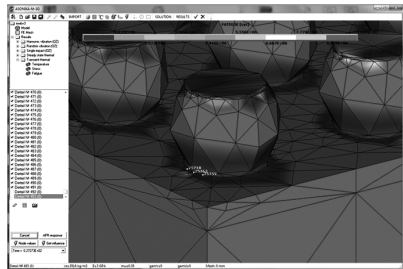
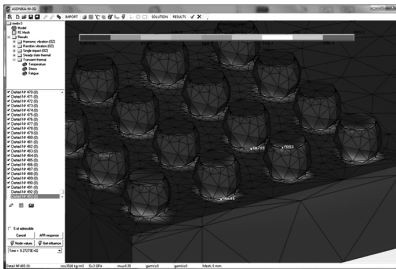
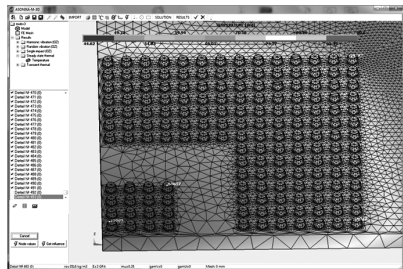
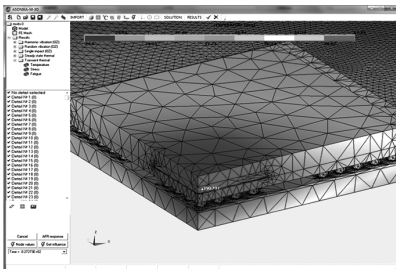




Accelerated Simulation of Thermal and Mechanical Reliability of Electronic Devices and Circuits

Prof. Alexander Shalumov
Ph. D. Evgeny Pershin



UDK 621.38
BBK 32.85
S18

Reviewer:
U.S. Saharov, prof.,
Head Department "Personal Electronics",
Moscow State University of
Instrument Engineering and Informatics

S18

- A. Shalumov, E. Pershin. **Accelerated Simulation of Thermal and Mechanical Reliability of Electronic Devices and Circuits.** - Moscow: Printing by PrintLETO.ru , 2013. – 128 c.
The press digital. A format 60 x 90 / 16. Paper offset, 80 g/m2. Circulation 300 copies.
The Order p Moscow, 2013

ISBN 978-5-91978-017-5

The monograph presents results by professor Dr. A. Shalumov's Research School of Modeling, Information Technology and Automated Systems (Russia). The program, ASONIKA, developed by the school is reviewed here regarding reliability and quality of devices for simulation of electronics and 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 number of cycles to fatigue failure under mechanical loads as well as under cyclic thermal effects. Simulation results for reliability analysis are taken into account. Models, software interface, and simulation examples are presented.

For engineers and scientists involved in design automation of electronics.

**Technical Editor: Anatoly Korkin, Research
Professor Arizona State University (USA)**

ISBN 978-5-91978-017-5

© Authors, 2013

TABLE OF CONTENTS

INTRODUCTION	3
1. MATHEMATICAL MODELS OF MECHANICAL PROCESSES IN MICROCIRCUITS	4
1.1. The model for the analysis of accelerations and stresses in chips exposed to vibration and shock	4
1.2. The model for the analysis of time to fatigue failure in chips exposed to vibration and shock	6
1.3. Models for the analysis of time to fatigue failure in microcircuits with transient thermal effects	52
2. AUTOMATED SUBSYSTEM ASONIKA-M-3D	53
2.1 Subsystem Interface	53
2.2 Projects in ASONIKA-M-3D	56
2.3 ASONIKA-M-3D Structure	58
2.3.1 ASONIKA-M-3D Preprocessor	58
2.3.1.1 Importing three-dimensional models	59
2.3.1.2 Specifying materials	60
2.3.1.3 Setting the mechanical effects	62
2.3.1.4 Setting endurance conditions	63
2.3.1.5 Setting thermal boundary conditions	65
2.3.1.6 Setting the finite element partitioning	67
2.3.1.7 Generation of the finite element mesh	68
2.3.2 Solver	68
2.3.3 Post-processor	73
2.3.3.1 Output fields of output values	73
2.3.3.2 Preparation of output of frequency response graph and Nyquist plot	74
2.3.3.3 Output of absolute and relative values of the output variables	75
2.3.3.4 Receiving variables of output values at the nodes of finite element mesh	75
2.3.3.5 Inspection of the deformed structure taking into account the enhancement factor	76
2.3.3.6 Response animation design on the mechanical effects	77
2.3.3.7 Export of mechanical effects to external files	78
2.3.4 Important features of ASONIKA-M-3D functions	78
3. EXAMPLE OF AN INTEGRATED CIRCUIT SIMULATION In ASONIKA-M-3D	79
3.1 Work in the preprocessor (preparing models for the calculation)	81
3.1.1 Creating a Project	81
3.1.2 Importing a model	81
3.1.3 Specifying materials	82
3.1.4 Setting the mechanical effects	90
3.1.5 Setting endurance conditions	92

3.1.6 Setting thermal boundary conditions	92
3.1.7 Setting finite element partitioning	96
3.1.8 Generation of the finite element mesh	96
3.2 Calculations	97
3.3 Review and results	99
3.3.1 Viewing the results of harmonic vibration analysis	100
3.3.2 Viewing the results of random vibration analysis	102
3.3.3 Viewing results of the single impact analysis	104
3.3.4 Viewing the results of steady-state thermal analysis	108
3.3.5 Viewing the results of transient thermal analysis	109
CONCLUSION	117
References	118
APPENDIX	119

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 Shalunov - 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 (1σ 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.

1.2. The model for the analysis of time to fatigue failure in chips exposed to vibration and shock

At present time, there are many approaches to simulate fatigue fracture. Among them are the three main ones:

1. High-cycle fatigue model.
2. Low-cycle fatigue model.
3. Approaches based on fatigue crack growth simulation.

The first approach requires low loads (stresses) and large number of cycles. The strains remain mostly elastic.

The second approach requires relatively high loads (stresses and strains) and a relatively small number of cycles. Here, plastic component of total deformation cannot be ignored.

A third approach is suitable for certain types of designs and uses various statistical hypotheses based on the analysis of experimental data.

The first two approaches are versatile and suitable for any design. Because the subsystem ASONIKA-M-3D was developed to simulate arbitrary constructions, only modes high-cycle and low-cycle fatigue will be considered.

High-cycle fatigue model.

In 1910, Basquin found that the curves of the alternating stress amplitude and the number of cycles to failure (curves Wohler) plotted on a logarithmic scale can be approximated by straight lines [1]. On this basis, he suggested the following formula for these named variables (so-called Basquin equation):

$$\frac{\Delta\sigma}{2} = \sigma_f' (2N_f)^b \quad (1.1)$$

where $\frac{\Delta\sigma}{2}$ – alternating stress amplitude,

N_f – number of cycles to failure,

σ_f' – fatigue strength coefficient, (a material property),

b – fatigue strength exponent (a material property).

Basquin equation is one of the basic equations used in the high-cycle fatigue, although in practice the following relationship is used more often:

$$N_f = N_b \left(\frac{\sigma_0}{\sigma_a} \right)^m \quad (1.2)$$

where N_f – number of cycles to failure,

N_b – base number of cycles (usually $N_b = 10^7$),

σ_0 – fatigue strength limit for material,

σ_a – cyclic alternating stress,

m – constant (a material property).

Parameter m characterizes the slope of fatigue. Figure 1.1 shows the experimentally obtained fatigue curves (Wohler curves). Curve 1 is characteristic for the steel of small and medium strength as well as for titanium alloys tested without influence for corrosion at normal temperature, curve 2 - non-ferrous metals and high-alloy steels. Curve 1 has a sharp change in the number of cycles $N_f \approx 10^6$, after which it is almost parallel to the x-axis. Therefore, as a base for testing, these materials are accepted $N_b = 10^7$.

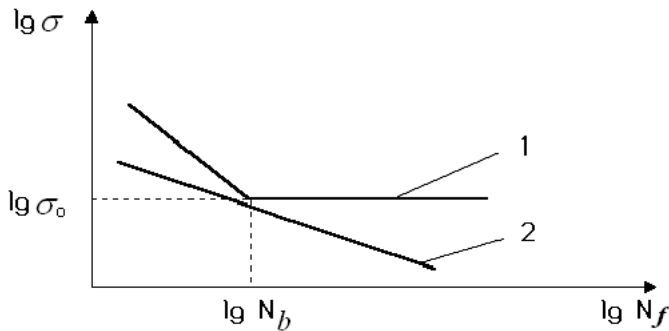


Figure 1.1 - Fatigue curves for different materials

Physically, this means that if the amplitude σ of the cycle variable stresses is less than the stress σ_0 , corresponding to the point of fracture, the fatigue failure does

not occur at an infinite number of cycles. This tension σ_0 is known as fatigue limit of the material in a symmetric cycle. Some materials, such as nonferrous metals and multicomponent alloys, don't have pronounced fatigue limit: with increasing number of cycles N_f strength continues to decrease (curve 2). For such materials, the fatigue resistance is characterized by a limited endurance at a given base (number of cycles) N_b , which is usually assumed to be 10^7 cycles. This fully applies to joints of electronic components (EC), as they are made of non-ferrous metals and their alloys.

Thus, high cycle fatigue defined parameters (m, σ_0, N_b) , or parameters (σ_f', b) . These models are equivalent and are related by the following relations:

$$\begin{aligned} m &= -\frac{1}{b} \\ \sigma_0 &= \sigma_f' (2N_b)^b \\ N_b &= 10^7 \end{aligned} \tag{1.3}$$

Low-cycle fatigue model.

During fatigue failure two main stages appear: the occurrence of cracks and fatigue crack growth until fracture. The high-cycle fatigue model does not distinguish between these two stages, and calculates the total time to failure. Methods of low-cycle fatigue are applied only during the first stage. Thus, the result of the calculation for low-cycle fatigue is the time to occurrence of cracks. Later, we will simply use generic term "time to failure" for both models of fatigue.

The basic equation for low-cycle fatigue is the Manson-Coffin-Basquin equation:

$$\frac{\Delta \varepsilon}{2} = \underbrace{\frac{\sigma_f'}{E} (2N_f)^b}_{\text{elastic component}} + \underbrace{\varepsilon_f' (2N_f)^c}_{\text{plastic component}}, \tag{1.4}$$

where $\frac{\Delta \varepsilon}{2}$ – total cyclic strain amplitude,

N_f – number of cycles to failure,

σ_f' – fatigue strength coefficient, (a material property),

b – fatigue strength exponent, (a material property),

ε_f' – fatigue ductility coefficient, (a material property),

c – fatigue ductility exponent, (a material property),

E – modulus of elasticity.

This equation includes two independent equations, representing, respectively, the elastic and plastic components of the total strain:

$$\frac{\Delta \sigma}{2} = \sigma_f' (2N_f)^b \quad (\text{Basquin's Equation}),$$

$$\frac{\Delta \varepsilon_p}{2} = \varepsilon_f' (2N_f)^c \quad (\text{Coffin-Manson equation}).$$

Coffin-Manson-Basquin equation can be represented graphically (Figure 1.2):

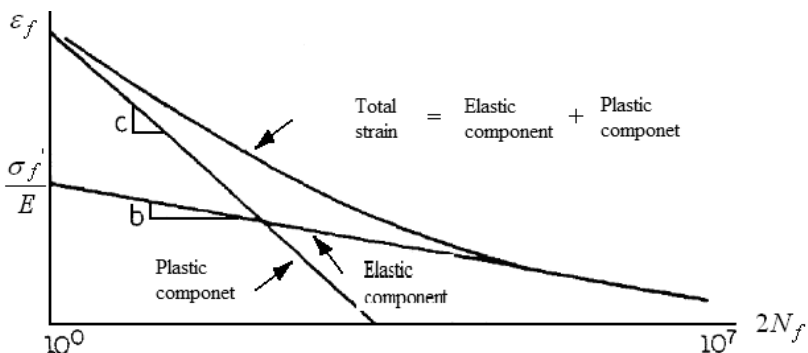


Figure 1.2 - Graphical representation of Manson-Coffin-Basquin equation on a logarithmic scale

Figure 1.2 shows that during significant deformations fatigue curve approaches the plastic curve (defined by Coffin-Manson equation), and at low strains - to the elastic curve (defined by Basquin's equation). This feature allows to approximately consider the basic equation of low-cycle fatigue as a general case against the high-cycle fatigue, and using the parameters ϵ_f' and b for high-cycle fatigue curve (Wohler curve) based on the Basquin's model.

Value equivalence.

Both models were developed for testing of prototypes in a simple loading (cyclic tension-compression, torsion). The result was the experimental curve describing the dependence of the cyclically varying parameter (stresses or total deformation) on the number of cycles to failure. High-cycle fatigue data curves are called fatigue curves or Wohler curves.

For their use in complex (multi-axis) loading the so-called equivalent value (equivalent stresses or equivalent strain) is applied. For example, to apply Wohler curve (built for testing under cyclic tension-compression) in the case of multiaxial cyclic loading at a critical point of the structure, it is necessary to calculate the amplitude of the equivalent stress based on one of the known criteria. The number of

cycles to failure for a given amplitude of equivalent stress is equal to the number of cycles to failure under cyclic tension-compression with the same stress amplitude.

There are many criteria for determining equivalent values. They are based on the use of static strength criteria for fatigue analysis under complex loading. Here are the most well-known criteria in terms of the principal stresses ($\sigma_1, \sigma_2, \sigma_3$):

1. The criterion of maximum principal stress.

The equivalent stress is equal to the maximum principal stress.

$$\sigma_1 = \sigma_e.$$

2. The criterion of maximum shear stress (Tresca criterion).

$$\left| \frac{\sigma_1 - \sigma_3}{2} \right| = \tau_e.$$

3. von Mises criterion.

$$\frac{1}{\sqrt{2}} \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2} = \sigma_e.$$

If the loading history is a random process (i.e. not cyclic), then there are special approaches for application of fatigue models that depend on loading history.

Cyclic curves of stress versus strain.

As noted, the low-cycle fatigue model takes into account plastic component of total strain. To use this model for fatigue calculations, one must consider characteristics of plastic deformation of materials under cyclic loading.

Total strain is the sum of the elastic and plastic components:

$$\varepsilon_{total} = \varepsilon_{elastic} + \varepsilon_{plastic}.$$

Plastic deformation, unlike elastic, retains its deformation state after the load is removed (Figure 1.3).

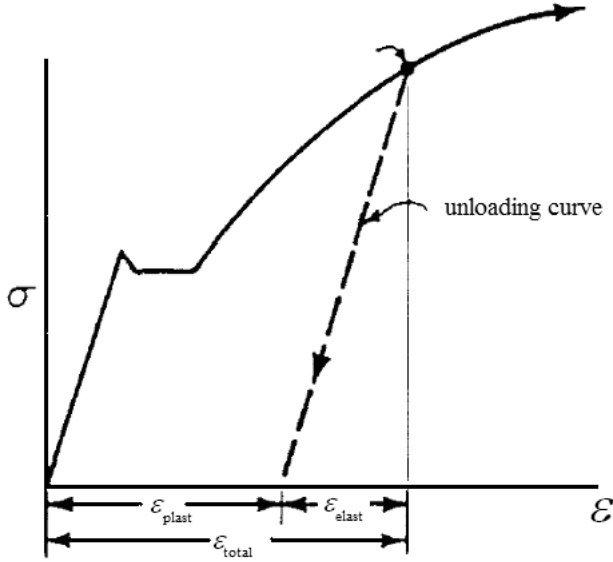


Figure 1.3 - The elastic and plastic deformation

The elastic part of total deformation is expressed by Hooke's law. For most metals, stress dependence of plastic deformation can be approximated by straight line on a logarithmic scale. As a result, this relationship can be represented by a power equation:

$$\sigma = K(\epsilon_{pl})^n$$

$$\text{or} \quad \epsilon_{pl} = \left(\frac{\sigma}{K} \right)^{\frac{1}{n}},$$

where K – modulus of plasticity,

n – strain hardening parameter.

The equation for the total strain can be written as:

$$\epsilon_{tot} = \epsilon_{el} + \epsilon_{pl} = \frac{\sigma}{E} + \left(\frac{\sigma}{K} \right)^{\frac{1}{n}}.$$

The equation describes the behavior of the material under monotonic loading with conditions where the residual strain is initially zero. Under cyclic loading behavior of the material corresponds to the hysteresis curve (see figure 1.4).

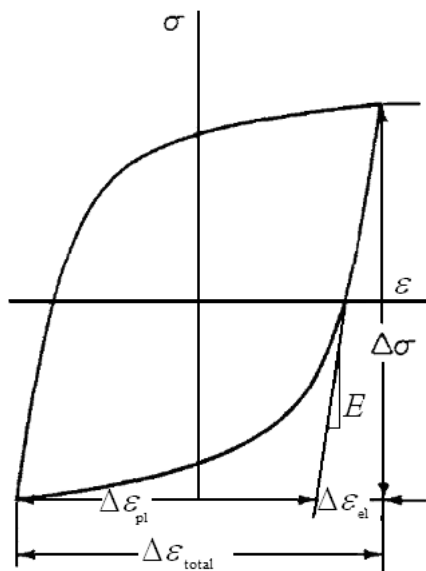


Figure 1.4 - Hysteresis curve

The equation for the hysteresis curve:

$$\frac{\Delta \varepsilon_{nonl}}{2} = \frac{\Delta \sigma}{2E} + \frac{\Delta \varepsilon_{nl}}{2}.$$

Material behavior under cyclic loading often varies from cycle to cycle. Depending on the properties of the material several behaviors may be observed [2]:

- 1) cyclic hardening (Figure 1.5)
- 2) cyclic softening (Figure 1.6)
- 3) cyclic stability,
- 4) hardening and softening depending upon the degree of deformation.

In Figures 1.5 and 1.6 on the left shows the dependence of the stress and strain on time and on the right - the corresponding hysteresis curves under cyclic hardening and softening.

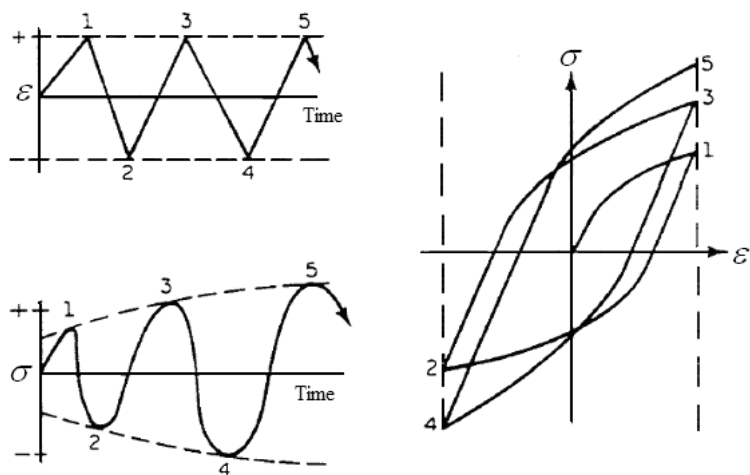


Figure 1.5 - Cyclic hardening

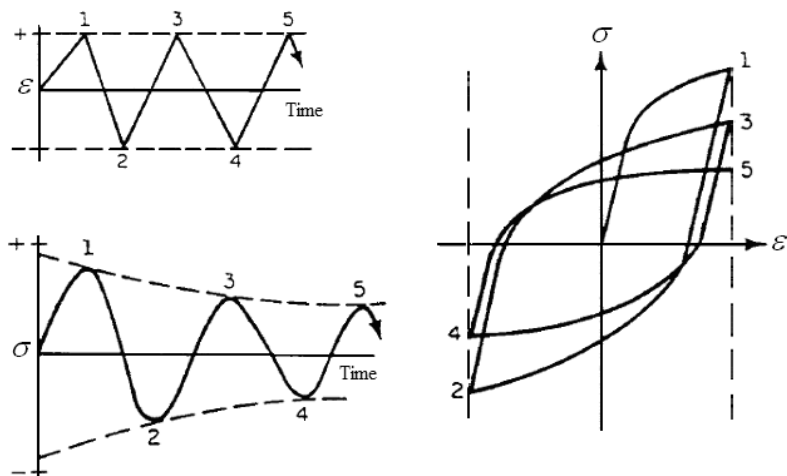


Figure 1.6 - Cyclic softening

Usually transient process of hardening or softening is observed in the early stages (cycles) of cyclic loading (20-40% of the total time to fatigue). Then the behavior of the material is stabilized following the so-called hysteresis curve.

To determine the total strain amplitude for a stable cycle, which is necessary for a low-cycle fatigue model, we introduce the so-called "cyclic" or "stable" curve of the stress-strain (cyclic stress-strain curve), whose equation can be written as:

$$\varepsilon_{nom} = \frac{\sigma}{E} + \left(\frac{\sigma}{K'} \right)^{\frac{1}{n'}} \quad (1.5)$$

where K' – cyclic elastic modulus,

n' – cyclic rate of hardening.

For different materials cyclic curve will vary depending on the respective monotonic behavior of the material in the transition process (softening or hardening).

Fatigue analysis is introduced with the following assumption: the behavior of the material during initial loading should have a cyclic curve of the stress-strain behavior of the material, followed by stabilized hysteresis curve [2].

The equation of the stabilized hysteresis curve is determined in accordance with the Massing's hypothesis with the doubling of the stress-strain cyclic curve:

$$\frac{\Delta \varepsilon_{nom}}{2} = \frac{\Delta \sigma}{2E} + \left(\frac{\Delta \sigma}{2K'} \right)^{\frac{1}{n'}} \quad (1.6)$$

Figure 1.7 is an example of cyclic loading with zero initial condition and constant amplitude (left) and the corresponding curve of stress versus strain (right). Section 0-1 corresponds to the initial application load, and describes the cyclic curve, section 1-2-3 is a closed cycle of stabilized hysteresis curve.

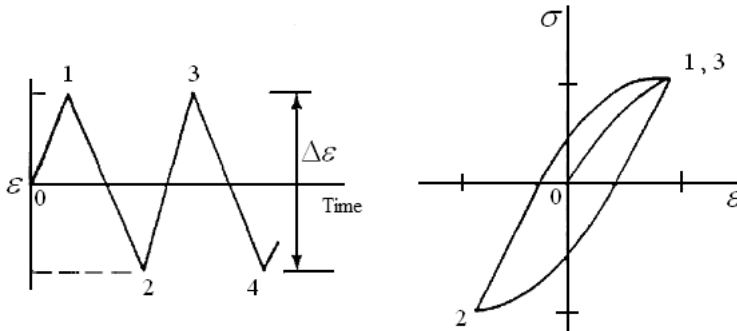


Figure 1.7 - Example of the constant amplitude cyclic load

Algorithms for counting cycles.

At random (non-cyclic) loading the stress (or strain) at any point of the design is also a random variable. Fatigue models discussed earlier were developed under cyclic loading with constant amplitude stress or strain. In a random loading (Figure 1.8) the term "loading cycle" becomes less certain.

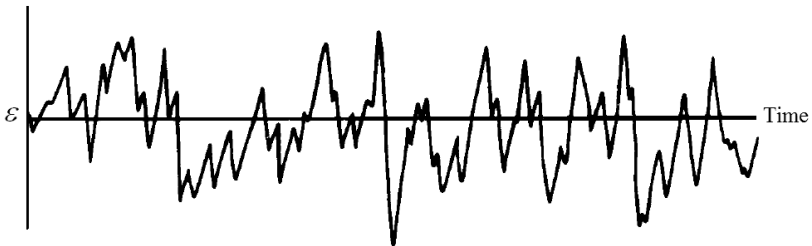


Figure 1.8 - Example of random loading history

To overcome this difficulty various researchers offered different interpretations of the "cycle" with the random loading. At present time, a number of algorithms exist that allow from random sample of loading (such as the one in Figure 1.8) to isolate the individual cycles with different amplitudes. Typically, the output of these algorithms are arrays whose elements contain entries like:

<amplitude, the number of cycles at a given amplitude>

The most sophisticated algorithms counting cycles are now the so-called rain algorithms, or rainflow algorithms [3 - 6].

Among the precursors of rain algorithms, the following can be distinguished: an algorithm of counting zero crossings, local extremes counting algorithm, a simple algorithm to calculate the range and others [5]. A common drawback of these algorithms is that they do not take into account the sequence of application cycles [4]. Because of the nonlinear relationship between stress and strain (in the region of plastic deformation), the sequence of cycles has a decisive influence on the response of the material, i.e. the shape of the hysteresis loops (see Figure 1.9).

