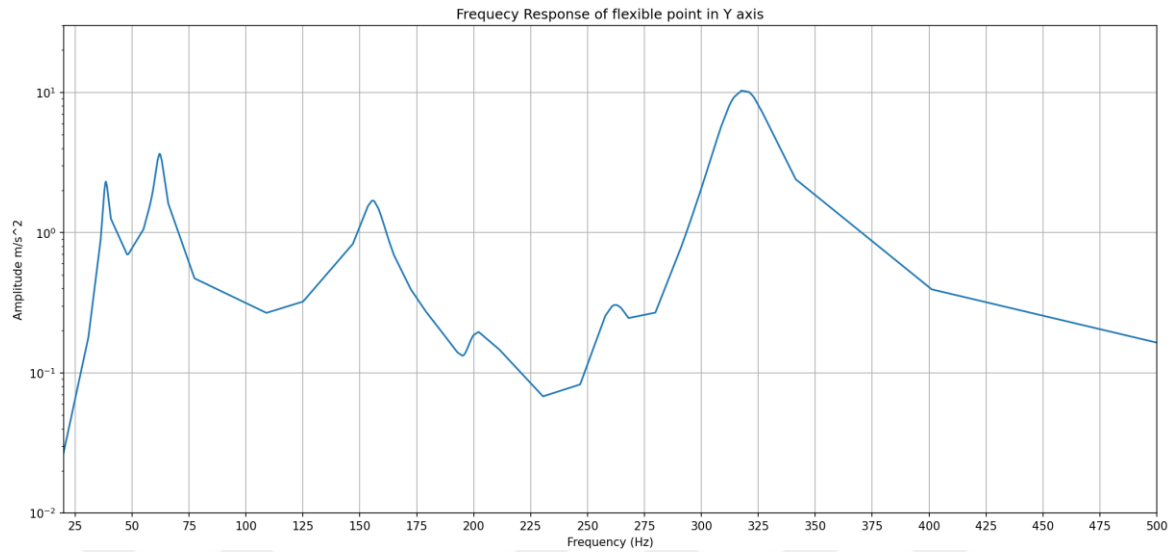


Figure 32 Y axis frequency response of flexible point by ANSYS

3.4. Power Spectrum Analysis of Machine Tool's with Vibration Tests

In the vibration test, impulse motor torque was applied to each axis and acceleration was read from the point with two different accelerometers. After that, PSD analysis on motor torque and acceleration was performed.

3.4.1. Positioning (air cut) test

Firstly, the axes were driven simultaneously to vibrate the machine corners. In the position control mode, axes are driven +200 mm and -200 mm with 150 mm/s with non-stop cornering to excite the machine vibrations. The servo system readings from the test are below.

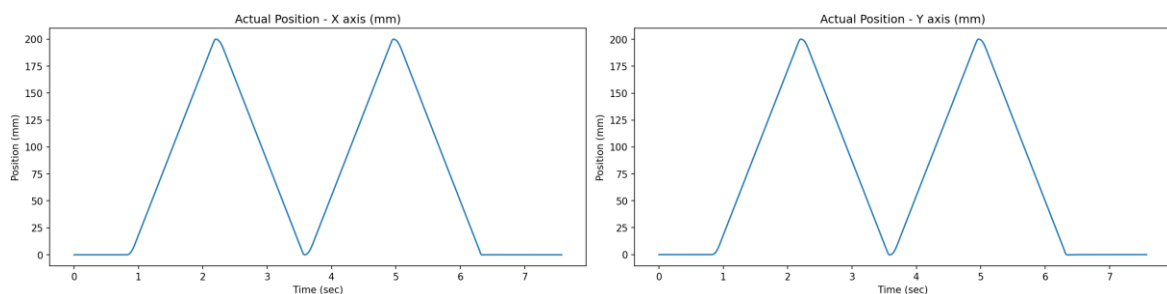


Figure 33 Positions of the axes while air cutting

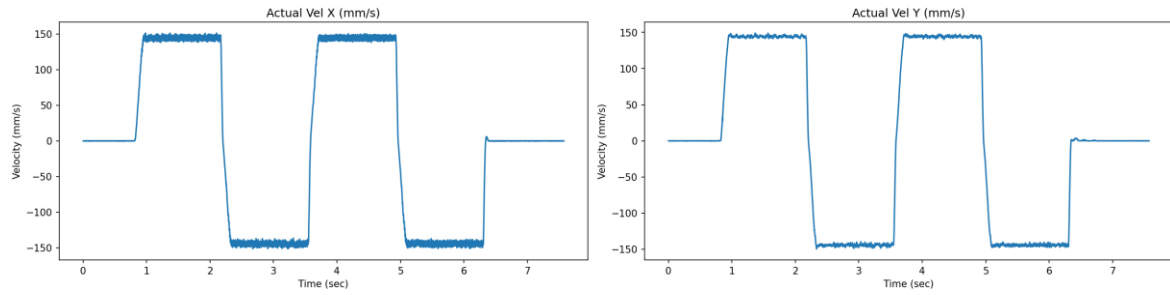


Figure 34 Velocities of the axes while air cutting

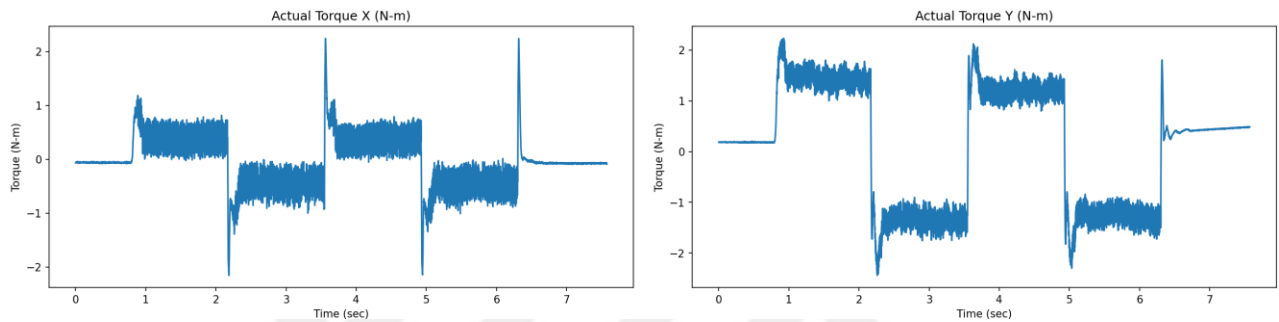


Figure 35 Motor torques of the axes while air cutting

According to these graphs, while the axes are moving between 0 and 200 mm non-stop cornering results in sudden torque changes. Before showing acceleration data from the sensors, the frequency content of torques should be revealed to see what range of frequency is excited in the axes. The simple FFTs of single peak of motor torques and velocity are below.

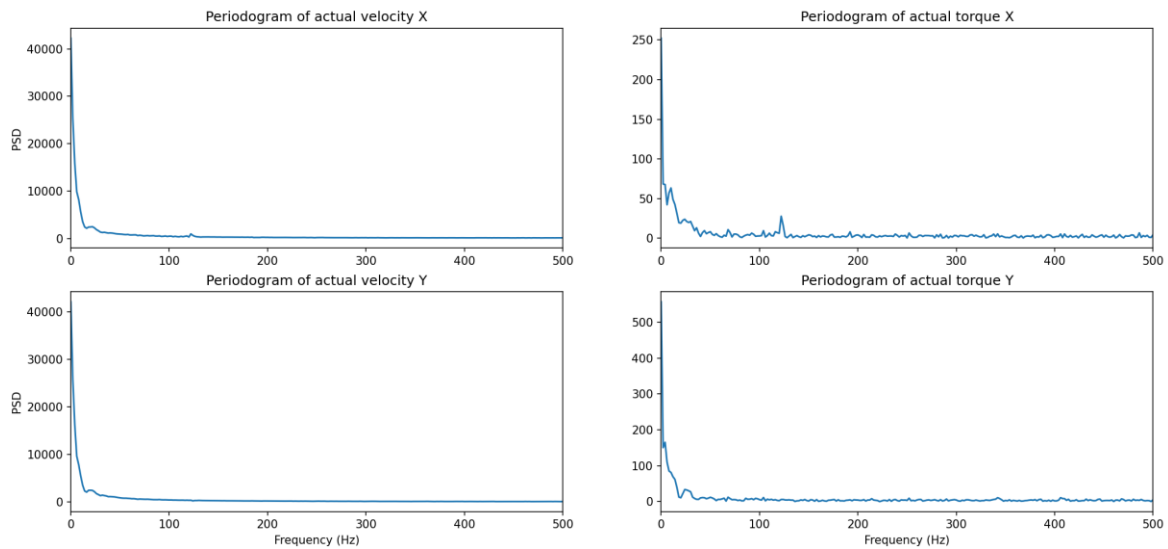


Figure 36 Periodograms of single peaks in velocity and torque readings in positioning test

From the results it is clear that the power is low for a narrow band of frequencies in the motor torque measurements. Therefore, the acceleration data will not be useful as the frequencies are not excited over a wide range. The reason for this low excitation is that during air cutting (positioning) in the corners, the trajectory generator smooths the movement and cannot be completely deactivated. Also, while the axes are moving, the velocity and torque readings from servo and acceleration reading from the sensor are very noisy due to the poor design of machine, geometric misalignments and the random friction on the axes. Therefore, instead, impulse test could be a better option since the axes will not move during the tests.

3.4.2. Impulse torque tests

Rather than exciting the axes in positioning (air cut), the axes are exposed to impulse torques that are frequently utilized in FRF studies thanks to their high range of frequency excitation. The servo drivers are fed with 9 N-m peak torque reference, but it is filtered in torque control loop that has motor electrical dynamics inside it, so the actual peak torque shows variety. The servo and sensor readings are shown below.

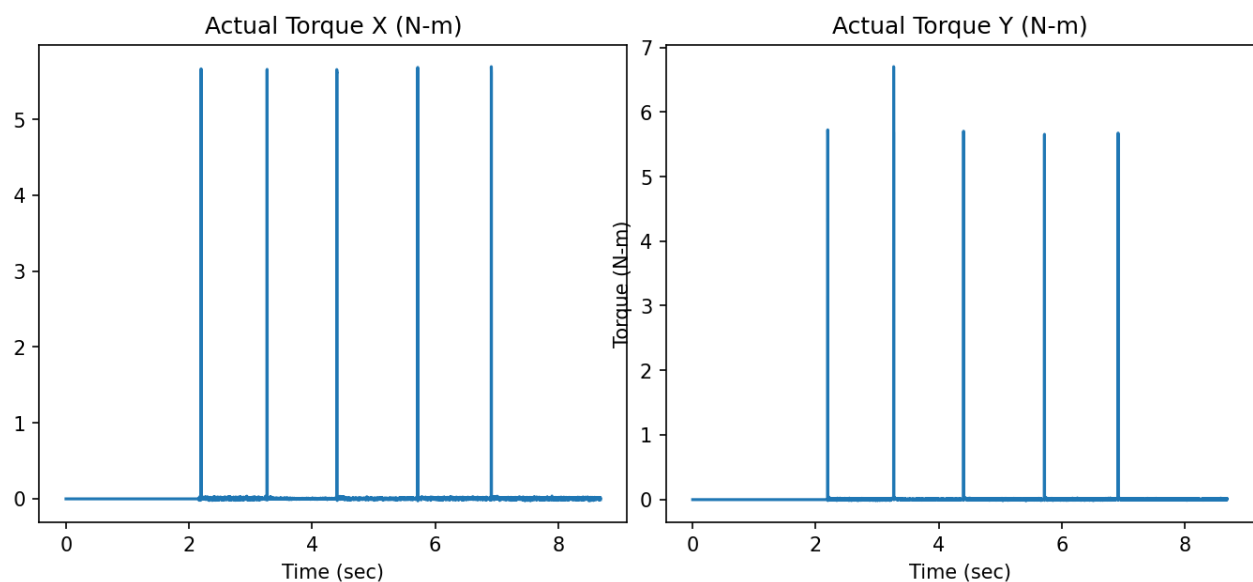


Figure 37 Applied torque impulses repeatedly on the axes

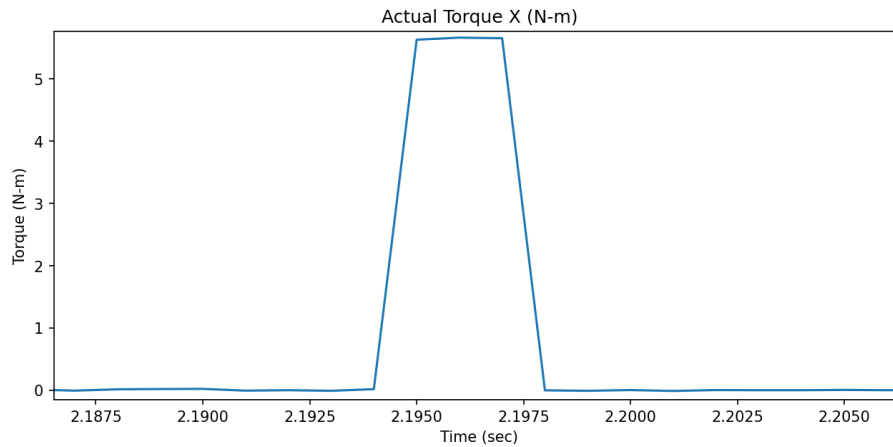


Figure 38 One of the impulses (zoomed)

