wbk
Institut für Produktionstechnik

Diploma Thesis

cand. ing. Juan Francisco Sánchez Alacid

Parameter Studies of a Machine Feed Axis Testbed in Time Domain by Application of Multibody Simulation

wbk
Institut für Produktionstechnik
Universität Karlsruhe (TH)
Kaiserstraße 12
D-76131 Karlsruhe

Prof. Dr.-Ing. J. Fleischer
Prof. em. Dr.-Ing. H. Weule

wbk
Institut für Produktionstechnik
Universität Karlsruhe (TH)
Kaiserstraße 12
76131 Karlsruhe
Prof. Dr.-Ing. J. Fleischer
Prof. em. Dr.-Ing. H. Weule

wbk
Institut für Produktionstechnik

Diploma Thesis (Diplomarbeit)

for Mr. cand. ing. Juan Francisco Sanchez Alacid, Matrikelnr. 1339071,
Brazal de la terraza, Nº 18, Patiño (Murcia), Spain

Parameter Studies of a Machine Feed Axis Testbed in Time Domain by Application of Multibody Simulation

The productivity of a machine tool is determined by its dynamical properties. Hence, it is important to determine those properties as early as possible when designing a new product. Because machine tool manufacturers are forced by an increasing competition to reduce the time to market, virtual prototypes can be used for simulation of the machine tool behavior instead of building and testing cost-extensive physical prototypes.

Since a large variety of parameters, such as component stiffness or damping of guides, couplings, ball screw drives and the like, influence the machine tool's behavior, the influence of those parameters has to be studied. To avoid extensive and expensive hardware testing, simulation is ideally suited to investigate a large number of different parameters and their importance.

The aim of this thesis is to investigate the dynamic behavior of a machine tool feed axis testbed by application of multibody simulation to study the effects of parameter variations in time domain. The following tasks have to be carried out:

- Illustration of the theoretical background of multibody simulation and parameter varation,
- modeling and simulation of the testbed with a multibody simulation tool,
- validating the model with experimental data
- simulation of parameter variations.

Interne Nr. der Arbeit:	WHT-
Tag der Ausgabe:	02.07.2007
Tag der Abgabe:	21.12.2007
Betreuer:	Dipl.-Ing. Alexander Broos

Karlsruhe, 03.12.2007

Prof. Dr.-Ing. Jürgen Fleischer

Declaration of Autonomy

Herewith I confirm that I wrote this report on my own without using any forbidden help. All the additives I used are completely listed in the bibliography. I marked everything that I absorbed from other papers with or without changes.

Karlsruhe, 21.12.2007

Juan Francisco Sánchez Alacid

Acknowledgements

First of all, I would like to thank Dipl.-Ing. Alexander Broos for his trust in me to realize this thesis, despite my difficulties with the English and German language. Without his support, it would have been impossible for me to create my thesis. Dipl.-Ing. Alexander Broos was always there when I had questions or any kind of problems and I want to thank him for his encouragement, support and patience.

I would also like to express my gratitude to Mr Xavier Rosel. He spent a lot of time helping with the English during the large period of the realization of this thesis.

Last but not least, I want to thank my parents, Juan Sánchez and Paquita Alacid. Without their support and help, it would have been impossible for me to finish my studies.

Thanks!

Table of Contents

List of Figures

List of Tables

Nomenclature

A	transformation matrix
B_{rs}	non-holonomic constrains coefficient
c	viscous damping coefficient
c_i	stiffness in i-axis
$[C]$	damping matrix
d_i	damping in i-axis
D_i	Lehr\`sches damping factor
$f(t)$	external force
$F(w)$	Fourier transform of $f(t)$
$\vec{F}_{e_i}$	forces in rigid bodies
$G(w)$	frequency response function
$\lvert G(w) \rvert$	dynamic compliance
i	rigid body
${}_I I_i^{(S)}$	moment of inertia tensor
j	$\sqrt{-1}$
J_{T_i} , J_{R_i}	Jacobian matrix
k	stiffness coefficient
$[K]$	stiffness matrix
$\vec{L}_i^{(S)}$	derivation of the angular momentum
m	mass
m_i	mass of each rigid body i
m_{ie}	mass in i-axis

M	non-holonomic constrains
$\vec{M}_{e_i}$	moments in rigid bodies
$[M]$	mass matrix
n_b	rigid bodies number
n_c	independent constrain equations number
N	holonomic constrains
$\vec{p}_i$	derivation of the momentum
P^i	arbitrary point belong the rigid body i
$\vec{q}$	generalized coordinates
$\dot{q}_i$	generalized velocity
Q_s	general forces
r^i	global position of the point P^i respect to the inertial frame system
$\dot{r}^i$	global velocity of the point P^i respect to the inertial frame system
$\ddot{r}^i$	global acceleration of the point P^i respect to the inertial frame system
$\vec{r}_{i_s}$	position vector of each rigid body with respect to its center of mass
R^i	global position vector of the body refence respect to the origin O^i
$\dot{R}^i$	global velocity vector of the body refence respect to the origin O^i
$\ddot{R}^i$	global acceleration vector of the body refence respect to the origin O^i
T	kinetic energy
u^i	vector position of P^i respect to the body reference
w	exciting frequency
w_d	damped natural frequency
$\vec{w}_i$	angular velocity vector of each rigid body with respect to its center of mass
w_n	natural angular frequency

w^i	angular velocity vector of the body i
$x(t)$	mass displacement
$\dot{x}(t)$	mass velocity
$\ddot{x}(t)$	mass acceleration
X	initial displacement of the motion
$X(w)$	Fourier transform of $x(t)$
$X^iY^iZ^i$	rigid body i coordinates
λ	lagrange multipliers
ϕ^i, θ^i, ψ^i	three angles Euler angles
α	angular acceleration vector
θ	rotation angle in the rotation matrix
$\Phi(w)$	phase angle
ζ	damping ratio

Abbreviations

2D	2-dimensional
3D	3-dimensional
ADAMS	Automatic Dynamic Analysis of Mechanical Systems
CAD	Computer Aided Design
CAE	Computer Aided Engineering
CAM	Computer Aided Manufacturing
Catia	Computer Aider Three dimensional Interactive Application
DMU	Digital Mock Up
DOF	Degrees of Freedom
FEM	Finite Element Method
FRF	Frequency Response Function
ISBN	Internationale Standard-Buchnummer
MBS	Multi-Body Simulation
MDOF	Multi Degrees of Freedom
SDOF	Single Degree of Freedom
SimCAT	Integration of CA-Technologies Towards a Holistic Simulation and Optimization Approach for Machine Tools
PSD	Power Spectral Density
UBKA	Universitätsbibliothek Karlsruhe
WBK	Institut für Produktionstechnik

1 Introduction

1.1 Motivation

The machine tool is a highly sophisticated technological product, which purpose is to provide capabilities to manufacture other products. This type of machine usually is not publicly recognized and, therefore, the advances and researches in the machine tool environment are widely unnoticed in the public. In recent years, the pressure on the manufacturers is increasing due to the necessity to improve the quality of these products while reducing the product development time and cost. For this reason, the use of virtual prototypes in the product development processes is being well accepted in this field. The benefit of using virtual prototypes is that it makes possible the simulation and the optimization of the machine properties in early stages. For it, the introduction of advanced engineering software tools is essential. Of these, the most commonly used in the machine tool industry is the finite element analysis (FEM), although at the moment, the multi-body simulation (MBS) is also being widely used to carry out many researches [Broos-2006].

As reference to all the previously expounded, have been created in the Institut für Produktionstechnik (wbk) of the Universität Karlsruhe, under the Bundesministerium für Bildung und Forschung, the German research project SimCAT: Integration of CA-Technologies Towards a Holistic Simulation and Optimization Approach for Machine Tools. The SimCAT project has as the overall objective of the research project, to create an integrated simulation environment which allows to simulate the dynamic behavior of machine tools using holistic virtual models, resulting in the ability to study the machine tool behavior and properties from the frequency domain over the time domain up to real time simulation in order to optimize single parts as well as the machine overall configuration and behavior [SimCAT-2006]. This challenge includes the design of parametric simulation models used for improvement of the behavior of a technical system by modifying single parameters by means of a parameter optimization process. These parametric simulation models are automatically diversified and evaluated by means of optimization programs. Methods for the enhancement of dynamic machine tool behavior and for the integration of process stability into the optimization loop are developed as part of the SimCAT-project [SimCAT-2006].

1.2 Objective

The main objective of this project is the study of the parameters in the dynamic behavior of a machine tool feed axis. In the Institut für Produktionstechnik have been carried out similar works with the same objective. Thanks to it, a validation of the method used for the development of this work can be realized. In this case, the method used for the analysis of dynamic behavior is by means of multibody simulation, where flexible connectors have been used to give elasticity to the system. The analysis is going to be realized in two fields, frequency domain and time domain analysis. With the utilization of these two analysis methods, the model can be studied in two different stages of the simulation process. Therefore, in the case of frequency domain, the results are obtained in an invariant time and for the case of time domain the results are obtained simulating over longer time periods. To obtain the results, has been taken as base the testbed, rebuilt by the SimCAT project, of which a simplified model has been obtained.

This project can be used as basis of other works which objective is the improvement of the dynamic behavior using optimization methods. In fact, this was the second objective of this thesis, where a frequency and time domain improvement of the model should have been realized, although, by different reasons, it has not been possible to carry it out.

1.3 Structure of this Thesis

In this work is described the study of the dynamic behavior of a model using multibody simulation. In chapter 2, it can be found, in first place, a theoretical development about the basis of the static, cinematic and dynamic behaviour of a machine tool. Then, an approximation to the multibody simulation is realized. And, finally, is realized a description of the utilized software.

In chapter 3 is described the state of the art. In it, is showed a brief revision of the literature about the simulation methods in the machine tool development process and, moreover, is given an overview of the researches realized about this topic.

An individual approach of this thesis and the followed steps to realize this project are shown in chapter 4.

In chapter 5 are described the different steps utilized to build the model. In the first part, a simplification of the real model is designed. Then, it is proceeded to the creation of the model with Catia and the mechanical modelling by means of MSC.ADAMS, both in general followed by each component individually. And, in the last part of this chapter, are offered the settings of the analysis tools necessary to obtaining results.

The results of the analysis are shown in chapter 6. In this chapter can be seen the results obtained in the Table when different parameters of the model are changed. These results are divided in two parts, frequency domain and time domain. Also an discussion of them is realized in the last part.

Finally, chapter 7 provides a summary of the work and gives a view about possible future investigations.

2 Theoretical Background

In this work is realized the study of the dynamics behavior of the machine tool feed axes, using multibody simulation. Therefore, in this first part of the thesis, it is developed the theoretical background that is necessary to carry it out. The chapter begins with a general introduction to the machine tool behavior. In the next part it is explained the basis used for design and simulation of the model using multibody simulaton. And finally, it is realized a brief description about the software necessary to carry out the analysis.

2.1 Machine Tool Behavior

The machine tool behavior is one of the most importart aspects in the creation of machine tool since it can decide the efficiency of the machine. That is due to important things like the accuracy, possible damage or wear of different parts of machine tool are directly influenced by its behavior. In this section it has been distinguished three type of behavior, static, kinematic and dynamic although other important types of behavior exist like, for example, thermal behavior. In the following part they are developed these three types of behavior.

2.1.1 Static Behavior

The problems derivated of the static behavior are produced due to the deformations of the transmision elements. This deformations are created by static friction forces, force action, weights and by means of accelerating forces in the drive. The deformation resistance of a structure under the influence of an external force in conditions of steady-state is denominated static stiffness [Weck-2001]. The measurents of flexibilitly or stiffness for the analysis of the static behavior can be realized in the interface between the tool and the workpiece, where the force is introduced. The problem of these measurements is that their results are refered to the genaral behavior of the machine and to its global deformation. Therefore, the influence of each part of the structure involved in the transmision or the contact point between the diffrerent parts can not be analized with these measurements. To analyze the static deformation of individual parts or several parts of the structure it has to be measured the relative or absolute movements between the differents parts in a lot of points of the structure. That is the only way, known the flexiblity of the parts, it can be determined the total static behavior of the system.

2.1.2 Kinematic Behavior

The geometrical variations are the cause of the problems produced in the kinematic behavior in a machine tool. This geometrical variations can be caused by faulty manufacturing, for instance spindel pitch error, incorrect installation or wear of the different parts which form the machine tool [Weck-2002]

2.1.3 Dynamic Behavior

In the machine tool task of metal cutting, the accuracy is determined by the desviation between the tool and the work piece in the interface respect to the defined trajectory. The deformation of all the structure parts in movement, like the spindels, couplings, or bearings, are produced by static and dynamic forces which create those desviations. A good method to solve this problem consists in the premature determination of the static and dynamic feature of the machine [Weck-2002].

The determination of the static stiffness during the construction time is not a problem nowadays. But it does not happen the same with the determination of the dynamic stiffness of a system which is influenced by numerous interactions of difficult estimation. The physical complexity of machine tool and the cutting process, the difficulty of estimating of the dynamic properties in the joints between structural components and the fact that the system is time varying since the components move relative to each other during the process, all it linked to other factors like ignorance of the exactly damping behavior lead to a complicated analysis of the dynamic behavior in the machine tool.

The imbalance of the dynamic feature of the mechanims generate vibrations. These vibrations in the metal cutting can cause defect in the machined surface texture, affecting significantly in the fine finishing operations. Moreover, this vibrations can cause an acceleration of the wear and breakage of the tool, acceleration of the machine tool wear, and damage to the machine tool and its parts. Therefore, the analysis of the dynamic behavior and the minimization of the vibrations play an important role in the future of this machine type and the machined processes [Weck-2002].

The dynamic behavior of the machine can be considered as a vibrator system of multiple degrees of freedom (MDOF) since it is composed of single structural parts, that can be treatred as a single degree of freedom systems connected by springs and dampers [Stephenson-2006].This idealization, where the dynamic behavior of a system can be considered as a single degree of freedom system under the influence of dynamic load, is the basis for many experimental methods whose objective is the analysis of the dynamics and stability in machine tool.

Different methods can be proposed to examine the dynamic behavior and system stability. In this project, the methods used are two, the time domain and frequency domain. In the frequency domain, the frequency response of a system is the steady-state response when a sinusoidal input signal is introduced. The output of a linear system to such an input is also sinusoidal with the same frequency in the steady state and it only differs from the input waveform in the amplitude and phase angle. For the analysis and desing of machine tool, the frecuency response has the advantage that it is a analyze method based on the amplitude and phase equations and curves but it has the disadvantage that is limited in applicability to linear time invariant systems. In the time domain method, the system response is given for a set of variables that determine the future behavior of a system. This variables are known as state variables. For the development of this method is necessary to know the present state of the system and the excitation signals. Some of the time domain advantages are that it can be used for nonlinear systems, time varying and multivariate systems[Stephenson-2006].

For an easy understanding of the dynamic behavior, it is explained the analysis of a single degree of freedom system (SDOF), which is a brief review of the basic vibration theory. In the first part of this analysis, any external force is introducced to the system. The movement is generated by a displacement or initial velocity, desviating the system from its static equilibrium and as a consequence of it, the system vibrates freely. This SDOF system can be modeled as the Figure 2.1 not taking into account the *Dynamic loading* shown in the ilustration.

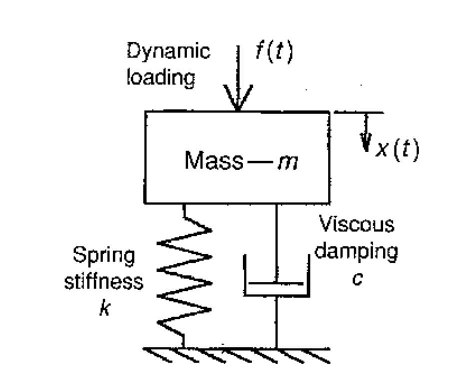


Figure 2.1: Ilustration of a single degree of freedom system [Stephenson-2006]

In the ilustration can be observed a system composed by a mass *m* sopported with a spring and viscous damper in parallel. The stiffness coefficient of the spring possess a value of *k* and the viscous damping coefficient of the damper is *c*. When the system is desviated of the equilibrium position its motion is described by the following differential equation:

$$m\ddot{x}(t)+c\dot{x}(t)+kx(t)=0 \tag{2.1}$$

where $\ddot{x}(t)$, $\dot{x}(t)$ and $x(t)$ are the acceleration, the velocity and the displacement of the mass, respectively.

When it is assumed that the initial displacement of the motion is defined by X, the solution of the diferential equation (2.1) for machine tool, which is supossed as a underdamped system, can be described according to [Altintas-2000] as

$$x(t)=Xe^{-\zeta w_n t}\cos(w_d t) \tag{2.2}$$

where, ζ is the damping ratio.This damping ratio is just a ratio of the actual damping over the amount of damping required to reach critical damping. In machine tool, this coefficient is usually very tiny and never greater than one, and in most of the cases its is even below 0,05. The formula for the damping ratio of the mass-spring-damper model is

$$\zeta=\frac{c}{\sqrt{km}}, \tag{2.3}$$

w_n is defined as the natural angular frequency of the system when the damping coefficient is equal to zero (c=0). The natural angular frequency is defined by

$$w_n = \sqrt{\frac{k}{m}}, \qquad (2.4)$$

and w_d is the damped natural frequency of the structure and is related with the undamped natural frequency by

$$w_d = w_n \sqrt{1-\zeta^2} \qquad (2.5)$$

The illustration graphic of the movement versus time for a machine tool when the damping ratio is ζ =0,1 and ζ =0,3 can be seen in the Figure 2.2. In these two figures can be observed how the logarithmic decrease of the vibrator movement is influenced by the damping. For this reason a good method, to determine the damping value in the system, will be the measure of the decreased range in the free vibrations.

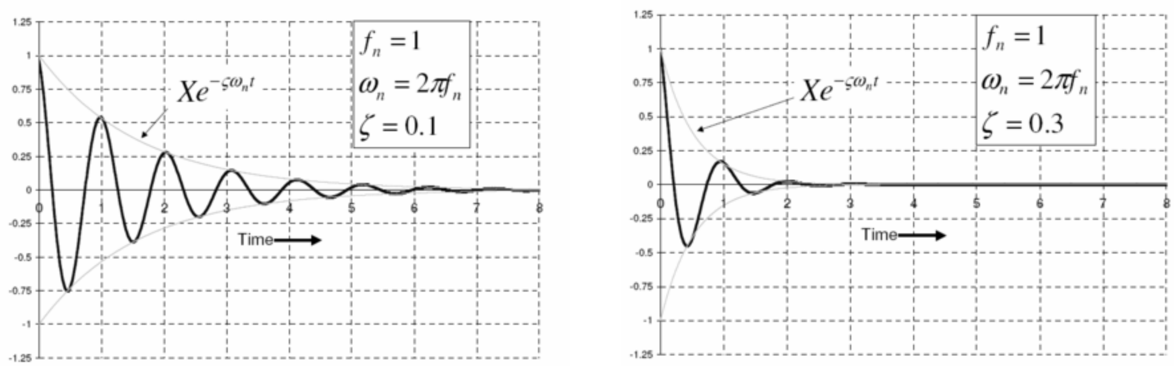


Figure 2.2: Representation grafic of the free vibration fo an SDOF system with ζ <1.

In the second part of the analysis, the single degree of freedom system is excited by an external force $f(t)$, see Figure 2.1. If it is assumed that the external force is harmonic, i.e., it can be represented by sine or cosine functions, the force can be described as $F_0 \cos(wt)$. When this harmonic force is introduced in the motion equation (2.1) the result is the following:

$$m\ddot{x}(t) + c\dot{x}(t) + kx(t) = F_0 \cos(wt) \qquad (2.6)$$

Taking into account that $f(t)$ can be also defined as complex harmonic functions such as $f(t) = Fe^{jwt}$ and taking the Fourier transform of both sides of the equation (2.6), will yield [Stephenson-2006]

$$(-w^2 m + jwc + k)X(w) = F(w) \qquad (2.7)$$

where $F(w)$ belong to the Fourier transform of $f(t)$, $X(w)$ belong to the Fourier transform of $x(t)$, w is the exciting frequency and $j=\sqrt{-1}$. Therefore, while the force is active in the system the steady-state can be given by

$$G(w)=\frac{X(w)}{F(w)}=\frac{1}{-w^2m+jwc+k} \tag{2.8}$$

The frequency response function ($G(w)$) is the ratio of the complex amplitude of the displacement produced by a force unit at the frequency w. The frequency response function can be also called the *transfer function* or *receptance* of the SDOF structure in other literature. This function correspond to a complex quantity and, therefore, it is composed of two parts, a real and a imaginary part. The real part represents the mobility of the system, while the imaginary part represents the inertance. The magnitude, which represents the dynamic compliance of the system, and the phase angle of the frequency response function are given, when $r=\frac{w}{w_n}$ by [Stephenson-2006]

$$\left|G(w)\right|=\sqrt{\frac{\left|X(w)\right|}{\left|F(w)\right|}}=\frac{1}{k\sqrt{(1-r^2)^2+(2\zeta r)^2}}$$

$$\Phi(w)=\tan^{-1}\left[\frac{2\zeta r}{1-2r}\right] \tag{2.9}$$

In the graphical illustration shown in Figure 2.3 it can be seen the phase angle $\Phi(w)$ and the dynamic compliance $\left|G(w)\right|$ of the system as functions of the frequency ratio r. In this representation can be observed the damping influence on the amplitude and phase angle in a SDOF system. This influence is maximum if the resonance appears in the system, which is produced when the response frequency has the same value that the natural frequency. In an undamped system this resonance generates a maximum dynamic compliance that has not end and, moreover, produces an instantaneous change in the phase angle from 0º to -180º. In a real system with a damping bigger than 0, the resonance will have an end and the phase angle variation will be changed gradually. The damping effect on a system decreases insofar the damping ratio value rises. The complete disappearance of the resonance in a system will be produced when its value is bigger than one. One of the major reasons to do vibration analysis is to predict when it may appear this phenomenon and to determine which steps follow to prevent it from occurring, since a too high amplitude value can be very harmful, leading to eventual failure of the system.

The compliance of the FRF can be divided in three control regions, see Figure 2.3. The central region, that is centered in the resonance frequency, is influenced mainly by the damping of the system, causing a decrease of the resonance when damping ratio value increases. At lower frequencies than the damping-controlled region, it is located the region influenced by stiffness, where it can be determined the stiffness value when the excitation frequency is zero. Finally, at higher frequencies than the two previous regions, it is situated the mass-controlled region, where the bigger influences in the dynamic compliance are produced by the mass.

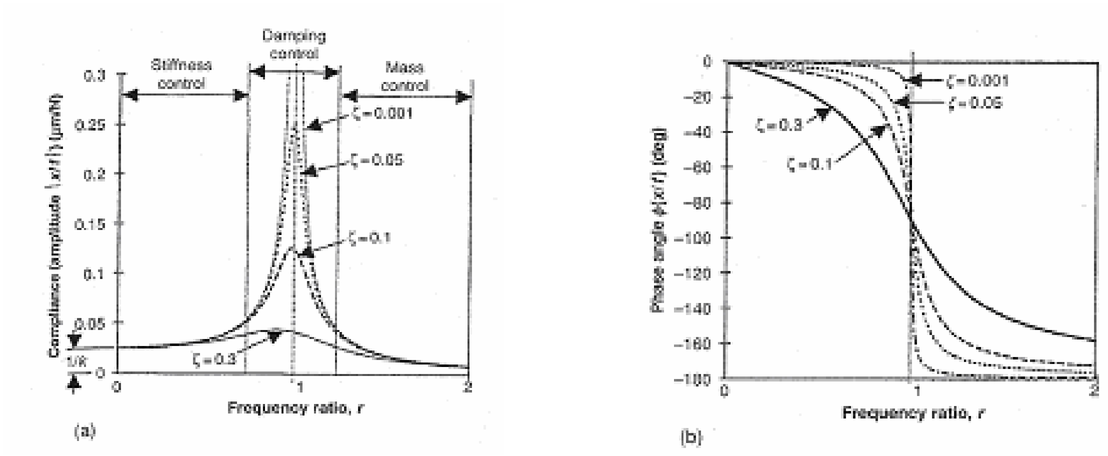


Figure 2.3: Response curves.(a) Compliance/amplitude versus frequency.(b) Phase versus frequency[Stephenson-2006]

In the graphical representation of the compliance versus frequency of the excitation force, the increase of the stiffness produces an increase of the stiffness-control part, displacing the resonance to higher excitation frequency values and reducing the mass-controlled area. The same happens when the mass value increases, but in this case, the other way around. The basis of these variations is the change of the natural frequency which is directly proportional to the stiffness and inversely proportional to the mass variations. Therefore, the resonance frequency, that is the center in the damping-controlled area, changes when it is increased or reduced the natural frequency of the system.

The analysis of a SDOF system idealizes a structure as a superposition of lumped masses. This mean that, the resonance frequency of the single parts is determined as the only resonance frequency of the structure as it was described previously. Moreover, the total vibration frequency of the system can be considered as the addition of all the single vibrations of each structure part. With the objetive to avoid distorsions, would be important that, the studied single oscillations of the structure were very obvious and with a very separated resonance frequency between themselves [Weck-2002].

In many structures, the techniques of analysis of a SDOF system are not enough. In this case, other method of analysis can be used. A method, which is a natural extension of the SDOF, would be the study of a system as a MDOF system. The MDOF method consists in idealize the structural elements of a machine tool as discrete lumped masses connected by springs and dampers or a distributed prameter system [Stephenson-2006]. This method has one natural frequency for each dregree of freedom (or mass) and it specifies the damping, the resonant positions and the oscillation amplitude for several oscillation forms concurrently. This makes it posible to determine the coupling oscillation and severely dampered form. Because of these reasons, a more exact description of the dynamic behavior of the machine tool is posible [Weck-2002]. The basis of the analysis in a MDOF system is in the main formula of the modal analysis. In a MDOF systems, the equation of motion is given by

$$[M]\{\ddot{x}\}+[C]\{\dot{x}\}+[K]\{x\}=\{f(t)\} \tag{2.10}$$

where $[M],[C]$, and $[K]$ are, respectively, the mass, damping, and stiffness matrices for the system.

TYPES AND CAUSES OF VIBRATION

In the feed rate and cutting movement in machine tools it appears overlaid relative movements between the cutting tool and the workpiece. Those relativ movements are generally classified in self-excited vibrations and separately excited vibrations. The reason because this vibrations are generated is described in the Table 2.1.

Self-excited vibrations	*Separately excited vibrations*
-Noise of the cutting forces -Cascading F-v-Characteristic -Formation of build-up edges -Bearing interconnection -Regenerative effect	-Impacts caused by incipient cut -Interrupted cutting -Disturbance force -Unbalances -Alternating cutting forces

Table 2.1: Vibrations in machine tools [Weck-1992]

Separately excited Vibrations

The inaccuracy of the construction elements, damaged elements, disturbance forces or alternating cutting forces produced by inhomogeneities in the workpiece, interrupted cutting, built-up edge or changes in the chip cross section can cause this type of vibration. The separately vibrations are characterized by vibrations in the machine with the same frequency that the excitation forces when they are periodically excited. Therefore, the magnitude of this vibration can be especially high when the excitation frequency is near the natural frequency of the system. If the machine is not excited by periodical forces, the structure vibrates with the natural frequency as it happens with impuls or step forces. This vibration types are decreased exponentialy in relation with the amount damping in the system.

The elimination or the reduction of the separately excited vibration in most of the cases can be realized eliminating the disturbance force or moving the periodic excitation frequency far from eigenfrequency of the structure.

Self-excited Vibrations

Opposite to the separately vibrations, the self excited vibration is not produced by disturb external forces. This type of system vibration is always in one or several eigenfrequencies.

The noise of the cutting process is a form of vibration that is situated between the separately vibrations and the self excited vibrations. This vibration is generated by the surface waviness, which is a consequence of the separately vibration in the metal cutting, creating self excited vibrations in the system. The noise of the cutting process is not very important since the spectrum procuced by the metal cutting has small amplitudes.

Another form of self excitation vibration is produced by the decreasing cutting forces when the cutting speed increases. In this case it can be produced imbalances in the system caused by the negative influence of the damping. Nowadays it almost never appears these vibrations because they appear in low cutting speed which are rarely used. Another vibration which only matters in low cutting speeds inducing alternated cutting forces is the build-up edges.

