A method to select the finite element models for the structural analysis of machine tools

A N Polyakov and S V Kamenev
Orenburg State University, 13, Pobedy Av., Orenburg, 460018, Russia
E-mail: kamenev_sergey@mail.ru

Abstract. This paper presents a method to select the finite element models for the structural systems of machine tools. The method based on the analysis of modelling results obtained for three different machine tools using the different variants of models. Performed numerical experiments have been shown that the most accurate model is one that involves modelling of feed drives and joints. Increased computational costs, however, may reduce applicability of the model. It was found that the elimination of the feed drives from the numerical model of a machine tool increases the error of modelling (up to 25 %) when static analysis performed. The results of modal and harmonic analysis are more stable to changes introduced to the numerical model of a machine tool, as the error of calculated parameters does not exceed 10 %. Moreover, it was found that the numerical models, which include preloaded joints with stiffness more than 16000 N/μm, are different a little from the models with ideal contact. Thus, it was shown that the use of the machine tool’s numerical model without non-ideal contact and feed drives is more preferable if the acceptable error of modelling does not exceed of 25 %.

1. Introduction
Traditionally, in the design of machine tools the CAE-systems are used to estimate the different performances of a machine tool’s structural system: static, dynamic and thermal.

The structural system of a machine tool strongly affects the productivity of machining. Therefore, the mechanical performances of the machine tool’s structural system have a great importance, in particular, for high precision equipment. For example, Liang et al. [1] proposed a new method to design the mechanical structures of machine tools. This method was applied for the designing of an ultra-precision fly-cutting machine tool. Method was realized using the integrated dynamical model of machine tool structure that used to optimize a hydrostatic slide. The model was developed with use of a commercial CAE system for finite element analysis (FEA). The implementation of this model allowed reducing the weight of a machine tool carriage and investigating the influence of machine tool dynamics on machining topography.

Liu et al. [2] denoted a need to optimize the structure of a crossbeam as one of the most important structural elements of a gantry machine tool. For this purpose, an optimization model was developed using grey relational analysis and analytic hierarchy process. This model included the results of finite element analysis performed for various crossbeam structures. As a result of optimization, a best crossbeam design was found.

In another paper, Liu [3] proposed a new multi-objective optimization design method. This method was applied for the improvement of the static and dynamic performances of machine tool’s structural
parts. An optimization model was established using orthogonal experimental design method and response surface method. The solution of this model was achieved using modified particle swarm optimization algorithm and grey relational analysis method. The use of the proposed method allowed optimizing the parameters of a gantry machine tool’s slide-seat.

Fang et al. [4] pointed out to the importance of a possibility to predict the thermo-mechanical behaviour of machine tool’s structure on its design stage. As a tool for this prediction, the finite element model of a machine tool is proposed. This model takes both the thermal contact resistance of joints and the contact stiffness of joints into account and allows estimating the distortions of machine tool’s structural components induced by force and thermal loading.

Similar researches presented by different authors in references [5-10]. Each of them in some way related to the FE modelling of machine tool’s structural systems.

Based on the presented overview, following conclusions may be drawn:

- finite element method as the basis of CAE systems remains the primary analysis technique in the worldwide practice of scientific researches;
- there is a trend towards the integration of CAE systems with the various systems for automated calculations, e.g. SIMULINK;
- in the finite element analysis of machine tools there is a trend towards the complication of used geometrical models and the description of different physical processes affecting the performance parameters of machine tool;
- one of the most significant directions of the researches is the multi-objective optimization of machine tools, which is performed using finite element analysis;
- quadratic solid finite elements are mainly used to create FEA models;
- the numerical models of machine tool’s structures consider the non-ideal surface contact in joints and the couplings of machine tool components.

These conclusions as well as our own practical experience in the area of the mathematical modeling of machine tools [11] allowed us to define the problem of the research. This problem can be formulated as increasing of efficiency of the simplification of FEM models. In this case, an efficiency criteria is the calculation error for simplified model of a machine tool in comparison with the full and most detailed model of a machine tool. Quantitative metrics was set to be equal 10%.

The solution of this problem, on the one hand, is related to the fact that finite element analysis is characterized by significant calculation efforts. On the other hand, a large number of worldwide researches relied to the solution of different optimization problems, which efficiency has the strong dependence on the number and duration of performed iterations. In general, it requires optimization of a finite element model for each iteration.

2. Researching methodology
The subjects of the research were the numerical models of three CNC machine tools, namely, vertical machining center 400V (Scientific-Production Association Stankostroenie Ltd., Russia), vertical machining center TM-1P and lathe HAAS ST-10Y (HAAS, USA). At the first stage of the research, the plan for calculations was designed (table 1).

