Applied Computer Systems
doi: 10.1515/acss-2015-0019
2015/18

Numerical Method for Determining Stiffness Characteristics of an Arbitrary Form Superelement

Alexander Tsybenko[1], Alexander Konyukhov[2], Hanna Tsybenko[3]
[1–3] *National Technical University of Ukraine "Kyiv Polytechnic Institute", Ukraine*

Abstract **– As part of the superelement approximation technology for fragments (subsystems) of the analyzed structures, a numerical method of determining the characteristics of arbitrary type superelements was developed. The examples of simulation models with two-node superelements demonstrated the efficacy of the method in the structural analysis of elastic systems.**

Keywords **– Finite element method (FEM), finite element (FE), superelement, set of equations FEM, element stiffness matrix, element compliance matrix, two-node superelement, beam FE.**

I. Introduction

An important stage of current technical product development is computational determination of a state of investigated mechanical systems in both static and dynamic analysis. It is necessary to obtain information about such strain-stress state in order to make a further estimation of strength, reliability and durability of mechanisms and machines as well as to select strength and kinematic load parameters for them.

The possibility of consideration an elastic system as the one that consists of mechanically connected subsystems, i.e., elements with known characteristics, can significantly facilitate the task of investigation strain-stress state by computational methods. Within this framework, rational approximation of fragments of the model under consideration is proposed in the form of their superelemental analogues. This approach eliminates the need of unreasonably computationally expensive resources in static and dynamic analysis of mechanical systems.

Work objective is to create a universal approach to the strain-stress state analysis of elastic systems with the usage of superelements, which approximate the subsystems, and also to develop a method for numerical determination of stiffness (and mass-inertia) characteristics of superelements.

II. Materials and Methods

Let us consider a static equilibrium, which is compatible with kinematic limitations, of an elastically deformable body (construction) under the action of external forces. Using the finite element method [1], [2] we accomplish a presentation of this body as discrete spacial model, being formed of various types FE, which are interfaced in nodes, with kinematic limitations and applied to the external forces of these nodes. Suppose that each node corresponds to the r degrees of freedom, the total number of nodes is N_n, then the number of degrees of freedom of the discrete model $N = N_n \cdot r$.

Introducing the FE approximation of generalized displacement on node values, we get a set of linear algebraic equations, which takes into account equilibrium conditions, linear relationship between strain and stress, conditions of compatibility of strain components and kinematic limitations [3]

$$\left[K^N\right] u^N = f^N , \tag{1}$$

where $\left[K^N\right]$ is the stiffness matrix of order Nx N; u^N is 1xN vector of generalized displacement of nodes; f^N is vector 1xN of generalized external load of nodes.

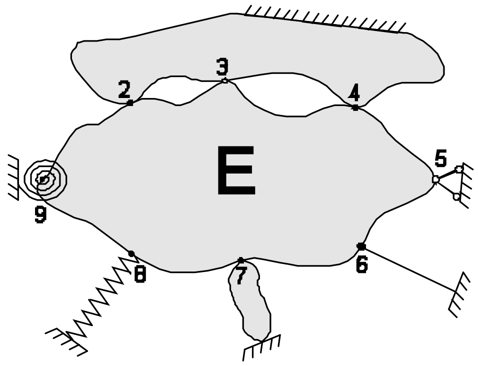


Fig. 1. Element E with interface nodes on boundary.

Let an element (Fig. 1) be connected to the rest of the construction in a limited number N^E of interface nodes. Since the degrees of freedom for the interface nodes are determined by the form of the minimum divider [4] graph of topological nodes relationships of FE system, it is possible to represent $\left[K^N\right]$ as a sum of independent components $\left[K^N\right]$ and $\left[K^E\right]$, associated with divider $\left[K^e\right]$:

$$\left[K^N\right] = \left[K^{-E}\right] + \left[K^e\right] + \left[K^E\right]. \tag{2}$$

In the absence of loading for inner degrees of freedom N^E of an element E, that describe only its internal status, it is possible to exclude from the system (1) equations that relate to u^E :

$$\begin{cases} u = u^N - u^E \\ f = f^N - f^E \\ K = \left[K^N\right] - \left[K^E\right] = \left[K^{-E}\right] + \left[K^e\right]. \end{cases} \tag{3}$$

The set of equations which does not contain internal degrees of freedom of the element E is given by

$$\begin{cases} K \quad u \;=\; f \;, \\ K \;= \left[K^{-E}\right] + \left[K^{e}\right]. \end{cases} \tag{4}$$

That way, we come to the concept of superelement, namely, an associated with the rest of construction single element, and accordingly, FE model in limited quantity of interface nodes $(N^e << N)$, which is fully characterized by its local stiffness matrix $\left[K^e\right]$ of order $(rN^e)\times(rN^e)$.