An important dynamic problem is the self excited vibration caused by regenerative effect. This type of vibration is linked with the noise of the cutting procces because it creates waviness on the workpiece. The appearance of inestability in the system depends on the stiffness of the machine and on the cutting process conditions.

2.2 Dynamics of Multibody Systems

The need of a better design of the system like in machine tool, robotics, and space structures has led to the development of methods for the dynamic analysis of multibody system. A multibody system consists of a collection of subsystems called bodies, components or substructures, interconnected by joints and forces where each one of those may undergo large translational and rotational displacements. The analysis of a multibody system can be divided into three differents fields according to the motion of the material of the subsystems. In the first field, the deformation of the body does not affect on the motion because the distance between any two particles of it is considered constant. In the second field, this distance between the particles will be considered and therefore, the deformation of the body will affect on the motion. And the third field of study, called continuum mechanics, will be formed by the union of the two previous fields [Shabana-2005a]. The study realized in this proyect is going to be centrated in the first field, where it is analyzed the behavior of a rigid body.

2.2.1 Rigid Body Kinematics

Two reference system types are used to determinate the locations and orientations of the rigid bodies in the multybody system.

·The first system, called inertial frame of reference, is a coordinate system fixed in time, which is unique for all the bodies of the mutibody system. This inertial frame of reference can be represented by three orthogonal axes where the point of connection between themselves is denominated origin.

·The second coordinate system used in the orientation and location of multibody systems is known as body reference. Each body refence is fixed to a rigid body, and as result of it , this system will change its location and orientation in time respect to the inertial frame system.

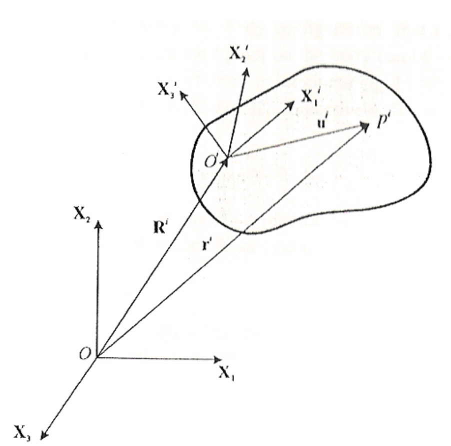


Figure 2.4: Body and inertial frame references [Shabana-2005a]

These two reference systems can be seen in the Figure 2.4 where appart from theses reference system, it has been represented a rigid body i. If in the rigid body i, it is selected an arbitrary point P^i, its representation can be related with three vectors. This three vectors are resepented in the ilustration as R^i, which is the global position vector of the body refence respect to the origin O^i, r^i that is the global position of the point P^i respect to the inertial frame system, and u^i that is the vector position of P^i respect to the body reference. Therefore, the global position of P^i can be defined as

$$r^i = R^i + u^i \tag{2.11}$$

As the R^i and r^i vectors are represented in global coordinates and only the u^i vector is not referenced to this coordenates, it would be important to exprese the u^i vector as a function of the goblal coordinate system. This transformation of the u^i vector from the globlal coordinate system to the body coordinate system can be carried out thanks to the transformation matrix A. Therefore , the globlal position of the point P^i can be expressed as

$$r^i = R^i + A^i \overline{u}^i \tag{2.12}$$

2.2.1.1 Rotation Matrix

The transformation matrix A is used to reference a vector from a body coordinate system to a global coordinate system.

$$r = A\overline{r} \tag{2.13}$$

For an easy development of A, it can be assumed that the axis of the two coordinate systems are parallel and that their origin is the same point. The rotation matrix referenced to the Rodriguez formula and expresed in terms of the angle of rotation, it can be written, after some trigonometric and mathematical transformations, according to [Shabana-1992] as

$$A = \left[I + \tilde{v} \sin\theta + 2\left(\tilde{v}\right)^2 \sin^2 \frac{\theta}{2} \right] \tag{2.14}$$

where I is a 3×3 indentity matrix and θ is the angle between b_2 and Δr showed in the Figure 2.5.

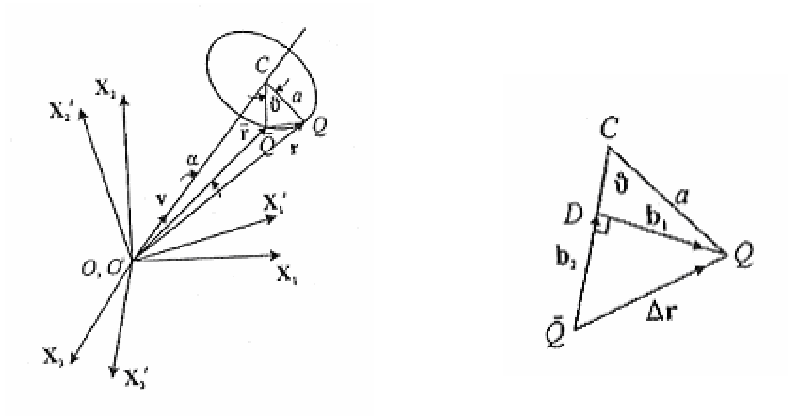


Figure 2.5: Rotation of the coordinate system [Shabana-1998]

This transformation matrix can be also expressed in terms of the four Euler parameters:

$$\left.\begin{aligned}\theta_0 = \cos\frac{\theta}{2}; \theta_1 = v_1 \sin\frac{\theta}{2} \\ \theta_2 = v_2 \sin\frac{\theta}{2}; \theta_3 = v_3 \sin\frac{\theta}{2}\end{aligned}\right\} \tag{2.15}$$

where, for the case of planar motion, the rotation matrix can be written as

$$A = \begin{bmatrix} \cos\theta & -\sin\theta & 0 \\ \sin\theta & \cos\theta & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{2.16}$$

2.2.1.2 Velocity and Acceleration

Another part of the kinematic analysis is the determination of the velocity of a body in the multibody system. For this, it is realized the differentiation of the equation (2.11) of the position of a point P^i in a body with respect to the time. The result of this differentation will be

$$\dot{r}^i = \dot{R}^i + w^i \times u^i \tag{2.17}$$

where $\dot{r}^i$ is the velocity vector with respect to the global coordinate system of the P^i point, $\dot{R}^i$ is the velocity vector with respect to the body coordinate system and w^i is the angular velocity vector of the body i.

Another form to obtain the velocity vector is using the equation (2.12) and to obtain with it the velocity with respect to the global coordinate system, using the transformation matrix. In this case the velocity vector will be

$$\dot{r}^i = \dot{R}^i + A^i\left(\overline{w}^i \times \overline{u}^i\right) \tag{2.18}$$

Regarding to the acceleration of the arbitrary point P^i that belongs to a body i, it can be got making the differentiation of the velocity $\dot{r}^i$ with respect to the time, whose result will be the following

$$\ddot{r}^i = \ddot{R}^i + w^i \times\left(w^i \times u^i\right) + \alpha^i \times u^i \tag{2.19}$$

being α the angular acceleration vector. And if, like in the case of the velocity, it is used the transformation matrix, the result obtained is

$$\ddot{r}^i = \ddot{R}^i + A^i\left[\overline{w}^i \times\left(\overline{w}^i \times \overline{u}^i\right)\right] + A^i\left(\overline{\alpha}^i \times \overline{u}^i\right) \tag{2.20}$$

2.2.1.3 Euler Angles

Euler angles are the most common method to describe the orientation in three dimensional rotational of multibody simulation [Shabana-2005a]. The objective of this part is to develop the transformation matrix in terms of Euler angles. The method consists of three axes that are not orthogonal in general and that are involved in three successive rotations.

For it, two coordinate systems, which initially coincide, are considered, XYZ and $X^iY^iZ^i$. The development of the method begins when the coordinate system $X^iY^iZ^i$ rotates around the axis Z^i with an angle ϕ^i. The result of this rotation is a transformation matrix as

$$A_1^i = \begin{bmatrix} \cos\phi^i & -\sin\phi^i & 0 \\ \sin\phi^i & \cos\phi^i & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{2.21}$$

Later, another rotation is produced in the system. This new rotation is realized around the axis X^i and with an angle of θ^i, which offers a new matrix that can be described as

$$A_2^i = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos\theta^i & -\sin\theta^i \\ 0 & \sin\theta^i & \cos\theta^i \end{bmatrix} \tag{2.22}$$

Finally, another rotation with an angle ψ^i is produced around the axis Z^i and the result is

$$A_3^i = \begin{bmatrix} \cos\psi^i & -\sin\psi^i & 0 \\ \sin\psi^i & \cos\psi^i & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{2.23}$$

From the multiplication of these three rotation matrix A_1^i, A_2^i and A_3^i, it can be defined the final orientation of the system $X^iY^iZ^i$ in terms of Euler angles, which result is

$$A^i = \begin{bmatrix} \cos\psi^i\cos\phi^i - \cos\theta^i\sin\phi^i\sin\psi^i & -\sin\psi^i\cos\phi^i - \cos\theta^i\sin\phi^i\cos\psi^i & \sin\theta^i\sin\phi^i \\ \cos\psi^i\sin\phi^i + \cos\theta^i\cos\phi^i\sin\psi^i & -\sin\psi^i\sin\phi^i + \cos\theta^i\cos\phi^i\cos\psi^i & -\sin\theta^i\cos\phi^i \\ \sin\theta^i\sin\psi^i & \sin\theta^i\cos\psi^i & \cos\theta^i \end{bmatrix} \tag{2.24}$$

The three angles ϕ^i, θ^i and ψ^i are the Euler angles, with whose rotation it can be oriented any rigid body in the space. The rotation sequence can be any one desired, for instance Z^i, Y^i and X^i.

2.2.1.4 Degrees of freedom, Constrains and generalized Coordinates

The unconstrained motion of a rigid body can be described by six independent coordinates. This coordinates are divided beetwen three translational coordinates, which describe the lineal movement of the system in the space, and three rotational coordinates, which describe the rotational movement of the system in the space. Therefore, the set of independent coordinates that describe the unconstrained motion of a rigid body is usually named degrees of freedom [Shabana-2005a].

A mechanical system consists of a collection of bodies interconnected by a different number and type of joints and force elements. The force elements usually do not prevent totally the movement between the bodies although they affect their motions. An example of these elements could be spring or damper. However, the joints are elements that prevent the motion in some directions of the system, therefore, they lead to a reduction of the degrees of freedom in a system. The joints are used to define the desired kinematic motion in a mechanic system, eliminating some degrees of freedom. This method is used in ADAMS with the same objetive. Examples of some joints, that appear in mechanical systems and that are also used by the mencioned software, are shown in the figure 2.6. Each joint reduces different types of motion and consequently different number of degrees of freedom, for instance:

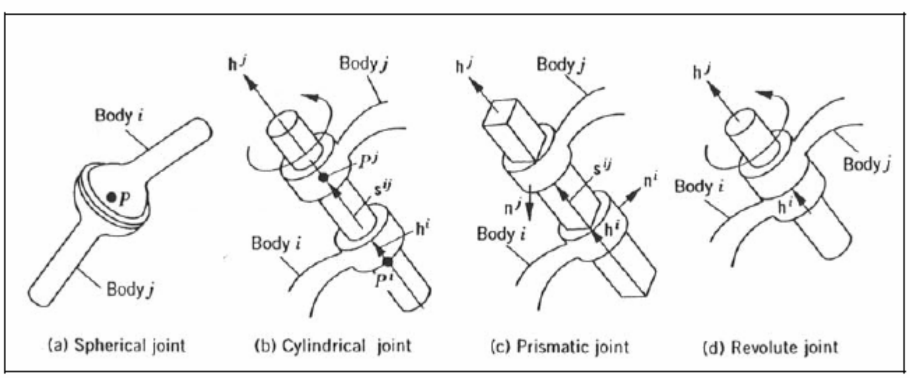


Figure 2.6: Spatial joints [Shabana-2005b]

-Spherical joint allows only relative rotations and is considered a joint of three degrees of freedom, Figure 2.6(a).

-Cylindrical joint possess two degrees of freedom, allowing only two relative displacements, one rotational and one translational, Figure 2.6(b).

-Prismatic joint has a single degree of freedom consisting in a translational relative displacement, Figure 2.6(c).

-Revolute joint has also a single degree of freedom, but this consists in a rotational relative displacement, Figure 2.6(d)

The motion of a system is reduced by mechanical joints, because the motion between the different rigid bodies is not independent between them. This motion can be described by algebraic equations related to the mechanical joints. Therefore, the number of system degrees of freedom is defined by the number of the system coordinates minus the number of independent constrain equations [Shabana-2001]. Then, a system, with n_b rigid bodies and n_c independent constrain equations, possess a number of degrees of freedom given by $DOF = 6 \times n_b - n_c$.

The generalized coordinates are denominated as the set of variables necessary to identify a multibody system, since they define the location and orientation of each body in the system. In some cases, the system can be defined by six coordinates as it has been explained previously, but in other cases, the number of variables changes depeding on the method used for its definition; Euler angles, Rodriguez parameters, an so on. The vector $\vec{q} = [q_1, q_2, q_3, q_4, ..., q_i]^T$ is the set of generalized coordinates of a system and the term i is the number of generalized coordinates in this vector. The generalized coordinates are related by the n_c constrains of a system, which number is equal or smaller than the generalized coordinates.

The constrains can be classified on many ways. One of this ways is related with its integrability. With respect to this, the constrains can be divided into two types, holonomic and nonholonomic. The holonomic constrains are the constrains that can be written in the vector form $C(\vec{q}, t) = 0$. In this case, the number of variables necessary to define a spatial system with n rigid bodies and N holonomic constrains is $v = 6n - N$. The other type of constrains are the nonholonomic, which also depend on the velocity, but in this case they can be written in the vector form $C(\vec{q}, \dot{\vec{q}}, t) = 0$. The M non-holonomic constrains cause that only $p = v - M$ of the $\dot{q}_i$ generalized velocity are independient between themselves and therefore, this set of nonintegrable kinematic constrains can not be reduced to geometric constrains.

2.2.2 Kinetic

2.2.2.1 Newton-Euler method

A posibility to obtain the motion equations for computer is to use the Newton-Euler method in the support of d´Alambert. The Newton-Euler method of deriving dynamic equations involves treating every body as a separate *free body* and solving for all forces and torques reaction. The system is formed by n rigid bodies with N holonomic constrains, whose vector $\vec{q}$ contains v generalized coordinates q_i. The translational equations come directly from the linear momentum principle, while the rotational equations come directly from the angular momentum principle. The Newton-Euler method in its compact form is

$$\sum_{i=1}^{n}\left[J_{T_i}^{T}\left(\dot{\vec{p}}_i - \vec{F}_{e_i}\right) + J_{R_i}^{T}\left(\dot{\vec{L}}_i - \vec{M}_{e_i}\right)\right] = 0 \tag{2.25}$$

where $\vec{p}_i = m_i \dot{\vec{r}}_{i_s}$ and $\vec{L}_i^{(S)} = {}_I I_i^{(S)} \vec{w}_i$ are the derivation of the momentum and of the angular momentum respectively, being $\vec{r}_{i_s}$ the position vector of each rigid body with respect to its center of mass, $\vec{w}_i$ the angular velocity of each body, ${}_I I_i^{(S)}$ the moment of inertia tensor and m_i the mass of each body. Moreover, this method contains the forces $\vec{F}_{e_i}$ and the moments $\vec{M}_{e_i}$ applied to the system and the Jacobian matrix $J_{T_i} = \frac{\partial \dot{\vec{r}}_{i_s}}{\partial \dot{\vec{q}}}$ and $J_{R_i} = \frac{\partial \dot{\vec{w}}_{i_s}}{\partial \dot{\vec{q}}}$.

2.2.2.2 Lagrange II

However, in MSC.ADAMS it is used another method which in the literature is named as Lagrange II, althougt it is known in MSC.ADAMS as Euler-Lagrange equation. The system is composed in this case of n bodies, M nonholonomic constraints and N holonomic constraints, with v generalized coordinates q_i, whose differential with respect to the time $\dot{q}_i$ has only $p = v - M$ independent generalized coordinates. Then, this case, due to the nonholonomic constraints, appear aditional lineal constraints in the generalized velocity given as

$$\sum_{s=1}^{v} B_{rs}\dot{q}_s + B_r = 0; \qquad r = 1,.....,M \tag{2.26}$$

In this case, the dynamic behavior of the system is described in terms of the general forces $Q_s = \sum_{i=1}^{n}\left(\vec{F}_{e_1}\cdot\frac{\partial \dot{\vec{r}}_i}{\partial \dot{q}_s} + \dot{\vec{M}}_i \cdot \frac{\partial \dot{\vec{w}}}{\partial \dot{q}_s}\right)$ and the kinetic energy $T = \frac{1}{2}\sum_{i=1}^{n}\left(m_i\left(\dot{\vec{r}}_{i_s}\right)^2 + \vec{w}_i \vec{L}_i^{(S)}\right)$. Furthermore, if the lagrange multipliers λ are taken into account, the lagrange equation can be written as

$$\frac{d}{dt}\left(\frac{\partial T}{\partial \dot{q}_s}\right) - \frac{\partial T}{\partial q_s} = Q_s + \sum_{m=1}^{M}\lambda_m B_{ms}\ ; \qquad s = 1,....,p \tag{2.27}$$

which, linked with the equation (2.26), represents the $p + M$ equations for the unknown λ_m and q_r.

2.3 Software

2.3.1 Catia V5

Catia V5 (Computer Aider Three dimensional Interactive Application) is a multi-platform PLM/CAD/CAM/CAE commercial software suite developed by Dassault Systemes. The program is developed giving support from the design stage (CAD), to the production (CAM) and the analysis (CAE) of products. In Catia V5, the geometry is not created only with the geometric definition, but also with parametric features. For each task type that can be realized in Catia exists a specific Workbench. A Workbench in Catia is the denomination of the specific work environment for different tasks. Each Workbench contains a toolbar collection and drop down menu that are specific for each task realized [Catia-2007a].

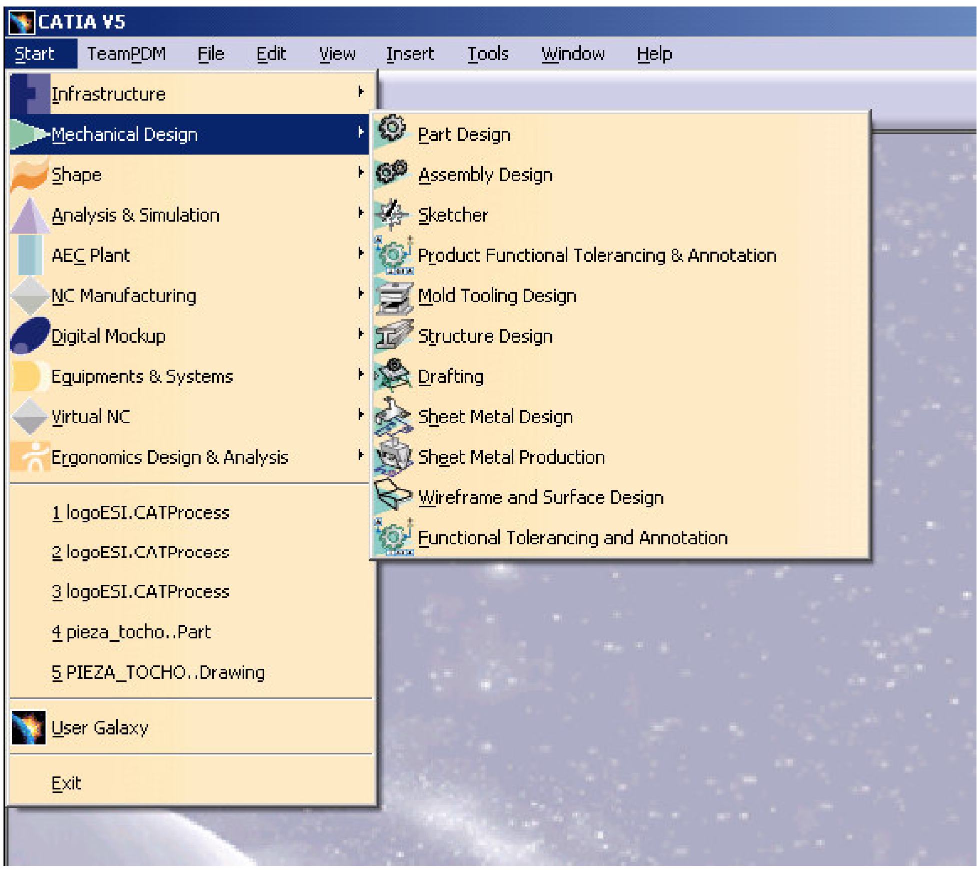


Figure 2.7: Catia environtment

The Workbenchs are divided in different categories, the main ones from which are explained briefly below:

-Mechanical Design

-*Part Design*: Workbench used for the construction of 3D parts, the toolbar of which contains elements such as PAD (filled), POCKED (emptied), or SCHAFT. This tool is a volume modeler and is used to manipulate geometric elements.

-*Assembly design*: Workbench used to assemble the different parts by means of joints, it helps in the production of a logical product structure to assemble under circumstances, where it can exist several hierarchies and hundreds of components.

-*Sketcher*: Workbench with a work environment to the create outlines in 2D and define the conditions for an exact determination of the geometric basis for a later modelling in 3D.

-*Shape Design and Styling*

-*Generative Shape Desing*: Workbench to construct the surface

-*Analysis*: It is used to integrate Finite Element Models (FEM) analysis.

-*Product Synthesis*: Digital Mock Up (DMU) Method

-*NC Manufacturing*: NC programming

A complex model is represented in a structure of components like products, parts and assemblies. It is possible to copy and paste components and joints, to introduce models, to replace components of the assembly, to modify components features, an so on. The designs in 2D are used in 3D to create parts, that are used to form products by means of joints and the final model is the union of all the products. This hierarchical structure of the model is defined in a tree structure, as it can be seen in the Figure 2.8. This tree structure is situated at the left side of the screen and the logic of each geometric object is defined placing the elemental objects in its branches. One of the usefulness that can be realized with the tree is the access and the possibility to isolate, visualize or partially hide the elements to simplify the work of a complex design.

Figure 2.8: Hierarchical structure defined in tree structure

2.3.2 MSC.ADAMS

ADAMS (Automatic Dynamic Analysis of Mechanical Systems), developed by MSC.Software Company, is a program based in the simulation of multi-body systems. It enables the user the production of virtual prototypes, a realistic simulation of the full-motion behavior of complex mechanical systems and a quickly analysis of multiple design variations until an optimal design is achieved [MSC-2005c]. In an early development stage, the dynamic behavior can be studied, without building a real prototype. Therefore, it reduces the number of physical complicated prototypes, improves the design quality, and dramatically reduces the product development time. The MSC.ADAMS contains software products classified in four main groups:

The core of MSC.ADAMS consists of three programs

-ADAMS/View (modelling)

-ADAMS/Solver (simulation)

-ADAMS/PostProcessor (evaluation)

Other products are:

-ADAMS/AutoFlex (elastic body)

-ADAMS/Controls (control technique)

-ADAMS/Durability (lifetime calculation)

-ADAMS/Flex (integration of FME-Bodies)

-ADAMS/Insight (design of experiments)

-ADAMS/Linear (linearization)

-ADAMS/Vibration (vibration analysis)

Some industry-specific products:

-ADAMS/3D Road (virtual auto-test track)

-ADAMS/Aircraft (test, improvement and optimization of aircrafts)

-ADAMS/Car (vehicle simulation with chassis, engine, drive train, control)

-ADAMS/Chassis (experimental design specific for motor vehicles)

-ADAMS/Rail (rail cars)

And CAD interface products:

-ADAMS/Exchange (data transfer between software)

2.3.2.1 ADAMS/View

ADAMS/View is a program that allows the building of models of mechanical systems and simulates the full-motion behavior of the models. It can also be used to quickly analyze multiple design variations until the optimal design is found [MSC-2005b]. The steps, which ADAMS/View uses to create a model, are the same, which would be used to create a physical prototype. The Figure 2.9 shows the steps used to create models and simulate them.

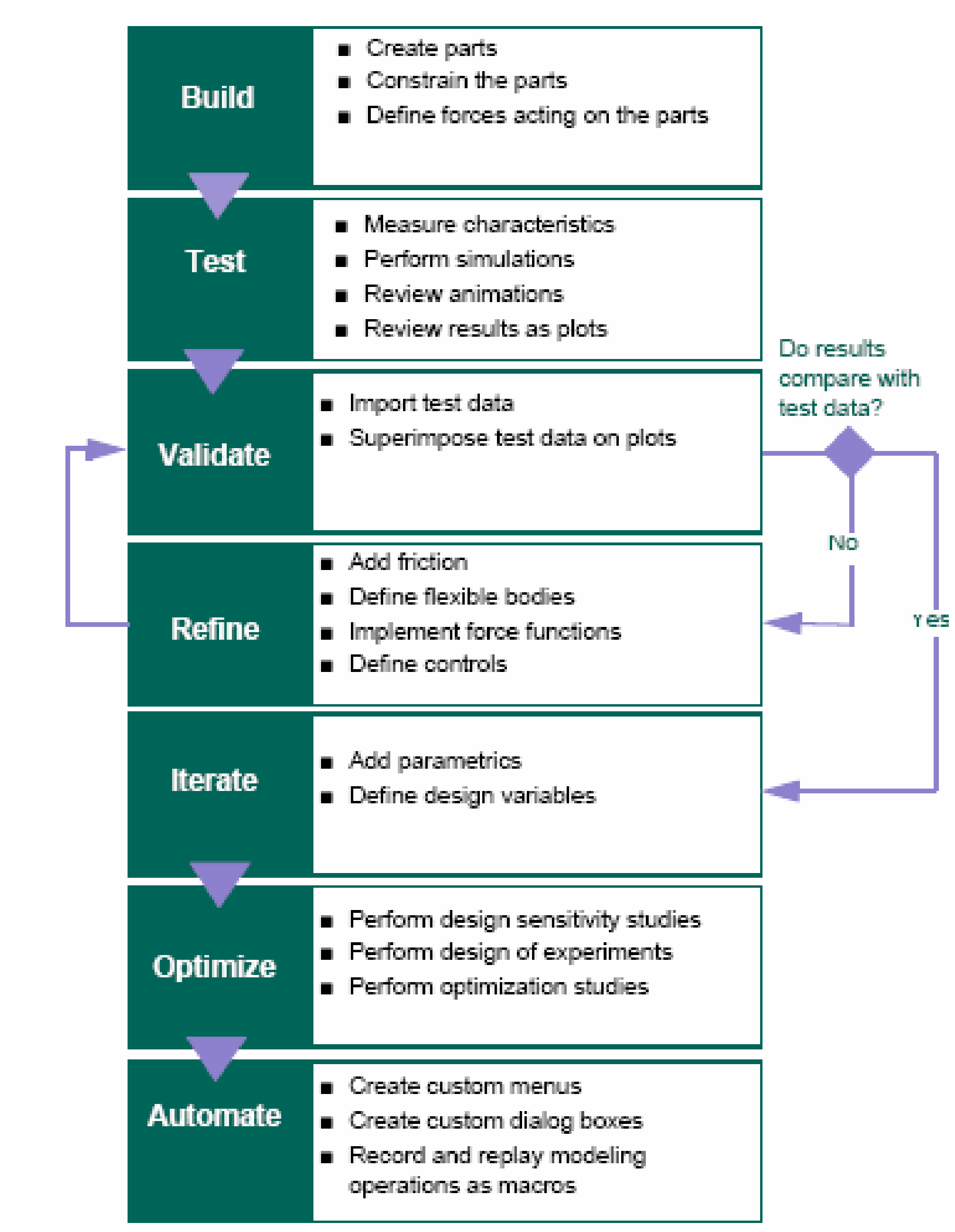


Figure 2.9: Step in modeling and simulation in ADAMS/View [MSC-2005b]

The first step to create a model is the building of the different parts that form the model, attaching them by means of constraints which allow the relative movements between each other and later apply forces on the model that will affect the part motion and reaction forces on constraints. Then, some tests of the model can be run to verify the system characteristics and to validate that the model is correctly created, comparing the results of the simulation with the physical tests of the mechanical systems. After running the initial simulations to validate the model, it can be improved adding more complex elements like friction or general state equations. Afterwards, the optimization of the model can be realized, which objective is to find the best combination of the design parameters, running automatically several simulations and varying the design parameters in each simulation. Finally, an automation of the model enables a quick change of different designs [MSC-2005b].

