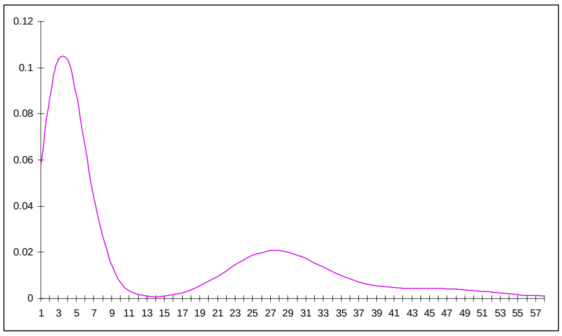


Fig. 3. Distribution probabilities of a number of events for the SM-flow at δ = 0,00001

In this research we have found the distribution of the asymptotic probability for a number of events occurring in the SM-flow in time *t*. By reducing the parameters δ, the deviation results of the asymptotic analysis for the prelimited varies: at $\delta \leq 0{,}0001$ it is equal to less than 1 %. Also, it is necessary to notice, that the obtained distribution is multimodal.

References

1. Lopukhova S. V. The asymptotic and numerical methods of studying special currents of homogenous events : candidate paper in physical-mathematical sciences. Tomsk, 2008.
2. Nazarov A. A., Terpugov A. F. The theory of probabilities and accidental processes : textbook. Tomsk : NTL publishers, 2006.
3. Gorbatenko A. E. Studying quasi-decomposable semi-Markov flow потока // The probability theory, mathematical statistics and appendix : journal of scientific article to an international science conference. Minsk, 2010. P. 53–59.
4. Nazarov A. A., Moiseeva S. P. Methods of asymptotic analysis in the theory of mass services. Tomsk : NTL publishers, 2006.

A. A. Hvalko, S. B. Suntsov
JSC “Academician M. F. Reshetnev” Information Satellite Systems”, Russia, Zheleznogorsk

V. G. Butov, A. P. Zhukov, A. A. Yaschuk
Tomsk State University, Russia, Tomsk

MECHANICAL ANALYSIS OF ONBOARD EQUIPMENT AND THE ISSUE OF FINITE-ELEMENT MODEL ADEQUACY

In this article we have considered issues related to the applied software package, designed to conduct onboard mechanical analysis of spacecraft equipment using FEM, together with an algorithm that simplifies the aforementioned models and estimates their correctness.

Keywords: hardware and software complex, mechanical analysis, satellite-borne equipment, simplified model, finite-element model.

The design of a complicated high-tech product, which includes space onboard equipment, is based on the currently used analysis methods. This allows building complicated structures and analyzing them thoroughly. Today these methods are successfully used is computers and modern CAE systems (Computer-Aided Engineering – engineering analysis systems). By applying this method, difficulties during the design and the strength analysis could be resolved, by bringing design models much closer to the existing structures.

The Finite Element Method (FEM) is one of the most universal methods of mechanical analysis, which is performed for different structures. It is, in fact, a variation method used to calculate detailed irregular models. Its capabilities of constructing overall systems, which can calculate all structural parts without separating the elements of the structure, give this method more advantages. The design of onboard equipment is characterized by the tendency of its design model improvements; this can be explained in two ways:

– in some cases it is possible to significantly reduce the mass of the onboard equipment, owing to more a detailed analyses of voltage and the distorted construction;

– this analysis is often the only method to estimate structure behavior in space conditions and on the way to space after onboard equipment simulation; ground operations in most cases are extremely limited.

Nevertheless, together with a number of beneficial characteristics FEM also has some disadvantages. The continuous progress in computational technologies stipulates the building of more detailed models, in the size of which there could be approximately 10^5 equation members. It is easier to perform a dynamic analysis of a reduced model while calculating the detailed models; this is still restricted in terms of the obtained results validity.

