# MODELLING OF STATIC PROPERTIES OF LOAD-CARRYING SYSTEM OF MACHINES TOOLS USING HYBRID FINITE ELEMENT METHOD

**Daniel Jastrzębski**

Summary

The paper presents a static analysis method for load-carrying systems of machine tools. Modelling concept, assumptions and theoretical fundamentals of the hybrid finite element method applied to the analysis of machine tool load-carrying systems are described.

Keywords: load-carrying system, machine tool, guideway, simulation investigations, modelling, static analysis

**Modelowanie właściwości statycznych układów nośnych obrabiarek hybrydową metodą elementów skończonych**

Streszczenie

W artykule przedstawiono metodę projektowych badań symulacyjnych układów nośnych obrabiarek w zakresie analizy ich właściwości statycznych. Podano koncepcję modelowania, założenia i podstawy teoretyczne hybrydowej metody elementów skończonych, stosowanej do analizy układów nośnych obrabiarek.

Słowa kluczowe: układ nośny, obrabiarka, prowadnice, badania symulacyjne, modelowanie, analiza właściwości statycznych

## 1. Introduction

A good design of the load-carrying system is an important task in the construction of machine tools. Load-carrying system of a machine tool is a conceptually isolated set of machine tool components that are connected by guideways. Such isolation results from the participation of these components in carrying the loads resulting from machine working processes [1]. Design of these systems requires a special attention to provide them with abilities to counteract undesired phenomena that limit machines performance. A large stiffness of these system is an essential criterion that must be met. An ever increasing machine tools end-user demands regarding the stiffness criterion

Address: Daniel JASTRZĘBSKI, D.Sc., Eng., Technical University of Szczecin, Institute of Manufacturing Engineering, Al. Piastów 19, 70-310 Szczecin, e-mail: Daniel.Jastrzebski@ps.pl

prompts designers to apply modern methods including computer simulations. Typical analysis methods, (classical finite element method) due to the specific character of load-carrying systems resulting from the guideway connections, are not commonly used in the machine tool industry. Optimum design of the load-carrying system considers various constructional variants and constructional parameters within particular variants. According to the Author there is a need for an analysis method that is a trade-off between modelling effectiveness (model preparation time and computing time) and the accuracy of the analysis from the construction analysis standpoint. Hybrid method that can handle contact joints modelling [2] presented in the paper may be an approach that is effective and adequate to give accurate results.

## 2. Concept of modelling load-carrying systems using hybrid finite element method

The paper considers load-carrying systems of machine tools that consists of frame components, usually of complicated geometry, connected by fixed or bolted joints and sliding joints. A sample machine tool load-carrying system structure is shown in Fig. 1.

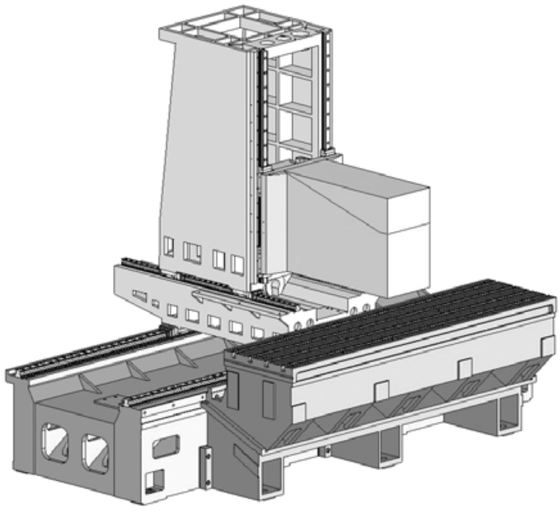

Fig. 1. Construction of load-carrying system of machine tool

To provide reliable results of the load-carrying static analysis a physical model of the system should consider, with an adequate accuracy, a series of contact phenomena and also elements deformability. Contact phenomena have a nonlinear character which implies usage of special techniques to solve these problems.

Following assumptions to the hybrid method are made to define an approach to the load-carrying system modelling. Because of the applicability range the method should provide:

- Versatility that allows to analyze load-carrying systems of different constructional forms,
- Flexibility, i.e. providing capability of carrying out analysis at various stages of the design process
- Modelling accuracy with respect to the most important physical phenomena occurring in contact joints and frame system components.

The postulated features of the method are the basis for a physical model construction.

## 3. Physical model

In modelling using hybrid method, the first thing is to assume that the model of the construction with contact joints consists of two structures: body and contact structure. Isolation of these structures results from the different approach to the modelling and analysis of physical phenomena occurring in bodies and contact joints. Body structure is responsible for the modelling of deformations of large components (tables, saddles, knees, beds, columns etc.) and small auxiliary elements (wedges, clamping elements etc.). Contact structure is responsible for the modelling of contact joints that are within load-carrying system (guideway connections). A general form of the hybrid finite element model is shown in Fig. 2.

Construction of the physical model considers the aforementioned division because both structures constitute isolated submodels in terms of theoretical fundamentals of modelling and computational algorithms.