The main element used in Adams/View is the Main Toolbox. It contains tools used to create and simulate a model. The tools in the Main Toolbox are divided in submenus, which can be displayed using the right mouse button and only a left mouse button click is necessary to use a tool of those menus. The main menus used to create a model are shown in the figure 2.10:

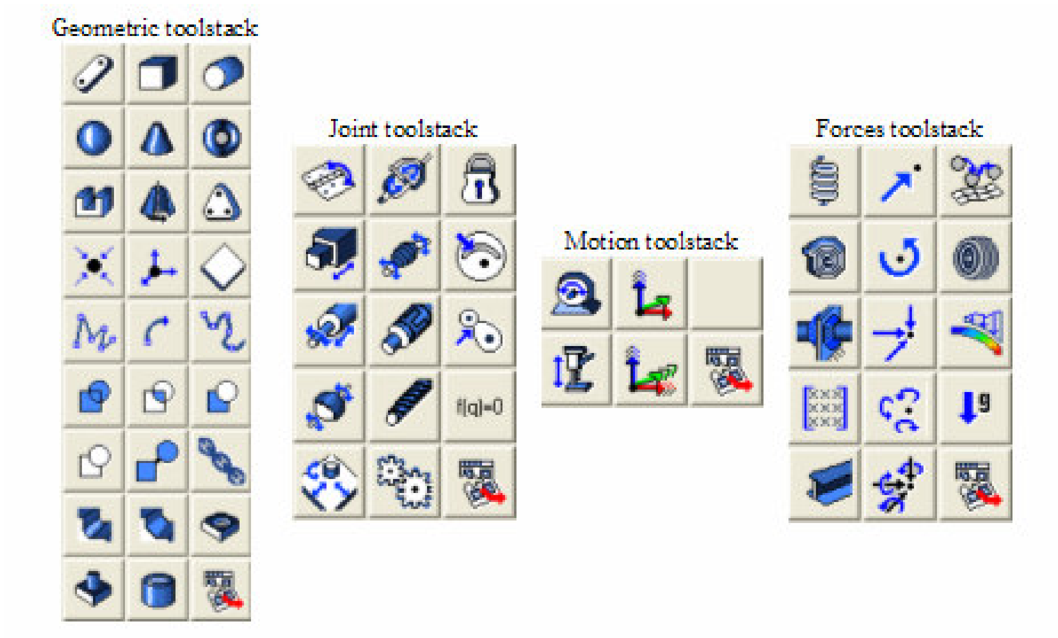


Figure 2.10: ADAMS/View Toolbox

·The geometric elements menu provides the necessary tools to create rigid bodies. With it, it can be introduced pre-defined geometric simple elements, apart from other elements as holes, curves or fillets. The mass and inertia properties are introduced automatically by ADAMS/View according to the body geometry and the materials, although other values can also be introduced manually if it is necessary.

·The joints are used to limit the movements by means of the elimination of degrees of freedom in singles components. The joints can be divided in two parts, idealized joints and joint primitives. The joint primitives have no physical counterparts and create only a restriction on the relative motion, contrary to the idealized joints, such as revolute joints or translational joints, which have physical counterparts.

·The motion menu is used to provide movement as a function of time, of pats, idealized joints or between a pair of parts of the system. It applies the force, rotational or translational, necessary to satisfy the desired movement.

·With the force and momentum menu, it can be created different types of forces. This different force types cover from applied forces and special forces to flexible connections.

The steps that ADAMS/View follows to create a model can be divided in two parts, one will be the creation of the model, making it as similar as possible to the real model and the other part will be the improvement of the real model with the help of the model created in ADAMS/View. To improve the model, it is realized a parametric analysis where ADAMS [MSC-2002a] runs a series of automatic simulations with different values for the design variables. This process helps to investigate the influence of the design variables on the model performance, offering the effects of the produced changes. ADAMS offers three types of parametric analyses:

-Design study

-Design of Experiments (DOE)

-Optimization

Design study

The design study analysis helps to understand the design variable behavior. The development of a design study comprises a series of simulations with different values for the design variable and gives the performance obtained in each simulation. Using a design study analysis, it can be determined:

-How the changes in the design variable varies the performance.

-Which of the simulated values is the best for the design variable.

-How is the sensibility of the design variables, understanding the sensibility as the relationship of change of the performance measure with respect to the variable.

Design of Experiments (DOE)

While in the design study only one variable can be varied in each simulation, in the DOE various variables can be varied in each simulation. Therefore, a DOE helps to know how several variables are related between them.

When the model is complex and involves many design variables, it should be considered to use an approach structure based on DOE techniques, because choosing the runs by intuition or trial-and-error can lead to confusion. The field of experimental design provides a set of procedures and static tools to plan the experiments and analyze the results.

Using DOE, it can be done:

-find the design variables that affect the performance of the model, acting alone or in a combination of several design variables.

-it can be realized a robust design or the Taguchi method, studying the effects of variations in manufacturing and operating processes.

Optimization

The optimization is a sophisticated tool that helps to improve the design. Moreover, it also helps to accelerate the design cycle when a good design is had in early stages. Most of the designs have specific requirements, for example, a specific vibration or a specific motion. Some aspects of the model, like that it is too expensive or too heavy, can to help to distinguish a poor model from a good model. Furthermore, the model can possess some restricting conditions such as material availability or work space.

When in a design process all the objectives and all the restrictions can be satisfied by means of the manipulation of the design variables and, moreover, all this features can be quantified, the optimization techniques can be used to analytically obtain the theoretical best solution. Also the process provides additional information to this questions:

- Are there too many adverse conditions that prevent a solution?

- Are there unnecessary design constraints, which dictate the results?

- To what design parameters is the design sensitive?

2.3.2.2 ADAMS/Solver

ADAMS/Solver is a software used to check the model, and formulate and solve the equations of motion for cinematic, static, quasi-static, and dynamic simulation [MSC-2005a]. The method used by ADAMS/Solver to solve the equations of motion is the Euler-Lagrange. ADAMS/Solver contains several numerical integrators to solve nonlinear differential-algebraic equations, for example, some which are suited to solve systems with differential natural frequencies. In ADAMS/Solver, force or motion can be described writing complex functions by means of a built-in function expression language and, moreover, it supports user-written subroutines. Two methods are used to call ADAMS/Solver, one method is in interactive mode where it is directly called, and another method is in batch mode by means of a dataset file (file extension *.adm).

2.3.2.3 ADAMS/Postprocessor

ADAMS/Postprocessor software is a powerful post processing tool that allows the view of the results of simulations performed using other products in the MSC.ADAMS suite of software [MSC-2005b]. ADAMS/Postprocessor can represent the results of multiples simulations allowing making comparations between them. The results can be showed on different ways such as graphs in 2D and 3D or animations.

2.3.2.4 ADAMS/Vibrations

ADAMS/Vibrations is a plugin of ADAMS/View where it can be performed frequency-domain analysis. It is used to study the forced vibrations in the model. The process followed by ADAMS/Vibration to realize the analysis is showed in the figure 2.11.

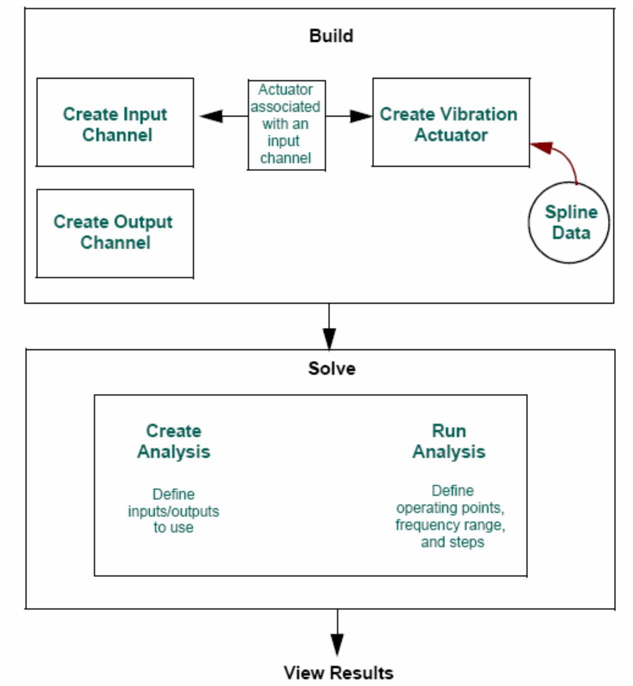


Figure 2.11: Process to realize an analysis with ADAMS/Vibration [MSC-2002b]

Furthermore, with ADAMS/Vibrations it can be performed:

- study, over different operating points, of the forced response.

-study of the frequency analysis, including hydraulic effects, controls and user subsystems.

-the computation of the eigenvalues, eigenmodes, transfer function, and power spectral density (PSD) that linearize the system.

-different types of input functions, such as swept sine, power spectral density, and rotational imbalance.

-frequency-based forces.

-frequency range to analyse system modes.

-frequency response analysis for magnitude and phase characteristics by means of ADAMS/Postprocessor.

3 State of the Art

In this chapter is presented an overview of the literature that has a similar research topic to this project one, the modelling in the machine tool environment.

3.1 Modelling in the Mechanics Field

In the development of machine tools virtual prototypes are more and more used with the aim to increase the product quality while reducing time-to-market and production costs. The machine tool design leads beyond analyzing given part structures and optimization of their stiffness and mass properties within a pre-described design space. However, most important are the geometrical dimensions and the interfaces between the parts because they influence the machine tool's static and dynamic behavior [Broos-2006].

The 3D-CAD-Systems are the central development tool in modern mechanics. They are increasingly used for simulations and optimizations in the field of computer-based technologies. The CAD used in the system design is the basis of the derivation of several mechanical calculation tools. Among the main calculation tools, there are the Finite-Element-Method (FME) and Multibody Simulation (MBS). The Finite-Element-Method (FME) has been a method of choice for decades when it came to determinate vibration modes or static stiffness and it is the most commonly used method throughout the machine tool industry for the simulation and optimization [Broos-2006]. In CAD is not necessary, for instance, to delete, restructure or attach parts, the insertion of coordinate systems or the regrouping of assemblies, since it is specific simulation methods prepared . However, in the topology optimization by means of FEM, it can have some disadvantages such as the load points, which often must be manually modified. Therefore a good method is the coupling of FEM into MBS. For this purpose, FE-bodies are introduced in a MB-System, creating a hybrid MBS-Model and by means of it, the simulation of the static and dynamic behavior of one model components on several positions in the workroom can be realized [Neithardt-2004]. Apart from MSC.ADAMS used in this project, in the market are other software packages as DADS, WorkingModel or SIMPACK.

The development of the Multibody Simulation Method was very important in the aerospace and automotive industry and, moreover, it was a priority in the biomechanics and robotics research, therefore, its use is there firmly established nowadays. However, in the machine tools industry, it is not yet totally established and it is relatively few used compared with the before mentioned industries. The reason of this low use is that the number of pieces is minor and the computation department in machine tool industry is smaller. Due to time pressure and the cost, is seldom possible to widely verify the models.

In a virtual vision of the product development process, the widely used MBS-program,is the central interface for a multidisciplinary simulation. However, the coupling with other suitable programs is still necessary. These important couplings are used, for instance, in fluid mechanics research or thermal dynamic problems [Ardnol-2004] [Veitl-2001].

The use of Multibody Systems in the machine tool analysis is increasing. It can be seen especially in the university environment, where are being realized numerous works and research. One example can be the modeling of feed axis like an oscillator with multiple degrees of freedom, which can be found in books as for example [Weck-2001a]. The study of the dynamic system in a hexapod-milling machine, by means of rigid body and taking into account the chipping process, is realized by Grossmann [Grossmann-2004]. Also the integration of flexible bodies is more and more extended in the mechanic researches. One example of this is the investigation of a machine tool with parallel kinematics by means of the combination of flexible and rigid bodies [Neithardt-2004]. Baudisch [Baudisch-2003] and Zäh [Zäh-2004] researched the dynamics of the conventional machines. In [Zäh-2004] is in foreground the modelling of a dynamic structure and of the chipping process. And in [Baudisch-2003] the integration of all the system elements in the environment simulation is in the foreground.

4 Individual Approach and Proceeding

4.1 Individual Approach

The goal of this thesis is the study of the parameter in the dynamic behavior of a machine tool feed axes. The study of the dynamic behavior represents an important rule in the manufacturing quality of a machine tool due to its influence in the accuracy of the final product. It is important to introduce as soon as possible the study of the machine behavior in the product development process and with it the possibility to realize improvements on the machine tool in early stages of its design. The analysis of mechanic systems is developed in different steps, where, each next step is closer to the real behavior of the studied systems. The first step in the analysis is the study of the behavior of the machine tool in the frequency domain. This step is an important start point in the analysis, since can be done a response study of different parameters with regard to one or several input signal. In the following step is realized a time domain analysis. With the time domain analysis can be studied the parameters with different motion of the machine and therefore a study closer to the real machine behavior.

The model design is realized in Catia and for the multibody simulation is used the software MSC.ADAMS. To link these two programs, the model is exported from the CAD-System Catia to ADAMS/View by means of SimDesigner, which realizes a smooth and easy export in most of the cases. If the model needs to be optimized after the analysis in MSC.ADAMS, it can be exported to optimization software as Optislang by means of Adams/View Command File and the Batch-Mode.

The results of this project can to be used to determinate the utilized methods quality to show the dynamic behavior of a system. Is possible to do this determination comparing the results with others results obtained by means of others methods, although, because of cost and time problems this validation is often not realized. Furthermore, one more step can be realized using this project as base. That step would be the search of the optimum parameters to improve the dynamic behavior of the machine using optimization software.

4.2 Proceeding

The first step of this thesis is the design of the simplified model, whose properties are probably the most similar to the machine testbed. That requires carrying out the following steps:

- Study of the original model.
- Creation of a simplified model in the CAD-Software (Catia V5).
- Processing and exporting the CAD-Data from Catia V5 by means of SimDesigner interface.
- Importing the model in ADAMS and validation of the imported elements.

• Change of the component properties to the original model properties and creation of the most important variable.

• Development of the flexibility of the model using rigid and flexible connections between the contact points of the different parts and validation of the introduced elements with force and simulation tests.

• Creation of the friction between the parts in movement.

The second step is the development of the analysis and obtaining the results. This is achieved following the procedure given below:

• Creation of the motion and the measure points to realize the time domain analysis.

• Creation of the input and output channel for the frequency domain analysis.

• Validation of the proceeding and first approach to the results with the study of each flexible connection individually introduced in the rigid model.

• Analysis of the elements that below to the transmission in the flexible model such as bearings, linear guides or coupling.

• Evaluation of the results.

5 Building the Model

In the first part of this chapter is presented the building of the complete model, which is a simplification of a real model, and the process to export it to the analysis software. Then, the preparation of the model is done to analyze it, first in a general way and later individually for each one of its components. Finally, is realized a description of the settings and analysis tools used in MSC.ADAMS to analyze the model.

5.1 Original Model and Simplified Model

The machine testbed is a reused component from a former testbed, which has been rebuilt in SimCAT-Project framework. The testbed base was built of concrete, where were placed functional surfaces for the bearings and guide rails support. In the former configuration, were embedded two steel bodies into the concrete to the place the bearing blocks. In the new configuration, the surfaces of the two steel bodies are too far away to allocate the new bearings. Therefore a new surface is built to satisfy it. This new surface screwed onto the two steel bodies is a longitudinal beam that possess a large surface where are screwed the bearing supports. Moreover, two steel bars were glued onto the base concrete whose objective is to be the functional surfaces for the linear guides. And to separate the testbed from the ground were used four wedge shoes [Hennrich-2007].

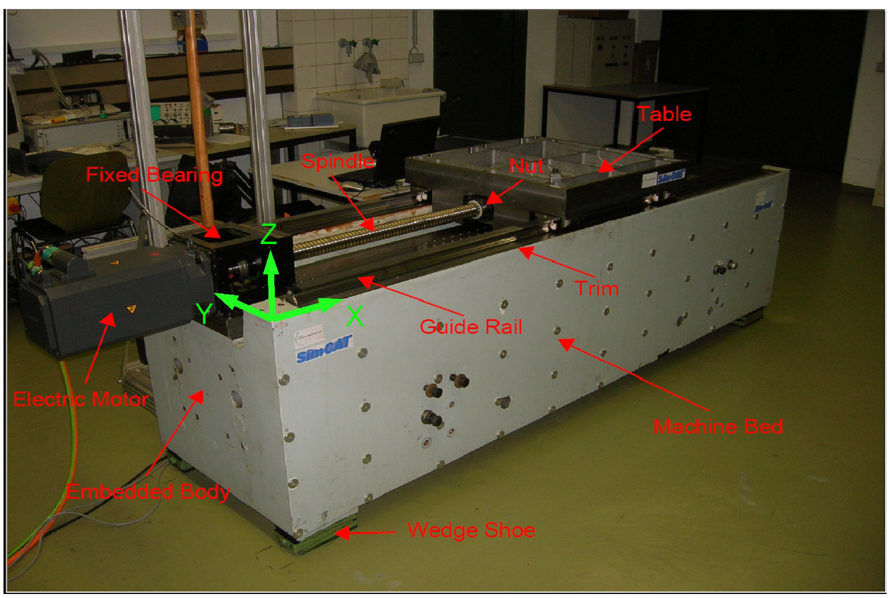


Figure 5.1: Testbed front [Hennrich-2007]

Inside of each of the two bearing blocks are placed different types of bearing. In the bearing block closest to the motor is introduced a fixed bearing and in the bearing block furthest from the motor is introduced a loose bearing. The bearing block with the fixed bearing is also used to hold the electric motor, which is screwed onto it. The bearings support both sides of the spindle, which is connected to the electric motor inside of the bearing block by means of a coupling [Hennrich-2007].

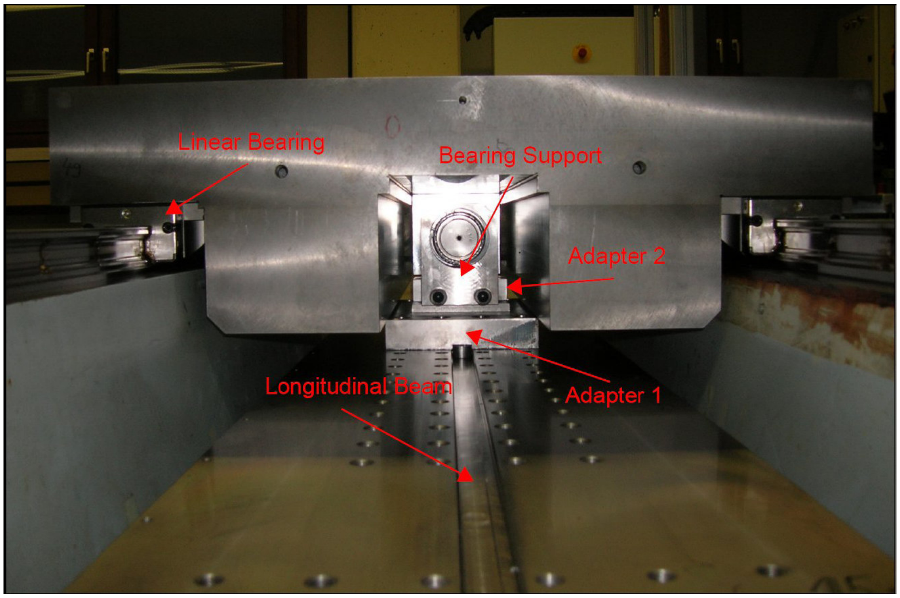


Figure 5.2: Testbed rear [Hennrich-2007]

In the table are allocated four linear bearings, two in each side, used to support the table upon the base by means of two rails screwed to the steel bars. Moreover, the table has a spindle nut, which is adapted to the table with an adapter. This nut transmits the rotational movement from the spindle to a translational movement in the table [Hennrich-2007].

Based on the original model, a simplified model has been developed. The reason why the original design is not used are the different simulation problems that this model presents, with whose results can not be obtained a reliable analysis and, moreover, the work with this model was very thorny and complicated. It was attempted to solve these problems for a long time, but because of time reasons and other different causes it was given up. Therefore, the solution to these problems was building a simplified model.

The simplified model shown in the Figure 5.3 was created carrying out a drastic reduction of the parts contained in the original model. For it, all the parts linked to other parts by means of fixed joints were designed as a unique part, which form was as simple as possible. With all this, the main objective was that the simplified model had a behavior as similar as possible to the original model. With these bases was designed a model which is formed of five main parts: the machine bed, the spindle motor, the spindle, the table and the ground. In the machine bed part are integrated some parts of the original model such as the base, the longitudinal beam, the steel bodies, the steel bars, the guide rails, the bearing blocks or the bearings as a single part linked with fixed joints to the ground and to the motor. The union of the spindle nut, the adapter, the four linear bearings and the original table led to the table, which is linked to the machine bed by means of flexible joints and to the spindle by a screw joint. The spindle is supported by the machine bed with flexible connections and at the same time coupled to the spindle motor by a flexible coupling.

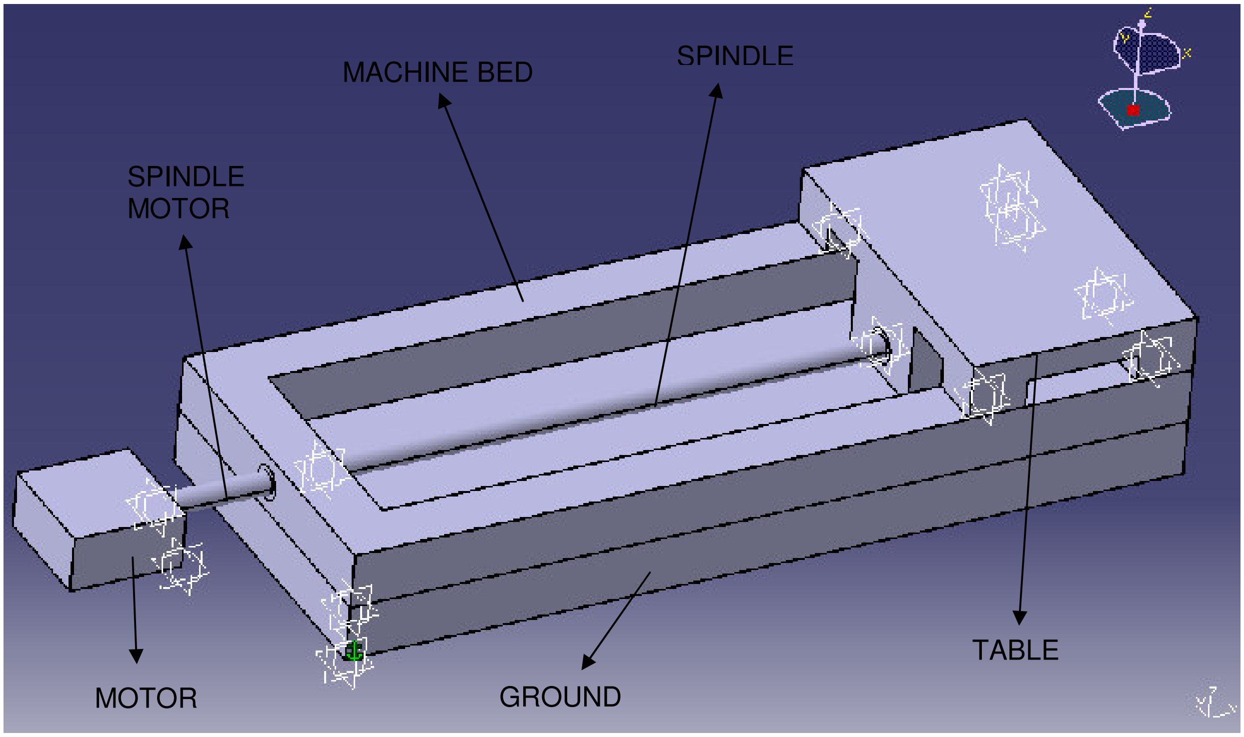


Figure 5.3: Simplified model

5.2 Modeling the Mechanics

The simplified model, based on the original model of the testbed, is designed as a CAD-Model. In order to get a model as similar as possible to the real model and the simulation analysis, the modeling of this CAD-Model has been realized using the same work-type specific programs. The procedures to export and import the model between these programs are the same as in other thesis, as [Inderbirken-2006], in which the same type of modeling have been followed. When the model is imported it has to be reviewed with the objective to avoid certain errors of exportation and certain parameters have to be recalculated. The model is imported as a rigid model which is given the elasticity and the desire motion by means of elements such as bushing, forces and joints connected with help bodies. The parameters introduced in this elements, in this case obtained of Klotzbuecher [Kotzbuecher-2007], influence highly the results in a multibody simulation. Therefore, the introduction of the appropriate parameters is essential to obtain optimal results. The values of these parameters (which is obtained mainly by the manufacturers) are not exact values because they present uncertainties due to different reasons. The results of the simulations obtained with these values, according to different researches, show that some differences are found in the system behavior. However their correspondence with a real model is sufficiently valid according to Klotzbuecher [Kotzbuecher-2007]. In this chapter, the choice of the joints and the determination of their appropriate parameters are developed.

5.2.1 Processing the CAD-Model

The initial design called Simplified Model, and shown in the Figure 5.3, was carried out in the design software denominated Catia V5. For the construction of the model in CATIA it is important to understand some essential terms (see also 2.3.1):

·The Parts are the smallest pieces of a model, which can be formed with a lot of parts.

·The Products are the biggest elements that form a model and therefore are the highest level of construction in Catia. The products can be formed by one or more single parts linked by means of constraints.

The building of the model has been created designing each part individually. Initially, in the *Sketcher* Workbench has been drawn each part in 2D, to later give it volume in the *Part Desing*. These two Workbenches can be used connected until the design is exactly as desired. When the parts have been built, it is proceeded to fix these parts by means of constraints using the *Assembly Design* Workbench. Before exporting it from Catia to ADAMS it has been reviewed if the model still contained individual parts called *Open Shell Elements*. In this case, they have to be deleted because in ADAMS these parts will not be created correctly. For it, can be opened this part in Part Design Mode and by means of Tools → Unnecessary Elements delete it.

5.2.1.1 Exporting the Model from Catia

To export the CAD-model was used an interface developed by MSC.Software denominated SimDesigner. In this program it is also possible to realize multibody simulations in Catia, but it does not contain all the options available in ADAMS. For this reason, the model must be exported to ADAMS opening it in the SimDesigner environment with the function Start → Digitale Modellerstellung → SimDesigner Motion. In the menu bar appears an additional menu denominated SD Motion. With this menu and using SD Motion → Add Ground Part, a part or an assembly is defined. This part or assembly is fixed to the inertial system. All the other parts or assemblies have to be selected with SD Motion → Add Moving Part.

Now begins the exportation action strictly speaking. The export is started with SD → Export to ADAMS/View and in the dialog window, Option With Geometry and Results Export it is put No activated

5.2.1.2 Importing the Model in ADAMS/View

The model data is exported from SimDesigner as a Command File (*.cmd) and as a Shell-Dateien (*.shl). The model data can be red in ADAMS/View with the Command File by means of: File → New Database → Import a file → OK. Subsequently, the data type and the data are chosen. The bodies imported to ADAMS/View are not solid bodies but shells. Therefore, to have solid bodies in ADAMS the model has to be exported by means of File → Export as parasolid (*.xmt_txt) and with the import function File → New Database → Import a file → OK to be imported again. When the model is imported in Parasolid-Geometry, the units must be adjusted by means of Settings → Units because in ADAMS millimeter-kilogram-second are used. The bodies also have to be checked just in case have not been really imported as solid.

5.2.1.3 Assignation of the Mass and Inertial Properties

When the model is imported in parasolid-Format, it loses the mass and inertial properties. They can be manually introduced again for each part. These properties are introduced by means of the density and, then, taking into account the basic geometry of the body, ADAMS recalculates them.

5.2.2 General Remarks on Modeling

The feed axis model was initially created as a pure rigid body model. The method to give elasticity to the model was the introduction of discrete spring-damper-elements and, therefore, Finite Element Method was not used for this purpose. Therefore, all the bodies are rigidly created. All the elements that belong to the transmission such as bearings, linear guides or spindle nut were modeled elastically and for it, help bodies were used. The friction was introduced in bearings, linear guides and spindle nut with the help of forces, translational ones in the case of the linear guides or rotational forces in the case of the bearings and spindle nut.