To eliminate such difficulties, the created applied programs packages are used to perform mechanical analysis, including analysis of the reduced models. ASONIKA-TM is the most popular product used for these purposes. It is used for analyzing mechanical characteristics of the upper level structures (cabinet, rack, and unit). Printed circuit boards with lumped EEE parts and associated material data package are not taken in account for this analysis. More over, it does not foresee a possibility to generate a design model automatically.

Due to the aforementioned, a dedicated program package had to be developed. In result of this necessity, the hardware & software system (HSS) was developed to perform onboard equipment mechanical analysis. This software has been generated to determine eigenfrequencies, effective masses, stressed-deformed states of models being exposed to linear acceleration, sine and wideband random vibrations, shocks. The objects of the HSS analysis are reduced Finite Element Models from onboard equipment accounting PCBs with EEE parts simulators.

CAE-system ANSYS and program "onboard equipment mechanical analysis" are the main parts of this HSS, since they are generated to analyze the mechanical characteristics of unspecified and unified onboard equipment structures. The dynamic analysis of onboard unit linear models is performed to verify if they can be reduced and implemented as a numerical mathematical model. Based on the analysis results, the main ways of reducing (simplifying) onboard structural elements have been defined. Also considered was the fact that implemented modifications should not cause any unacceptable errors of numeric analysis results.

For the EEE parts to be included into the onboard equipment models a special algorithm was developed; it can generate the APDL file (ANSYS Parametric Design Language) and allow automating some time-consuming modeling procedures. This algorithm substitutes EEE parts to use lumped mass elements, but not the associated increased density of materials used for frame parts; as it is done in similar calculation systems. HSS adaptation for the final user is also ensured by a friendly interface and database of onboard equipment unified structures.

Based on the requirements specified for HSS, a relay type database has selected with access of being performed via ODBC (Open Data Base Connectivity) under Microsoft Access control. Database comprises service tables, providing a description of the contents and links, existing within the data tables; and the data tables as well. The database is designed accounting its possible enlargement.

Service tables contain data referred to other tables, types of information in the database (unit, frame, fixation, material, data source, etc.), and data fields of all tables (size, variables range, description of meaning, etc.).

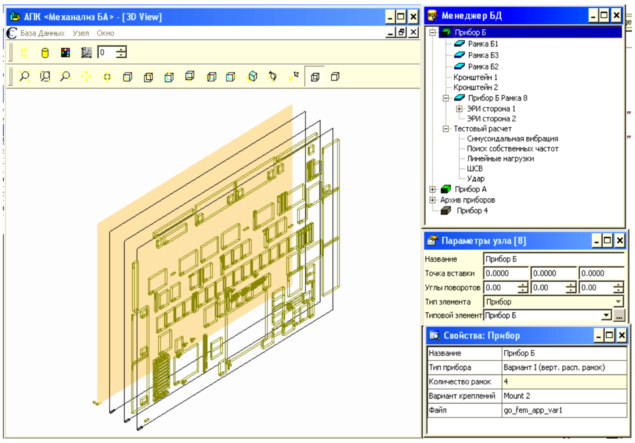
The following informational blocks are included in the data tables:

– devices;
– independent units;
– materials;
– unit structural elements;
– location and contact of units in each device;
– mechanical loads;
– performed analyses (calculations);
– EEE parts (automatically imported from PCB design systems).

Onboard equipment database of the structural design can be seen as an example, since it has a "forest" structure, comprising different "tree" elements. Thus, the top of the tree is the onboard equipment – a structural element that has a reference to its description in a dedicated line of the table. Each line in the onboard equipment table shall refer to record of layout schematics.

Onboard equipment layout schematics may have the following affiliated (daughter) elements: units comprised, additional structural elements (bracket, plank, etc.), and the calculation results. It should be noted, that each affiliated element shall have a line of its own description and its own affiliated elements. Thus, a unit with affiliated elements may have EEE parts that can be finally automatically accommodated into random groups, to be further mounted on different unit functional parts (PCBs, modules, sides of the basic plate (aluminum plate under the PCB), etc). Affiliated element calculation may result in onboard equipment reaction to particular types of mechanical loads.

