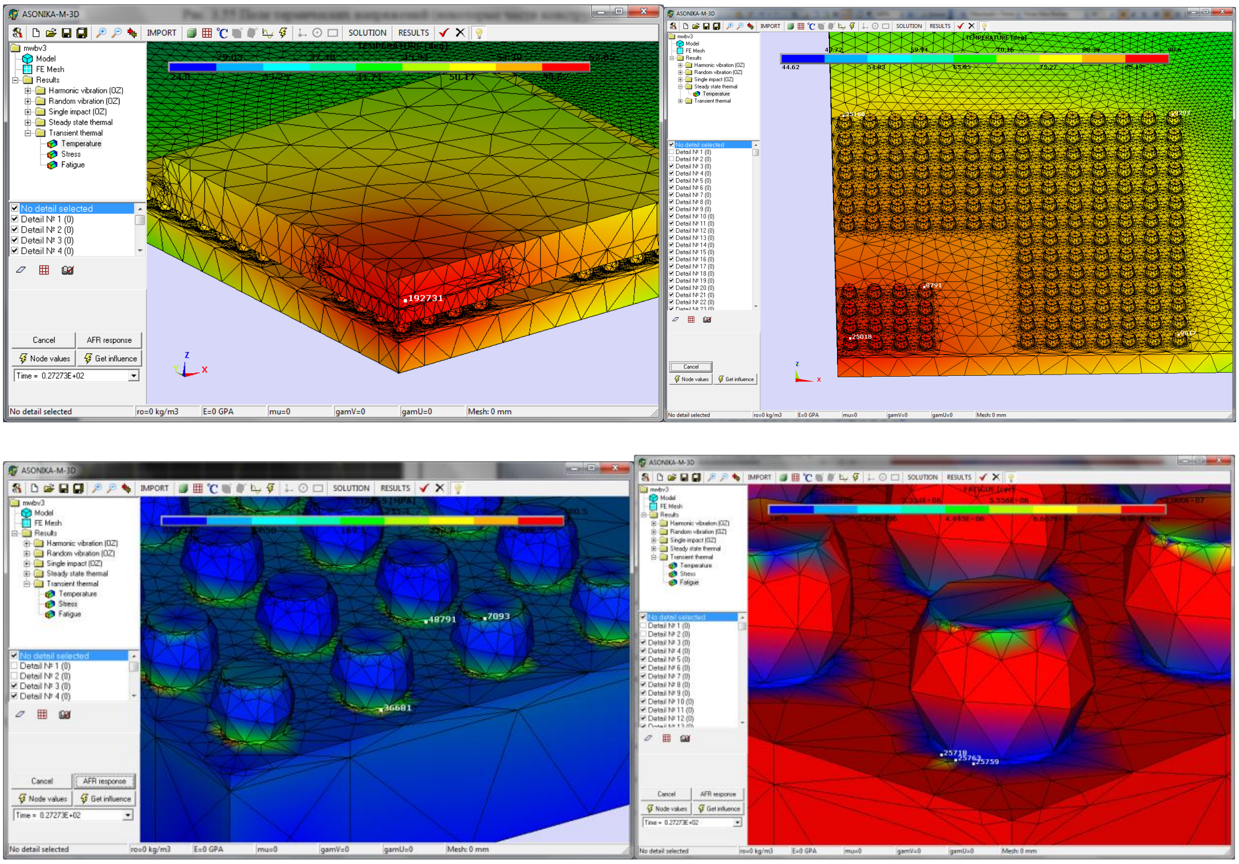
Accelerated Simulation of Thermal and Mechanical Reliability of Electronic Devices and Circuits

Prof. Alexander Shalumov

Ph. D. Evgeny Pershin



2013

INTRODUCTION

Currently, when creating electronic devices and integrated circuits (ICs), the following problems arise: 1) electronic devices and ICs in space and aviation are subjected to intense heat and mechanical effects. Because of this, their reliability is drastically reduced and there is a risk of failure of the object (rockets, planes), which is controlled electronically. It is necessary to predict reliability of thermal and mechanical effects before manufacture of electronic devices and ICs; 2) programs for rapid modelling simulation are required; 3) important design details must be taken into account during modeling. There is a need for a database with material properties and electronic components; 4) There is a need for programs with a simple and intuitive interface, so they can be used by designers; 5)the model should be created automatically. Meanwhile, 3D-models of chips should be imported from widely used CAD systems such as ProEngineer, SolidWorks, Inventor and others having either IGES or SAT file format. Models of printed circuit boards should be read automatically from the known CAD systems - PCAD, Mentor Graphics, Altium Designere, OrCAD, and others.

All these problems are solved by our program - Automated System of Analysis, ASONIKA, developed by a team of Research Institute "ASONIKA" (Russia).

The development of ASONIKA is directed by professor, Dr. Alexander Shalumov - Director General of the research institute "ASONIKA" (Russia) and the Chair of the Information Technologies Department at the Russian Presidential Academy of National Economy and Public Administration. He is also the founder and director of the Research School of Simulation, Information Technology and Automated Systems.

The system is registered in the U.S. and is currently being tested in a number of universities and companies in the USA. There are positive reviews. The system is used by Russian companies for more than 30 years. This book examines the use of ASONIKA system for modeling chips during harmonic and random vibration, single and multiple impacts, linear acceleration and acoustic noise, and steady-state and transient thermal effects. Calculations are done for thermal stress during changes in temperature and power in time. Calculations are done for the number of cycles to fatigue failure under mechanical as well as cyclic thermal effects. Simulation results for reliability analysis are taken into account.

1. MATHEMATICAL MODELS OF MECHANICAL PROCESSES IN MICROCIRCUITS

1.1. The model for the analysis of accelerations and stresses in chips exposed to vibration and shock

For the microcircuit analysis of mechanical and thermal effects the method of finite elements is used. Mathematical modeling of mechanical processes with finite element method comprises the following steps:

- obtaining a finite element model,
- setting properties of materials,
- entering loads and endurance conditions,
- carrying out simulation,
- review and analysis of results.

Finite element model (FEM) of the chip can be obtained on the basis of solid modeling (Solid model), built with one of the CAD-systems (SOLIDWorks, ProEngineer, etc.). ANSYS allows to get these results not only through interactive interface, but also using command macros. With meshing finite elements, the following challenges arise:

1. Ensure coherence of a solid model imported into the finite element environment of ANSYS. Coherence refers to the definition of the geometric forms of contact between different parts that make up the chip (substrate, solder ball). To make the binding process successful, the original solid model should not have overlapping parts.

2. The mesh should have an optimum size: not too big to avoid calculation errors and not too small to avoid wasting computer resources.

Binding and partition processes can be automated. Software interface between systems of ASONIKA and ANSYS has been developed for exporting, binding, and FEM meshing with minimal effort from the user. The interface is based on the automatic generation of ANSYS' macros. A macro is a sequence of ANSYS-commands written to file for multiple reuse.

The geometric complexity of microcircuits on the solder balls also greatly complicates the process of finite element modeling. This complexity is expressed primarily in a plurality of smaller elements (e.g., parts of solder balls), thin layers (copper layer on the surface of the board). The number of finite elements required to cover the above-mentioned geometric nonlinearities may be too large. Typically, modeling always involves some simplification of the design. In this book, simplifying circuit design is based on the following considerations:

– the model is symmetrical (two planes of symmetry or quarter-symmetry). Therefore, it is enough to simulate only a fourth of the chip. It is important to set the correct endurance conditions, limiting the degree of freedom in the plane of symmetry;

– a layered structure of the substrate chip (package substrate), and PCB (printed circuit board) can be modeled by a material with average physical properties. This should not significantly affect the accuracy of the results, since the principal stress concentrators, as well as the most vulnerable parts from the point of view of fatigue are solder balls;

– The solder balls on the FE model should maintain their geometry making simplifications inappropriate.

Material properties settings.

The model of the circuit board with solder balls consists of various parts (solder balls, silicon chip, substrate chip, board, etc.) with different physical properties. Some parts (e.g., the substrate) may have orthotropic properties. ANSYS allows all this into account, however, the lack of the database with values of different materials complicates its use.

Loads and endurance conditions settings.

The harmonic vibration and shock are given by the inertia (Inertia Loads) in ANSYS.

For example, harmonic vibration is related to acceleration of resonant frequency. Each frequency and acceleration requires the use of harmonic analysis (Harmonic Response Analysis in terms ANSYS). The results are then combined for all frequencies in the ANSYS postprocessor. It is important that the frequency spectrum of harmonic vibration is wide enough to avoid missing the resonant frequency of the structure.

Shock load is modeled similarly, with the difference that instead of using dependent frequency we use transient excitation.

Carrying out the calculation.

Calculations are also automated with macros. It is important to choose the right type of calculation and its parameters.

To carry out the calculation for the harmonic vibration we use Harmonic Response Analysis.

Implemented in ANSYS, analysis for random vibration (so-called PSD analysis) outputs standard deviations (1 σ values) of displacements, accelerations, stresses and strains in all nodes of FE mesh. Post-processor (Time-History Postprocessor) system of ANSYS can also calculate the spectral density of the output variables (displacements, accelerations, stresses and strains).

Dynamic analysis is used to calculate impact.

1.3. Models for the analysis of time to fatigue failure in microcircuits with transient thermal effects

Ambient temperature changes, transient heat power sources, and the difference in thermal expansion coefficient in different elements of circuits cause a large deformation and high stresses in solder balls. Cyclical repetition of the process leads to accumulation of fatigue damage and eventually to fatigue failure in solder balls.

Thermal cycles (i.e. power cycles or temperature of the environment) forming loading history in arbitrary nodes of FE mesh, which may have a complex shape, in other words, will differ from cyclic loading. Typically, fatigue failure occurs in a limited number of thermal cycles , within which there are significant jumps in stresses and strains , as well as plastic deformation. The most suitable model for fatigue analysis here is low-cycle model.

The algorithm for calculating fatigue with transient thermal effects will be similar to the algorithm with multiple collisions and minor changes:

- after transient thermal analysis in ANSYS, equivalent deformations are obtained depending on the nodes of the finite element mesh of time, forming the loading history in these nodes. Next, the calculation is made according to the algorithm for multiple shocks up to section 6;

- after the summation of fatigue damage by the Palmgren-Miner rule, we count the number of thermal cycles to failure. Knowing the number of thermal cycles, which formed a loading history - NC (this parameter should be entered by the user of the algorithm), and the fatigue damage accumulated during these cycles - D(NC), the number of thermal cycles to failure is calculated by forming NCF ratios:

$$\frac{D(NCF)}{D(NC)} = \frac{NCF}{NC}.$$

Given that fatigue failure is related to damage $D(NCF)=1$, we get:

$$NCF = \frac{NC}{D(NC)}.$$

2. AUTOMATED SUBSYSTEM ASONIKA-M-3D

ASONIKA-M-3D subsystem was developed for simulation of mechanical and thermal processes in arbitrary modelling of electronic equipment.

2.1 Subsystem Interface

The main window of the subsystem (Figure 2.1) consists of several parts: working area (1), toolbar (2), object tree (3), list of parts (4), and post-processor panel (5).

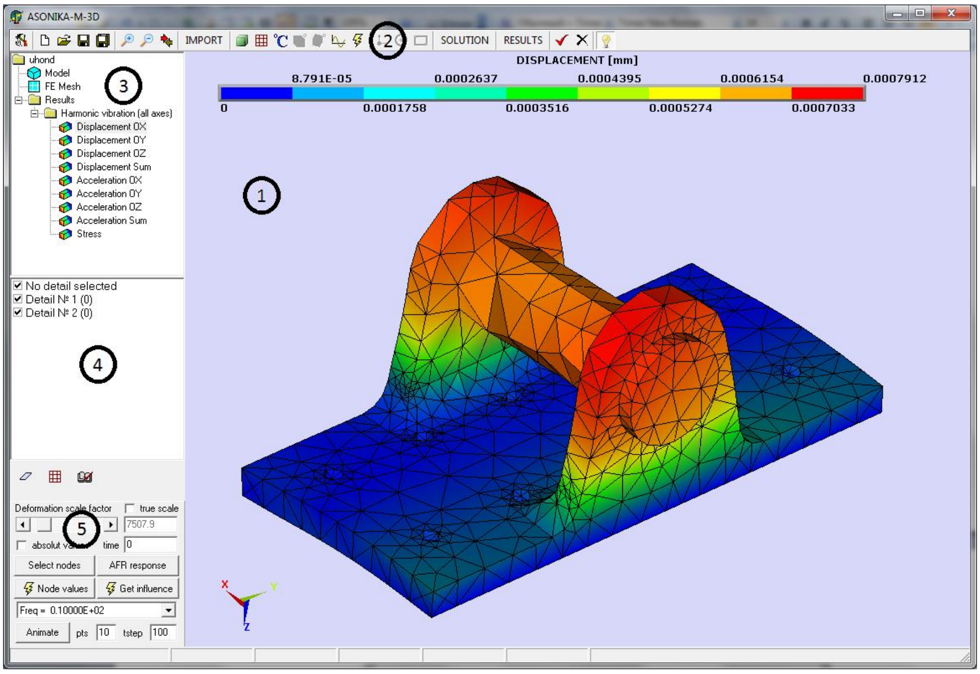


Figure 2.1 - Window subsystem during operation

The working area displays the geometry of the model. The working area is scaled either with the toolbar buttons  or the mouse wheel.

