APM Structure3D

User's Guide

APM Structure3D

Parts and structures calculation and design system using finite element method

Version 15

User's Guide

Research and Software Development Center APM Ltd,

Oktyabrsky boulevard 14, office №6, Korolev, Moscow Region, 141070, RUSSIA

Tel/fax: +7 (498) 600-25-10, +7 (495) 514-84-19.

http://www.apmwm.com, e-mail: com@apmwm.com

Printed in Russia.

Contents

Introduction

Preliminaries

APM Structure 3D is a universal system for calculation of various rod, plate, shell, solid, and hybrid constructions.

Using the system one can calculate arbitrary three-dimensional constructions consisting of rods of arbitrary cross-sections, plates, shells and solid elements under arbitrary loading and restraints. The joints between elements may be either rigid, or hinged.

As a result of the calculations, performed by **APM Structure3D** system, the following information is available:

- Loads at the ends of the structure elements
- Stress fields in the structure
- Strain at arbitrary point
- Stress fields in any section of a rod
- Diagrams of bending and twisting moments, transverse and axial forces, etc.
- Structure elements' bearing capacity estimation
- Euler buckling safety factor of the structure
- Stress-strain state of the structure at large displacements (geometrically nonlinear problem)
- Frequencies and modes of natural oscillations of the structure
- Changes in stress-strain states of the structure under dynamic loads

Hardware and software requirements

Minimal:
Operating System: Windows Vista/7/8/10;
dual-core processor, supporting 64-bit addressing.

RAM - 4 GB.

Hard drive: 500 MB free space.
Video display adapter: Radeon or NVidia with OpenGL hardware support.

Recommended:
OC: Windows 7/8/10;
4-core processor, supporting 64-bit addressing. RAM – more than 12 GB.

Hard drive: 1,5 TB free space.

Video display adapter: Radeon or NVidia with OpenGL hardware support.

Brief guidebook

In the **Introduction** (this section) general information about **Structure 3D** is given.

Chapter 1, Construction Editor describes the functions and operation of construction editor **APM Structure 3D**.
Chapter 2, Command Reference gives a complete description of all commands, menu options, and dialog boxes available in the construction editor.
Chapter 3, Cross-section Editor describes the process of creating new cross-sections, and work with cross-section libraries.
Chapter 4, Calculations gives a complete description of all calculations made by **APM Structure3D**.
Chapter 5, Design Elements, Soil Bases and Foundations describes working principles with steel, wood and reinforced concrete elements, cross-sections characteristics and reinforcement design (columns, girders, plates, foundations) according to SNiP.
Chapter 6, Results gives a complete description of all results obtained with **APM Structure3D.**

Chapter 7, Design of Structure Steel Joints describes the creation process and the basic properties of the drawings of steel elements' joints.

Chapter 8, Function Editor gives a complete description of all commands available in the function editor for setting graph of dynamic load.

Chapter 9, Finite Element Analysis gives overview of the finite element method as implemented in the system.

Fonts used in this book

To help you reading this book we used the following set of fonts

`a:\setup`	This font represents the text the way it occurs on the screen, and also text, which user should enter with the keyboard
SETUP.EXE	We use all capital letters for the names of files and keys
Help	**APM Structure3D** command names and buttons of dialog boxes are shown in boldface
Results	Italics is used for the frame names of dialog boxes and controls

How to contact APM

To contact APM you can use one of the following ways:

Send fax. Our Moscow fax number is +7(498) 600-25-10.

Call by phone +7(498) 600-25-10, +7(495) 514-84-19 (Moscow).

Write a letter and send it to

Research and Software Development Center APM Ltd
Oktyabrsky boulevard 14, office №6
Korolev, Moscow Region, 141070, Russia

e-mail: com@apmwm.com
internet: www.apmwm.com

Chapter 1. Structure editor

APM Structure3D editor allows you to enter construction geometry, place supports and hinges, apply static loads, assign cross-sections, thickness, and material parameters to elements.