Generating the database content, performing database operations, constructing onboard equipment models, and mechanical analysis procedures are performed via the user interface, which is the program shell of the HSS "onboard equipment mechanical analysis" demonstrated in fig. 1.



Название	Прибор Б		
Точка вставки	0.0000	0.0000	0.0000
Углы поворотов	0.00	0.00	0.00
Тип элемента	Прибор		
Типовой элемент	Прибор Б		

Название	Прибор Б
Тип прибора	Вариант I (верт. расп. рамок)
Количество рамок	4
Вариант креплений	Mount 2
Файл	go_fem_app_var1

Fig. 1. Graphic user interface

The program shell has 4 main windows:

– the Main window of the HSS "onboard equipment mechanical analysis" comprises the Manu with the main functions, allowing to manage database information; it is responsible for the visualization of the built models and the coordination of the other windows operations;

– the window "Data base Manager" ensures operations with the onboard equipment layout tree and automated building of a model with the accommodation of EEE part simulators;

– the window "Nod parameters" specifies the location of any structural element in relation to a parental, and displays these parameters;

– the window "Properties" allows the user to review and specify the parameters of the structural elements, referring to a layout element specified in the "Database Manager" window.

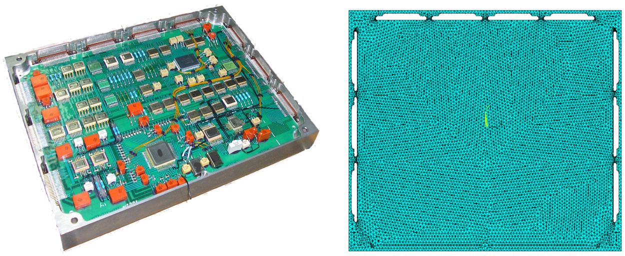
Reduction of onboard equipment structural elements 3D-models. Based on the experienced mechanical analyses of different structure FEMs built with CAD, it can be stated that it is not reasonable to use detailed CAD 3D-models to perform numerical analysis, since the recourses required for that overcome the computer's capabilities. HSS foresees the following algorithms to be used for the unit models' reduction:

– reduction of onboard equipment structural elements' 3D-models;

– automatic generation of reduced onboard equipment FEMs with EEE parts' mass simulators, accommodated in it.

The reduction of a complete onboard equipment model is performed by reducing its structural elements (frames, brackets, planks, etc), and their further connection. For the onboard equipment unified structures, recorded into the HSS database, it is predictable to automatically generate a point or allocated contacts by using additional elements which produce a mechanical join between structural parts, which are compliant with the actual equipment connected on a sufficient extent.

The operation of the reduction algorithm is demonstrated with an aluminum frame of the onboard equipment unit given in fig. 2, *a*; its complete FEM is demonstrated in fig. 2, *b*. This model is based on 92 074 solid-state elements (2 order tetrahedron SOLID92) and comprises 172 615 nods.



a *b*

Fig. 2. Appearance of the onboard unit (*a*); detailed FEM of its frame (*b*)

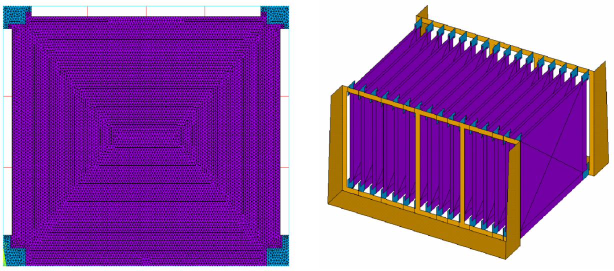
The reduction of structural elements is performed in two ways: the first one is by decreasing the task size; the second one is by excluding insignificant details. The result of this structure's modification was beam elements BEAM188 (frame) and BEAM188 (supplementary elements), shell elements SHELL181 (plate). It should be noted that model of this plate was built up as a multi-layer structure with a number of layers referring to the number of structural elements on it – such as glue, insulating spacers, PCB, coating, etc. The result of the 3D-model reduction is demonstrated in fig. 3, *a*. The reduced FEM of the frame structure comprises 21 278 elements and 11 684 nods.