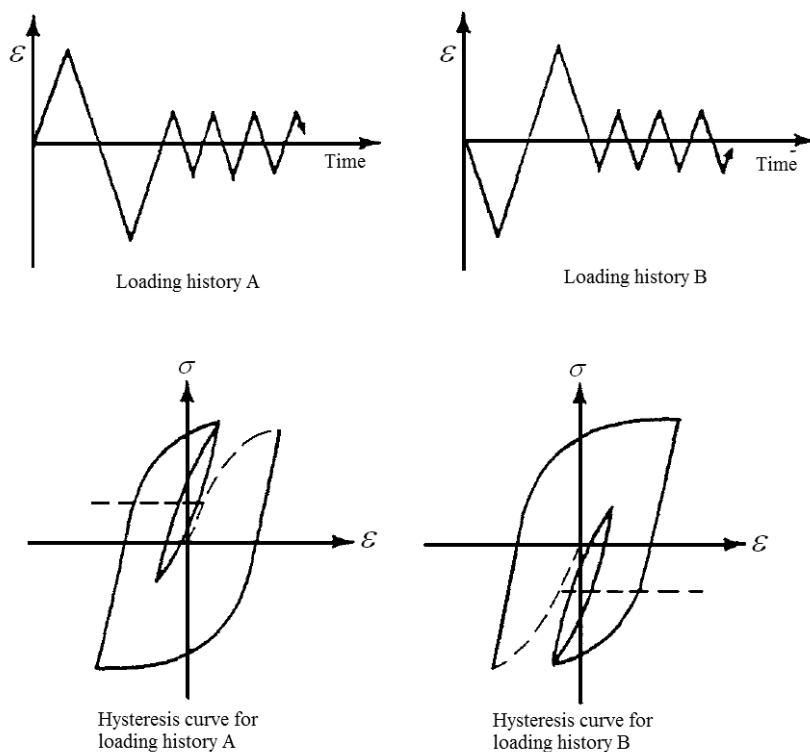


Figure 1.9 - Effect of application sequence of cycles on the response of the material

Figure 1.9 shows two histories of loading A and B. It can be seen that a change in the sequence of loading leads to a variety of hysteresis curves for cycles with a lower amplitude (in the first case - the positive value of the average stress in the second - with a negative). Using simple algorithms to count cycles will give the same results for the two histories of loading (A and B), although it is evident that the time to failure under load A is less. Features of the rainflow algorithms allow to take into account the effect of a sequence in a cycle.

There are several varieties of rainflow algorithms [2]. Their common name comes from analogy with the streams of rain motion on the roofs of Buddhist pagodas (pagoda roofs). The analogy becomes clear if we plot the stress history, placing time scale on the vertical axis (Figure 1.10). Cycles counting is conducted by the same rules and principles that rain droplets move across rooftops. These rules are as follows:

A trickle of rain moves from the top of each loading history and continues for as long as either of the following conditions are met:

1. A trickle flowing out of the point of local maximum falls into another point of local maximum with a deformation value greater than the original.
2. A trickle flowing out of the local minimum point falls into another point of local minimum with a deformation value less than the original.
3. A trickle joins another stream flowing from the previous summit.

Consider the example of counting cycles for loading history shown in Figure 1.10.

A. A trickle flowing out of the top of A through B & D, continues to flow until the end of the loading history, since none of the conditions are fulfilled from stopping the flow.

B. A trickle flowing out of the top of B through C, stops in front of the top of D, because B and D are local maximums and the deformation value of D is greater than B (condition 1).

C. A trickle flowing from the top of C, stops when meeting with a trickle flowing from the top of A (condition 3).

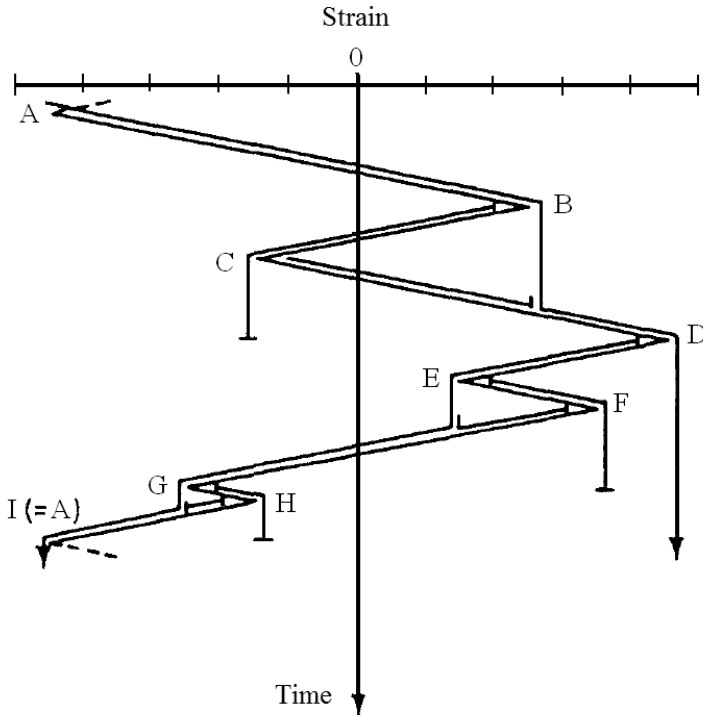


Figure 1.10 – Rainflow algorithm

D. A trickle of flowing from the top of D across the tops of E & G, continues to flow until the end of the loading history, as it does not fulfill any one of the conditions for stopping the flow.

E. A trickle flowing out of the top of E through the top F, stops in front of the vertex of G. E and G are local minima and the value of the deformation of G is smaller than in E (condition 2).

F. A trickle of flowing out of the top of F, stops at the junction with a trickle flowing from the top of D (condition 3).

G. A trickle of flowing out of the top of the top G through H, stops in front of the top I, since G and I are local minima and the value of the deformation for I is less than for G (condition 2).

H. A trickle of flowing from the top of H, stops at the junction with a trickle flowing from the top of D (condition 3).

Now we can use combination of events to form cycles. Events A-D and D-A will make one complete cycle. Event B-C is combined with C-B (part of C-D), also forming a cycle. Similarly cycles (E-F F-E) and (G-H H-G) are formed.

The algorithm of falling rain is able to recognize closed hysteresis loops. Figure 1.11 shows the stress-strain relationship for an example to be considered. It can be seen that the events of A-D, B-C, E-F and G-H form closed hysteresis loops. These same cycles are obtained using the falling rain algorithm. Note that this result occurs only if the condition of loading history begins and ends at the point where the strain is maximum (in absolute terms) value. In other cases, the the calculations become more complicated, and the results of different versions of the algorithm can be a little different. In this book we consider one of the variants of the algorithm, which is a standard of the American Society for Testing and Materials.

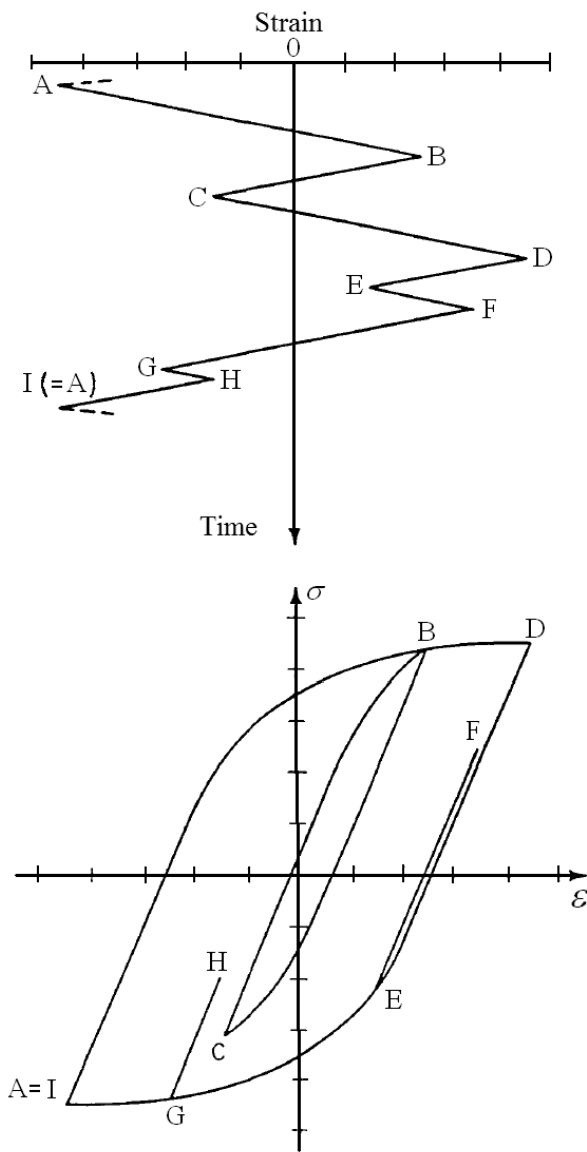


Figure 1.11 - Stress-strain relationship for the example considered

Spectral methods for calculating time to fatigue failure.

One way to calculate fatigue for random (non-cyclic) loading when the loading history is characterized by a graph of the spectral density of the stress or strain on the frequency, is to directly generate the equivalent of 'temporary' stress history for a given spectral density, and then use one of the algorithms for counting cycles, and finally summing up fatigue damage calculated for all cycles using one of the methods of summation for fatigue damage.

To get a good result from such an algorithm, it is necessary to generate a history of loading for a great enough length of time. This requires significant computational resources, and in practice is not often used [7].

Figures 1.12 and 1.13 are examples of loading history graphs in the time domain with corresponding spectral densities for the narrowband and broadband random processes.

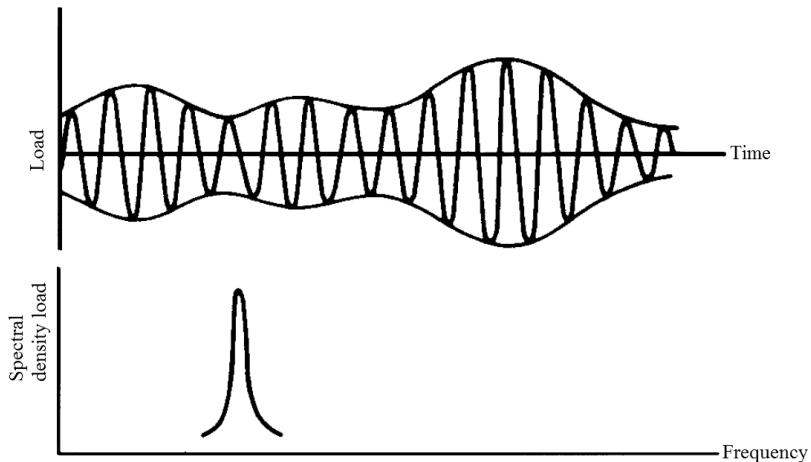


Figure 1.12 - Narrowband random process

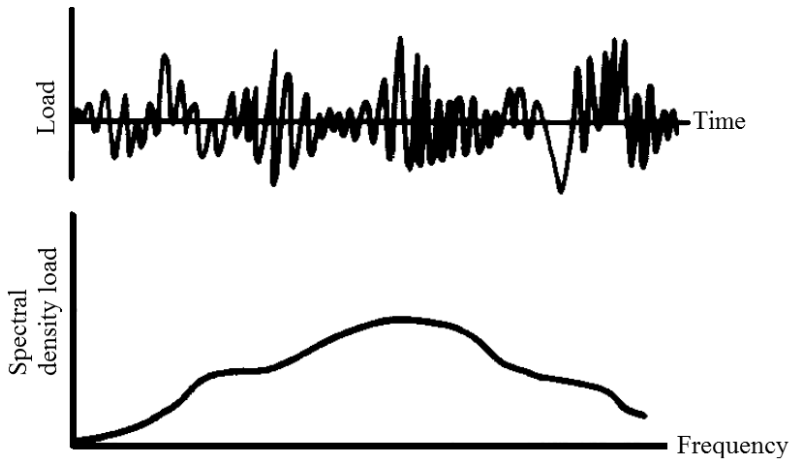


Figure 1.13 - Broadband random process

Another approach for calculating fatigue for random loading is calculation of timing immediately before the destruction of the spectral density. These calculation methods are called spectral methods.

The basis for the spectral methods are theoretical distributions of local extremes, amplitudes, and mean values in the loading history. From the point of view of fatigue, the distribution of local extremes (or peaks distribution) is the most important.

In 1954, one of the first theoretical papers on the spectral methods, Miles put forward the hypothesis that the density of local extrema is equal to the density of the amplitude distribution in the loading history [8]. This assumption is valid only for narrow-band loading history, which can be characterized by the following sequence of cyclically repeating steps (Figure 1.13): local maximum, crossing the zero bound in the negative direction, local minimum and zero-crossing boundaries in a positive direction. Moreover, if the loading history has a normal distribution:

$$p(x) = \frac{1}{\sqrt{2\pi\mu_x}} \exp\left[-\frac{x^2}{2\mu_x}\right],$$

the corresponding distribution of local extremes can be described as the Rayleigh distribution:

$$p(a) = \frac{a}{\mu_x} \exp\left[-\frac{a^2}{2\mu_x}\right].$$

Consider the derivation of the general formula to calculate the time of fatigue failure, using the model of high-cycle fatigue. For general formula, it is also necessary to use one of the hypotheses of summing fatigue damage. Most use the so-called rule of summation Palmgren-Miner, under which the accumulated damage D for the loading history with variable amplitude power is:

$$D = \sum_{i=1}^k D_i = \sum_{i=1}^k \frac{n_i}{N_{fi}} \quad (1.7)$$

where n_i – the number of cycles of i -th amplitude,

N_{fi} – number of cycles to failure with the amplitude of the i -th load,

k – number of different amplitudes in the loading history,

D_i – partial damage of cycles with the i -th amplitude.

It is assumed that fatigue failure occurs when $D = 1$.

If the probability of a local extremum is $P(\sigma_a)$ (at high-cycle fatigue loading we have alternating stresses), the number of cycles with amplitude σ_{a_i} with time T_0 is equal to:

$$n_i = n(\sigma_{a_i}) = M^+ T_0 P(\sigma_{a_i}),$$

where M^+ – expected number of local extrema per unit time.

Using the summation rule Palmgren-Miner, an accumulation of damage over time T_0 :

$$D(T_0) = \sum_{i=1}^k D_i = M^+ T_0 \int_0^{\infty} \frac{p(\sigma_a)}{N_f(\sigma_a)} d\sigma_a$$

where $p(\sigma_a)$ – density distribution of extremes,

$N_f(\sigma_a)$ – dependence of the number of cycles to failure of the stress amplitude of the curve given by Wohler.

Taking into account that the time to fatigue T , expressed in seconds corresponds damage $D(T) = 1$, from proportions

$$\frac{1}{D(T_0)} = \frac{T}{T_0}$$

obtain a general formula for calculating the time to fatigue failure:

$$T = \frac{T_0}{D(T_0)} = \frac{1}{M^+ T_0 \int_0^{\infty} \frac{p(\sigma_a)}{N_f(\sigma_a)} d\sigma_a}.$$

If the distribution of local extremes, according to the Miles hypothesis approximates Rayleigh distribution, and for dependence $N_f(\sigma_a)$ use according to one of the variants of the model high-cycle fatigue, then after mathematical transformations we obtain the following formula for calculating the time to failure:

$$T = \frac{\sigma_0^m N_b}{M^+ (2m_0)^{\frac{m}{2}} \Gamma\left(\frac{m+2}{2}\right)}, \quad (1.8)$$

where m – fatigue parameter,

$\Gamma()$ – gamma function,

N_b – base number of cycles,

σ_0 – fatigue limit.

For a history of narrow-band loading of local extremes in a unit of time M^+ is equal to the number of zero crossings N_0^+ in the positive direction of the border or dominant frequency f_0 :

$$M^+ = N_0^+ = \sqrt{\frac{m_2}{m_0}} = f_0,$$

where m_0 и m_2 – zero and second moments of the spectral density of the stress at an arbitrary point of the structure respectively. Moments are defined by the following formula:

$$m_k = \int_0^{\infty} f^k S(f) df, \quad (1.9)$$

where $S(f)$ – stress dependence of the spectral density with frequency.

The resulting formula is suitable only for a narrow-band random process. For broadband random process, density of local extrema in general will be different from the Rayleigh distribution while the number of local extrema per unit time equals the number of zero crossings in the positive direction of the border.

There are a number of different hypotheses to correct formula 1.8 in the case of broadband random process. Some of them will be discussed hereinafter.

Calculation of time for fatigue fracture in a harmonic vibration.

For the case of harmonic vibration we apply a model of cycle fatigue. This is due to the fact that, as compared, for example, with impact force, deformations occur primarily within the elastic limit.

The number of cycles to failure is calculated by one of the equations cycle fatigue (formulas 1.1 - 1.2).

Time to fatigue failure at any point during the construction of the harmonic vibration is determined by the formula:

$$T = \frac{N_f}{f}, \quad (1.10)$$

where N_f - number of stress cycles to failure; f - current frequency of oscillation.

The problem of calculating the time to fatigue failure at a harmonic vibration is quite trivial in relation to the following circumstances:

- the amplitude of the stress points of the structure is constant, therefore no additional algorithms for counting cycles with different amplitudes are required, the use of hypotheses summation of fatigue damage is excluded as well;
- no need for further adjustment of the average stress in the cycle as it is zero.

Accelerations exerted at all attachment locations are equal in phase and amplitude and cause a proportional loading in all points of the structure. The components of the stress tensor at the point oscillate in phase with constant amplitude within the cycle. Since the stress condition is not a single-axis at an arbitrary point of the structure, for application of the high-cycle fatigue model we need to calculate the equivalent stress value. Von Mises criterion, as the most common, is used by default.

The nodes of finite element mesh, in which finite elements have contact, representing different materials, there is some complexity in the calculation of fatigue. The fact that different materials have different fatigue properties can lead to calculation results in the node for the two adjoining elements to vary considerably. Therefore, after averaging the results of the calculation for all adjacent to the node finite elements will lead to a result devoid of any meaning. There are various approaches to overcome these difficulties.

One possible algorithm is presented below:

1. For each node, where elements of different types cross (ie different material properties), adjacent elements are separated into groups according to the type of material.
2. Each group of elements is determined by averaging equivalent stresses to all the elements of the group.

3 . Based on equivalent stresses and the fatigue properties of the materials for each group of elements time to fatigue is calculated.

4 . As a result the time to failure in the node, maximum time result for groups is applied.

The algorithm for fatigue design of printed circuits with a harmonic vibration.

1. Conducting modal analysis in ANSYS to obtain the natural frequencies of the structure. Analysis is performed in the frequency range of master control.

2 . Conducting harmonic analysis in ANSYS for frequency range of master control with the specified steps , including custom design frequency falling in the range and received at step modal analysis.

3 . Calculating time to fatigue fracture on the basis of harmonic analysis results. The calculation is carried out for the frequency range of master control with the specified steps, including the natural frequencies of the structure falling in the range from process and modal analysis.

3.1. Creating an array of FE mesh nodes , which will be calculated for fatigue. This array will include all external (i.e. located on the surface of the model) FE mesh nodes . In addition, if finite elements converge in a node with different material properties , the node is stored in the array as many times as it intersects materials . Material index is also written to the array for each node.

3.2. For all elements in the array of nodes ,using the basic equation cycle fatigue (formula 1.2) , we calculate number of cycles to failure . For an equivalent stress, we use the von Mises equivalent stress, namely its peak value . The peak value is defined as the square root of the sum of the real and imaginary parts of the complex solutions in a harmonic vibration.

3.3. For each frequency, the time to failure is calculated by dividing the number of cycles to failure at the desired frequency.

3.4. In FE mesh nodes, which converge in finite elements with different material properties, multiple different values of time to fatigue occur corresponding to different materials. As a result, the time to failure values of these nodes receive maximum applicable value of time.

3.5. Formation of output arrays containing allocation of time to failure at the grid nodes of FE mesh for each frequency. If the time to failure in the node exceeds a certain limit, the resulting array is stored as a special object, symbolizing that the destruction does not come with an unlimited number of cycles.

Calculation of fatigue failure in random vibration.

Random vibration is divided into steady-state and transient parts. steady-state random vibration is normally distributed. The deviation from the amplitude of acceleration of the normal distribution for transient vibration is caused by the presence of periodic vibrations in a total random process.

Modeling transient random vibration requires many function types containing acceleration versus time and is not applied in practice.

A measure of random vibration is a spectral density (Power Spectral Density or PSD), which determines the distribution of RMS values or power of the random process of a frequency $S(f)$.

Implemented in ANSYS analysis for random vibration (so-called PSD analysis), provides a standard deviations (1σ values) of displacements, accelerations, stresses, and strains in the finite element mesh of nodes [9]. All these values in a steady-state random process are normally distributed with a mean of zero. Therefore, RMS (root mean square) value in this case will be equal to the variance, and the root of RMS will be equal to the square root of the variance, that is, the standard deviation σ .

Post-processor (Time-History Postprocessor) system of ANSYS can also calculate the spectral density of the output variables (displacements, accelerations, stresses, and strains) in all FE mesh nodes, as well as mutual spectral density (Cross Spectral Density) of the two output values (for example, two components of the stress

tensor) . The integral of the frequency of the spectral density is equal to the rms value (and variance if the expectation is zero), and the integral of the frequency of the relative spectral density is equal to the correlation time (the time of the mixed second-order or covariance).

Acceleration exerted on constrained regions causes stress at any point of the design, characterized by stress and strain tensors:

$$\boldsymbol{\sigma}(\mathbf{t}) = \begin{bmatrix} \sigma_{xx}(t) & \sigma_{xy}(t) & \sigma_{xz}(t) \\ \sigma_{yx}(t) & \sigma_{yy}(t) & \sigma_{yz}(t) \\ \sigma_{zx}(t) & \sigma_{zy}(t) & \sigma_{zz}(t) \end{bmatrix},$$

$$\boldsymbol{\varepsilon}(\mathbf{t}) = \begin{bmatrix} \varepsilon_{xx}(t) & \varepsilon_{xy}(t) & \varepsilon_{xz}(t) \\ \varepsilon_{yx}(t) & \varepsilon_{yy}(t) & \varepsilon_{yz}(t) \\ \varepsilon_{zx}(t) & \varepsilon_{zy}(t) & \varepsilon_{zz}(t) \end{bmatrix}.$$

With random vibration all tensor components are random variables. Thus, the stress state at the point characterizes the system of nine random variables.

Since stress and strain tensors have symmetric matrices, in which:

$$\begin{aligned} \sigma_{xy}(t) &= \sigma_{yx}(t) & \sigma_{xz}(t) &= \sigma_{zx}(t) & \sigma_{zy}(t) &= \sigma_{yz}(t) \\ \varepsilon_{xy}(t) &= \varepsilon_{yx}(t) & \varepsilon_{xz}(t) &= \varepsilon_{zx}(t) & \varepsilon_{zy}(t) &= \varepsilon_{yz}(t) \end{aligned},$$

then a six-dimensional stochastic process that characterizes the stress state can be represented by two vectors:

$$\boldsymbol{\sigma}(\mathbf{t}) = [\sigma_{xx}(t) \quad \sigma_{yy}(t) \quad \sigma_{zz}(t) \quad \sigma_{xy}(t) \quad \sigma_{xz}(t) \quad \sigma_{yz}(t)]$$

$$\boldsymbol{\varepsilon}(\mathbf{t}) = [\varepsilon_{xx}(t) \quad \varepsilon_{yy}(t) \quad \varepsilon_{zz}(t) \quad \varepsilon_{xy}(t) \quad \varepsilon_{xz}(t) \quad \varepsilon_{yz}(t)].$$

For six-dimensional stationary normally distributed random process, the joint density is given by [7]:

$$p_{x_1, \dots, x_6}(x_1, \dots, x_6, \tau) = \frac{1}{\sqrt{(2\pi)^6 |\boldsymbol{\mu}_x(\tau)|}} \exp \left[-\frac{1}{2} \mathbf{x} \boldsymbol{\mu}_x^{-1}(\tau) \mathbf{x}^T \right],$$

$$\text{where } \mu_x(\tau) = \begin{bmatrix} \mu_{x11}(\tau) & \mu_{x12}(\tau) & \mu_{x13}(\tau) & \mu_{x14}(\tau) & \mu_{x15}(\tau) & \mu_{x16}(\tau) \\ \mu_{x21}(\tau) & \mu_{x22}(\tau) & \mu_{x23}(\tau) & \mu_{x24}(\tau) & \mu_{x25}(\tau) & \mu_{x26}(\tau) \\ \mu_{x31}(\tau) & \mu_{x32}(\tau) & \mu_{x33}(\tau) & \mu_{x34}(\tau) & \mu_{x35}(\tau) & \mu_{x36}(\tau) \\ \mu_{x41}(\tau) & \mu_{x42}(\tau) & \mu_{x43}(\tau) & \mu_{x44}(\tau) & \mu_{x45}(\tau) & \mu_{x46}(\tau) \\ \mu_{x51}(\tau) & \mu_{x52}(\tau) & \mu_{x53}(\tau) & \mu_{x54}(\tau) & \mu_{x55}(\tau) & \mu_{x56}(\tau) \\ \mu_{x61}(\tau) & \mu_{x62}(\tau) & \mu_{x63}(\tau) & \mu_{x64}(\tau) & \mu_{x65}(\tau) & \mu_{x66}(\tau) \end{bmatrix}$$

– the matrix of variances and correlation moments (covariance matrix) for a system of random variables $X_1, \dots, X_6$;

$\mathbf{x} = [x_1 - M(X_1), \dots, x_6 - M(X_6)]$, where $M(X_i)$ – mathematical expectation X_i .