Views

The work of the editor is based on the operation of projecting onto a plane. Such plane is called viewplane or simply view. When editing a construction, the user works with such viewplanes. The viewplane is characterized by two parameters: rotation and position. Rotation defines the direction of the vector normal to the viewplane and is set by two angles φ and θ like in spherical coordinate system. The other parameter – position in space – is set by a vector.

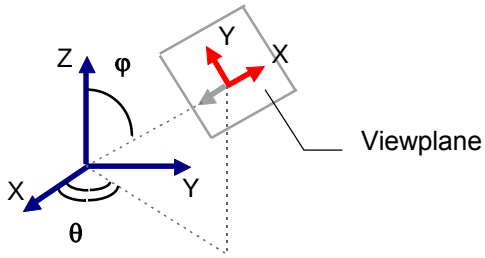


Fig. 1.1 Viewplane position in space

Sometimes, for example, when we move viewplane parallel, it is useful to define viewplane position with *depth*, a scalar value equal to distance between the viewplane and the center of the global coordinate system, similar to ρ in spherical coordinate system.

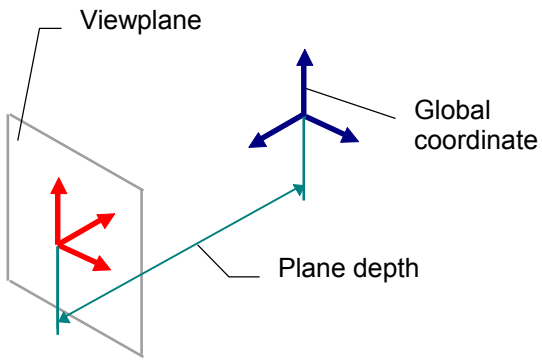


Fig. 1.2 Viewplane depth

Viewplane coordinate system has its Z axis coinciding with the normal vector of the plane, its X axis and Y axis lying in the viewplane. Hereafter viewplane coordinate system will be referred to as *local* while world coordinate system - as *global*. Sometimes it is more convenient to work with local coordinates.

Views which have normal vector coinciding with one of the axes of the global coordinate system are called *main* views. These views are *Top*, *Bottom*, *Right*, *Left*, *Front* and *Rear*. When the direction of the normal vector doesn't coincide with any of the global axes, the view is called *Custom*.

In editor, user is provided with four views, which represent separate windows that can be opened, closed and arranged to the user's convenience. By default, viewplanes are set as *Front*, *Left*, *Top,* and *Custom* or *Isometric* views.

In order to ensure the optimum position of the image on the screen, the user can *scale* and *scroll* it. *Image scale* is the value that shows the relation of the dimensions of the image to the real dimensions of the object. *Scrolling* is the operation of moving or shifting the image with respect to the window.

Also you can use **View Plane** toolbar for convenience.

Fig. 1.3 View plane toolbar

Plane Rotation – command invokes dialog box where you can set view plane with the help of *Phi* and *Theta* angles. Toolbar buttons for standard views: - front; - top; - left; - right; - bottom; - rear; - isometric; - isometric; - isometric; - isometric.

For the optimal view you can *move*, *scale* and *rotate* view with the mouse without breaking a current command.

It is convenient to use the Cube for accelerated dynamic rotation of working field. It appears by holding Ctrl key. Cube sizes increase in working field when mouse cursor is approaching to cube. To activate one of standard views left click on required face, edge or corner of cube; to rotate view use corresponding arrows.