The input excitation to axes consists of many impulses repeatedly. One of the impulses has a duration of 5 milliseconds. Against these impulses, the output acceleration read from sensors are below,

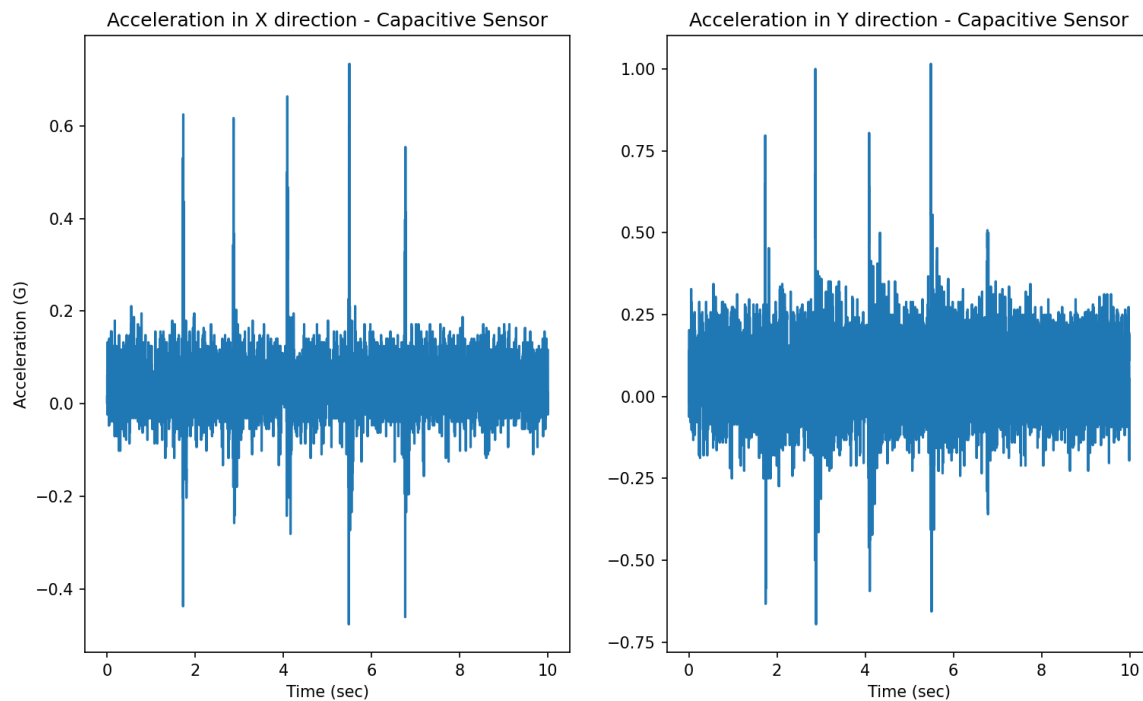


Figure 39 Acceleration read by capacitive sensor (ADXL345)

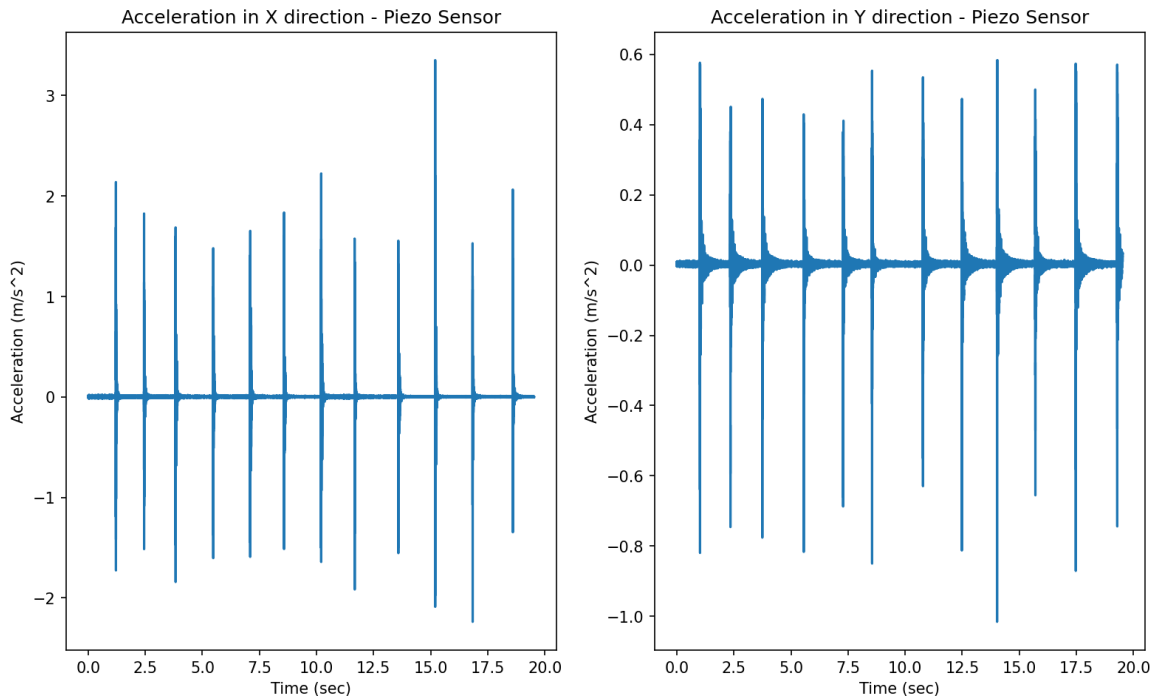


Figure 40 Acceleration read by piezo electric sensor (received from a different impulse test)

As it is clearly seen that the acceleration read by capacitive sensor is much noisier than by piezo sensor. The reason can be the sensitivity of the sensor. The capacitive sensor is less sensitive than piezo sensor. The torque applied to motors is damped in machine frame resulting in lower acceleration in capacitive sensor. So, the signal to noise ratio in capacitive sensor is obtained too low. On the other hand, utilized piezo sensor size was increased during the test until the signal to ratio becomes high and data becomes clearer. Using larger piezo sensor increases the sensitivity and it can even read smaller acceleration with high signal to noise ratio. However, the PSD analyzed are performed with two sensors despite the noise. The noise can be eliminated in the Welch's and Blackman-Tukey's methods for especially in capacitive sensor. The Welch's and Blackman Tukey method results in capacitive sensor's X and Y axis readings are below.

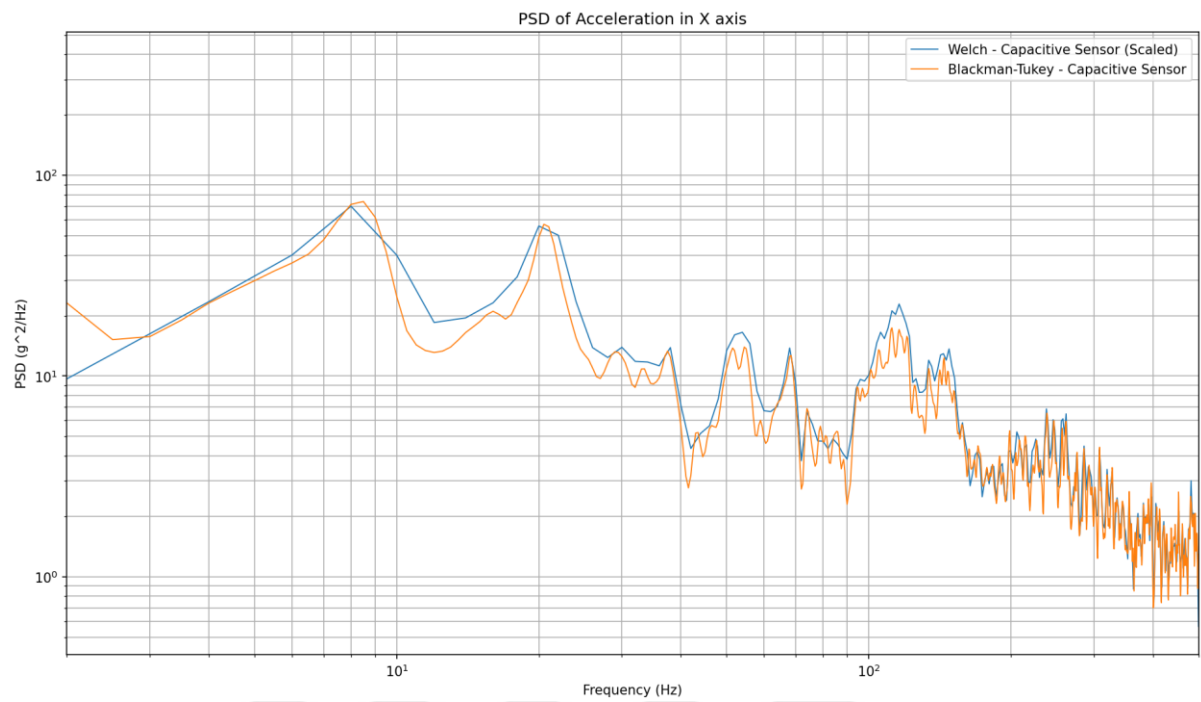


Figure 41 PSD of acceleration in X axis read by capacitive sensor

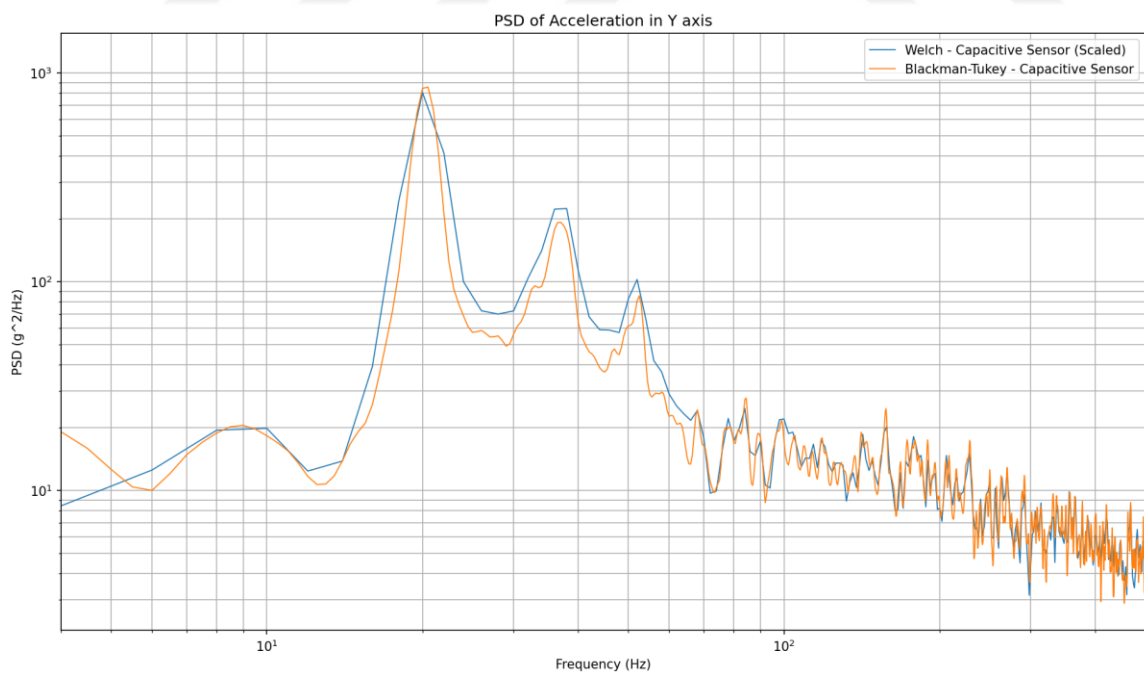


Figure 42 PSD analysis of acceleration in Y axis read by capacitive sensor

The results were averaged over many impulses. The two methods yield similar results so, they are reasonable. The comparison of capacitive sensor and piezo sensor is below.