The rotation of the design is accomplished by moving the mouse cursor on the workspace and holding down CTRL + LMB (left mouse button).

To panning of the design is accomplished by moving the mouse cursor over the workspace and holding down CTRL + RMB (right mouse button).

The list of parts lists all parts which make up the design. After importing the mode, the names of parts on the list have the following format: "Item № N”, where

N is a serial number. Picking out a separate item in the list (the left mouse button or the "up" and "down" on the keyboard), it also stands out (highlighted in yellow frame) in the work area (Figure 2.2).

If the checkmark is removed next to the items, the item is no longer displayed in the work area (Figure 2.3). In this way the user can see an internal structure of the model.

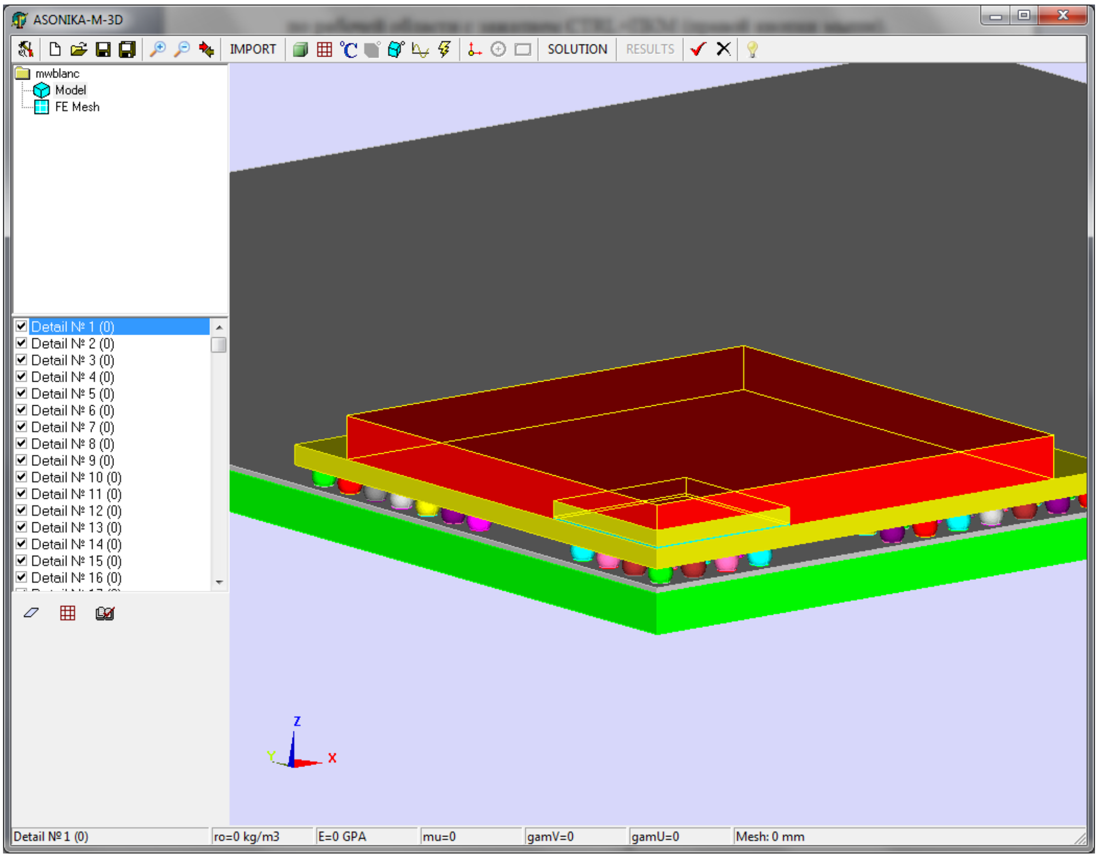


Figure 2.2 - Highlighting of parts in the list (highlighted part number 1)

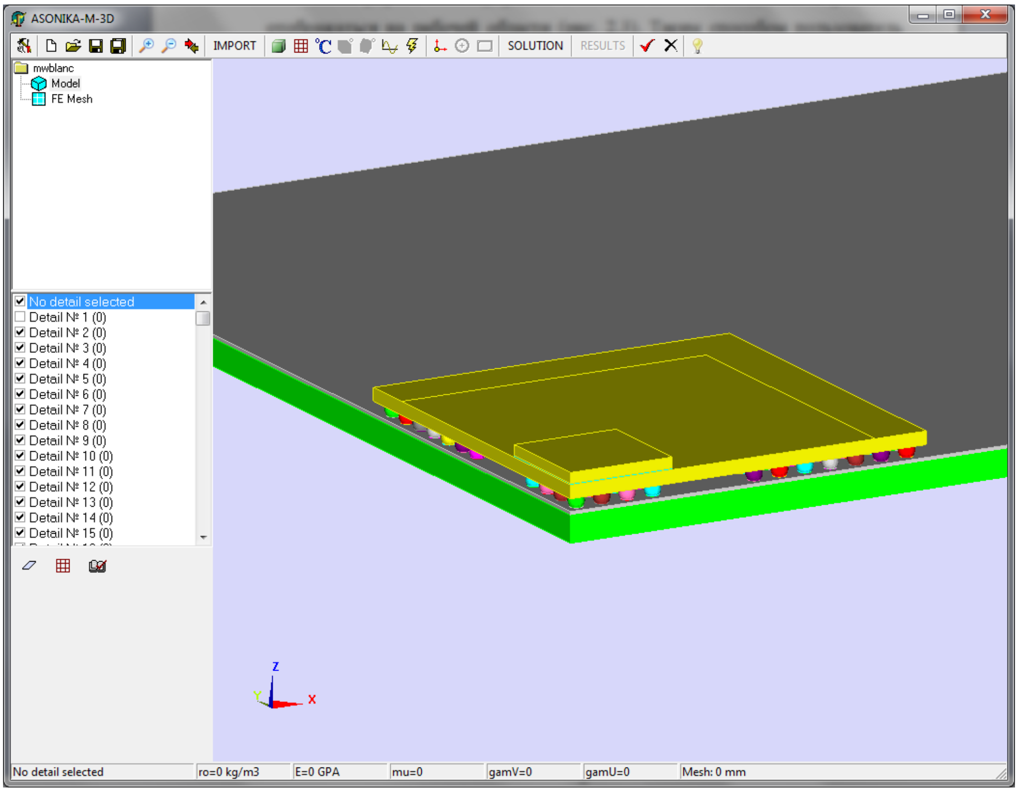


Figure 2.3 - Hiding of parts (hidden part number 1)

The geometry of the model displayed in the work area can be roughly represented as a hierarchical structure of "volume-surface-line-point." Every item is represented by a separate volume. Each volume includes a surface (flat or having a curvature) that it covers. The surfaces are formed by the lines (straight or curved). Each line has point with a beginning and an end.

Volumes (i.e. details) in the work area are displayed in different colors. When you move the mouse cursor on the workspace, the surface and the line below the cursor are automatically highlighted (blue border). Left-clicking on a surface or a line while holding the ALT key, highlights (highlighted in yellow frame) a detail, including a given surface or line. The item is also highlighted in the list.

The object tree, which has a hierarchical structure and is located to the left of the work area and top of the window allows you to select an object to be displayed in the workspace. In ASONIKA-M-3D, the display objects are of three types:

1. *Model* – a solid model design, which is imported into the subsystem.

2. *FE mesh* – finite element model is generated by means of the subsystem.

3. *Results* – these are stress fields, displacement, acceleration, temperature, and other variables, which are the results of analysis of structures on the thermal and mechanical effects . All results are grouped by the type of analysis, that is, for each analysis, there is a different set of results, contained under the tree node with the name of the analysis.

Postprocessor panel, located to the left of the work area at the bottom of the window, contains a set of commands that are used to manipulate the results of calculations: output of frequency response graph and Nyquist plot, selection of nodes, inspection of results for individual nodes, output of the absolute values of the results, animation of the results, etc.

2.2 Projects in ASONIKA-M-3D

Working with the program is carried out mainly by pressing command buttons on the toolbar at the top of the window.

The work with each model is done within a framework of a separate project, representing a single information block about geometry of the model, the finite element mesh, material properties, effects , etc. The project is created by the user before starting work or it is imported at any time by using the program.

Commands  in the toolbar (looking left to right) are used to create a new project, open an existing project, save the project, and save the project under a new name.

Figure 2.4 shows the main program window that appears when you run it.

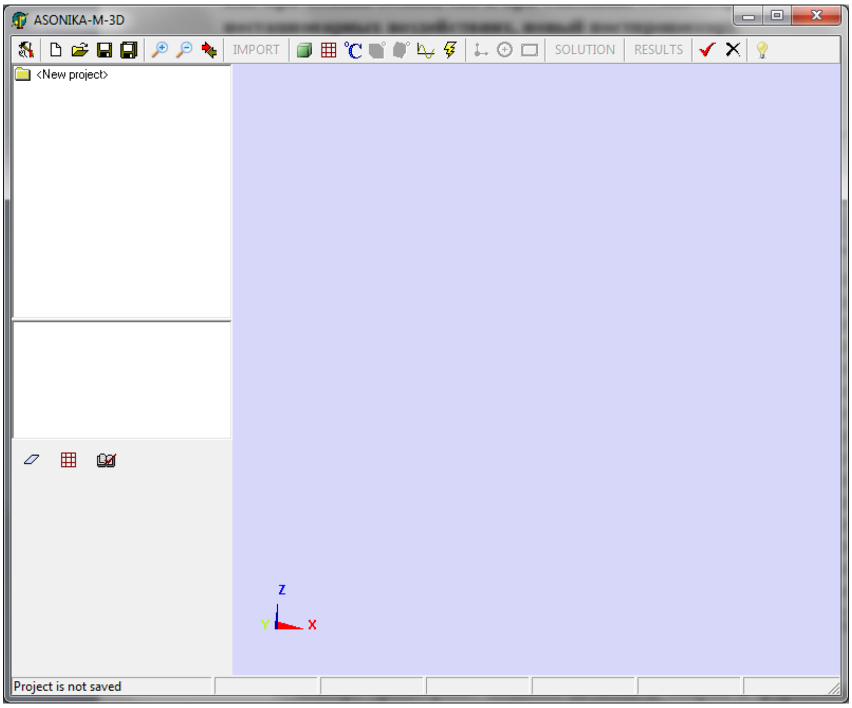


Figure 2.4 - Window subsystem after its launch

After starting the program, the user can:

– open an existing project by clicking the button and in the dialog box specify the path to the project file with the extension ". rea" (Figure 2.5);

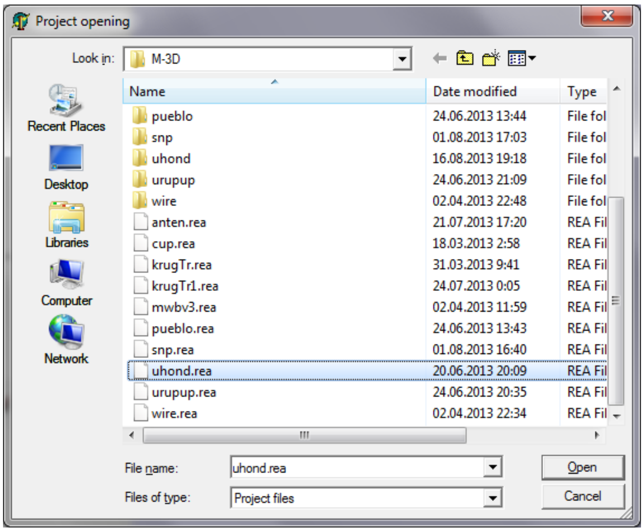


Figure 2.5 - Opening the project

– save an empty project by clicking the button and entering the name of the project in the dialog box (Figure 2.6).

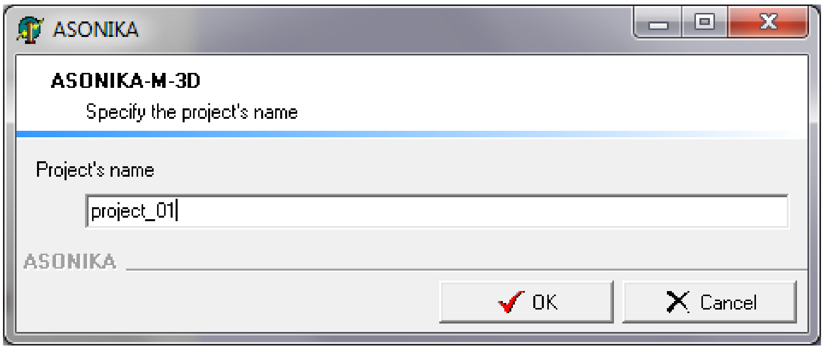


Figure 2.6 - Setting the name for the new project

If the project has already been created, in order to start a new one, you must press the button 🗋 to clear the memory and create a blank project. After this, the new project is saved, as previously described.