5.2.2.1 Orientation

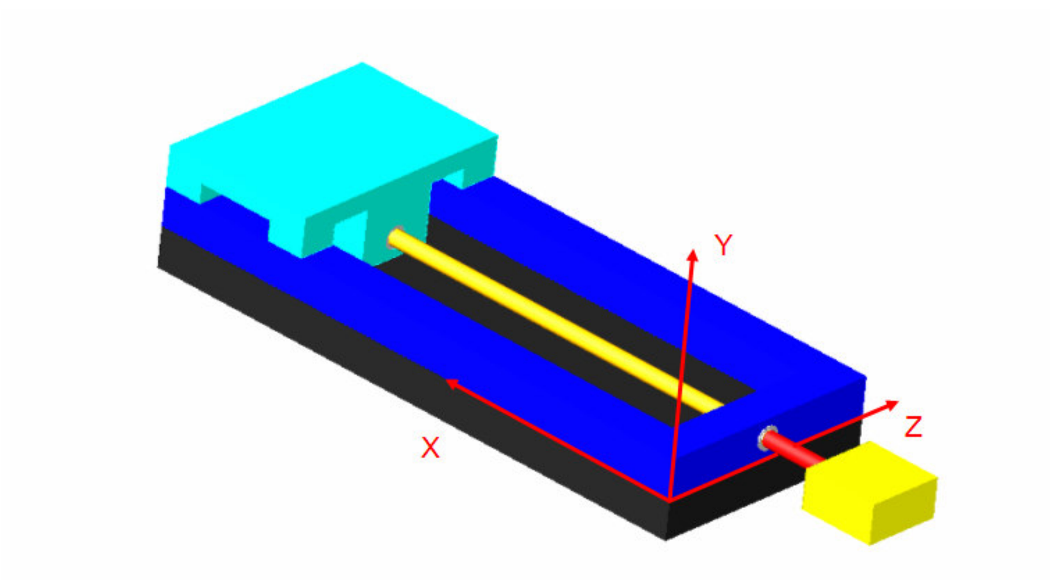


Figure 5.4: Orientation of the model in ADAMS/View

In the Figure 5.4 is presented the orientation of the model in the global coordinate system. All the motions and loads are defined in this orientation. Therefore, it can be seen that the trajectory followed by the table is along the X-axis. When it is possible, all the elements of the model are oriented in this coordinates, although this is not always possible. When that happens, it can also be hand changed.

5.2.2.2 Mass Properties

The mass properties are automatically introduced by ADAMS, taking into account the material type and the geometry of the rigid body. With the objective to do this model realer and with lower variations than the original model, the mass properties have been manually introduced. Therefore, the mass value of each rigid body that forms the simplified model is the sum of all the masses which constitute this element in the original model. A summary of all the mass properties introduced in the simplified model is shown in the appendix C.

5.2.2.3 Damping

To search the damping values exist a lot of difficulties and uncertainties. Numerous models have been published whose objective was the search of the damping analytical value. Although, according to Bürgel [Bürgel-2001], it has not been satisfactorily solved yet, since these uncertainties can cause differences in the vibration amplitude.

The values introduced in the simplified model for the damping have been got of experimental values and measured values or calculated by Klotzbuecher [Kotzbuecher-2007] following the methods used in others projects which have the same objetive.

When is not possible to use experimental values or measured values, can be taken into account a very good idea to calculate the damping realized by Eubert [Eubert-1992] which have been follow by Klotzbuecher [Kotzbuecher-2007]. This idea is based in the definition of a single mass vibrator:

$$d_i = 2D_i\sqrt{c_i m_{ie}} \quad \text{or} \quad d_i = 2D_i\sqrt{c_i J_i} \tag{5.1}$$

In the Table 5.1 are shown some reference values for D_i.

D	Description
0,1...0,15	Damping of (hydrodynamics) sliding guides
0,03...0,05	Damping of bearing etc.

Table 5.1: D values Klotzbuecher [Kotzbuecher-2007]

5.2.2.4 Modeling the Friction

The friction can be directly modeled in ADAMS with idealized elements. This option can be found in some idealized joints where the static and dynamic coefficient of the friction force can be introduced. This option has been rejected in this project and the friction has been introduced by means of another additional elements according to Klotzbuecher [Kotzbuecher-2007]. This additional elements used in this thesis are the *General Forces*, with which can be designed forces and moments in the direction of the coordinate axes. To place idealized elements is necessary to create at least two *Markers*. The I-Marker is the part where the force flow is directed to, therefore, the part that receives this force flow and by defect it is placed in the action-part. And the J-Marker is the one situated in the reaction-body. With a General force can be defined the friction by means of a function and that way can be introduced the addition of the different parts friction and the nonlinear process can also be represented [Kotzbuecher-2007].

Other method used by Klotzbuecher [Kotzbuecher-2007] to model the friction moment, when it is dependent on the velocity, or rotation speed or it is nonlinear, is using *Splines*. With these elements can be realized a diagram in ADAMS, where can be represented the friction moment characteristics as an input parameter. For the utilization of this parameter is only necessary to realize *Build → Data Element → Spline → New*. Then, can be introduced the data table of preview plots and adjust it with different options.

To know better the friction behavior and the characteristics of differents method used for its definition it can be seen in the Apendix H.

5.2.2.5 Help Bodies

In the modeling of flexible connections is advisable the utilization of help bodies which have been already used in other researches as in [Neithardt-2004]. These help bodies have not connection with the real model, therefore their mass should be as small as possible. In this case help bodies have been introduced in all the connections. The design of theses bodies is different according where are they situated. In the linear guides, these bodies are spherical and in the bearings and ball screw, their form is cylindrical. On one hand, these bodies have connected the bushings which give flexibility to the connection and on the other hand are connected the friction forces and idealized joints which give motion to the connection. To delete the non-desired degrees of freedom that this type of connection has, the bushing stiffness in this direction should be very high. For this motive and because of the small mass of these bodies, sometimes this method is not advisable due to numerical calculation.

5.2.3 Modeling of Transmission Parts

The information with which have been modeled the transmission parts, have been obtained from Klotzbuecher [Kotzbuecher-2007].

5.2.3.1 Modeling the Bearings

The modeling of the fixed and the floating bearings is done by means of one *Bushing* and one *General Force*. The *Bushings* are flexible connections with which can be defined the stiffness and the damping in all the coordinates of the space, the three translational coordinates and the three rotational coordinates. And with the *General Force* are introduced forces and torques in the six coordinates of the space. For the modeling of the bearings are used help bodies and two revolution joints connected between the spindle and the help bodies.

5.2.3.1.1 Stiffness

In the bearings, the motion around the rotation axis is permitted, therefore, the stiffness value in these coordinates is zero. In the case of the floating bearing, the movement in the axial direction is also permitted, therefore, this value is zero too. For the other stiffness values, the information has been offered by INA/FAG company. For the fixed bearings, it is used an Axial-Schrägkugellager (fixed bearing) model ZKLN-3072 and for the floating bearings is a Nadellager (floating bearing) model NA 6908-ZW. Their values are shown in the Table 5.2.

Bearing type	x-Direction	y-Direction	z-Direction	Unit
NA 6908-ZW	0	7,69 × 1E5	7,69 × 1E5	[N/mm]
ZKLN-3072	1,4 × 1E6	2,4 × 1E5	2,4 × 1E5	[N/mm]

Table 5.2: Bearings stiffness value [Kotzbuecher-2007]

5.2.3.1.2 Damping

The damping values used in the simplified model have been taken from the original model, which calculations have been realized by Klotzbuecher [Kotzbuecher-2007] using the equation (5.1). Like in the case of the stiffness, the bearings do not have damping in the rotation axis neither in the axial direction of the floating bearing. The results of the calculations can be seen in the Table 5.3.

Bearing type	x-Direction	y-Direction	z-Direction	Unit
Fixed bearing	195,3 · D	113,1 · D	113,1 · D	[N·s/mm]
Floating bearing	0	176,6 · D	176,6 · D	[N·s/mm]

Table 5.3: Bearing damping value [Kotzbuecher-2007]

5.2.3.1.3 Friction

The friction torque is modeled with a *General Force* connected to a help body and to the machine bed. The calculation for the determination of the friction quantity of a bearing is offered by the manufacturer, according to Kotzbuecher [Kotzbuecher-2007]. For the realization of these calculations, it is necessary to know the bearing characteristics; the load, the rotation speed and the lubricant viscosity.

The formula to calculate the total torque friction M_R is written as:

$$M_R = M_0 + M_1 \tag{5.4}$$

where the torque M_0 is directly related with rotation speed of the following form:

$$M_0 = \begin{cases} f_0 \cdot 160 \cdot d_M^3 \cdot 10^{-7}, v \cdot n < 2000 \\ f_0 \cdot (v \cdot n)^{\frac{2}{2}} \cdot d_M^3 \cdot 10^7, v \cdot n \geq 2000 \end{cases} \tag{5.5}$$

being f_0 the additional value that depends on the rotation speed, d_M the average value of the bearing diameter, v the cinematic viscosity and n the rotation speed.

And the torque M_1 is directly related with the load following:

$$M_1 = f_1 \cdot F_r \cdot d_M \quad \text{for needle bearing,}$$
$$M_1 = f_1 \cdot P_1 \cdot d_M \quad \text{for angular ball bearing with} \tag{5.6}$$
$$P_1 = X \cdot F_{a\mathrm{Res}} - Y \cdot F_r$$

being f_1 the additional value that depends on the load, F_r the radial load, P_1 the total load for the friction load, X and Y additional values and F_a the axial load.

The creation of the friction function in ADAMS is realized by means of the *Function Builder*. In the first part of the function is defined the torque M_0 that depends on the rotation speed. Because of that is necessary the creation of a *Spline* following: *Build → Data Element → Spline → New*. On it is written the value of M_0 in relation with the rotation speed. The call to the Spline in ADAMS is realized using the syntax AKISPL (VARVAL(.model_JUAN.VARIABLE_VELOCITY_LOSLAGER) , 0 ,SPLINE_Loslager). With VARVAL(.model_JUAN.VARIABLE_VELOCITY_LOSLAGER) are obtained the current values of a *state variable,* which contains the multiplication of the rotation speed and the cinematic viscosity of the lubricant.

In the case of the torque M_1, it is necessary to know the resulting force in the radial direction. For it, exists a function in ADAMS called BUSH which returns the force values resulting in the *bushing elements*. The syntax of this function is BUSH(Force_Name , On_Body , Component , Axes). With another function denominated SIGN is connected the part that depends on the load with the rotation speed of the system. This is not necessary with the part dependent on the speed rotation because these dates have been kept in the *Spline*, where ADAMS can read them directly.

5.2.3.2 Modeling the Ball Screw

To model the Ball Screw has been utilized one help body. On one hand, to connect the spindle and the help body has been used a *Screw joint*, which possess a pitch of 25, therefore, per each revolution of the *spindle* the *table* is moved 25mm. On the other hand, between the table and the help body there is a *Bushing* and a *Translational joint,* allowing only the relative motion between these two parts in the X-axis of the model. Therefore, the movement in this axis is regulated by the stiffness and the damping introduced in the *Bushing*. The stiffness and the damping values are shown in the Table 5.4.

Transmission part	Stiffness		Damping	
	X-axis	Unit	X-axis	Unit
Ball Screw	1,153 × 1E6	[N/mm]	1081 · D	[N·s/mm]

Table 5.4: Ball screw stiffness and damping values [Kotzbuecher-2007]

5.2.3.2.1 Friction

The friction torque in the ball screw nut is created by means of a *General Force* connected to the help body and to the machine bed, therefore, parallel to the *Screw joint*. This friction is placed as a torque in the X-axis and its constant friction torque, according to [Kotzbuecher-2007], is 2,04Nm. The definition of the friction is realized in the *Function Builder* of the following manner:

```
SCALE_BALL_SCREW
*(-1*(2.04E+003)
*STEP(VARVAL(.model_JUAN.VELOCIDAD_MUTTER),-1,-1,1,1)
*STEP(ABS(VARVAL(.model_JUAN.VELOCIDAD_MUTTER)),1,1,1.001,0)
-1*(2.04E+003)
*STEP(ABS(VARVAL(.model_JUAN.VELOCIDAD_MUTTER)),1,0,1.001,1)
*SIGN(1,VARVAL(.model_JUAN.VELOCIDAD_MUTTER)))
```

Figure 5.5: Definition of the Friction in the ADAMS/View *Function Builder*

In the first part of the function can be found the coefficient SCALE_BALL_SCREW which is a *design variable*. This coefficient is used to change the friction value, which objective is realize an analysis about the friction influence in the system behavior. In this case it has a standard value of one. The first Step Function creates a smooth characteristic curve of the friction function in the point zero in a range from -1 to +1. The second Step Function is used to set to zero the lines 1 and 2 when the function is not in the interval [-1, +1]. The third Step Function introduces a constant friction value when the velocity is bigger than 1.001. And finally, the Sign Function is directly related with the motion direction, because the friction torque should work in opposite direction to the input torque.

5.2.3.3 Modeling the Linear Guides

The linear guides have been modeled using also help bodies between the table and the machine bed. The table motion along the X-axis is allowed by means of four *Translational Joints* situated two in each side of the machine bed, between this and the help bodies. And to get the flexibility on the joints, four *Bushings* are placed between each table leg and the help bodies. In the case of the linear guides, the rotation is allowed around the three axes and, moreover, exists also relative movement in the Y and Z direction. For the movement in the X-axis, as this is given by the four *Translational joints*, a very high stiffness value has been introduced in this coordinate, which objective is to delete this movement.

The stiffness values, according to Klotzbuecher [Kotzbuecher-2007], have been offered by Bosch-Rexroth company and they are shown in the Table 5.5 with respect to the global coordinate system used in the simplified model.

Direction	Stiffness value	Unit
Y-axis	7,14 × 1E5	[N/mm]
Z-axis	8,57 × 1E5	[N/mm]
Rotational X	1,5 × 1E5	[N·mm/º]
Rotational Y	1,5 × 1E5	[N·mm/º]
Rotational Z	1,5 × 1E5	[N·mm/º]

Table 5.5: Linear guide stiffness values [Kotzbuecher-2007]

For the damping values the Eubert method [Eubert-1992] is not good, therefore, the values have been obtained from the SimCAT-Project according to Klotzbuecher [Kotzbuecher-2007]. These results are in the Table 5.6 and, like in the stiffness, they are indicated respect to the global coordinate system of the model.

Direction	Damping value	Unit
Y-axis	6,25 · D	[N·s/mm]
Z-axis	4,389 · D	[N·s/mm]
Rotational X	587,34 · D	[Nmm·s/º]
Rotational Y	248,7 · D	[Nmm·s/º]
Rotational Z	516,62 · D	[Nmm·s/º]

Table 5.6: Linear guides damping values [Kotzbuecher-2007]

5.2.3.3.1 Friction

The linear guide friction is made up of the gasket friction, which is the most important, the rolling friction, the dynamic friction and the lubricant friction.

The friction has been calculated according to Kotzbuecher [Kotzbuecher-2007] with the following formula:

$$F = \mu \cdot F_{res} + f \tag{5.7}$$

where μ is the friction coefficient which possess a value of 0.002, f with a value of 20.4N is the gasket resistance and F_{res} is the equivalent load of the linear bearing in the main direction of the load (horizontal in Z-axis and vertical in Y-axis), $F_{res} = |F_y| + |F_z|$. The other parts of the total friction are not taken into account because their influence is small.

The friction function has been created in the ADAMS/View *Function Builder* in a similar manner as in the case of the ball screw nut. In this case, the loads have been red in ADAMS as state variables. And the behavior friction is realized by means of the following function:

$$F(v) = -(f + F_{res}) \cdot step(v,-1,-1,1,1) \qquad \text{for } |v| \le 1$$

$$F(v) = -(f + F_{res}) \cdot sign(v) \qquad \text{for } |v| > 1 \tag{5.8}$$

5.2.3.4 Modeling the Motor

To model the Motor have been necessary a *revolute joint* and a *Force*. With the *Revolute joint* has been got the relative motion between the Motor and the Spindle Motor in the X-axis. On the other hand a rotational force, *Torque*, has been also introduced to introduce the movement in the system. For it, this *Torque* has been placed in the X-axis of the Spindle Motor. With it, in the Spindle Motor is introduced a motion which will give movement to the Table by means of the transmission elements. To make the Table describe the desired trajectory, has been used a *Spline* whose values have been designed to obtain this movement.

5.2.3.5 Modeling the Coupling

The coupling has been modeled deleting all its degrees of freedom except the rotation around the X-axis. For it, a revolute joint has been used. Moreover, a bushing is introduced to give flexibility in these axis. The real coupling is produced by KTR Kupplungstechnik GmbH company and the type is Rotex GS38 98Sh A-GS. Its values are in the Table 5.7.

Transmission part	Stiffness		Damping	
	Rotational X	Unit	Rotational X	Unit
Coupling	2,05942 × 1E4	[Nmm/º]	147,48 · D	[Nmm·s/º]

Table 5.7: Coupling stiffness und damping values [Kotzbuecher-2007]

5.3 Setting the Analysis Tools

5.3.1 Developing the Time Domain Analysis

The realization of the analysis in the time domain can be easily carried out in ADAMS. The first step is the design of the desired motion in the system that, as it has been explained in 5.2.3.4, has been realized by means of a *Spline.* With this method, any motion can be introduced to the model. In the time domain can be obtained the measurement of the magnitudes that are necessary. Various forms can be used for the acquisition of the results such as *Requests*, *Expression Functions* or *Measures*. In this case, has been used the *Measure Points* method. For it, several *Markers* have been created in the points where the results have to be collected. Next, to create a *Measure Point,* the desired *Marker* has to be clicked with the right button of the mouse, then a dialog box appears where, *Measure*, has to be selected. With it, a window is displayed, which is showed in the Figure 5.6. In it can be seen the settings realized to create a measure.

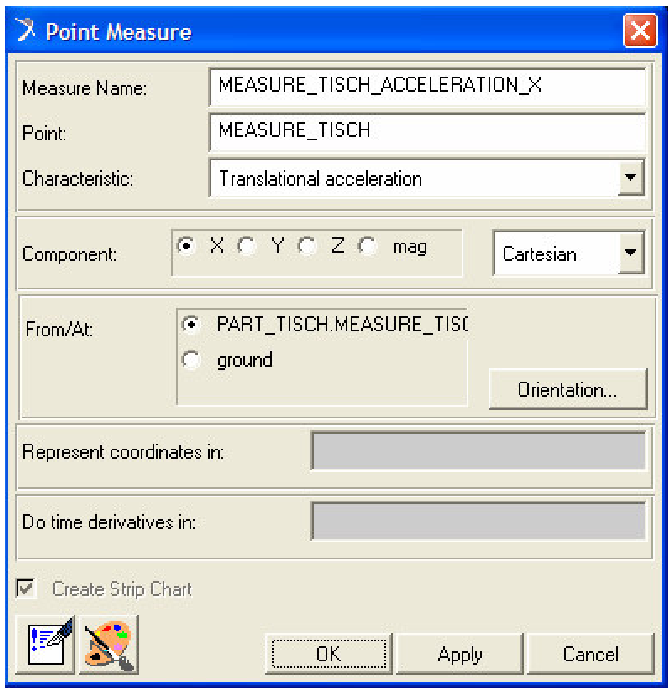


Figure 5.6: Point measure window

With all the steps previously realized, is possible to proceed to the time domain analysis. The simulation can be developed directly in the *Main Toolbox* but it is desired to use the Simulation Control because of the multitude of settings that can be realized with it. All this settings can be seen in the Figure 5.7. In this case the duration time of the simulation is 0.5 sec. and the simulation steps are 200. When it is necessary that the simulation is externally produced, as in the case of an external optimization, can be used the option *Scripted,* where is only necessary to previously create a *Script*.

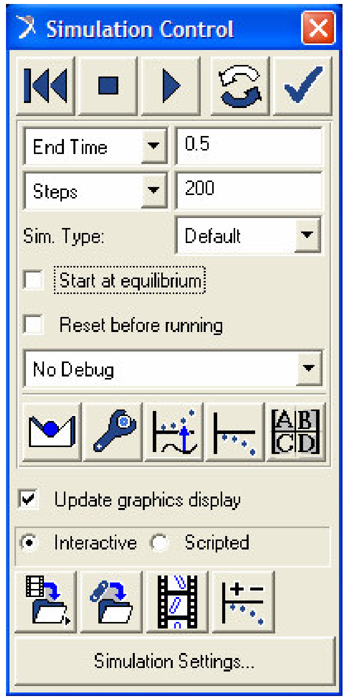


Figure 5.7: Simulation control window

5.3.2 Developing the Frequency Domain Analysis

In this part is shown the procedure to develop frequency domain analysis in ADAMS environment. A frequency domain analysis consists on the introduction of one or several inputs that will be used to excite the system with the objective to collect, in a wide frequency range, the results in any desired point of the system by means of outputs. Therefore, through the analysis of the input and the output signals can be calculated the frequency response. For this type of investigation is necessary the utilization of the module ADAMS/Vibration, which is included in MSC.ADAMS.

In the case of the input, to excite the system has been used kinematic input in the X-axis of the *Spindle Motor*. For the creation of it, has been followed this procedure, *Build* → *ADAMS/Vibration* → *Input Channel* → *New*. The settings realized in the *input channel* are shown in the Figure 5.8. For the *input channel* can be used *forces*, *kinematic* or *User-Specified State Variable* inputs. Obviously, the input magnitude will affect the magnitude of the obtained response frequency, which will be greater for a bigger excitation magnitude and smaller for a smaller excitation magnitude. Therefore, for a good study of the mechanism, only one magnitude value is used in these simulations.

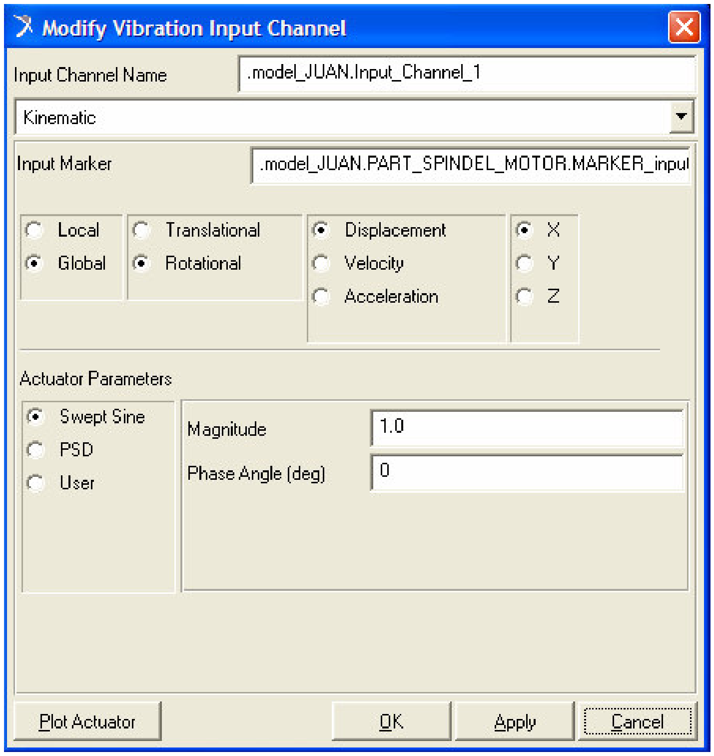


Figure 5.8: Input channel window

The analysis of the response of the system is realized with *output channel*, *Build → ADAMS/Vibration → Output Channel → New*. The results can be obtained in any desired point of the system. For it, only has to be introduced the markers in the desired places. In this case is used a predefined *output channel* but can also be introduced a function expression. All the *output channels* have been created the same way, only varying the predefined magnitude and the place in each case. In the Figure 5.9 are shown the settings realized in one of these *output channels*.

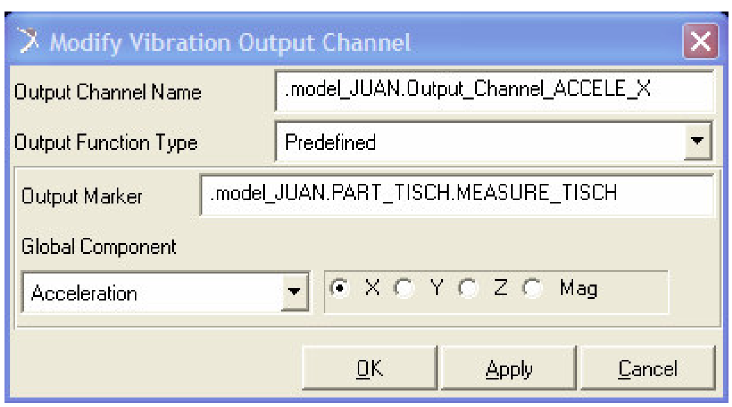


Figure 5.9: Output channel window

The execution of the frequency domain analysis is realized by means of *Simulate → ADAMS/Vibration → Vibration Analysis*, and, afterwards, an emergent window appears. This window can be seen in the Figure 5.10 and in it can be realized all the settings for the vibration analysis. In the Pull-Down menu on the top-left part is possible to realize a New Analysis, or import the settings of previous analysis if other analyses have been developed and the settings are valid for this analysis. In this case the Operating Point used is Assembly but there is the option to realize an external analysis using Script. Moreover, also Forced Vibration Analysis and Damping have been selected. The input and output channels can be easily called with the right button of the mouse. To obtain the results in different frequency values, the Frequency Range option has to be checked. In this case, the used range varies from 10 to 10000, divided on 10000 logarithmic steps.

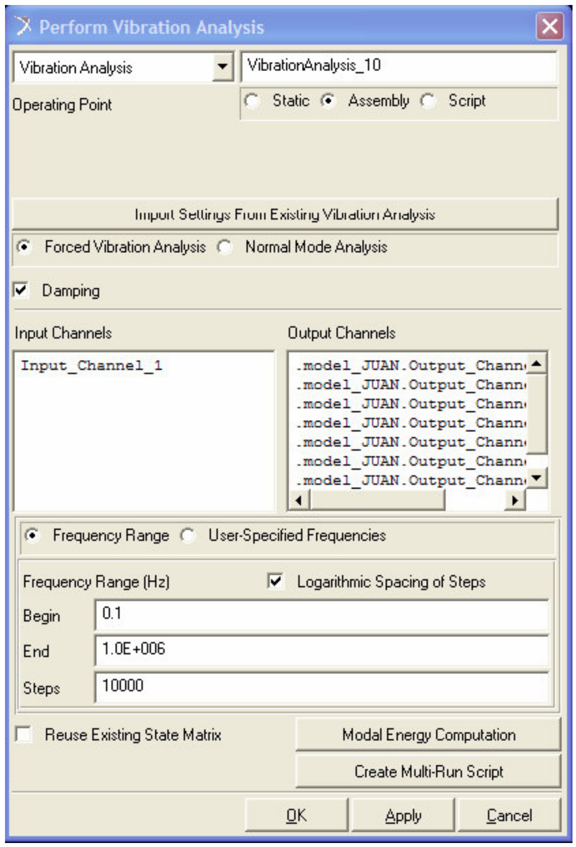


Figure 5.10: Vibration analysis window

5.3.3 Preparing the Parameter Analysis

In ADAMS a parameter analysis can be realized using Simulate →Design Evaluation. With a parametric analysis can be investigated the design variables influence on model. During a parametric analysis, ADAMS runs a series of simulations with different design variables values and gives the feedback on the effects of the changes [ADAMS ayuda]. The study of the variables can be realized individually as in the case of a Study Design or several variables can be combined in the same analysis as in a Design of Experiments (DOE).

To create a parameter analysis in frequency domain is necessary to realize a vibration analysis, as has been explained in 5.3.2. It is necessary because the information used in that vibration analysis is utilized to create a script simulation. Then, when an analysis vibration has been previously realized, to create a script simulation has to be opened again a vibration analysis window. In this window, the option Create Multi-Run Script must be clicked, with it a new window appears in the screen, Figure 5.11. In it, the information of the vibration analysis has been introduced and confirmed clicking *OK*.