A random process of a spectral density is given by the matrix:

$$\mathbf{S(f)} = \begin{bmatrix} S_{11}(f) & S_{12}(f) & S_{13}(f) & S_{14}(f) & S_{15}(f) & S_{16}(f) \\ S_{21}(f) & S_{22}(f) & S_{23}(f) & S_{24}(f) & S_{25}(f) & S_{26}(f) \\ S_{31}(f) & S_{32}(f) & S_{33}(f) & S_{34}(f) & S_{35}(f) & S_{36}(f) \\ S_{41}(f) & S_{42}(f) & S_{43}(f) & S_{44}(f) & S_{45}(f) & S_{46}(f) \\ S_{51}(f) & S_{52}(f) & S_{53}(f) & S_{54}(f) & S_{55}(f) & S_{56}(f) \\ S_{61}(f) & S_{62}(f) & S_{63}(f) & S_{64}(f) & S_{65}(f) & S_{66}(f) \end{bmatrix},$$

where the diagonal elements are the components of the six-dimensional spectral density of a random process, and the off-diagonal elements are the mutual spectral density.

Equivalent values with random vibrations.

Impacts at endurance points are treated as random variables with the same statistical parameters, however, at any moment of time their instantaneous values may vary. This will show in any point of the structure a complicated multi-axis and generally disproportionate loading. In other words, the direction and proportional of principal stress (or strain) can vary over time. For the analysis of fatigue in the face high stress state method of critical surfaces is generally used. Methods of critical planes take into account accumulation of fatigue damage in separate planes, in which it is assumed that the destruction is most likely to occur. To determine the position of the critical plane a certain criterion is given. The criteria applied to the probabilistic

analysis (i.e. analysis of a random vibration) determined by spectral density are presented in [7].

The main criteria are divided into two groups depending on the model of fatigue:

1. Generalized criterion of maximum normal and shear stress in the critical plane (used for high-cycle fatigue).

2. Generalized criterion of maximum linear and angular deformity in the critical plane (used for low-cycle fatigue).

The position of the critical plane is given by the directional cosines of three mutually perpendicular vectors (usually vectors of principal stresses or strains):

$$l_n, m_n, n_n, \quad (n = 1, 2, 3).$$

The following are particular criteria for the above two groups. For each criterion a formula of equivalent value is obtained.

1. The criterion of the maximum normal stress in the critical plane.

The critical plane is determined by the average direction of the maximum principal stress $\sigma_1(t)$:

$$\sigma_e = l_1^2 \sigma_{xx}(t) + m_1^2 \sigma_{yy}(t) + n_1^2 \sigma_{zz}(t) + 2l_1 m_1 \sigma_{xy}(t) + 2l_1 n_1 \sigma_{xz}(t) + 2m_1 n_1 \sigma_{yz}(t)$$

2. The criterion of maximum shear stress in the critical plane.

The critical plane is determined by the average position of one of the two planes in which the maximum shear stress acts $\tau_1(t)$:

$$\begin{aligned} \sigma_e = & (l_1^2 - l_3^2) \sigma_{xx}(t) + (m_1^2 - m_3^2) \sigma_{yy}(t) + (n_1^2 - n_3^2) \sigma_{zz}(t) + \\ & + 2(l_1 m_1 - l_3 m_3) \sigma_{xy}(t) + 2(l_1 n_1 - l_3 n_3) \sigma_{xz}(t) + 2(m_1 n_1 - m_3 n_3) \sigma_{yz}(t) \end{aligned}$$

3. The criterion of the maximum linear strain in the critical plane.

The critical plane is determined by the average direction of the maximum principal strain $\varepsilon_1(t)$:

$$\varepsilon_e = l_1^2 \varepsilon_{xx}(t) + m_1^2 \varepsilon_{yy}(t) + n_1^2 \varepsilon_{zz}(t) + 2l_1 m_1 \varepsilon_{xy}(t) + 2l_1 n_1 \varepsilon_{xz}(t) + 2m_1 n_1 \varepsilon_{yz}(t).$$

4. The criterion of maximum angular deformation in the critical plane.

The critical plane is defined by the average position of one of the two planes in which there is a maximum shear strain $\gamma_1(t)$:

$$\varepsilon_e = \frac{1}{1+\nu} \left[(l_1^2 - l_3^2) \varepsilon_{xx}(t) + (m_1^2 - m_3^2) \varepsilon_{yy}(t) + (n_1^2 - n_3^2) \varepsilon_{zz}(t) + 2(l_1 m_1 - l_3 m_3) \varepsilon_{xy}(t) + 2(l_1 n_1 - l_3 n_3) \varepsilon_{xz}(t) + 2(m_1 n_1 - m_3 n_3) \varepsilon_{yz}(t) \right],$$

where ν – Poisson's ratio

The above formulas for the equivalent stresses σ_e and strains ε_e include the directional cosines of the vectors defining the critical plane and the corresponding tensor components of stress and strain. The linear form of these criteria can be written in the form of a generalized formula of equivalent value, considering the coefficients of the components of a tensor as a set of constant coefficients:

$$x_e(t) = \sum_{k=1}^6 a_k x_k(t),$$

where $\mathbf{a} = [a_1, \dots, a_6]$ – vector of coefficients, depending on the chosen criteria;

$x_k(t)$ – component of the stress tensor or deformation.

The equivalent value, as well as a tensor, is normally distributed. It follows from the above linear relationship between the magnitude and the equivalent components of the tensor. The density distribution of the equivalent value is expressed as:

$$p_{x_e}(x_e) = \frac{1}{\sqrt{2\pi\mu_{x_e}}} \exp \left[\frac{-(x_e - M(x_e))^2}{2\mu_{x_e}} \right],$$

where $M(x_e)$ – expectation of equivalent value:

$$M(x_e) = \sum_{k=1}^6 a_k M(x_k);$$

μ_{x_e} – dispersion of equivalent value:

$$\mu_{x_e} = \sum_{k=1}^6 \sum_{l=1}^6 a_k a_l \mu_{x_{kl}}.$$

The formula shows that if the expectation component of the tensor is zero, then the expectation of an equivalent amount is also equal to zero. This result cannot be obtained for nonlinear criteria for determining the equivalent values (such as Tresca and von Mises hypothesis).

The spectral density of the equivalent value expressed in terms of the spectral density tensor components:

$$\begin{aligned}
 S_{xe}(f) &= \sum_{k=1}^6 \sum_{l=1}^6 a_k a_l S_{kl}(f) = \\
 &= \sum_{k=1}^6 a_k^2 S_{kk}(f) + 2 \sum_{k < l}^6 a_k a_l S_{kl}(f) .
 \end{aligned}$$

Formulas for spectral density and variance of the random variable in the equivalent matrix form have a more compact form:

$$\begin{aligned}
 \mu_{xe} &= \mathbf{a} \boldsymbol{\mu}_x \mathbf{a}^T , \\
 S_{xe}(f) &= \mathbf{a} \mathbf{S}_x(\mathbf{f}) \mathbf{a}^T ,
 \end{aligned}$$

where $\boldsymbol{\mu}_x$ – matrix of variances,

$\mathbf{S}_x(\mathbf{f})$ – spectral density tensor matrix,

$\mathbf{a}$ – vector of coefficients, depending on the selected criteria.

The use of ANSYS to implement a set of vector and matrix operations, can significantly speed up executable macros.

The algorithm for determining the position of the plane of the critical criterion for the maximum normal stress.

There are several methods for determining the position of the critical plane. The following describes a method called variance. This method can be used when the tensor components have the same probability distribution. In the method of dispersion is assumed that the plane in which the dispersion reaches the value equivalent to the maximum, is critical for the material . To find the maximum dispersion, it is necessary to determine a maximum value for the expression:

$$\mu_{xe} = \sum_{k=1}^6 \sum_{l=1}^6 a_k a_l \mu_{xkl}.$$

For the criterion of maximum normal stress the critical plane is the direction orthogonal to it the unit vector $\bar{\eta}$, defined by the direction cosines (l_1, m_1, n_1) . The vector of coefficients **a** equal to:

$$\mathbf{a} = \begin{bmatrix} l_1^2 & m_1^2 & n_1^2 & 2l_1m_1 & 2l_1n_1 & 2m_1n_1 \end{bmatrix}.$$

A vector orthogonal to the critical plane, may be arranged randomly in space. Considering all the possible positions of the vector within the hemisphere (Figure 1.14), setting a definite step in the sample space, we can get the directional cosine matrix, with each row representing the directional cosines for the particular position of the vector. The matrix will have a dimension equal to the number of positions in space of $\times 3$.

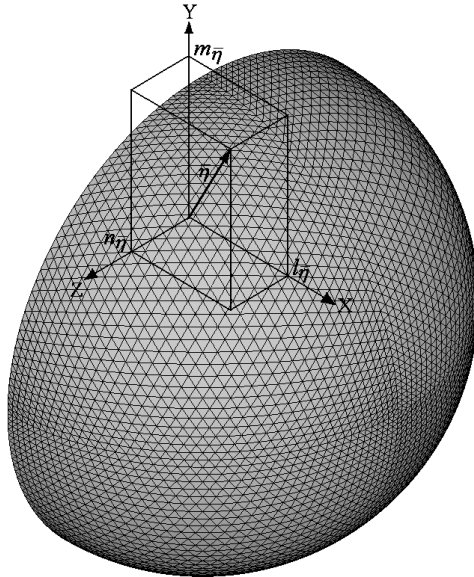


Figure 1.14 - The triangulated hemisphere

For each row of the directional cosine vector, coefficients of vector $\mathbf{a}$ and variance of equivalent value are calculated. Minimum variance for all positions of the vector $\bar{\eta}$ determines the location of the critical plane.

This algorithm is efficiently implemented in ANSYS, using the built-in matrix operations. The algorithm uses the pre-harvested directional cosine matrix, the size of which can be varied. By default, the matrix of 4141 positions is used.

To construct the matrix of directional cosines a simple hemisphere triangulation algorithm is used. Figure 1.14 shows a triangulated hemisphere containing 4141 vertices.

The use of spectroscopic methods for the calculation of the time to failure.

Since the result of analysis on a random vibration define the structure spectral density stress nodes, then in order to calculate the time until the fatigue failure spectral methods need to be used. Formula 1.8 is suitable for fatigue calculation only for the narrowband random vibration. For an arbitrary, including broadband random vibration, various hypotheses adjusting the formula 1.8 are used. Consider two of the most well-known hypotheses:

1. Reicher's Hypothesis.

Reicher proposed the following formula for calculating the random vibration fatigue:

$$T = \frac{\sigma_0^m N_b}{\left[\int_0^\infty S_0(f) f^{\frac{2}{m}} df \right]^{\frac{m}{2}} (2m_0)^{\frac{m}{2}} \Gamma\left(\frac{m+2}{2}\right)}, \quad (1.11)$$

where $S_0(f) = \frac{S(f)}{m_0}$ – reduced spectral density

The formula shows that the number of local extrema per time unit M^+ depends on the fatigue parameter m . This adjustment results from Reicher conclusions based on the analysis of experimental data.

2. Dirlik formula.

Dirlik, based on numerous simulations of the Monte Carlo method, derived an empirical formula that approximates the distribution of the amplitudes obtained at the output of the rainflow algorithms[5]. Usually the result of rainflow algorithms is not the distribution of the actual amplitude, but the distribution of ranges (a range which is twice the amplitude $\Delta\sigma = 2\sigma_a$), so the formula for the distribution Dirlik written in terms of ranges:

$$p(\Delta\sigma) = \frac{1}{2\sqrt{m_0}} \left[\frac{G_1}{Q} e^{\frac{-Z}{Q}} + \frac{G_2}{R^2} e^{\frac{-Z^2}{2R^2}} + G_3 Z e^{\frac{-Z^2}{2}} \right],$$

$$\text{where } Z = \frac{\Delta\sigma}{2\sqrt{m_0}}, \quad G_1 = \frac{2(x_m - I^2)}{1 + I^2},$$

$$G_2 = \frac{1 - I - G_1 + G_1^2}{1 + R}, \quad G_3 = 1 - G_1 - G_2,$$

$$R = \frac{I - x_m - G_1^2}{1 - I - G_1 + G_1^2}, \quad Q = \frac{1.25(I - G_3 - G_2 R)}{G_1},$$

$$x_m = \frac{m_1}{m_0} \left(\frac{m_2}{m_4} \right)^{\frac{1}{2}}, \quad I = \frac{m_2}{\sqrt{m_2 m_4}}.$$

Dirlik distribution is a combination of one exponential and two Rayleigh distributions. Also, the formula assumes that the zero point of the spectral density is a critical parameter.

Time to fatigue formula can be obtained from the conversion as previously described:

$$T = \frac{2^m \sigma_0^m N_b}{M^+ \int_0^\infty p(\Delta\sigma) \Delta\sigma^m d\Delta\sigma},$$

where $M^+ = \sqrt{\frac{m_4}{m_2}}$.

Assuming that the Dirlik distribution in terms of amplitude equal to:

$$p(\sigma_a) = \frac{1}{\sqrt{m_0}} \left[\frac{G_1}{Q} e^{\frac{-Z}{Q}} + \frac{G_2}{R^2} e^{\frac{-Z^2}{2R^2}} + G_3 Z e^{\frac{-Z^2}{2}} \right],$$

the following formula is obtained:

$$T = \frac{\sigma_0^m N_b}{M^+ \int_0^\infty p(\sigma_a) \sigma_a^m d\sigma_a}.$$

Performing the transformations associated with the integration by using gamma function machinery, the formula for time to fatigue failure can be written more simply as follows [3]:

$$T = \frac{\sigma_0^m N_b}{M^+ m_0^{m/2} \left[G_1 Q^m \Gamma(1+m) + 2^{m/2} \Gamma\left(1 + \frac{m}{2}\right) \left(G_2 |R|^m + G_3 \right) \right]} \quad (1.12)$$

General algorithm for calculating the time to fatigue failure in random vibration.

Summarizing all of the above, we present a general algorithm for calculating the time to fatigue failure. Calculation according to the algorithm will be performed for the nodes of FE mesh.

1. Getting the spectral densities for the components of the stress tensor, including mutual spectral densities using the ANSYS post-processor (Time-History Postprocessor).

The result of the first step may be represented in matrix form:

$$\mathbf{S}_\sigma(\mathbf{f}) = \begin{bmatrix} S_{\sigma_{xx}}(f) & S_{\sigma_{xx}\sigma_{yy}}(f) & S_{\sigma_{xx}\sigma_{zz}}(f) & S_{\sigma_{xx}\sigma_{xy}}(f) & S_{\sigma_{xx}\sigma_{xz}}(f) & S_{\sigma_{xx}\sigma_{yz}}(f) \\ & S_{\sigma_{yy}}(f) & S_{\sigma_{yy}\sigma_{zz}}(f) & S_{\sigma_{yy}\sigma_{xy}}(f) & S_{\sigma_{yy}\sigma_{xz}}(f) & S_{\sigma_{yy}\sigma_{yz}}(f) \\ & & S_{\sigma_{zz}}(f) & S_{\sigma_{zz}\sigma_{xy}}(f) & S_{\sigma_{zz}\sigma_{xz}}(f) & S_{\sigma_{zz}\sigma_{yz}}(f) \\ & & & S_{\sigma_{xy}}(f) & S_{\sigma_{xy}\sigma_{xz}}(f) & S_{\sigma_{xy}\sigma_{yz}}(f) \\ & & & & S_{\sigma_{xz}}(f) & S_{\sigma_{xz}\sigma_{yz}}(f) \\ & & & & & S_{\sigma_{yz}}(f) \end{bmatrix}$$

Here the spectral density matrix is a symmetric matrix. In general, the mutual spectral density (off-diagonal elements of the matrix) contain imaginary components, which in the calculation vanish when summing in step 4 [7].

3. Calculation of the elements of the matrix (μ_σ) variances and correlation moments (covariance matrix) by integration of the corresponding spectral densities

$$\mu_\sigma = \begin{bmatrix} \int S_{\sigma_{xx}}(f)df & \int S_{\sigma_{xx}\sigma_{yy}}(f)df & \int S_{\sigma_{xx}\sigma_{zz}}(f)df & \int S_{\sigma_{xx}\sigma_{xy}}(f)df & \int S_{\sigma_{xx}\sigma_{xz}}(f)df & \int S_{\sigma_{xx}\sigma_{yz}}(f)df \\ & \int S_{\sigma_{yy}}(f)df & \int S_{\sigma_{yy}\sigma_{zz}}(f)df & \int S_{\sigma_{yy}\sigma_{xy}}(f)df & \int S_{\sigma_{yy}\sigma_{xz}}(f)df & \int S_{\sigma_{yy}\sigma_{yz}}(f)df \\ & & \int S_{\sigma_{zz}}(f)df & \int S_{\sigma_{zz}\sigma_{xy}}(f)df & \int S_{\sigma_{zz}\sigma_{xz}}(f)df & \int S_{\sigma_{zz}\sigma_{yz}}(f)df \\ & & & \int S_{\sigma_{xy}}(f)df & \int S_{\sigma_{xy}\sigma_{xz}}(f)df & \int S_{\sigma_{xy}\sigma_{yz}}(f)df \\ & & & & \int S_{\sigma_{xz}}(f)df & \int S_{\sigma_{xz}\sigma_{yz}}(f)df \\ & & & & & \int S_{\sigma_{yz}}(f)df \end{bmatrix}$$

4. The integration is performed over the entire range of master control frequencies.

3. Determination of the critical position of the plane according to one of the criteria for dispersion method using the algorithm which was described earlier.

For the criterion of maximum normal stress, a result of this step will be a vector of directional cosines orthogonal to the critical plane (l_1, m_1, n_1) , and also a vector of coefficients:

$$\mathbf{a} = \begin{bmatrix} l_1^2 & m_1^2 & n_1^2 & 2l_1m_1 & 2l_1n_1 & 2m_1n_1 \end{bmatrix}.$$

4. Spectral density calculation of the equivalent value.

The spectral density of the equivalent stress is calculated by the formula:

$$\begin{aligned} S_{\sigma_e}(f) = & a_1^2 S_{\sigma_{xx}}(f) + a_2^2 S_{\sigma_{yy}}(f) + a_3^2 S_{\sigma_{zz}}(f) + a_4^2 S_{\sigma_{xy}}(f) + \\ & + a_5^2 S_{\sigma_{xz}}(f) + a_6^2 S_{\sigma_{yz}}(f) + 2a_1a_2 S_{\sigma_{xx}\sigma_{yy}}(f) + 2a_1a_3 S_{\sigma_{xx}\sigma_{zz}}(f) + \\ & + 2a_1a_4 S_{\sigma_{xx}\sigma_{xy}}(f) + 2a_1a_5 S_{\sigma_{xx}\sigma_{xz}}(f) + 2a_1a_6 S_{\sigma_{xx}\sigma_{yz}}(f) + \\ & + 2a_2a_3 S_{\sigma_{yy}\sigma_{zz}}(f) + 2a_2a_4 S_{\sigma_{yy}\sigma_{xy}}(f) + 2a_2a_5 S_{\sigma_{yy}\sigma_{xz}}(f) + \\ & + 2a_2a_6 S_{\sigma_{yy}\sigma_{yz}}(f) + 2a_3a_4 S_{\sigma_{zz}\sigma_{xy}}(f) + 2a_3a_5 S_{\sigma_{zz}\sigma_{xz}}(f) + \\ & + 2a_3a_6 S_{\sigma_{zz}\sigma_{yz}}(f) + 2a_4a_5 S_{\sigma_{xy}\sigma_{xz}}(f) + 2a_4a_6 S_{\sigma_{xy}\sigma_{yz}}(f) + \\ & + 2a_5a_6 S_{\sigma_{xz}\sigma_{yz}}(f) \end{aligned},$$

where $a_1..a_6$ – components of the vector of coefficients $\mathbf{a}$.

5. Calculating time to fatigue failure with formula (1.11) or (1.12).

Moments of equivalent values needed to be inserted in the time to failure formula, is calculated in accordance with:

$$m_k = \int_0^{\infty} f^k S_{\sigma_e}(f) df,$$

where m_k – k -th moment of the spectral density of the equivalent stress.

Full and abbreviated calculation algorithms.

The presented general calculation algorithm is time-consuming to apply to all nodes of FE mesh. For complex printed circuit board assemblies, the amount of nodes may exceed 100,000. Getting the spectral density matrix (step 1 of the general algorithm) takes most of the time, while the remaining steps of the general algorithm are efficiently implemented using matrix operations of ANSYS.

To reduce the time of calculation two algorithms were developed in which the calculation is carried out for certain finite element mesh nodes, which are of greatest interest from the point of view of fatigue failure.

Since fatigue is a superficial phenomenon, that is, the emergence and development of fatigue cracks occur on the surface of the structural elements, it is sufficient to calculate the time to fatigue failure for external FE mesh nodes. Full algorithm implements this approach.

However, despite the significant reduction in the time for the full calculation, it can take 1-2 hours for complex printed circuit boards. Therefore, an abbreviated algorithm was developed and will be described hereinafter.

PSD- analysis in ANSYS produces equivalent stress and strain according to von Mises hypothesis. For this purpose, a special algorithm (Seegalman-Reese Algorithm) was built into ANSYS [10 , 11]. It should be noted that the use of components of the stress tensor for calculating the equivalent stress by the known formula

$$\sigma_e = \frac{\sqrt{2}}{2} \sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2)}$$

is impossible for the probabilistic PSD- analysis because all the output of this analysis are random variables that can not be simply summed, squared, or taken root of.

Equivalent stresses calculated by ANSYS are difficult to use for the analysis of time to fatigue failure, since the distribution of equivalent size on von Mises hypothesis differs from the distribution of the tensor components (normally distributed). This is due to the nonlinearity of the von Mises hypothesis. Moreover, it is more difficult for a given hypothesis to obtain the distribution of the amplitude or local extremas of equivalent values needed to calculate fatigue.

Nevertheless, calculations of equivalent stresses outputted with ANSYS, allows to allocate nodes FE mesh in which time to fatigue failure is most likely maximum. For each radioelement a number of these "dangerous " sites are selected for which the time to failure is calculated from the algorithm. Node in which the time

to failure will be minimal, will determine the time to failure for electronic components. An abbreviated algorithm implements this approach.

Calculation of time to fatigue failure of structure by impact.

Shocks occur during transportation, installation, operation, or under the action of the shock wave. During impact, load is applied to structural elements for a short time, but often with significant amplitude, thereby accelerations and stresses on structural elements may attain large values, and cause various damage until mechanical failure. The intensity of the impact depends on the shape, amplitude, and duration of the pulse impact.

Fatigue failure on impact can occur in a very short time due to the high stresses and large plastic deformations. Therefore, the high-cycle fatigue model is not applicable here.

The use of a low-cycle fatigue encounters certain difficulties. This is due, first, to the nature of the non-cyclic loads at the nodes of finite element model of the design, as the form of a shock pulse can be arbitrary; and secondly, the simulation of plastic deformation in ANSYS can be very complex.

Non-cyclic loading involves the use of algorithms for counting cycles. Each cycle is derived from the loading history, contributes to the accumulation of fatigue damage. To calculate the total damage from all known cycles, one of the rules of summation of fatigue damage is applied. Here, Palmgren - Miner rule (see formula 1.7) is most common.