2.3 ASONIKA-M-3D Structure

The structure of the subsystem can be described conventionally by dividing it into 3 functional parts, typical for CAE engineering simulation systems:

– preprocessor,

– solver (or processor),

– post-processor.

2.3.1 ASONIKA-M-3D Preprocessor

The preprocessor is designed to prepare models for carrying out mechanical and thermal simulation.

The pre-processor addresses the following objectives:

– import three-dimensional models of parts and assemblies in the formats of IGES and SAT, created with different CAD-systems,

– setting materials,

– setting the task of mechanical impacts,

– setting endurance conditions,

– setting the task of thermal boundary conditions,

– setting parameters of the finite element partition,

– generating finite element mesh.

Let us examine each item in detail.

2.3.1.1 Importing three-dimensional models

Command  on the toolbar, a dialog box appears (Figure 2.7), which indicates the type of file and imported file model.

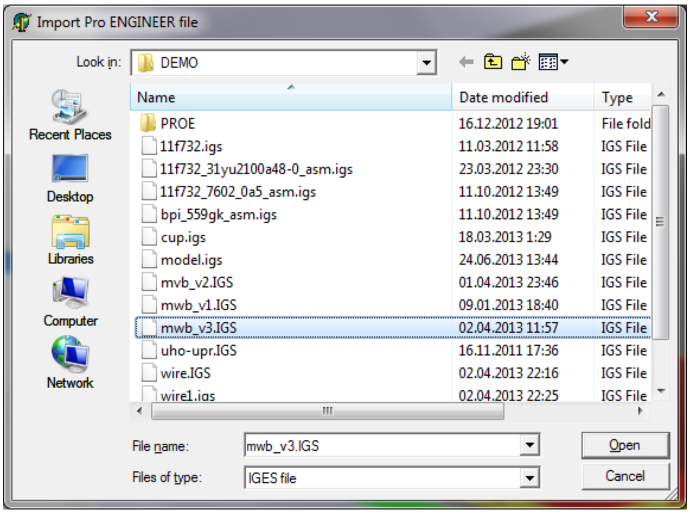


Figure 2.7 - Import model window

By pressing the "Open" button, the process of importing the model starts, at the completion of which, the model is displayed in the working area of the program (Figure 2.8).

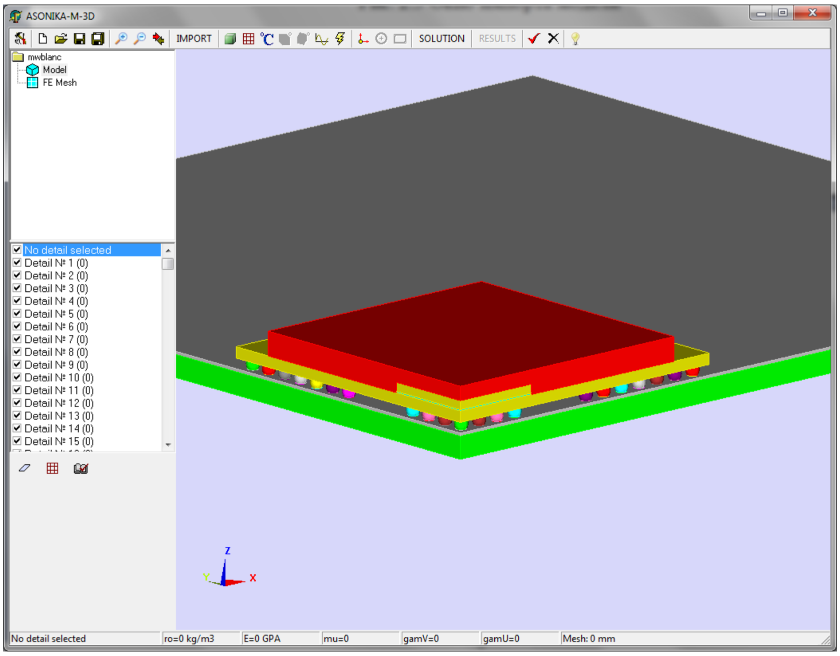


Figure 2.8 - The display of the imported model of the microcircuit

2.3.1.2 Specifying materials

The command  on the toolbar is used to set parameters of the material for the entire structure (Figures 2.9 and 2.10).

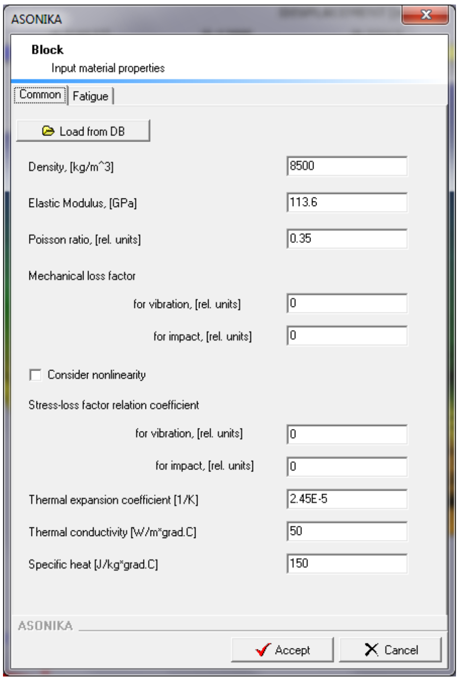


Figure 2.9 - Setting the parameters of the material (“General” tab)

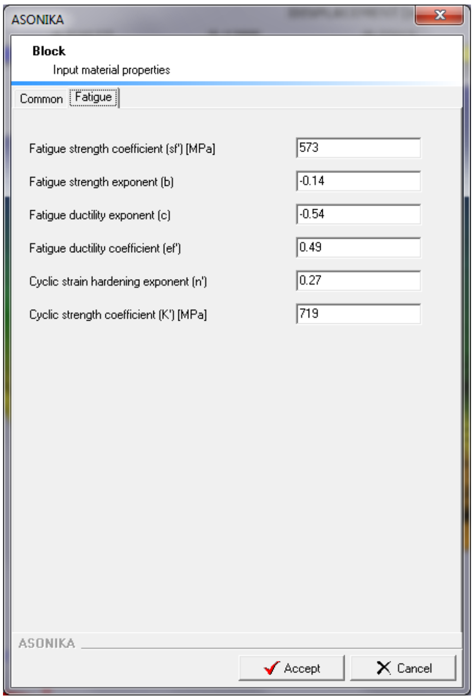


Figure 2.10 - Setting the parameters of the material ("Fatigue" tab)

The material parameters are grouped into two tabs: "General" tab, containing the basic thermal and mechanical properties, and the "Fatigue" tab containing fatigue parameters. It should be noted that each type of analysis requires a limited set of material parameters (e.g. vibration harmonic analysis requires setting only the Young's modulus , Poisson's ratio , density, and mechanical losses). This issue will be discussed later in a greater detail.

To select a material from the database, press the Загрузить из БД button (Figure 2.11).

The "nonlinearity" parameter allows you to take into account the dependence of the mechanical losses from stresses in dynamic analysis of structures.

Besides being able to set the parameters for the entire structure of the material, using  button on the toolbar, the program has the ability to set the parameters of the material for specific details. To do this, first select an item (by any method described in paragraph 2.1). The material parameters details are entered by pressing , button located in the area below the list of parts.

2.3.2 Solver

Solver (or processor) is used to carry out mechanical and thermal design calculations..

The calculation process may take quite a long time. Its duration depends on the complexity of the model and the power of the computer. A critical parameter which directly affects the calculation time is the density of the finite element mesh.

The calculation itself does not require any intervention by the user, who only needs to specify the type and parameters of the calculation before its execution by using SOLUTION command on the toolbar (Figure 2.21)

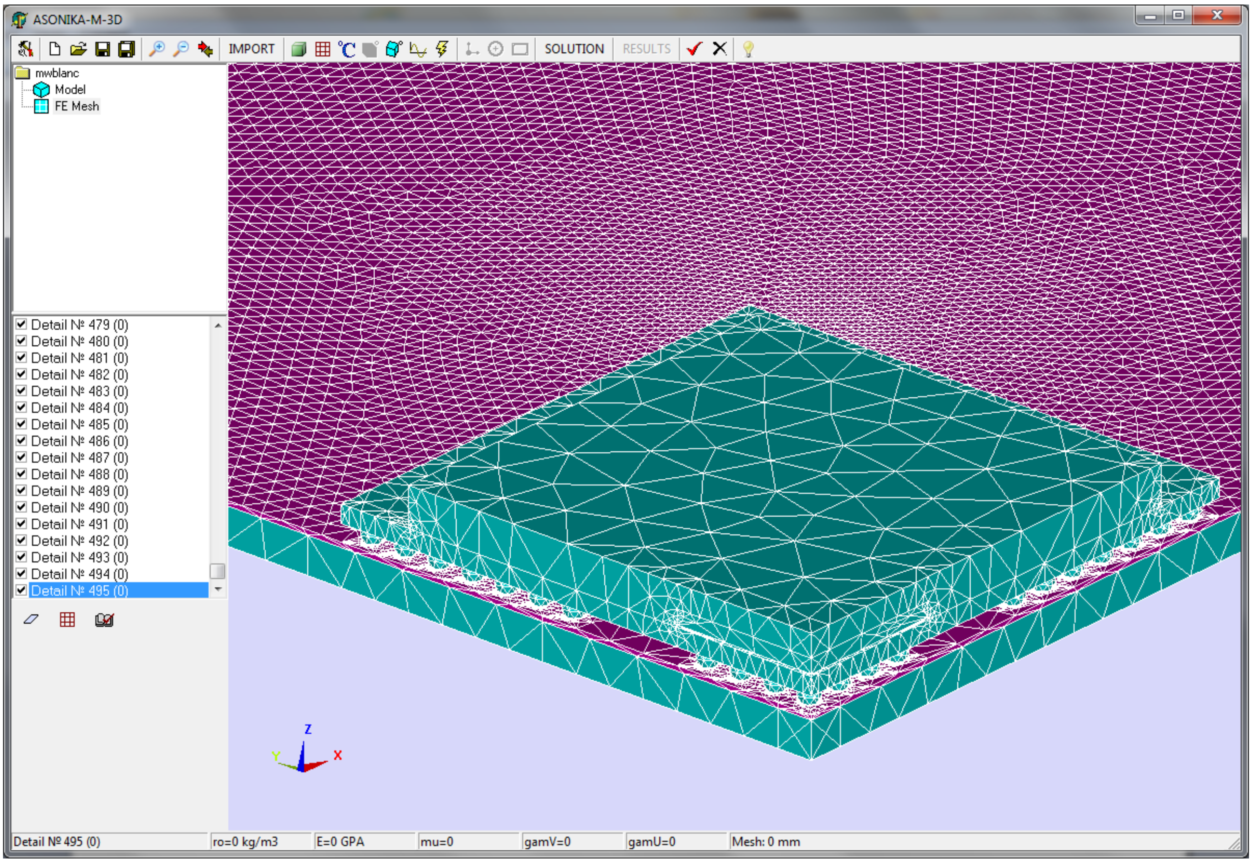


Figure 2.20 - FE mesh

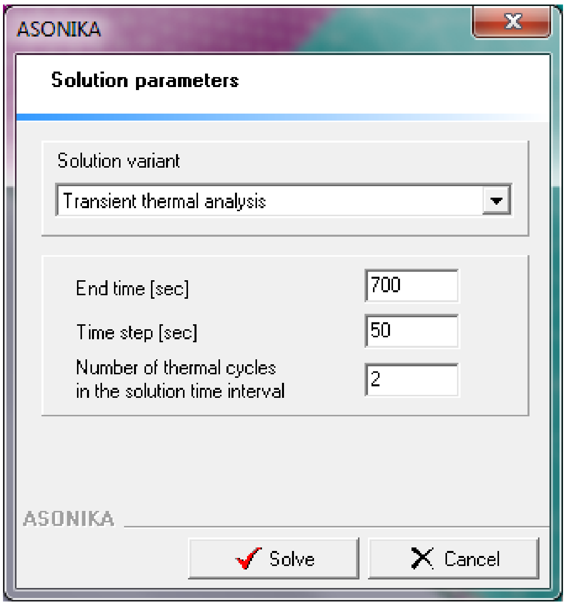


Figure 2.21 - Types and parameters of the calculations

In ASONIKA-M-3D, the following types of calculations are carried out:

- harmonic vibration

- random vibration

- single shock
- multiple shock
- linear acceleration
- thermal expansion
- steady-state thermal analysis
- transient thermal analysis.

2.3.3 Post-processor

Post-processor is designed to view the results of the mechanical and thermal calculations.

In the pre-processor, the following objectives are addressed:

- output fields of output values: accelerations , displacements, stresses, temperatures, time to fatigue, etc.;
- obtaining amplitude-frequency and time-frequency characteristics (frequency response graph and Nyquist plot) required for output values;
- display of absolute and relative values of accelerations and displacements (mechanical calculations only);
- obtaining required values of output variables in the individual nodes of the finite element mesh;
- view strained state of constructions taking into account the enhancement factor;
- animation design response to mechanical stress (for structural oscillations of different frequencies with harmonic vibrations, as well as shocks);
- Export of mechanical stress effects to external files.