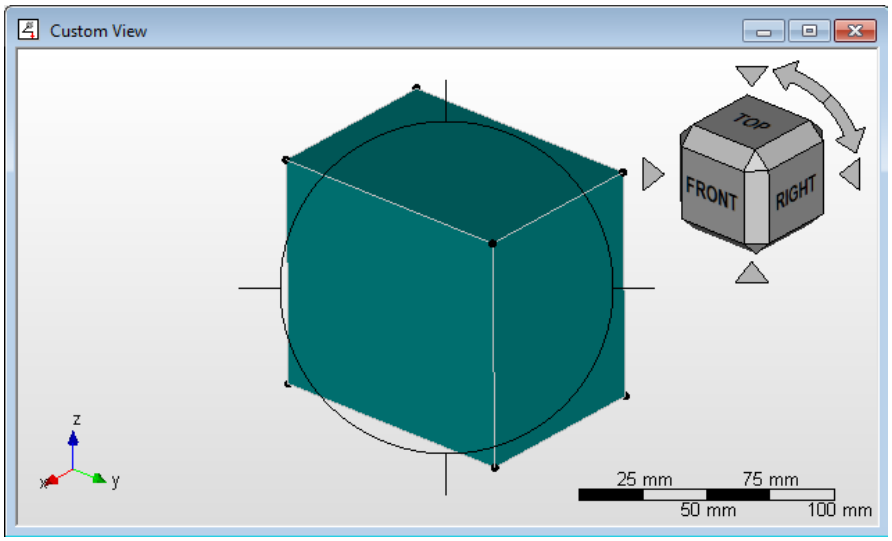


Fig. 1.4 View plane cube

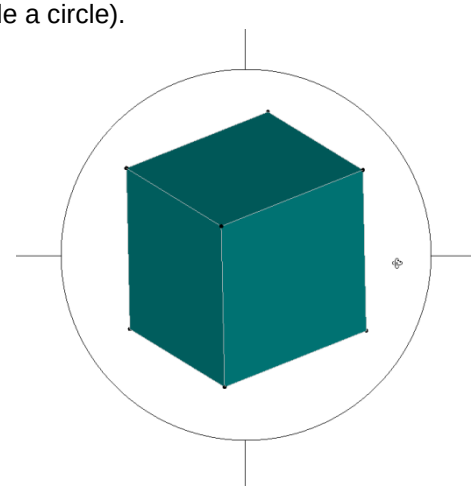
Table 1.1 – View operations with a model

Operation	Mouse
Move view	Holding mouse scroll
Scale view	Rotating mouse scroll
Rotate view	Moving mouse with pressed left button and Ctrl key

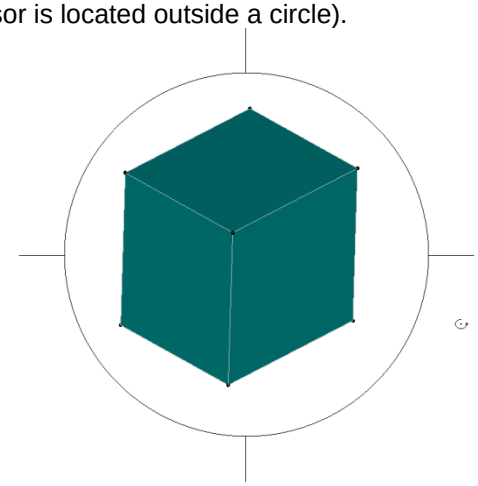
Scale can be activated with the help of toolbar commands: – enlarge, – decrease, – zoom, – fit to view.

View rotation can be realized with holding Ctrl key. Thus there is a circle on the screen with which it is possible to change view in required plane.

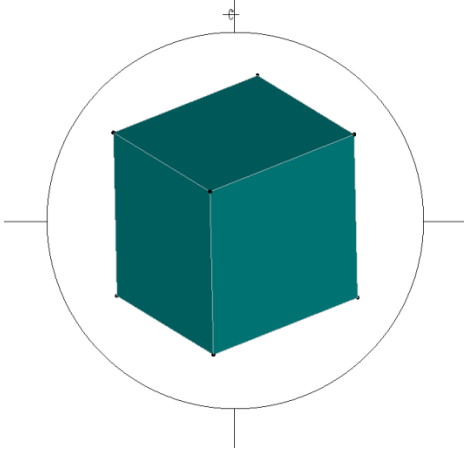
– arbitrary rotation (mouse cursor is located inside a circle).