The general mathematical description of element types used for complete and reduced models may be demonstrated in the form of a matrix equation:

$$[M]\left\{\ddot{u}\right\}+[C]\left\{\dot{u}\right\}+[K]\{u\}=\{F\}, \qquad (1)$$

where {u} is a vector of nodal displacement for the entire unit body; $\left\{\dot{u}\right\}$, $\left\{\ddot{u}\right\}$ are velocity vectors and vectors of each body nod velocity [*K*], [*C*], [*M*] are "global" matrixes of stiffness, damping, and masses of the entire body, {F} is a vector of equivalent nodal forces for the entire body. As it has been shown in equation (1), the mathematical representation of the reduced model is more beneficial for numerical modeling.

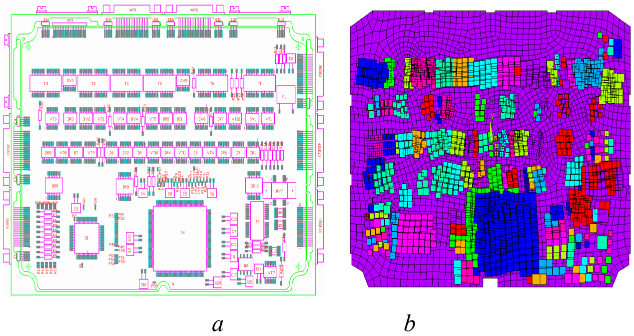
Further, the entire onboard equipment model is automatically built by comprising 16 units as shown in fig. 3, *b*. After that, the EEE parts' mass simulators are automatically accommodated onto units of the reduced onboard equipment model. This accommodation can be demonstrated with one of the units, used as an example.



a *b*

Fig. 3. Unit reduced model (*a*); onboard equipment reduced model (*b*)

The general view of the unit with PCBs and EEE parts accommodated on PCBs is shown in fig. 4, *a*. The generation of the data file for the parts' automatic accommodation on PCB is based on their attributes, generated accounted on the topology of the developed PCB. For the mass simulators to be included in the unit FEM, a special algorithm and input file APDL for ANSYS have been developed. The presence of the elements on PCB has been considered by including additional masses in the dedicated nods of the FEM. For this purpose, their each size and location onboard were used to define the network nods that were covered by the element plane. Then mass of each element was evenly distributed between two selected nods. In these nods, elements as MASS21 were generated; they had a distributed mass. The results of this algorithm operation are demonstrated in fig. 4, *b*.



a *b*

Fig. 4. General view of the unit with PCB and EEE parts (*a*); unit plate with automatically generated EEE parts' mass simulators (*b*)

Estimated conformity of the developed models. Eigenfrequencies of one of the units were calculated; the obtained results were compared to the experimental in order to estimate the conformity of the onboard equipment reduced model. One of the main parameters of the reduced model is a number of finite elements (FE), comprised into the model. Consequently, when changing the number of FE, it is possible to vary deviations of the eigenfrequencies of the design model relatively to the empirical one. The following equation was used to calculate the relative deviation:

$$\varepsilon = \frac{f_e - f_P}{f_e} \cdot 100\,\%, \tag{2}$$

where f_e stands for frequencies obtained by applying empirical methods; f_P stands for frequencies, obtained while calculating the reduced model.

Regarding fig. 3, *a*, the main quantity of the nods and of the reduced model is accommodated on a plate; this means that it is reasonable to use the plate to perform the experiment with a changing number of FE. Fig. 5 demonstrates relative deviations' dependability on the number of elements, comprised in the model, when determining eigenfrequencies, used in modeling the plate.