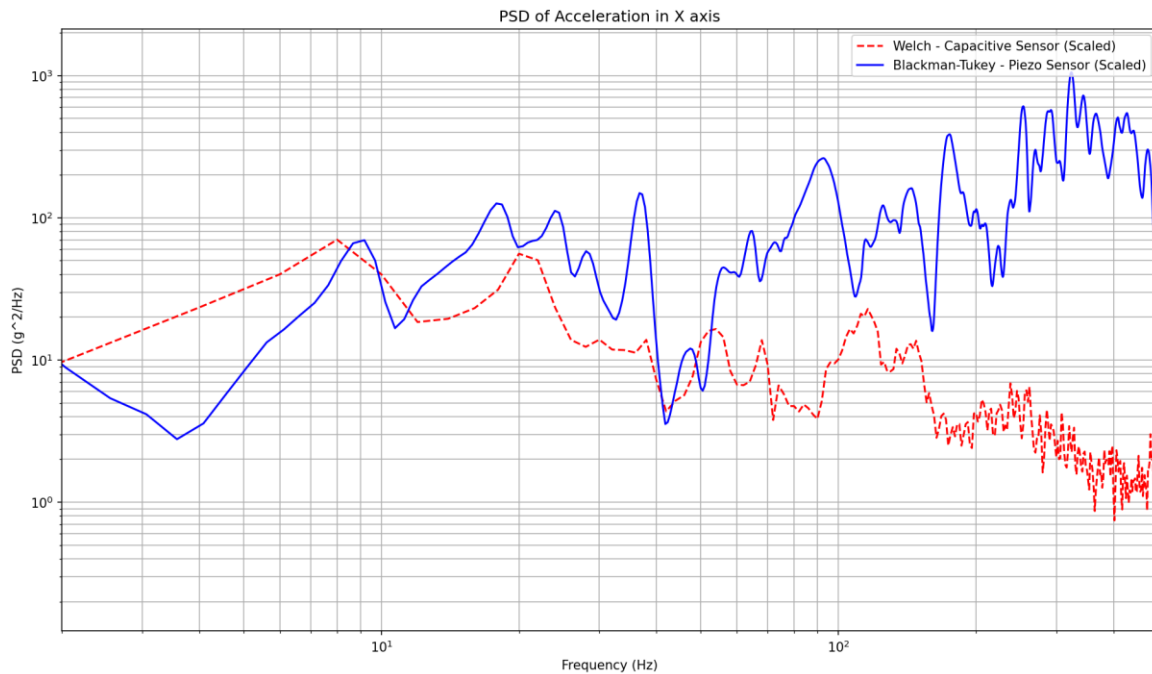


Figure 43 Comparison of capacitive sensor and piezo sensor PSDs in X axis

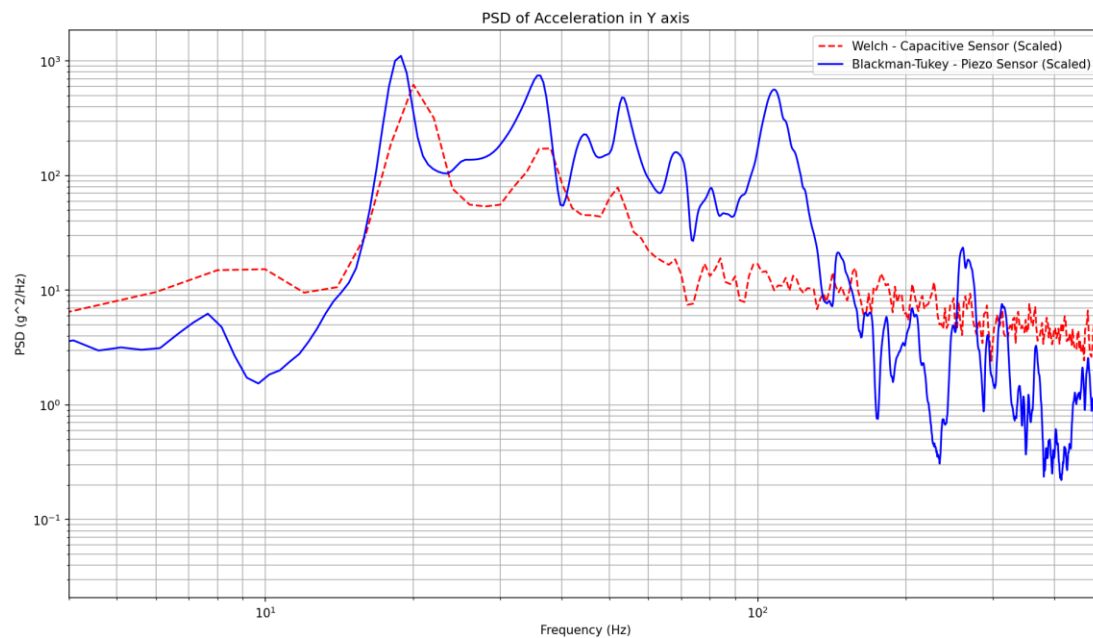


Figure 44 Comparison of capacitive sensor and piezo sensor PSDs in Y axis

Before reviewing the PSDs, the frequency content of input torque should be examined. PSDs of input torques obtained with Welch's method are below.

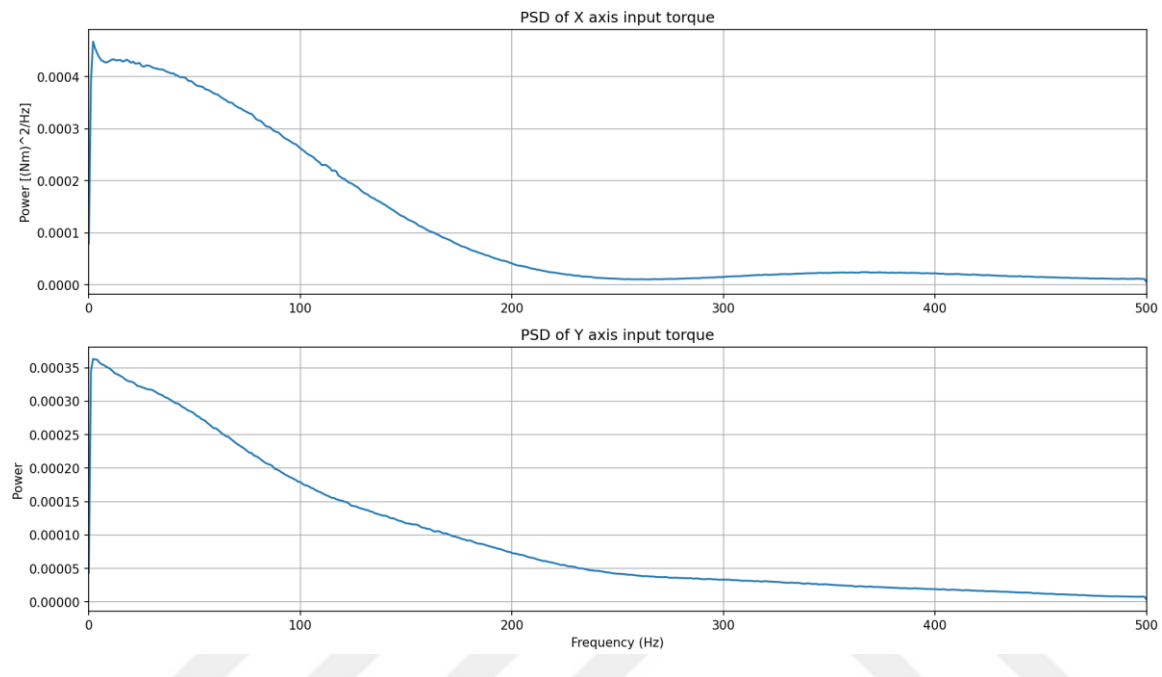


Figure 45 PSDs of input torques

Figure 45 illustrates that frequencies below ~200 Hz are excited in X axis and below ~350 Hz in Y axis. By analyzing Figure 43 and Figure 44, the trend is more similar in Y axis for both sensors than in X axis. In X axis, especially the low frequency responses are similar to each other in both sensors. Furthermore, piezo sensor captured higher frequencies than capacitive sensor did. However, frequencies above 350 Hz are not reliable because of the applied torque power spectrum (Figure 45).

The final comparison of commercial tool analysis and PSD analyzes of two sensors are below.

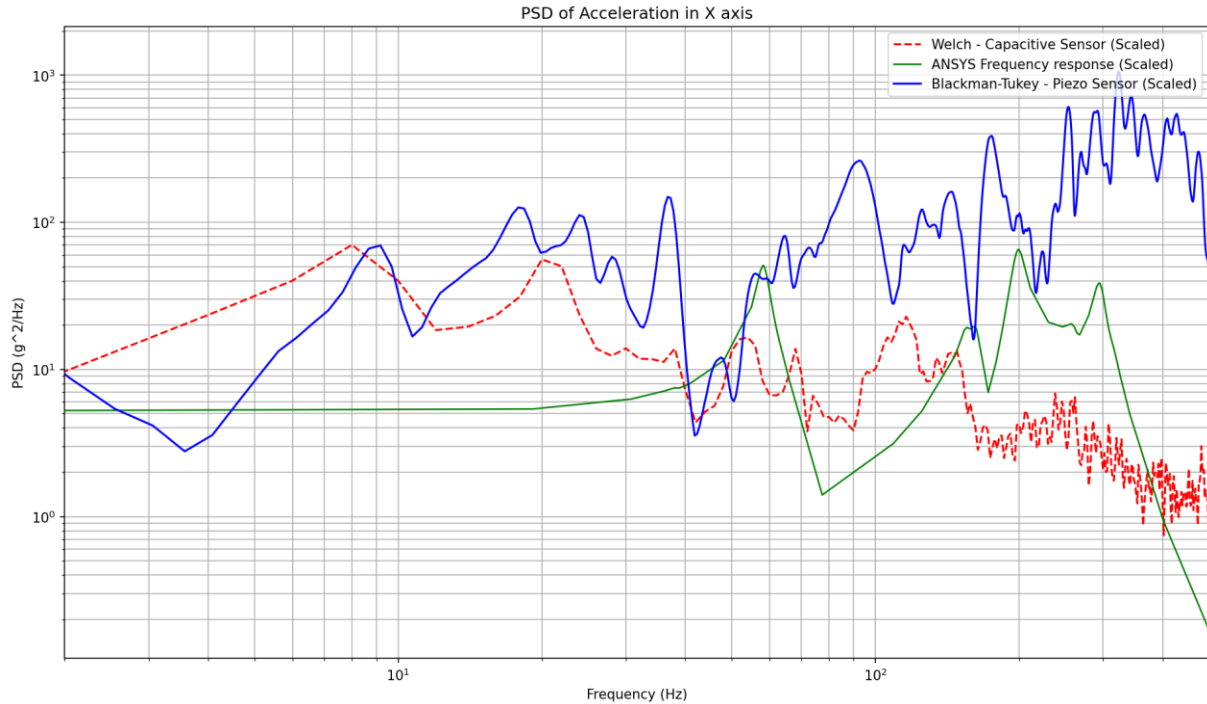


Figure 46 Comparison of ANSYS FRF and PSD analyzes of two sensor outputs in X axis

It should be noted that ANSYS result is an FRF plot whose unit is m/s^2 . The Power spectrum lines are different with the unit of g^2/Hz (Power/Frequency). So, exact fitting of the ANSYS FRF with the PSDs is not expected. However, this comparison gives common information about the locations of natural frequencies.

