Efficient mechatronic evaluation of machine tool designs using model reduction

P. Maglie*, R. Carbini, S. Weikert, K. Wegener

Institute for Machine Tools and Manufacturing IWF
Swiss Federal Institute of Technology ETH
Zurich, Switzerland

Abstract

This paper addresses the use of a reduced FE-representation based on a model that was originally created and used in a standard FEM-Analysis environment for static and dynamic investigations. The applied reduction method takes into account the specific requirements for machine tools: the modelling of guideway systems and drive components. In order to obtain a complete model which a) includes the entire machine structure, b) offers information about the resulting behaviour between tool and work-piece and c) incorporates the properties of the control loop, the following steps are carried-out: After exporting the needed model information (system matrices and coupling characteristics) from a commercial FE-software, a widely automated reduction procedure is applied. The structural axis components are then transferred into a state-space representation where, firstly, the global structural behaviour is analysed and secondly, the control system is added. Based on this complete mechatronic model, a variety of investigations can be performed, covering time- and frequency-domain. Based on measurements on an existing test-bed, the calculation results are compared with reality and interpreted.

1. Introduction

With the increasing demand for high-productivity and high-accuracy machine tools, the evaluation of the mechatronic behaviour of these systems is required in order to verify the system's ability to meet the given requirements *[1]*. The use of a complete FE-model with a fine mesh, including all the design details, is not yet possible with acceptable efforts in time and calculation power. For the investigation of the system's behaviour in the time- and frequency-domain, the use of a reduced FE-representation of the machine tool structure is proposed. In this paper, the derivation of the reduced structural model for a 2-axes machine tool is shown and discussed in detail. The different steps of this application-oriented method are explained, showing the advantages of a highly automated process. Only the essential parameters and configuration inputs are required from the ANSYS Workbench user in order to obtain the reduced model using the stand-alone algorithm MOR for ANSYS. The resulting reduced system matrices are then imported into MATLAB, where the desired analyses are carried out in a small fraction of the time needed for a full FE-simulation.

2. Model building in ANSYS Workbench

The CAD-interfacing and pre-processing capabilities of ANSYS Workbench are used in a first step in order to import the geometry of a machine tool structure and prepare it for simulation. In its simulation environment, the different axes of the structure, as well as their coupling properties need to be defined.

2.1. Component definition

The structure of a machine tool is composed of several parts, which are either solidly joined by standard fixings (screws, bolts, etc…) in case they belong to a same body, or joined by rotating bearings resp., by linear guides when defining the interface between two axes with a relative degree of freedom. In the example covered in this paper, the structure is divided into three assemblies corresponding to the basis of the machine (Volume_2), a linear axis X (Volume_1) and a linear axis Z (Volume_3). In Figure 1 these model components are highlighted in red and the component names can be found on the "project tree" on the left.

* Pascal Maglie: Tel.: +41 (0)44 632-0944; Fax: +41 (0)44 632-1125; E-mail: maglie@iwf.mavt.ethz.ch

The area components defining the contact faces between a guide carriage and its guideway, respectively between a drive nut and its ball screw, are shown on the right in Figure 1. These area components are used during the model preparation phase to automatically create two mass elements at each coupling location and set the corresponding "net-shaped" constraint equations which connect one mass element to the moving part (carriage or drive nut) and one mass element to the fixed part (guideway or ball screw).