If the number of degrees of freedom for each node of the finite element model is the same, the given superelement can be directly connected to any other FE or to other superelements, and conjugation conditions correspond to the equality of kinematic and force factors at corresponding nodes (of interfacing).

Let us assume that FE model containing superelement E is characterized by stiffness matrix $K = \left[K^{-E}\right] + \left[K^e\right]$. It is also suggested that the stiffness matrix $\left[K^{-E}\right]$ is known, in other words, for every FE that is distinguished from E physical, topological and geometrical properties are defined.

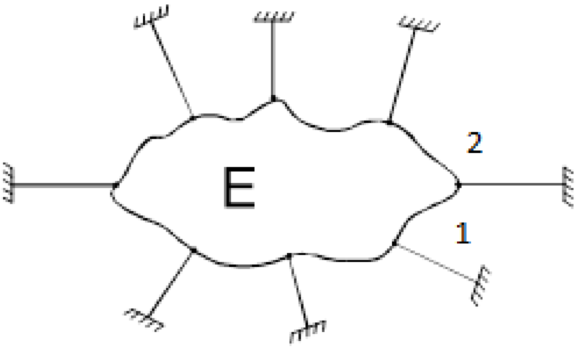


Fig. 2. Superelement E cooperating with beam type elements.

Without generality restriction, in this context, we consider a system (Fig. 2), composed of superelement E and N^e beam two-node FE [5], each of which connected with one node to a corresponding interface node of the superelement, and the other node is rigidly fixed. We shall call these beam elements "a trial".

Every element of the introduced system compliance matrix P is essentially linear $\bar{u}_{\beta i}^{\alpha j}$ or, respectively, angular $\bar{\theta}_{\beta i}^{\alpha \varphi}$ displacement of i node in the direction β under the influence of a single generalized force factor, which is applied at j node in the direction α. The system of equations (4) can be rewritten as

$$P \quad f \;=\; u \quad ,$$

where $P = \left[K^{-1}\right]$. (5)

For spatial continuum models in every node of superelement interfacing the conditions of compatibility for the three degrees of freedom, which are corresponding to linear displacements u_x, u_y, u_z, should be fulfilled. In the case illustrated in Fig. 2, the given conditions have to be supplemented by matching three rotation angles $\theta_x, \theta_y, \theta_z$, as there is a conjugation of three-dimensional continuum with beam elements having three and six degrees of freedom in every node [4].

Conjugation of the three-dimensional continuum on the six degrees of freedom can be represented as a compound of the small contact area on the surface, the stress-strain state of which can be integrally characterized by generalized force factor at the point (node interface) belonging to a given area. Thus, considering the six degrees of freedom $(u_x, u_y, u_z, \theta_x, \theta_y, \theta_z)$ in each of the nodes the system is changing the (5) as:

$$\left|\begin{matrix}
\bar{u}_{x1}^{Px1} & \bar{u}_{x1}^{Py1} & \bar{u}_{x1}^{Pz1} & \bar{u}_{x1}^{Mx} & \bar{u}_{x1}^{My} & \bar{u}_{x1}^{Mz} & \bar{u}_{x1}^{Px2} & \bar{u}_{x1}^{Py2} & \bar{u}_{x1}^{Pz2} & \bar{u}_{x1}^{Mx} & \bar{u}_{x1}^{My} & \bar{u}_{x1}^{Mz} & \cdots \\
\bar{u}_{y1}^{Px1} & \bar{u}_{y1}^{Py1} & \bar{u}_{y1}^{Pz1} & \bar{u}_{y1}^{Mx} & \bar{u}_{y1}^{My} & \bar{u}_{y1}^{Mz} & \bar{u}_{y1}^{Px2} & \bar{u}_{y1}^{Py2} & \bar{u}_{y1}^{Pz2} & \bar{u}_{y1}^{Mx} & \bar{u}_{y1}^{My} & \bar{u}_{y1}^{Mz} & \cdots \\
\bar{u}_{z1}^{Px1} & \bar{u}_{z1}^{Py1} & \bar{u}_{z1}^{Pz1} & \bar{u}_{z1}^{Mx} & \bar{u}_{z1}^{My} & \bar{u}_{z1}^{Mz} & \bar{u}_{z1}^{Px2} & \bar{u}_{z1}^{Py2} & \bar{u}_{z1}^{Pz2} & \bar{u}_{z1}^{Mx} & \bar{u}_{z1}^{My} & \bar{u}_{z1}^{Mz} & \cdots \\
\bar{\theta}_{x1}^{Px1} & \bar{\theta}_{x1}^{Py1} & \bar{\theta}_{x1}^{Pz1} & \bar{\theta}_{x1}^{Mx} & \bar{\theta}_{x1}^{My} & \bar{\theta}_{x1}^{Mz} & \bar{\theta}_{x1}^{Px2} & \bar{\theta}_{x1}^{Py2} & \bar{\theta}_{x1}^{Pz2} & \bar{\theta}_{x1}^{Mx} & \bar{\theta}_{x1}^{My} & \bar{\theta}_{x1}^{Mz} & \cdots \\
\bar{\theta}_{y1}^{Px1} & \bar{\theta}_{y1}^{Py1} & \bar{\theta}_{y1}^{Pz1} & \bar{\theta}_{y1}^{Mx} & \bar{\theta}_{y1}^{My} & \bar{\theta}_{y1}^{Mz} & \bar{\theta}_{y1}^{Px2} & \bar{\theta}_{y1}^{Py2} & \bar{\theta}_{y1}^{Pz2} & \bar{\theta}_{y1}^{Mx} & \bar{\theta}_{y1}^{My} & \bar{\theta}_{y1}^{Mz} & \cdots \\
\bar{\theta}_{z1}^{Px1} & \bar{\theta}_{z1}^{Py1} & \bar{\theta}_{z1}^{Pz1} & \bar{\theta}_{z1}^{Mx} & \bar{\theta}_{z1}^{My} & \bar{\theta}_{z1}^{Mz} & \bar{\theta}_{z1}^{Px2} & \bar{\theta}_{z1}^{Py2} & \bar{\theta}_{z1}^{Pz2} & \bar{\theta}_{z1}^{Mx} & \bar{\theta}_{z1}^{My} & \bar{\theta}_{z1}^{Mz} & \cdots \\
\bar{u}_{x2}^{Px1} & \bar{u}_{x2}^{Py1} & \bar{u}_{x2}^{Pz1} & \bar{u}_{x2}^{Mx} & \bar{u}_{x2}^{My} & \bar{u}_{x2}^{Mz} & \bar{u}_{x2}^{Px2} & \bar{u}_{x2}^{Py2} & \bar{u}_{x2}^{Pz2} & \bar{u}_{x2}^{Mx} & \bar{u}_{x2}^{My} & \bar{u}_{x2}^{Mz} & \cdots \\