According to Figure 46, In X axis, there are some natural frequencies which are 8, 20 Hz caused by swaying of the machine tool on its legs. These frequencies are successfully captured by two sensors. However, in the modal analysis, the machine is estimated as fixed to the ground, so those frequencies could not be captured in ANSYS analyzes. The best match in natural frequency is around 60 Hz for three approaches. This frequency is the most dominant frequency according to mode shapes and FRF in ANSYS (Figure 30, Figure 31). Both sensors have an increase in amplitude of 38 Hz which is the dominant mode in Y axis although the measurements are on the X axis. Since the ANSYS measurement point is symmetric, 38 Hz vibration cannot be seen in ANSYS FRF of X axis (Figure 31). In actual tests, symmetry is not valid and the sensor axes can affect each other due to axis misalignment so 38 Hz vibration which is a dominant mode in Y axis, can be reflected to sensor in X axis PSD analyzes.

Furthermore, the piezo sensor PSD and ANSYS FRF tend to increase after approximately 150 Hz with a similar trend. Because the damping model in ANSYS is not well modelled, the ANSYS FRF amplitudes does not show similarity with sensor PSDs. ANSYS model can be improved by further detailing the damping, material and contact properties. For example, in the analyzes the rail cars are constrained with planar joint, however in real, cars have balls inside and have some stiffness. Also, the bolted connections are simplified to face bonding somewhere in the model to lower the solver load. Care should also be taken to lubricate moving parts while performing tests. So, they all affect the analysis accuracy of the machine. As well as, the X axis is more rigid than Y axis, so exciting the modes of by simply applying impulse torque is not easy with the current servo motors. Additionally, more tests can be performed to verify high frequencies among two sensors. The comparison of natural frequencies analyzed with ANSYS and two sensors may be shown as below table.

Table 9 Comparison of natural frequencies in X axis

Power Spectrum Density – Capacitive Sensor	8.3 Hz	20.6 Hz	38 Hz	52.9-68 Hz		113-144 Hz	-	247 Hz	-
Power Spectrum Density – Piezo Sensor	8.9 Hz	20.8 Hz	36 Hz	64 Hz	91 Hz	142 Hz	175 Hz		~330 Hz
ANSYS Harmonic	-	-	-	58 Hz		158 Hz	200 Hz	258 Hz	294 Hz

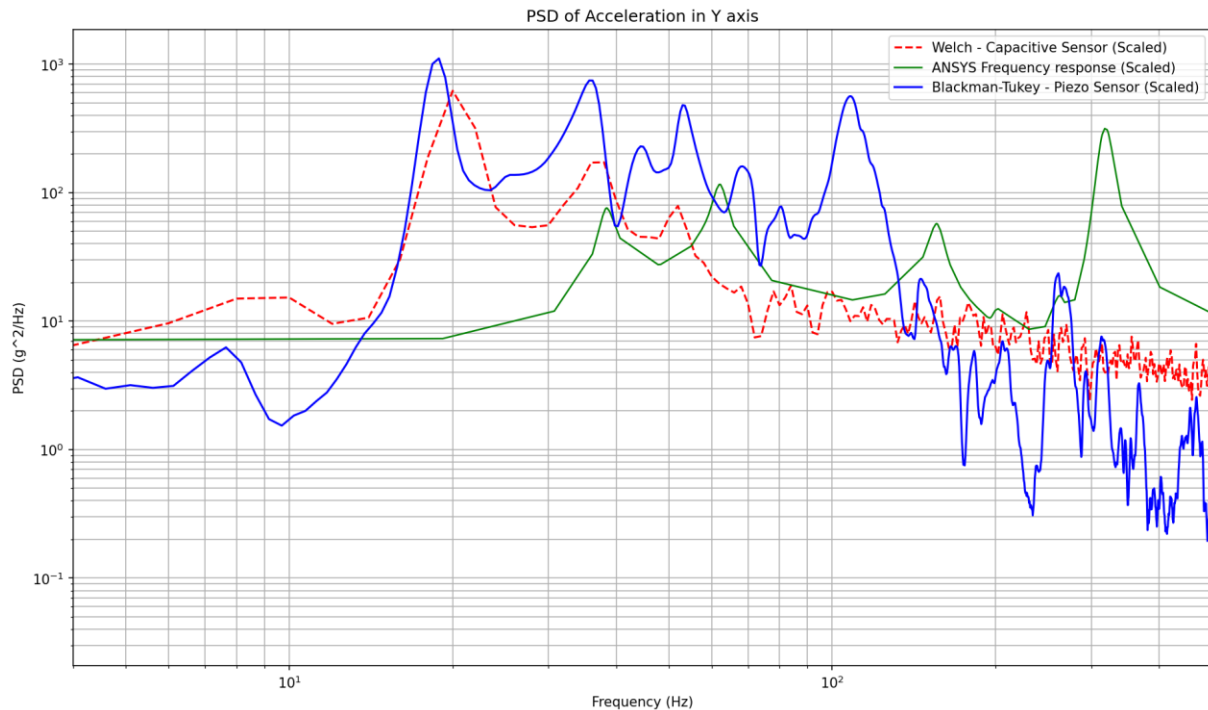


Figure 47 Comparison of ANSYS FRF and PSD analyzes of two sensor outputs in Y axis

From Figure 47, it can be stated that the ANSYS results are more similar to tests in Y axis than X axis. Because the Y axis flexibilities are larger so that can be easily read by sensors. Because of the machine's sway, natural frequencies of 8 Hz and 20 Hz read again from the sensors. Since the measurement direction is changed, amplitude of 8 Hz is decreased and amplitude of 20 Hz is increased compared to X axis. The best matches between three results are around 38 Hz and 50-65 Hz. Above 100 Hz, the results do not show a high similarity so, it is hard to comment for high frequencies. However, 107 Hz of piezo sensor and 150 Hz of ANSYS results may be related. As well as, a match analyzed in 260 Hz for piezo sensor PSD and ANSYS. According to Appendix E Figure 56, there is a twisting mode shape of Z table plate in 260 Hz in ANSYS. Its affect is seen as a small peak in ANSYS FRF of Y axis (Figure 32) due to symmetry of measurement point. However, the symmetry is not valid for real test and 260 Hz vibration reveals itself in piezo sensor PSD results (Figure 47). The final comparison of Y axis PSDs is below.

Table 10 Matching natural frequencies of Y axis by PSD and ANSYS

Power Spectrum Density - Capacitive sensor	8.7 Hz	20.3 Hz	37.4 Hz	51.1 Hz	-	-	-	-	-
Power Spectrum Density - Piezo sensor	9 Hz	19 Hz	36.1 Hz	53.12 - 66 Hz	107 Hz	-	-	258 Hz	-
ANSYS Harmonic	-	-	38 Hz	62 Hz	-	155 Hz	200 Hz	261.7 Hz	317 Hz

To conclude, three approaches show similarity in especially low frequencies. Both two sensors succeeded to capture the machine sway due to unfixed legs of machine. 38 Hz and 50-65 Hz natural frequencies successfully captured by all the three approaches. The higher frequencies generally do not follow a trend but in some frequencies, there are similarities.

In order to improve the PSD results, more appropriate excitation can be applied like hammer impulse. This test could be applied to directly to flexible point which is at the top of Z axis table plate. In this method the force applied force hammer must be measured and this requires advanced test equipment. However, if an FRF between input torque and output acceleration is desired, hammer may be a disadvantageous method since the excitation should be a torque on feed drive mechanism.

Another and most powerful method of analyzing FRF is testing in discrete frequencies separately. The system is separately exposed to one harmonic torque at a time. After each constant frequency torque test, the acceleration data is read and FRF is created with discrete test results. Each discrete torque frequency corresponds to an acceleration amplitude.

Also, the machine is not stable because it is manufactured and assembled from many parts and has too much nonlinearities. This might affect the impulse torque test because of high Coulomb friction since it creates discontinuities in motion and directly affects FRF. These discontinuities

might cause high variance between the tests. It makes it difficult to simulate the real machine in ANSYS while the machine consists of too many parts and fittings.

There are several methods available to analyze the ball screw mechanism only. Some of them were attempted during this study. Defining contact in ball screw-nut interface makes the model too complex and makes difficult to solve. Also, it requires detailed CAD model. Instead of defining contact, in order to define the flexibility in ball screw and nut interface, an axial and torsional springs between ball screw and nut can be placed, however this model is too simple, so it lacks of defining coupling between axial and torsional dynamics. Under the light of these attempts, in this study, it is decided that the MPC184 constraint element is very useful and not too complex to model the coupling between axial and torsional dynamics in ball screw-nut interface. Therefore, the MPC184 element in ANSYS APDL is more efficient for modelling the ball screw.

As a future work, some studies will be performed. Since the piezo sensor is widely used in industry, more tests will be realized with it until the repeated tests satisfy each other. Coherence will be observed for results. On the other hand, tests with capacitive sensor can be realized to compare with piezo sensor. Also, only one axis will be excited during the tests because the axes affect the sensor in different directions makes it more difficult to analyze spectrum. Furthermore, the spectrum of torque should be divided to spectrum of acceleration to get a real relation between them. After getting more reliable spectrums, transfer function between input torque and output acceleration can be calculated via Rational Fraction Polynomial method from discrete frequency-amplification data. This model can be used for simulations and also damping of vibrations.

REFERENCES

- [1] A. H. H. Hosseinabadi, "Modeling and Active Damping of Structural Dynamics in CNC Machine Tools," The University of British Columbia, Vancouver, 2011.
- [2] K. Erkorkmaz and Y. Altıntaş, "High speed CNC system design. Part I: jerk limited trajectory generation and quintic spline interpolation," *International Journal of Machine Tools & Manufacture*, vol. 41, pp. 1323-1345, 2001.
- [3] B. Sencer and N. Suzuki, "Trajectory generation for CNC machine tools," *Journal of the Japan Society for Precision Engineering*, vol. 79, pp. 631-637, 2013.
- [4] P. Besset and R. Bearee, "FIR filter based online jerk constrained trajectory generation," *Control Engineering Practice*, 2017.
- [5] M. H. Turhan, G. W. G. Tseng, K. Erkorkmaz and B. Fidan, "Dynamic model identification for CNC machine tool feed drives from in-process signals for virtual process planning," *Mechatronics*, vol. 72, no. 102445, 2020.
- [6] M. S. Tsai, Y. C. Huang, M. T. Lin and S. K. Wu, "Integration of input shaping technique with interpolation for vibration suppression of servo-feed drive system," *Journal of the Chinese Institute of Engineers*, 2017.
- [7] C. C. Liu, M. S. Tsai, M. Q. Hong and P. Y. Tang, "Development of a Novel Tuning Approach of the Notch Filter of the Servo Feed Drive System," *Journal of Manufacturing and Materials Processing*, 2020.
- [8] K. Erkorkmaz and A. Kamalzadeh, "High Bandwidth Control Of Ball Screw Drives," *Annals of the CIRP*, vol. 565/1, 2006.
- [9] Y. Hosseinkhani, *Control Methods for Improving Tracking Accuracy and Disturbance*

Rejection in Ball Screw Feed Drives, Waterloo, Ontario, Canada: The University of Waterloo, 2013.