Since the state of stress on impact is not a single-axis at an arbitrary point of the structure, to apply the model of low-cycle fatigue we need to calculate the equivalent value of the total strain. Von Mises criterion, by default, is used more often. Von Mises equivalent strain is calculated using the formula:

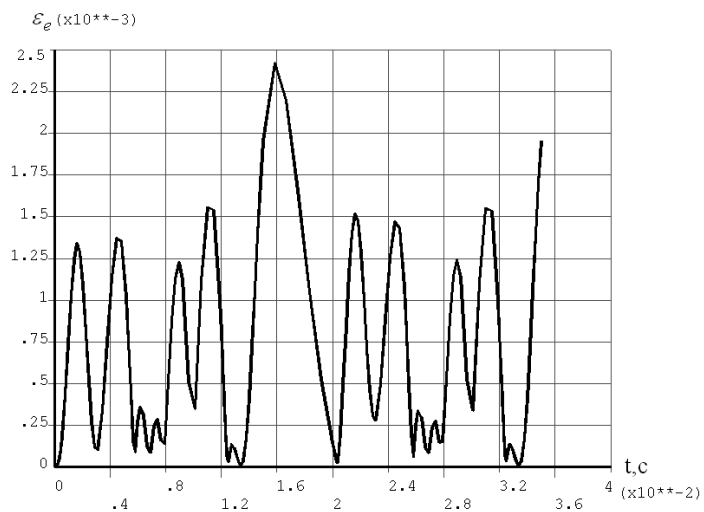
$$\varepsilon_e = \frac{1}{\sqrt{2}(1+\nu)} \sqrt{(\varepsilon_x - \varepsilon_y)^2 + (\varepsilon_y - \varepsilon_z)^2 + (\varepsilon_z - \varepsilon_x)^2 + \frac{3}{2}(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{xz}^2)}.$$

However, this formula is not suitable for the calculation of fatigue loading with variable amplitude, since the negative “compressive” component of the total strain is lost after applying the square root [9]. Therefore, when calculating fatigue, the so-

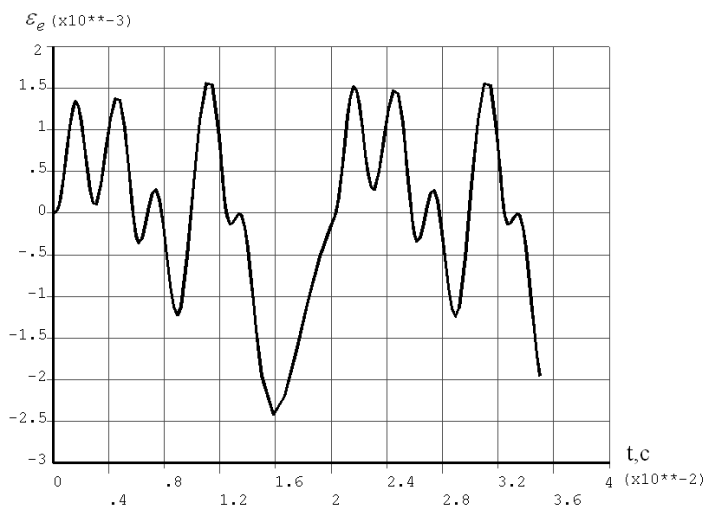
called von Mises equivalent strain is applied with sign consideration. The sign of complete deformation is defined by the sign of the maximum absolute principal stress. The formula becomes:

$$\varepsilon_e = \frac{\sigma_1}{|\sigma_1|(1+\nu)\sqrt{2}} \sqrt{(\varepsilon_x - \varepsilon_y)^2 + (\varepsilon_y - \varepsilon_z)^2 + (\varepsilon_z - \varepsilon_x)^2 + \frac{3}{2}(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{xz}^2)}$$

Figure 1.15 shows graphs of related equivalent strains in FE mesh node on impact from time ((a) - without the sign, (b) - with the sign). The second graph, as can be seen from the figure, most accurately describes the history of loading.



(a)



(b)

Figure 1.15 - Relationship of equivalent strains in the FE mesh node with time

Counting cycles.

Earlier, the main features of rainflow algorithms serving to count cycles with different amplitudes in a random loading history were reviewed. It was noted that the results of different versions of the algorithm may vary slightly.

Next, we considered a variant of the algorithm, which is the standard of the American Society for Testing and Materials (ASTM). Let X - current segment (range) of loading history, Y - adjacent to the previous segment (range) X of loading history, S - initial node (local maximum or minimum) of loading history. The algorithm consists of six steps:

1. Read the next node (local maximum or minimum) in the current array of nodes. If there are no nodes, go to step 6.
2. If the array of current nodes has less than three nodes, go to step 1. Generate range X and Y , using the last three nodes in the array.
3. Compare absolute range values X and Y .
 - 3a. If $X < Y$, go to step 1.
 - 3b. If $X > Y$, go to step 4.
4. If the segment (range) Y contains the initial node S , proceed to step 5; otherwise, count segment Y as one cycle, remove the top which make up section Y from the current array of nodes and move on to step 2.
5. Count segment Y as half the cycle, remove the first node of segment Y from array of current nodes; the initial node S will now be equal to the second node of segment Y ; go to step 2.
6. Count each remaining segment (range) as half the cycle.

To illustrate the algorithm, we consider the steps shown in the example of the loading history in Figure 1.16 (a).

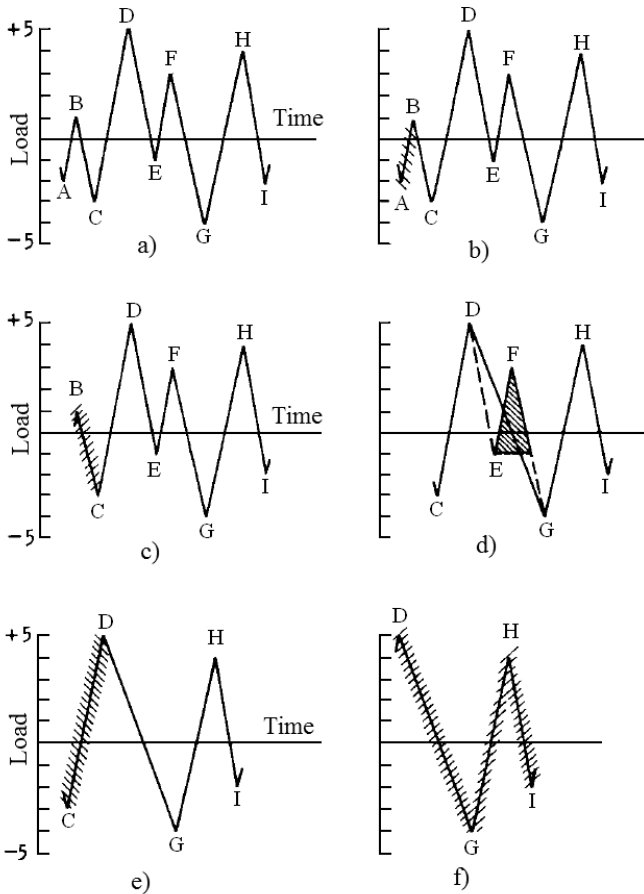


Figure 1.16 - Rainflow algorithm (standard ASTM)

1. $S = A$; $Y = |A-B|$; $X = |B-C|$; $X > Y$. Y contains S , which is node A . Count $|A-B|$ as a half cycle and eliminate the node A from current array of nodes; $S = B$ (Figure 1.16 (b))
2. $Y = |B-C|$; $X = |C-D|$; $X > Y$. Y contains S , which is node B . Calculate $|B-C|$ as a half cycle and eliminate B from current array of nodes; $S = C$ (Figure 1.16 (a))
3. $Y = |C-D|$; $X = |D-E|$; $X < Y$
4. $Y = |D-E|$; $X = |E-F|$; $X < Y$

5. $Y = |E-F|$; $X = |F-G|$; $X > Y$. Count $|E-F|$ as one cycle and eliminate nodes E and F from current array of nodes (Figure 1.16 (d)). It should be noted that the full cycle is formed with segments E-F and F-G.

6. $Y = |C-D|$; $X = |D-G|$; $X > Y$. Y contains S, which is node C. Count $|C-D|$ as a half cycle and eliminate node C from the current array of nodes ; $S = D$ (Figure 1.16 (d)).

7. $Y = |D-G|$; $X = |G-H|$; $X < Y$

8. $Y = |G-H|$; $X = |H-I|$; $X < Y$. There are no more nodes for counting.

9. Count $|D-G|$ as half cycle, $|G-H|$ as half cycle, and $|H-I|$ as half cycle (Figure 1.16 (e)).

Cycles counting results can be summarized in a table:

Range	Number of cycles	Events
10	0	
9	0.5	D-G
8	1.0	C-D,G-H
7	0	
6	0.5	H-I
5	0	
4	1.5	B-C,E-F
3	0.5	A-B
2	0	
1	0	

Correction for plasticity.

To account fully for deformations at impact with the plasticity, we need to perform nonlinear transient analysis (Nonlinear Transient Analysis) in ANSYS. Nonlinear analysis of complex structures in ANSYS is time consuming. During analysis, problems may arise with the convergence of results. This leads to the need to reduce the integration step, the convergence conditions change, etc. Furthermore, each material will need its own stress-strain curve of points, and also a model of hardening.

There are approximate methods, which are based on the linear analysis the results of which are elastic deformation, that allow one to obtain corresponding values of total strain with plastic component.

The most widely used method for the correction of plasticity is Neuber rule [12].

Neuber rule is written as:

$$\varepsilon \sigma = K_t^2 e S,$$

where ε – local total strain,

σ – local stress,

e – nominal elastic deformation,

S – nominal stress,

K_t – theoretical stress concentration factor.

If the FE mesh is fine enough to account for any effects of stress concentration, then $K_t = 1$ may be used.

Now Neuber rule is:

$$\varepsilon \sigma = e S.$$

Combining Neuber rule and model plastic deformation of materials under cyclic loading (see formulas 1.5 and 1.6), we can form a system of equations and solve for unknown variables. Solving the system:

$$\begin{cases} \varepsilon \sigma = e S \\ \varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{K'} \right)^{\frac{1}{n'}} \end{cases},$$

we obtain a corrected value of the total strain ε and stress σ on the basis of the relevant nominal values for e и S during the initial loading.

With further cyclic loading, corresponding to stable hysteresis curve, range adjusted stresses and strains are determined from the system of equations:

$$\begin{cases} \Delta \varepsilon \Delta \sigma = \Delta e \Delta S \\ \Delta \varepsilon = \frac{\Delta \sigma}{E} + 2 \left(\frac{\Delta \sigma}{2K'} \right)^{\frac{1}{n'}}. \end{cases}$$

In carrying out the substitution, the system can be reduced to two nonlinear equations:

$$\begin{aligned} \left[\frac{\sigma}{E} + \left(\frac{\sigma}{K'} \right)^{\frac{1}{n'}} \right] \sigma - eS &= 0 \\ \left[\frac{\Delta \sigma}{E} + 2 \left(\frac{\Delta \sigma}{2K'} \right)^{\frac{1}{n'}} \right] \Delta \sigma - \Delta e \Delta S &= 0 \end{aligned} \quad (1.13)$$

Both equations are solved by numerical methods (such as the method of bisection, Newton's method, etc.).

General algorithm for calculating the time to fatigue fracture on impact.

Summarizing all the above, we present a general algorithm for calculating the time to fatigue fracture on impact, periodically repeated after a specified period of time.

The algorithm consists of the following steps:

1. Obtaining, using ANSYS post-processor, dependencies of equivalent deformations in the FE mesh nodes in time, which form the history of loading in these nodes. These dependences characterize the state of stress in the FE mesh nodes for a specified amount of shock pulses. Equivalent deformations with a sign according to von Mises are determined in reference with section.

2 . Obtaining extremes for dependency equivalent deformations of time. This step is necessary because the loading history must contain only the extremes of equivalent values for counting cycles in step 3.

3 . Counting cycles, using a rainflow algorithm.

The result of this step is an array with elements containing equivalent strain range values, and also the number of cycles corresponding to a particular range:

$$\langle (\Delta e_1, n_1) \quad (\Delta e_2, n_2) \quad \dots \quad (\Delta e_{k-1}, n_{k-1}) \quad (\Delta e_k, n_k) \rangle,$$

where Δe_k – k -th equivalent strain range, isolated from the loading history during cycle counting;

n_k – number of counted cycles with range Δe_k .

4. Plasticity correction for counted cycles.

Elongations Δe_k , calculated in the previous step correspond to the nominal elastic strains in equation 1.13 and linked with the range of nominal stresses according to Hooke's law:

$$\Delta S_k = E \Delta e_k$$

Adjusted stresses and strains are determined by solving equations 1.13 for all elements of the array generated in the previous step.

The result of this step is the adjusted array:

$$\langle (\Delta \varepsilon_1, n_1) \quad (\Delta \varepsilon_2, n_2) \quad \dots \quad (\Delta \varepsilon_{k-1}, n_{k-1}) \quad (\Delta \varepsilon_k, n_k) \rangle,$$

where $\Delta \varepsilon_k$ – k -th equivalent strain range after correction for plasticity;

5. Calculating the number of cycles to failure based on the equation of low cycle fatigue (formula 1.4) for each deformation range.

The result is an array:

$$\langle N_{f_1} \quad N_{f_2} \quad \dots \quad N_{f_{k-1}} \quad N_{f_k} \rangle,$$

where N_{f_k} – number of cycles to failure for k -th equivalent strain range (i.e., the number of cycles to failure under cyclic loading with the range of $\Delta \varepsilon_k$).

6. Calculation and total damages from counted cycles with Palmgren-Miner rule:

$$D(T_0) = \sum_{i=1}^k D_i = \sum_{i=1}^k \frac{n_i}{N_{f_i}},$$

where $D(T_0)$ – fatigue damage accumulated in a specified number of shock pulses in a particular node of finite element mesh.

7. Calculation of time to failure. Knowing fatigue damage $D(T_0)$, accumulated during a given number of shocks, as well as the total duration T_0 of these shocks, the time to failure T can be determined by simple ratio. Given that fatigue failure is related to damage $D(T) = 1$, the ratio becomes:

$$\frac{1}{D(T_0)} = \frac{T}{T_0}.$$

Therefore:

$$T = \frac{T_0}{D(T_0)}.$$

It should be noted that the calculation is performed according to the algorithm for all FE mesh nodes, which is necessary to determine the time to fatigue. As mentioned earlier, fatigue is a surface phenomenon, therefore, it is sufficient to account for finite element mesh only on the surface FE mesh nodes.

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).

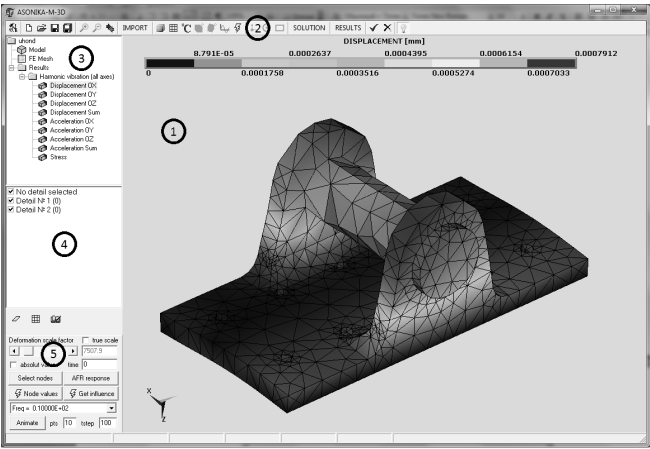


Figure 2.1 - Window subsystem during operation

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.

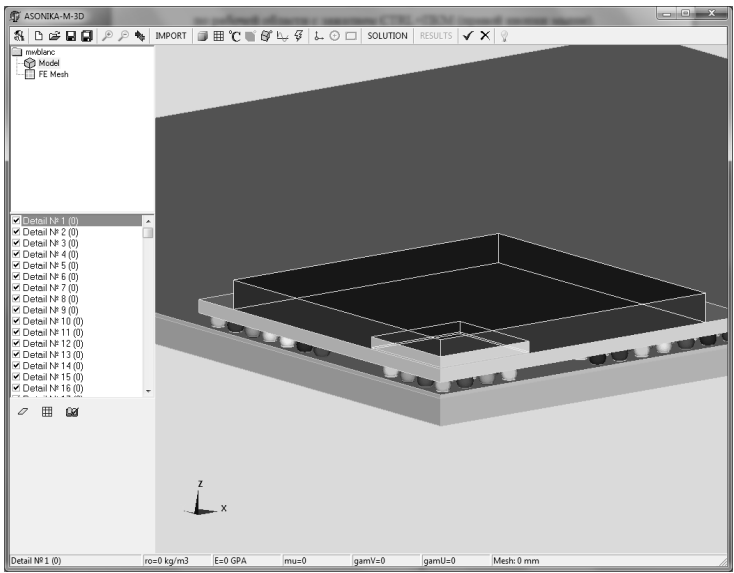


Figure 2.2 - Highlighting of parts in the list (highlighted part number 1)

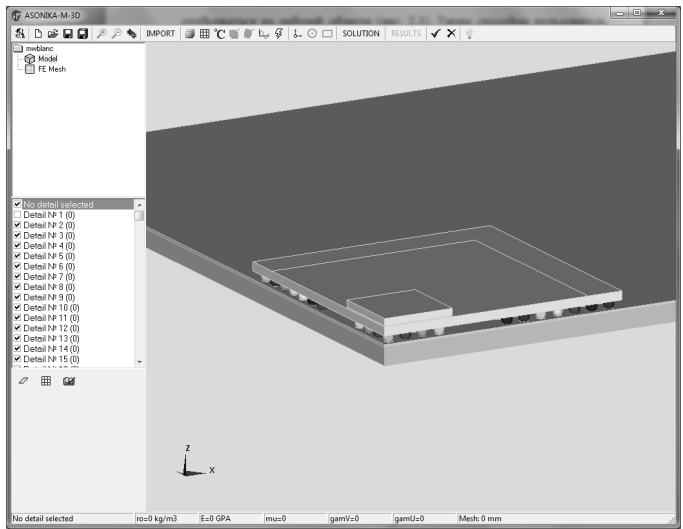


Figure 2.3 - Hiding of parts (hidden part number 1)

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.

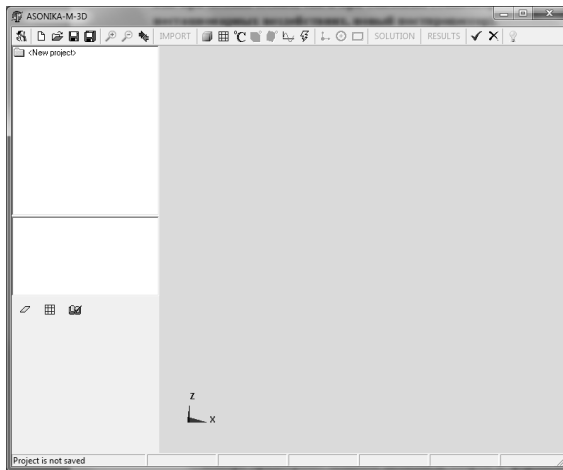


Figure 2.4 - Window subsystem after its launch

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);

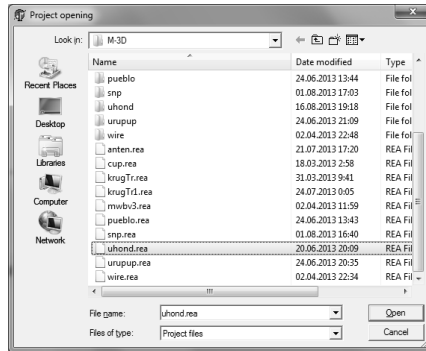


Figure 2.5 - Opening the project

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

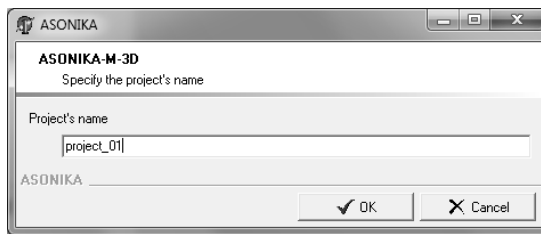


Figure 2.6 - Setting the name for the new project

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

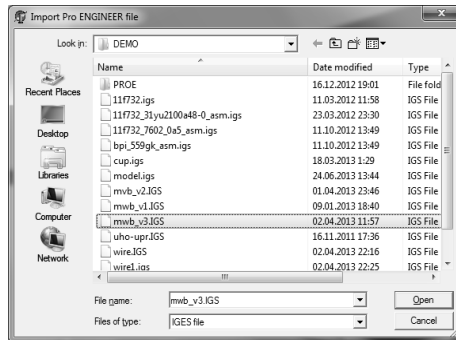


Figure 2.7 - Import model window

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).

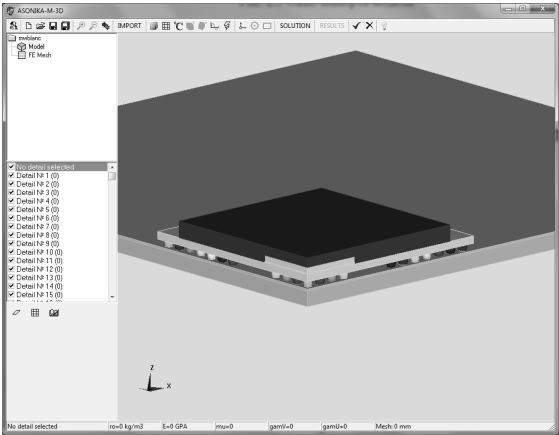


Figure 2.8 - The display of the imported model of the microcircuit

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).

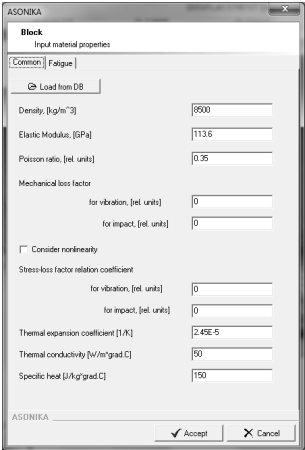


Figure 2.9 - Setting the parameters of the material (“General” tab)

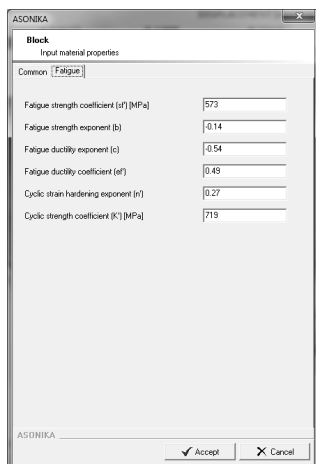


Figure 2.10 - Setting the parameters of the material ("Fatigue" tab)

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.

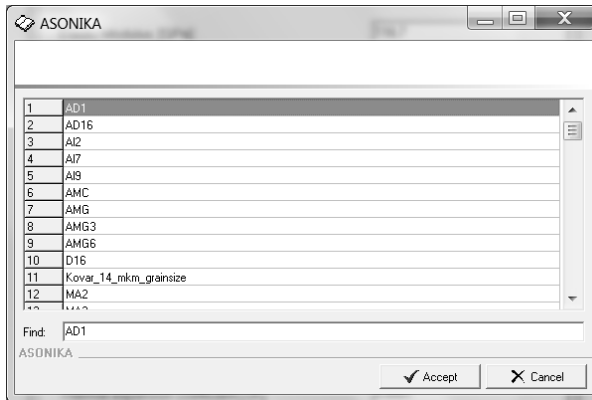


Figure 2.11 - Selection of the material database

Attention : If you do not specify a separate piece of material parameters, then this part will be used by the parameters set for the entire structure (using  button on the toolbar).

2.3.1.3 Setting the mechanical effects

The interface for setting mechanical effects is activated by  button on the toolbar (Figure 2.12).

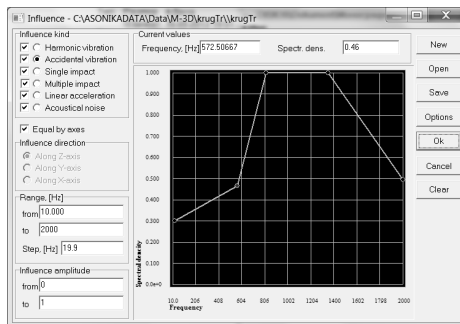


Figure 2.12 - Interface to specify mechanical effects

Depending on the type of impact, frequency or time dependent acceleration is entered in dot-sequence. For random vibration frequency dependence of spectral density of acceleration is entered. You want to specify a range and a step beforehand with which the solver will choose a point within the range for calculations and write

results to a file. Setting the step will determine to a great extent the accuracy and computation time.

2.3.1.4 Setting endurance conditions

The software has the ability to set constraint types:

- line constraint,
- round constraint,
- rectangular constraint.

Constraining the line is set by pointing the mouse cursor on the line and pressing Shift + LMB. The line is highlighted in black color (Figure 2.13)

Defined endurance in this way can be removed by pressing Shift + RMB when you hover your mouse over the fixed line.

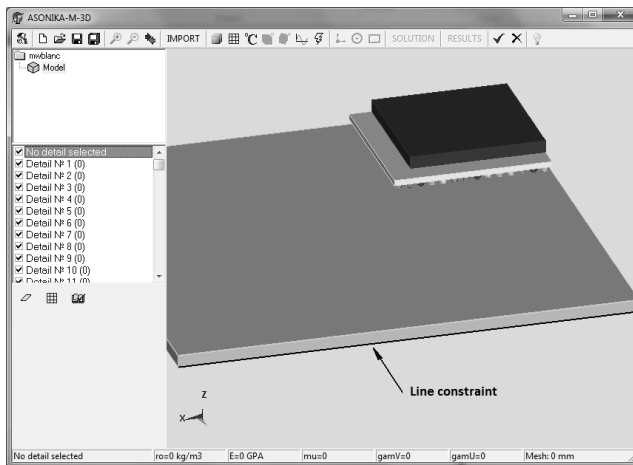


Figure 2.13 - Constraining the line

Round and rectangular constraints are created in the following way:

1. Select the plane by clicking LMB (the plane gets highlighted in blue).
2. Button  coordinate system (CS) setting becomes available. You have to press this button.
3. Points within the selected plane are displayed in which you can create the CS.