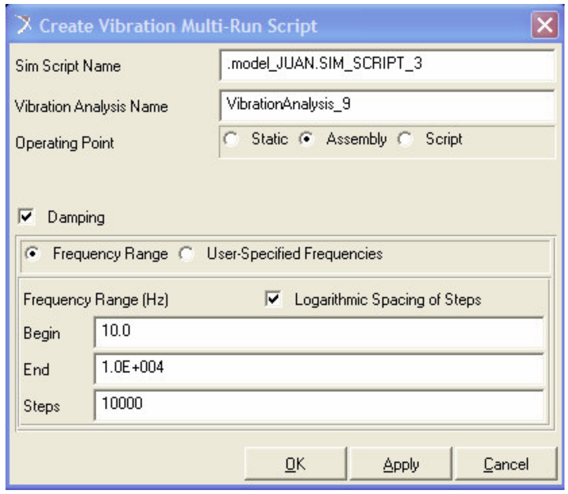


Figure 5.11: building a Vibration Multi-Run Script in ADAMS

Apart of this, to realize a Design Evaluation in frequency domain it is necessary to create an objective, Simulate → Design Objective → New. In this objective window, in the *Definition by* Pull Down menu of the Figure 5.12 must be selected */View Variable and Vibration Macro*, then a new dialog window will appear. The settings of this new dialog window, called the Design Objective Macro, are shown in the Figure 5.13. In it has been set *1 Input, Multiple Outputs* in *Target Vibration Data* to obtain the results in various output channels.

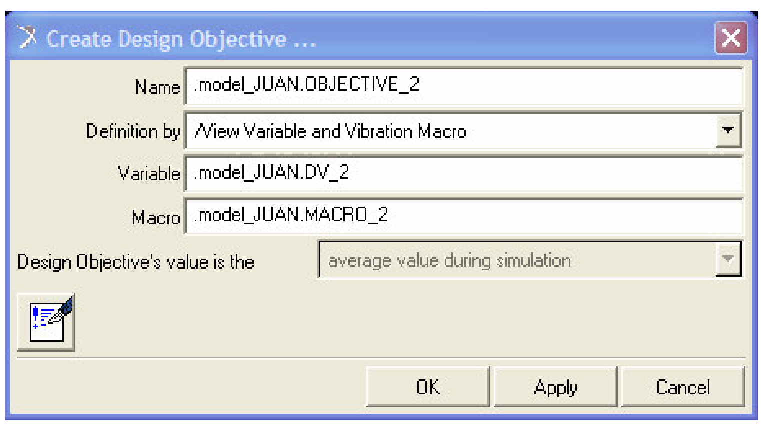


Figure 5.12 : defining the objective

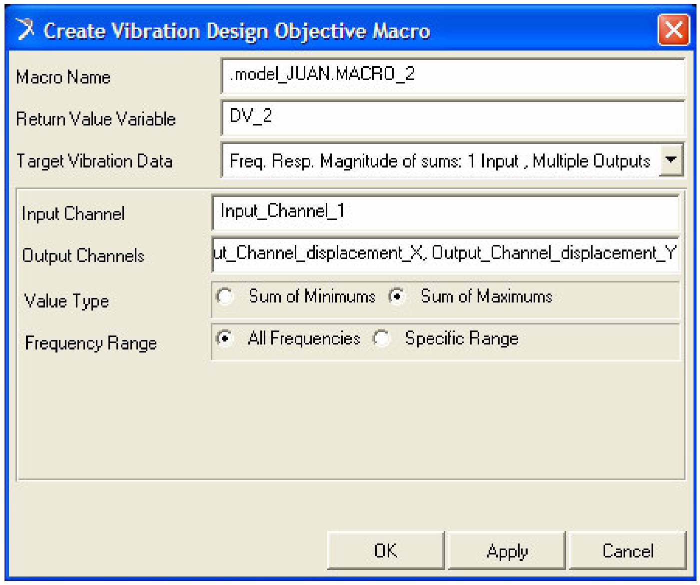


Figure 5.13: defining the Ojetive Macro

Now, the Design Evaluation has to be opened. Depending on if it is desired to realize a frequency domain or a time domain analysis, the settings in the Design Evaluation window, Figure 5.14, are different. In the frequency domain case, the vibration multi-run script and the objective are introduced in the corresponding place. Next, are introduced the Design Variables, in which dialog window has been introduced the desired range to be varied in the analysis and, finally, the number of simulation runs is adjusted in Default Levels.

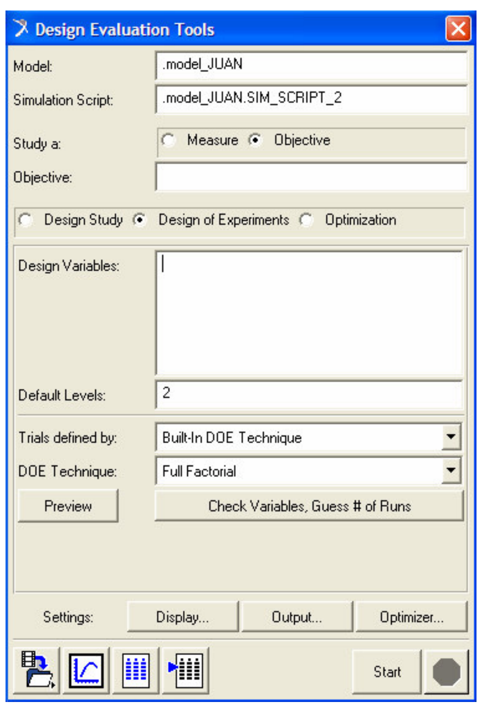


Figure 5.14: Design Evaluation Tools window in ADAMS

In the case of a time domain analysis is not necessary to create an objective, only introducing a measure it can be realized, but in this case, another script simulation has to be created with Simulate → Simulation Script → New. The settings of this new script are shown in the Figure 5.15, where can be seen that the simulation duration is 0.5 sec and is divided in 100 steps.

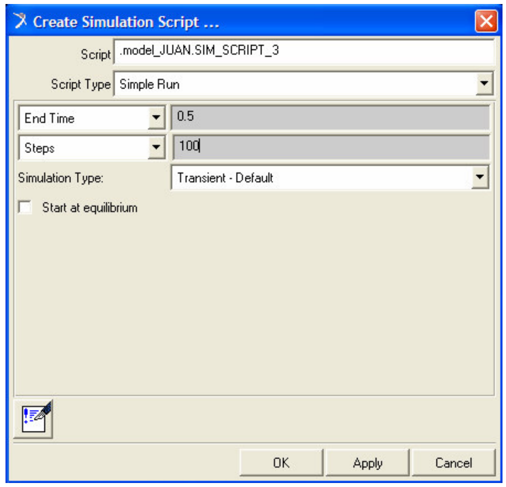


Figure 5.15: building a Simulation Script for the time domain

6 Results

In this chapter are shown the results obtained with two analysis types, time domain and frequency domain. In the first part is realized an introduction to the dynamic behavior. On it, can be seen the results obtained with the rigid model and with only elasticity in the coupling. In the following point, are shown the results with the variation of different parameters such as stiffness, damping and friction. The variations are produced in the interface components when the model is totally elastic. And, finally, is carried out an evaluation of the results.

6.1 Relation of the Results with Previous Thesis

The results obtained in this work are an extension of the results obtained with the original model realized by Klotzbuecher [Klotzbuecher-2007]. In the Klotzbuecher thesis development, the objective was the study of the influence of the parameters using MATLAB/Simulink to obtain the results. The study of the influence of the parameters in that thesis was centered only on obtaining the results in the frequency domain analysis, therefore, having these results as a basis, in this thesis has been realized the analysis of the parameter influence in the frequency domain by means of ADAMS/Vibration and an additional study of the parameter influence in the time domain analysis. With the utilization of these two analysis methods can be realized a greater study of the dynamic behavior of the testbed and, moreover, an approach to the real behavior of the machine is given by means of the time domain analysis. Although in this section will be shown and commented the results of both analysis methods, frequency domain and time domain, the results obtained in frequency domain will be a confirmation of the results obtained by [Klotzbuecher-2007] in MATLAB/Simulink and, furthermore, can be analyzed what happens when the spindel and the shoes elasticity is eliminated. Moreover, the results in frequency domain can be useful to understand better the results in the time domain and with them can also be realized comparisons between the two methods.

With the objective to obtain a dynamic behavior as similar as possible to the original model studied by Klotzbuecher, the modeling of the simplified model is realized of the same way, see section 5. If the simplification of the model is not taken into account, the most important differences between this model and the original one is the elimination of the spindel and the shoes elasticity, as it has been said in the previous paragraph, and the utilization of help bodies for the modeling and the construction of the model.

6.2 Simplified Model Results with the Calculated Parameters

The frequency response obtained with the simplified model in the frequency domain is shown in the Figure 6.1. These results of the frequency response of the model have been obtained in the output channel designed to measure the displacement of the X-axis. With these results can be confirmed the similarity of the simplified model to the original one studied by Klotzbuecher [Klotzbuecher-2007] which results are shown in the Figure 6.2. Although, between the two Figures it can be seen that the frequency which is producing the resonances is not completely similar. This is due to the natural frequency of the system, equation (2.4). Therefore, the main reason of this variation can be the difference of mass to the original model due to the simplification of its parts, in the single parts, in the simplified model or a no-exactly stiffness introduced. To make possible the comparison between the simplified model and the original model of Klotzbuecher has been necessary the elimination of the spindel and the shoes elasticity. Moreover, to verify the similarity of simplified model to the original model, the comparison of the model with the results obtained in the real testbed research of the other thesis has been realized with positive results. Therefore, with this, the first objective of this project which was the construction of a simplified model with the same behavior than the original model has been achieved. The obtained results about the parameter influence in the frequency domain have been contrasted with the original model, used to obtain the Figure 6.2.

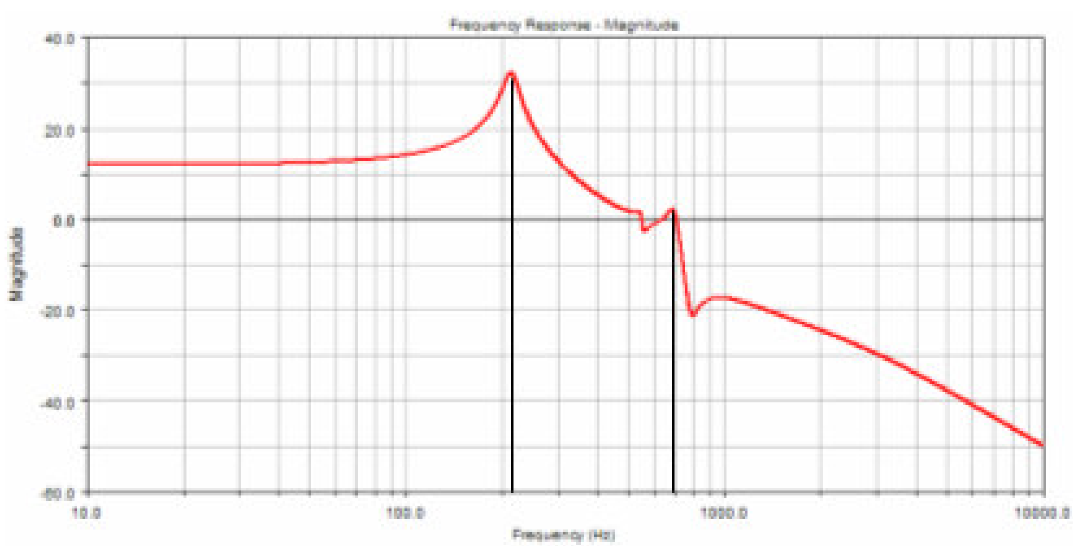


Figure 6.1: frequency response in the X-axis of the simplified model

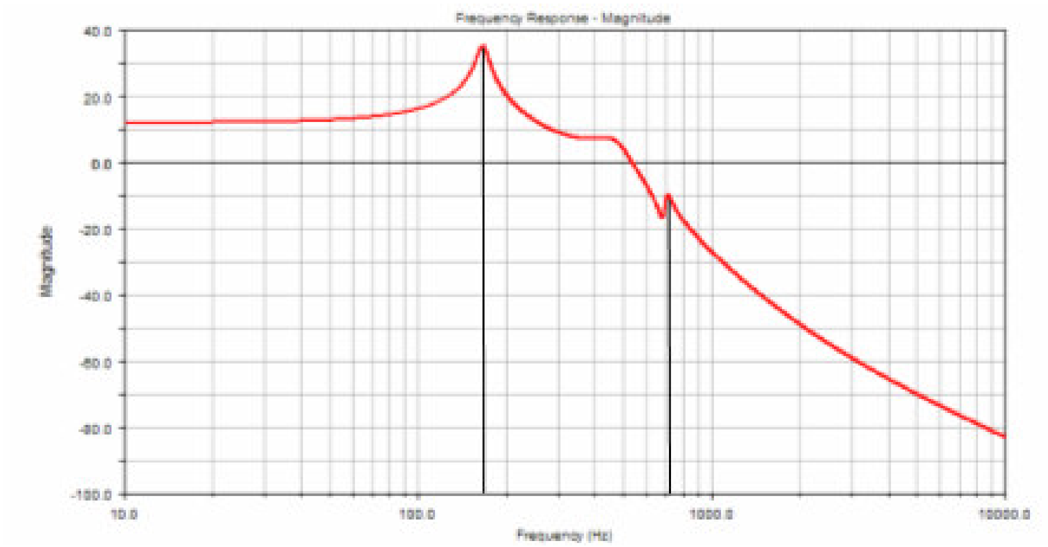


Figure 6.2: frequency response in the X-axis of the original model

To develop the time domain analysis has been necessary the introduction of a motion on the Table in the work direction, see section 5.2.3.4. The motion has been designed aiming to simulate all the movements that the Table can realize during its operation. Therefore, the Table realizes an initial motion along all the length of the machine bed, then a movement up to half of the machine bed is realized, to finally stop almost on the opposite side to where the motion has begun. This motion can be seen for an easily understanding in the Figure 6.3. The time in which the Table has to cover this displacement is 0.5 sec. This time range is the one used to obtain all the results in the time domain.

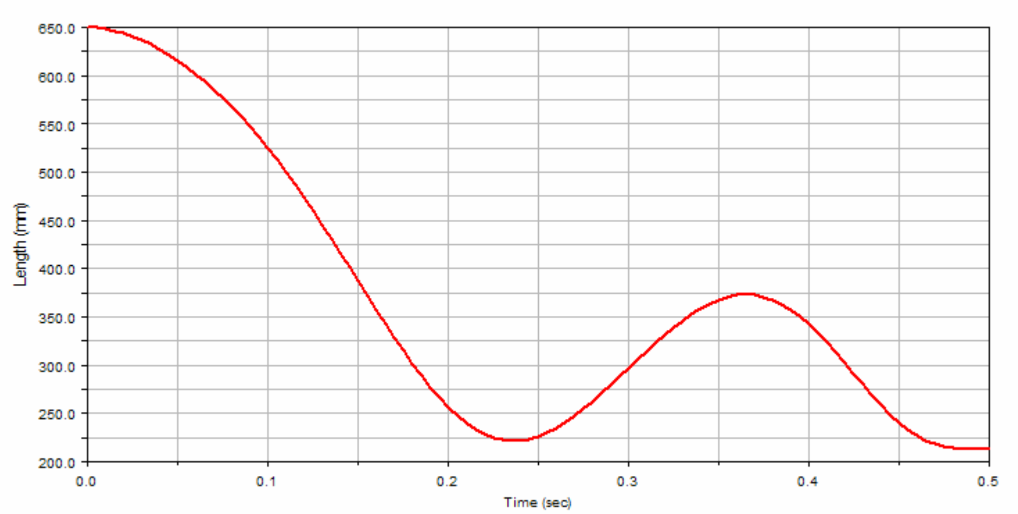


Figure 6.3: displacement of the Table in the X-axis

The consequence of this long displacement in so little time is an elevated acceleration value during the route, Figure 6.4. With these great accelerations, approx. between 50000 and -50000 m/s², is expected that the obtained results are clear. Other interesting results about the parameter influence obtained in the time domain analysis can be seen in the following sections, although not all are shown. On these figures is compared the calculated standard value with the results obtained in the Design Evaluation realized in ADAMS.

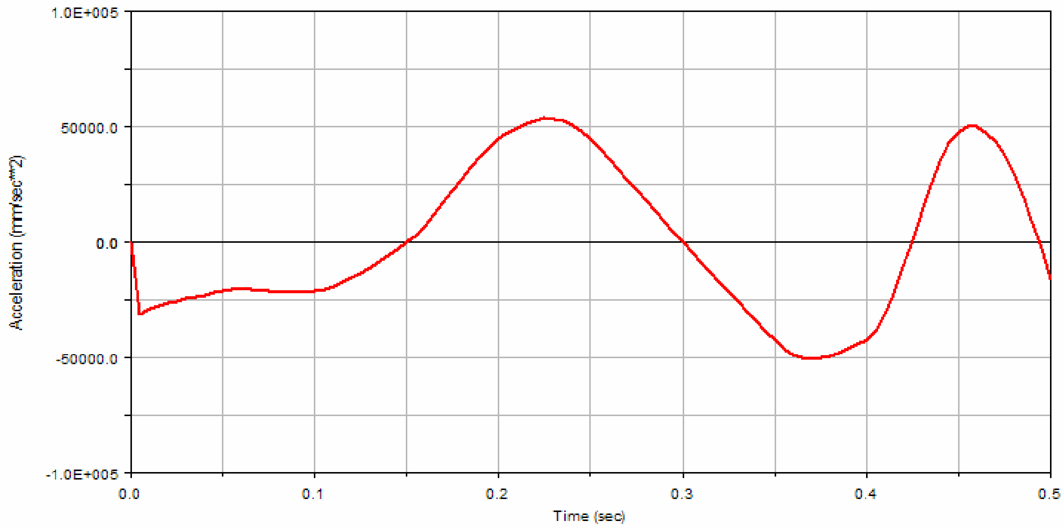


Figure 6.4: acceleration of the Table in the X-axis

6.3 Parameter Variation Influence

The main objective of this analysis is to study the individual parameter behavior of each element in the frequency domain and, above all, in time domain. The study is realized individually for each parameter of each element because, if several parameters would be analyzed at the same time, the influence of a parameter or element could be influenced by others and therefore, this type of analysis could create confusion. The objective of this individual analysis is to determine the influence of each parameter on the system and how this can give information about which could be the critical design or which could be the best combination of parameters to improve the machine behavior. Furthermore, thanks to the utilization of the two analysis methods, can be obtained information about the advantages and disadvantages of each method and the results in time domain and frequency domain can be complemented for a better analysis.

In the case of the frequency domain, the results are obtained as a frequency response, which predefined magnitudes are the displacement and the acceleration in the three axes of the coordinate system. In the time domain the results obtained are measured directly on the model, in which measure points are obtained the displacement and acceleration in the X, Y and Z axes. The point where the measure points and output channels are placed is the central point of the Table surface.

In these results is only realized the variation of the parameters of the elements which influence is most important in the system behavior. Five simulations are carried out for each varied parameter and only the most important results are shown in the following Figures.

6.3.1 Coupling Influence on the Rigid Model

In this section is realized an introduction to the dynamic behavior. For it, is analyzed the coupling behavior in the rigid model. With this purpose, has been deleted the elasticity of the other interface of the transmission, such as the spindel nut, linear guides or bearings and, moreover, the friction forces have been set to null. Thanks to it, any other influences in the analysis are eliminated. The results obtained are divided in two parts, frequency domain and time domain, as it can be seen in the following point.

6.3.1.1 Time Domain

The purpose of these simulations is to find the differences that appear in the rigid model with respect to the displacement in the X-axis of the Table, produced by variations of the stiffness in the coupling. The differences of the values are very small compared with the displacement values of the Table. In order to show the differences found in the results obtained, it will be proceeded to realize the difference between the result obtained in a rigid coupling, which is used as reference, and the results obtained in the simulations realized in an elastic coupling with different stiffness values. An example of it can be seen in the Figure 6.5.

To obtain the result of the rigid coupling, used as reference, an extremely high stiffness value has been introduced, 1E+15 Newton*mm/deg, which can be considered as rigid. In this first part, the analysis is centered in the stiffness and, therefore, the damping has been deleted, setting it to a value of zero. The used stiffness values are comprised from 1E+2 to 1E+13 increasing logarithmically its value.

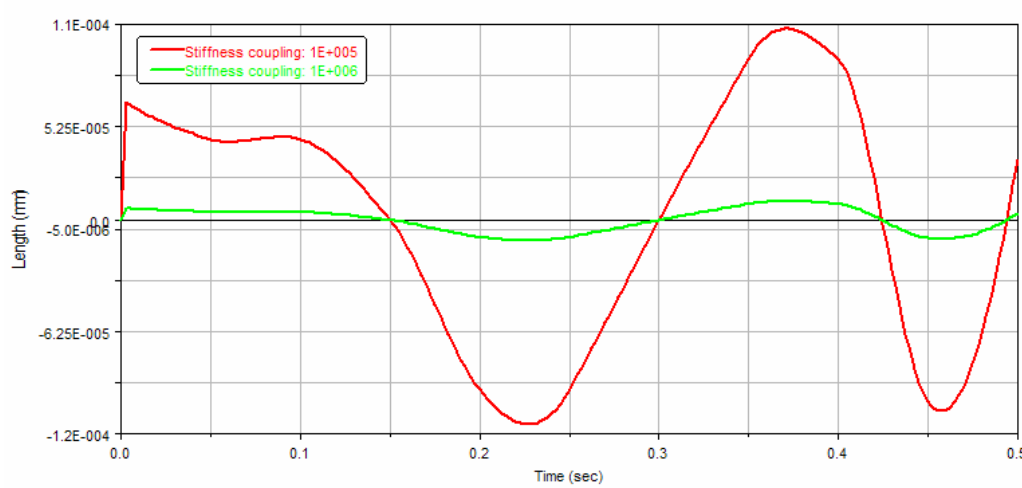


Figure 6.5: displacement difference between the rigid coupling and an elastic coupling with different stiffness values.

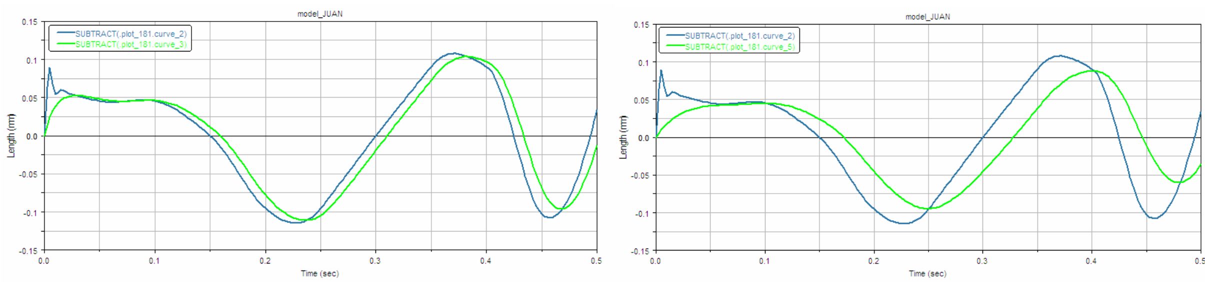
In the Figure 6.5 are shown two examples of what happens in all the obtained results. As it can be seen in the illustration, the value of the obtained curves decreases logarithmically when the stiffness value in the elastic coupling is logarithmically increased. With these results, can be created a relation between the amount that the stiffness value increases and the amount that the difference between the rigid and elastic coupling decreases. This relation can be defined with the following equation derived from the results obtained

$$Y = C \cdot e^{-10X} \tag{6.1}$$

Where Y is the value of the magnitude, in this case the displacement, X is the value of the stiffness and C is a constant. This C constant is different in each moment of the simulation and its value is equal to the displacement when the coupling stiffness value is equal to 1. It can not be obtained directly, since when the stiffness value is 1 the Table does not almost have motion. So that its value must be found of using another method, for example, for a fixed time of t=0,0025 and a stiffness X=100 the magnitude is Y=0,064 and for the same time and a stiffness X=10, the magnitude is Y=0.64, therefore, it can be supposed that for a stiffness whose value is X=1 the resulting magnitude would be Y=6.4, which is the calculated constant value introduced in the approximated equation. This equation can only be used when the stiffness values are not excessively low or excessively high. It can, also, be seen how at the beginning of the movement, the displacement increases quickly, starting from zero. This is due to the elasticity of the coupling which allows the gradual transmission of the torque, unlike in a rigid coupling, where the force will be transmitted instantly. The form of the curves is directly related to the acceleration of the Table, according to the equation (2.1). For this reason, for high acceleration values in the Table are also produced high values in the obtained curves and vice versa.

The obtained results for the acceleration possess the same characteristics as the case of the displacement. Therefore, all the previously said, can be also attributed to the acceleration. The unique difference found between the acceleration and the displacement is in the initial part of the motion. In this part, a very high initial acceleration, positive and negative, takes place in a very short time. It is produced by the stabilization of the system when the force is initially applied to the spindel motor.

In the next step of this analysis, the damping has been introduced in the coupling. As in the previous case, the results have been compared with a rigid coupling, which is used as a reference. This is because the variations produced by the damping can not to be considered directly due to the high displacement values. The introduced damping values are low, with the objective to be able to do a better study of its effects. In the obtained results has been observed how the damping affects mainly when the stiffness values are very low. Therefore, the bigger is the stiffness value, the lower is the influence of the damping in the results. The damping influence is bigger when increase its value. It can be noticed because of different reasons, for example, when the stiffness is high, a high damping value is required to appreciate its influence in the system. Another reason is that the phase displacement produced between the two curves is higher with 3 Newton-mm-sec/deg than with 1 Newton-mm-sec/deg, as it can be seen in the Figure 6.6. And another observation can be the slight difference decrease produced between the rigid and the elastic couplings when the damping increases, which is a similar effect to the stiffness increase effect. For the looked for of a equation to relate the differences between the rigid and the elastic coupling, as it has been realized in the case of the stiffness influence, will be necessary a great effort and the obtaining of a high numbers of accurate results because, in this case, this relation will depend of the stiffness values, as well as the damping values and a constant. In the results can be seen how with the same stiffness value, the damping values increase produce a increase of constant value in the difference between the rigid and elastic coupling.



(a) (b)

Figure 6.6: displacement difference between undamping coupling (blue curve) and a damping coupling (green curve), (a) with 1 Newton-mm-sec/deg, (b) with 3 Newton-mm-sec/deg

If the difference between the two previous curves, (blue curve) and (green curve) is done, the result is a curve which has a great approach to the velocity difference between a rigid, used previously as guideline, and an elastic coupling. Furthermore, this curve increases its value when the damping value is more elevated. This relation between the damping and the velocity is given by the damping force, which relates the velocity with the damping constant. This damping force has a direct influence on the vibrations of an elastic system with damping, as is corroborated by the equation (2.1) and shown in the Figure 6.7. Another important effect produced by the damping is the elimination of a very high difference in the initial displacement. Thus, the initial displacement in a damping system is more harmonic than in an undamping system, where it is sharper.

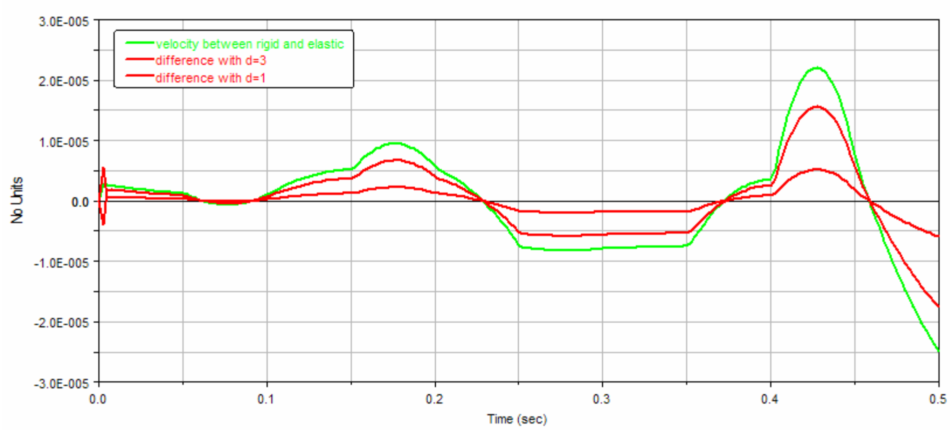


Figure 6.7: differences increase between displacement with different damping values and its relation with the speed.