- [10] K. Erkorkmaz and Y. Altintas, "High speed CNC system design. Part II: modeling and identification of feed drives," *International Journal of Machine Tools & Manufacture*, vol. 41, pp. 1487-1509, 2001.
- [11] H. B. Armstrong, P. Dupont and A. Canudas De Wit, "Survey of models, analysis tools and compensation methods," *Automatica*, vol. 30, no. 7, pp. 1083-1138, 1994.
- [12] K. K. Varanasi and S. A. Nayfeh, "The Dynamics of Lead-Screw Drives: Low-Order Modeling and Experiments," *Journal of Dynamic Systems Measurement and Control*, vol. 126, 2004.
- [13] C. Pislaru, D. G. Ford and G. Holroyd, "Hybrid modelling simulation of a compute numerical control machine tool feed drive," *Journal of Systems and Control Engineering*, vol. 218, no. 11, pp. 111-120, 2004.
- [14] C. E. Okwudire and Y. Altıntaş, "Hybrid Modelling of Ball Screw Drives With Coupled Axial, Torsional, and Lateral Dynamics," *Journal of Mechanical Design*, vol. 131, 2009.
- [15] P. Hong, Y. Pan and T. Zhang, "Dynamic Characteristics Analysis of Ball Screw Based on Workbench," *American International Journal of Contemporary Research*, vol. 5, no. 3, 2015.
- [16] L. X. Zhou and P. Y. Li, "Finite element analysis of the axial stiffness of a ball screw," *International Conference on Material Strength and Applied Mechanics*, vol. 372, no. 012023, 2018.
- [17] M. F. Zaeh, T. Oertli and J. Milberg, "Finite Element Modelling of Ball Screw Feed Drive Systems," *CIRP Annals*, vol. 53, pp. 289-292, 2004.
- [18] D. A. Smith, *WIDE BANDWIDTH CONTROL OF HIGH-SPEED MILLING MACHINE*

FEED DRIVES, Doctoral Dissertation, Florida: University of Florida, 1999.

- [19] S. Technics, "SM08-DX-075BX," SMB Technics, [Online]. Available: <https://www.smbtechnics.com/urun/sm08-dx-075bx-2/>. [Accessed 10 August 2023].
- [20] GTEN, "Lead in manufacture of precise ball screw hardware".
- [21] E. Demir, A. İ. Köse, S. Ç. Başlamışlı, B. N. Yüksel, B. Doğan, O. Tunçer, U. Şimşir and Y. Altıntaş, "CNC System Design Part II: Construction of a Three Axis Test Rig and System Identification," in *Otomatik Kontrol Ulusal Toplantısı*, Elazığ, 2022.
- [22] J. S. PRZEMIENIECKI, THEORY OF MATRIX STRUCTURAL ANALYSIS, New York: DOVER PUBLICATIONS INC, 1968.
- [23] ANSYS Inc., "ANSYS Mechanical APDL Element Reference," Canonsburg, 2010.
- [24] Y. Altintas, A. Verl, C. Brecher, L. Uriarte and G. Pritschow, "Machine tool feed drives," *CIRP ANN-MANUF TECHNOL*, 2011.
- [25] Wikipedia contributors, "Bartlett's method," Wikipedia, 4 May 2023. [Online]. Available: https://en.wikipedia.org/w/index.php?title=Bartlett%27s_method&oldid=1153184680. [Accessed 15 August 2023].
- [26] R. Whalley, M. Ebrahimi and A. Abdul-Ameer, "Hybrid modelling of machine tool axis drives," *International Journal of Machine Tools & Manufacture*, vol. 45, pp. 1560-1576, 2005.

APPENDIX

A. Ball Screw Mechanism Finite Element Analysis Codes in MATLAB

```
clc,clear,close all
format shortG
% Ref: Chinedum Okwudire: Finite element modelling of ballscrew feed drive
% systems for control purposes
% Ref: David Alan Smith: Wide bandwidth control of high-speed milling
% machine feed drives

% Analysis Settings

% Lead screw properties
E = 2.1e+11; % Modulus of elasticity, Pa
G = 8.0e+10; % Shear modulus, Pa
rho = 7850; % Density of steel, kg/m^3
de_ballscrew = 0.0225; % Mean Dia of ballscrew, m
d_rotor = 0.02956; % Dia of rotor, m
L_BallScrew = 1.2; % Length of ballscrew, m
L_Rotor = 0.18648; % Length of rotor, m
mt = 64.151; % Table mass, kg
r = 0.005/(2*pi); % transmission ratio of ball screw , ws * L =
2pi * vn , wn = R * vn
kBN = 3.6E7; % Thrust stiffness of nut N/m ---- 3.6e7 N/m
from Smith's thesis
kCoupler_Axial = 1e12; % Coupler stiffness properties
kCoupler_Torsional = 5e12;
kBearing = 4e9; % Thrust bearing stiffness

% Element numbers
N_elm_BallScrew = 200;
N_elm_Rotor = 10;

L_elm_BallScrew = L_BallScrew / N_elm_BallScrew; % Lenght of BallScrew elements,
m
L_elm_Rotor = L_Rotor/ N_elm_Rotor; % Lenght of Rotor elements, m

% Create nodes
Nodes_BallScrew = 1:(N_elm_BallScrew+1); % Ballscrew nodes
Nodes_Rotor = Nodes_BallScrew(end):(Nodes_BallScrew(end)+N_elm_Rotor+1); % Rotor
nodes
BearingNode = [1,N_elm_BallScrew]; % Bearing node position
NutSwitch = 1; % 1: there is ball nut, 0: no ball-nut, only the ball screw
NutNode = N_elm_BallScrew/2; % Ball nut node position
x_nut = L_elm_BallScrew * (NutNode-1); % Ball nut distance from the origin
% Area and area moment of inertias
A_elm_ballscrew = pi * (de_ballscrew / 2)^2; % Cross sectional area of screw,
m^2
A_elm_rotor = pi * (d_rotor / 2)^2; % Cross sectional area of rotor,
m^2
J_ballscrew = pi / 2 * (de_ballscrew / 2)^4; % Area moment of inertia or second
moment of area of inertia, m^4
```

```

J_rotor = pi / 2 * (d_rotor / 2)^4;

% DOF number
DOF = (N_elm_BallScrew + N_elm_Rotor + 2) * 2 + 1 * NutSwitch;

% Create Stiffness matrix of model,
%%%%%%%%% Euler Bernoulli beam
K_elm = zeros(DOF);
M_elm = zeros(DOF);

for currentElement=1:N_elm_BallScrew           % Ballscrew
    n=2*(currentElement-1);

    % Euler-bernoulli: Stiffness
    k11 = E*A_elm_ballscREW/L_elm_BallScrew;
    k12 = 0 ;
    k13 = -E*A_elm_ballscREW/L_elm_BallScrew;
    k14 = 0;

    k21 = 0;
    k22 = G*J_ballscREW/L_elm_BallScrew;
    k23 = 0;
    k24 = -G*J_ballscREW/L_elm_BallScrew;

    k31 = -E*A_elm_ballscREW/L_elm_BallScrew;
    k32 = 0;
    k33 = E*A_elm_ballscREW/L_elm_BallScrew;
    k34 = 0;

    k41 = 0;
    k42 = -G*J_ballscREW/L_elm_BallScrew;
    k43 = 0;
    k44 = G*J_ballscREW/L_elm_BallScrew;

    K_elm(1+n,1+n) = K_elm(1+n,1+n) + k11;
    K_elm(1+n,2+n) = K_elm(1+n,2+n) + k12;
    K_elm(1+n,3+n) = K_elm(1+n,3+n) + k13;
    K_elm(1+n,4+n) = K_elm(1+n,4+n) + k14;
    K_elm(2+n,1+n) = K_elm(2+n,1+n) + k21;
    K_elm(2+n,2+n) = K_elm(2+n,2+n) + k22;
    K_elm(2+n,3+n) = K_elm(2+n,3+n) + k23;
    K_elm(2+n,4+n) = K_elm(2+n,4+n) + k24;
    K_elm(3+n,1+n) = K_elm(3+n,1+n) + k31;
    K_elm(3+n,2+n) = K_elm(3+n,2+n) + k32;
    K_elm(3+n,3+n) = K_elm(3+n,3+n) + k33;
    K_elm(3+n,4+n) = K_elm(3+n,4+n) + k34;
    K_elm(4+n,1+n) = K_elm(4+n,1+n) + k41;
    K_elm(4+n,2+n) = K_elm(4+n,2+n) + k42;
    K_elm(4+n,3+n) = K_elm(4+n,3+n) + k43;
    K_elm(4+n,4+n) = K_elm(4+n,4+n) + k44;

    % Euler-bernoulli: Mass
    m11 = 2*A_elm_ballscREW*rho*L_elm_BallScrew/6;
    m12 = 0 ;

```

```

m13 = A_elm_ballscREW*rho*L_elm_BallScrew/6;
m14 = 0;

m21 = 0;
m22 = 2*J_ballscREW*rho*L_elm_BallScrew/6;
m23 = 0;
m24 = J_ballscREW*rho*L_elm_BallScrew/6;

m31 = A_elm_ballscREW*rho*L_elm_BallScrew/6;
m32 = 0;
m33 = 2*A_elm_ballscREW*rho*L_elm_BallScrew/6;
m34 = 0;

m41 = 0;
m42 = J_ballscREW*rho*L_elm_BallScrew/6;
m43 = 0;
m44 = 2*J_ballscREW*rho*L_elm_BallScrew/6;

M_elm(1+n,1+n) = M_elm(1+n,1+n) + m11;
M_elm(1+n,2+n) = M_elm(1+n,2+n) + m12;
M_elm(1+n,3+n) = M_elm(1+n,3+n) + m13;
M_elm(1+n,4+n) = M_elm(1+n,4+n) + m14;
M_elm(2+n,1+n) = M_elm(2+n,1+n) + m21;
M_elm(2+n,2+n) = M_elm(2+n,2+n) + m22;
M_elm(2+n,3+n) = M_elm(2+n,3+n) + m23;
M_elm(2+n,4+n) = M_elm(2+n,4+n) + m24;
M_elm(3+n,1+n) = M_elm(3+n,1+n) + m31;
M_elm(3+n,2+n) = M_elm(3+n,2+n) + m32;
M_elm(3+n,3+n) = M_elm(3+n,3+n) + m33;
M_elm(3+n,4+n) = M_elm(3+n,4+n) + m34;
M_elm(4+n,1+n) = M_elm(4+n,1+n) + m41;
M_elm(4+n,2+n) = M_elm(4+n,2+n) + m42;
M_elm(4+n,3+n) = M_elm(4+n,3+n) + m43;
M_elm(4+n,4+n) = M_elm(4+n,4+n) + m44;
end
for currentElement=(N_elm_BallScrew+1):(N_elm_BallScrew+N_elm_Rotor) % Rotor
n=2*currentElement+1;

% Euler-bernoulli: Stiffness, Rotor
k11 = E*A_elm_rotor/L_elm_Rotor;
k12 = 0 ;
k13 = -E*A_elm_rotor/L_elm_Rotor;
k14 = 0;

k21 = 0;
k22 = G*J_rotor/L_elm_Rotor;
k23 = 0;
k24 = -G*J_rotor/L_elm_Rotor;

k31 = -E*A_elm_rotor/L_elm_Rotor;
k32 = 0;
k33 = E*A_elm_rotor/L_elm_Rotor;
k34 = 0;