When you click the LMB on one of the points - it creates the CS, with the X-axis of the new CS co-directed (or tangent) with one of the lines coming out of the point. If you click on the point again, the X-axis is co-directed with another line, starting from the point.

4. After the establishment of the CS button, settings of round  and rectangular  fixtures become available. You must click on one of these buttons.

5. Clicking with LMB indicates the starting point of attachment, then hold down the mouse button and moving the cursor to increase / reduce the size of attachment (Figure 2.14).

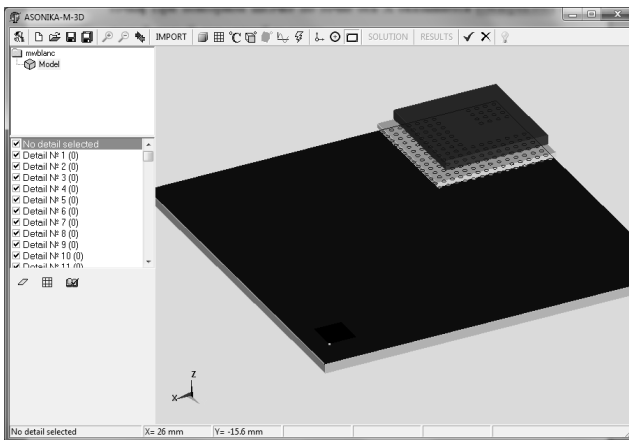


Figure 2.14 - Rectangular constraint

6. When a constraint is created, you can change the settings by double-clicking on the attachment: a dialog box appears in which you must specify the required parameters (Figure 2.15).

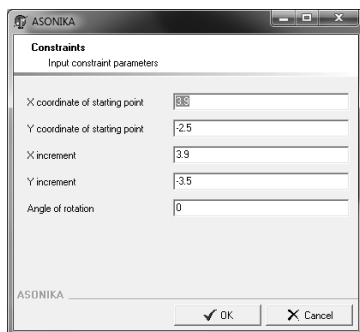


Figure 2.15 – Constraints parameters

7. To remove the attachment select it with LMB and press Delete.

All manipulations with attachments are possible only in creating fixities or CS within a specific plane.

2.3.1.5 Setting thermal boundary conditions

To carry out thermal analysis (steady-state or transient), you must set the correct thermal boundary conditions.

In ASONIKA-M-3D, it is possible to set the volume and surface thermal boundary conditions using the buttons  and  toolbars, respectively. In order for the buttons  and  to become active, first select the surface or volume. These buttons call the corresponding dialog boxes (see Figures 2.16 and 2.17), which indicate the type of boundary conditions and parameters.

For surfaces thermal boundary conditions are defined with the following types:

- convection with the external environment,
- the temperature source,
- radiation into the environment,
- convection + radiation into the environment.

For volumes (individual items), thermal boundary conditions are defined with the following types:

- source of power,
- source of heat.

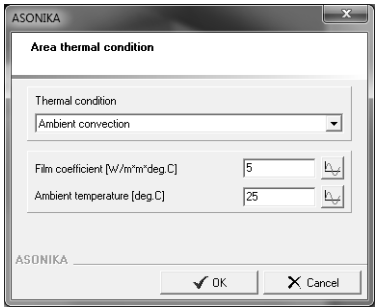


Figure 2.16 - Surface thermal boundary condition

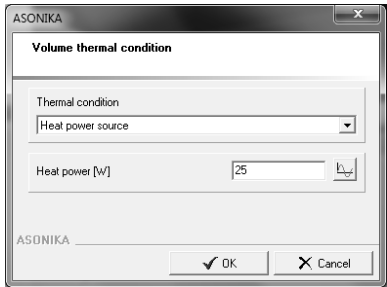


Figure 2.17 - Volume thermal boundary condition

If transient thermal calculation is planned, it should be borne in mind that the main parameters of the boundary conditions (heat capacity, sustained temperature, heat transfer temperature coefficient) can be time-dependent. In this case, you must enter the appropriate graph distribution, using the input interface (Figure 2.18), with a button , located to the right of the corresponding parameter.

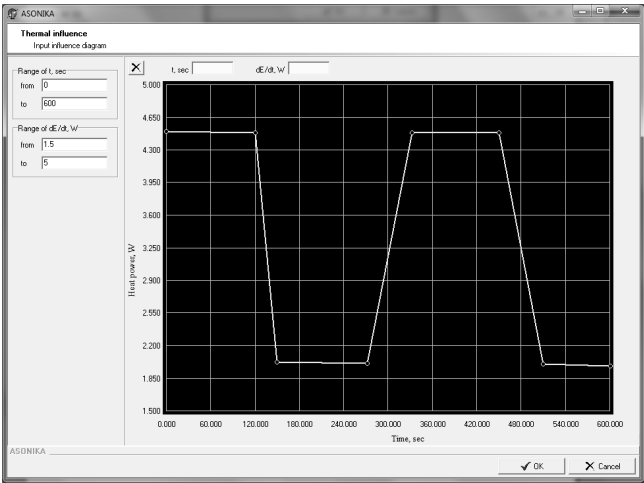


Figure 2.18 - Input parameters for the plot of the boundary conditions

2.3.1.6 Setting the finite element partitioning

The command  on the toolbar is used to specify the size of the finite element mesh for the entire structure (Figure 2.19). In this case, you can manually enter the average size of a finite element, or let the program automatically select the size (set the corresponding check mark). The larger the grid, the faster the calculation is carried out, but with less accuracy.

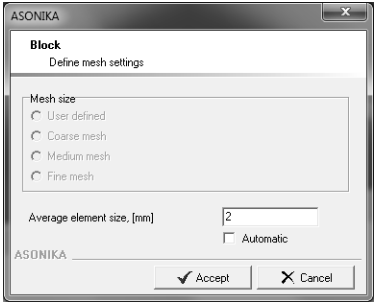


Figure 2.19 - Setting parameters of the mesh

Besides being able to specify the size of the finite element mesh for the entire structure, using the  button on the toolbar, the program has the ability to set the mesh settings for individual parts. To do this, first select the part (in any method

described in paragraph 2.1). Parameters for the mesh are entered by pressing the  button located in the area below the list of parts.

Warning: if the parts are not set with mesh parameters, then this part will be used by the parameters set for the entire structure (using the  button on the toolbar).

2.3.1.7 Generation of the finite element mesh

Before carrying out the calculations, we need to obtain FE model construction. FE mesh generation process started by the  command from the toolbar.

If, during the generation of an error occurs, the program will inform the user with part number that could not be broken down into finite elements. To successfully complete the partition, the user has the ability to change grid settings for this part, usually by reducing its size (see section 2.3.1.6), and start the process of re-meshing. This sequence of user actions can be repeated (including for other items that can not be immediately divided into finite elements) until the program displays a message about successful partition, after which the FE mesh is displayed on the screen (Figure 2.20).

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)

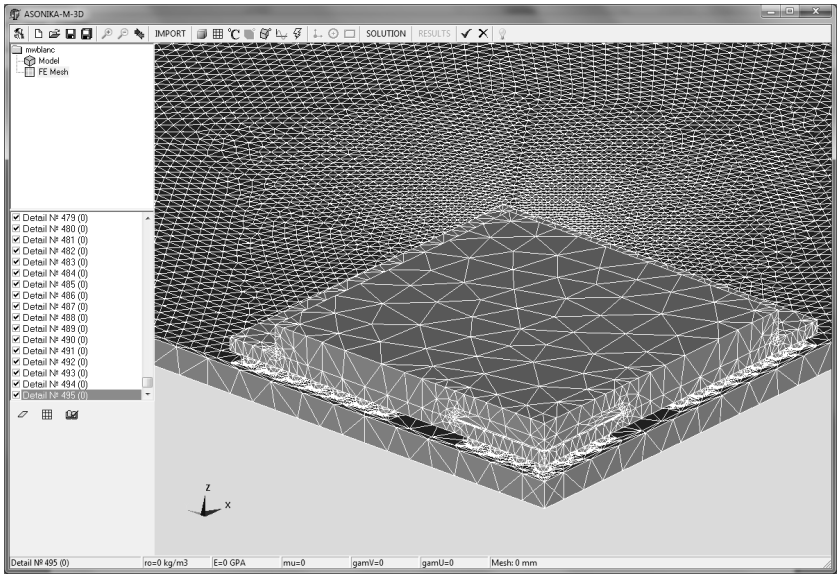


Figure 2.20 - FE mesh

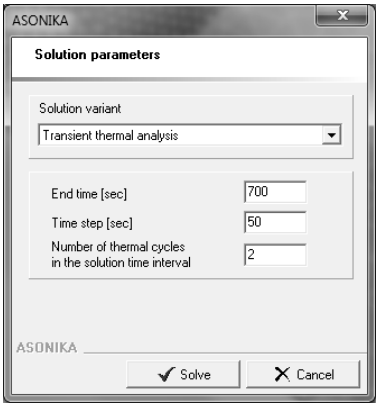


Figure 2.21 - Types and parameters of the calculations

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.

Before starting the calculation, the user has to perform a sequence of actions in the preprocessor, which depend on the type of the proposed calculation.

Mechanical calculations (for vibration and shock) require the following actions in the preprocessor:

1. Setting the parameters of the material.

For all design and / or for individual parts, material parameters are given which are required for mechanical calculations (density, elastic modulus, Poisson's ratio, mechanical loss factor (MLF)). If nonlinear analysis is required (taking into account the dependence of the mechanical loss factor from stresses), you need to also specify "stress MLF index" and set the checkbox "take into account the nonlinearity" for individual parts and / or for the entire structure. Nonlinear calculations will be carried out, provided that at least one piece is set with "stress MLF index" and "take into account the nonlinearity" is checked.

To calculate the time to fatigue failure under mechanical loads, fatigue material parameters must also be specified.

2. Setting the mechanical effects.

A graph corresponding to the mechanical action (harmonic vibration , random vibration, single or multiple impact) is introduced.

3. Setting endurance conditions

4. Setting the finite element partitioning

For all designs and / or to individual components, finite element mesh parameters are specified.

5. Generation of finite element mesh.

Calculation of thermal expansion requires the following steps in the preprocessor:

1. Setting the parameters of the material.

For the whole model and / or individual parts, material parameters necessary for calculating the thermal expansion are defined (modulus of elasticity, Poisson's ratio, coefficient of thermal expansion).

2. (Optional) Setting endurance conditions.

5. Setting thermal boundary conditions

The temperature distribution is defined: all items specified with initial and final temperatures.

4. Setting parameters of finite partition,

For the whole model and / or to individual parts, parameters of finite element mesh are specified.

5. Generation of finite element mesh.

Steady-state thermal calculations require the following steps in the preprocessor:

1. Setting the parameters of the material.

For all designs and / or individual parts, parameters of materials are given. For steady-state heat transfer calculations, it is sufficient to specify only the value of thermal conductivity.

2. Setting thermal boundary conditions.

Volume and / or surface thermal boundary conditions are specified.

3. Setting finite element partition.

For all designs and / or to individual components, finite element mesh parameters are specified.

4. Generation of finite element mesh.

Transient thermal calculations require the following steps in the preprocessor:

1. Setting parameters of the material.

For all designs and / or to individual components, material parameters are specified.

Transient thermal analysis requires specification of the following material parameters:

- thermal conductivity,
- specific heat,
- density.

If thermal stresses are required for calculation, the following parameters are also specified:

- modulus of elasticity,
- poisson's ratio,
- coefficient of thermal expansion

Calculation of fatigue in (cyclic) thermal stress conditions require specification of the parameters of low-cycle fatigue:

- coefficient of fatigue (sf'),
- fatigue strength (b),
- fatigue ductility (c),
- fatigue ductility coefficient (ef'),
- cyclic hardening index (n'),
- cyclic elastic modulus (K').

2 . (Optional) Setting endurance conditions.

3 . Setting thermal boundary conditions.

The volume and / or surface thermal boundary conditions are specified. Basic boundary conditions (heat capacity, sustained temperature, ambient temperature, heat transfer coefficient) may be time dependent. In this case, enter the appropriate relationship graph.

Initial temperature for the entire structure and / or for individual parts is specified.

4. Setting finite element partition.

For all designs and / or individual components, parameters of finite element mesh are specified.

5. Generation of finite element mesh.

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)

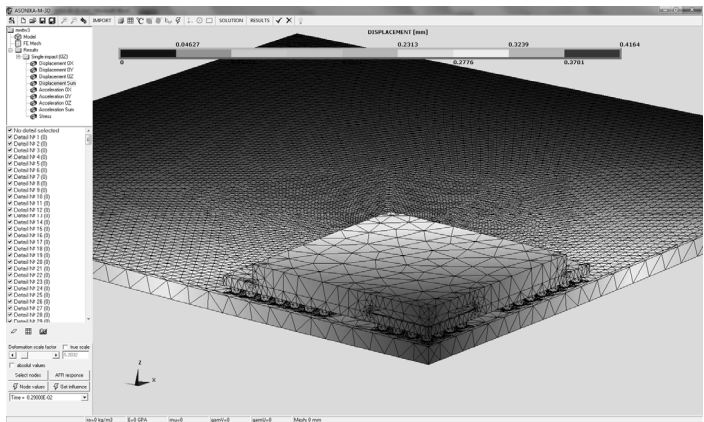


Figure 2.22 - Total displacement field

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).

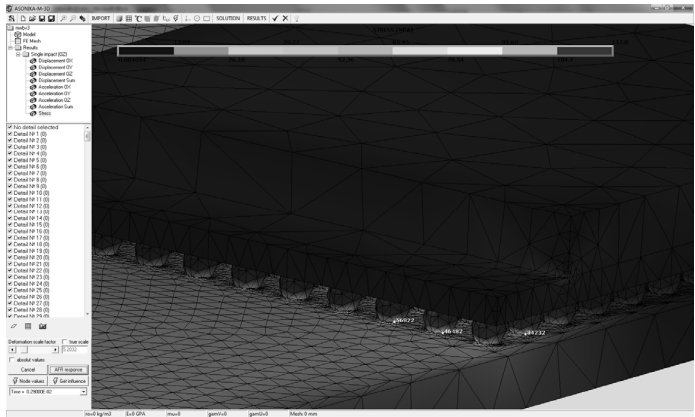


Figure 2.23 - Selection of 3 nodes

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

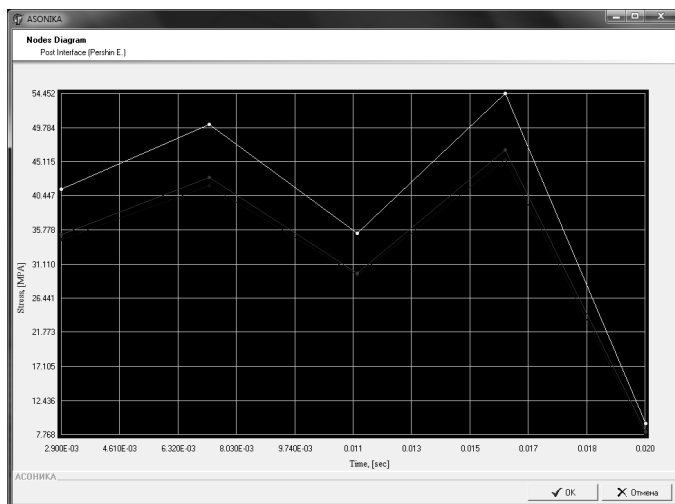


Figure 2.24 - Node charts

2.3.3.3 Output of absolute and relative values of the output variables

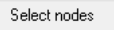
In ASONIKA-M-3D analysis on mechanical effects, and above all - displacements can be viewed in two ways: on the one hand - these are deflections of the structure, provided that the place of attachment is at rest (have zero displacement), on the other hand - these are deflections of structure points, 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.

2.3.3.4 Receiving variables of output values at the nodes of finite element mesh

To obtain the values of the output variables in the nodes of a finite mesh, the user should take the following steps:

1. In the object tree, select the desired output value.

2. Using the  **Select nodes** command on postprocessor panel, the nodes, values of the output variables that are interested by the user, are selected.
3. Command  **Node values** on the panel postprocessor displays a dialog box with the values of the output variables in the selected nodes (Figure 2.25)

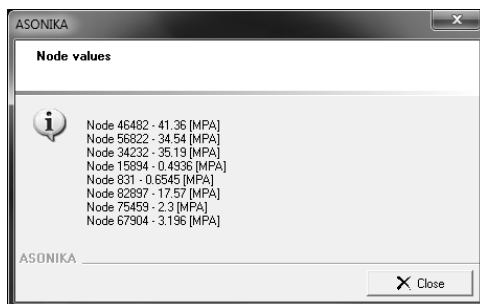


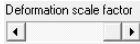
Figure 2.25 - Display of node values

4. *Results* – these are stress fields, displacements, accelerations, temperatures, and other variables, which are the results of analysis of structures on the thermal and mechanical effects. All of the results are grouped by the type of analysis, i.e. for each analysis, there is a different set of results, contained under the node tree with the name of the analysis.

Postprocessor panel, to the left of the work area at the bottom of the window contains a set of commands that are used to manipulate calculation results: creating frequency response graphs and Nyquist plots, nodes selection, inspection of results in the individual nodes, output of absolute values of the results, the animation of results, etc.

2.3.3.5 Inspection of the deformed structure taking into account the enhancement factor

The deformed state of the actual design, if the values of stress and deformation are too large, usually is not amenable to visual observation. In ASONIKA-M-3D, there is an opportunity to strengthen the deformations so as to assess the shape and direction of the deflection of structures under mechanical loads.

By moving the slider  the deformation can be strengthened or reduced. In this case, the coefficient of enhanced deformation is displayed in a special box of postprocessor panel (Figure 2.26).

If the strain state needs to be seen in true scale (coefficient of enhancement = 1), "true scale" needs to be ticked in the panel postprocessor.

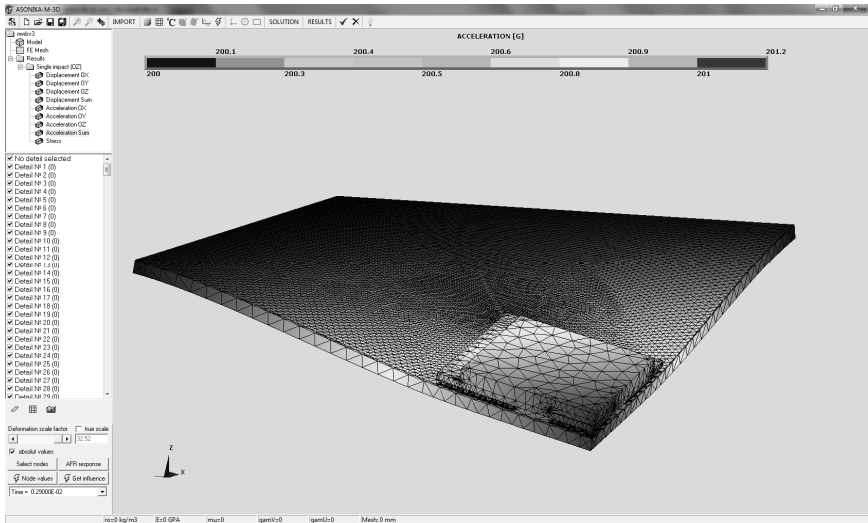


Figure 2.26 - Enhanced deformations

2.3.3.6 Response animation design on the mechanical effects

In ASONIKA-M-3D, animation provides an opportunity to present results of the calculation of harmonic vibrations and shock impacts.

Animation allows one to simulate a real experiment, see oscillation form of the structure, and thus better understand the physics of the process.

Before using an animation, it is desirable to set an enhancement factor (see section 2.3.3.5), in order for harmonic motion or response from the impact to be available for inspection.

The command  on the panel of the post processor, allows to run the process of results animation. In addition, the parameters of the animation can be changed as well:

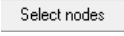
- (for harmonic vibrations only) setting “pts” determines the number of times (or frames) within a period of harmonic oscillation structure to be used for animation.
- parameter “tstep” sets the amount of delay in milliseconds between successive frames.

2.3.3.7 Export of mechanical effects to external files

Subsystem ASONIKA-M-3D is a software package of ASONIKA, designed for complex analysis of structures found in electronics.

The command  on panel postprocessor, is designed to transfer data between the subsystems, which belong to the ASONIKA software package. When activated, a text file is created containing a description of mechanical effects (harmonic vibration , random vibration, or shock), which would cause the same response in the selected node of a finite element mesh, in the event of applicability to the particular node, that is, if design had a set endurance point at this node.

In addition to the activation command , used to create the file containing description of the effect, following conditions are required:

- initially, using the command  on panel postprocessor, a node is selected (choose only one node) in output effect is required;
- it’s necessary to select the axis along which the output effect is sent.

In the subsystem ASONIKA-TM, which is part of ASONIKA software package, text file created this way can be used as an input to calculate the impact of printed circuit assemblies.

2.3.4 Important features of ASONIKA-M-3D functions

- The subsystem uses ANSYS software package to carry out core calculations, interacting with it through a system of automatically generated macros.

- To speed up the calculations, the ANSYS kernel can take advantage of multiprocessor data processing, when a computing machine meets all of the requirements. This feature can be disabled in ASONIKA-M-3D settings.
- The complexity of the calculated models is limited only by the capabilities and resources of the ANSYS kernel and machine specifications.
- Post-processor, to speed up access to the results, reads data in large blocks . If the possibility of the computer's memory is limited, it is possible to set the maximum size of the data in the program data, which are simultaneously loaded into RAM.

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.

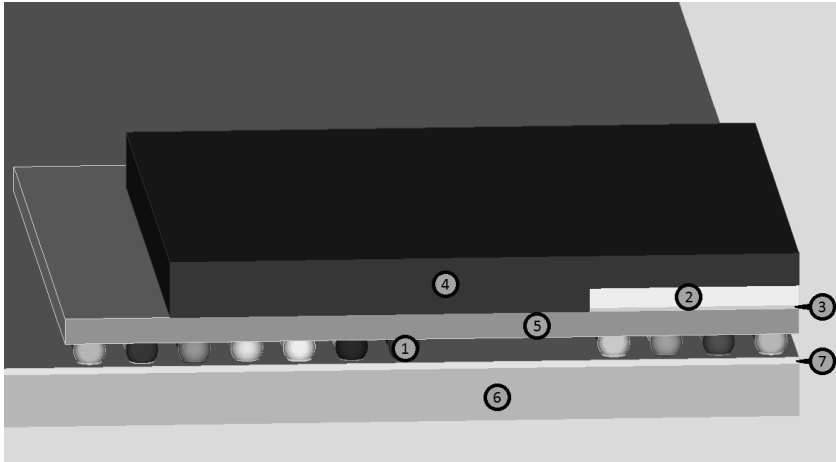


Figure 3.1 - Structure of the solid model

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 Work in the preprocessor (preparing models for the calculation)

3.1.1 Creating a Project

Start the subsystem ASONIKA-M-3D and create a new project. To do this, press the  button on the toolbar and enter the project name in the dialog box (Figure 3.2)

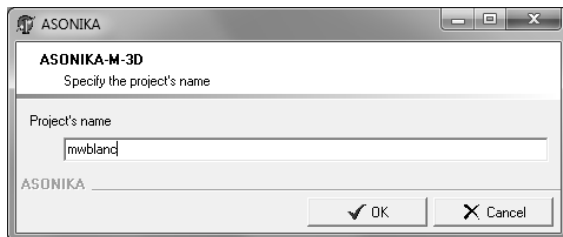


Figure 3.2 - Creation of a project

3.1.2 Importing a model

1. Once the project is created, import the model by pressing the **IMPORT** button on the toolbar. In the dialog box that appears (Figure 3.3) specify the name and file type.

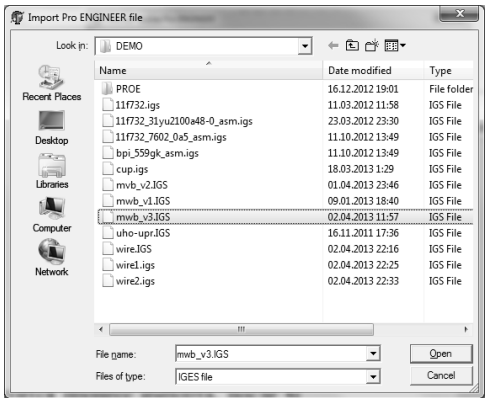


Figure 3.3 - Selection of an IGES file

Start the import process, after which the model is displayed on the screen (Figure 3.4).