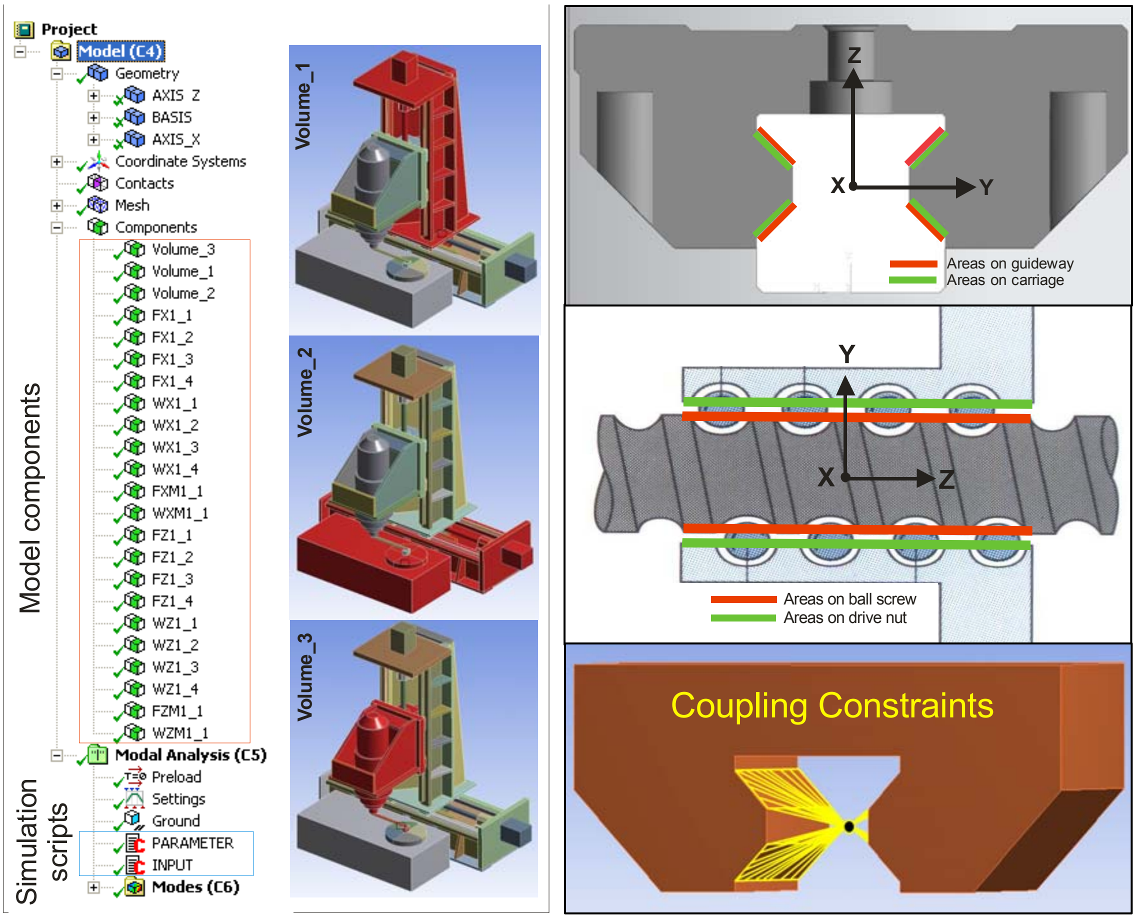

Figure 1: Illustration of the component definition in an ANSYS Workbench simulation model

2.2. Simulation Scripts

Two simulation scripts then need to be included in the modal analysis pre-processing pipeline (see the project tree in figure 1). One script sets all the parameters of the coupling properties: for each interface a six-dimensional vector has to be defined which contains the three translational and the three rotational stiffness values between the two corresponding mass elements. The second script then executes a series of self-programmed ANSYS macros which use the inputs from the named components (volumes and areas) created in the simulation model and the parameter values defined in the parameter script.

3. Exporting the Model

The standard ANSYS simulation run is stopped before entering the solution phase and the resulting outputs of the analysis are the so-called .full files containing the system matrices of the model (stiffness, damping and mass). For each interface location between two axes, one .full file per load direction has to be exported in order to be able to transfer every force and moment contribution between two adjacent bodies (see Figure 2). Additionally to these files, a set of text files containing the coupling parameters (physical stiffness and damping values) and the corresponding node numbers are automatically exported.