\bar{u}_{y2}^{Px1} & \bar{u}_{y2}^{Py1} & \bar{u}_{y2}^{Pz1} & \bar{u}_{y2}^{Mx} & \bar{u}_{y2}^{My} & \bar{u}_{y2}^{Mz} & \bar{u}_{y2}^{Px2} & \bar{u}_{y2}^{Py2} & \bar{u}_{y2}^{Pz2} & \bar{u}_{y2}^{Mx} & \bar{u}_{y2}^{My} & \bar{u}_{y2}^{Mz} & \cdots \\
\bar{u}_{z2}^{Px1} & \bar{u}_{z2}^{Py1} & \bar{u}_{z2}^{Pz1} & \bar{u}_{z2}^{Mx} & \bar{u}_{z2}^{My} & \bar{u}_{z2}^{Mz} & \bar{u}_{z2}^{Px2} & \bar{u}_{z2}^{Py2} & \bar{u}_{z2}^{Pz2} & \bar{u}_{z2}^{Mx} & \bar{u}_{z2}^{My} & \bar{u}_{z2}^{Mz} & \cdots \\
\bar{\theta}_{x2}^{Px1} & \bar{\theta}_{x2}^{Py1} & \bar{\theta}_{x2}^{Pz1} & \bar{\theta}_{x2}^{Mx} & \bar{\theta}_{x2}^{My} & \bar{\theta}_{x2}^{Mz} & \bar{\theta}_{x2}^{Px2} & \bar{\theta}_{x2}^{Py2} & \bar{\theta}_{x2}^{Pz2} & \bar{\theta}_{x2}^{Mx} & \bar{\theta}_{x2}^{My} & \bar{\theta}_{x2}^{Mz} & \cdots \\
\bar{\theta}_{y2}^{Px1} & \bar{\theta}_{y2}^{Py1} & \bar{\theta}_{y2}^{Pz1} & \bar{\theta}_{y2}^{Mx} & \bar{\theta}_{y2}^{My} & \bar{\theta}_{y2}^{Mz} & \bar{\theta}_{y2}^{Px2} & \bar{\theta}_{y2}^{Py2} & \bar{\theta}_{y2}^{Pz2} & \bar{\theta}_{y2}^{Mx} & \bar{\theta}_{y2}^{My} & \bar{\theta}_{y2}^{Mz} & \cdots \\
\bar{\theta}_{z2}^{Px1} & \bar{\theta}_{z2}^{Py1} & \bar{\theta}_{z2}^{Pz1} & \bar{\theta}_{z2}^{Mx} & \bar{\theta}_{z2}^{My} & \bar{\theta}_{z2}^{Mz} & \bar{\theta}_{z2}^{Px2} & \bar{\theta}_{z2}^{Py2} & \bar{\theta}_{z2}^{Pz2} & \bar{\theta}_{z2}^{Mx} & \bar{\theta}_{z2}^{My} & \bar{\theta}_{z2}^{Mz} & \cdots \\
\cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots
\end{matrix}\right| \times \left\{\begin{matrix} P_{x1} \\ P_{y1} \\ P_{z1} \\ M_{x1} \\ M_{y1} \\ M_{z1} \\ P_{x2} \\ P_{y2} \\ P_{z2} \\ M_{x2} \\ M_{y2} \\ M_{z2} \\ \cdots \end{matrix}\right\} = \left\{\begin{matrix} u_{x1} \\ u_{y1} \\ u_{z1} \\ \theta_{x1} \\ \theta_{y1} \\ \theta_{z1} \\ u_{x2} \\ u_{y2} \\ u_{z2} \\ \theta_{x2} \\ \theta_{y2} \\ \theta_{z2} \\ \cdots \end{matrix}\right\}. \tag{6}$$