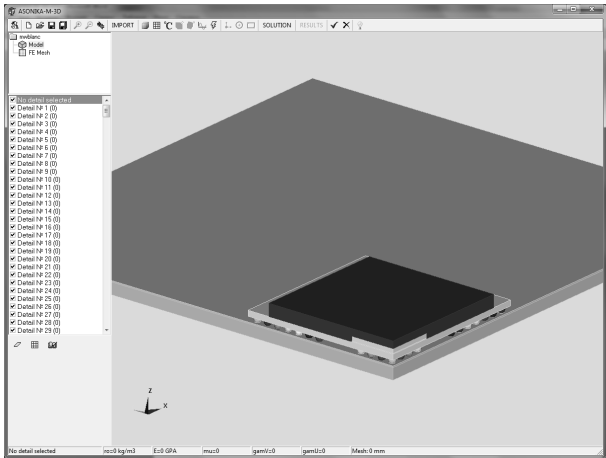
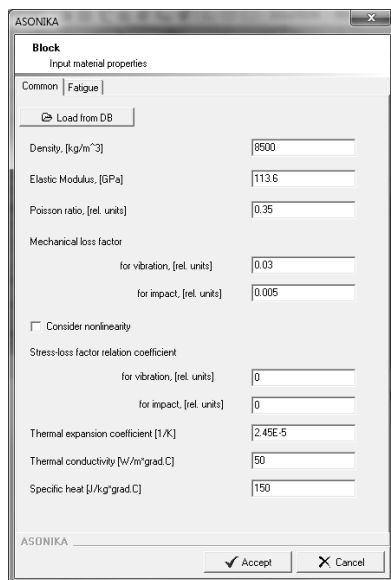


Figure 3.4 - Imported model

3.1.3 Specifying materials

The model of the microcircuit with solder balls consists of various parts (solder balls, silicon chip, chip mounting, epoxy compound, substrate chip, circuit board, a copper layer on the board, etc.) with different physical properties.

To begin with, we define material parameters for the entire structure by default. Press the  button on the toolbar and in the dialog box that appears on the "General" tab (Figure 3.5), enter the values of the parameters:



The screenshot shows a dialog box titled 'ASONIKA' with a sub-header 'Block' and 'Input material properties'. It has two tabs: 'Common' (selected) and 'Fatigue'. A 'Load from DB' button is at the top left. The 'Common' tab contains the following parameters and their default values:

Parameter	Value
Density, [kg/m ³]	9500
Elastic Modulus, [GPa]	113.6
Poisson ratio, [rel. units]	0.35
Mechanical loss factor	
for vibration, [rel. units]	0.03
for impact, [rel. units]	0.005
☐ Consider nonlinearity	
Stress-loss factor relation coefficient	
for vibration, [rel. units]	0
for impact, [rel. units]	0
Thermal expansion coefficient [1/K]	2.45E-5
Thermal conductivity [W/m*grad.C]	50
Specific heat [J/kg*grad.C]	150

At the bottom, there are 'Accept' and 'Cancel' buttons.

Figure 3.5 - "General" default settings for the material

These parameters (the values are taken for tin-lead alloy) will be used for solder balls, since due to the large number, it is impractical to set the parameters for each output separately. Since most interest from the point of view of fatigue come from solder balls, go to the tab " Fatigue " (Figure 3.6) and set the parameters of low-cycle fatigue.

Input material properties	
Common	Fatigue
Fatigue strength coefficient (σ_f') [MPa]	573
Fatigue strength exponent (b)	-0.14
Fatigue ductility exponent (c)	-0.54
Fatigue ductility coefficient (σ_f'')	0.43
Cyclic strain hardening exponent (n')	0.27
Cyclic strength coefficient (K') [MPa]	718

Figure 3.6 - "Fatigue" default parameters for the material

For the remaining parts of the model, that is, excluding solder ball, the material parameters are set separately.

For example, select the item that represents a silicon chip with a click ALT + LMB (Figure 3.7), then select the , button located in the area below the list of parts, to enter the parameters of the material (Figure 3.8).

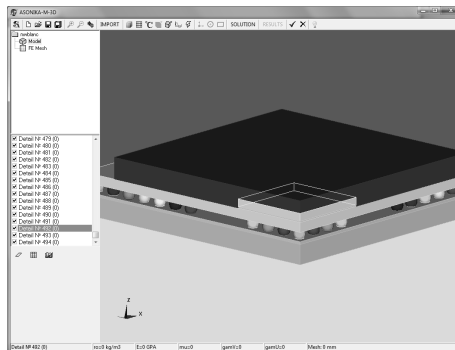


Figure 3.7 - Selected part (silicon chip)

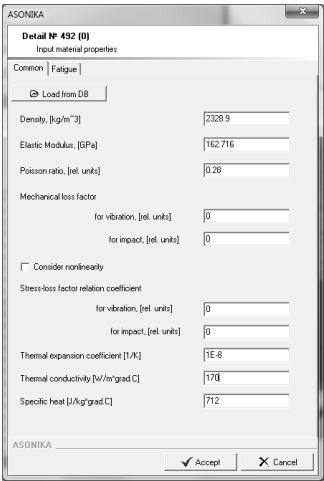


Figure 3.8 - Material parameters for the silicon chip

Next, select the part that represents the epoxy compound (Figure 3.9), and similarly set material parameters (Figure 3.10).

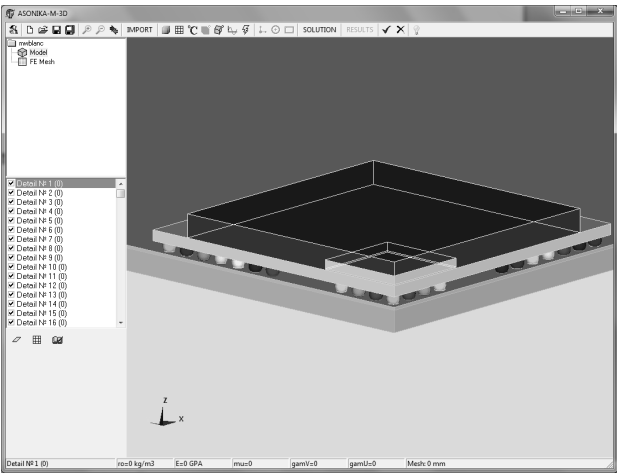


Figure 3.9 - Selected part (epoxy compound)

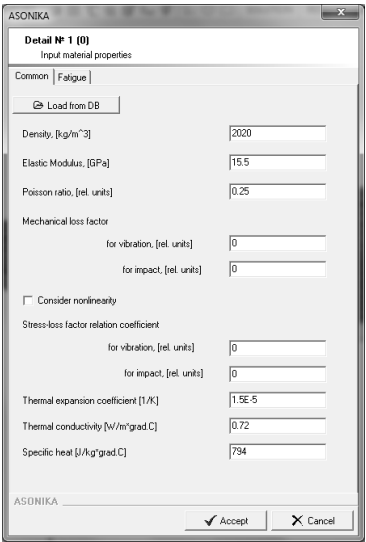


Figure 3.10 - Material parameters for epoxy compound

Select the part that represents endurance of the chip (Figure 3.11) and set material parameters (Figure 3.12).

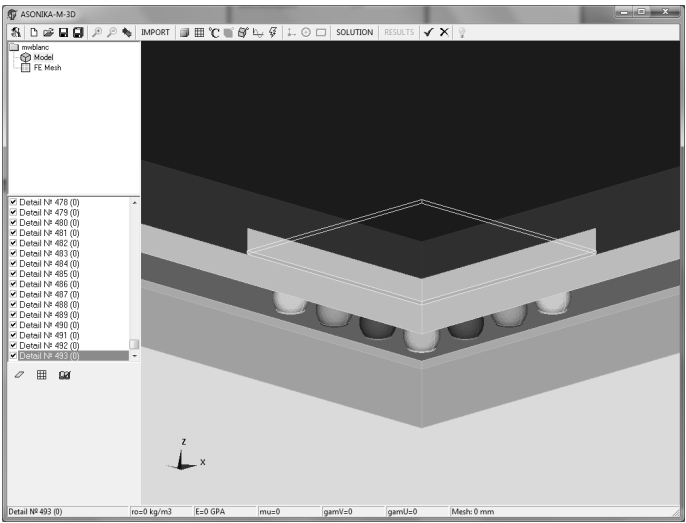


Figure 3.11 - Selected part (constraining the chip)

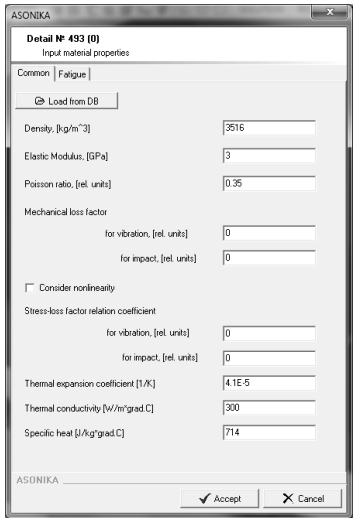


Figure 3.12 - Material parameters for endurance of the chip

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

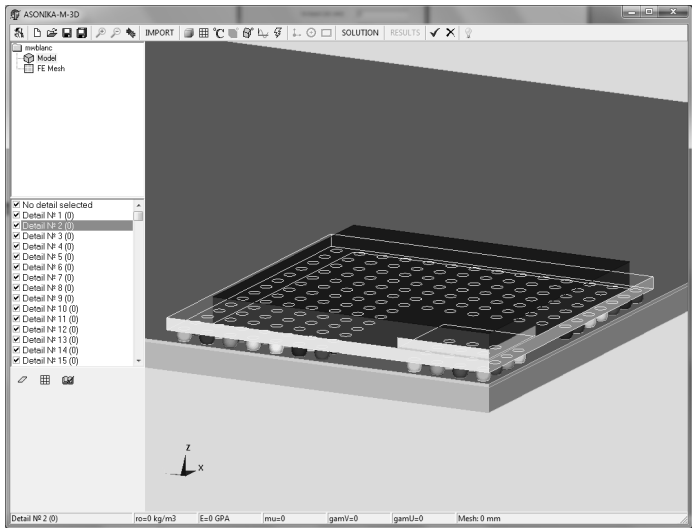


Figure 3.13 - Selected part (chip substrate)

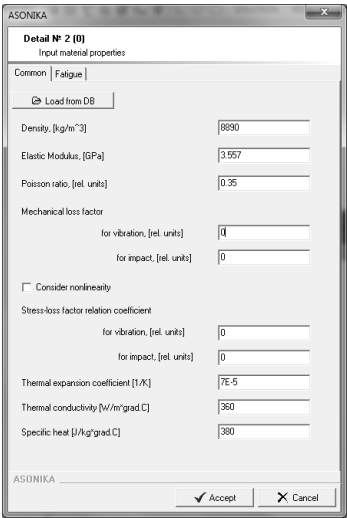


Figure 3.14 - Material parameters for the chip substrate

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

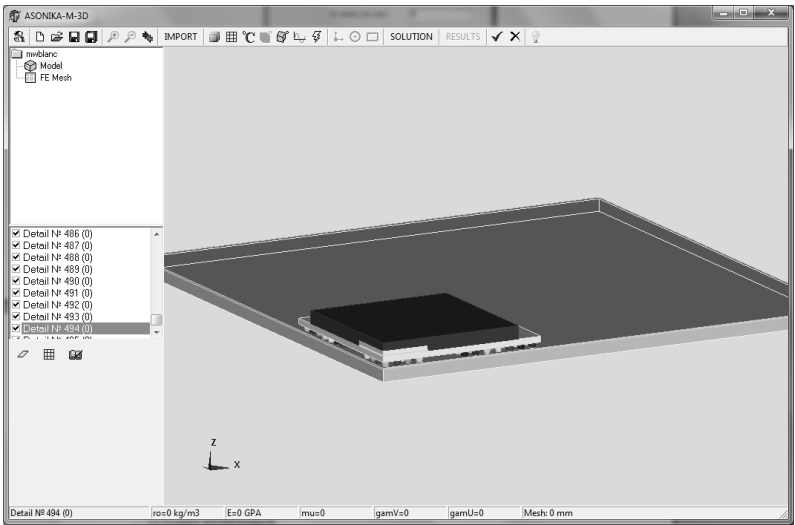


Figure 3.15 - Selected part (printed circuit board)

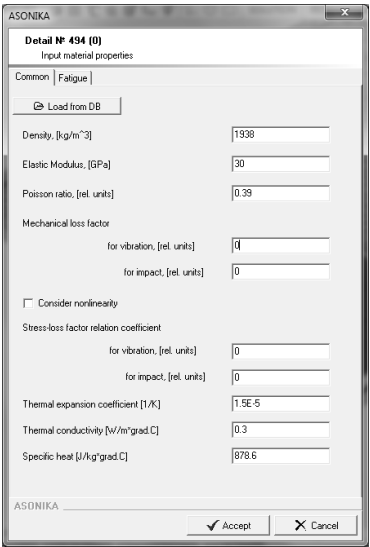


Figure 3.16 - Material parameters for printed circuit board

Finally, select the part that represents the copper layer on the board (Figure 3.17) and set material parameters (Figure 3.18).

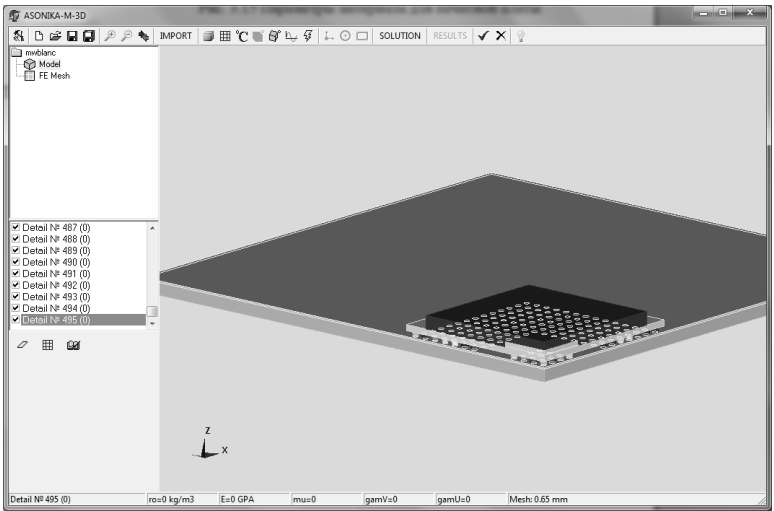


Figure 3.17 - Selected part (copper layer on the board)

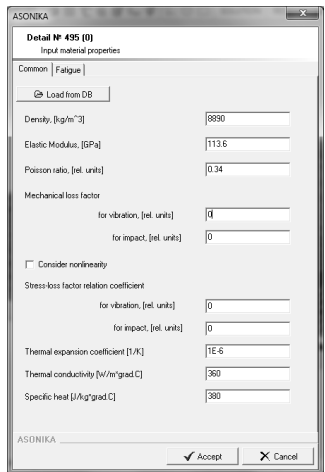


Figure 3.17 - Selected part (copper layer on the board)

3.1.4 Setting the mechanical effects

We run the interface of inputting mechanical effects by pressing the  button on the toolbar.

The harmonic vibration given by the graph of acceleration versus frequency (Figure 3.19). Frequency effect - from 10 to 1200 Hz, the amplitude of acceleration is constant and is within a range of 5g.

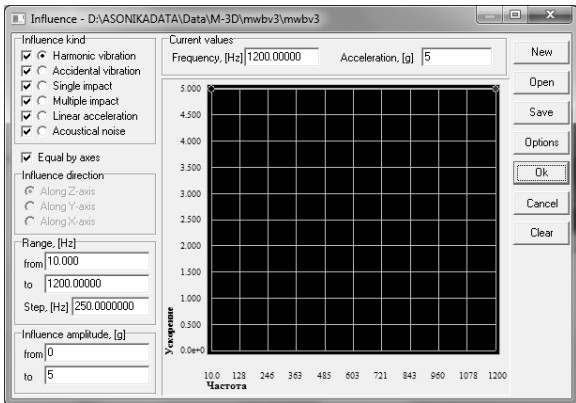


Figure 3.19 - Harmonic vibration

We introduce a graph of random vibration (Figure 3.20). Frequency range - from 10 to 2000 Hz, spectral density acceleration - from 0 to 1 g * g / Hz.

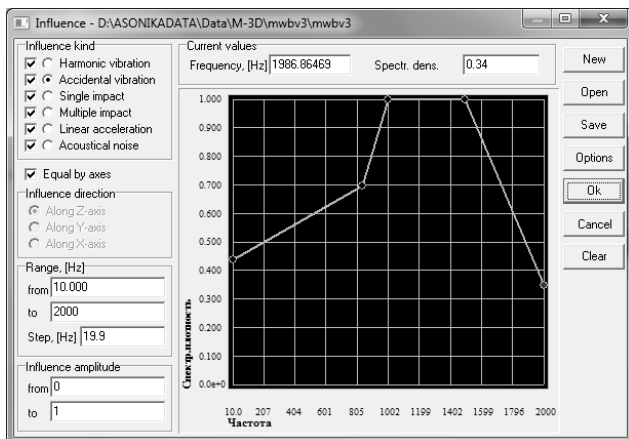


Figure 3.20 - Random Vibration

Shock impulse is given by the following graph (Figure 3.21):

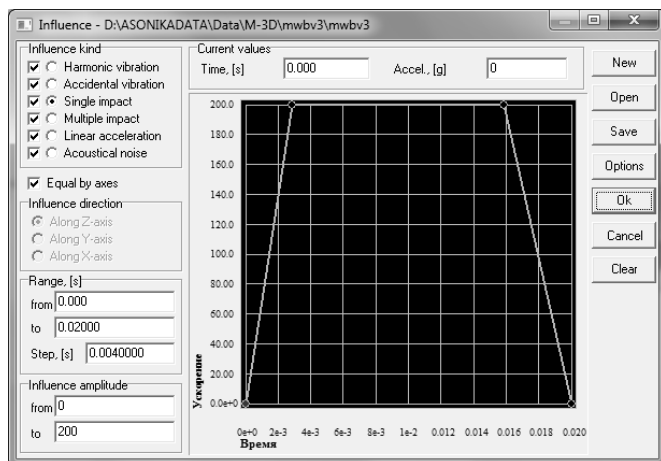


Figure 3.21 - Shock impulse

Temporal range of the shock - from 0 to 20 msec, the acceleration is increased from 0 to 200g.

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).

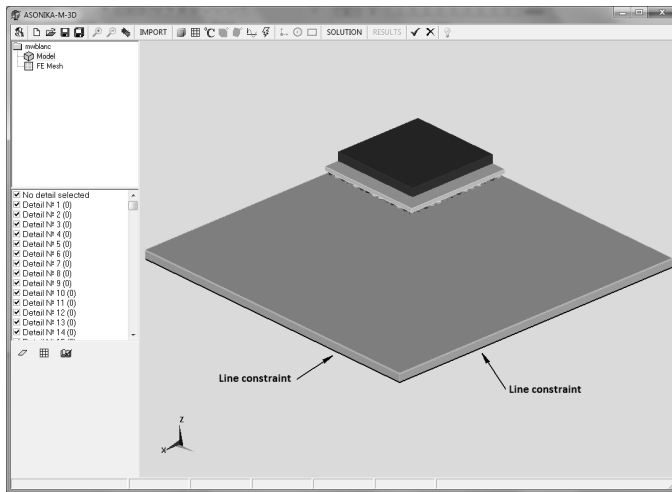


Figure 3.22 - Constraints

Note: The symmetry planes are not used for constraints.

3.1.6 Setting thermal boundary conditions

Using LMB (in sequence) to select external surface of the structure. Then press the  button to set thermal boundary condition for the outer surfaces. Introduce the parameters of thermal boundary condition (Figure 3.23):

- type of boundary conditions - convection + radiation into the environment,
- convective heat transfer coefficient = $10 \text{ W} / \text{m}^2 \cdot \text{Grad}$
- emissivity = 1,
- slope = 1,

- ambient temperature = 20 °.

Thereafter, the surface of thermal boundary conditions are conventionally marked by the mesh (Figure 3.24):

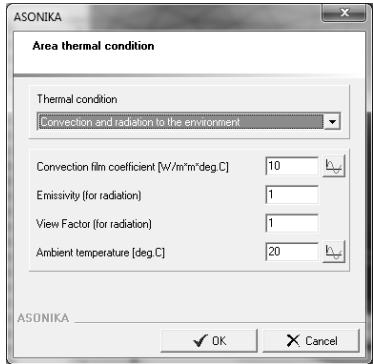


Figure 3.23 - Thermal boundary condition

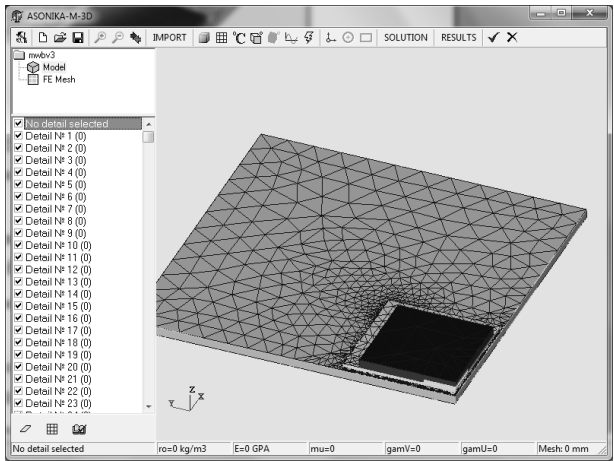


Figure 3.24 - The model after setting thermal conditions

Select the part that represents silicon chip, click ALT + LMB (Figure 3-25). Then press the  button to set the volumetric thermal boundary condition for the part. Introduce the parameters of the thermal boundary condition (Figure 3.26):

- type of boundary condition - the source of power,
- power output = 4.5 (for steady-state thermal calculation).

For transient thermal analysis, option "Thermal Power" will depend on time. Press the  button to generate the graph of time dependence of this parameter. First, specify a range of time (x-axis) and power (y-axis) using the appropriate boxes (Figure 3.27). Then the schedule is entered on the points.

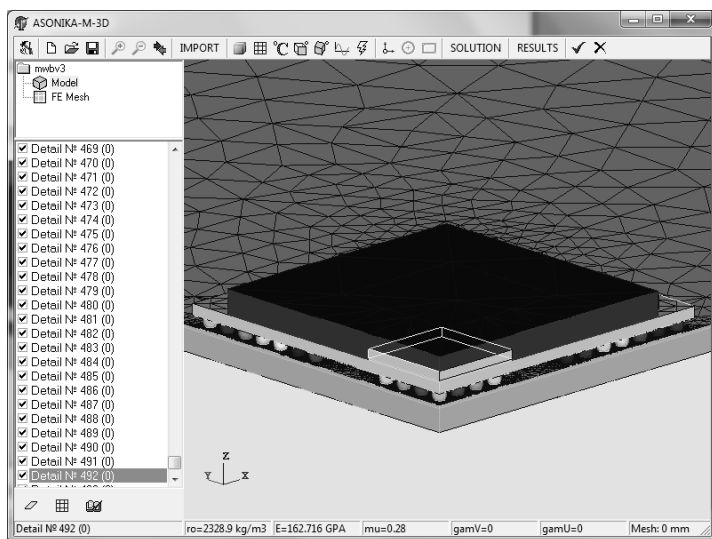


Figure 3.25 - Selected part

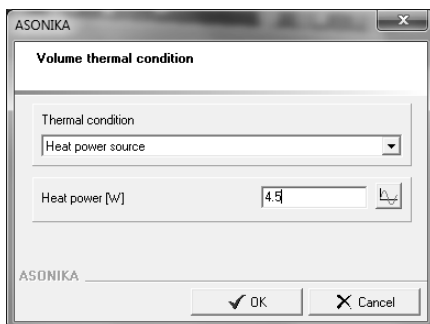


Figure 3.26 - Thermal boundary condition parameters

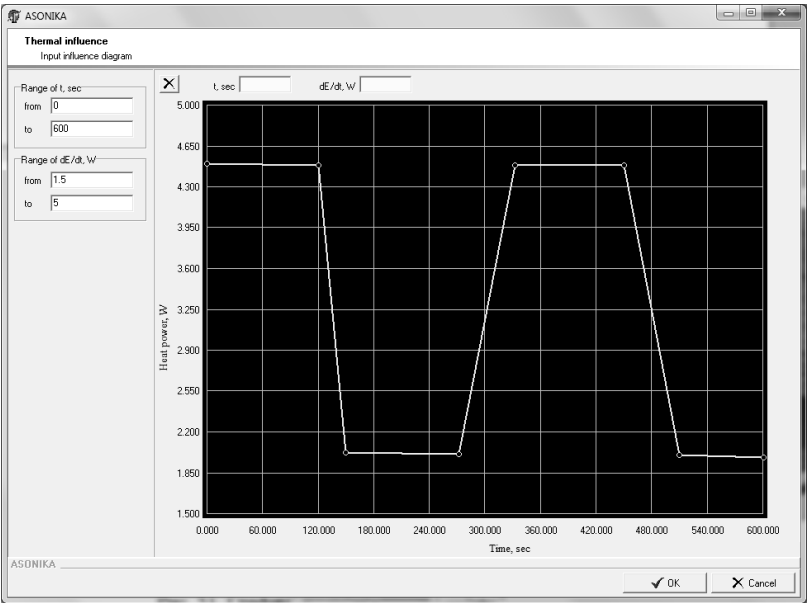


Figure 3.27 - Power heat profile

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.