The same effects produced by the damping in the displacement are produced also in the acceleration. When the damping is introduced, the most important aspect related with the acceleration magnitude is the elimination of an initial acceleration that occurs during the first instants of the Table movement. It can be also mentioned how the damping changes the acceleration, removes the sharp values and, therefore, makes it more harmonic. The difference between the two curves with different damping values provides a curve similar to the difference between the rigid, used as guideline, and the elastic coupling accelerations, but changes its sign.

6.3.1.2 Frequency Domain

In first place, the frequency response in the elastic coupling is analyzed with respect to the stiffness value variation, to know the influence of the stiffness on the behavior of the system in the frequency domain. To achieve that, the stiffness value is increased gradually, like in the time domain, multiplying it by 10 in each simulation. In this first part, the studied model is an undamping model, thus the damping value is equal to 0.

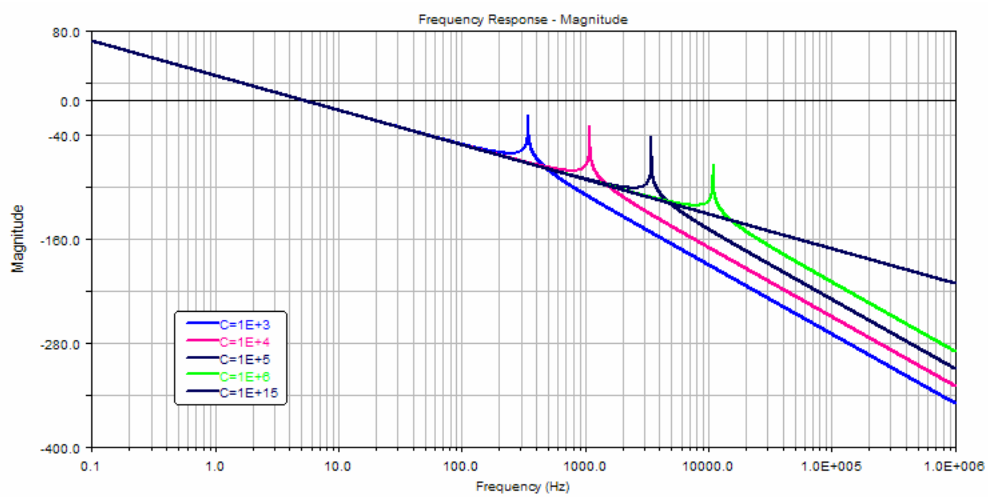


Figure 6.8: frequency response of the X-axis displacement of the Table.

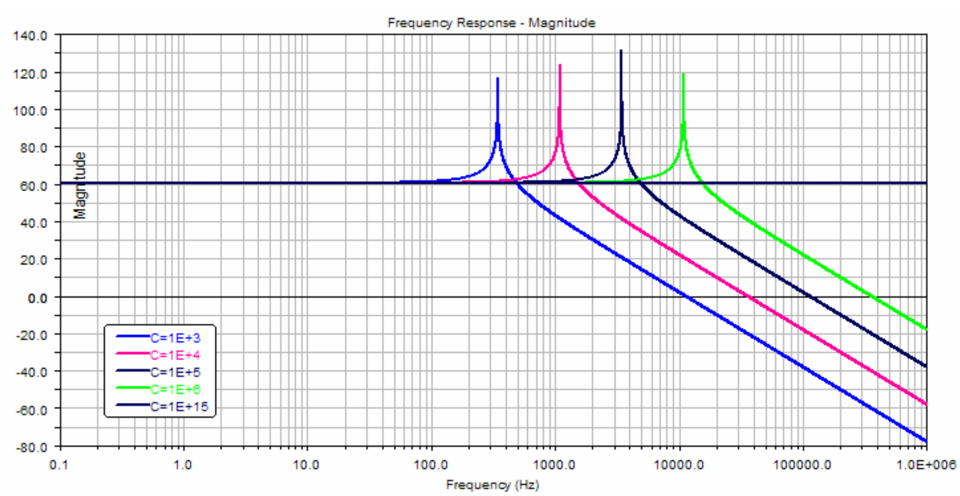


Figure 6.9: frequency response of the X-axis acceleration of the Table.

The results shown in the Figures 6.8 and 6.9 are the expected ones. In them, the phenomenon known as resonance is obvious. These results prove that when the stiffness varies, the resonance is produced in different frequencies. The system performance is that the higher is the stiffness value, the higher is the frequency where the resonance is produced. That is because the resonance happens when the frequency response has the same value that the natural frequency of the system. Furthermore, the natural frequency of a system increases its value when the stiffness value increases or when the mass in movement diminishes. Therefore, having this system a constant mass, it can be thought that the resonance variation is caused only by stiffness changes.

In the Figure 6.8 about the displacement, it can be seen that the magnitude value is bigger when the frequency is smaller. In addition, in these low frequency values, the coupling stiffness does not influence on the results. It can also be seen how, after the resonance, the displacement falls toward negative values. Therefore, for high frequencies, the most negative values are the values that have a minor stiffness because the resonance is produced for lower frequencies.

In the analysis of the acceleration is shown that the highest positive values are produced by the resonance. Just like in the analysis of the displacement, it can be observed how this phenomenon takes place for elevated frequency values, when the stiffness is more elevated. In all the carried out simulations, before the resonance takes place, the acceleration has a constant value. After the resonance, this acceleration value changes, decreasing toward negative values, as it happened in the displacement.

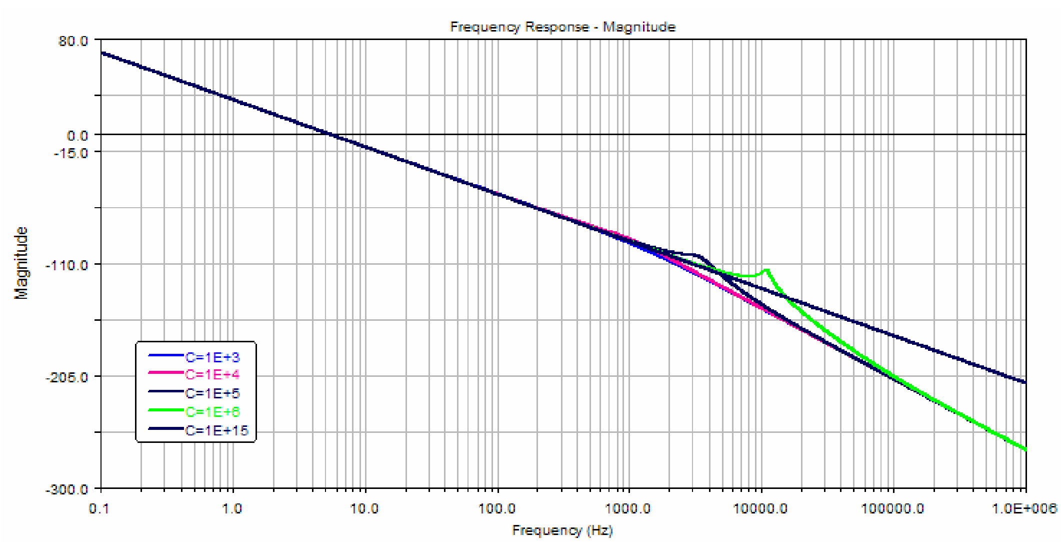


Figure 6.10: frequency response of the displacement of the Table in X-axis with a damping value of 2 Newton-mm-deg/sec.

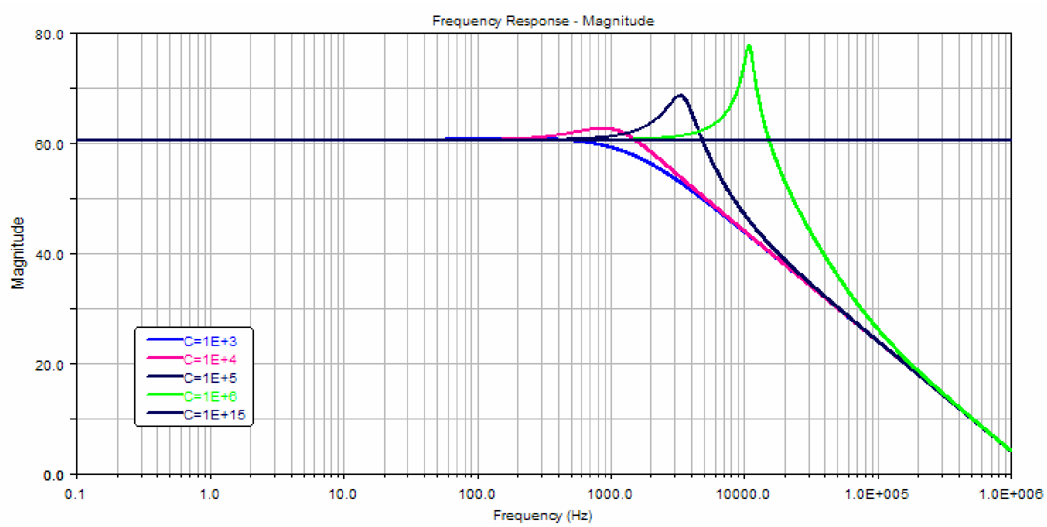


Figure 6.11: frequency response of the Table acceleration in X-axis with a damping value of 2 Newton-mm-deg/sec.

In the Figures 6.10 and 6.11 the damping has been introduced in the coupling and as it can be seen, its influence is very important. The results obtained show how the introduction of the damping produces a reduction of the resonance phenomenon. The damping influence is higher when its value is higher.

The damping influence disappears when the stiffness increases. This disappearance does not depend only on the stiffness increase, it depends also on the introduced damping amount. Therefore, for higher damping values higher stiffness values are needed so that the damping influence disappears totally of the system.

The slope produced after the resonance is less negatively steep when the damping influence is greater, as it can be perceived for low stiffness values. This is due to the decrease of the resonance phenomenon. The displacement magnitude has the same slope for all the frequencies when the resonance totally disappears, as it happens for a rigid model. The acceleration magnitude possesses the same value in all the frequency range when the resonance phenomenon is eliminated.

6.3.2 Parameter Variation Influence on the Simplified Model

6.3.2.1 Friction Influence

The introduction of the friction is realized individually for each element. The objective of it, is to have a better vision of the friction influence on each of the parts, without coupling between them. It is done both in time domain and in frequency domain. In the case of the frequency domain, the friction influence is produced in all the axes of the system and both displacement and acceleration. An example of it is shown in the Figure 6.12 with the friction on FL(fixed bearing) and LL(flotaing bearing), the friction on ball scre and, the friction on linear guides. In the friction case, the attenuation of the magnitude is produced in the first positive resonance of the results. The influence of the bearings (FL and LL) friction is very low with respect to the first resonance on the results, however, it does not happen for the linear guides and ball screw. In the case of the linear guides friction, it produces a very important decrease of the magnitude in this point and in the case of the ball screw friction, this reduction is complete, deleting totally this resonance.

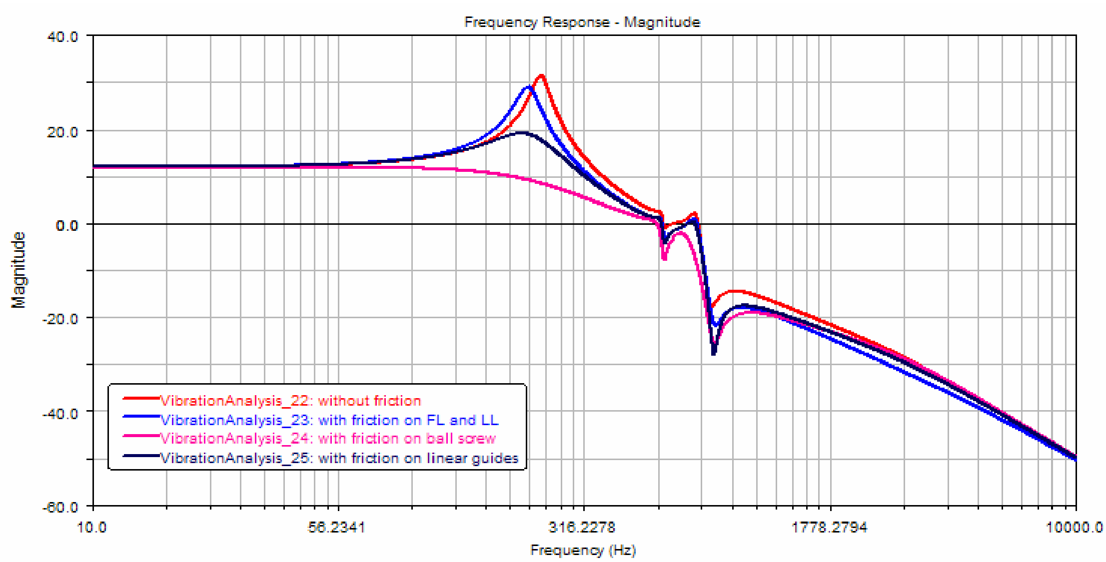


Figure 6.12: Friction influence on the X-axis displacement of the Table in the frequency domain

In the case of the time domain, the friction influence produces only a significant change in the X-axis displacement of the Table, because the direction of the friction forces is oriented only in this direction of the Table. This change, as it can be seen in the Figure 6.13, is produced in different intensity depending on each element. The biggest friction influence in the displacement is in the ball screw and next, with a little smaller friction influence comes the friction of the four linear guides. The bearing friction does not almost produce changes when it is introduced in the system. These changes in the displacement are created by the friction forces with opposite direction to the torque generated in the motor and, therefore, with an increase of this torque can be solved the changes. Another small influence found in the results of the other axes is the decrease of the initial displacement and acceleration produced by the stabilization of the system, which is almost completely eliminated when the friction influence is high.

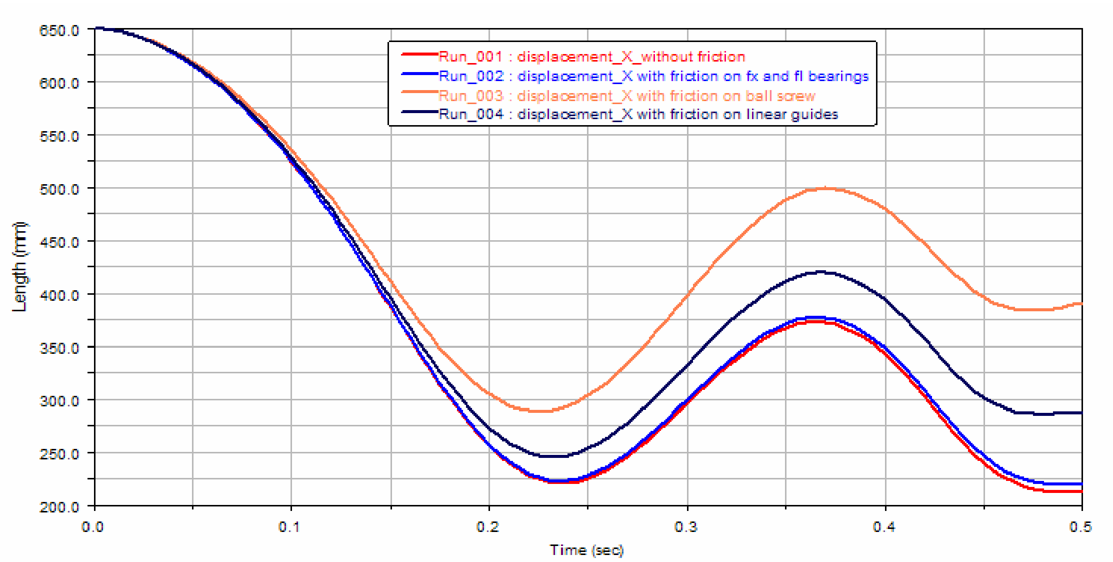


Figure 6.13: Friction influence on the X-axis displacement of the Table in the time domain

6.3.2.2 Bearings Influence

For the stiffness analysis in the frequency domain, the used value range varies between -50% and +200% of the standard calculated value and, in the damping case, the utilized D values are between 0.01 and 1. These ranges have been used in the axial and radial magnitude values of the bearings and in both fixed and floating bearings. This variation of the values shows that the value change in the bearings does not impact on the model. In the bearings, the variation of the stiffness values does not introduce any change in the results in the frequency response. In the case of the time domain, the values have been varied the same way that in the frequency domain. The obtained results on this analysis show that any change is produced in the system, as it happened in the frequency domain. An example of this, in the case of the time domain, can be seen in the Figure 6.14.

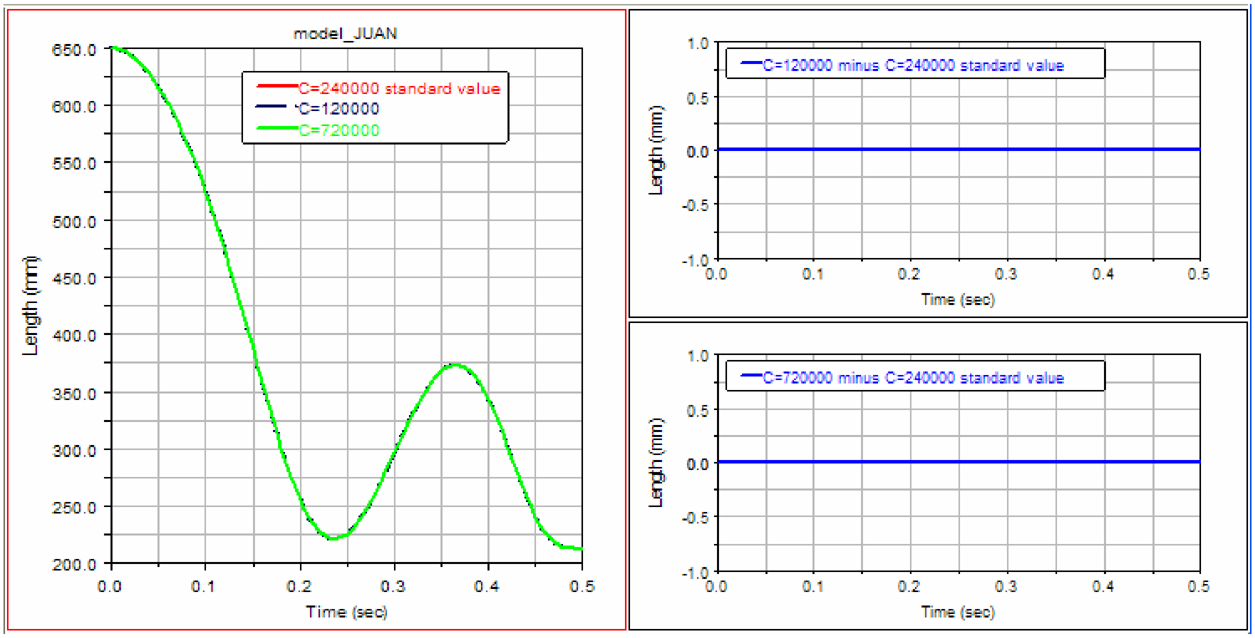


Figure 6.14: axial stiffness influence of the bearings on the X-axis of the Table

6.3.2.3 Coupling Influence

In the study of the coupling influence it is realized, in first place, the frequency domain analysis. The stiffness values have been varied from the standard value, c=2.06 Nmm/º, between -50% and +200%. In the Figure 6.15, in which are showed the results obtained about the displacement in X-axis, can be seen an example of what happened in the three axes of the coordinate system and in both displacement and acceleration. In the results is produced a displacement of the first resonance, which happens in lower frequency values when the stiffness value is smaller than the standard value. The same happens when the stiffness value is higher but the other way around.

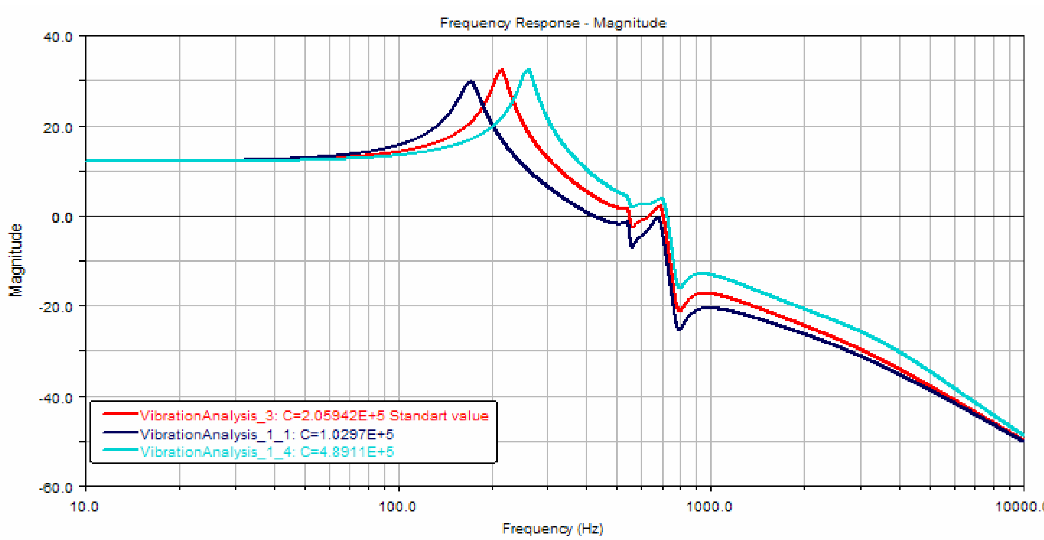


Figure 6.15: coupling stiffness influence on the X-axis of the Table

The damping values have been varied between D=0.05 and D=1, being 0.1 the standard value. In the Figure 6.16, about the displacement no the X-axis of the Table, can be observed the reduction of the first resonance that is influenced by the damping value, which reduces the maximum value of the resonance when the damping value increase. The same happens with the first resonance in the other axes of the system.

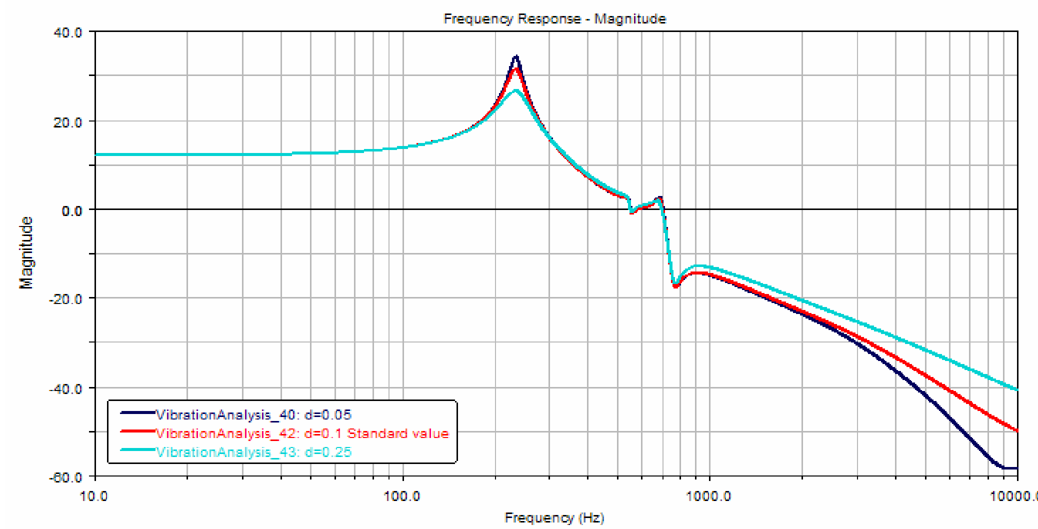


Figure 6.16: coupling damping influence on the X-axis of the Table

The time domain analysis is realized with the same values that in the frequency domain analysis. Due to the large displacement and the high acceleration produced in the motion of the Table, the results obtained in the X-axis are shown as the difference between the results obtained in the Design of Experiments (DOE) and the calculated standard values of the stiffness and damping, which are used as guideline, the same way that in the section 6.1.1 but in this case with the standard values. The coupling results in the elastic model possess the same structure that the obtained results in the rigid model. Like in the rigid model, the results of the displacement of the X-axis are directly related with the acceleration of the Table, having bigger differences between them when the stiffness value is bigger, as it is shown in the Figure 6.17. Therefore, the previously developed equation (6.1) can also be used to relate the differences between the results with the stiffness value, although adjusting it to the standard values. It can also be observed in the illustration, that the curves have a different sign between them. This is due to the fact that the DOE values are some higher and other lower than the standard value and, therefore, to realize the difference with the stardard value some are positives and other are negatives .

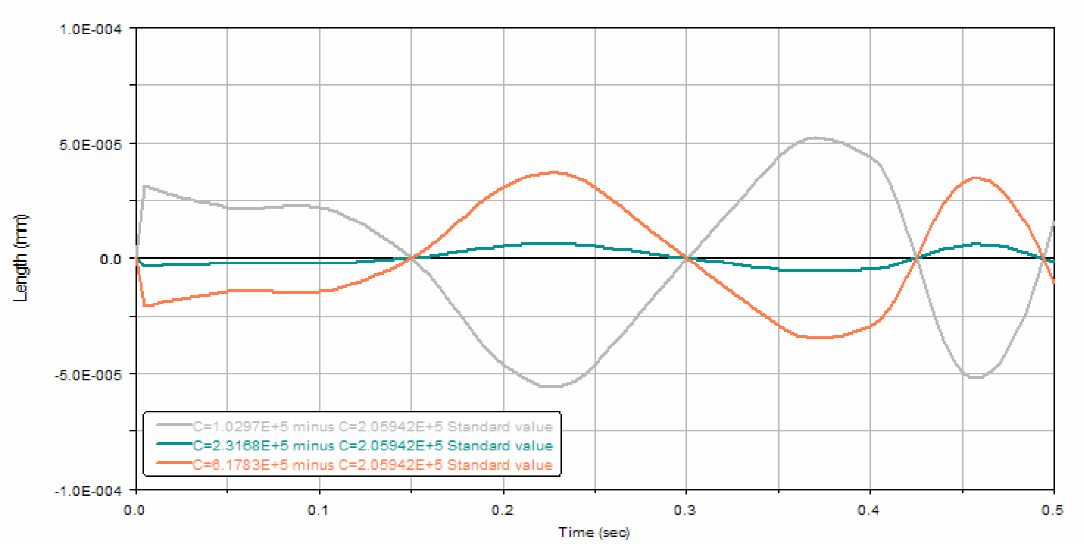


Figure 6.17: displacement difference between different stiffness values of the DOE and the standard value in the coupling

In the case of the damping influence, the results are also similar to the obtained results on the rigid model. If the Figure 6.18, there is the difference between the results of the analysis and the results of the standard value. Comparing it with the Figure 6.7, can be seen the relation between the obtained results and the velocity of the Table. As in the stiffness, there are bigger magnitudes in the results when the range of the damping increases its values and the curves have a different sign when the values are bigger or lower than the standard value, it also happens in the case of ball screw, as later can be seen. In both cases, stiffness and damping, is proved the link between these results and the equation about the dynamics behavior (2.6). With respect to the acceleration of the X-axis of the Table the results are similar to the results obtained with a rigid model, for both stiffness and damping. It can be thought that the change of the value in the coupling has not impact on the others system coordinates, Y- and Z- axes, because there is any variation between the results.