– rotation in current view plane (mouse cursor is located outside a circle).



– rotation in vertical plane relative to current view plane (mouse cursor is located near the horizontal axis of a circle).



– rotation in horizontal plane relative to current view plane (mouse cursor is located near the vertical axis of a circle).

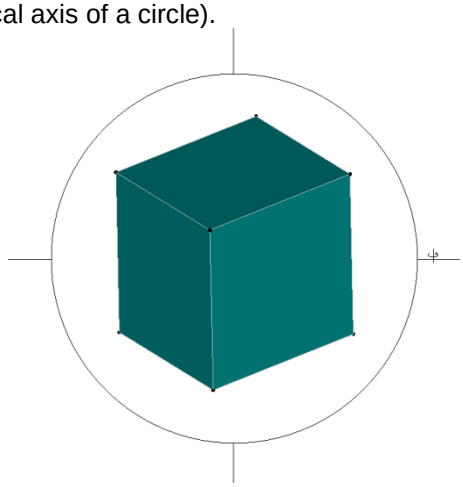


Fig. 1.5 Use of view rotation tool for changing view plane

– Rotate view command which can disable current command.

Editor elements

Construction editor consists of view windows, menus, toolbars and status bar. View elements are nodes, rods, load of various types, auxiliary points such as rotation center and global coordinate system center, etc. All view elements are depicted in a different color. User can change the colors of all view elements and save these color settings in groups called *Palette* (**View / Palette** command). Editor and view elements are shown below.