During the selection of physical model form one may distinguish two substructures: rigid and deformable [2]. The rigid substructure includes the components that are modelled as rigid bodies. The deformable substructure consists of these components that deform under applied forces. Modelling of both substructures is performed according to the principles of appropriate versions of finite element methods [3]. Physical model of this structure reflects the construction geometry while preserving the most important dimensional features of its elements and their location. Engineering experience and preliminary knowledge about the object may allow for some simplification that do not influence significantly the analysis results, e.g. neglecting holes, small ribs, chamfers. Example of model simplifications is illustrated in Fig. 3.

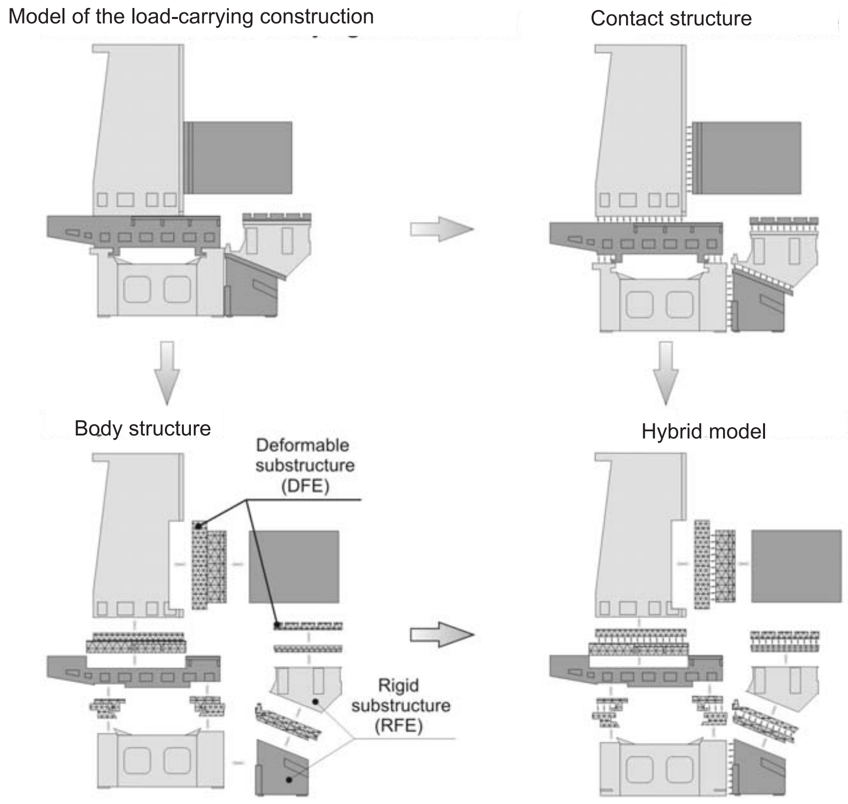


Fig. 2. General form of the hybrid finite element model

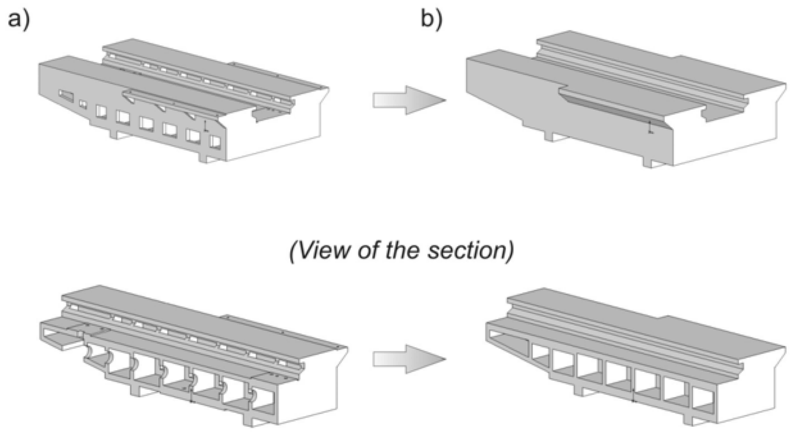


Fig. 3. Example of geometry simplifications of the load-carrying system model
a) model before simplification b) model after simplifications

Strongly preloaded joints may be treated as solid, homogeneous bodies (Fig. 4).

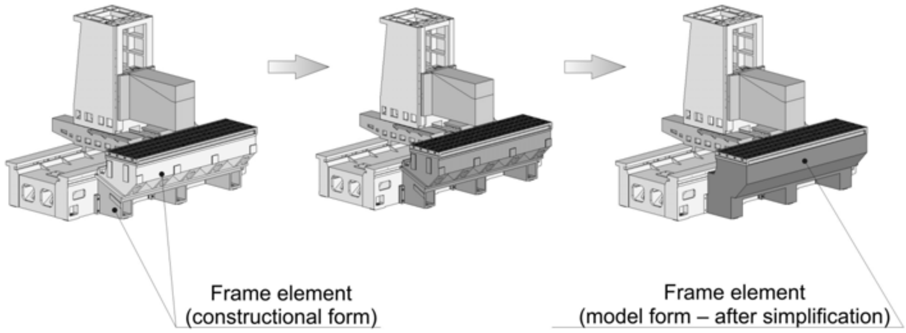


Fig. 4. Example of the load-carrying system model simplifications – strongly preloaded joints treated as solid, homogeneous bodies a) constructional form of the system, b) model after implifications

Assumed certain distribution of deformability, i.e. selection of the construction fragments where deformability is considered or neglected, is followed by the discretization of these fragments using rigid finite elements (RFE) or deformable finite elements (DFE) respectively.