In the presented research, a FE model is considered as a complete model if it includes the most detailed geometrical model of a machine tool, the models of leadscrews and the models of contact pairs. The following variants of numerical model’s simplifications are considered: the absence of leadscrew models (category “no leadscrew” in table 1), the absence of contact models at the joints of components (category “without contact pairs” in table 1). The rigid supports are used (i.e. the supports with zero compliance) at the preliminary stage of machine tool’s structure modelling. This stage was used only to verify the accuracy of the developed model marked as category “on elastic supports” in table 1.
Additional variants of models were constructed taking into consideration all the leadscrews or the specific leadscrew only (aligned along a selected coordinate axis). The quantitative parameters of stiffness in joints and supports were also varied. The values of contact stiffness were varied in the range between 15 and 16,000 N/μm as confirmed by experimental researches [6, 12]. The stiffness of roller slideways was assumed to be approximately 2000 N/μm [6]. The maximum stiffness values (observed in [6]) were taken for the preloaded joints. The stiffness of supports was varied in the range between 3 and 300 N/μm. The basic calculation was performed for the supports with stiffness equals 30 N/μm.

Table 1. Design of experiments.

| Machine tool       | Model types and analysis types | On elastic supports without contact pairs | On elastic supports with contact pairs |
|--------------------|---------------------------------|------------------------------------------|----------------------------------------|
|                    |                                 | With leadscrews                          | Without leadscrews                     |
| 400V, TM-1P,      |                                 | A<sup>d</sup> X<sup>c</sup> Y<sup>f</sup> | Z<sup>g</sup>                           |
| ST-10Y             |                                 | S                                        | S                                      |
|                    |                                 | M<sup>b</sup>                            | M                                      |
|                    |                                 | H<sup>e</sup>                            | H                                      |
|                    |                                 | A                                        | X                                      |
|                    |                                 | X                                        | Y                                      |
|                    |                                 | Z                                        |                                        |

<sup>a</sup> S: static;
<sup>b</sup> M: modal;
<sup>c</sup> H: harmonic
<sup>d</sup> A: no leadscrews;
<sup>e</sup> X: leadscrew on X-axis;
<sup>f</sup> Y: leadscrew on Y-axis;
<sup>g</sup> Z: leadscrew on Z-axis.

The second stage of the research involves a series of calculations with the variation of different stiffness parameters. The static analysis (S) of the models without contact pairs (models of joints) was performed using the three stiffness values of supports k<sub>s1</sub>, k<sub>s2</sub> and k<sub>s3</sub> (k<sub>s1</sub> = 30 N/μm, k<sub>s2</sub> = 3 N/μm and k<sub>s3</sub> = 300 N/μm). The static analysis of the models with contact pairs was realized when the stiffness parameters of joints and supports are varied (the three parameters for joints k<sub>j1</sub> = 16000 N/μm, k<sub>j2</sub> = 2000 N/μm and k<sub>j3</sub> = 200 N/μm and the two parameters for supports k<sub>s1</sub> and k<sub>s2</sub>). The variation of stiffness parameters, in this case, was a series of calculation for the three stiffness parameters k<sub>j1</sub> and k<sub>j2</sub> when the stiffness of support k<sub>j1</sub> was constant. The forth calculation was performed for the stiffness parameter k<sub>j3</sub> and the stiffness parameter k<sub>s2</sub>.

The modal analysis (M, table 1) of the models without contact pairs was performed with the stiffness parameters k<sub>j1</sub> and k<sub>j2</sub>. The modal analysis of the models with contact pairs was performed with a same set of stiffness parameters, which used for the static analysis.

The harmonic analysis (H, table 1) was performed with a smaller number of stiffness parameters variations due to large computational efforts. The only two variants of the stiffness parameters for joints (k<sub>j1</sub> and k<sub>j2</sub>) and the two stiffness parameters for supports (k<sub>s1</sub> and k<sub>s2</sub>) were used for calculation.

The analysis of obtained results was performed during the third stage of research. This analysis allowed to make conclusions regarding the efficiency of the simplified models and the complete models.

3. Application of modelling method

The developed numerical models of the three machine tools are presented in figure 1. To identify applied loads, they are numbered labeled as it is shown in figure 1. The CAE system “ANSYS Mechanical APDL” was used to build the models. The FEM models were developed using the two standard elements “Solid185” and “Combin14”. The “Solid185” is a solid eight-node element used for all the types of the structural analysis of machine tools. The “Combin14” is a linear spring used by
both model machine tool’s supports and model the non-ideal joints of machine tool’s components [13].