2.3.3.1 Output fields of output values

In the object tree, the name of the analysis is selected, the results of which you want to display, as well as the name of the output variable. Field values of this quantity appear in the work area (Figure 2.22)

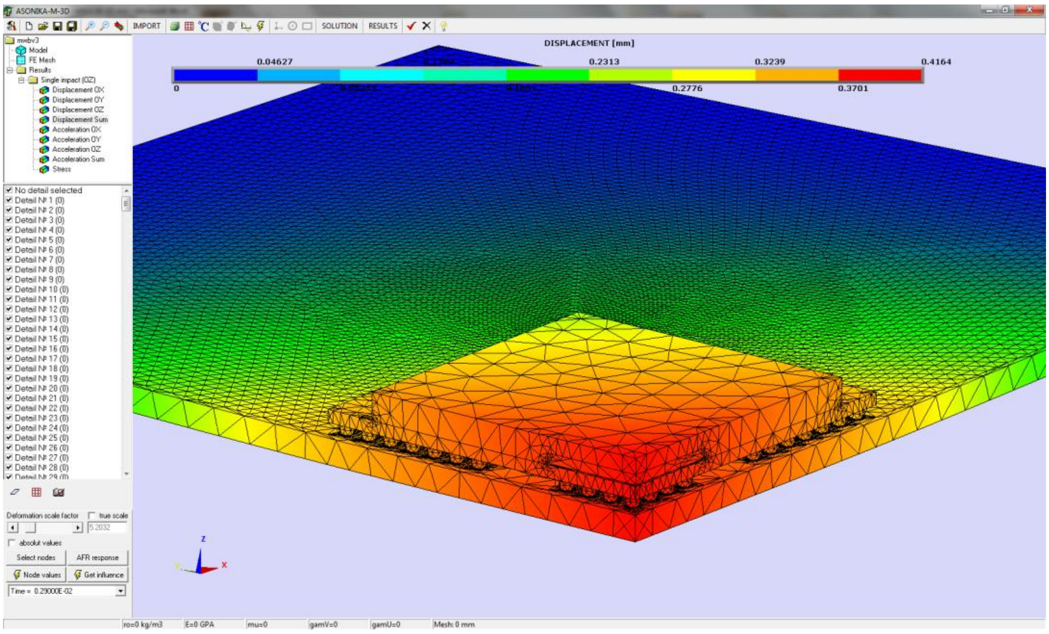


Figure 2.22 - Total displacement field

2.3.3.2 Preparation of output of frequency response graph and Nyquist plot

Using command [Select nodes] on the postprocessing panel, select a node (or several nodes) for which frequency or time dependence of output variable is needed (Figure 2.23).

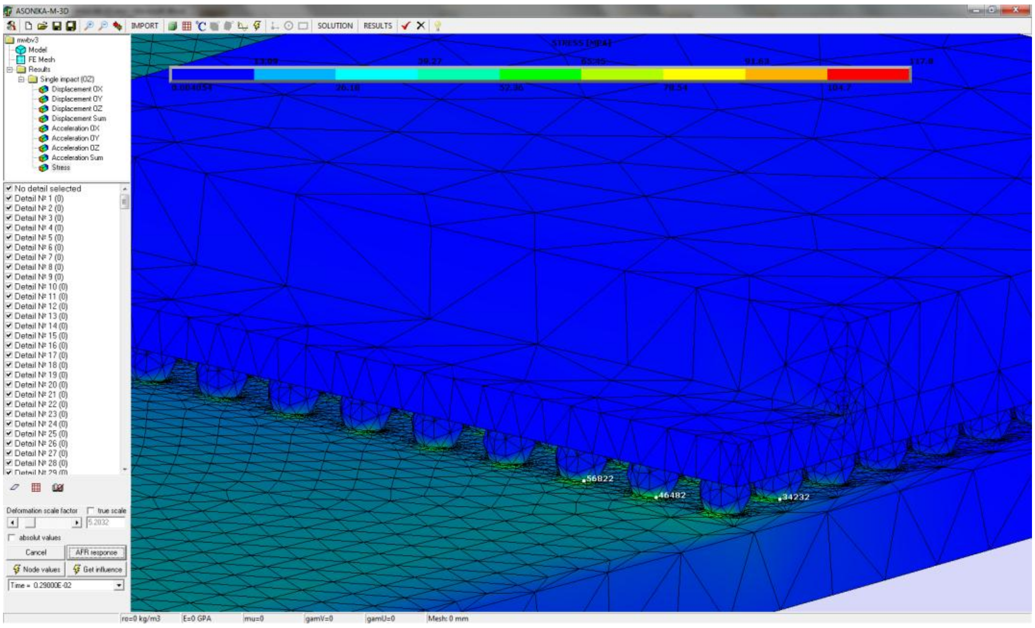


Figure 2.23 - Selection of 3 nodes

Next, with the command [AFR response] on the postprocessor panel, we get a display of corresponding graphs with the selected nodes (Figure 2.24).

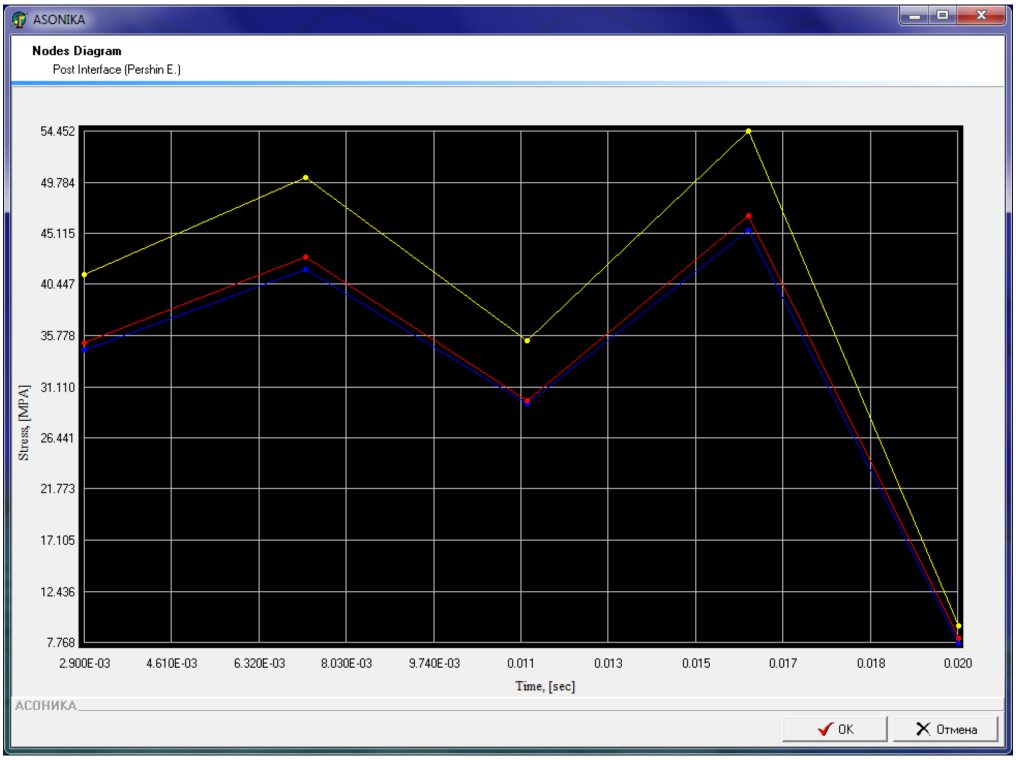


Figure 2.24 - Node charts

2.3.3.3 Output of absolute and relative values of the output variables

In ASONIKA-M-3D analysis on mechanical effects, and above all - displacements can be viewed in two ways: on the one hand - these are deflections of the structure, provided that the place of attachment is at rest (have zero displacement), on the other hand - these are deflections of structure points, provided that the deflections at fixities coincide with the given effects (not zero). Acceleration may be viewed similarly. In the terminology of ASONIKA-M-3D, displacement and acceleration with respect to fixities are called relative (displacements and accelerations), and the same values with the reference variable called absolute (displacements and accelerations).

Switching between absolute and relative values is accomplished by setting the appropriate checkboxes on the panel postprocessor.

3. EXAMPLE OF AN INTEGRATED CIRCUIT SIMULATION In ASONIKA-M-3D

This section describes an example of the analysis of the mechanical and thermal processes of a microcircuit using ASONIKA-M-3D. The model contains a chip with the solder balls located on a circuit board.

Since the circuit board is symmetrical (with two planes of symmetry), then the analysis is taken only a quarter of the model.

Complex modeling includes the analysis on vibration and shock, as well as steady-state and transient thermal calculations. In ASONIKA-M-3D, there is no need to create a separate project for various types of analysis, it is enough to set all the necessary data as a single project, generate finite element model, and then run calculation processes for all types of effects.

Solid model of the microcircuit set up a third-party CAD-system and the process of geometric modeling is not considered here.

Structurally solid model (Figure 3.1) consists of the following parts:

1 - solder balls,

2 - silicon chip,

3 - endurance chip,

4 - epoxy compound,

5 - substrate chip,

6 - printed circuit board,

7 - copper layer on board’s surface.

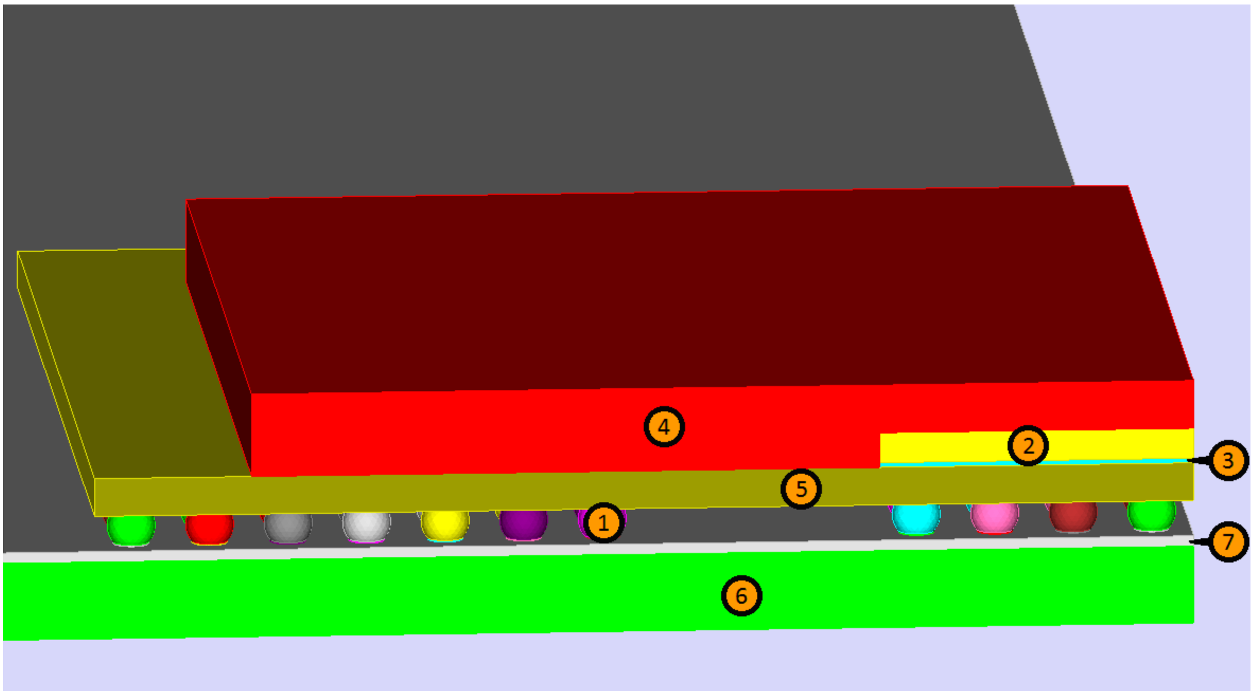


Figure 3.1 - Structure of the solid model

From a physical point of view, analysis of vibration and shocks, considered in this section is as follows:

- printed circuit board is fixed on the edges;
- mechanical effects occur at endurance points (harmonic vibration, random vibration, or shock impulse);
- due to structural oscillations, deflections, and resonances in some places there are high stresses and accelerations, which are to be determined in the analysis;

Thermal analysis (steady-state and transient) can be described as follows:

- The silicon chip, as a source of thermal power (constant power for steady-state and variable for transient analysis) generates heat;
- Through the heat-removing elements of the integrated circuits (chip mounting, substrate) and solder balls, the heat is transferred to the surface of board’s copper layer, which also has a high thermal conductivity;

– With external surfaces of various structural components (an epoxy compound, the board, the copper layer, etc.) the heat transferred by convection and radiation is given to the external environment;

– Temperature distribution and thermal stresses induced by the difference in thermal expansion coefficients, are values to be calculated;

– Cyclic variation of power (thermal cycles) causes an accumulation of fatigue damage in vulnerable areas of the structure (usually a solder ball). The number of thermal cycles to fatigue failure is one of the most important values calculated during transient thermal analysis.