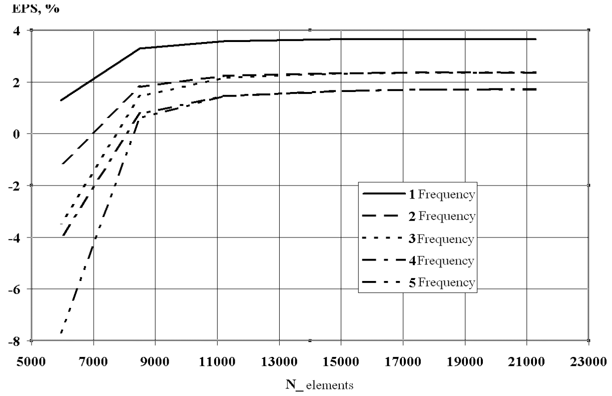


Fig. 5. Relative deviations' dependability upon the number of the model FE

As it is shown in the figure above: with the growth of FE quantity, the deviations at first sustain their actual rate, but as soon as they are equal to 9 000, their quantity falls down abruptly, and goes the same way up to 20 000. There is no sense in the further increase in the quantity of FE in the model, since the mathematical model becomes significantly complicated, while its stability does not improve much. Consequently, it is acceptable to decrease the number of FE to 9 000. Fig. 6 demonstrates the results of the dynamic analysis, performed with the unit model under sine vibration, and with 20 000 of FE at the plate of the unit. The acceleration was calculated at the point of sensor installation which is perpendicular to the unit plane. The damping factor was equal to 2 %. The figure also provides results obtained during the tests of this unit.

The calculation results performed for the entire unit were compared with the results of the actual experiment in order to confirm the application of a correct philosophy, used for design model reduction and adjustment. The vibration test and mechanical analysis were performed using real onboard equipment, comprising 16 units, and using its numeric model, provided in fig. 3, *b*. It should be noted that the mechanical analysis was performed by using two numerical models: I – less-weight model with less than 20 000 FE on the unit plate; II – light model with at least 9 000 FE on the unit plate. Only clearly marked peaks were considered. The frequencies, defined by experience were placed into compliance with the design frequencies, the values of which were closest to the experienced ones.

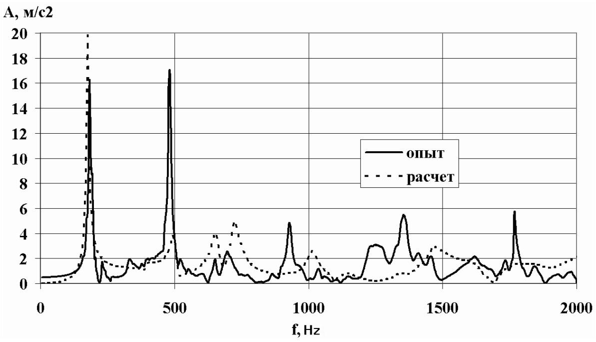


Fig. 6. Results of the design and experimental models' dynamic analysis

Table demonstrates the design and experimental frequencies for type I and II models. Besides that, the effective masses are also provided. If the effective masses are analyzed it can be seen, that some frequencies induced during the experiment have an effective mass close to 0 during calculations. The probable reason can be that the boundaries specified for the calculations do not fully comply with the experimental conditions.

The results compared for type I and II models have demonstrated that the type II light model can provide positive results as well. More evident changes could be observed for some effective masses. For instance, is type II model: the frequency effective mass with number 6 increased approximately by two times, while the frequency effective mass with number 7 decreased.

Besides the implementation of functions and tasks assigned to the developed HSS, it can also smooth interaction between the designer and CAE-system (for example, ANSYS). The developed structure of the database, interface, and algorithms of design models' reduction allow the designer to be rapid with the necessary data, and perform the mechanical analysis of onboard equipment when using unified structures. The schematics of model conformity estimations provided above, permitted the provision of qualitative content for the HSS database, comprising unified structures of onboard equipment and performing valid calculations. There is also a possibility to perform a mechanical analysis for the models created with other CAD programs. The activities performed in terms of this analysis, have demonstrated the effectiveness of the applied methods in resolving difficulties of Designer/CAE-system operation; thus, activities related to the content of the database and HSS improvement are in progress.