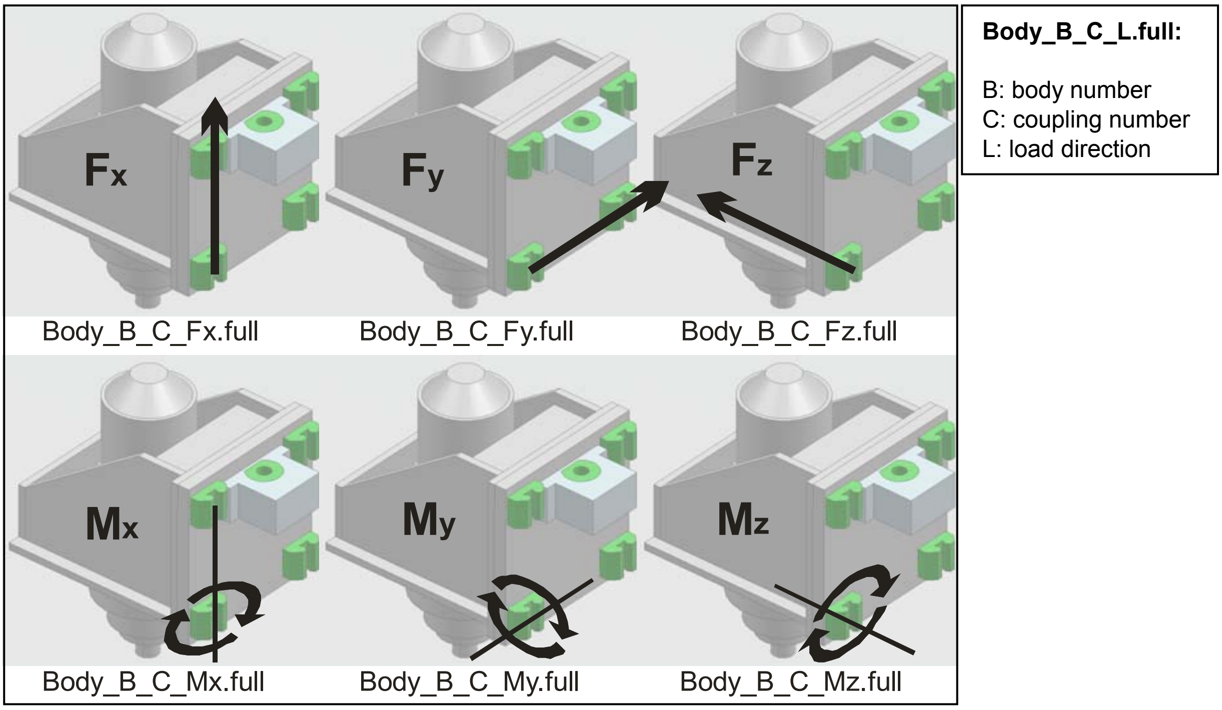

Figure 2: Load definition for the coupling locations: one .full file per load direction

4. MODEL REDUCTION

The goal of the model order reduction is to produce systems representing the relevant dynamical properties of the original systems, but with much smaller dimension. For this purpose, MOR for ANSYS *[2]*, a stand-alone program based on the Krylov-Method and implementing the Arnoldi algorithm, has been used: starting from the first order equation system in equation 1:

$$\begin{aligned} A\dot{x}(t) &= x(t) + Bu(t) \rightarrow sAX(s) = X(s) + BU(s) \\ y(t) &= C^T x(t) \qquad \rightarrow Y(s) = C^T X(s) \end{aligned} \tag{1}$$

The transfer function can be formulated as follows in equation 2:

$$G(s) = Y(s)/U(s) = C^T (I - sA)^{-1} B \tag{2}$$

The Taylor series expansion of G(s) about the expansion point $s_0 = 0$ is given by equation 3:

$$G(s) = -C^T (I + sA + s^2 A^2 + \ldots)B = \sum m_i s^i \tag{3}$$

With $m_i = -C^T A^i B$ the so-called moments about s_0.

The basic idea behind the Krylov-subspace based block-Arnoldi algorithm is to find a reduced system whose transfer function $G_r(s)$ has the same moments as $G(s)$ up to a chosen degree.

Once all the needed files from ANSYS are grouped into one directory, the order reduction of each axis can be performed separately, leading to a number of reduced systems corresponding to the number of bodies in the machine tool. For the structure considered above, which is composed of three bodies, three state-space systems are generated, each having its own reduced dimension. In order to be able to analyse the system globally, the three systems need to be combined back into one unique global state-space system (see figure 3) *[3]*:

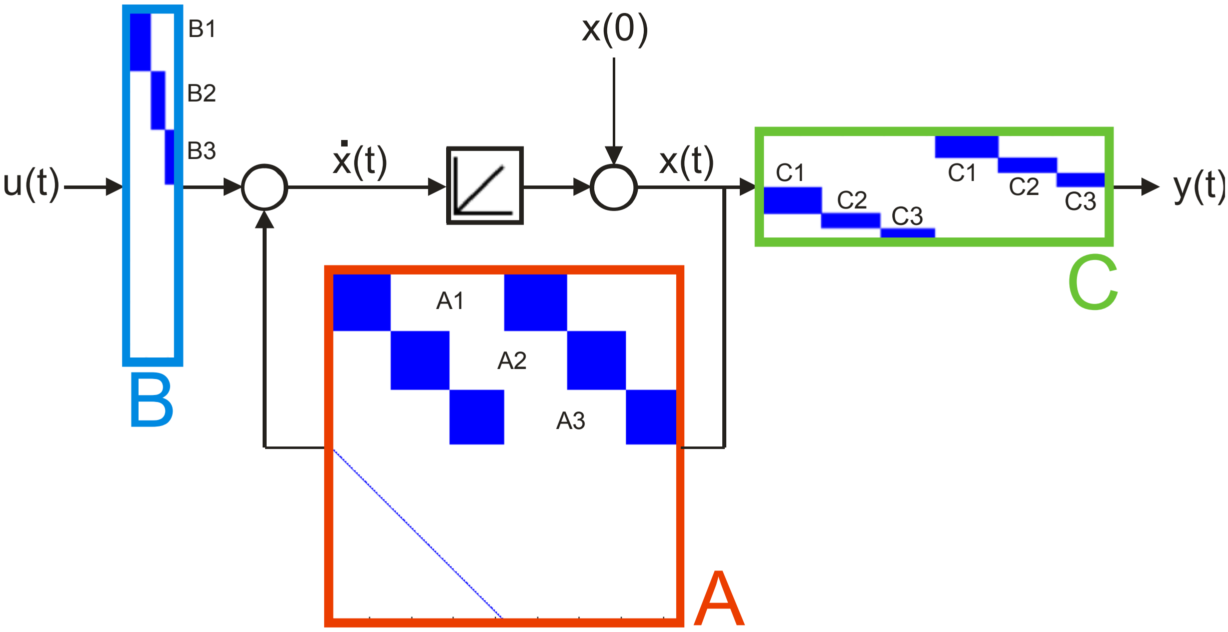

Figure 3: Combination of the three state-space systems (A_1,B_1,C_1,D_1) , (A_2,B_2,C_2,D_2) and (A_3,B_3,C_3,D_3) into one global state-space space system by assembling the individual matrices

5. Analysis in MATLAB

The combination of the state-space systems described above contains the reduced dynamical characteristics, as well as the input/output information of the single bodies. In addition to this, in order to correctly set the connections between the machine axes, the physical stiffness and damping values, contained in the coupling parameters text file, have to be incorporated into the system. With help from a “Spring-Damper” matrix (figure 4 left), the displacement and velocity outputs are automatically returned to the corresponding system inputs as forces (figure 4 right):

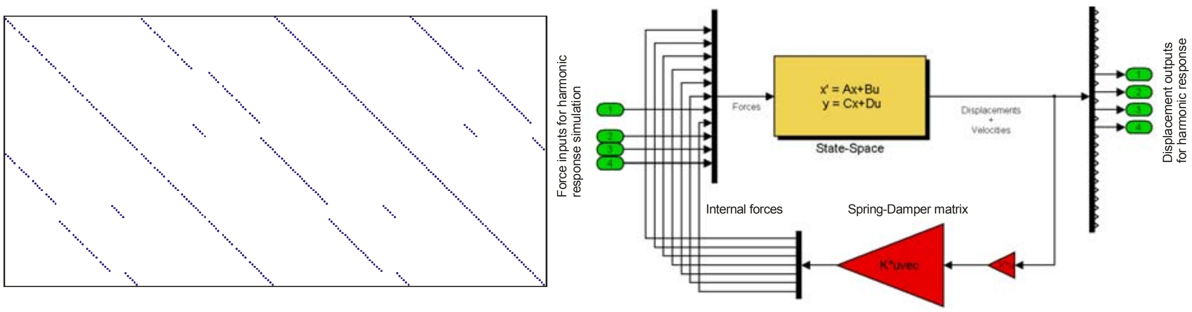

Figure 4: “Spring-Damper” matrix (left) and Simulink model (right) for a harmonic response simulation including the assembled state-space system and the additional Spring-Damper matrix specifying the internal coupling characteristics

At first glance, this whole method of reducing the single axes separately can appear to be laborious. If one is interested in the frequency response between two points of the global system, previous studies have shown that it is actually more adequate to operate a single model order reduction of the whole system (see figure 5) *[4]*. However this brings a certain number of disadvantages: firstly for complex machines having up to several millions degrees of freedom, the reduction of the full system matrices can be extremely time consuming and requires notable computing resources.