3.1.3 Specifying materials

Select the item that represents the chip substrate (Figure 3.13) and set material parameters (Figure 3.14).

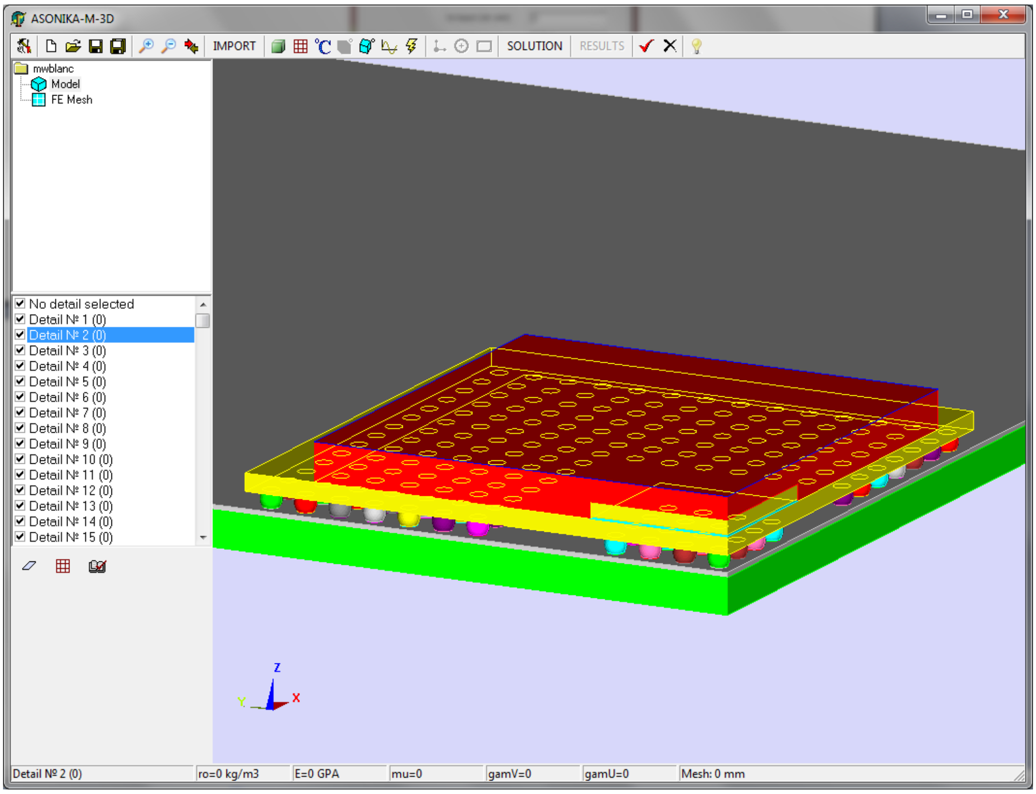


Figure 3.13 - Selected part (chip substrate)

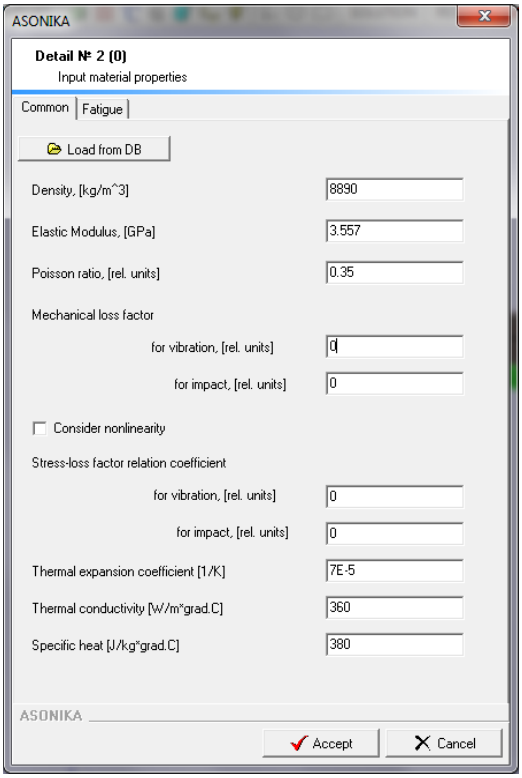


Figure 3.14 - Material parameters for the chip substrate

Select the item that represents the printed circuit board (Figure 3.15) and set material parameters (Figure 3.16).

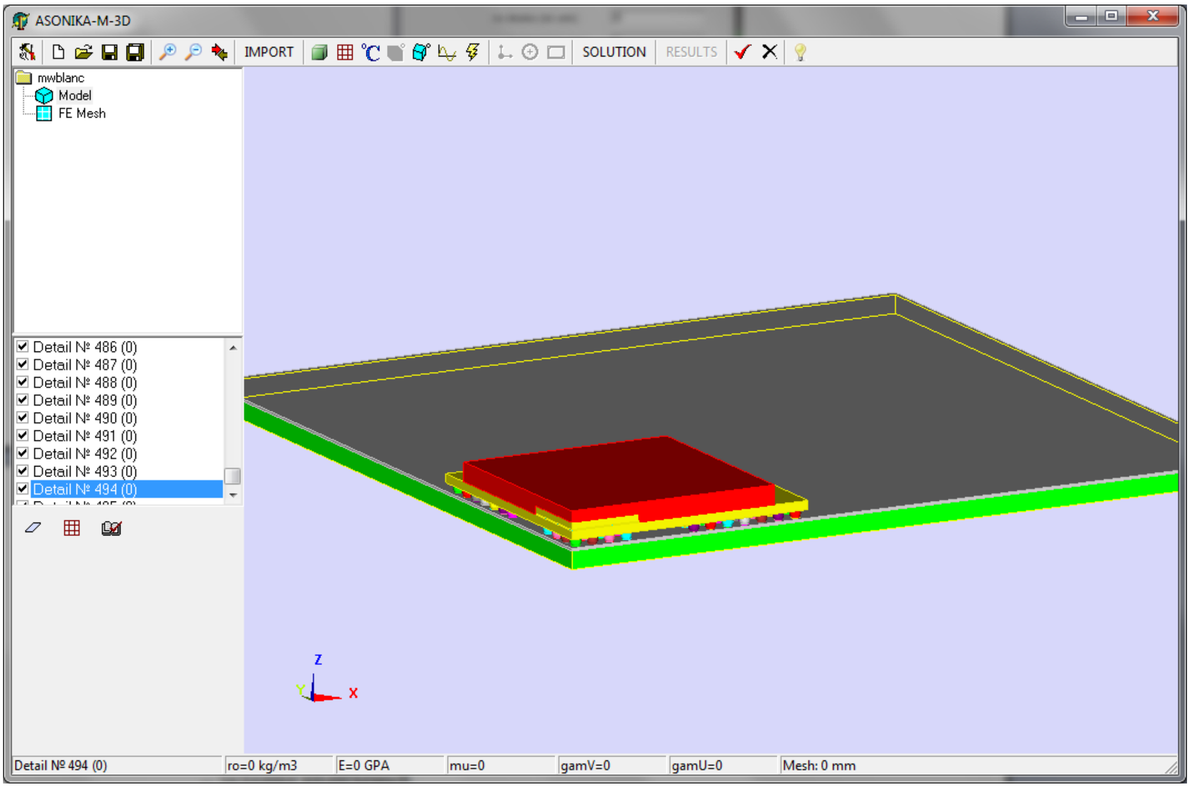


Figure 3.15 - Selected part (printed circuit board)

3.1.5 Setting endurance conditions

Constrain the circuit board on the edges. We select line attachment as a fastening type. Select the desired line by positioning the cursor and pressing Shift + LMB (Figure 3.22).