To determine the physical meaning of P exert a unit force P_{x1} in the first node in the direction of the first degree of freedom (linear displacement along x). The rest of the nodal loads are considered to be zero. From (6) it can be seen that the vector u is equal to the first column of the matrix P and represents generalized displacements of all nodes of the system under the action of the unit force applied in the direction u_{x1}. Then, having loaded the first node in the direction of its second degree of freedom (linear movement to y) with the force $P_{y1}=1$, the remaining nodal loads are taken as zero, and, in this case, vector u is equal to the second column of the matrix P in (6). Applying a single moment M_{x1} at the first node in the direction of its fourth degree of freedom (angular displacement x), the rest of node loads will remain zero. Then the vector corresponds to the fourth column of the matrix P, and so on.

It should be mentioned that, according to the displacement reciprocity theorem, for the elastic mechanical system compliance matrix P is symmetric [1]. Consequently, when loading system in each of the nodes with generalized unit forces in the direction of all degrees of freedom and determining generalized displacements of nodes, we get all the columns of the system compliance matrix P.

Altogether, it is obviously necessary to perform rN_n computation.

Inverting compliance matrix, we find the stiffness matrix of the system:

$$K = P^{-1}. \tag{7}$$

Since the stiffness matrix of the system with excluded superelement $\left[K^{-E}\right]$ composed of stiffness matrices of trial elements is known, then knowing K, we can obtain from (3) the local stiffness matrix $\left[K^{e}\right]$ at the ranging mark that coincides with the selected global coordinate system.

$$\left[K^{e}\right]=\left[K\right]-\left[K^{-E}\right]. \tag{8}$$

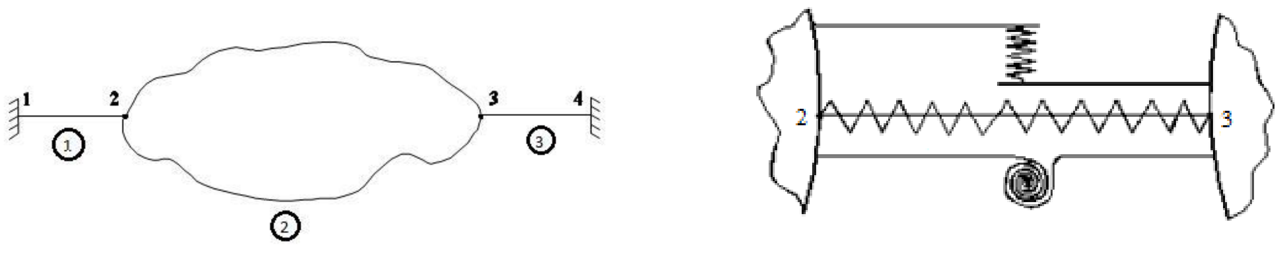
It is to be noted that the stiffness characteristics (8) of the superelement E can be used in any other finite element model with the coincidence of the topology of connections and degrees of freedom of interfacing nodes.

Finding the compliance matrix of the system is convenient to realize according to the calculations in any computing environment: Ansys, Nastran, etc. [6]. Let us consider a two-node superelement (of generalized beam type), effectively used for the approximation of attachment points of the side blocks to the central of dynamic beam models of liquid carrier rockets (CR) of packet layout [7], [8]. To find the stiffness characteristics of the superelement we consider the system of FE (Fig. 3), consisting of two beam elements № 1 (1.2 nodes) and № 3 (3.4 nodes), the local stiffness matrices $\left[K^{1}\right]$ and $\left[K^{3}\right]$ of which are known and the superelement № 2 (nodes 2.3).

Stiffness matrix of FE system (Fig. 3) is defined as the sum of elemental contributions:

$$K=\left[K_e^1\right]+\left[K_e^2\right]+\left[K_e^3\right]. \tag{9}$$

Fig. 4 shows the formation of a global stiffness matrix of the system (Fig. 3) from the local FE matrices with non-zero blocks of size 12x12 (highlighted in light gray). Components of elements № 1 and № 2 are summed up for degrees of freedom of node 2 (highlighted in dark gray), and similarly for elements № 2 and № 3 at node 3. Since the nodes 1 and 4 are restrained, the generalized displacements for them are identically zero. In this regard, by excluding 12 occurring identities of the form $0=0$, we come to the FE system stiffness matrix of size 12x12, which includes a local stiffness matrix of the superelement with two additives 6x6 in the upper left and bottom right segments (Fig. 5).



a b

Fig. 3. The calculation scheme for determining the stiffness characteristics of two-node superelement (a) and its approximation (b).

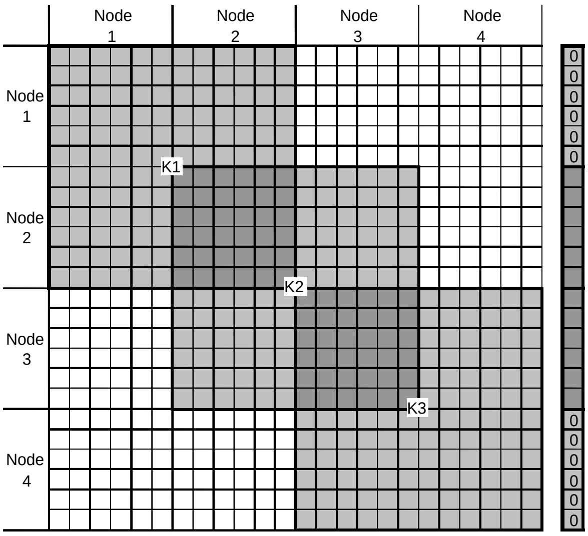


Fig. 4. Type of the global and the local stiffness matrices of FE (Fig. 3).