Another drawback of a global model order reduction is the impossibility of integrating additional effects/functions within the reduced system. The connections between the moving axes are especially sensitive in this regard. Most FE-analyses (modal and harmonic analyses e.g.) are performed under the assumption that the system is linear. This is acceptable for the structural investigation of the single bodies, but if couplings are involved (guideways, drives, transmissions or any other system implying a relative motion between two parts) it only constitutes a linearized approximation affecting the dynamical behaviour of the whole machine tool.

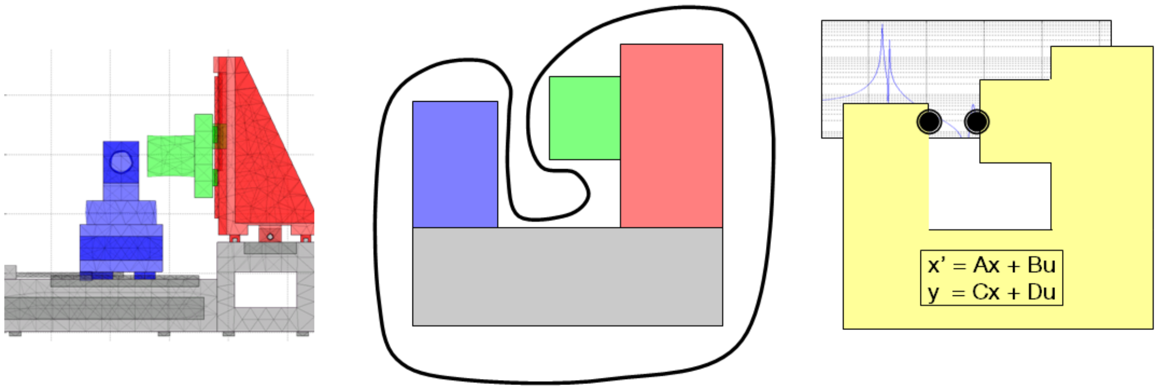

Figure 5: Standard approach consisting in performing a global model order reduction of a machine tool

By reducing single bodies separately, more detailed modelling steps can be added subsequently in the environment of the reduced system: carriages for example can be complemented by non linear phenomena like hysteresis, or transmission gears can incorporate backlash effects.

In our case, the interest of the presented method consists in having the possibility of integrating a control algorithm to the single machine tool axes (figure 6) and thus being able to study the structural behaviour under motion and the corresponding behaviour at the Tool Centre Point (TCP). This allows the investigation of dynamical effects in function of velocity, acceleration and jerk settings, as well as the optimization of the control system layout in an early stage of the development of a machine with drastically reduced computing time.

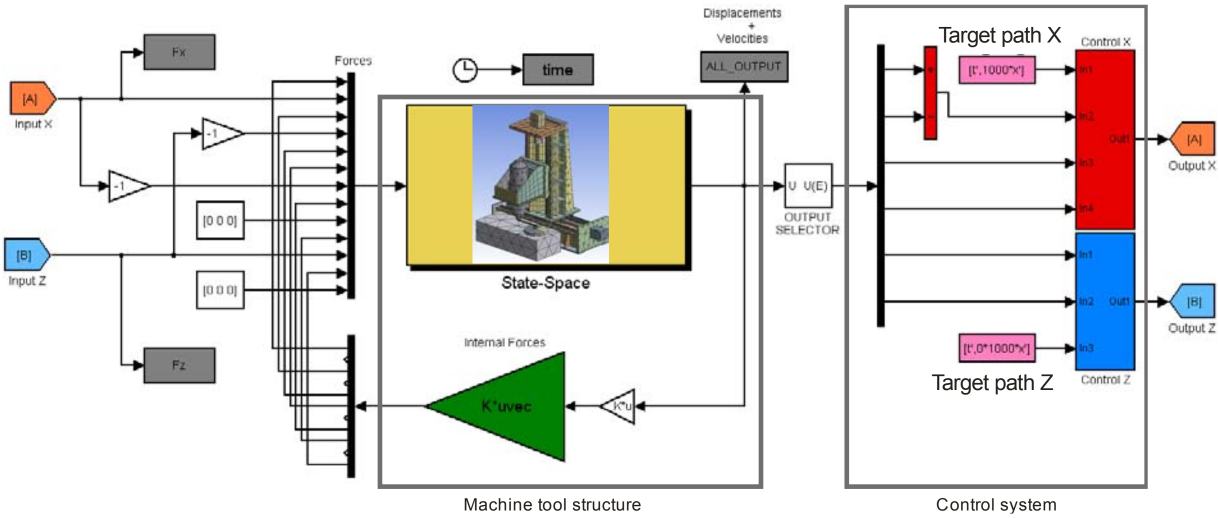

Figure 6: The complete Simulink model containing the model of the machine tool structure, as well as two basic cascaded control systems for the two motion directions X and Z.