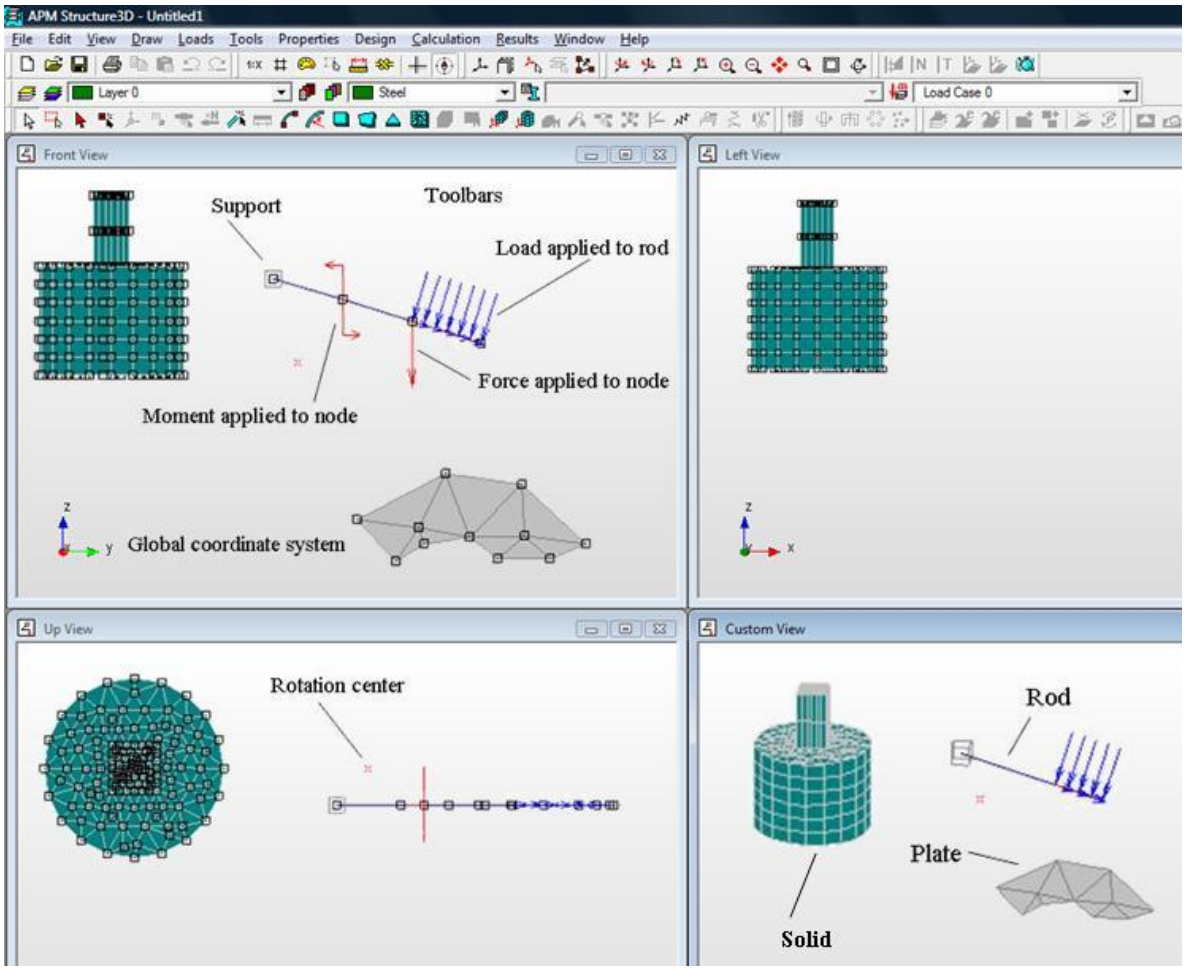


Fig. 1.6 Editor and view elements

Status bar is used for representation of primary information necessary for current work. This information includes: length measurement units, cursor coordinates, current operation parameters (for example radius, when you draw circle) and current operation name. Structure editor status bar is shown below.

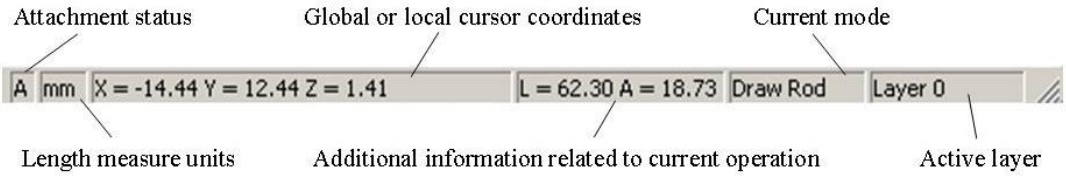


Fig. 1.7 Structure editor status bar

User interface settings

To set toolbars it is necessary to right click on one of toolbars. In displayed context menu make left click to on/off required toolbar. When Settings option is selected the standard dialog box appears on the screen. Let's consider tabs of this dialog in details.

Commands tab contains all commands and its description.

Toolbars tab allows to on/off toolbars.

Keyboard tab allows to create shortcuts to accelerate command activation.

Menu tab allows to set animation and shadows for menu.

Parameters tab allows to set displaying of command prompts and keyboard shortcuts.