We set the initial temperature for the entire structure at once.

Press the  button on the toolbar, and then in the dialog box, press the  button and enter the default initial temperature (Figure 3.28):

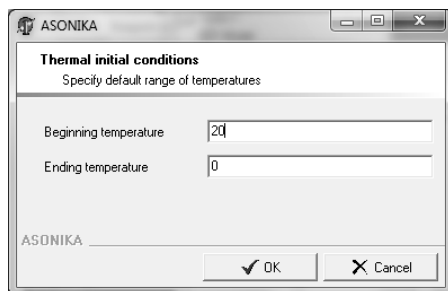


Figure 3.28 - Initial temperature

3.1.7 Setting finite element partitioning

Press the  button on the toolbar and after dialog box appears (Figure 3.29), check "automatic" to let the program select the size of the grid for the entire structure automatically.

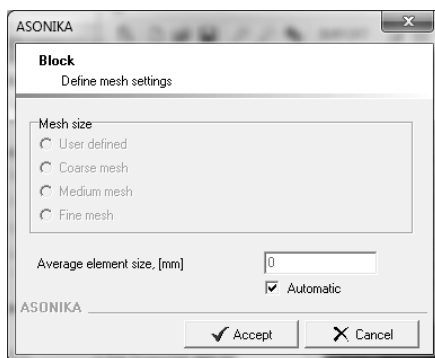


Figure 3.29 - Automatic mesh size

3.1.8 Generation of the finite element mesh

Start the process of finite element mesh generation with  command on the toolbar. The resulting mesh is shown in Figure 3.30

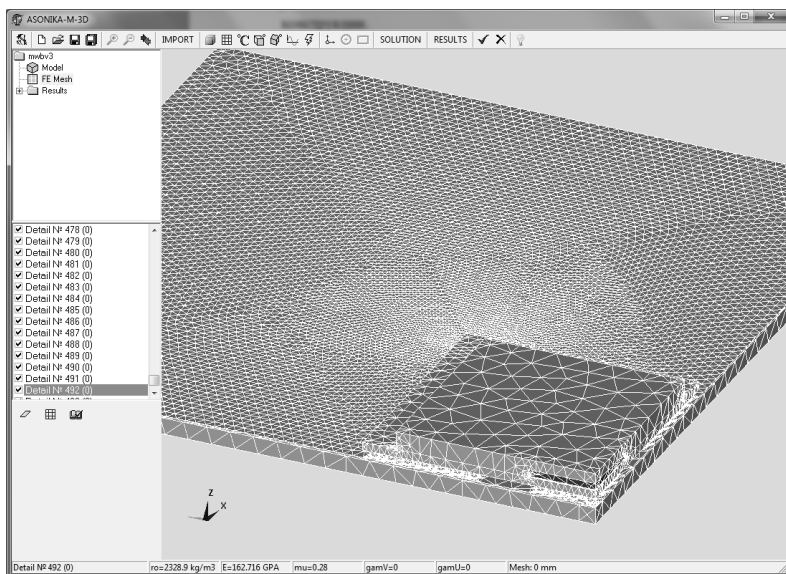


Figure 3.30 - FE mesh

3.2 Calculations

Data entered in the preprocessor are adequate for the following types of calculations:

- harmonic vibration,
- random vibration
- single shock,
- steady-state thermal analysis,
- transient thermal analysis.

Using the **SOLUTION** command on a toolbar, run the 5 series of calculations. The calculation parameters are shown in Figure 3.31-3.35.

Mechanical effects are directed along Z axis, i.e. perpendicular to the plane of the board:

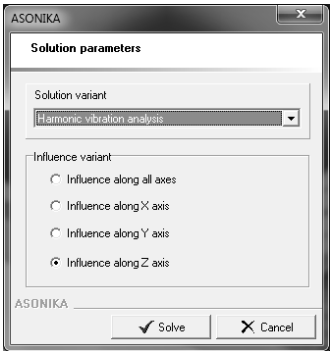


Figure 3.31 - Parameters for harmonic vibration analysis

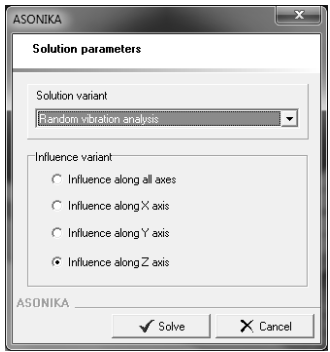


Figure 3.32 - Parameters for random vibration analysis

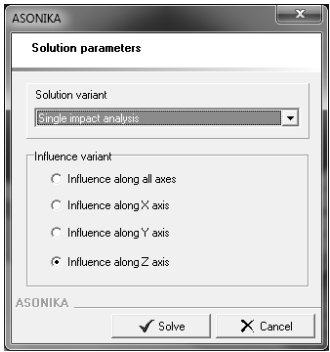


Figure 3.33 - Parameters for single impact analysis

Steady-state thermal analysis does not require a set of parameters:

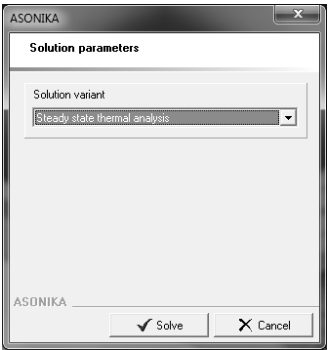


Figure 3.34 - Parameters for steady-state thermal analysis

Transient thermal analysis is carried out in the time interval of 600 seconds at intervals of 50 seconds. At this interval, the number of simulated thermal cycles is two - these are two cycle variations of power heat in the silicon chip.

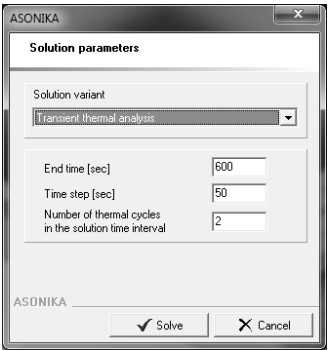


Figure 3.35 - Parameters of transient thermal analysis

3.3 Review and results

The following briefly discusses the results and draws some conclusions about stability of the structure due to mechanical and thermal effects.

3.3.1 Viewing the results of harmonic vibration analysis

Calculation of a harmonic vibration showed that the interval 10-1200 Hz has one resonant frequency of 435 Hz. At this frequency, the maxima of accelerations of displacements and stresses whose fields are shown in Figures 3.36-3.38.

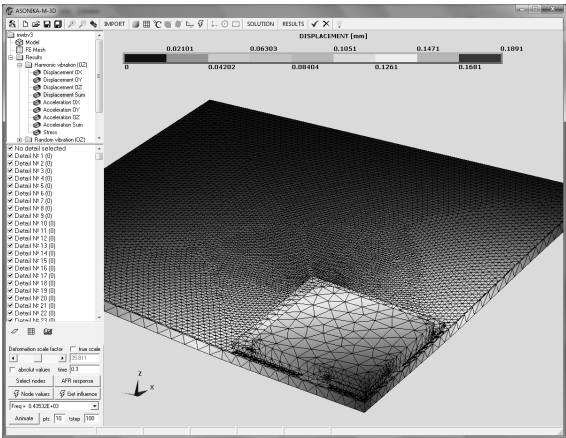


Figure 3.36 - Field of total displacements

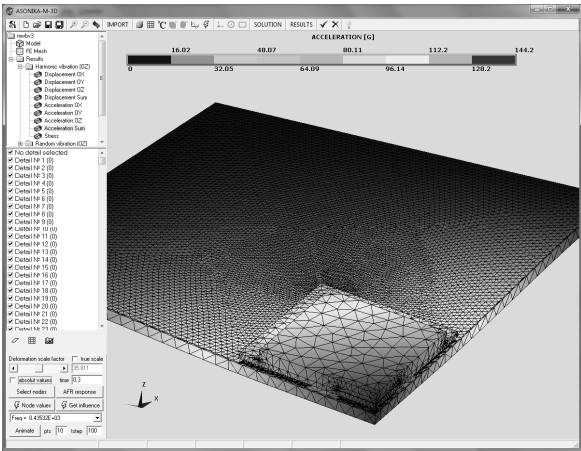


Figure 3.37 - Field of the total acceleration

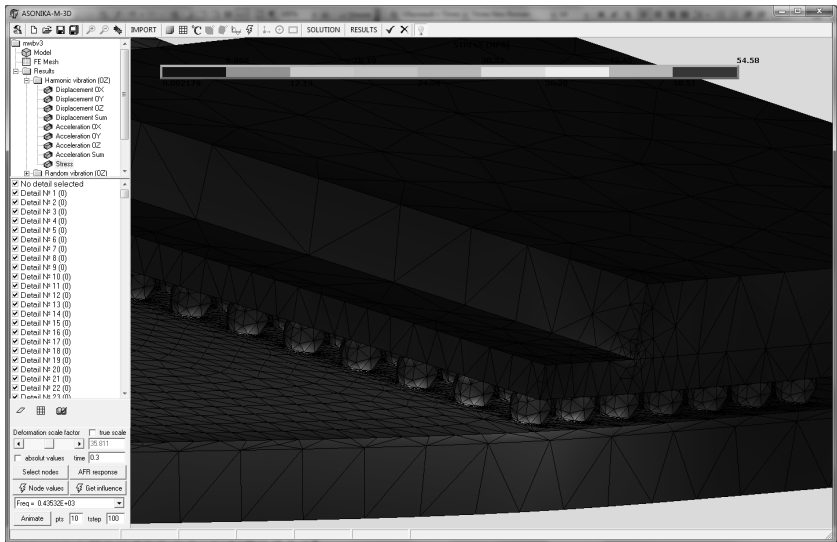


Figure 3.38 - Stress field

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

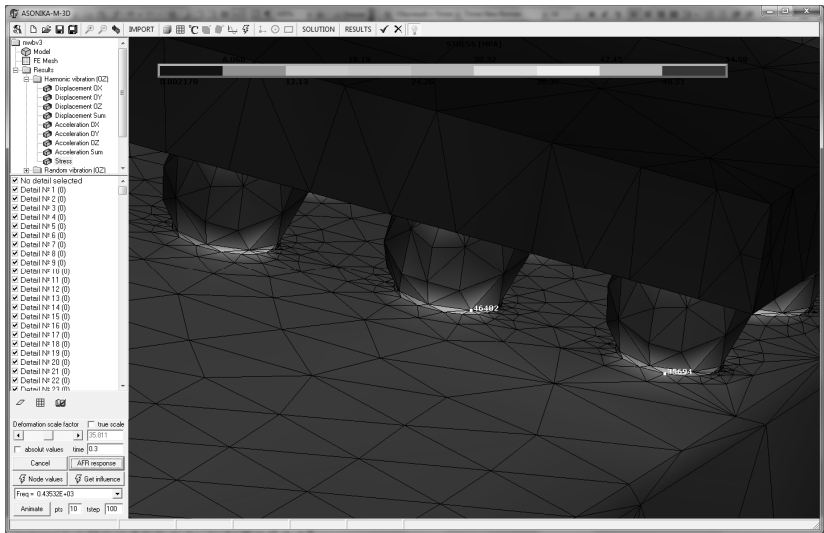


Figure 3.39 - Selection of nodes

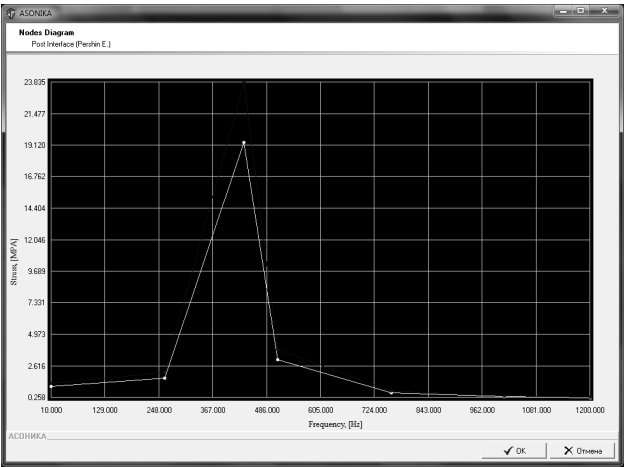


Figure 3.40 - Relationship between stress and frequency for two nodes

3.3.2 Viewing the results of random vibration analysis

Fields of displacements, accelerations, and stresses at random vibration are presented in Figures 3.41-3.43.

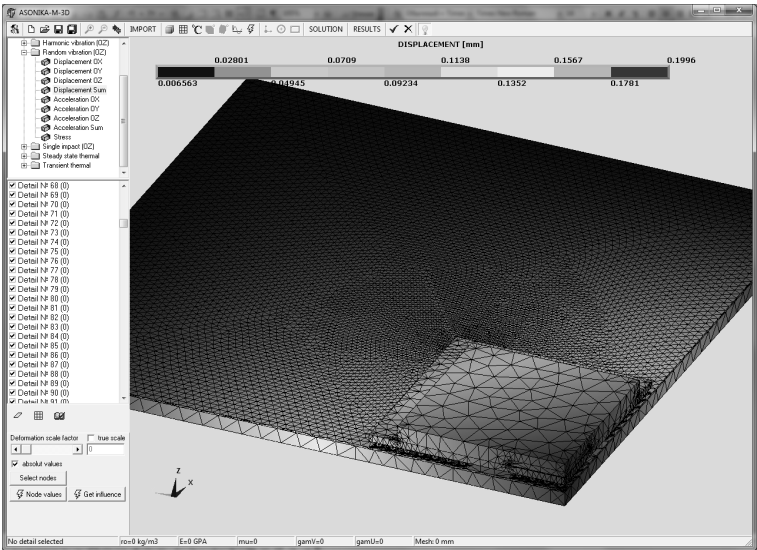


Figure 3.41 - Field of total displacements

The results indicate high values of the stress in the lower sections of the solder balls. For more detailed investigation of the stress distribution along solder balls, we hide some parts of the structure, such as a substrate chip and a compound (Figure 3.44). Now select the nodes in the lower part of a solder balls and we get stresses at these nodes (Figure 3.45).

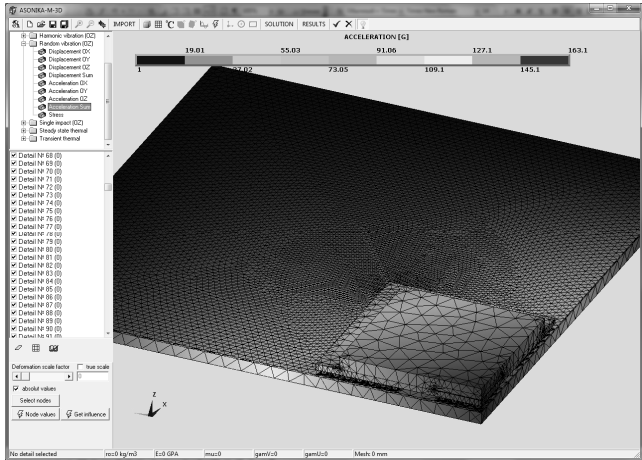


Figure 3.42 - Total acceleration field

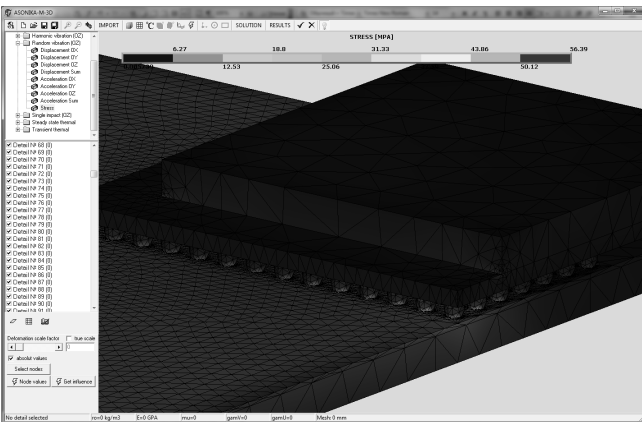


Figure 3.43 - Total stress field

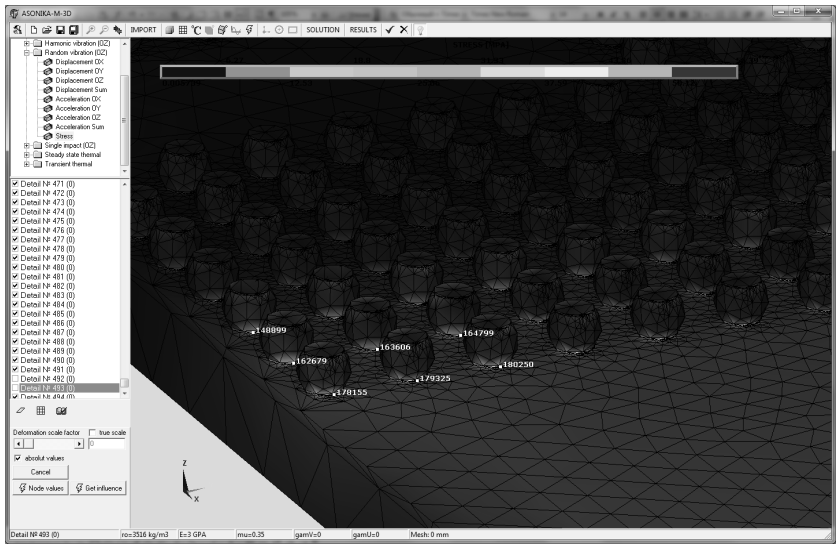


Figure 3.44 - Selection of nodes

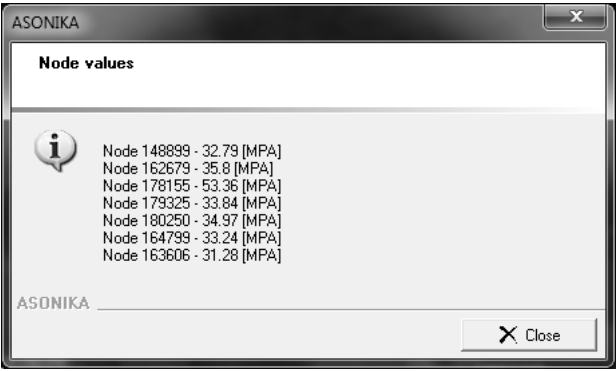


Figure 3.45 - Values of node stresses

3.3.3 Viewing results of the single impact analysis

The impact of a short shock pulse (its duration is equal to 20 ms) causes a sudden power surge. To display the output values of fields, select time corresponding to the start of shock impulse damping (15.9 ms), since maximum values of stresses

appear at this moment. Fields of displacement, acceleration, and stresses are presented in Figures 3.46-3.48.

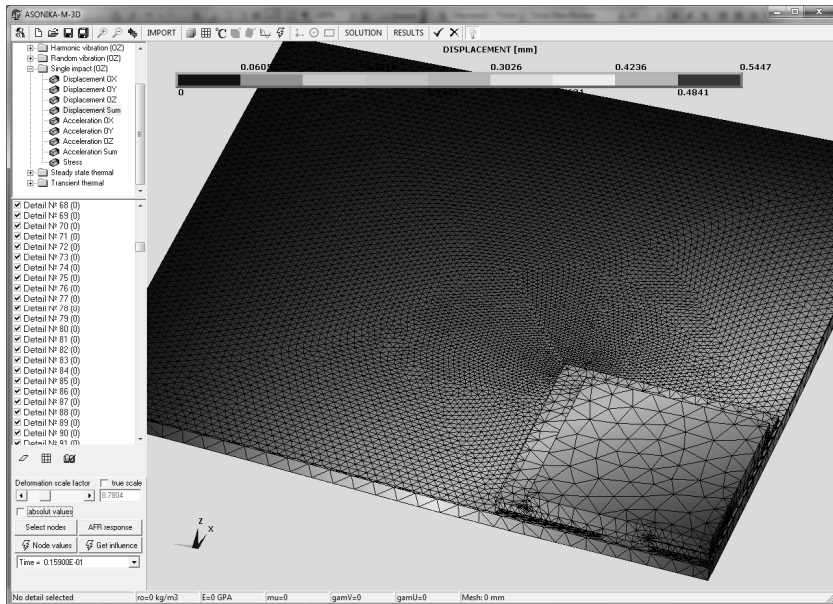


Figure 3.46 - Field of total displacement

Figure 3.47 depicts "relative" acceleration, the maximum of which is about 1.5g. However, one must understand that the shock impulse amplitude at a given moment of time is 200g and the "absolute" speed will be largest at more than 200g.

As an example, we select several nodes (see Figure 3.49) to obtain graphs of stress versus time at these sites (Figure 3.50).

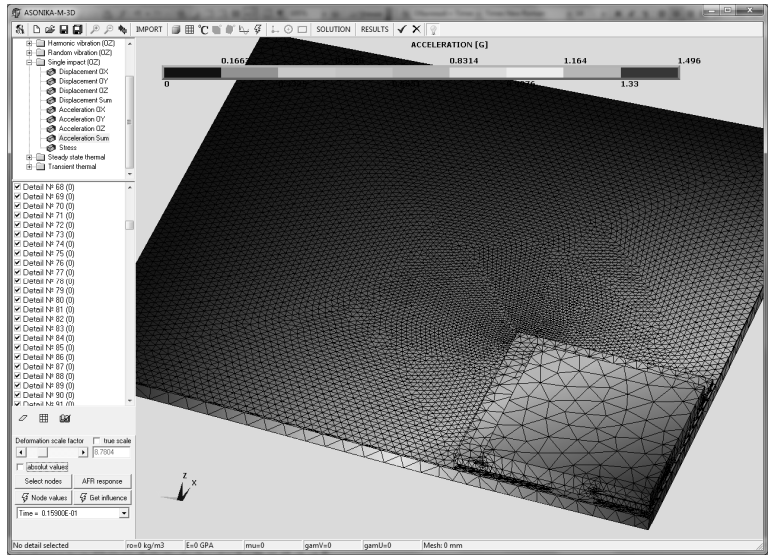


Figure 3.47 - Field of the total (relative) acceleration

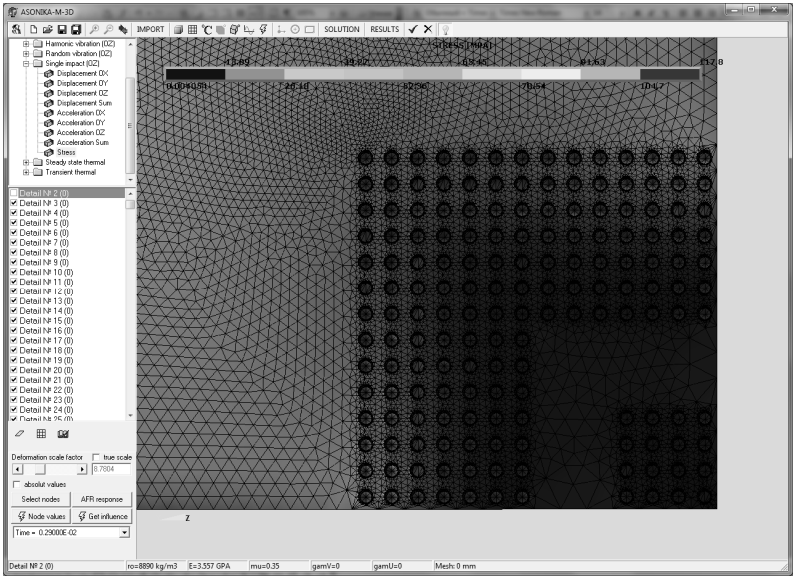


Figure 3.48 - Stress field (some parts of the structure are hidden)

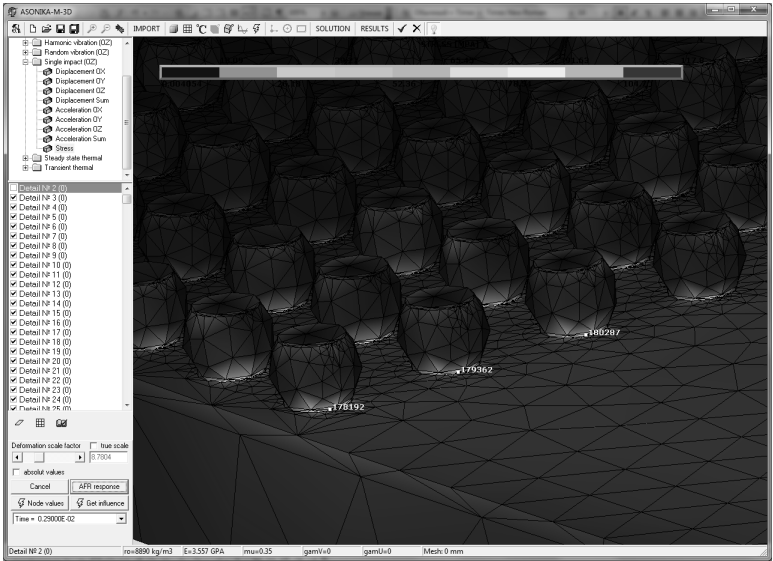


Figure 3.49 - Selection of nodes

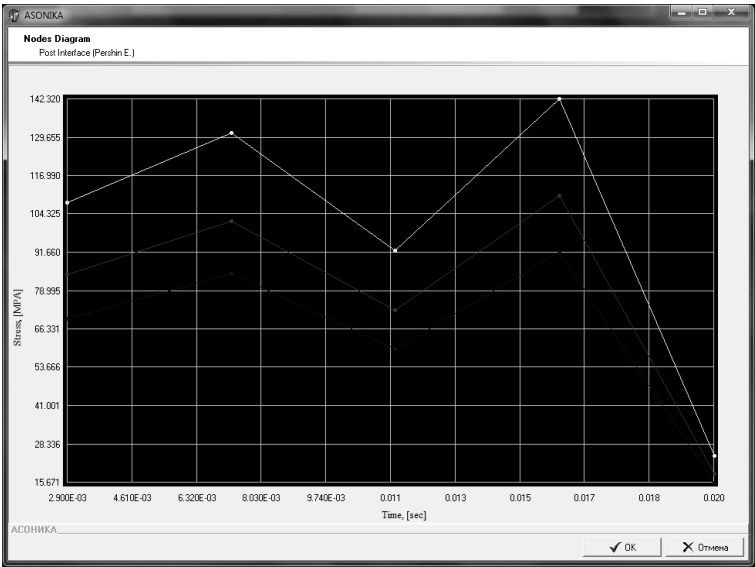


Figure 3.50 - Stress versus time for the three nodes

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.