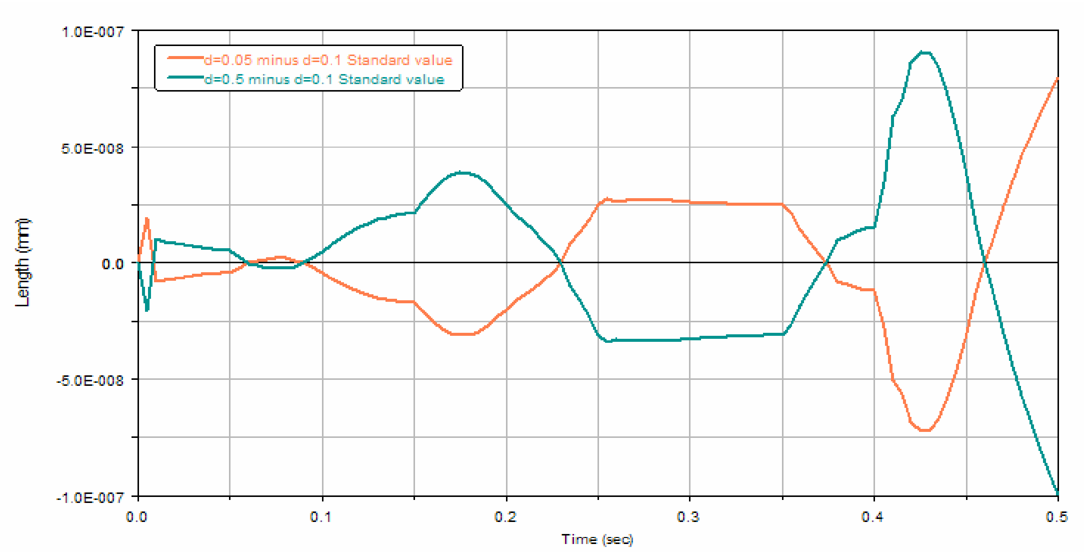


Figure 6.18: displacement difference between different stiffness values of the DOE and the standard value in the coupling

6.3.2.4 Ball Screw Influence

The Design of Experiments has been realized, as in the previous cases, with a value range of -50% and +200% of the standard value. The obtained results in the frequency domain follow a structure similar than the coupling results. The Figure 6.19, in which can be seen the results obtained about the displacement on the X-axis of the Table, shows that the variations are mainly produced in the first resonance, apart from the small changes produced in the different results after of this resonance. In the other axes, the behavior is also the same.

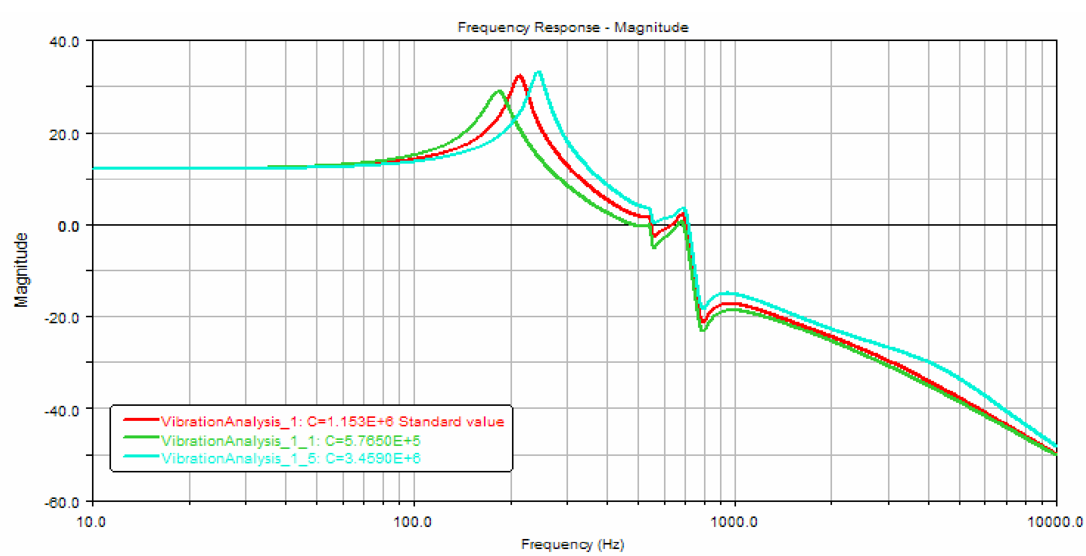


Figure 6.19: ball screw stiffness influence on the X-axis of the Table

The damping value varies between D=0.05 and D=1. The results are also the same that in the coupling, Figure 6.20. Although, in this case the high damping value prevents that other resonances can appear, as it has been checked in others results with damping equal to 0 .

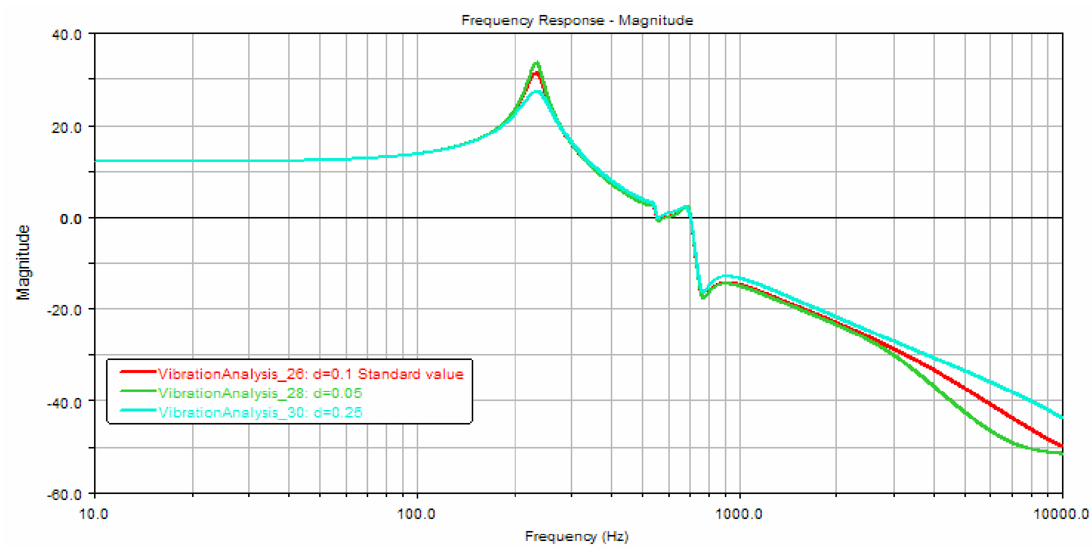


Figure 6.20: ball screw damping influence on the X-axis of the Table

In the case of the time domain, the value variations are the same that previously and in the results about the stiffness and damping influence in the displacement on the X-axis of the Table, shown in the Figure 6.21 and 6.22, can be seen that, also in this case, happens the same that in the coupling. The only difference between them is the magnitude of the results, being the differences between the values in the ball screw higher, the reason why will be commented in the discussion of the results, but the form is the same.

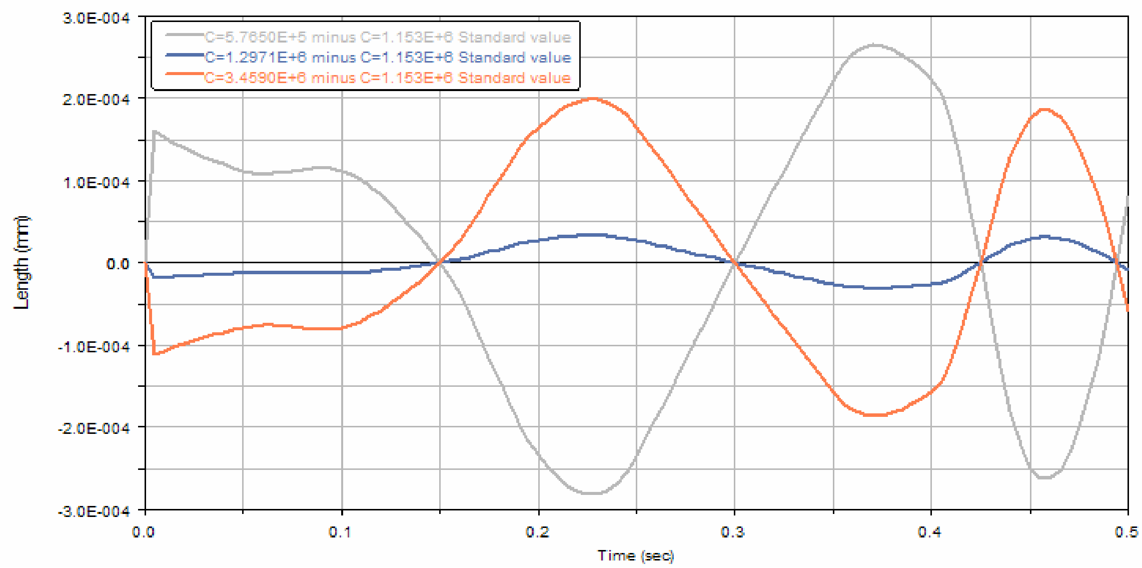


Figure 6.21: displacement difference between different stiffness values of the DOE and the standard value in the ball screw

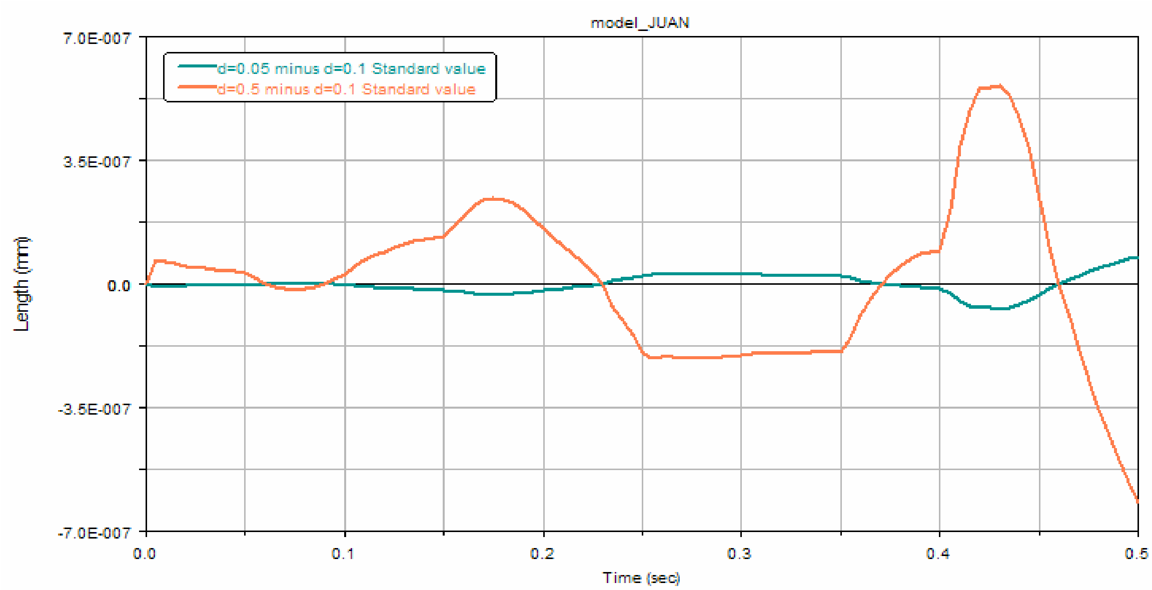


Figure 6.22: displacement difference between different damping values of the DOE and the standard value in the ball screw

6.3.2.5 Linear Guides Influence

In the frequency domain analysis of the linear guides influence, the DOE has been realized varying the translational stiffness of the Y- and Z-axes together. Although, in a previous analysis where the two axes have been analyzed individually is deduced that the biggest influence is produced by the Y-axis. The mean value introduced in the two axes is 8E+5 N/mm and the variations are realized between -50% and +200% of this value. As it can be seen in the Figure 6.23 about the displacement on the X-axis, the other resonances of the system are influenced by the change of the stiffness value in the linear guide, mainly by the Y-axis stiffness. These resonances are in higher frequencies when the stiffness value is bigger and the resonances are in lower frequency when the stiffness value is smaller. In the case of rotational stiffness, the standard value is 1.5E+5 N·m·m/º and the variation in the DOE have been realized for the same range that in the translational stiffness. In the obtained results of the rotational stiffness analysis for this value range, there are not changes in the results. Moreover, in other stiffness analysis has been found that the rotational stiffness influence is only important when its value is very high. Then the variations are produced in the same resonance than in the translational stiffness. These changes in the obtained results in both translational and rotational stiffness are produced in the three axes of the coordinate system and the same way for both the displacement and the acceleration.

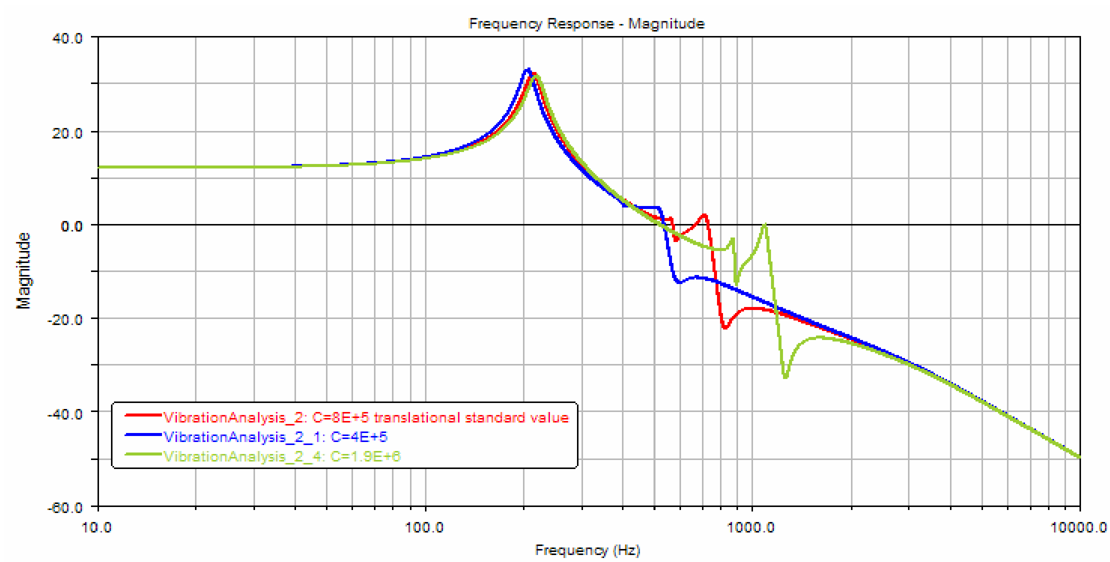


Figure 6.23: linear guide translational stiffness influence on the X-axis of the Table

The damping influence analysis shows that the variations are produced in the same resonance that in the stiffness. In the Figures 6.24 and 6.25 are shown these variations related to the D value. In the case of translational damping, the influence in this resonance is very high, however, in the case of the rotational damping the values have very different to be able to observe these influences. In both cases when the damping value is very elevated the resonance is almost eliminated, above all in the rotational damping, where the values must bigger of 5000 N·m·m·s/º.

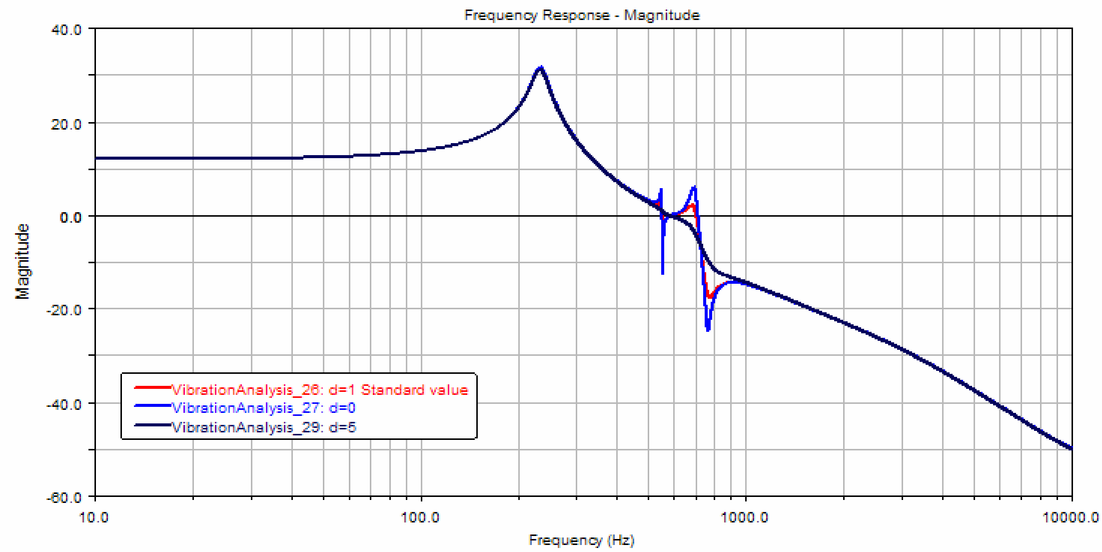


Figure 6.24: linear guide translational damping influence on the X-axis of the Table

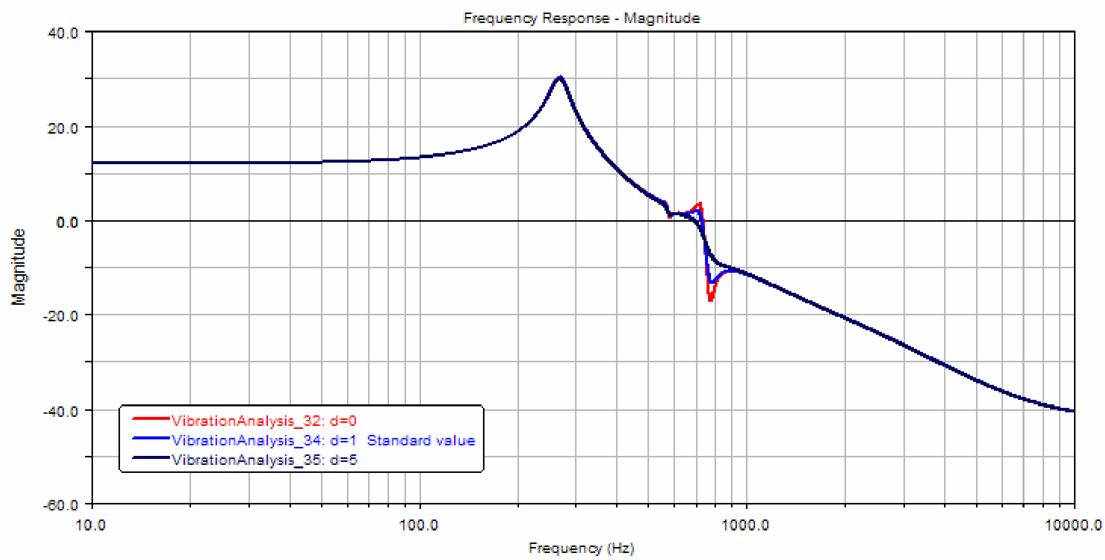


Figure 6.25: linear guide rotational damping influence on the X-axis of the Table

The time domain analysis has been realized like in the frequency domain, varying the translational and rotational values separately and for the same value range. The influence of each magnitude is different in each axis. In first place, as in the case of the frequency domain, the influence of the translational stiffness is produced mainly by the stiffness of the Y-axis of the linear guides. This influence is produced in all the axes of the coordinate system. In the X-axis, the results are similar that the results obtained in the coupling and the ball screw for that axis. However, in the Y and Z axes, the influence is most important in both the displacement and the acceleration. An example of these variations can be seen in the Figure 6.26 about the displacement on the Z-axis of the Table, where the displacement in this axis is bigger when the stiffness value is smaller. The same effect is produced also in the Y-axis displacement.

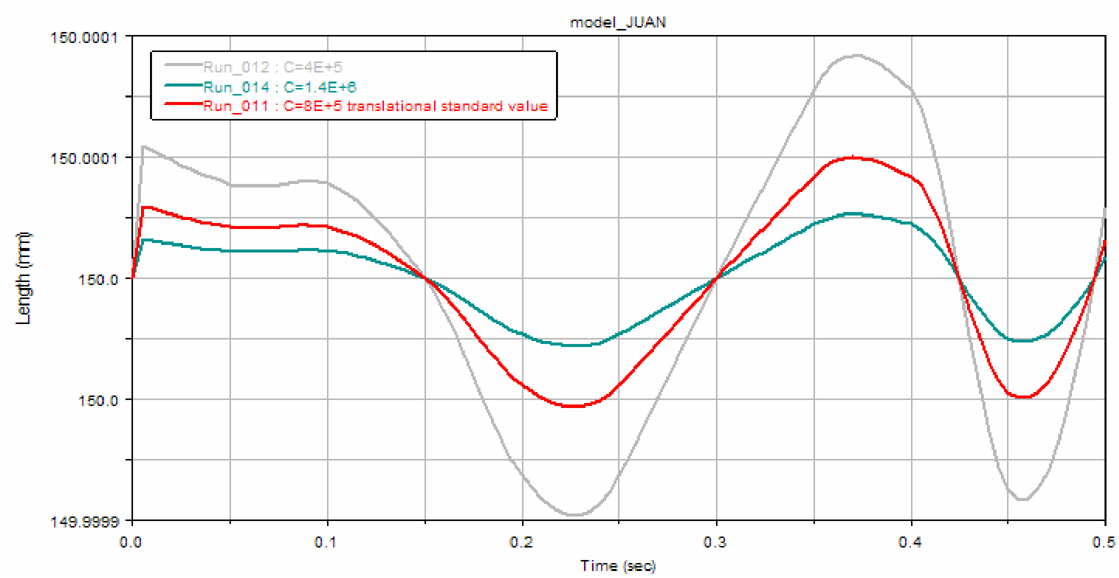


Figure 6.26: translational stiffness influence of the linear guides in the displacement Z-axis of the Table

With respect to the results of the acceleration in the three axes of the coordinate system, it is also influenced the machine behavior. In the X-axis acceleration, the results have the same structure that in the coupling and ball screw. In the case of the Y- and Z-axis acceleration, the results show that the higher is the stiffness value, the smaller are the vibrations in both axes. Another influence produced by the translational stiffness is that the elevated values produced by the stabilization of the system at the beginning of the motion decrease when the stiffness increases. In the time domain, the damping values do not influence in the results of the displacement and the acceleration of the Y- and Z- axis. The influence of the stiffness and damping rotational do not produce important changes in the results of the time domain analysis.

6.3.3 Discussion of the Results

In the first part is realized the analysis of the coupling elasticity in the rigid model. Although, those are the unique shown results, the elasticity of all the others elements has been analyzed in the rigid model. This study type is a good method to know the behavior of each element individually and to obtain better results. With this type of analysis, moreover, is possible to easily find some problems that can happen in the modeling of the machine tool and it is a basis that can avoid errors in the results when the elastic model is analyzed. In the analysis of the coupling in the rigid model, with regard to the time domain, can be emphasized the great relation between the stiffness and the displacement difference in the X-axis of the Table. This relation is so great and so strict that even an equation can be created to define its behavior. Furthermore, this equation can also be applied to the acceleration in the X-axis. Another important point of this analysis type is that the results of each element can be easily related with the theoretical background of the dynamic behavior, for instance, the relation between the acceleration, the stiffness and the results about the displacement. The damping analysis in the time domain shows also that exists a great relation between it and the velocity of the Table in the X-axis, which results are directly related. In the obtained results in the frequency domain analysis there is also the influence of the stiffness and the damping on the three regions in which can be divided the compliance results. With it, is totally proved that the frequency in which is produced the resonance varies according to the stiffness value and that this resonance is bigger or smaller according to the damping value.

In the second part the results are obtained with an elastic model. In the first point are shown the effects of the friction in the model. As it can be seen in the results, the definition of the friction is not easy. This can be deduced from the great differences between the bearings effect and the linear guide and spindle nut effect. The influence of the bearings friction is very low if it is compared to the influence of the linear guide friction and, above all, if it is compared to the influence of the ball screw friction. Therefore, it can be thought that the definition of the friction is not very accurate for maybe because some important parts of the friction are not defined or because there are problems with the model in ADAMS. Although with this definition an approach to the friction effects can be done.

Observing the results in the frequency domain, about the stiffness effects, is shown that the coupling and the ball screw influence on the same resonance, although these two elements have not the same influence on the results. For a range of -50% to +200% the coupling influence is higher than the ball screw influence. The other resonances found in the results are influenced by the linear guide stiffness that displaces the position of these resonances to different points in the frequency range. And for the same values of the translational stiffness, the effect produced by the Z-axis stiffness is almost insignificant compared to the effect produced by the Y-axis stiffness, which causes most of the variations. Something similar to the Z-axis stiffness happens with the rotational stiffness, which influence on the systems is very low, although in this case the effects can be noticed in the model when their values are very elevated, i.e., bigger than the introduced value range.

With respect to the stiffness influence in the time domain, the coupling and the ball screw have also the same type of influence. The influence of these two elements is exerted on the results of the X-axis displacement, which effect is bigger when the difference between the stiffness values is bigger. In the time domain, the influence of both is similar, although in the shown results, the magnitude of the difference between the obtained displacements in the analysis and the standard value is higher in the ball screw. This is due to the difference between the stiffness values and the standard stiffness value is higher also in the ball screw. The greatest influence on the results is produced by the linear guides because its effect is exerted to the three axes of the coordinate system. These influences are, as in the case of the frequency domain, mainly because of the translational stiffness of the Y-axis. The translational stiffness of the Z-axis has only some effect on the results of the acceleration and the displacement on the Z-axis of the coordinate system. The smallest effect on the system is the one produced by the rotational stiffness, which is not visible in the obtained results of the Y- and Z-axis. In all the obtained results, with respect to the stiffness in the time domain, is very obvious the relation between the stiffness and the displacement and the acceleration of the Table in the X-axis.

This analysis of the damping gives an approach to its behavior in the system. Although, due to the complexity of the damping analysis, a bigger calculus support would be necessary to get results nearer to the real behavior. Taking all this into account, in the obtained results in the frequency domain can be observed that the attenuation of the first resonance is produced by the coupling and the ball screw damping. And, as in the case of the stiffness, the influence of the coupling is higher than the influence of the ball screw. In the other resonances that appear in the results, the attenuation is exerted by the translational and rotational damping of the linear guides. In the case of the translational damping, the Z-axis damping is not important. However, in the case of the rotational damping, their effect is important, although it is smaller than the produced by the translational damping.

In the time domain are obtained positive results taking into account the previously said about the accuracy of the damping analysis. In this results can be observed the relation between the damping and the velocity of the Table in the X-axis of the motion. Only in this axis are found damping effect in the results, which are bigger in the case of the coupling and the ball screw than the linear guides. In the other axes of the coordinate system is not found almost any damping influence to the analyzed elements.

An important conclusion that can be drawn from all the obtained results is that in the case of the frequency domain analysis, the influence of each element to the displacement and the acceleration magnitude is the same in all the axes of the system and the direction in which is varied the stiffness or the damping in this element do not matter. However, in the case of the time domain, in the analysis of the variation of the stiffness or the damping in each element, the obtained results are not the same in the three axes of the coordinate. For instance, if the rotational stiffness of the X-axis is changed in the coupling, the effect of this change is only produced in the X-axis of the results and the results of the other axes do not change.

According to the obtained results in this thesis and other thesis like [Klotzbuecher-2007], the bearings do not influence in the system behavior. This lead to think that can appear possible problems to obtain good results in this type of elements with this type of modeling in the multibody simulations.

Below can be seen some Tables where is described the influence of each element in the results of both frequency domain and time domain. In the case of frequency domain, the resonance has been divided in two groups which are shown in the Figure 6.27 and the results are given in percentage. In the Table of the time domain the results are classified between the displacement and the acceleration in each axis. In this case the percentage can not be calculated due to the simulations are of variant time and its percentage is not uniform in all the points of the simulation, therefore the classification will be carried out by degree of influence.