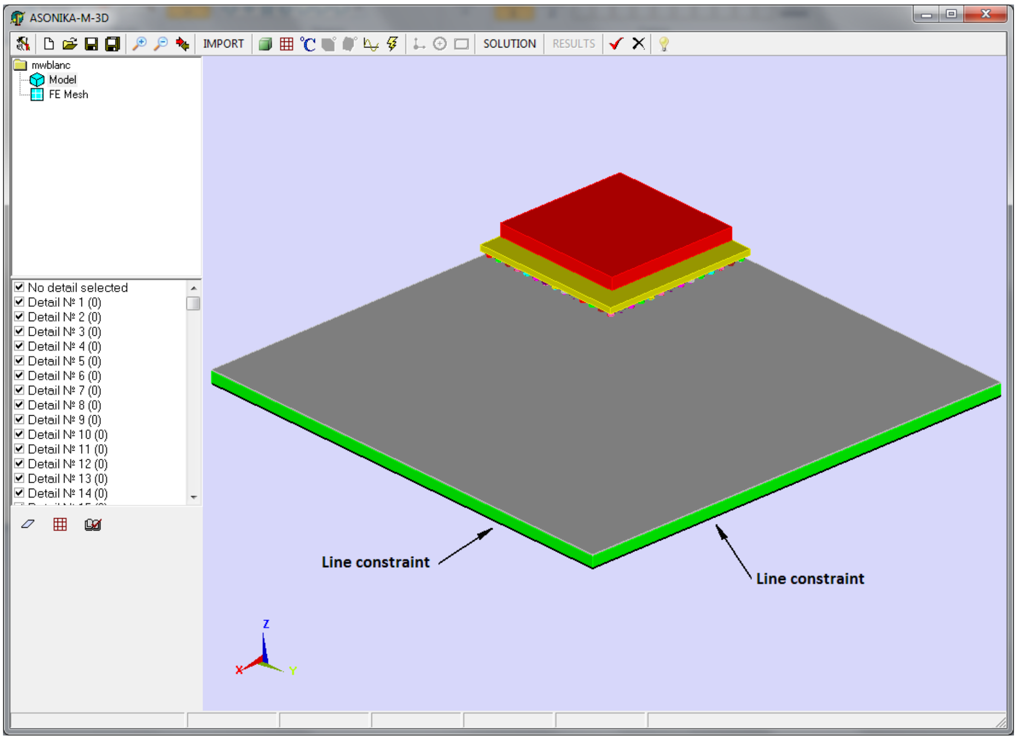


Figure 3.22 - Constraints

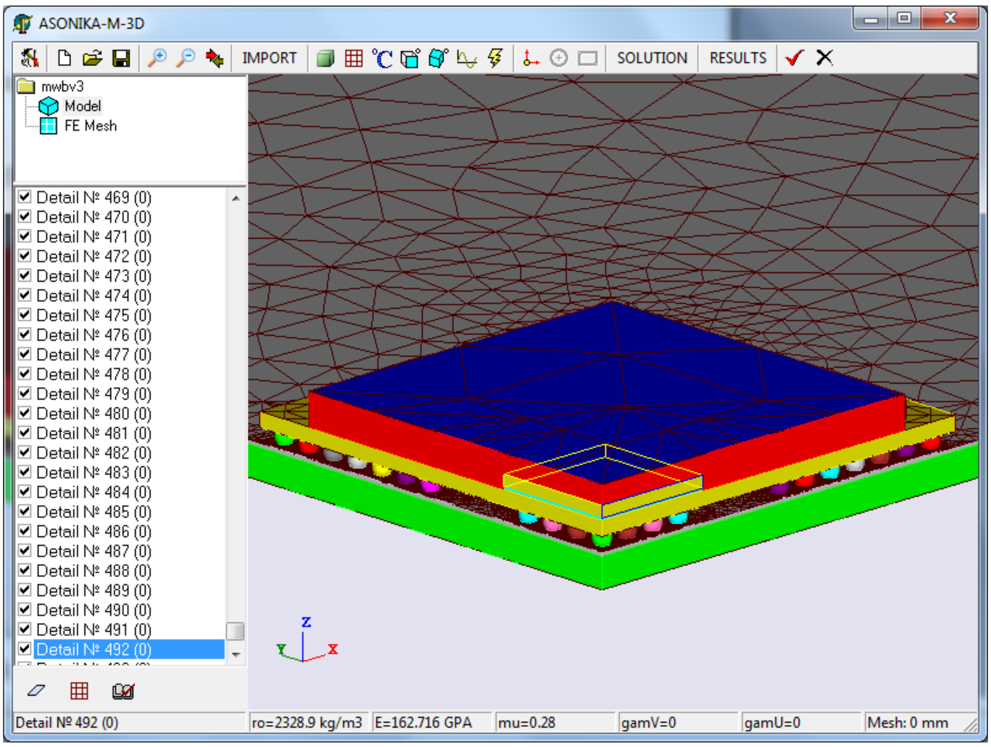


Figure 3.25 - Selected part

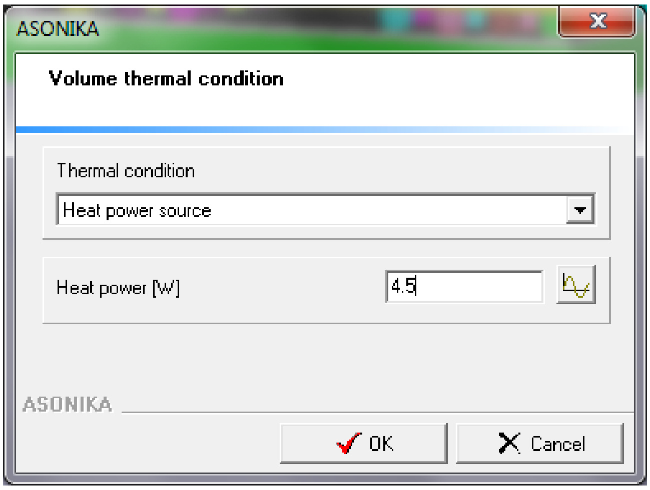


Figure 3.26 - Thermal boundary condition parameters

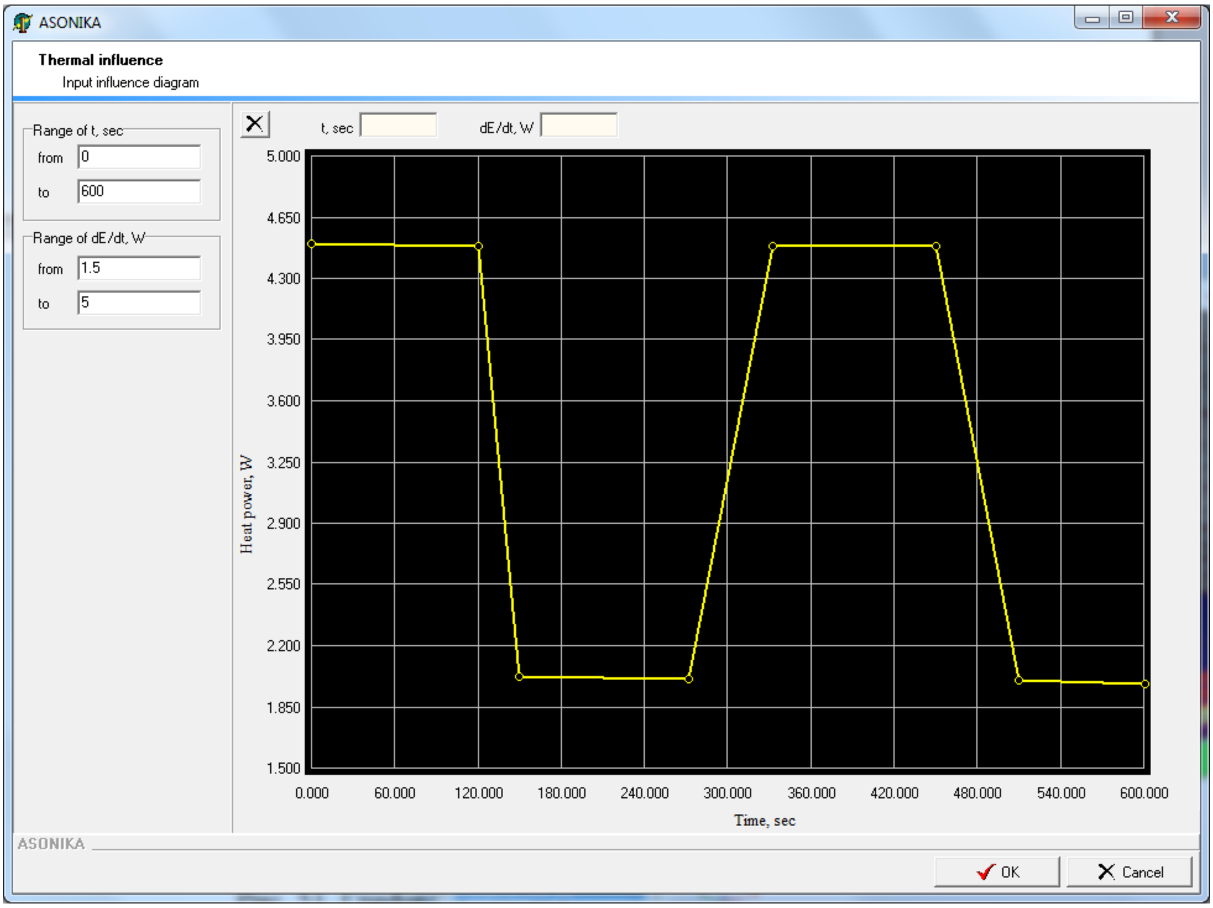


Figure 3.27 - Power heat profile

The graph in Figure 3.27 simulates two thermal cycles. In other words, the power of heat varies cyclically, and transient thermal analysis will be performed on a time interval equal to 600 seconds, and includes two cycles of change in capacity.

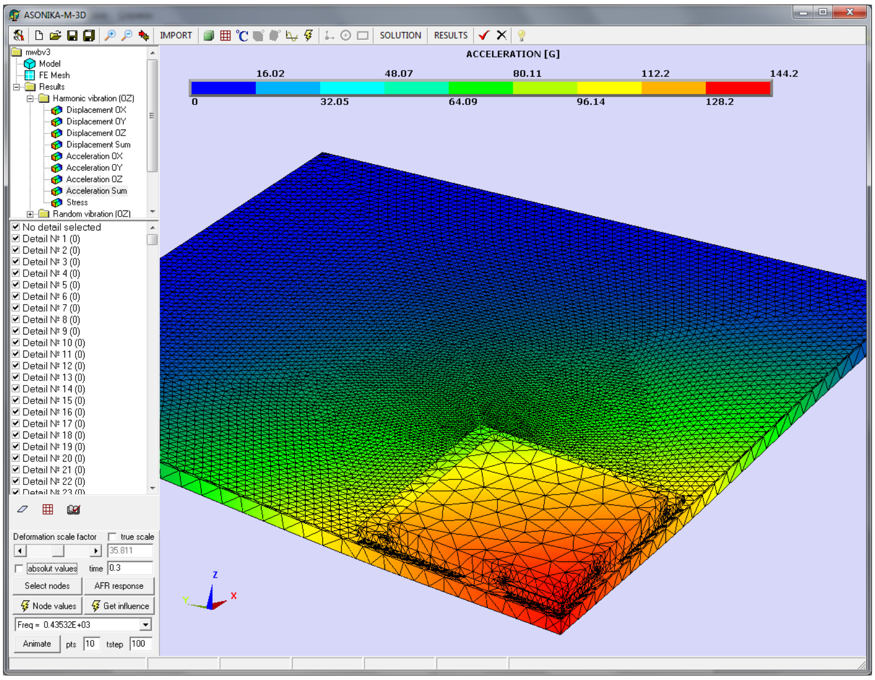


Figure 3.37 - Field of the total acceleration

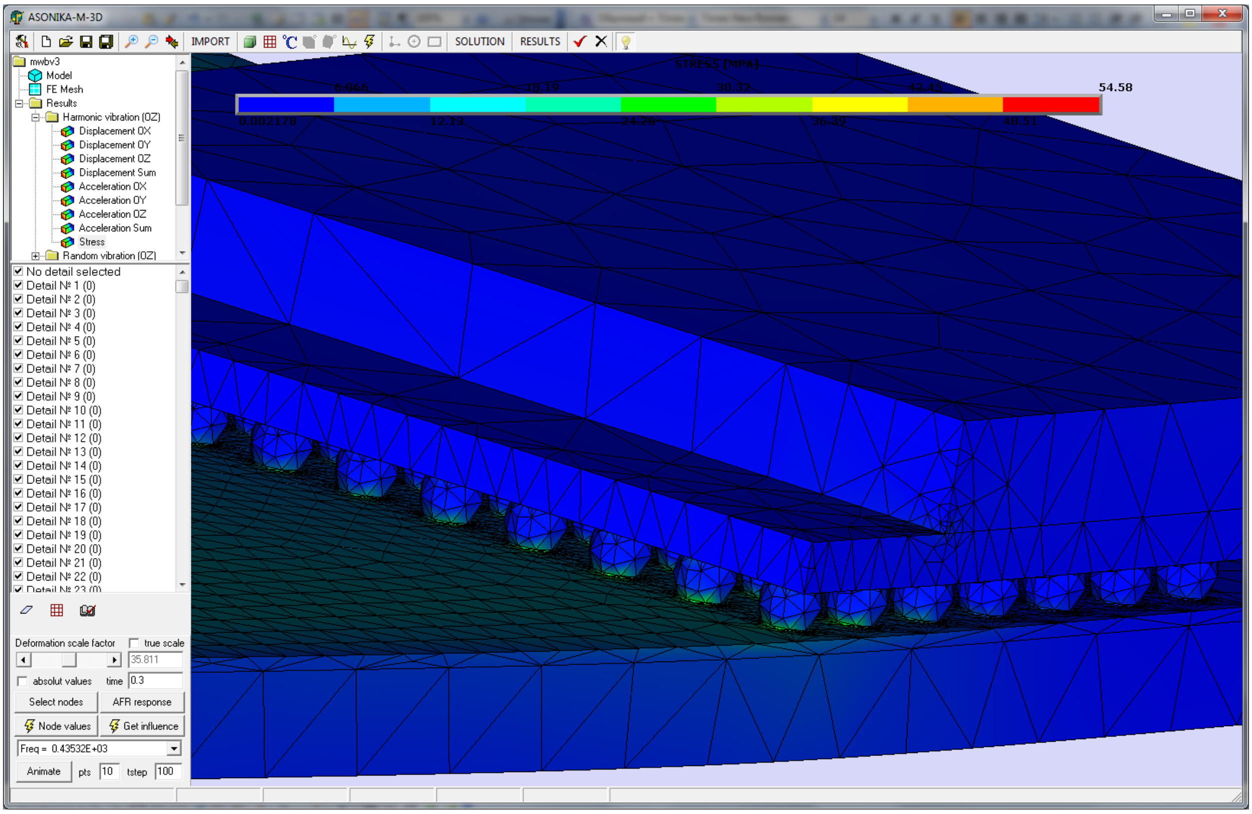


Figure 3.38 - Stress field

As an example, we select two nodes (figure 3.39) and obtain graphs of stress versus frequency in these sites (Figure 3.40).