It is suggested to discretize components of the structure treated as rigid bodies using natural divisions. This means that RFEs correspond to the constructional elements (or their fragments) bounded by real external surfaces including contact surfaces and surfaces located across the elements in the regions of direct interaction with DFEs. It is recommended to avoid so called conceptual divisions used in the classical rigid finite element method. These divisions are made across constructional elements at the location of spring elements and the divided parts are then modeled using RFEs. These division may be replaced by the coupling of DFE and RFE methods and also adopting contact joints modeling method.

According to the RFE method, the rigid substructure is defined in local orthocartesian coordinate system. However, in order to simplify the model of the load-carrying system it is recommended to use a single global coordinate system. Global coordinate system corresponds to the constructional datums of the load-carrying system.

Discretization of the deformable part of the construction should be performed according to the procedure and principles used in the classical DFE method [4]. At first, discretized fragment of the construction is treated as a free-body not connected with other bodies. Then nodes are selected within the body between which finite elements are spanned to fill the modelled fragment. Number of nodes depends on the geometrical complexity of the body and selected type of the finite element. Definition of the substructure topology is related to the finite element type.

The constructed fragment may be a separated construction element that interacts with other bodies of the structure through the contact structure or it may be a portion of the body that is complemented (after interaction region is defined) by the fragment of the same element modelled as a rigid body. Components of these substructure, i.e. deformable and rigid, interact directly without any intermediate elements. These interaction is conditioned by the displacements continuity determined in the nodes that belong to the both substructures. Also the postulate of the nondeformability is imposed on the aforementioned nodes, i.e. these nodes can realize only displacements that results from the rigid body behavior.

Nodes that belong to the regions of the direct interaction are excluded during model solving. This results in hybrid model size reduction and improvement of numerical properties of the problem. These nodes are activated when analyzing results to determine their contribution to the states and behavior of the load-carrying system model.

Introduction of boundary conditions to the hybrid model can be done within the body or contact structure. It results in reducing model dimensions of the body structure. One has a possibility to lock RFE and single nodes. Such flexible constraining capabilities of the method are very useful in model construction, e.g. in connecting the model to the foundation.

Size of the physical model of the body structure is determined by the total number of degrees of freedom of its two substructures. Size depends not only on the number of particular components of the substructures and locked degrees of freedom but also includes the number of nodes that belong to the region of direct interaction of both substructures. Numbers of particular elements are:

$$s_{RFE} = 6n_{RFE} \tag{1}$$

$$s_{nod} = 3n_{nod} \tag{2}$$

$$s_{RFE}^{nod} = 3n_{RFE}^{nod} \tag{3}$$

were: $s_{RFE}$ – number of DOFs of all RFEs, $n_{RFE}$ – number of all RFEs, $s_{nod}$ – number of DOFs of all nodes, $n_{nod}$ – number of all model's nodes, $s_{RFE}^{nod}$ – number of DOFs of all nodes that belong to the region of direct interaction of both substructures, $n_{RFE}^{nod}$ – number of all nodes that belong to the region of direct interaction of both substructures.

After including a complete description of the physical model the total number of active DOFs of the hybrid model is

$$s_m = s_s - s_{RFE}^o - s_{nod}^o \tag{4}$$

$$s_s = s_{RFE} + s_{nod} - s_{RFE}^{nod} \tag{5}$$

where: $s_m$ – total number of active DOFs of the hybrid model, $s_s$ – total number of DOFs before imposing the boundary conditions, $s_{RFE}^o$ – number of inactive DOFs of RFEs that are locked completely or partially, $s_{nod}^o$ – number of inactive DOFs of nodes that are locked completely or partially.

Contact structure includes conceptual contact layers. Height of the contact structure isolated from the bodies of constructional elements is limited by a depth of roughness and waviness of the modelled surface. Contact layers are regions of direct contact of body elements. These regions are located at surfaces of slideways, contact regions of the track and cylindrical or ball rollers. This structure includes also mechanisms such as ball screws.

Modelling and analysis of the contact structure is based on the concepts of the method of external loads correction [5]. Physical model of the structure is subjected to the following assumptions and simplifications:

- preserves real dimensions, shapes and location of contact surfaces;
- real contact surfaces are approximated using fragments of perfect geometrical surfaces;
- contact properties are defined in the contact layers;
- stresses and strains in the neighboring points are independent (Winkler hypothesis);
- contact properties are uniformly distributed across the contact surface;
- contact layer has a one-sided nature in terms of normal contact strains and can work only in compression;
- normal deformations are nonlinear function of the contact pressure according to the relationships formulated by Sokołowski and Lewina [3, 6, 7, 8]:

$$\delta_n = C p_n^m \tag{6}$$

where: $\delta_n$ – normal contact deformations, $p_n$ – normal contact pressure, $C,m$ – empirical parameters that are dependent on machining, surface microgeometry etc. [1, 3, 8].