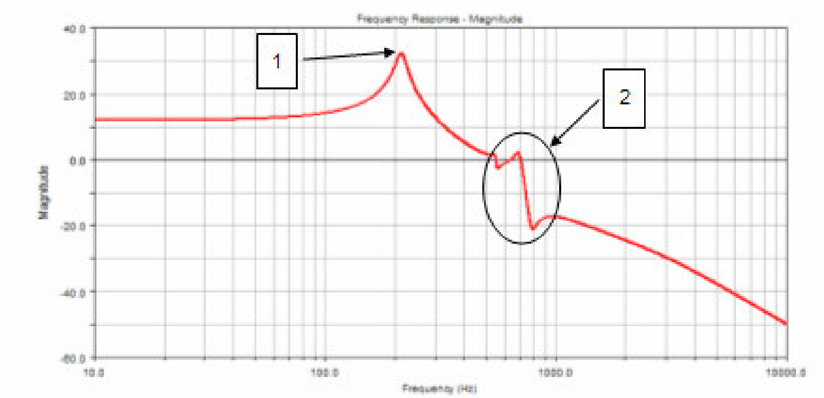


Figure 6.27: resonance found in the frequency domain results

Table 6.1: summary of the results of the elastic model in frequency domain

ELEMENTS	VALUES		INFLUENCE IN ALL AXES (%)	
	STANDARD	**VARIATION (-50% +200%)**	**RESONANCE 1**	**RESONANCE 2**
STIFFNESS				
COUPLING_x_rot	2.06E+5 N·mm/º	1.03E+5	21%	2%
		6.18E+5	26%	0%
BALL_SCREW_x	1.15E+6 N/mm	5.76E+5	14%	0%
		3.46E+6	14%	0%

L.G._y	7.40E+5 N/mm	4.00E+5 2.40E+5	3% 0%	30% 72%
L.G_z	8.57E+5 N/mm	4.00E+5 2.40E+5		
L.G._x_y_z_rot	1.50E+5 N·mm/º	7.60E+4 4.56E+5	0% 0%	0% 0%
DAMPING				
COUPLING_x_rot	147.5× D (0.1)	D =0.05 D =0.5	28% 9%	0% 1%
BALL_SCREW_x	1081× D (0.1)	D =0.05 D =0.5	23% 7%	0% 0%
L.G._y	6.5× D (1)	D =0.1 D =10	0% 0%	15% 51%
L.G_z	4.4× D (1)	D =0.1 D =10		
L.G._x_y_z_rot	587.34× D (1) 248.70× D (1) 516.62× D (1)	D =0.1 D =10	0% 0%	12% 40%

Table 6.2: summary of the results of the elastic model in time domain

ELEMENTS	VALUES		INFLUENCE			
	STANDARD	VARIATION (-50% +200%)	DISPLACEMENT		ACCELERATION	
STIFFNESS						
COUPLING_x_rot	2.06E+5 N·mm/º	1.03E+5 6.18E+5	**X**	high	**X**	high
			Y	null	**Y**	null
			Z	null	**Z**	null
BALL_SCREW_x	1.15E+6 N/mm	5.76E+5 3.46E+6	**X**	high	**X**	high
			Y	null	**Y**	null
			Z	null	**Z**	null
L.G._y	7.40E+5 N/mm	4.00E+5 2.40E+5	**X**	average	**X**	average
			Y	high	**Y**	high
			Z	average	**Z**	high
L.G_z	8.57E+5 N/mm	4.00E+5 2.40E+5	**X**	null	**X**	null
			Y	null	**Y**	null
			Z	low	**Z**	low
L.G._x_y_z_rot	1.50E+5 N·mm/º	7.60E+4 4.56E+5	**X**	average	**X**	average
			Y	null	**Y**	low
			Z	null	**Z**	very low
DAMPING						
COUPLING_x_rot	147.5× D (0.1)	D =0.01 D =1	**X**	high	**X**	high
			Y	null	**Y**	null

			Z	null	**Z**	null
BALL_SCREW_x	1081× D (0.1)	D =0.01 D =1	**X**	high	**X**	high
			Y	null	**Y**	null
			Z	null	**Z**	null
L.G._y	6.5× D (1)	D =0.1 D =10	**X**	average	**X**	average
			Y	null	**Y**	low
			Z	null	**Z**	low
L.G_z	4.4× D (1)	D =0.1 D =10	**X**	null	**X**	null
			Y	null	**Y**	null
			Z	null	**Z**	low
L.G._x_y_z_rot	587.34× D (1) 248.70× D (1) 516.62× D (1)	D =0.1 D =10	**X**	average	**X**	average
			Y	null	**Y**	low
			Z	null	**Z**	low

7 Summary and Outlook

7.1 Summary

This work consisted in the study of the effects of the parameter variation on the dynamic behavior of a testbed. The analysis of the machine axis testbed was realized by means of the multibody simulation, designing a model that is a simplification of the original model used in other thesis. The objective of this simplified model is to get a model with a similar behavior to the original model using for it the same proceeding of construction and the same values.

The design was realized initially as a CAD-Model which was exported by means of SimDesigner to ADAMS as a rigid model. In ADAMS/View has been developed the elastic model from the rigid model previously imported. For it, in the interface of the elements belonging to the transmission of the motion are used flexible connectors. In the interface have been used help bodies to make the modeling of these elements easier, which in this case have been a fixed bearing, a floating bearing, a spindle screw nut, a coupling and four linear guides. On one side of these help bodies has been connected a bushing which gives elasticity to the link and on the other side have been connected the joints, which allow the desired motion. To create the friction have been utilized *general forces* that were introduced as a function by means of the *Function builder* to describe the friction behavior. In some cases as in the bearings were used *Splines* to construct the function. And another force was introduced in the model to introduce the desired motion to the mechanics.

To realize the parameter analysis was necessary, in the case of the frequency domain, the creation of input and output channels in some points of the machine, which contain the predefined magnitudes used for the study. In the case of the time domain, apart of the creation of the measure points in the central part of the Table surface, was necessary to introduce a *Spline* in the force utilized to give motion to the mechanics, which objective was to determine the desired motion.

Initially, the results were centered in the effects of the elasticity of each element in the rigid model, in both frequency domain and time domain, introducing fixed joints in the necessary elements. With this analysis was obtained a first approach to the dynamic behavior of the simplified model, being able to correct possible errors in the modeling. Subsequently, was proceeded to obtain the results of the effect of the parameters variation in the elastic model. The results in frequency domain were obtained to compare the behavior of this model with other thesis realized previously and, moreover, to realize a compared analysis of the results in both domains. In the first step, the analysis was centered in the effects of the introduction of the friction on the system, to later center the analysis individually in each element of the transmission. The analysis of each element is realized in frequency domain and in time domain, varying the stiffness and the damping a specific range. Finally is realized an analysis of all the obtained results and a summary of the results is carried out in two Tables.

7.2 Outlook

In future work, this model could be used as a base to realize a model with higher elasticity introducing, for instance, elasticity in the spindle and creating the shoes as in the original model. With it, an approach to the dynamic behavior of the original model could be realized.

In the case of the friction, due to the non-exact definition defining only some parts of the friction and due to the problems related with ADAMS the obtained results do not totally correspond to the real behavior. Another problem with the modeling can be found in the results of the bearings, which have not any variation. Therefore, an improvement of the modeling method could be carried out in future projects, in which could be achieved results closer to the real behavior.

Appendix

Appendix A. Correspondence between the simplified and the original model parts.

-GROUND	-Ground_part
-MACHINE BED	-Fuss_1 -Fuss_2 -Fuss_3 -Fuss_4 -Eingusskoerper_1 -Eingusskoerper_2 -Stahlleiste_1 -Stahlleiste_2 -Laengstraeger -Linearfuehrung_1 -Linearfuehrung_2 -Festlager -Loslager
-MOTOR	-Motor
-SPINDLE MOTOR	-Motor_welle -Kopplung_motorseite
-SPINDLE	-Kopplung_spindelseitig -Spindel_motorseitig -Spied_mutter

	-Spindel
-TABLE	-Tisch -Mutterflansch -Linearlager_1 -Linearlager_2 -Linearlager_3 -Linearlager_4

Table A.1: Correspondence between the simplified and the original model parts.

Appendix B. Name of the parts in Catia and in ADAMS

Name in Catia	Name in ADAMS
-GROUND	-ground
-MACHINE BED	-MASCHINENBETT
-MOTOR	-MOTOR
-SPINDLE MOTOR	-SPINDEL_MOTOR
-SPINDLE	-SPINDEL
-TABLE	-TISCH
-DUMMY_1	-DUMMY_1
-DUMMY_2	-DUMMY_2
-DUMMY_3	-DUMMY_3
-DUMMY_4	-DUMMY_4
-DUMMY_FIXED_BEARING	-DUMMY_FESTLAGER
-DUMMY_FLOTAING_BEARING	-DUMMY_LOSTLAGER
-DUMMY_BALL_SCREW	-DUMMY_MUTTER

Table A.2: Name of the parts in Catia and in ADAMS

Appendix C. Mass and mass inertia tensor properties

PARTS	MASS (kg)	MASS INERTIA TENSOR (kg·mm²)
-ground	-	-
-MASCHINENBETT	36.50	2.75 E+6 ; 2.26 E+6 ; 4.96 E+5
-MOTOR	3.9	6500 ; 4063 ; 4063
-SPINDEL_MOTOR	11	210 ; 210 ; 12
-SPINDEL	20	8.62 E+4 ; 8.62 E+4 ; 91.9
-TISCH	229.99	2.61 E+6 ; 1.82 E+6 ; 1 E+6
-DUMMY_1	3.26 E-5	1.3 ; 1.3 ; 1.3
-DUMMY_2	3.26 E-5	1.3 ; 1.3 ; 1.3
-DUMMY_3	3.26 E-5	1.3 ; 1.3 ; 1.3
-DUMMY_4	3.26 E-5	1.3 ; 1.3 ; 1.3
-DUMMY_FESTLAGER	1.5 E-5	44.35 ; 44.35 ; 24.89
-DUMMY_LOSTLAGER	1.5 E-5	44.35 ; 44.35 ; 24.89
-DUMMY_MUTTER	1.5 E-5	44.35 ; 44.35 ; 24.89

Table A.3: Mass and mass inertia tensor properties

Appendix D. Joints in the elastic model

<table>
<tr><th>PART NAME</th><th>JOINT TYPE</th><th>JOINT NAME</th></tr>
<tr><td>-ground</td><td rowspan="2">-Fixed Joint</td><td rowspan="2">-FIXED_MASCHINENBETT</td></tr>
<tr><td>-MASCHINENBETT</td></tr>
<tr><td>-ground</td><td rowspan="2">-Fixed Joint</td><td rowspan="2">-FIXED_MOTOR</td></tr>
<tr><td>-MOTOR</td></tr>
</table>

-SPINDEL -DUMMY_MUTTER	-Screw Joint	-SCREW_MUTTER
-MASCHINENBETT -DUMMY_1	-Translational Joint	-TRANSLATIONAL_DUMMY_1
-MASCHINENBETT -DUMMY_2	-Translational Joint	-TRANSLATIONAL_DUMMY_2
-MASCHINENBETT -DUMMY_3	-Translational Joint	-TRANSLATIONAL_DUMMY_3
-MASCHINENBETT -DUMMY_4	-Translational Joint	-TRANSLATIONAL_DUMMY_4
-TISCH -DUMMY_MUTTER	-Translational Joint	-TRANSLATIONAL_MUTTER
-MOTOR -SPINDEL_MOTOR	-Revolute Joint	-REVOLUTE_MOTOR
-SPINDEL -DUMMY_FESTLAGER	-Revolute Joint	-REVOLUTE_FESTLAGER
-SPINDEL -DUMMY_LOSTLAGER	-Revolute Joint	-REVOLUTE_LOSLAGER
-SPINDEL_MOTOR -SPINDEL	-Revolute Joint	-REVOLUTE_KOPPLUNG

Table A.4: Joints in the elastic model

Aditional joints in the rigid model

PART NAME	JOINT TYPE	JOINT NAME
-TISCH -DUMMY_1	-Fixed Joint	-FIXED_LINEARGUIDE_1
-TISCH -DUMMY_2	-Fixed Joint	-FIXED_LINEARGUIDE_2
-TISCH -DUMMY_3	-Fixed Joint	-FIXED_LINEARGUIDE_3
-TISCH -DUMMY_4	-Fixed Joint	-FIXED_LINEARGUIDE_4
-MASCHINENBETT -DUMMY_FESTLAGER	-Fixed Joint	-FIXED_FESTLAGER
-MASCHINENBETT -DUMMY_LOSTLAGER	-Fixed Joint	-FIXED_LOSLAGER
-TISCH -DUMMY_MUTTER	-Fixed Joint	-FIXED_MUTTER
-SPINDEL_MOTOR -SPINDEL	-Fixed Joint	-FIXED_KOPPLUNG

Table A.5: Aditional joints in the rigid model

Appendix E. Flexible Joints

PART NAME	FLEXIBLE JOINT TYPE	FLEXIBLE JOINT NAME
-TISCH -DUMMY_1	-Bushing	-BUSHING_LINEARGUIDE_1
-TISCH -DUMMY_2	-Bushing	-BUSHING_LINEARGUIDE_2
-TISCH -DUMMY_3	-Bushing	-BUSHING_LINEARGUIDE_3
-TISCH -DUMMY_4	-Bushing	-BUSHING_LINEARGUIDE_4
-MASCHINENBETT -DUMMY_FESTLAGER	-Bushing	-BUSHING_FESTLAGER
-MASCHINENBETT -DUMMY_LOSTLAGER	-Bushing	-BUSHING_LOSLAGER
-TISCH -DUMMY_MUTTER	-Bushing	-BUSHING_MUTTER
-SPINDEL_MOTOR -SPINDEL	-Bushing	-BUSHING_KOPPLUNG

Table A.6: Flexible Joints

Appendix F. Forces

PART NAME	FORCE TYPE	FORCE NAME
-MASCHINENBETT -DUMMY_1	-General Force	-FRICTION_LINEARGUIDE_1
-MASCHINENBETT -DUMMY_2	-General Force	-FRICTION_LINEARGUIDE_2
-MASCHINENBETT -DUMMY_3	-General Force	-FRICTION_LINEARGUIDE_3
-MASCHINENBETT -DUMMY_4	-General Force	-FRICTION_LINEARGUIDE_4
-TISCH -DUMMY_MUTTER	-General Force	-FRICTION_MUTTER
-SPINDEL -DUMMY_FESTLAGER	-General Force	-FRICTION_FESTLAGER
-SPINDEL -DUMMY_LOSTLAGER	-General Force	-FRICTION_LOSLAGER
-SPINDEL_MOTOR	-Single component Force	-FORCE_MOTOR

Table A.7: Forces.

Appendix G. Spline values of the Motor Torque in time domain

TIME (seg)	FORCE (Newton-mm)
0.0	-30000.0
5.0E-002	-20000.0
0.1	-20000.0
0.15	0.0
0.2	42000.0
0.25	42000.0
0.3	0.0
0.35	-40000.0
0.4	-40000.0
0.45	45000.0
0.5	-15000.0

Table A.8: Spline values of the Motor Torque in time domain

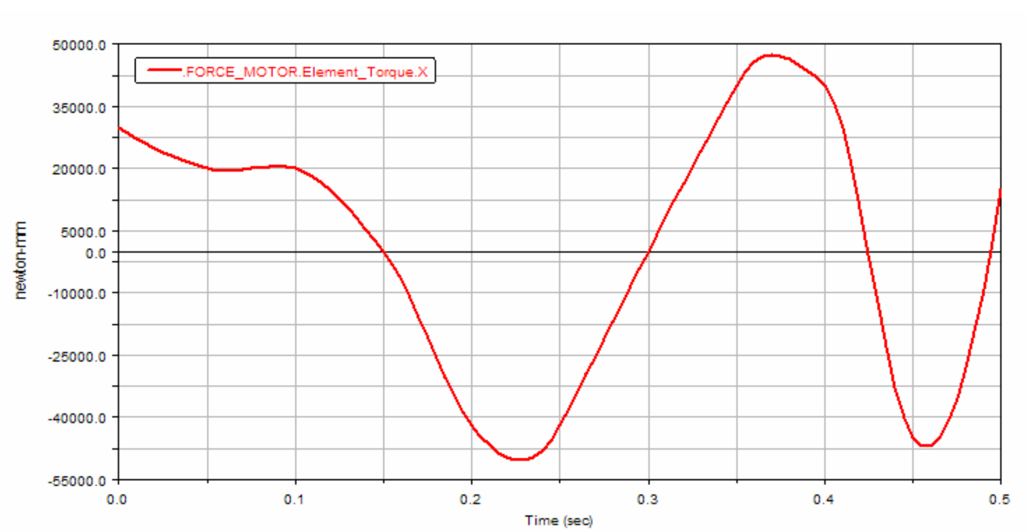


Figure A.1: Motor Torque in time domain

Appendix H. Modeling the Friction

The friction forces in bearings and guides are formed by the union of the following parts, according to Klotzbuecher [Kotzbuecher-2007]:

·The rolling friction between the surfaces of the rolling elements, whose value is the square of the relative velocity.

· Liquid or gas friction between the relative motion of the bodies, whose value increases with the relative velocity.

·Dry kinetic friction which is contrary to the movement trajectory.

·Static friction is recognized by the forces and moment break free, which only exist in stand state.

To realize the modeling of the friction is important, due to the different parts that compose the friction, to know the velocity range of the system when it is realizing its work and to find with it the dominant parts of the friction in this mechanism. To design a model the most exactly as possible is necessary a lot of work and, moreover, the nonlinear parts can cause instability in the simulation. Therefore, must be determinated which is the influence of the friction in the simulation and to choose with that which accuracy grad is required for the simulation.

When the model works at high speeds, the introduced friction can be of purely viscous type. And when the difference between the adhesion and dynamic friction is small and, therefore, low velocities, can be introduced a dynamic friction totally dry Klotzbuecher [Kotzbuecher-2007].

The friction can be directly modeled in ADAMS with idealized elements. This option can be found in some idealized joints where the static and dynamic coefficient of the friction force can be introduced. This option has been rejected in this project and the friction has been introduced by means of another additional elements. This additional elements used in this thesis are the *General Forces*, with which can be designed forces and moments in the direction of the coordinate axes. To place idealized elements is necessary to create at least two *Markers*. The I-Marker is the part where the force flow is directed to, therefore, the part that receives this force flow and by defect it is placed in the action-part. And the J-Marker is the one situated in the reaction-body. With a General force can be defined the friction by means of a function and that way can be introduced the addition of the different parts friction and the nonlinear process can also be represented.

The method used to define the friction is the same that uses ADAMS [MSC-2005] to define the idealized connection. In ADAMS is described the friction coefficient with regard to the velocity by means of a characteristic curve smooth which is shown in the Figure A.2, to delete discontinuities when the relative velocities disappear. The problem in this modeling is that the adhesion can not be represented, because when disappears the relative velocity also disappears the friction force. This method has been realized positively in others projects in the WBK institute as, for example, in [Inderbirken-2006].

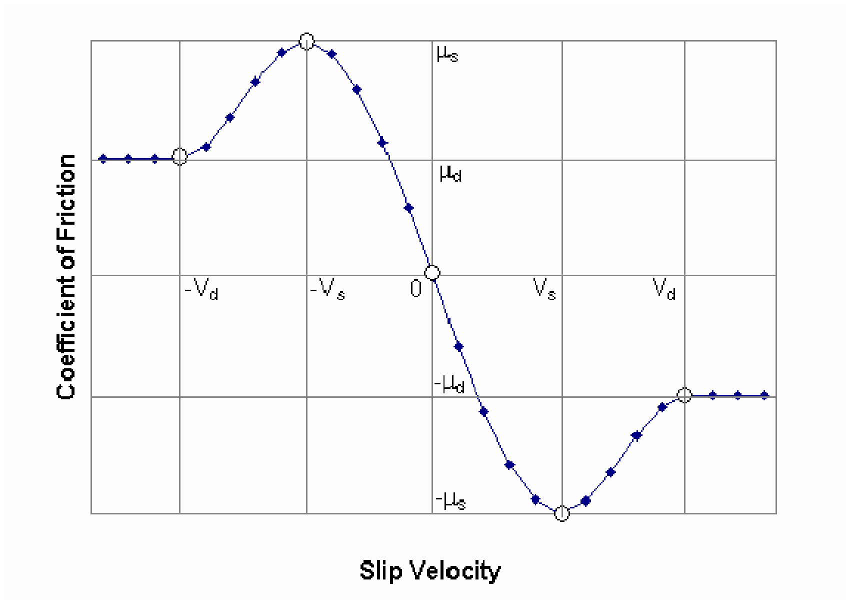


Figure A.2: Coefficient of friction varying with slip velocity [MSC-2005b]

To define the friction behavior is used the dialog box of the element denominated *Function Builder* on the following form:

$$\mu(v) = -sign(v) \cdot \mu_d \qquad \text{for } |v| > v_d$$

$$\mu(v) = -step(|v|, v_d, \mu_d, v_s, \mu_s) \cdot sign(v) \qquad \text{for } v_s \le |v| \le v_d \tag{A.1}$$

$$\mu(v) = step(v, -v_s, \mu_s, v_s, -\mu_s) \qquad \text{for } -v_s < |v| < v_s$$

When the velocity is in the point v_s, the friction is in the maximum point u_s which corresponds with the adhered friction. And when the velocity is v_d the friction is converted to constant with a value of u_d.

The step function is a function used to approximate the Heaviside step function with a cubic polynomial and it is defined in the function builder on the following manner [MSC-2005]:

$$STEP = \begin{cases} h_0 & : x \le x_0 \\ h_0 + a * \Delta^2 \ (3 - 2\Delta) & : x_0 < x < x_1 \\ h_1 & : x \ge x_1 \end{cases} \qquad a = h_1 - h_0 \qquad \Delta = (x - x_0)/(x_1 - x_0) \tag{A.2}$$

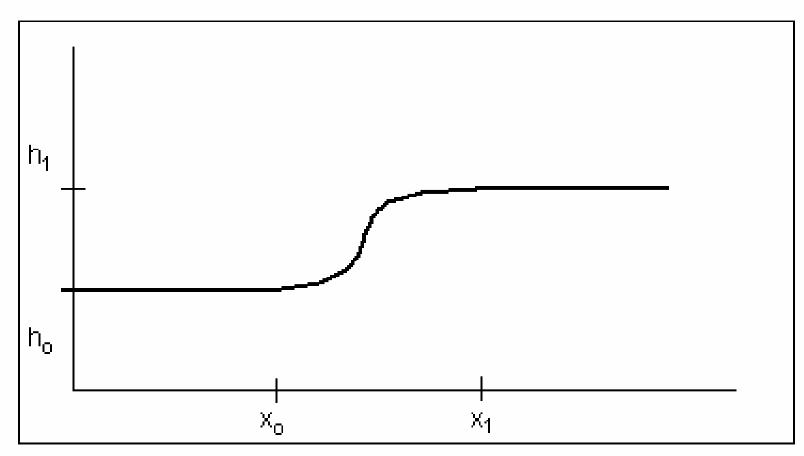


Figure A.3: Definition for the Step function [MSC-2005b]

Other method to model the friction moment, when it is dependent on the velocity, or rotation speed or it is nonlinear, is using *Splines*. With these elements can be realized a diagram in ADAMS, where can be represented the friction moment characteristics as an input parameter. For the utilization of this parameter is only necessary to realize *Build → Data Element → Spline → New*. Then, can be introduced the data table of preview plots and adjust it with different options.

To access the Spline data with the Function Builder to model the friction, is necessary the command AKISPL. The syntax used for this command is AKISPL(VARVAL(*id*),z,*id*), where the command VARVAI(*id*) is utilized to open the actual data of a *State Variable* in ADAMS. This is the first input parameter of the function, where ADAMS expects the X-value of the archived diagram and which returns the value of the corresponding function. In the second part of the syntax, the introduced value is 0 in this case, because it is a 2D plot [MSC-2006].

The advantage of this method is that it can be used with complicated friction calculations with multiple variables.

BIBLIOGRAPHY

- [Altintas-2000] Altintas, Yusuf: Manufacturing automation: metal cutting mechanics, machine tool vibrations and cnc desing. Cambrige University Press, 2000- IBSN 0-521-65029-1, WBK.

- [Arnold-2004] Arnold, Martin: Simulation Algorithms in Vehicle System Dynamics. Halle, 2004. www.mathematik.uni-halle.de/reports/sources/2004/04-27report.pdf

- [Baudisch-2003] Baudisch, Thomas: Simulationsumgebung zur Auslegung der Bewegungsdynamik des mechatronischen Systems Werkzeugmaschine. München: Herbert Utz Verlag, 2003 – ISBN 3-8316-0249-2, WBK D 373.029

-[Broos-2006] Fleischer, J; Broos, A; Will, J:Parameter optimization in combination with finite element analysis and multi-body-simulation: one way towards better machine tool design. Universitaet Karlsruhe (TH), Karlsruhe, Germany, 2006.

- [Bürgel-2001] Bürgel, Robert: Prozessanalyse an spanenden Werkzeugmaschinen mit digital geregelten Antrieben. München: Herbert Utz Verlag, 2001 – ISBN 3-8316-0021-X, WBK D 373.001

- [CATIA-2007a] FH Heilbronn, CSI-Entwicklungstechnik GmbH: CATIA V5 Homepage http://www.websites-factory.de/catia/index.html

- [CATIA-2007b] Dassault Systemes: Produkthomepage CATIA V5 http://www.3ds.com/de/corporate/about-us/brands/catia/

- [Eubert-1992] Eubert, Peter.: Digitale Zustandsregelung elektrischer Vorschubantriebe. Berlin: Springer, 1992 – ISBN 3-540-55541-2, UBKA 92 A 2300

- [Großmann-1996] Großmann, K.: Einfach Umrüsten? – Bewertung von Vorschubantriebskonzepten an Werkzeugmaschinen mittels Simulation. In Konstruktionspraxis 7, 1996

- [Hennrich-2007] Hennrich, Heiko: Experimental and Numerical Analysis of the vibrations of a Testbed for Machine Tool Feed Axes. Studienarbeit am Institut für Produktionstechnik (wbk), Universität Karlsruhe, 08.02.2007

- [Indenbirken-2006] Indenbirken, Marcel: Untersuchung einer Mikrohandhabungseinrichtung mit gekoppelter Mehrkörper- und Regelungssimulation. Diplomarbeit am Institut für Produktionstechnik (wbk), Universität Karlsruhe, 03.02.2006

- [MSC-2002a] Refining Model Designs in ADAMS/View. Mechanical Dynamics Inc., Ann Arbor, Michigan 2002, Part number: 20VWRF-01.

- [MSC-2002b] MSC.SOFTWARE CORP.; MSC.ADAMS Release Guide; Santa Ana, USA,2005

- [MSC-2005a] MSC.Software Corp.: MSC.ADAMS Product Catalog; Santa Ana, USA, 2005

- [MSC-2005b] MSC.Software Corp.: MSC.ADAMS Online Help; Santa Ana, USA, 2005

- [MSC-2005b] MSC.Software Corp.: MSC.ADAMS Online Help (Getting Started using ADAMS/Vibration); Santa Ana, USA, 2005

- [Neithardt-2004] Neithardt, Wolfgang: Optimierung von Werkzeugmaschinen in der Entwurfs- und Konzeptphase auf Basis der Mehrkörpersimulation. Karlsruhe, Universität (TH), Diss., 2004 – ISSN 0815-1234

- [Shabana-1991a] Shabana, Ahmed: Theory of vibration. Volume I:An Introduction. Springer-Verlag, 1991- ISBN 0-387-97276-5, UBKA 91 A 1366-1

- [Shabana-1991b] Shabana, Ahmed: Theory of vibration. Volume II: Discrete and continuous systems. Springer-Verlag, 1991- ISBN 0-387-97384-2, UBKA 91 A 1366-2

- [Shabana-2005a] Shabana, Ahmed: Dynamics of multibody systems. 2. Auflage. Cambridge: Cambridge University Press, 2005 – ISBN 0-521-85011-8, UBKA 2005 E 1109(3)

- [Shabana-2005b] Shabana, Ahmed:Computational dynamics. 2. Edition. Cambridge: Cambridge University Press, 2005 – ISBN 0-471-37144-0, UBKA 2005 A 2561(2)

- [SimCAT-2006] Steinebrunner, E. (Hrsg.): Abschlussbericht des Verbundforschungsprojekts SimCAT. www.simcat.org, 2007

-[Stephenson-2006] David A. Stephenson ; John S. Agapiou: Metal cutting theory and practice, 2 edition. CRC Taylor & Francis, 2006-ISBN 0-8247-5888-9, UBKA 2006 E 26(2)

- [Weck-2001] Weck, Manfred: Werkzeugmaschinen, Band 3: Mechatronische Systeme, Vorschubantriebe, Prozessdiagnose. 5. Aufl. Düsseldorf: VDI-Verlag, 2001 – ISBN 3-540-67614-7, UBKA 2001A 1169-3(5)

- [Weck-2002] Weck, Manfred: Werkzeugmaschinen, Band 5: Messtechnische Untersuchung. 5. überarb. Aufl. Düsseldorf: VDI-Verlag, 1996 – ISBN 3-18-401441-X, UBKA 78A 1590 4(5)f

- [Veitl-2001] Veitl, Armin: Integrierter Entwurf innovativer Stromabnehmer. Fortschrittsberichte VDI, Reihe 12, Band 449 Düsseldorf: VDI-Verlag, 2001-ISBN 3-18-344912-9

-[Zäh-2007] Zäh, M; Reinhart, G: Mechatronik Vorsprung durch Simulation.ISBN 978-3-89675-088-4. Garching,2007