Design and experimental frequencies for the models type I and II

Frequency №	Object				
	Real onboard equipment	Model I		Model II	
	Frequency, experimental value, Hz	Frequency, design value, Hz	Effective mass, kg	Frequency, design value, Hz	Effective mass, kg
1	183.0	171.1	0.291695	176.0	0.288 266
2	–	320.2	3.90×10^{-04}	324.6	6.96×10^{-04}
3	–	388.8	7.63×10^{-06}	398.0	1.31×10^{-03}
4	482.7	490.9	2.13×10^{-02}	498.2	1.83×10^{-02}
5	–	545.8	3.54×10^{-03}	554.7	2.77×10^{-03}
6	653.0	660.0	1.30×10^{-02}	655.5	2.53×10^{-02}
7	700.0	729.9	2.12×10^{-02}	722.9	1.80×10^{-02}
8	–	740.2	3.98×10^{-03}	745.5	2.00×10^{-03}
9	–	854.6	1.42×10^{-04}	857.2	1.06×10^{-04}
10	–	885.6	1.36×10^{-04}	890.0	7.68×10^{-05}
11	930.9	962.7	9.71×10^{-05}	943.5	2.19×10^{-04}
12	1 043.0	1 018.2	6.74×10^{-03}	1 013.5	6.64×10^{-03}
13	–	1 071.9	2.01×10^{-03}	1 083.9	1.23×10^{-03}
14	–	1 151.1	9.62×10^{-05}	1 121.4	9.41×10^{-04}
15	–	1 152.9	1.06×10^{-03}	1 148.3	1.02×10^{-03}
16	1 237.0	1 258.4	7.21×10^{-05}	1 259.3	6.51×10^{-07}
17	1 276.0	1 276.1	7.74×10^{-05}	1 288.8	2.98×10^{-04}
18	–	1 336.3	3.11×10^{-03}	1 341.6	4.63×10^{-06}
19	1 359.0	1 362.4	1.58×10^{-03}	1 343.1	3.83×10^{-03}
20	1 418.0	1 406.9	7.05×10^{-04}	1 410.4	1.15×10^{-03}
21	1 462.0	1 456.8	1.21×10^{-02}	1 460.1	1.13×10^{-02}

I. V. Ignatyev, A. E. Kovrov
Bratsk State University, Russia, Bratsk

ALGORITHM OF CHOICE FOR AUTOMATIC EXCITATION REGULATOR SETTINGS IN MULTIMACHINE ELECTRIC POWER SYSTEMS

A program-realized coordinated algorithm of choice for automatic excitation regulator settings has been developed. This algorithm is based on the resultant theory and applies a mathematical model which is synthesized by experimental frequency characteristics of an electric power system.

Keywords: automatic excitation regulator, resultant, stability.

The ensuring of stability in electric power systems (EPS) and the damping of fluctuation are realized by automatic excitation regulators (AER) which are placed in power station generators [1].

Today it is matter of current coordinated choices in AER stability coefficients, in conditions of ensuring multimachine EPS with the required intermediary processes quality.

To solve this problem, algorithms have been developed based on the D-dividing method [2–4]. These algorithms assume serial choices for AER settings and for each separate station by a calculated field of stability. In this case, transition from one station to another is realized by increasing the system stability extent. However, in connection with the complexity of using special-purpose functions, these algorithms do not allow us to ensure acceptable EPS property damp with a greater AER number.

Another way to solve this problem is to use algorithms based on the principal matrix calculation values of the Gorev-Park linear differential equation [5–7]. However, these algorithms have a complex operative control in multimachine EPS faults:

1) they are characterized by a high order of differential equation demanding significant calculations;

2) they admit average data value for elements of large EPS units and subsystem during large intervals. The last results in non-conformity mathematic model forming during a current mode situation.

In [8; 9] it has been shown that EPS mathematic models may be obtained through the result of experimental frequency characteristic stability parameters as a characteristic polynom. It allows avoiding of many admission and simulation errors, which are typical for calculated methods; the describing the upper and lower dimension of the AER choice problem settings to