```

```

k41 = 0;
k42 = -G*J_rotor/L_elm_Rotor;
k43 = 0;
k44 = G*J_rotor/L_elm_Rotor;

K_elm(0+n,0+n) = K_elm(0+n,0+n) + k11;
K_elm(0+n,1+n) = K_elm(0+n,1+n) + k12;
K_elm(0+n,2+n) = K_elm(0+n,2+n) + k13;
K_elm(0+n,3+n) = K_elm(0+n,3+n) + k14;
K_elm(1+n,0+n) = K_elm(1+n,0+n) + k21;
K_elm(1+n,1+n) = K_elm(1+n,1+n) + k22;
K_elm(1+n,2+n) = K_elm(1+n,2+n) + k23;
K_elm(1+n,3+n) = K_elm(1+n,3+n) + k24;
K_elm(2+n,0+n) = K_elm(2+n,0+n) + k31;
K_elm(2+n,1+n) = K_elm(2+n,1+n) + k32;
K_elm(2+n,2+n) = K_elm(2+n,2+n) + k33;
K_elm(2+n,3+n) = K_elm(2+n,3+n) + k34;
K_elm(3+n,0+n) = K_elm(3+n,0+n) + k41;
K_elm(3+n,1+n) = K_elm(3+n,1+n) + k42;
K_elm(3+n,2+n) = K_elm(3+n,2+n) + k43;
K_elm(3+n,3+n) = K_elm(3+n,3+n) + k44;

% Euler-bernoulli: Mass, Rotor
m11 = 2*A_elm_rotor*rho*L_elm_Rotor/6;
m12 = 0 ;
m13 = A_elm_rotor*rho*L_elm_Rotor/6;
m14 = 0;

m21 = 0;
m22 = 2*J_rotor*rho*L_elm_Rotor/6;
m23 = 0;
m24 = J_rotor*rho*L_elm_Rotor/6;

m31 = A_elm_rotor*rho*L_elm_Rotor/6;
m32 = 0;
m33 = 2*A_elm_rotor*rho*L_elm_Rotor/6;
m34 = 0;

m41 = 0;
m42 = J_rotor*rho*L_elm_Rotor/6;
m43 = 0;
m44 = 2*J_rotor*rho*L_elm_Rotor/6;

M_elm(0+n,0+n) = M_elm(0+n,0+n) + m11;
M_elm(0+n,1+n) = M_elm(0+n,1+n) + m12;
M_elm(0+n,2+n) = M_elm(0+n,2+n) + m13;
M_elm(0+n,3+n) = M_elm(0+n,3+n) + m14;
M_elm(1+n,0+n) = M_elm(1+n,0+n) + m21;
M_elm(1+n,1+n) = M_elm(1+n,1+n) + m22;
M_elm(1+n,2+n) = M_elm(1+n,2+n) + m23;
M_elm(1+n,3+n) = M_elm(1+n,3+n) + m24;
M_elm(2+n,0+n) = M_elm(2+n,0+n) + m31;
M_elm(2+n,1+n) = M_elm(2+n,1+n) + m32;
M_elm(2+n,2+n) = M_elm(2+n,2+n) + m33;

```

```

M_elm(2+n,3+n) = M_elm(2+n,3+n) + m34;
M_elm(3+n,0+n) = M_elm(3+n,0+n) + m41;
M_elm(3+n,1+n) = M_elm(3+n,1+n) + m42;
M_elm(3+n,2+n) = M_elm(3+n,2+n) + m43;
M_elm(3+n,3+n) = M_elm(3+n,3+n) + m44;
end

% Coupling of BallScrew and Rotor
K_coupler =[      kCoupler_Axial      0      -kCoupler_Axial      0
                0      kCoupler_Torsional      0      -
kCoupler_Torsional
                -kCoupler_Axial      0      kCoupler_Axial      0
                0      -kCoupler_Torsional      0
kCoupler_Torsional];
row =(N_elm_BallScrew+1)*2-1;
K_elm(row:(row+3),row:(row+3)) = K_elm(row:(row+3),row:(row+3)) + K_coupler;

% Assemble thrust bearing
for i=1:length(BearingNode)
    row = BearingNode(i)*2-1;
    column=row;
    K_elm(row,column)=K_elm(row,column) + kBearing;
end

% Assemble Ball Nut on ball screw
if NutSwitch == 1
    K_elm(2*NutNode-1,2*NutNode-1) = K_elm(2*NutNode-1,2*NutNode-1) + 1 * kBN;
    K_elm(2*NutNode-1,2*NutNode) = K_elm(2*NutNode-1,2*NutNode) - r * kBN;
    K_elm(2*NutNode-1,DOF) = K_elm(2*NutNode-1,DOF) - 1 * kBN;

    K_elm(2*NutNode,2*NutNode-1) = K_elm(2*NutNode,2*NutNode-1) - r * kBN;
    K_elm(2*NutNode,2*NutNode) = K_elm(2*NutNode,2*NutNode) + r^2 * kBN;
    K_elm(2*NutNode,DOF) = K_elm(2*NutNode,DOF) + r * kBN;

    K_elm(DOF,2*NutNode-1) = K_elm(DOF,2*NutNode-1) - 1 * kBN;
    K_elm(DOF,2*NutNode) = K_elm(DOF,2*NutNode) + r * kBN;
    K_elm(DOF,DOF) = K_elm(DOF,DOF) + 1 * kBN;

    M_elm(DOF,DOF) = mt;
end

% Eigen value analysis
eigs = eig(K_elm,M_elm);
[eigen_vecs,eigen_vals] = eig(K_elm,M_elm);
wn = sqrt(diag(real(eigen_vals))) / (2 * pi); % natural frequencies Hz
% scale eigen vectors
[big_eigen_vecs_linear,which_big_linear] = max(abs(eigen_vecs(1:2:DOF,:)));
[big_eigen_vecs_rotary,which_big_rotary] = max(abs(eigen_vecs(2:2:DOF,:)));
for i=1:length(wn)
    if (big_eigen_vecs_linear(i) > r * big_eigen_vecs_rotary(i))
        scale(i) = eigen_vecs(2*which_big_linear(i)-1 , i)/r;
    else
        scale(i) = eigen_vecs(2*which_big_rotary(i) , i);
    end
end

```

```

end
eigen_vecs = eigen_vecs * diag(1 ./ scale);
% Plot modeshapes
if DOF >= 10
    modeShapetoPlot = 10;
else
    modeShapetoPlot = DOF;
end
x_ballscREW= linspace(0,L_BallScrew,N_elm_BallScrew+1);
L_coupling = 0.05;
x_rotor=linspace(x_ballscREW(end)+L_coupling,x_ballscREW(end)+L_coupling+L_Rotor,N
_elm_Rotor+1);
x=[x_ballscREW x_rotor];
for i=1:length(x)
    if mod(i,2) == 0
        continue
    else
        x_compressed(1,(i+1)/2) = x(i);
    end
end
for i=1:modeShapetoPlot
    figure(i)
    subplot(3,1,1) % Axial mode shape
    y=eigen_vecs(1:2:(DOF-1),i)';
    plot(x,y,"b","linewidth",1.5)
    hold on
    if NutSwitch ==1
        plot(x_nut, eigen_vecs(DOF,i), "ro",'LineWidth',2.5)
    end
    yline(0)
    title(["Natural Freq. of The Mode Shape: " + wn(i)+" Hz","Axial Mode Shape"]),
grid on
    xlim([x(1),x(end)])
    xticks(x_compressed)
    subplot(3,1,2) % Torsional
    y=eigen_vecs(2:2:end,i)';
    plot(x,y,"b","linewidth",1.5)
    hold on
    yline(0)
    title('Torsional Mode Shape'), grid on
    xlim([x(1),x(end)])
    xticks(x_compressed)
    %% Plot geometry
    subplot(3,1,3)
    xlabel('Node Position (m)')
    hold on
    % BallscREW geometry
    for i=1:N_elm_BallScrew
        h=fill([x_ballscREW(i),x_ballscREW(i+1),x_ballscREW(i+1),x_ballscREW(i)],[0,0,de_b
allscREW/2,de_ballscREW/2], 'red');
        set(h,'FaceAlpha',0.3) ;
    end
    % Rotor geometry

```

```

    for i=1:N_elm_Rotor

h=fill([x_rotor(i),x_rotor(i+1),x_rotor(i+1),x_rotor(i)], [0,0,d_rotor/2,d_rotor/2]
,'blue');
        set(h, 'FaceAlpha',0.3) ;
    end

line(linspace(x_ballscREW(end),x_rotor(1),9),[de_ballscREW/4,de_ballscREW/2,0,de_b
allscREW/2,0,de_ballscREW/2,0,de_ballscREW/2,0])
    % Nut geometry
    h=fill([x_nut,x_nut+0.02,x_nut-
0.02],[de_ballscREW/2,de_ballscREW/2+0.01,de_ballscREW/2+0.01], 'green');
    h=fill([x_nut-0.08,x_nut+0.08,x_nut+0.08,x_nut-
0.08],[de_ballscREW/2+0.01,de_ballscREW/2+0.01,de_ballscREW/2+0.025,de_ballscREW/2
+0.025], 'green');
    % Bearing geometry
    x_bearing=(BearingNode-1)*L_elm_BallScrew;
    for i=1:length(BearingNode)
        h=fill([x_bearing(i)-
0.015,x_bearing(i)+0.015,x_bearing(i)+0.015,x_bearing(i)-
0.015],[de_ballscREW/2,de_ballscREW/2,de_ballscREW/2+0.005,de_ballscREW/2+0.005], '
blue');
        set(h, 'FaceAlpha',0.3) ;
    end
    set(h, 'FaceAlpha',0.3) ;
    grid on
    xlim([x(1),x(end)])
    xticks(x_compressed)

end

```

B. APDL Codes

1. COMBIN 14 Torsional Spring for Coupling

```
ET,_sid, combin14
KEYOPT,_sid,2,6 ! define z rotation freedom
K=5e12          ! torsional stiffness
r,_sid,K
```

2. MPC184 Screw Element for Ball Screw – Nut Interface

```
myjid=_jid
KEYOPT,myjid,1,17          ! MPC184 screw joint
SECTYPE,myjid,JOINT,SCRE
SECJOINT,LSYS,9999,9999    ! coordinate systems at nodes of MPC184
SECJOINT,PITC,7.9577e-4    ! meters per radians 5mm/rev
TB,JOIN,myjid,,,STIF
TBDATA,12,36000000         ! axial stiffness N/m
TBDATA,15,180000          ! axial-torsional stiffness Nm/m
TBDATA,21,143.239         ! torsional stiffness Nm/rad
TB,JOIN,myjid,,,DAMP       ! estimating corresponding damping values
TBDATA,12,0.002
TBDATA,15,0.002
TBDATA,21,0.002
```