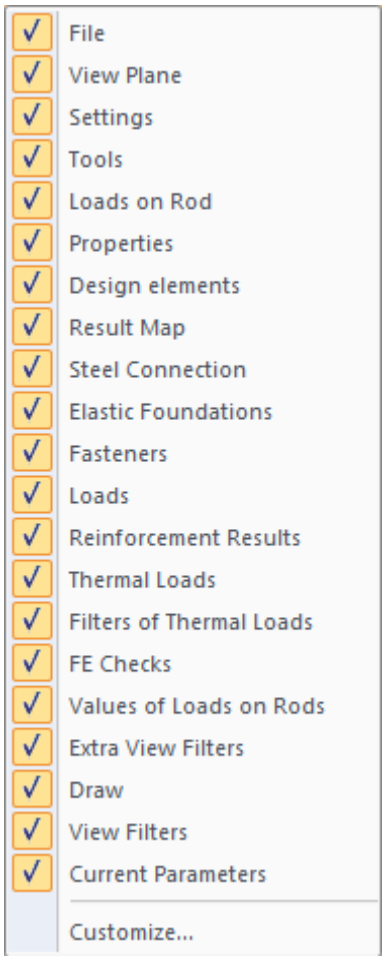


Fig. 1.8 Toolbars and settings context menu

Structure elements

The basic elements of the construction are *nodes, rods*, *plates* and *solid elements*. Nodes are connection points of the elements. Rod is a straight-line beam. Plate is planar 3 or 4-nodal polygon. Solid element is a three-dimensional element in the form of the 8-node hexahedron, 6-node triangular prism or 4-node tetrahedron. Special elements are rigid and elastic links, particular rod finite elements, nodes masses, etc.

The nodes, rods, plates and solid elements can be entered in arbitrary sequence. Setting nodes is a rather simple operation. At first, it is necessary to choose a viewplane that contains the point in which you want to place the node, and then, referring to the coordinates in the toolbar, specify the point with the desired coordinates by a mouse-click.

Rods

Rods can be created in two ways. The first way (**Draw / Rod / By coordinates** command) allows you to draw a rod using the coordinates of its nodes. You can either select existing nodes as starting points or create new ones. The second method allows drawing a rod based on the starting node, direction and length. To use this method, select **Draw / Rod / By Length and Angle** command. Besides, you can insert an additional node in rod or set it on the rod extension by **Draw / Node / On Rod** command. **Draw / Rod / Divide Rod into N Rods** command allows dividing a rod into arbitrary number of equal parts. For more detailed information see *Chapter 2 Command Reference*.

Rod cross-section

A cross-section must be assigned to each rod. It can be of arbitrary shape. Cross-sections are stored in cross-section libraries. *APM Structure3D* includes some cross-section libraries with standard profiles. These libraries files have *.*slb* extension.

Use **Properties / Cross-section to Selected Rods** or **Properties / Cross-section to All Rods** commands to assign cross-section to a rod or a group of rods, respectively.

Cross-section editor is used to make an arbitrary cross-section and it is launched by **File / New / Cross-section**. For more details see *Chapter 3 Cross-section Editor*.

Cross-section alignment point

Rod axis can pass through any point in a cross-section. Cross-section alignment point (or its offset) is set by **Properties / Cross-section Alignment Point** command. By default rod axis passes through center of mass of the cross-section, which corresponds to zero offset (Fig. 1.9 a). Fig. 1.9 b shows an example of rod with center-bottom cross-section alignment. See also *Chapter 2 Command Reference*.

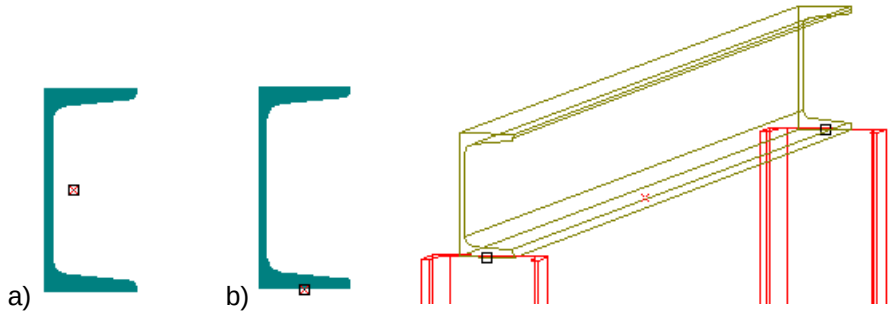


Fig. 1.9 Cross-section alignment point

Plates

A plate can be set as either a parallelogram, an arbitrary quadrangle, or as an arbitrary triangle. For these cases **Draw / Plate / Rectangular 4-noded**, **Draw / Plate / Arbitrary 4-noded** and **Draw / Plate / 3-noded** commands are used correspondingly. Besides, any rectangular plate can be divided into equal parts by **Draw / Plate / Divide Plate** command. Any area including multi-connected areas can be meshed with plates. This is achieved by using **Draw / Plate / Arbitrary with Mesh** command. For more detailed information see *Chapter 2 Command Reference*.