The most detailed numerical model of the 400V machine tool was composed with the 221500 finite elements and almost 50000 nodes. The most simplified model of this machine tool was composed with 76000 elements, which is nearly three times smaller than that in the complete model. The number of nodes in this model has been just over 21000, which is only half of the complete model. These reductions of numerical models result in decreasing of the computational efforts for both the duration of calculations and used amount of RAM. It has significant impact on performance of the harmonic analysis and multivariant calculations. Duration of a single iteration during the harmonic analysis of a most compact model was 10.2 s (the TM-1P machine tool, the number of nodes 14511). The amount of RAM used for 100 iterations during the harmonic analysis of the most detailed model (the 400V machine tool) has been just over 8 GB. The harmonic analysis of the complete model of the 400V machine tool required more than the 56 GB of RAM, while the duration of a single iteration was about 25 s (calculations are performed using a same computer).

![Figure 1. The FEM models of the machine tools with loading conditions.](image)

Static analysis results in the most detailed FE model taking into account the joints and the feed drives in all axes obtained for the 400V machine tool reveals maximum displacements of 29.2 μm. When ideal joints were used in this model (no contact pairs), the maximum displacements were found to be 28.3 μm. As it can be seen, the discrepancy of results has not exceeded 5%. The elimination of leadscrew models results, in fact, in increase of displacements of more than 20%. However, increase of the joints compliance induces the decrease of this discrepancy. Moreover, if one of feed drives is eliminated from the model, it does not lead to the stable and significant changes of machine tool’s deformations.

The results obtained for different variants of 400V machine tool show that the changes in the completeness of design and joints features do not induce the fundamental changes of the machine tool’s displacement field. For the other two machine tools, the results of the static analysis did not reveal the influence of non-ideal joints on the static deformations of machine tool when the stiffness of joints is high. The decrease of the joints stiffness in the range from 200 to 2000 N/μm leads to the significant grows of displacements.

The overview of results obtained from the modal analysis revealed following regularities:
the non-significant differences in the modal parameters of machine tool’s structural system were observed for the numerical models of the 400V machine tool without the models of feed drives and with preloaded joints;

- the introducing of feed drive into the numerical models with the non-ideal contact at the joints does not reveal significant changes in the modal parameters of machine tool’s structural system;

- the most significant changes in the modal parameters of machine tool’s structural system were observed with non-stiff joints with stiffness in a range of 200 - 2000 N/μm;

- the stiffness of supports has a greater importance in the changing of modal parameters than the stiffness of joints.

For the numerical models of the machine tools HAAS TM-1P and ST-10Y, it was found that the effect of modal parameters change variation is minimal in the transition from more detailed model to less detailed model. For example, the feed drives eliminated from the model with contact pairs do not produce the significant changes of modal parameters. For the models with non-ideal joints, the discrepancy of modal parameters did not exceed 10 %.

To investigate the results of the harmonic analysis, frequency response curves were built. Their analysis showed that the quantitative assessment of the response curves changes is small, despite the some qualitative divergences of these curves for models presented in table 1.

The modal analysis for the model of the machine tool’s structural system that includes the models of feed drives and assumes the ideal contact at joints shows that the range from 0 to 300 Hz contains two natural frequencies only. This observation was confirmed by the plotted response curve, which shows two peaks only on corresponding frequencies.

The analysis of response curves for the most detailed model of the machine tool’s structural system shows the existence of five distinct peaks, although the modal analysis reveals six natural frequencies. It means that the one of the natural frequencies, equal to 265 Hz was not reflected at the considered point of the structural system.

The decrease of the stiffness in supports shifts the spectrum frequencies to the left.

The response curve for model of the structural system with ideal contact and without feed drives did not differ drastically from the response curves obtained with the models including non-ideal contact.

On the basis of calculated results, the response curves for the four models of structural system built for the 400V machine tool. They differ in the terms of the feed drives presented at corresponding axes. All the models presented are built using the stiffness of joints $k_{j1}$.

The analysis of a response curve for model, which considers joints, but does not consider all the models of feed drives shows that the response curve in this case is only slightly different from a response curve obtained for the model with all the feed drives. The analysis of next curves shows that the elimination of a feed drive leads to some qualitative changes in the representation of the curves. A response curve obtained for the model, in which the feed drive of Z-axis is eliminated, has the most perceptible change.

In all cases, the maximum value of dynamic displacements calculated for the components of machine tool did not exceed 50 μm. This value fits well in the range of roughness values observed for rough milling (from Ra25 to Ra6.3 μm). It means that the rough machining is realized for all the variants of models regardless of what model is used, because dynamic performances were estimated with the maximum values of cutting forces.