C. Mode Shapes of Ball Screw Mechanism by MATLAB

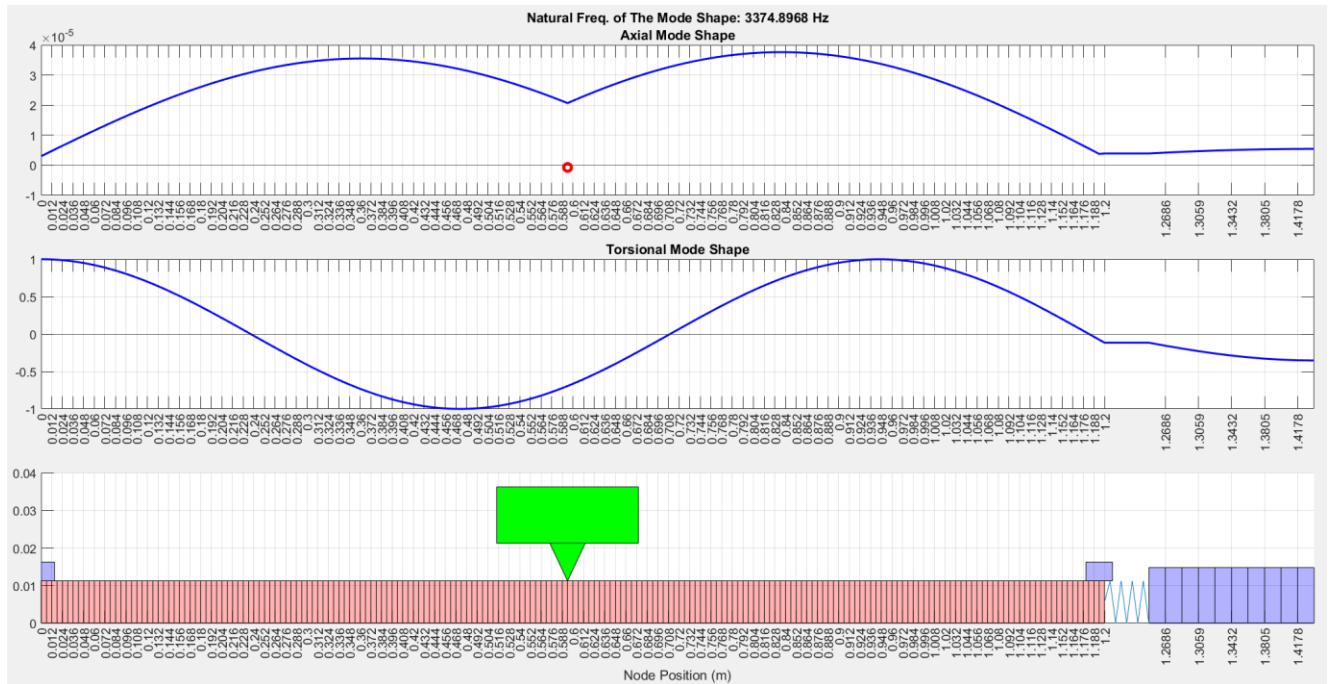


Figure 48 Mode shape of ball screw #5 – 3374 Hz

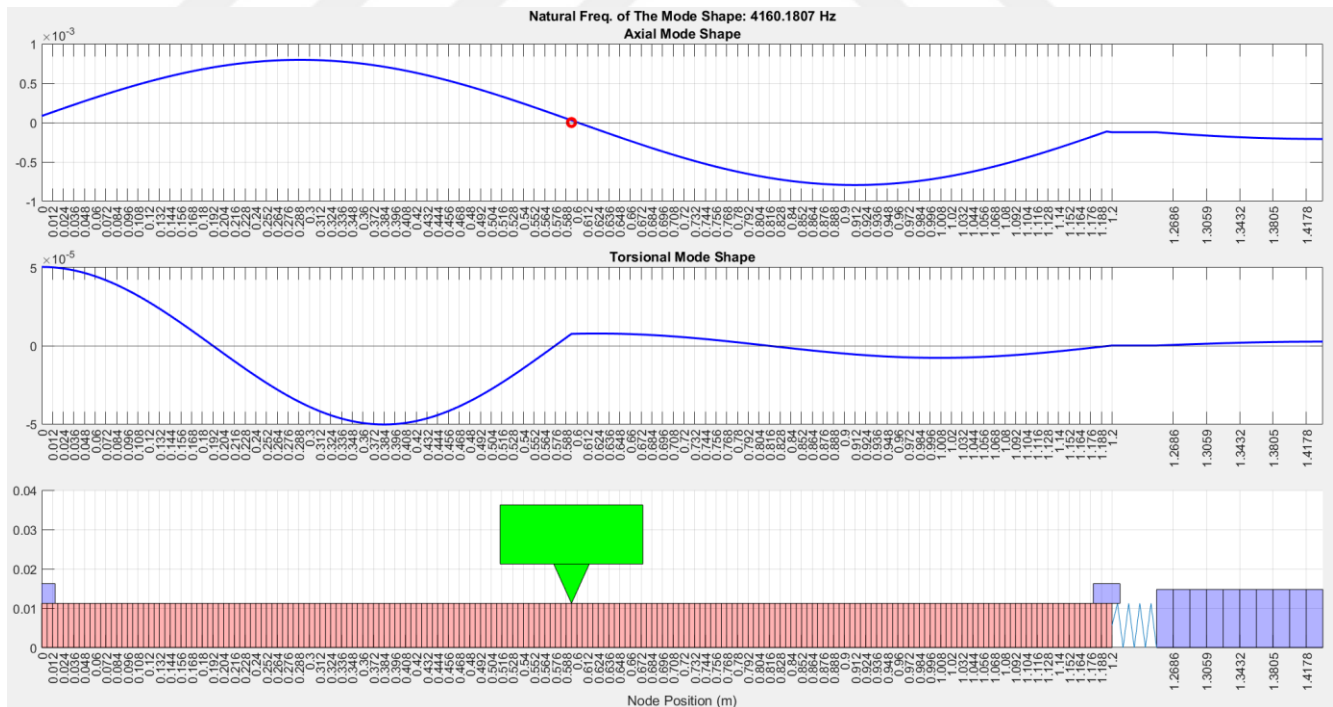


Figure 49 Mode shape of ball screw #6 – 4160 Hz

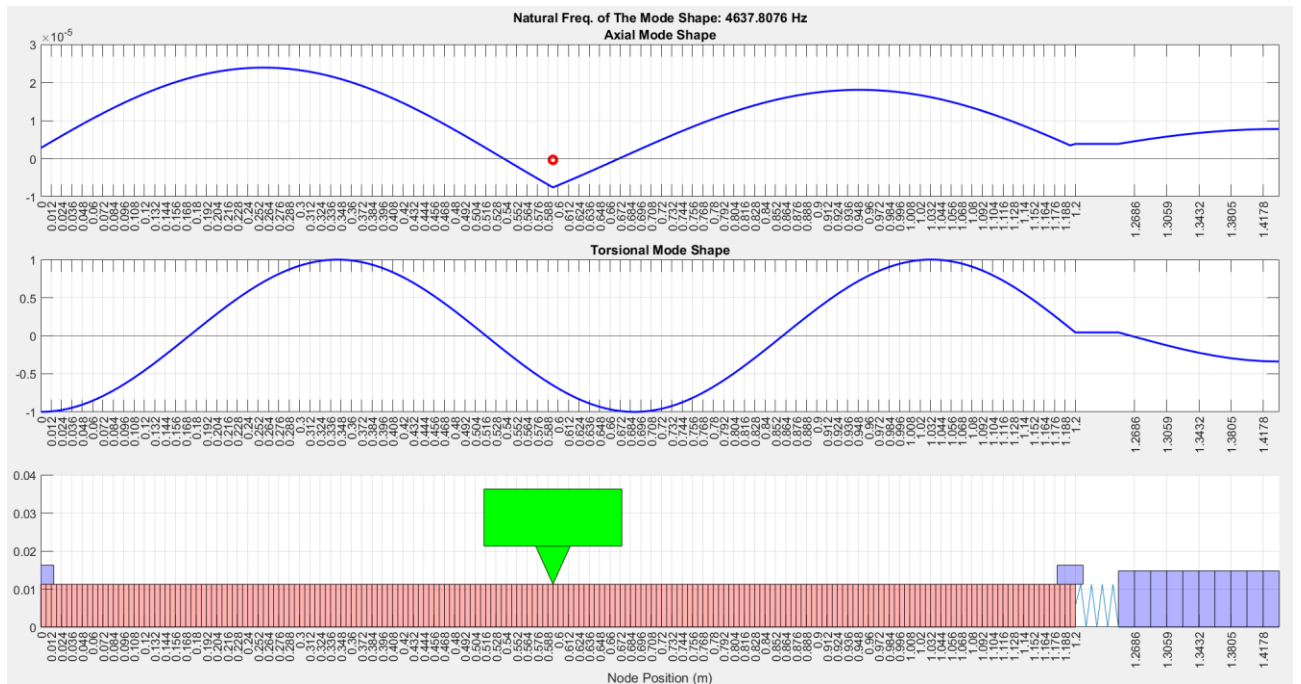


Figure 50 Mode shape of ball screw #7 – 4637 Hz

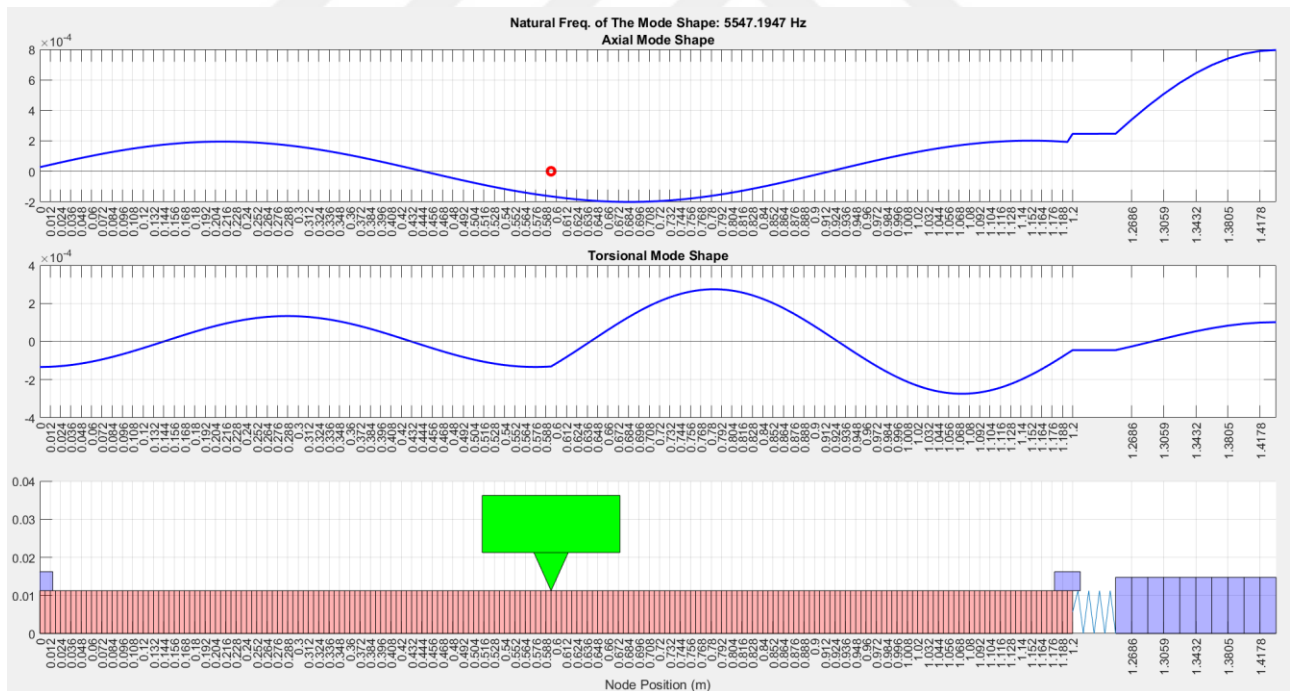


Figure 51 Mode shape of ball screw #8 – 5547 Hz

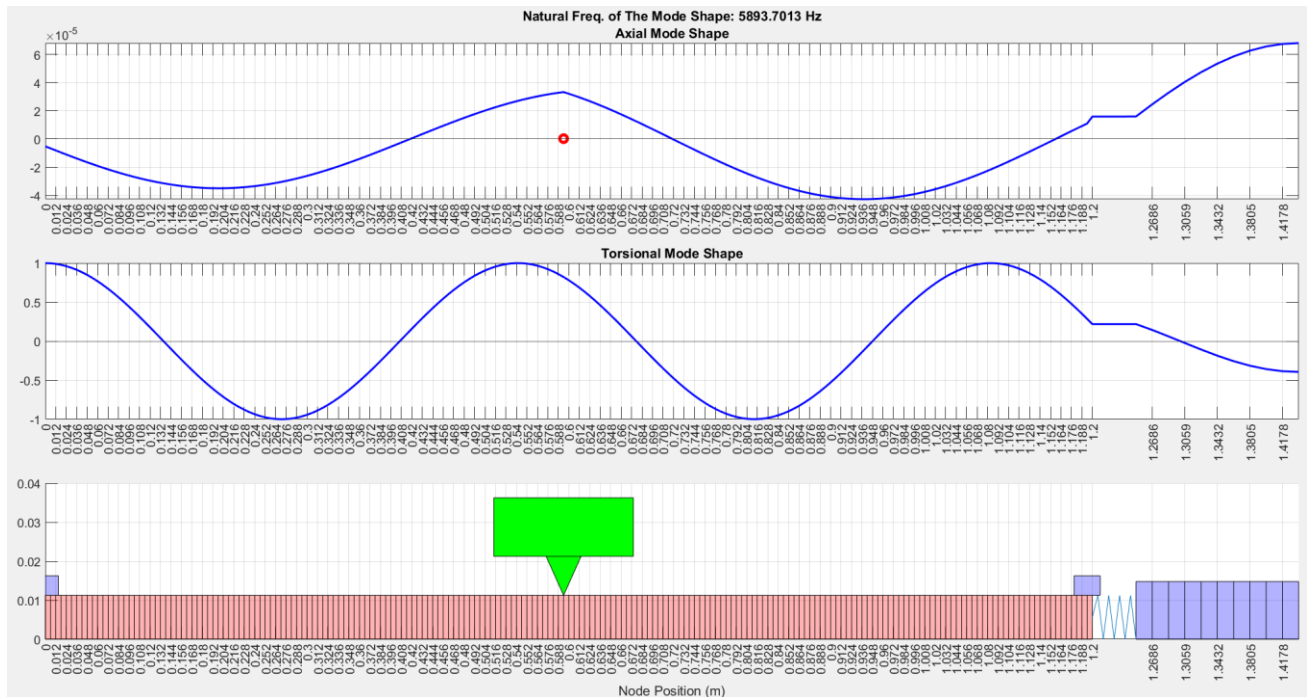


Figure 52 Mode shape of ball screw #9 – 5893 Hz