3.3.2 Viewing the results of random vibration analysis

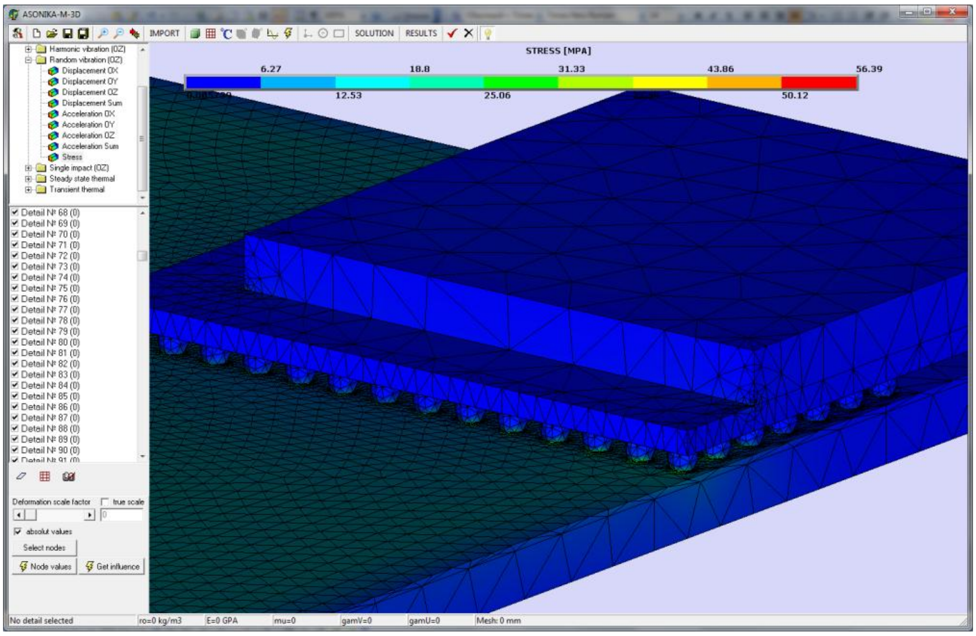


Figure 3.43 - Total stress field

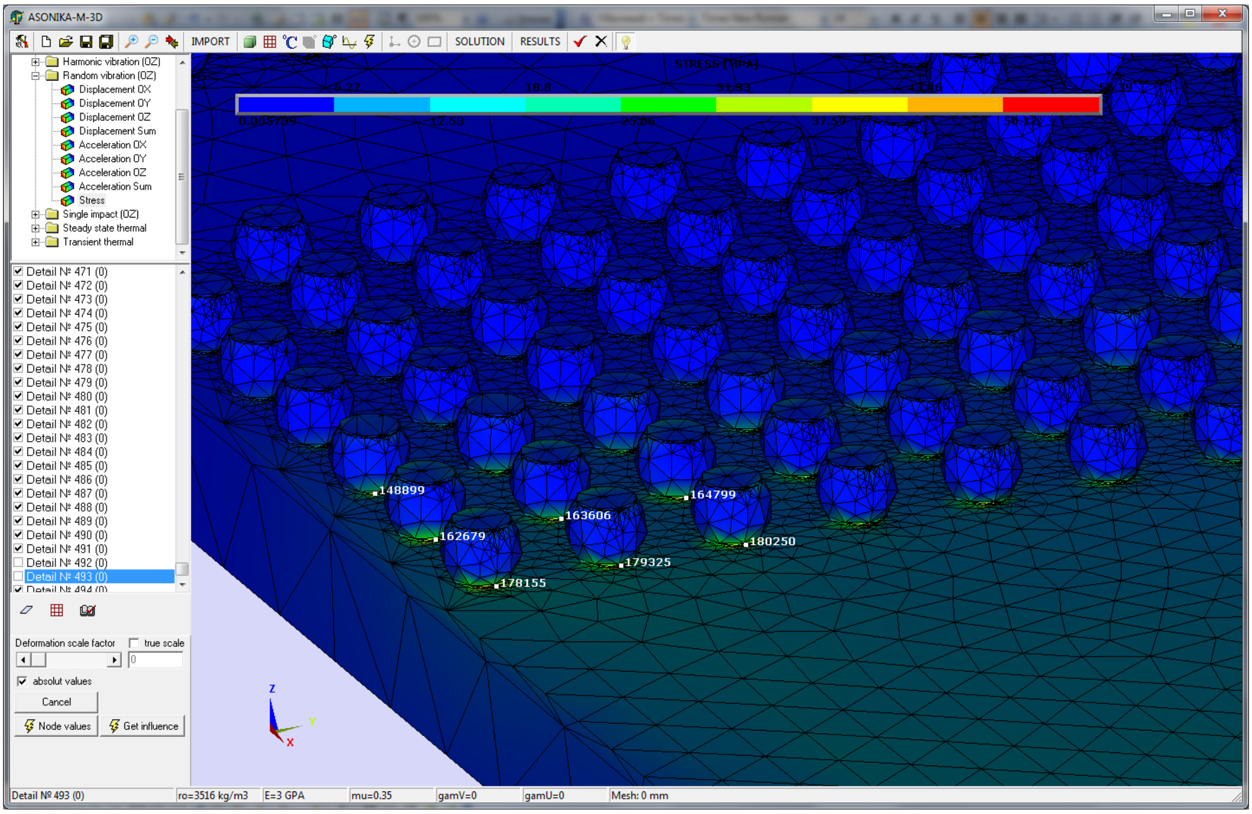


Figure 3.44 - Selection of nodes

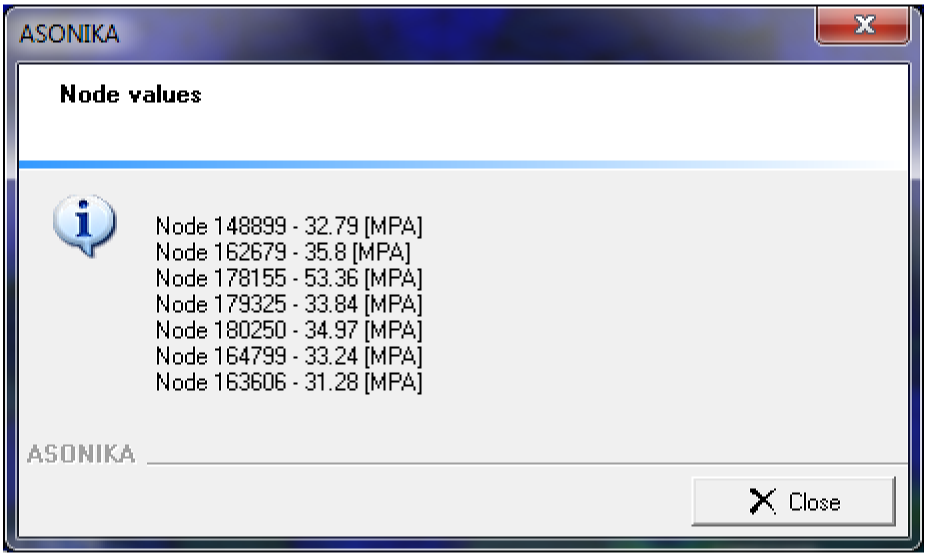


Figure 3.45 - Values of node stresses

3.3.4 Viewing the results of steady-state thermal analysis

The temperature field is shown in Figure 3.51. As expected, the temperature reaches its maximum (about 90 degrees Celsius) in the parts of the structure close to a source of thermal power - silicon chip.