A linear approximation of the formula (6) is acceptable [6, 9]:

$$\delta_n = e_{pn} p_n \tag{7}$$

where $e_{pn}$ – coefficient of normal contact elasticity.

It is assumed that for a small tangential displacements the contact has an elastic nature due to the interaction of micro-irregularities. After the limit tangential deformation is exceeded, friction forces are developed. This leads to the following mathematical notation of the phenomenon in the elastic regime:

$$\delta_s = e_{ps} p_s \tag{8}$$

where: $\delta_s$ – elastic deformation of the contact layer, $p_s$ – tangential pressure at the contact layer, $e_{ps}$ – coefficient of tangential contact elasticity.

Formula (8) is valid until the limit tangential deformation is reached, then the micro-irregularities contact is lost and sliding forces are fully developed. Then, in the friction regime, the following applies:

$$p_s = \mu\, p_n \tag{9}$$

where $\mu$ – friction coefficient.

Constructional clearances interpreted as a distance between contacting surfaces and preloads (negative clearances) can be introduced to the model. Modelling geometrical errors of contact surfaces (shape and relative surface location) consists in the introduction of variable clearances distribution.

Rolling elements of linear slides are modelled as discrete, translational spring elements that can only be compressed.

Mechanisms of feed drives are modelled as two-sided translational springs (compression and tension). Compliance of these elements results from the compliances of the components of the kinematic chain.

The contact structure, which is an isolated component of the load-carrying structure model, is subjected to autonomous idealization. The discretization of the contact surface does not depend on the discretization of the body structure, i.e. assumed body structure does not determine the approach to the contact structure discretization. This feature is advantageous when analyzing the machine tool construction in different points of the working space: the discretization of the contact surface is performed only once and only relative locations of the components of the load-carrying system are changed by moving them along the guideways.

Discretization of the contact structure is done using the contact finite element [5] (CFE). Physical form, modelling capabilities and numerical simplicity of this element is very convenient for the modelling of the load-carrying structure. The contact finite element may also be used for the modelling of machine tool slideways, linear rolling guides and fixed joints. When used for the modelling of linear rolling guides, the contact element replace naturally rolling element (balls, cylindrical rollers) or more complex rolling sub-assemblies. Modelling requires information about the stiffness characteristics of rolling elements that is usually delivered by manufacturers. If the contact elements are used to model a continuous surface (slideways) then a discretization must be performed. It consists in a division of the modeled surface into sectors of arbitrary shapes that fill the surface area (Fig. 5). Contact finite elements (CFEs) are placed in the geometrical centers of sectors.

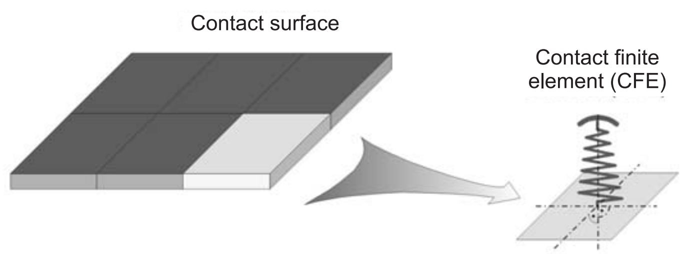


Fig. 5. Discretization of the contact surface using contact finite element

Mechanisms of feed drives are modelled as two-sided translational springs (compression and tension). Compliance of these elements results from the compliances of the components of the kinematic chain.

A complete CFE description includes the following data:

- coordinates of the element's placement in the global coordinate system;
- angular location of the element, i.e. normal direction of the surface tangential to the discretizad sector of the contact layer at the element's node;
- axial stiffness of CFE along the axis of its action;
- tangential stiffness of CFE defined in the direction of the relative tangential displacement of the interacting bodies;
- equivalent friction coefficient that describes both phenomenon of tangential friction and tangential contact deformations;
- constructional clearance or preload, measured the axis of the element's action at its node.

Modelling contact structure involves the usage of two-sided spring element (TSE). These elements can be compressed and tensed. TSEs are used to model kinematic chains of feed drives, fixed, bolted joints.

TSE is characterized by:

- coordinates of the element's placement in the global coordinate system;
- angular location of the element;
- axial stiffness.

Algorithmization of the contact structure modelling requires a formal preparation of the structure which consists in assuming an uniform enumeration of components. Number of all elements of the contact structure is given by:

$$n_k = n_{CFE} + n_{TSE} \tag{10}$$

where: $n_{CFE}$ – number of CFEs, $n_{TSE}$ – number of TSEs.

Assignment of the particular CFEs and TSEs to the elements of the body structure (to the rigid and deformable substructures) is necessary for the contact structure modelling. Hybrid method allows creation of arbitrary combinations of interacting elements that is illustrated in Fig. 6.

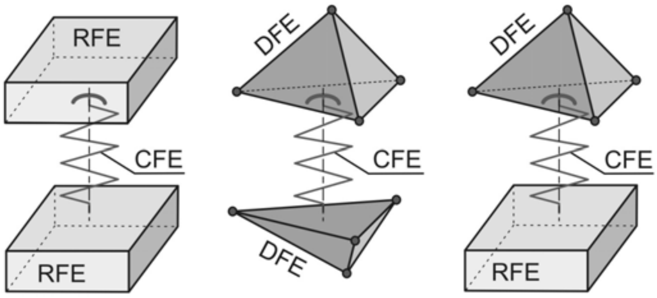


Fig. 6. Possible cases of the body and contact structure interaction

A description of the physical model of the load-carrying system is completed after loads characterization. Usually the system is subjected to forces resulting from the operational loads. The operational loads in machine tools can be classified as:

- coming from the cutting process – cutting force;
- gravitational loads of particular elements of the load-carrying system, tooling and workpiece.