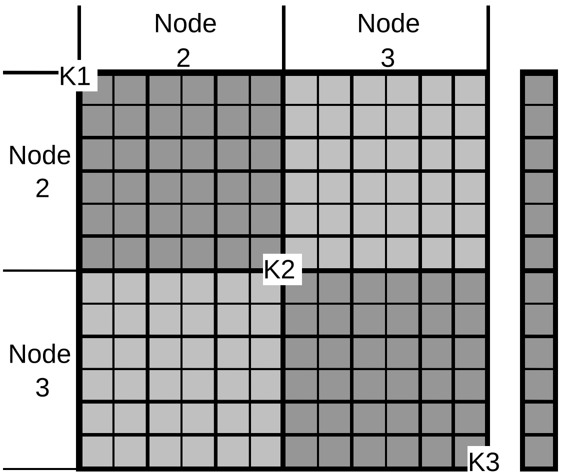


Fig. 5. Structure of the stiffness matrix of the system FE with the conditions of restraint for the 1st and 4th nodes.

Subtracting from the global stiffness matrix of the system (9) the local units of matrices FE № 1 and № 3, we obtain the sought stiffness matrix of a two-node superelement.

$$\left[K_e^2\right] = K - \left[K_e^1\right] - \left[K_e^3\right]. \tag{10}$$

To find the global stiffness matrix K of the FE system (Fig. 3) node 2 should be loaded sequentially by unit forces and moments in all six degrees of freedom, thereby determining the three linear and three angular displacements in each of the nodes 2 and 3. Thus, we derive the first six columns of the compliance submatrix 6x12 of the system.

Then in a sequential order the node 3 is loaded by unit forces and moments in all six degrees of freedom, fixing three linear and three angular displacements in each of the nodes 2 and 3. As the result we get the second six columns of the compliance system matrix, i.e. the second sub-matrix 6x12.

In total, obviously, in order to find a complete compliance matrix of FE system 12 calculations must be made; as a result of each of them 12 of generalized displacements are defined. Inverting the resulting matrix P, we obtain the global stiffness matrix K. Then, by subtracting the known blocks of stiffness matrices of beam FE we determine the local stiffness matrix of arbitrary form for the two-node superelement (Fig. 3).

Matrix $\left[K_e^2\right]$ calculated in such a manner can be used in any other model of FE without any changes with conformance of the degrees of freedom at interfacing nodes.

If the considered structural member is connected to the rest of the mechanical system in two local zones close to the point, the scheme of its two-node approximation can be used directly both in static and in dynamic analysis.

In practice, during interacting structural elements there can be a plurality of local conjugation points (bolted, riveted joints, etc.) or conjugation areas (welded joints, etc.). It is obvious, that the approximation with the usage of two-node scheme is not applicable here and simulation model with the actual number of nodes (zones) should be used, such as shown in Fig. 2.

An attempt can be made to average the obtained generalized displacements for the plurality of interfacing nodes, which leads superelement model to the conventional two-node analogue. The averaging method in each case should be selected out of structural, physical and other considerations.

Let us consider the case when there are M_1 points for interfacing with other elements of the elastic system on the surface S_1 of element E and M_2 of them on the surface S_2. By analogy with Fig. 2 we consider discrete model of system and perform $r\ M_1 + M_2$ calculations, loading in turn all the nodes belonging to the surfaces S_1 and S_2 with the generalized unit forces in the direction of all the generalized coordinates. As the result generalized displacements in each of the nodes on the surfaces S_1 and S_2 are obtained. Following (5) and (6), the compliance matrix of FE system is determined. Inverting it and subtracting the known local matrices of "trial" elements, we derive the local stiffness matrix of the superelement E of size $\left[r\ M_1 + M_2\ \right]^2$.