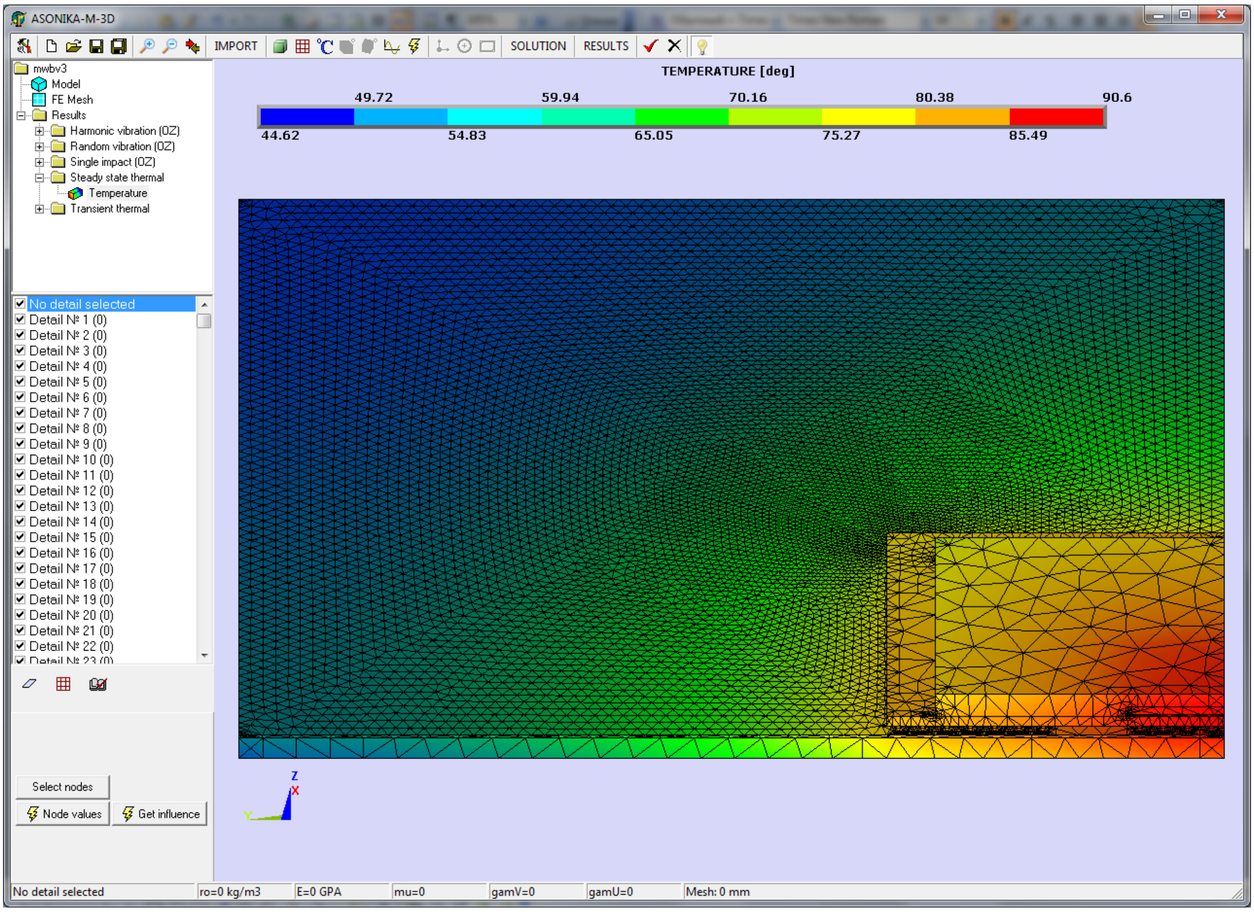


Figure 3.51 - Temperature field

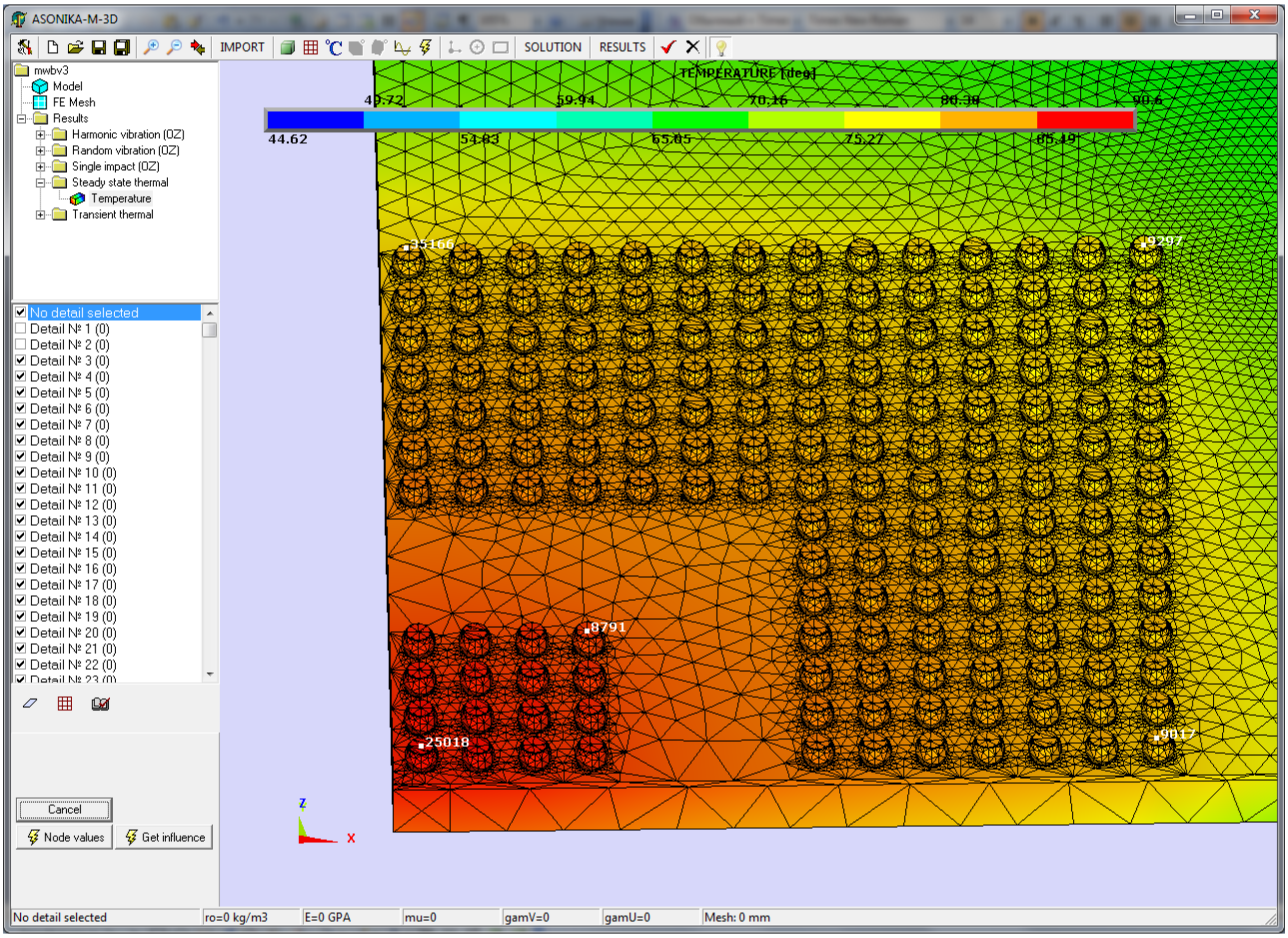


Figure 3.52 - Temperature field in solder balls

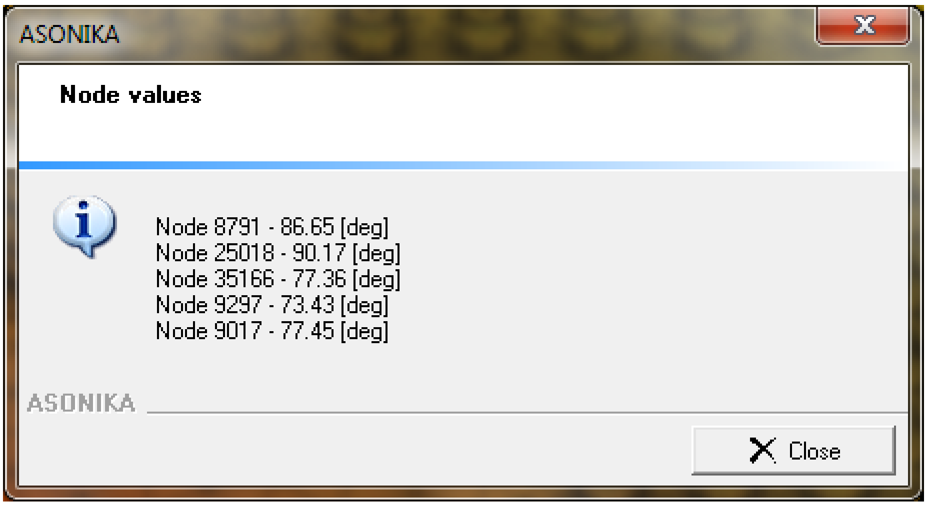


Figure 3.53 - Temperatures in the selected nodes

3.3.5 Viewing the results of transient thermal analysis

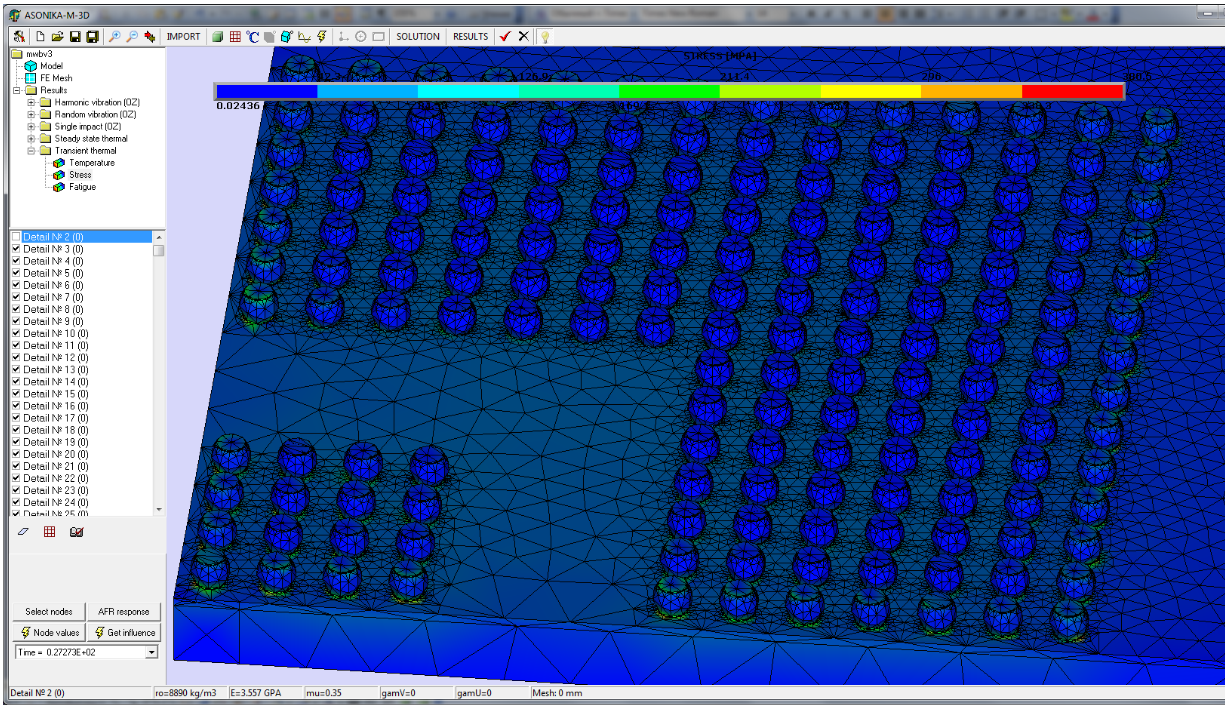


Figure 3.55 - Thermal stress field (some parts of the structure are hidden)

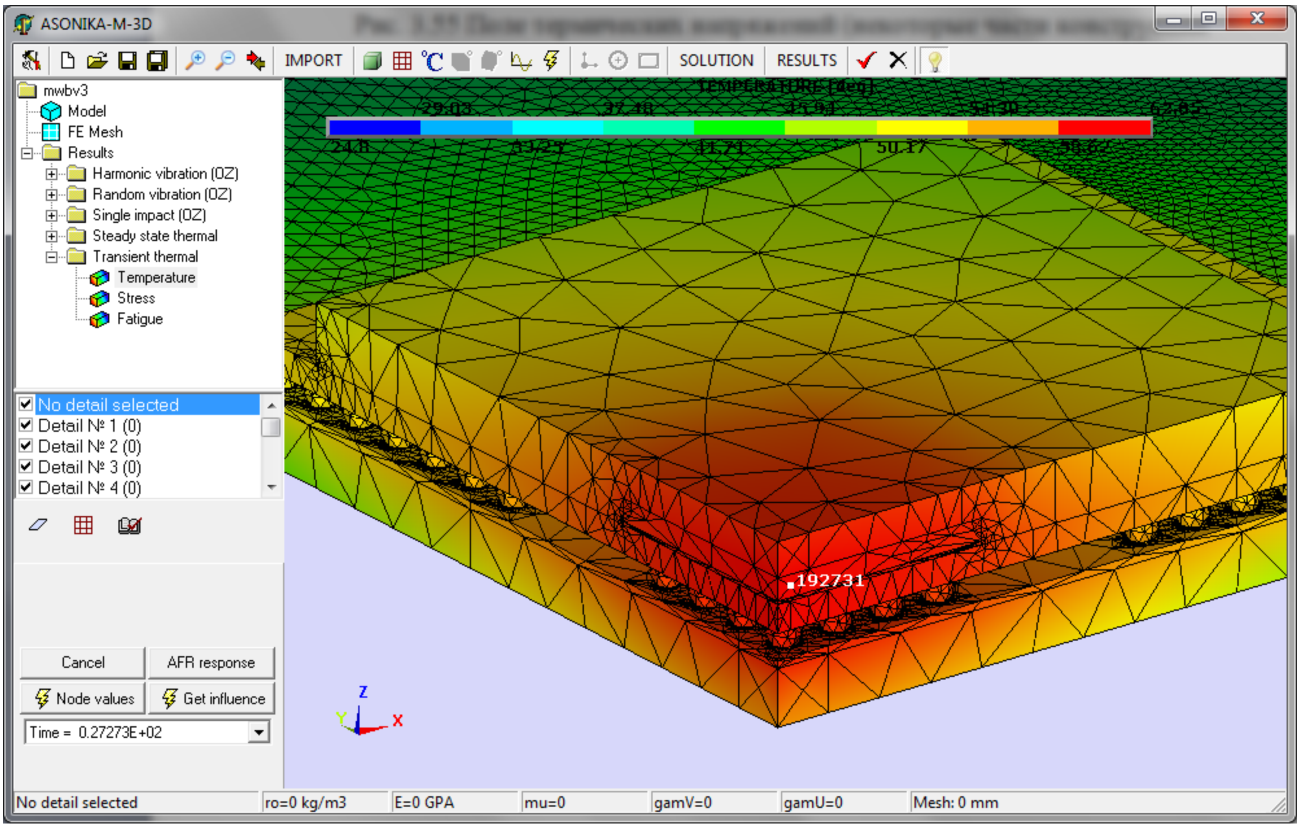


Figure 3.56 - Selected node

CONCLUSION

Currently, increasing demand on developing techniques, necessitates the use of virtual design in development of electronics. The principal of such requirements include: performance, dimensions, weight, and reliability.

During the field test of a prototype device, it is difficult to take into account many factors that are directly related to and influence each other. This, ultimately, will lead either to excessive factors of safety, or to failures in real operating conditions.

Under the guidance of the author of the book, professor Dr. Alexander Shalumov, the automated system, ASONIKA, is developed - designed to ensure reliability and quality of equipment. Virtual design, in ASONIKA system, takes into account the relationship of thermal, mechanical, and electrical factors in calculating both steady-state and transient regimes. There is a possibility, even before the prototype, to avoid possible failures and faults of the developed product by choosing the best option structure. This increases reliability of the developed equipment, as well as decreases time and expense of the project, which is also important in the growing competitive environment.

A virtual project represents a series of interlocking of design, technological, operational, and other documents with the virtual mockup of the product.

Virtual mockup is a structure, in a certain way related with complex patterns of electrical, thermal, and mechanical processes with the results of the analysis. Interaction with the virtual mockup is done by methods specified in CALS-technologies, through electronic document subsystem (Product Data Manager - using model properties with PDM-system) ASONIKA-UM.

Practice has shown that virtual design system based on ASONIKA reduces preparation time developed for production of electronics, minimizes the number of changes made to electronics, and increases reliability.

Designed virtual project provides a link of physical processes during their complex modeling, realization of a single information space, and support for subsequent phases of the product life cycle (production, service). There is an opportunity at any time to view, edit, and make new desired document in the project, which greatly simplifies subsequent modifications.

ASONIKA system is widely used for consulting, allowing one to give a qualified opinion on the conformity of electrical, thermal, mechanical, electromagnetic, and other device requirement characteristics.

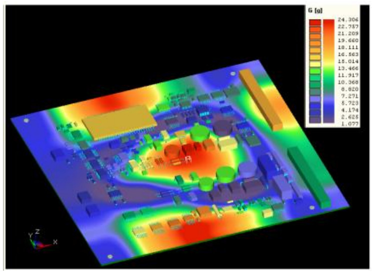
APPENDIX

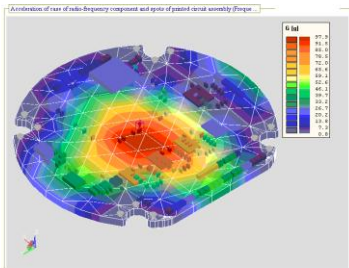
A S O N I K A

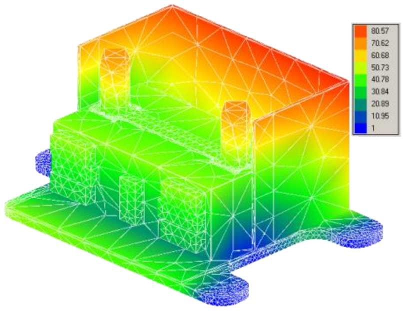
An automated system to ensure reliability and quality of the device, ASONIKA, is designed to simulate electronics and chips for complex thermal, mechanical, electromagnetic effects, and workflow automation in design; and can be used in enterprises that are developing devices which are installed on mobile objects.

ASONIKA is used as a replacement for testing electronics and chips with computer simulation of mechanical, thermal, electromagnetic, and other impacts before manufacture.

This represents significant cost savings and time reduction in terms of creation of electronics and chips while increasing the quality and reliability by reducing the number of tests.







ASONIKA system has been used for more than 30 years, by Russian companies, including for aerospace and aviation devices.

ASONIKA system consists of 13 sub-systems:

- *ASONIKA-T - modeling of thermal characteristics with arbitrary design in electronics*
- *ASONIKA-M - modeling of typical structural blocks with mechanical effects design in electronics*
- *ASONIKA-M-CABINET - modeling of typical cabinets and racks with mechanical effects design in electronics*
- *ASONIKA-M-3D - modeling of arbitrary structures of electronics and chips, created with ProEngineer, SolidWorks and other CAD-systems with formats of IGES and SAT, with*

mechanical and heat transfer designs, including cyclic effect and fatigue strength analysis

- *ASONIKA-ID - identification of physical and mechanical parameters of models in electronics*
- *ASONIKA-V - modeling of mechanical characteristics of electronic structures with vibration isolators*
- *ASONIKA TM - modeling of printed circuit assemblies with thermal and mechanical effects design in electronics*
- *ASONIKA-P - automated filling cards of electronic components operating modes*
- *ASONIKA-B - Analysis of reliability of electronics based on real modes of electronic components*
- *ASONIKA-UST - analysis of fatigue design of printed circuit boards and electronic components under mechanical effects*
- *ASONIKA-EMC - modeling of electromagnetic compatibility in electronics*
- *ASONIKA-DB - electronic components and materials reference database on the geometrical, physical, mechanical, thermal, electrical, and electromagnetic reliability parameters*
- *ASONIKA-UM - modeling management of the design in electronics*

ASONIKA system includes the following **converters with known CAD**:

- system integration module of PCAD, Mentor Graphics, Altium Designere, and OrCAD printed circuit assembly;

- 3-D model integration module created by CAD systems such as ProEngineer, SolidWorks, Inventor and other IGES and SAT file formats.

Proposed:

1. The introduction of ASONIKA to enterprises of electronic industry and institutions of higher education.

2. Providing consulting services to enterprises of electronic industry for modeling electronics and chips of mechanical, thermal, electromagnetic, and other effects with the help of ASONIKA.

3. Organizing training of professionals working with the system ASONIKA.

Prof, Dr. Alexander Shalumov is the Director General of the research institute "ASONIKA" (Russia) and the Chair of the Information Technologies Department at the Russian Presidential Academy of National Economy and Public Administration. Dr. Shalumov has received several prestigious awards in science technology and is recognized as a leader in the field of information technology and automated systems in Russia. He has more than 300 publications, including 9 books. During the last 10 years he was the supervisor of 17 PhD works.

ALS140965@mail.ru

www.asonika.com

Ph. D. Evgeny Pershin is the research assistant of the research institute "ASONIKA" (Russia) and the developer of system ASONIKA