## 4. Mathematical model

Mathematical model of the system is a system of algebraic equations that relate displacements and generalized forces. Assuming that the analyzed construction is a linear-elastic system, mathematical model can be formulated:

$$\mathbf{Kq} = \mathbf{Q} \tag{11}$$

where: $\mathbf{K}$ – $s_m$ x $s_m$ model stiffness matrix, $\mathbf{q}$ – $s_m$ x 1 vector of generalized displacements, $\mathbf{Q}$ – $s_m$ x 1 vector of generalized forces.

Solution of the model consists in finding the vector of generalized displacements for a given stiffness matrix when the load vector is known. This problem involves application of linear equations solving procedures, that may be formulated symbolically as:

$$\mathbf{q} = \mathbf{K}^{-1}\mathbf{Q} \tag{12}$$

If the contact structure contains nonlinear elements, then solution of the problem is more complicated and requires application of iterative methods that consist in multiple solving locally linearized models. Computing process is realized until reaching agreement of displacements and loads coupled by nonlinear relationships. Mathematical model is formulated according to the concept of external loads correction method so that factors originating from the nonlinearities (physical and geometrical) are transformed into fictional forces assigned to the load vector. Thus equation (11) becomes:

$$\mathbf{Kq}_j = \mathbf{Q}_{sj} \tag{13}$$

$$\mathbf{Q}_{sj} = \mathbf{Q} + \mathbf{Q}_{kj} \tag{14}$$

where: $j$ – iteration number added to variables in the iterative computing process of the external loads correction, $\mathbf{Q}_{sj}$ – vector of corrected forces, $\mathbf{Q}_{kj}$ – component of the vector of corrected forces

Stiffness matrix, according to the physical model of the load-carrying system, is:

$$\mathbf{K} = \mathbf{K}_B + \mathbf{K}_K \tag{15}$$

where: $\mathbf{K}_B$ – component of the stiffness matrix originating from the body structure, $\mathbf{K}_K$ – component of the stiffness matrix originating from the contact structure.

Since body elements are linear elastic, $\mathbf{K}_B$ is a constant number during the computing process. However, constancy of $\mathbf{K}_K$ component is linked to the existence of so called initial model in which each CFE is linear. Initially CFE is defined as a two-sided linear spring (tension and compression are allowed) that can not generate friction forces acting on the body structure. Also, clearances, preloads and geometry errors are not incorporated into the initial CFE. Algorithmic mapping of the initial model into the final model is realized by the external loads correction method. The final model incorporates all the features of physical model. This is illustrated in Fig. 7.

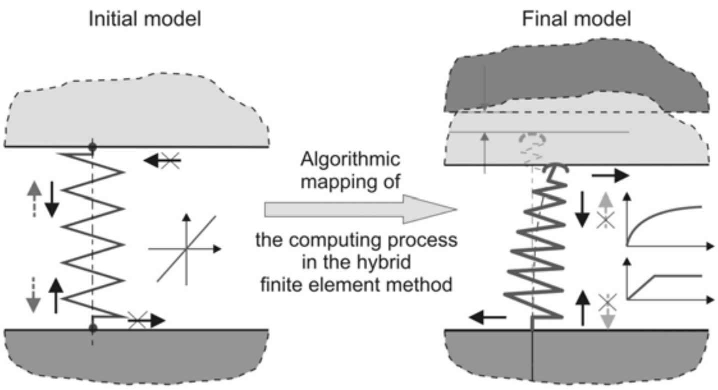


Fig. 7. Algorithmic mapping of the initial model into the final model

Stiffness matrix is a modelling synthesis of spring properties of all deformable finite elements (DFE, CFE, TSE). According to the physical model description stiffness matrix is a sum of two components (equation (15)) that synthesize spring properties of the body and contact structures. Construction of these two components consists in the appropriate aggregating block elements of stiffness matrix originating from the geometry-stiffness relationships performed with respect to particular deformable finite elements. However creation of these blocks for the body and contact structures is independent but aggregation is identical for both components of the structure. It results from the assumed discretization (division into rigid and deformable substructure) and enumeration of the structure elements.

Existence of two substructures in the hybrid model complicates the construction of the stiffness matrix that is caused by the occurrence of regions generated by the rigid substructure, deformable regions and regions of interactions between the substructures. Arranging and filling the aforementioned regions is a result of the assumptions (when DFE is understood as an element that does not interact with RFE): location of regions in the matrix is uniform and coherent; first the region of the rigid substructure is placed followed by the region of the deformable substructure (corresponding to the DOFs of all RFEs). Numeration of elements within each substructure is continuous. Global model numeration of DOFs is also continuous which means that the first DOF of the first DFE node has a number larger by one than the last DOF of the rigid substructure. Thus the number of an arbitrary DOF of an active DFE node is:

$$s_{nodi} = s_{RFE} + s'_{nodi} \tag{16}$$

where: $s_{nodi}$ – global number of DOF of the "ith" node, $s'_{nodi}$ – number of DOF of the "ith" node within deformable substructure, $s_{RFE}$ – total number of DOFs of all RFEs (1).