The static equivalent of generalized internal forces within the conjugating surfaces S_1 and S_2 can be defined as the arithmetic mean of the respective degrees of freedom in the form of vector of averaged forces f^{es}. Then, after finding the ratio of the arithmetic mean of the internal forces work to arithmetic means of the internal forces on the relevant degrees of freedom, we obtain the energetic form of the local compliance matrix $\left[P^e\right]$ of superelement. Inverting it, we get the local stiffness matrix $2r \times 2r$ of superelement with a block of the non-zero elements.

As can be seen, in the proposed method of averaging in the presence on the surfaces $S_1 + S_2$ of the element E points ($M_1 + M_2$) of conjugation it is necessary to make force calculations, and then a rather complex process for averaging the results, which is quite cumbersome. If one neglects the deformation of the conjugating surfaces, it is possible to implement a simpler engineering approach. Conventionally supplement the analyzed element with two thin weightless absolutely rigid fragments belonging to the mating surface and attached to them in the interface nodes. Let us choose on each of these fragments a single node of adduction with two "trial" elements attached to them.

The scheme corresponds to Fig. 3 and for its definition only calculations are required. Obviously, the stiffness characteristics of the thus modified superelement, when surfaces of conjugation are of small area or in case of smallness of relative strain on each of the surfaces, will be slightly different from the corresponding values of the original superelement.

III. Conclusion

1. An efficient numerical method for constructing the matrix ductility (in the form of the Green's matrix) as a result of the superposition of elastic solutions based on FEM for superelement (substructure) of any kind in its sequential loading generalized unit forces at the nodes of pairing was presented.
2. The method was developed, comprehensively tested and effectively implemented in the two-node versions of superelements used to approximate areas of the central module interfaces and side blocks in the simulation of dynamic models of liquid rockets with packet layout for commercial purposes.

References

[1] N. Bate, E. Wilson, *Numerical methods of analysis and finite element method*. Moskow: Stroyizdat, 1982.

[2] S. Klovanich, *Finite element method for nonlinear problems of mechanical engineering*. Zaporozhye: World of geotechnics, 2009.

[3] A. Tsybenko, N. Kryschuk, A. Vachev, V. Todorov, *Application program package Thermoelastoplasticity for simulation the transient thermal and non-isothermal thermal elasticoplastic stress-strain states of plane and axisymmetric bodies*. Gabrovo: VMEI, 1985.

[4] S. Pissanetski, *Technology of sparse matrices.* Moscow: Mir, 1988.

[5] A. Tsybenko, A. Konyukhov, *Simulation dynamic models of liquid carrier rockets*. Kiev: NTUU “KPI”, 2008.

[6] A. Tsybenko, M. Kryschuk, *Computer-aided design systems and engineering analysis in machine construction*. Kiev: NTUU “KPI”, 2008.

[7] A. Tsybenko, A. Konyukhov, M. Kryschuk, “Influence of the simulation model of the carrier rocket “Cyclone-4” on the calculated values of dynamic performance,” *Strenght of Materials*, vol. 3, pp. 99–105, 2015

[8] A. Tsybenko, A. Konyukhov, M. Kryschuk, “Method of interblock connections in a dynamic packet-rod model of liquid carrier rockets,” *J. of Mechanical Engineering the National Technical University of Ukraine “Kyiv Polytechnic Institute”,* vol. 73, pp. 15–21, 2015.

Alexander Tsybenko is a Doctor of Technical Sciences. He defended his thesis in 1985. He is a Professor with the Chair of Dynamics and Strength of Machines and Resistance of Materials of the Institute of Mechanical Engineering, National Technical University of Ukraine (Kyiv Polytechnic Institute). He is the author of more than 300 scientific papers. His main research interest is the development of numerical methods for solving problems in continuum mechanics.
Email: as-ts@ukr.net

Alexander Konyukhov received the PhD of Technical Sciences in 2002 . He has published over 40 scientific papers in the field of rocket and space technology. His main research interest is improvement of numerical calculation methods applied to the rocket and space technology, in particular to liquid boosters.
Email: krys@ukr.net

Hanna Tsybenko is a 4th year Master student with the Chair of Dynamics and Strength of Machines and Resistance of Materials of the Institute of Mechanical Engineering, National Technical University of Ukraine (Kyiv Polytechnic Institute). Her scientific interests include the improvement and development of methods for the numerical solution of applied problems of mechanics.
Email: as-ts@ukr.net