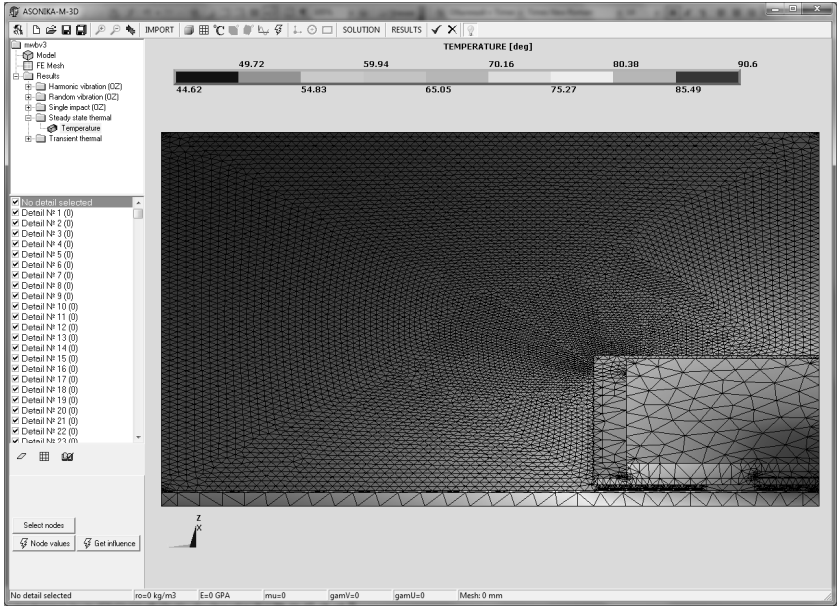


Figure 3.51 - Temperature field

Hide some parts of the structure to obtain the temperature field in the solder balls (Figure 3.52). Separating the components and deriving the temperatures in them (Figure 3.53), we establish that the range of temperature change along the solder balls is approximately 73-90 degrees Celsius.

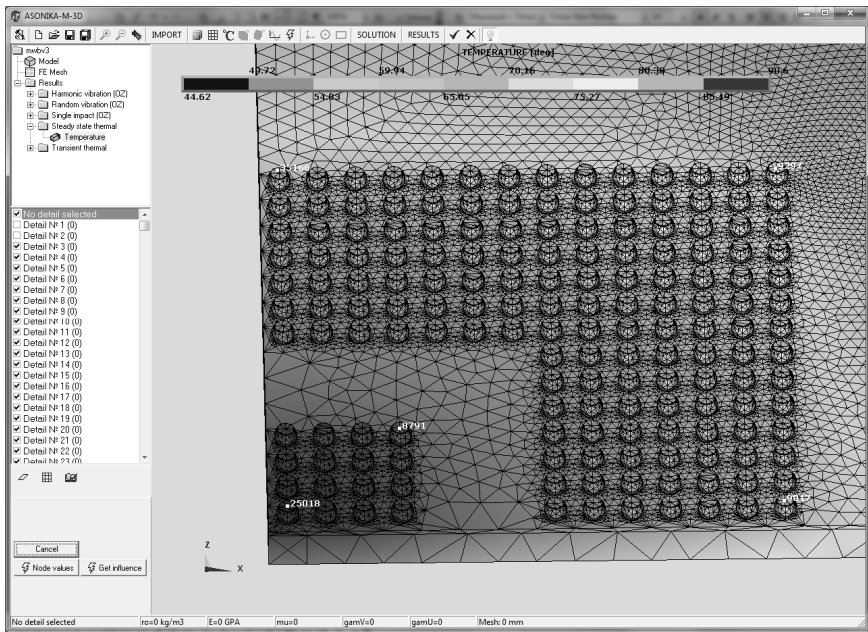


Figure 3.52 - Temperature field in solder balls

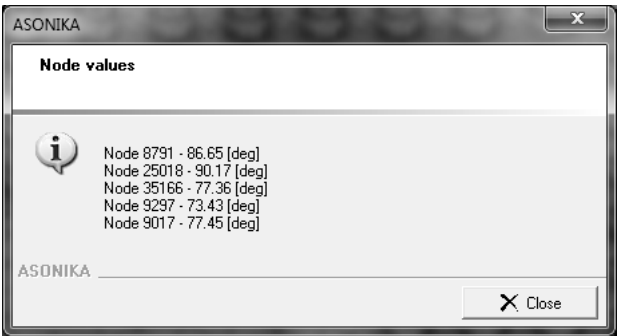


Figure 3.53 - Temperatures in the selected nodes

3.3.5 Viewing the results of transient thermal analysis

Due to the cyclic change of heat capacity and the temperature of the silicon chip, stresses must also change cyclically. The temperature field and thermal stress

for an arbitrary point in time are presented in Figures 3.54-3.55. For others, the amplitude of these values will be different, but the pattern of distribution will remain about the same.

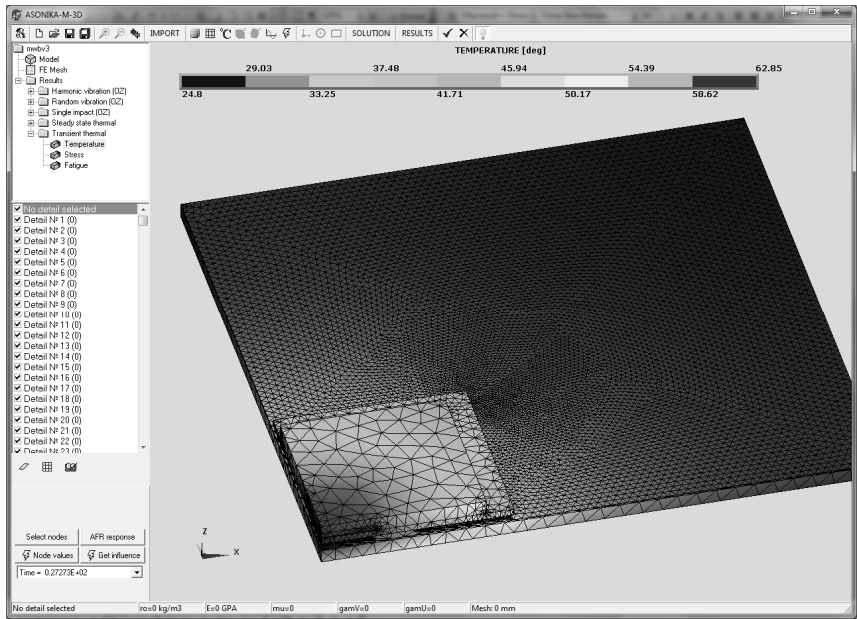


Figure 3.54 - Temperature field

Singling out an arbitrary node within a silicon chip (Figure 3.56), we obtain the graph of temperature versus time at the source of thermal power (Figure 3.57).

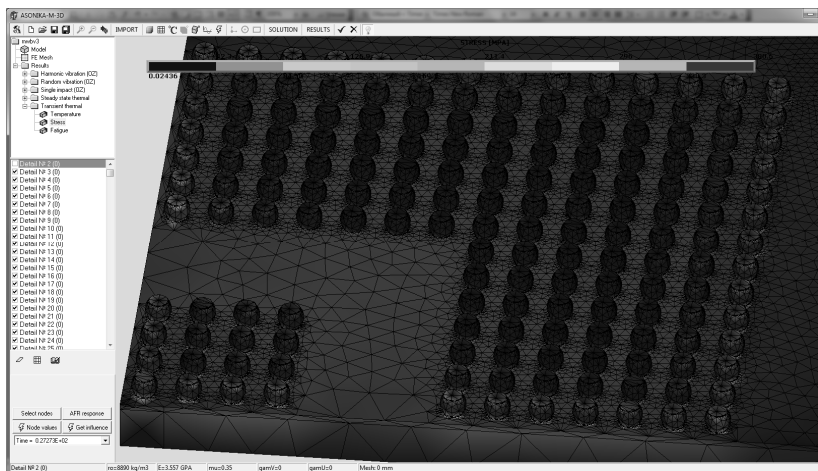


Figure 3.55 - Thermal stress field (some parts of the structure are hidden)

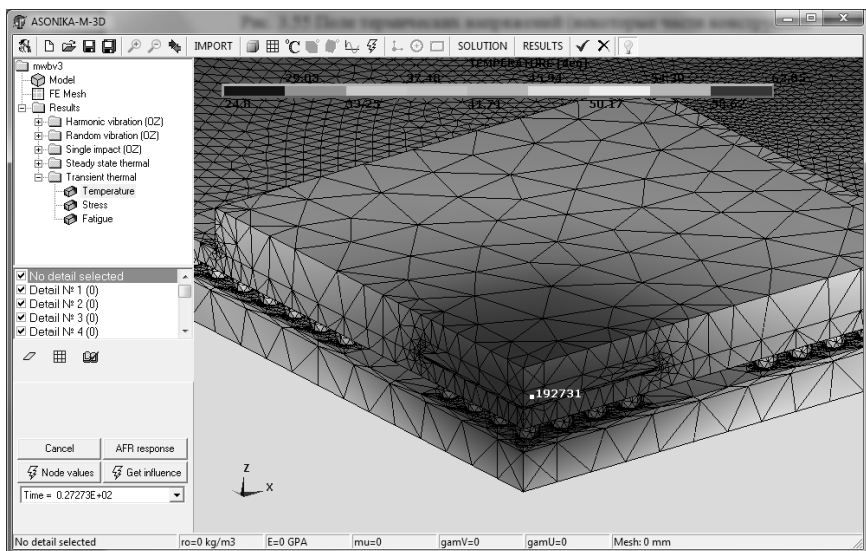


Figure 3.56 - Selected node

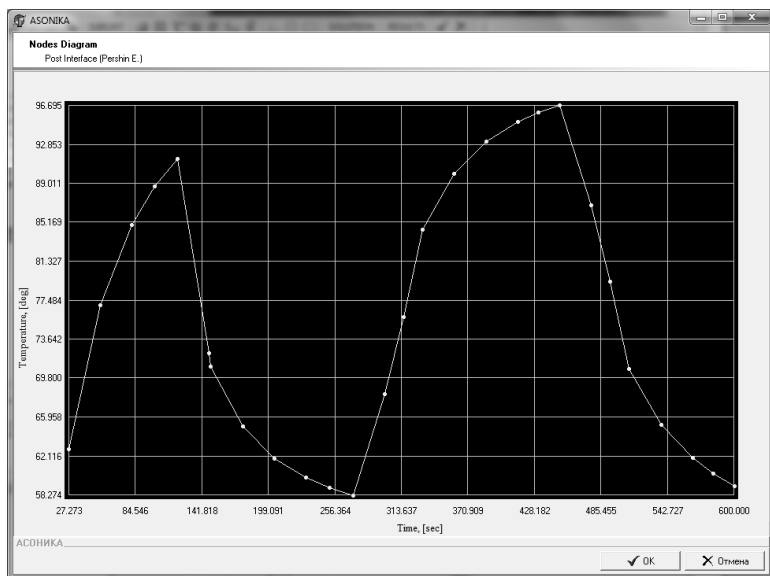


Figure 3.57 - Time dependence of the temperature at the source of power

As the results of the calculation show, thermal stresses reach a maximum in the upper and lower sections of the solder balls (Figure 3.55). To highlight a few examples of "dangerous" sites (Figure 3.58), we obtain the time dependence of stress and temperature in these nodes (Figures 3.59-3.60).

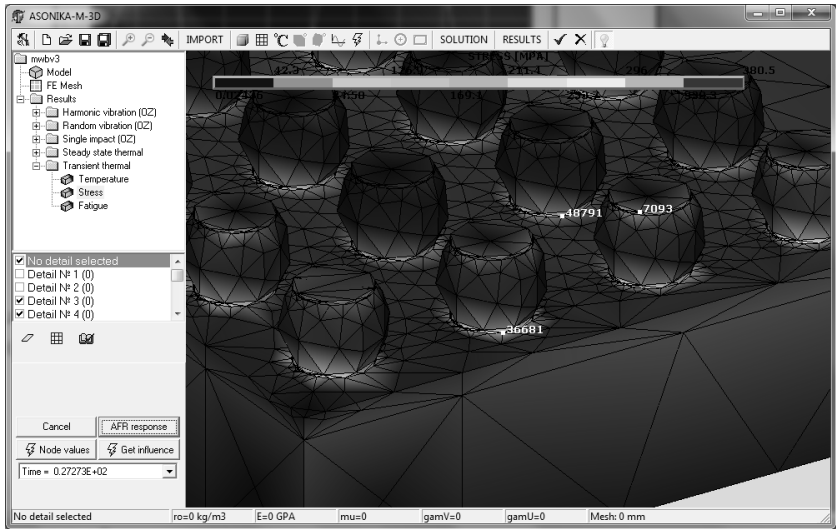


Figure 3.58 - Selection of nodes

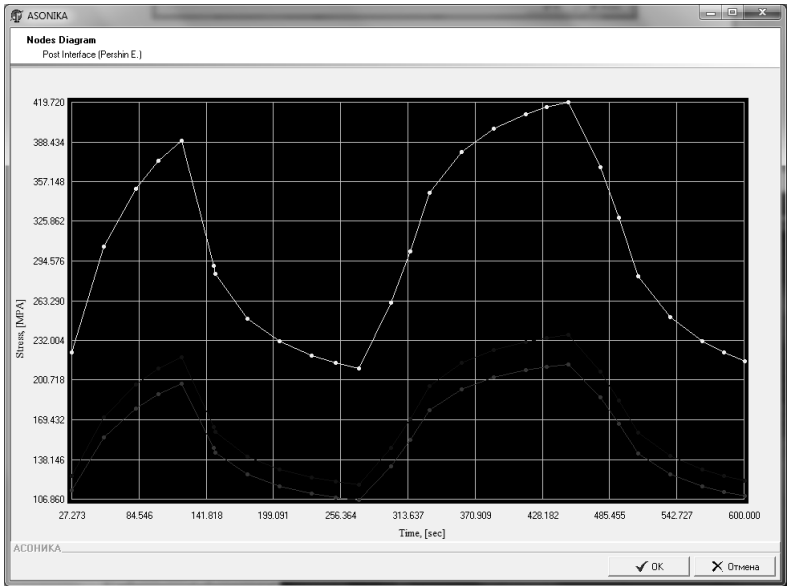


Figure 3.59 - Time-dependent stresses at the nodes

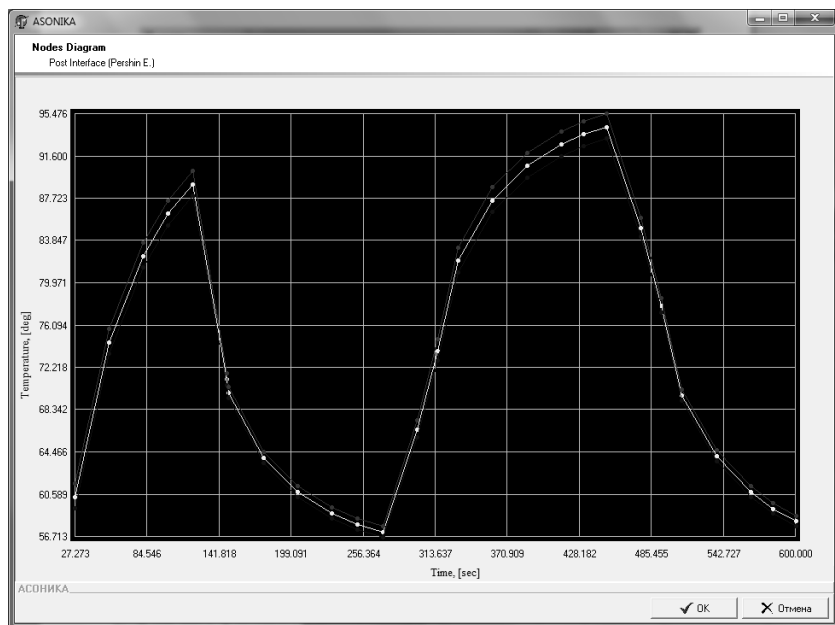


Figure 3.60 - Time dependence of the temperature at the nodes

Finally, we generate the field of fatigue for the structure (Figure 3.61). Time to fatigue failure under transient thermal analysis is measured in thermal cycles. The minima of thermal cycles to failure are observed, as expected, in sections solder balls. The absolute minimum of thermal cycles to failure is 1060 cycles.

We present a separate field for a separate solder ball (Figure 3.62), and also values of fatigue in critical nodes of the output section (Figure 3.63).

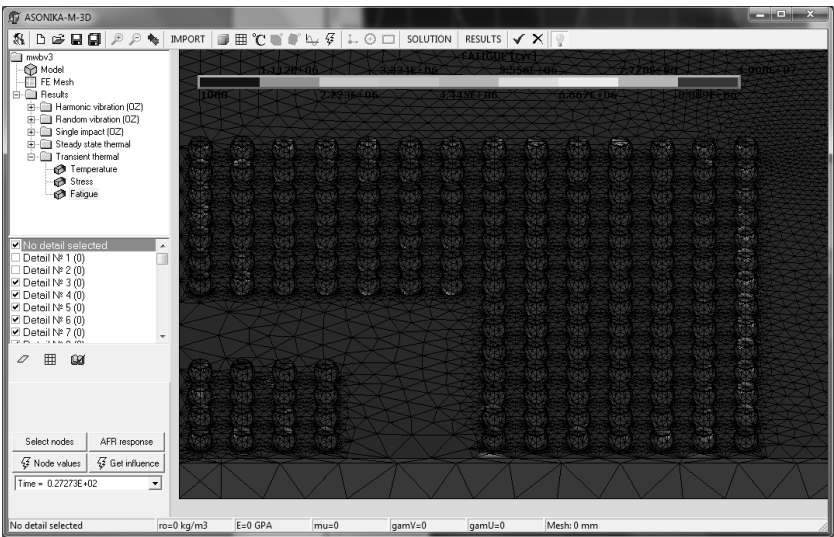


Figure 3.61 - Field of fatigue

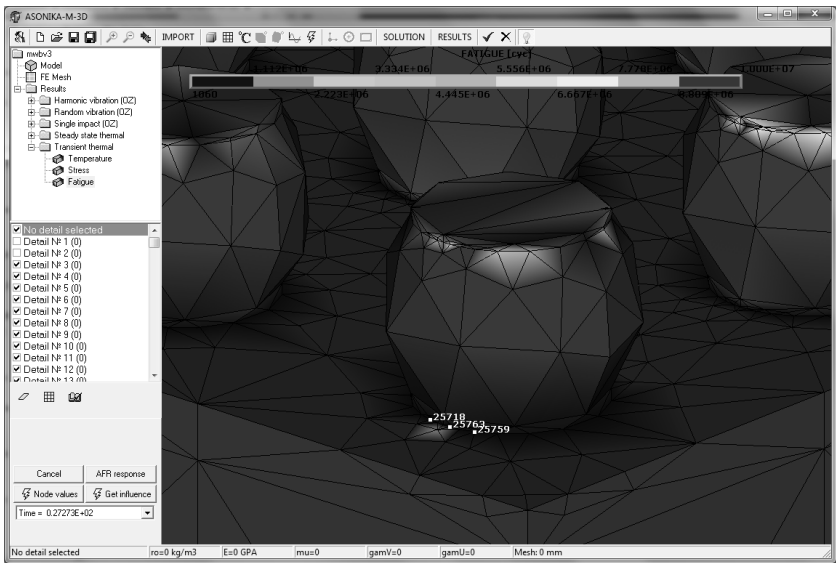


Figure 3.62 - Field of fatigue for a single output

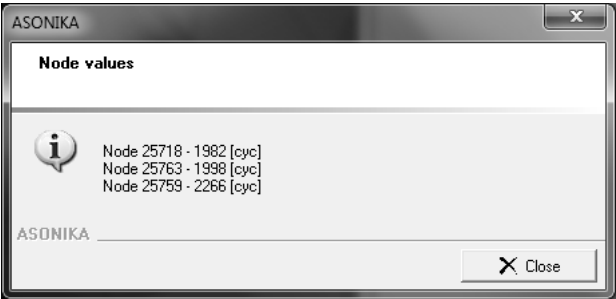


Figure 3.63 - Values of cycles to failure at the nodes

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.

References

1. *Niesłony A., Macha E.*, Spectral method in multiaxial random fatigue, Springer, Berlin, 2007.
2. *Segalman D.J., Reese G.M., Fulcher C.W., Field R.V.* An Efficient Method for Calculating RMS von Mises Stress in a Random Vibration Environment, Proceedings of the 16th International Modal Analysis Conference, Santa Barbara, CA, 1998, pp. 117-123.
3. *Зенкевич О.С.* Метод конечных элементов в технике. М., Мир, 1975.
4. *Benasciutti D., Tovo R.* Rainflow cycle distribution and fatigue damage in Gaussian random loadings. Report n. 129, Department of Engineering, University of Ferrara, Italy, 2004.
5. *Tovo R.* Cycle distribution and fatigue damage under broad-band random loading, International Journal of Fatigue, Vol. 24, 2002, pp. 1137–1147.
6. *Morrow J.D.* The effect of selected subcycle sequences in fatigue loading histories, Random Fatigue Life Prediction, ASME Publications, PVP 72, 1986, pp. 43–60.
7. Release 11.0 Documentation for ANSYS, ANSYS Inc., 2007. 92. nSoft V5.3 Online Documentation, 2003.
8. *Thomson W.T.* Theory of Vibration with Applications, 5th Ed, Prentice Hall, 1998.
9. U.S. Dept. of Defense, “MIL-HDBK-5H: “Metallic materials and Elements for Aerospace Vehicle Structures”, 1998.
10. *Dirlik T.* Application of Computers in Fatigue Analysis, PhD Thesis, UK: University of Warwick, 1985.
11. *Bishop N.W.M., Sherratt F.* A theoretical solution for the estimation of “rainflow” ranges from power spectral density data, Fatigue and Fracture of Engineering Materials and Structures, Vol. 13, No. 4, 1990, pp. 311–326.
12. *Miles J.*: On the structural fatigue under random loading, Journal of Aeronautical Science, Vol. 21, No. 11, 1954, pp. 753–762

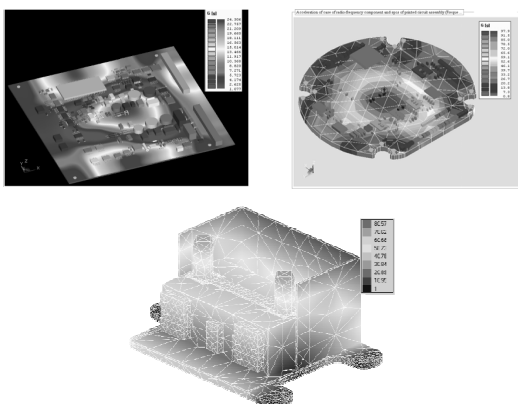
APPENDIX

ASONIKA

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.