This notation is valid until the boundary conditions are considered. This leads to the separation of four blocks in the global stiffness matrix. Dimensions of these blocks result from the number of columns and rows of the band of the rigid substructure (width is equal $s_{RFE}$) and the deformable substructure (width is equal $s_{nod}$). Structure of the stiffness matrix is shown in Fig. 8.

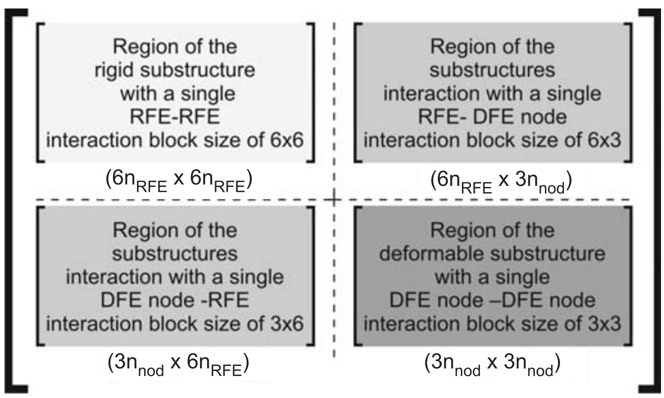


Fig. 8 Structure of the stiffness matrix of the hybrid model

Stiffness matrix contains stiffness blocks from the rigid and contact substructure. First the rigid components $\mathbf{K}_B$ and then components from the contact structure $\mathbf{K}_K$ are placed in the stiffness matrix.

Mathematical model of the load-carrying system is solved iteratively using the computing process based on the concept of the external forces correction [5]. After imposing boundary conditions the process is realized in the following steps (description of the single iteration).

**Step 1. Solving system of equations with respect to displacements**

The method is realized using a procedure of solving systems of linear equations. The proposed method employs Cholesky procedure [10] in which the stiffness matrix is decomposed during the iteration. Such form is remembered and used to determine displacements $\mathbf{q}$ in the next iteration.

**Step 2. Calculating deformations of the contact structure elements**

Deformations of the contact elements (CFE and TSE) are quantities calculated on the basis of the previously obtained solution of the system of equations, i.e. displacements of the body structure **q**. These quantities are:

$$\mathbf{u}_j = col\{u_1,...u_i,...u_n\} \tag{17}$$

where: $n = n_k$ – (according to equation (10)), $u_i$ – axial deformation of the "ith" element of the contact structure.

**Step 3. Calculating the reaction forces in the contact elements**

This step results in axial reaction forces in CFEs and SDE as proportional to their deformations. These forces are calculated as (negative sign denotes counter direction of the deformation and reaction forces):

$$\mathbf{R}_j = -\mathbf{k}_o \mathbf{u}_j \tag{18}$$

where $\mathbf{R}_j$ – matrix of reaction forces in the form:

$$\mathbf{R}_j = col\{R_1,...R_i,...R_n\} \tag{19}$$

$\mathbf{k}_o$ – matrix of axial stiffness coefficients of contact elements (obtained from the linearized characteristics assumed for the construction of the initial model) formed as:

$$\mathbf{k}_o = diag\{k_{o1},...k_{oi},...k_n\} \tag{20}$$

**Step 4. Selection of the reaction forces of contact elements that are tensed or compressed**

Reaction forces of tensed and compressed elements are selected from the $\mathbf{R}_j$ matrix according to the notation:

$$\mathbf{R}_{rj} = \mathbf{f}_{rj}\mathbf{R}_j \tag{21}$$

$$\mathbf{R}_{sj} = \mathbf{f}_{sj}\mathbf{R}_j \tag{22}$$

$\mathbf{R}_{rj}$ – vector of the reaction forces of the tensed elements, $\mathbf{R}_{sj}$ – vector of the reaction forces of the compressed elements, $\mathbf{f}_{rj}, \mathbf{f}_{sj}$ – operators of the selection of tensed and compressed elements respectively. The operators are defined as:

$$\mathbf{f}_{rj} = diag\{f_{r1},...f_{ri},...f_{rn}\} \tag{23}$$

$$\mathbf{f}_{sj} = diag\{f_{s1},...f_{si},...f_{sn}\} \tag{24}$$

elements $f_{ri}$ and $f_{si}$ are calculated according to the scheme:

$$\text{if} \quad R_i \geq 0 \vee i > n_k \quad \text{then} \quad \begin{cases} f_{si} = 1 \\ f_{ri} = 0 \end{cases} \tag{25}$$

$$\text{if} \quad R_i < 0 \wedge i \leq n_k \quad \text{then} \quad \begin{cases} f_{si} = 0 \\ f_{ri} = 1 \end{cases} \tag{26}$$

that yields

$$\mathbf{f}_{sj} + \mathbf{f}_{rj} = \mathbf{I} \tag{27}$$

where: $\mathbf{I}$ – is a $n_k \times n_k$ unit matrix.