The calculation results for the joints with stiffness $k_{j2}$ shows that the value of dynamic displacements was grown almost double in the some variants of models. However, the analysis of response curves shows that the rough machining in the range of spindle speed from 0 to 3000 rpm can be realized with roughness, which value is about the same for all the models of structural system used.

Additional calculations were performed to estimate the effect of cutting forces that are specific to finishing machining. In this case, the amplitudes of dynamic displacements decreased almost an order
of magnitude smaller than the amplitudes observed in the previous case. The nature of response curves can provide surface roughness at level Ra0.63 \( \mu m \) when spindle speed is not exceeding 12000 rpm. However, there are the areas of increased compliance in the range from 3000 to 7000 rpm, where surface roughness is grown up to Ra0.8 \( \mu m \).

The absence of vibrations on the 400V machine tool was confirmed in the whole range of frequencies by experiments described in paper [14].

The results of harmonic analysis obtained for the structural systems of the machine tools HAAS TM-1P and ST-10Y confirm the results obtained for the 400V. At the expense of simplifications used in the models of structural system (model considers two feed drive only), the corresponding response curves are less sensitive to the different structural and parametric variations of the models.

4. Conclusions
The present research allows to make overall conclusions regarding the efficiency of machine tool’s structural system modelling on the basis of the FEA.

- the most accurate model is provided when feed drives and joints are used in the model of machine tool’s structural system; in this case the model requires a lot of computational efforts;
- the elimination of feed drives from a numerical model may cause the increasing of modelling error up to 20–25 % when static analysis is performed; in this case the results of modal and harmonic analysis are less sensitive to variations introduced into a numerical model, because the error of estimated parameters does not exceed 10%; therefore, if the results of static analysis are little significant the numerical models with ideal contacts and without feed drives are more preferable;
- the numerical models with preloaded joints (stiffness is more than 16000 N/\( \mu m \)) show small difference in comparison to the models with ideal contact; therefore, the use of models with ideal contact is more preferable when such values of joint stiffness are used.

The analysis of presented conclusions allows formulating generalized conclusion regarding selection of the numerical model that describes the behaviour of machine tool’s structural system based on finite element analysis; when the maximum margin of modelling error is 25 %, it is more preferable using the models of the machine tool’s structural system without taking into consideration non-ideal contacts and feed drives.

Carried out researches demonstrate a possibility to use the simplified models during the computer-aided analysis of the machine tool’s structural system without the drastic loss of calculation accuracy.

Acknowledgments
The reported study was funded by RFBR and Orenburg region according to the research project № 19-48-560001 and with the financial support of the Government of Orenburg Region within the framework of the project entitled "Introducing the methodology of the design-engineering preparation of press working operations in economic use of the machine-building enterprises of Orenburg Region".

References
[1] Liang Y, Chen W, Sun Y, Lu L, and Liu H 2014 Int. J. Adv. Manuf. Technol. 70 1915–21
[2] Liu S, Li Y, Liao Y and Guo Zh 2014 Struct. Multidisciplinary Optim. 50 297–311
[3] Liu S 2015 Int. J. Adv. Manuf. Technol. 78 1053-65
[4] Fang B, Gu T, Ye D and Luo T 2016 Int. J. Adv. Manuf. Technol. 87 2581–92
[5] Zhao L, Diao G, Yan P Yao Y and Chen H 2016 Int. J. Adv. Manuf. Technol. 83 729–41
[6] Gao X, Li B, Hong J and Guo J 2016 Int. J. Adv. Manuf. Technol. 86 2093–106
[7] Zhou Z, Gu L, Tan Y, Liu M, Liu Y and Li R 2017 Chin. J. Mech. Eng. 30 1262–81
[8] Wang J, Niu W, Ma Y, Xue L, Cun H, Nie Y and Zhang D 2017 Int. J. Adv. Manuf. Technol. 91
545–68

[9] Jedrzejewski J and Kwasny W 2017 *Int. J. Adv. Manuf. Technol.* **93** 1051–68
[10] Zhao D, Bi Y and Ke Y 2018 *Int. J. Adv. Manuf. Technol.* **96** 4431–43
[11] Polyakov A N and Kamenev S V 2010 *Russ. Eng. Res.* **30** 67–71
[12] Boim A G, Pratusevich R M, Kaminskaya V V and Levina Z M 2009 *Russ. Eng. Res.* **29** 71–74
[13] Krutov A V and Molodtsov V V 2011 *Russ. Eng. Res.* **31** 270–75
[14] Polyakov A N, Goncharov A N and Kamenev S V 2018 *Russ. Eng. Res.* **38** 408–10