D. Mode Shapes of Ball Screw Mechanism by ANSYS

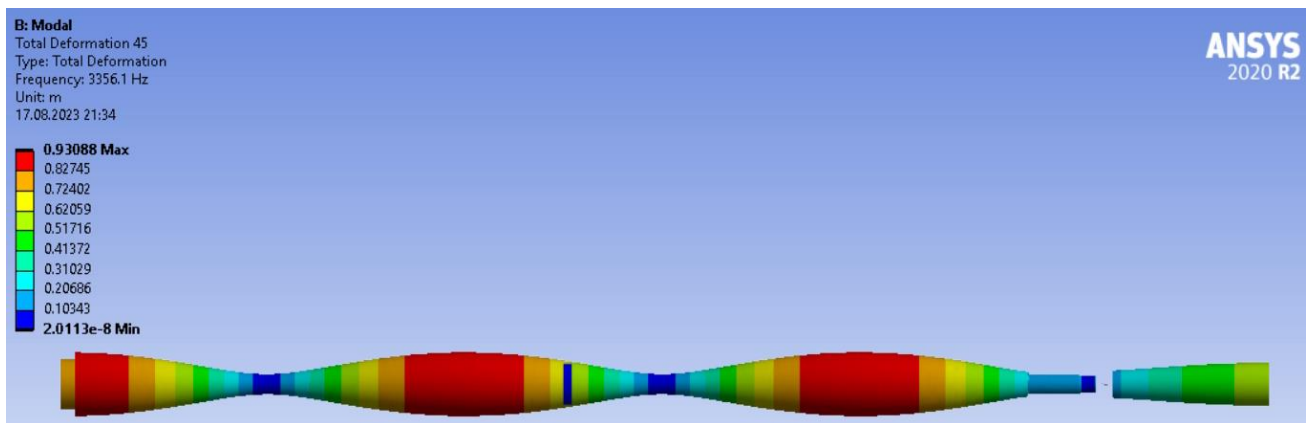


Figure 53 Mode shape of ball screw by ANSYS screw #4 – 3356 Hz

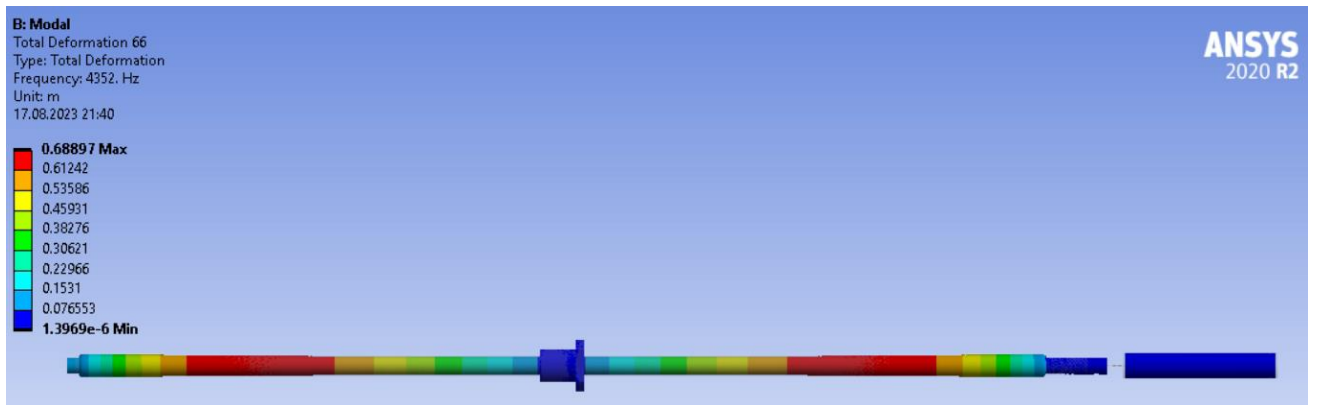


Figure 54 Mode shape of ball screw by ANSYS #5 – 4352 Hz

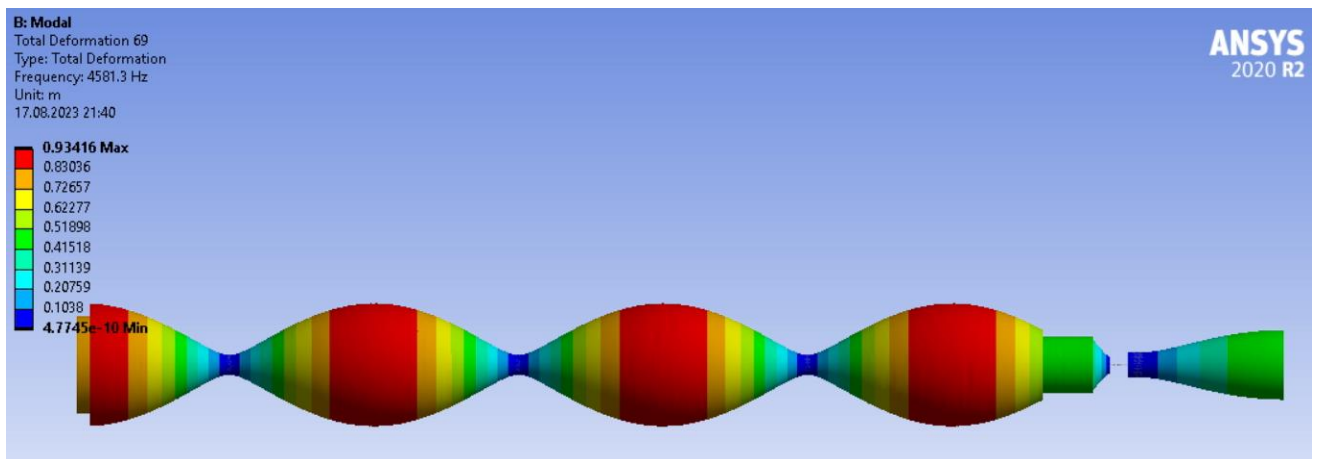


Figure 55 Mode shape of ball screw by ANSYS #6 – 4581 Hz

E. Mode Shapes of Machine Tool Frame by ANSYS

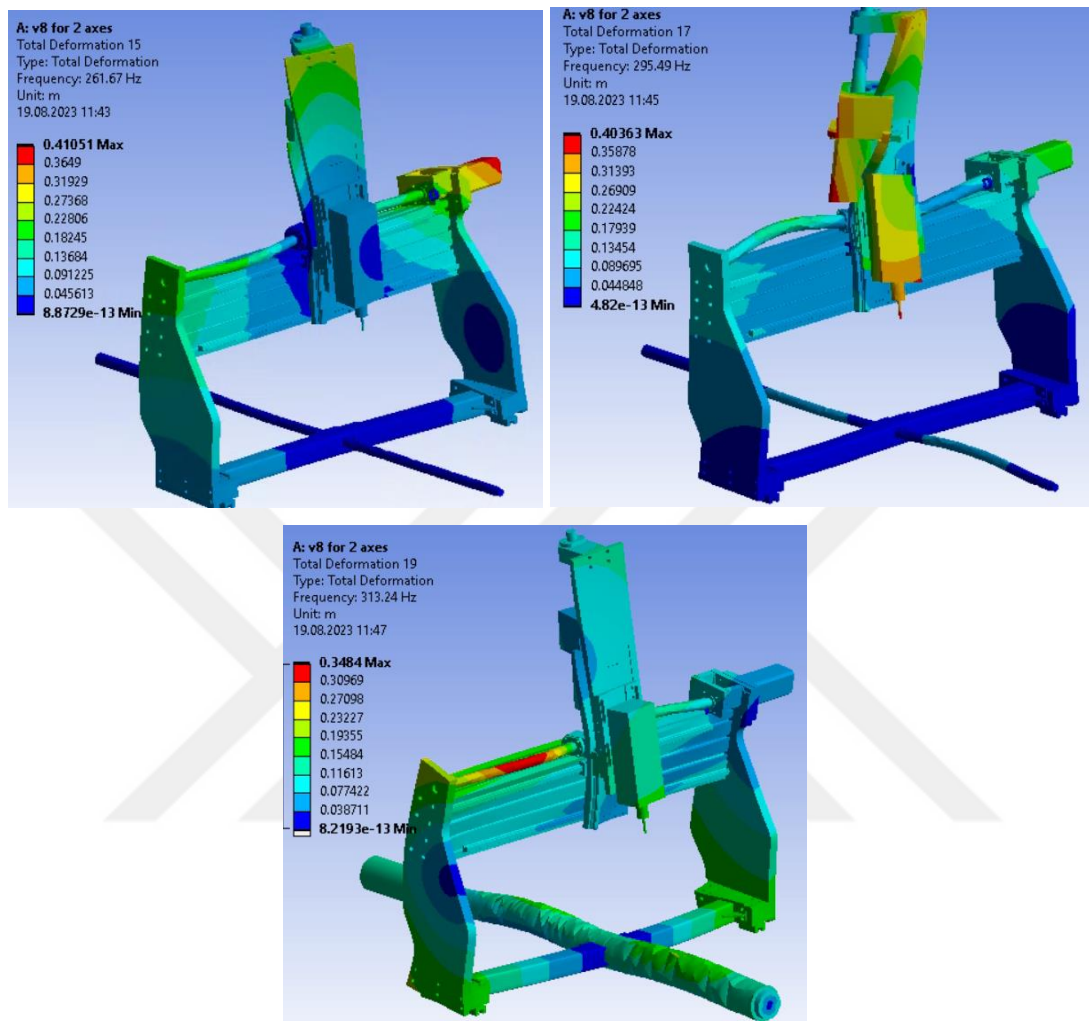


Figure 56 More mode shapes of machine frame by ANSYS