**Step 5. Taking into account clearances and preloads**

Denoting a preload or initial clearance assigned to CFE no. „i" as $\Delta_i$, one may arrange these quantities for the complete contact structure in the matrix form

$$\Delta = col\{\Delta_1,..\Delta_i,..\Delta_n\}; \quad n = n_k \tag{28}$$

Matrix locations that correspond to SDE (last elements of $\mathbf{\Delta}$ matrix for $i > n_{KES}$) are used to place values of $\Delta_i = 0$. If a preload is assigned to a CFE then $\Delta_i$ has a negative value (calculated from the relationship $\Delta = -R_z / k_n$, where $R_z$ – preload force at the given CFE). Clearances, preloads and contact geometry (treated as variable clearance distribution) consists in calculating the additional axial forces in contact elements $\mathbf{R}_d$ according to the formula:

$$\mathbf{R}_d = -\mathbf{k}_o \Delta \tag{29}$$

Given this variable matrices $\mathbf{R}_{rj}$ and $\mathbf{R}_{sj}$ are obtained from:

$$\begin{aligned} \mathbf{R}_{sj} &= \mathbf{f}_{sj}\mathbf{R}_j + \mathbf{f}_{sj}\mathbf{R}_d \\ \mathbf{R}_{rj} &= \mathbf{f}_{rj}\mathbf{R}_j - \mathbf{f}_{sj}\mathbf{R}_d \end{aligned} \tag{30}$$

**Step 6. Introducing a nonlinearity of the normal contact deformations characteristics**

Step 6 is executed when the parameters $C, m$ that characterize physical nonlinearity are set. Coefficients of axial stiffness $k_i$ are then calculated in each iteration using secant method for a current loading of contact elements. This can be written as:

$$k_i = A_i \left(u_i^{1-m} / C\right)^{1/m} \tag{31}$$

where: $A_i$ – area of the sector that is discretized by the "ith" contact element.

Such calculated coefficients $k_i$ are placed in the matrix $\mathbf{k}_j$ identical to the construction of $\mathbf{k}_o$ matrix. Both $\mathbf{k}_o$ and $\mathbf{k}_j$ matrices are used to calculate the operator which transforms reaction forces of the initial, linear CFE into the reactions of the final nonlinear model. The operator is obtained as:

$$\boldsymbol{\xi}_j = \mathbf{k}_j \mathbf{k}_O^{-1} \tag{32}$$

Matrix $\boldsymbol{\xi}_j$ is used to calculate the reaction forces of the compressed CFEs as follows:

$$\mathbf{R}_{sj} = \mathbf{f}_{sj} \boldsymbol{\xi}_j (\mathbf{R}_j + \mathbf{R}_d) \tag{33}$$

**Step 7. Considering static friction and elastic tangential deformations of the contact layers**

Vector of tangential forces $\mathbf{R}_{tj}$ in the similar to (19) form is obtained as:

$$\mathbf{R}_{tj} = \boldsymbol{\mu}_j \mathbf{R}_{sj} \tag{34}$$

$\boldsymbol{\mu}_j$ – matrix of equivalent friction coefficients (coefficients that consider friction and contact tangential deformations).

Since SDEs do not carry tangential loads, the matrix $\boldsymbol{\mu}_j$ has a form:

$$\boldsymbol{\mu}_j = diag\{\mu_1,...\mu_i,...\mu_n,...0,...0\}; \quad n = n_k \tag{35}$$

and $\mu_i$ is

$$\text{if} \quad u_{ti} \geq u_{tlt} \quad \text{then} \quad \mu_i = \mu \quad - \text{ sliding regime}$$

$$\text{if} \quad u_{ti} < u_{tlt} \quad \text{then} \quad \mu_i = u_{ti} / u_{tlt} \ - \text{elastic deformations regime} \tag{36}$$

where: $u_{ti}$ – relative tangential displacement of the “ith” CFE ends, $u_{tlt} = \mu R_n / k_t$ – limit tangential displacement ( $\mu_i$ – friction coefficient), $k_{ti} = A_i / e_{ps}$ – axial stiffness coefficient ( $e_{ps}$ – contact tangential compliance coefficient).

When modelling contact joints, directions of friction forces result from displacements of the tangential contact nodes. Denoting direction versor of such displacement for the "ith" CFE by $\tau_i$, we have:

$$\boldsymbol{\tau}_i = -\mathbf{u}_{ti} / u_{ti} \tag{37}$$

$\mathbf{u}_{ti}$ – matrix of components of the relative tangential displacement of the CFE ends, $u_{ti}$ – relative tangential displacement of the CFE ends.

Relative tangential displacements $\mathbf{u}_{ti}$ are:

$$\mathbf{u}_{ti} = \boldsymbol{\delta}_i - u_i \boldsymbol{\eta}_i \tag{38}$$

$\boldsymbol{\delta}_i$ – matrix of components of the relative displacement of the "ith" CFE ends, $u_i$ – as in (17), $\boldsymbol{\eta}_i$ – versor of the CFE axis in the direction normal to the surface sector.