Solid elements

Solid elements can be created as either an arbitrary hexahedron, a triangular prism or a quadrangle. For that **Draw / Solid / 8-noded solid**, **Draw / Solid / 6-noded solid** and **Draw / Solid / 4-noded solid** commands are used correspondingly. Besides, 8-node element can be divided into smaller elements by **Draw / Solid / Divide 8-noded solid** command. For creating pipes and parallelograms out of solid elements, editor has **Draw / Solid / Solid Pipe** and **Draw / Solid / Rectangular Parallelepiped** commands. For more detailed information see *Chapter 2 Command Reference*.

Cursor coordinates in the editor change according to *cursor step*. Cursor step determines the precision degree to which the coordinates are approximated when the cursor is moved in the view. Two cursor steps are used in the editor: *linear* and *angular*. Linear step determines the approximation precision of coordinates and lengths, while angular step determines approximation precision of angles in such operations as drawing an arc, or a rod using length-and-angle method. To change cursor step

values, use the **View /** **Cursor Step** command. Default linear cursor step is equal to 1 measurement unit and angular is equal to 1 degree.

To simplify the process of editing, the *APM Structure3D* editor operates in nodes *snapping* mode. Each node in the viewplane has its own *sensitivity zone*. When cursor is placed in the sensitivity zone of the node, the cursor coordinates are automatically assigned node coordinates. For example, while drawing circle you specify the center so that the cursor hits the sensitivity zone of some node, the center of the circle will be placed exactly in the projection of this node onto the viewplane. A node is highlighted in a different color when cursor is placed in its sensitivity zone. User can set the radius of the sensitive zone by **View /** **Cursor Step** command, while it is 10 pixels by default. Snapping is switched off when SHIFT key is pressed.

Before starting calculation, user should set the section and material parameters for every rod, thickness and material for every plate, and material for every solid element. A particular section can be set either for all rods or for selected ones only. Material parameters are set by default, but can be changed (just as the section) either for all, or for selected rods, plates and solid elements.

Material

Use **Properties /** **Materials** command to set material to structure elements. To assign a material to a group of elements these should selected beforehand, using **Edit /** **Select Object, Edit /** **Select Group** or **Edit /** **Complex Selection by Box** and **Edit /** **Complex Selection by Circle** commands. For more information see *Chapter 2 Command Reference*.

Plate object

Plate object serves to accelerate modeling and to set parameters of plate finite elements. Plate object is characterized by the same properties as a plate (thickness, material etc.). Plate object is meshed on finite elements according to the set parameters at calculation. One plate object can correspond to one design element that does creation of plate design elements (reinforced concrete and reinforced masonry) more simple and convenient.

To create plate object use the **Draw / Plate /** **Arbitrary with Mesh** command. After specifying of arbitrary contour it is necessary to check *Create plate* option in the appeared dialog box.

Using this command plate object creation has a number of features, for example, automatic search of intermediate nodes. It is not necessarily to specify al the nodes of closed contour (fig. 1.10), specifying only angular nodes enough: 1-2-3-4-5-1. Thus all internal nodes on perimeter of a contour and internal plate nodes will be considered automatically at FEM meshing.

Draw / Plate Object/ Create Plate command invokes *APM Graph* editor for creation of plate contour. Plate contour can be created by the following ways:

- Import from *APM Graph* file (*.agp) using **File /** **Open** command.
- Import from foreign graph editor through DXF file using **File / Import** command.
- Create by means of *APM Graph*.
- Create in *APM Graph* from block of current drawing or block library using **Draw /** **Insert Block** command.
- Create in *APM Graph* from parametric model using **Draw /** **Insert Block** command.