6. Results

The first step to validate the presented reduction method, leading from the FE-model originated in ANSYS to the state-space model designed in MATLAB/Simulink, is to compare the results of two basic analyses.

6.1. Modal analysis and harmonic response analysis

On the left in figure 7 the first ten eigenfrequencies of a modal analysis are shown for a full FEM computation in ANSYS and for a reduced computation in MATLAB. The depicted table outlines the quality of the reduced model, which seems to be able to quite exactly describe the modal behaviour of the structure up to ~300Hz. On the right in figure 7 the outcome of a full harmonic analysis in ANSYS and of a reduced simulation in MATLAB are superposed in one diagram. The graphic represents the X-displacement of the TCP under a load in X-direction applied at the same location. The results reveal an insignificant error between the two methods over the entire considered frequency range.

Modal analysis

Mode number	FEM	Reduced
1.	31.21 Hz	31.21 Hz
2.	39.46 Hz	39.45 Hz
3.	47.99 Hz	47.97 Hz
4.	58.82 Hz	58.82 Hz
5.	82.95 Hz	82.93 Hz
6.	112.61 Hz	112.61 Hz
7.	155.07 Hz	155.10 Hz
8.	215.02 Hz	214.96 Hz
9.	259.88 Hz	259.84 Hz
10.	275.92 Hz	275.83 Hz

Harmonic response analysis

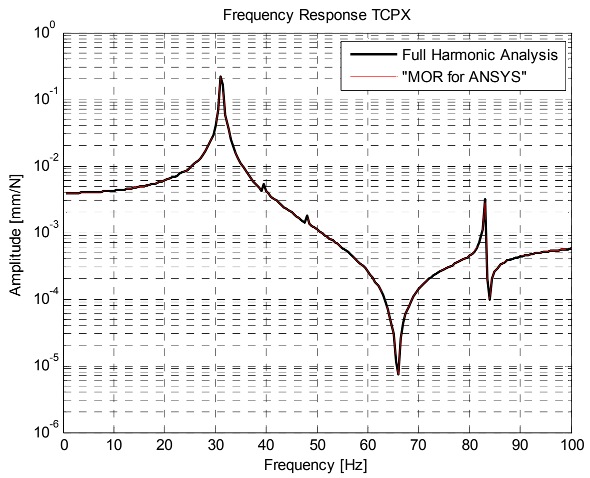

Figure 7: Comparison between full ANSYS simulation and reduced MATLAB simulation

The above considerations point out the efficiency and accuracy of the method. The quality of the results of the reduced model shows that it is possible to achieve an enormous diminution of computation time without deteriorating the dynamical behaviour of the machine tool, even in the case of a complex structure as the one illustrated in the previous chapters. In order to quantify the benefits obtained by a reduced simulation, table 1 summarizes the times required for a full harmonic analysis in ANSYS and the times required for the different steps of the reduced analysis. The advantages of the method are tangible, and become even more significative if one considers the fact that, for each further simulation run in ANSYS the time required is 144 minutes, whereas it remains limited to less than one minute for each further analysis in MATLAB.

Computing Times	Full simulation	Reduced simulation
Matrices writing from ANSYS	-	5 min
Order reduction with MOR for ANSYS	-	2.5 min
State-Space model building in MATLAB	-	0 min
Harmonic response analysis	144 min	0.5min
Total	144 min	8 min

Table 1: Comparison of simulation times for the full ANSYS model and the reduced model

6.1. CROSS-TALK

The basic mechanisms of inertial straightness deviations (cross-talk) are explained below: figure 8 shows the configuration parameters leading to the basic equations for a simplified 2D case (see equation 4). Regarding the magnitude of translational displacements at the TCP, the following factors need to be considered *[5]*:

- amount of inertia force F_a due to mass and acceleration
- offset of the inertia ΔY_{Fa} to the driving force
- offset ΔZ_{TCP} of the slide's centre of gravity to TCP in direction of motion
- tilt-stiffness $k_{rot,A}$ of the guideway system orthogonal to direction of motion given by the translational and rotational stiffness values $k_{Y,i}$ and $k_{A,i}$ of the guideway elements
- offsets of the guideway elements ΔZ_i (quadratic influence)

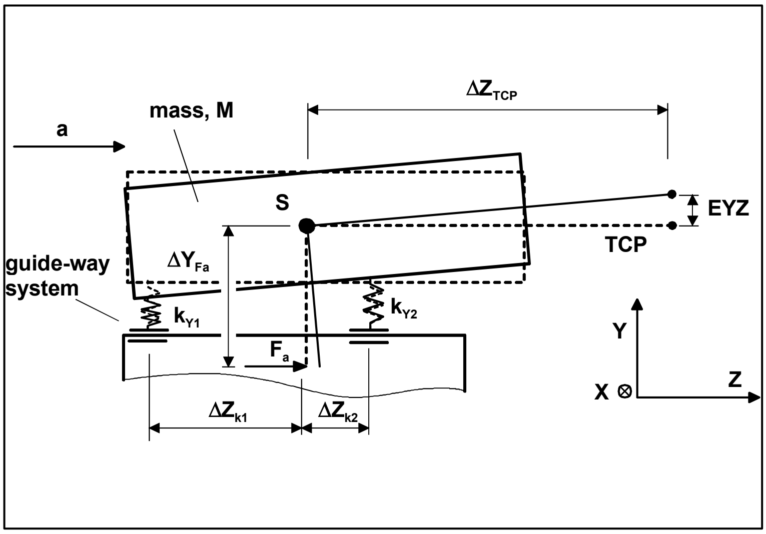

Figure 8: Schematic of the inertia tilt effects on the TCP

$$EYZ_{inertia} = \frac{F_a\, \Delta Y_{F_a}\, \Delta Z_{TCP}}{k_{rot,A}} = \frac{M\, a\, \Delta Y_{F_a}\, \Delta Z_{TCP}}{\sum_i k_{Y,i} \Delta z_i^2 + \sum_i k_{A,i}} \tag{4}$$

By means of cross-grid measurements on an existing machine, whose model has been used to demonstrate the reduced simulation method of the present paper, the effects described above should be evidenced. The required measuring devices were mounted on the machine as showed in figure 9. An offset of $X = 300mm$ has been added to the TCP in the direction of motion in order to magnify the cross-talk effects. A path of $s = 100mm$ was fed to the X-drive, and while keeping the programmed velocity at a constant value of $v = 15mmin^{-1}$ and the jerk value at $j = 300ms^{-3}$, the acceleration value has been successively set to $1ms^{-2}$, $2ms^{-2}$ and $3ms^{-2}$, in order to evaluate the effects of the inertial loads on the cross-talk deviations.

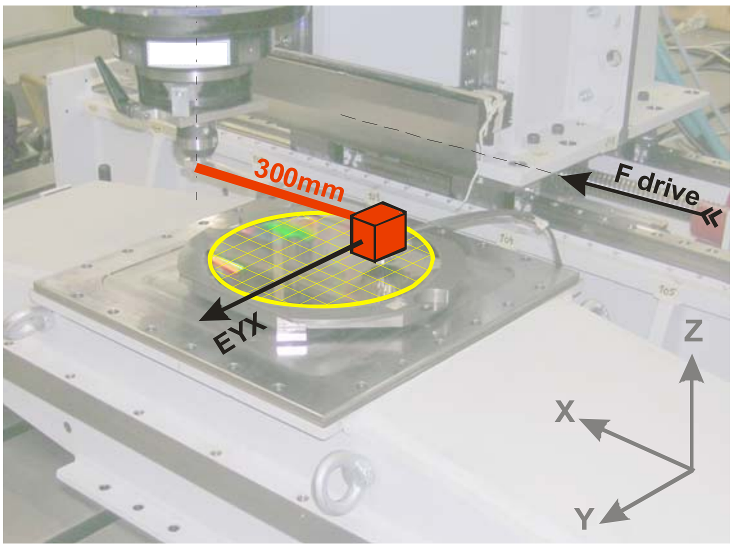

Figure 9: Cross-grid measurement set-up

The cross-talk measurements have then been reproduced using the reduced simulation model introduced in figure 6. The settings of the model controller have been approximately set to the values of the real controller and the three corresponding acceleration values were assigned to the path generator in Simulink.