Matrix $\boldsymbol{\delta}_i$ is obtained as:

$$\boldsymbol{\delta}_i = \boldsymbol{\delta}_{ai} - \boldsymbol{\delta}_{bi} \tag{39}$$

$\boldsymbol{\delta}_{ai}, \boldsymbol{\delta}_{bi}$ – vectors of displacements of the CFE ends resulting from the behavior of the interacting body structure.

Other quantities are:

$$u_{ti} = \sqrt{\delta_i^2 - u_i^2}; \quad \delta_i^2 = \boldsymbol{\delta}_i^T \boldsymbol{\delta}_i \tag{40}$$

Friction forces are calculated only for the compressed CFEs.

**Step 8. Calculating matrix of the computational load $\mathbf{Q}_{oj}$**

Computational load is a key quantity when realizing the iterative process of external forces correction because it is used to control the process as it considered in the stop criterion, and to calculate the corrected load (14-40)

Matrix of the computational load (in "jth" iteration) $\mathbf{Q}_{oj}$ collects information about reactions of the compressed CFEs ($\mathbf{Q}_{oj}^{ks}$), tangential forces in the contact nodes of these CFEs ( $\mathbf{Q}_{oj}^{t}$ ) and forces that result from deformations of DFEs in the body structure ( $\mathbf{Q}_{oj}^{os}$ ). Thus we have:

$$\mathbf{Q}_{oj} = \mathbf{Q}_{oj}^{ks} + \mathbf{Q}_{oj}^{t} + \mathbf{Q}_{oj}^{os} \tag{41}$$

This matrix may be described as a sum of appropriately transformed (in a counter direction) reaction forces of all contact and body components of the model. The iterative process is convergent when $\mathbf{Q}_{oj}$ tends to the matrix of given external load $\mathbf{Q}$. Thus, the stop criterion is defined as:

$$\left\| \mathbf{Q} - \mathbf{Q}_{oj} \right\| < \varepsilon_{gr} \tag{42}$$

where: $\left\| \mathbf{Q} - \mathbf{Q}_{oj} \right\|$ – norm of $\mathbf{Q}$ and $\mathbf{Q}_{\mathrm{oj}}$ matrices difference, $\varepsilon_{gr}$ – limit value of the convergence criterion.

Matrix of the corrected load used in the next ($j$+1) iteration is calculated recursively as:

$$\mathbf{Q}_{sj+1} = \mathbf{Q}_{sj} + \mathbf{Q} - \mathbf{Q}_{oj} \tag{43}$$

The described mathematical model of the hybrid finite element method has been applied to construct the computer algorithm for calculating static properties of the load-carrying system.

## 5. Conclusions

Presented hybrid finite element method was programmed and became an independent solver “Helicon”. This program has been used in numerous design projects completed by researchers from the Institute of Manufacturing Engineering at Szczecin University of Technology. These projects include design analysis of the tailstock assembly guideway connection completed for the Machine Tool Research and Construction Centre, Pruszków, Poland [4]; simulations of the FV2 milling centre for the Jarocin Machine Tools Factory JAFO; construction of the load-carrying systems of FS and FNU milling machine for the Jarocin Machine Tools Factory JAFO. Results of the performed analysis were validated by results of experiments carried out on machine tools prototypes.

## References

[1] L.T. WROTNY: Projektowanie obrabiarek. Zagadnienia ogólne i przykłady obliczeń. WNT, Warszawa 1986.
[2] G. SZWENGIER: Modelowanie i obliczenia projektowe układów prowadnicowych obrabiarek. Prace Naukowe ITM Politechniki Szczecińskiej nr 512, Szczecin 1994.
[3] J.KRUSZEWSKI i in.: Metoda elementów skończonych w dynamice konstrukcji. Arkady, Warszawa 1984.
[4] D. JASTRZĘBSKI, P. PAWEŁKO, M. RATYŃSKI, G. SZWENGIER: Design calculations of a lathe's rolling connection of tailstock system. *Advances in Manufacturing Science and Technology*, **31**(2007)2, 5-24.
[5] Z.M. LEVINA: Research on the Static Stiffness of Joints in Machine Tool. Proc. 8-th Int. Mach. Tool Des. and Res. Conf., London 1965, 737-758.
[6] N. BACK, M. BURDEKIN, A. COWLEY: Analysis of machine tool joints by the finite element method. Proc. of 14-th Int. Mach. Tool Des. and Res. Conf. Manchester 1973, London-Basingstoke 1974, 529-536.
[7] M.P. DOLBEY, R. BELL: The Contact Stiffness of joints at low apperent interface pressures. *Annals of the CIRP*, **19**(1971)1, 67-79.
[8] Z.M. LEVINA, D.N. REŠETOV: Kontaktnaâ žestkost' mašin. Mašinostroenie, Moskva 1971.
[9] K. GRUDZIŃSKI: Rola, modelowanie i obliczanie połączeń elementów w konstrukcjach maszyn. Mat. XIII Sympozjonu Podstaw Konstrukcji Maszyn, Świnoujście 1987, 61-87.
[10] M. DRYJA, J. i M. JANKOWSCY: Przegląd metod i algorytmów numerycznych. Część 2, WNT, Warszawa 1988.

*Received in January 2008*