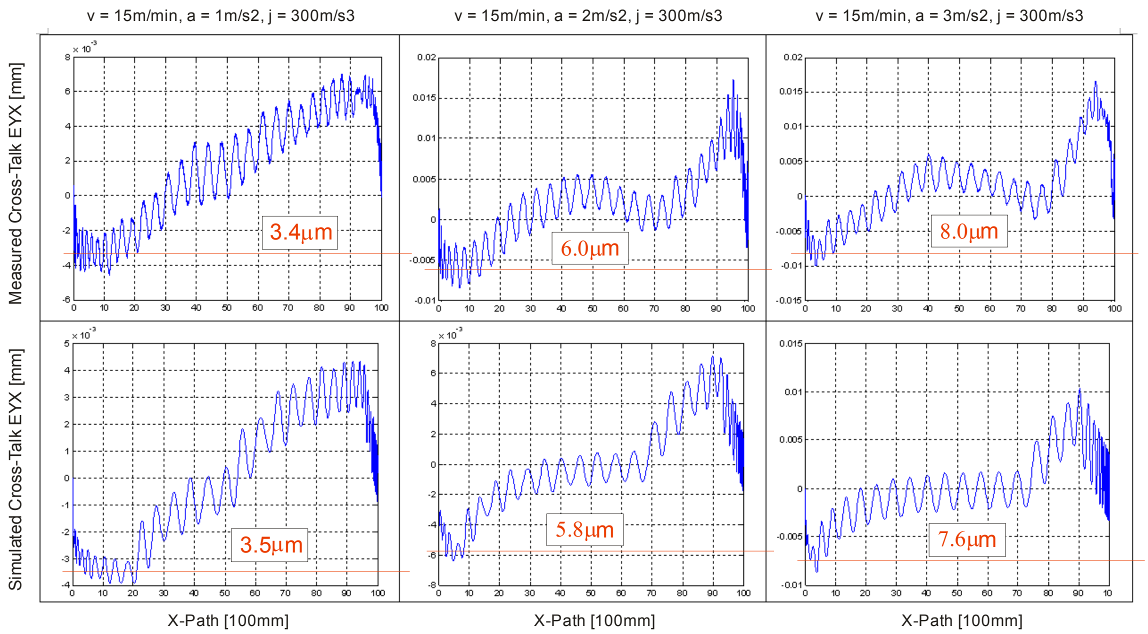

Figure 10: Measured and simulated cross-talk values by variable acceleration values

Figure 10 outlines two things: firstly, the quasi-linear correspondence between acceleration and cross-talk values, which has already been emphasized in several occasions on analogue structures, and secondly, asserting the conclusion drawn in chapter 6.1, the validity of the reduction method. The reduced model is capable of matching the measured cross-talk effects in the acceleration phase, despite rather inaccurate controller parameters, some modelling simplifications of the machine tool at CAD-level and the assumption that the global dynamic behaviour of the structure is constant over the considered X-position motion range.

7. Summary

This paper presents the step-by-step process leading from an elaborate and complex FE-model of a machine tool to its simplified and efficient reduced representation. After relating all the phases required to achieve the final model, the incontestable advantages brought by a greatly automated and powerful method are pointed out through the different analyses possibilities. The efficiency of this approach in terms of computing time and resources is supported by the validated comparisons of the results: on the one hand between the full finite-element and the reduced simulations (modal and harmonic analyses) and on the other hand between the cross-talk effects measured on the real machine and those obtained with the reduced simulation model with integrated controller.

References

[1] G. Kehl: Integrierte Simulation von Strukturdynamik und Regelungstechnik an Bearbeitungszentren, Symposium Simulation von Werkzeugmaschinen, ETH Zurich, 2008

[2] E. Rudnyi: www.modelreduction.com

[3] O. Zirn, R. Montavon: Gekoppelte Simulation von FE- und Mehrkörpermodellen für Werkzeugmaschinen, ASIM, 2008

[4] P. Maglie: Einsatz von Reduktionsmethoden für die Simulation von Werkzeugmaschinen, 13. Schweizer CADFEM Users' Meeting, 2008

[5] S. Weikert, S. Bossoni, K. Wegener: Evaluation of machine tool concepts under friction influences, Proceedings of the ASPE 2008 Annual Meeting, 2007