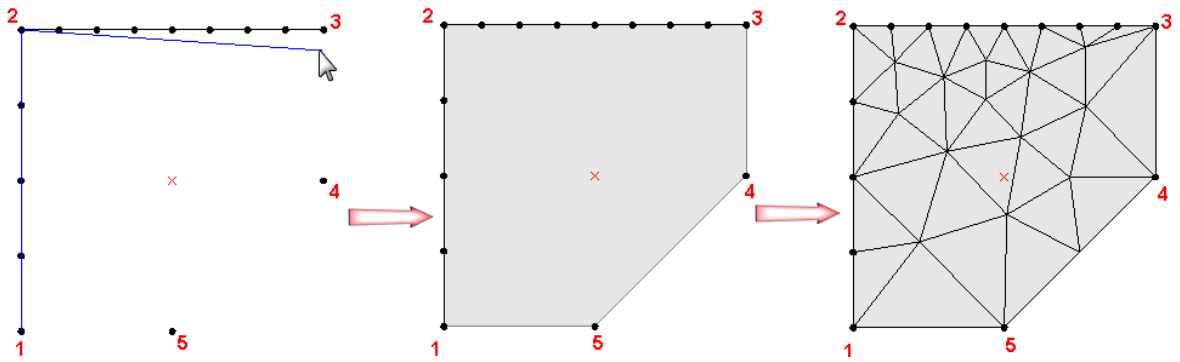


Fig. 1.10 Example of plate object creation with searching of intermediate nodes

To create additional nodes inside or on a contour, for example for further set of the concentrated loads, use one of the following *APM Graph* commands:

- **Draw / Point / Free Point**;
- **Draw / Point / Point on Object**;
- **Draw / Point / Point on Intersection**.

Use of *APM Graph* as the editor of plate objects has some features in comparison with its functioning as the independent 2D editor. In the prepared drawing it is necessary to specify contours using **Contour / Simple contour** and **Contour / User defined contour** commands.

After pressing **Simple contour** it is necessary first to specify elements of external contour, and then elements of internal contours (if they are). After that closed contours should be painted in dark blue color. Corresponding contours can be selected only in the event that they are closed. Simultaneously with pressing one of these buttons the *Contour selection* dialog box invokes, in which after specifying all contours it is necessary to press either **OK** button, or mouse right button, or SPACE key. The area between the selected contours will be painted in grey color. It means, that the program has adequately understood, what area will be a contour.

✍ *Note. Contour is not defined if it is created using lines connected not by means of control points, and, for example, by "Normal" object snap. To define such contour it is necessary to break a line on which the perpendicular has been dropped using **Modify / Break at Point** command.*

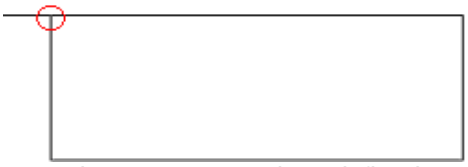


Fig. 1.11 Contour is undefined

User defined contour command is used when ambiguity of contour definition takes place. In this case it is necessary to select required contour elements clicking serially the mouse left button. If the previous element was highlighted, and the subsequent is not, there is no connection between these elements i.e. the contour is non-closed.

It is possible to set contour using hatch: **Draw / Hatch / Simple Hatch** or **Draw / Hatch / User Defined** commands. It is possible to save plate object in *APM Graph* file ***.agp** format.

After the contour will be painted in grey color select **File / Ready** command to transfer plate in *APM Structure3D*. In the appeared dialog box it is necessary to set height (z coordinate) where the plate will be created.

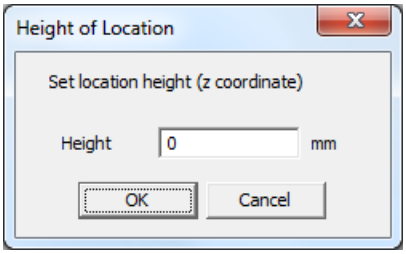


Fig. 1.12 Height of plate location dialog box

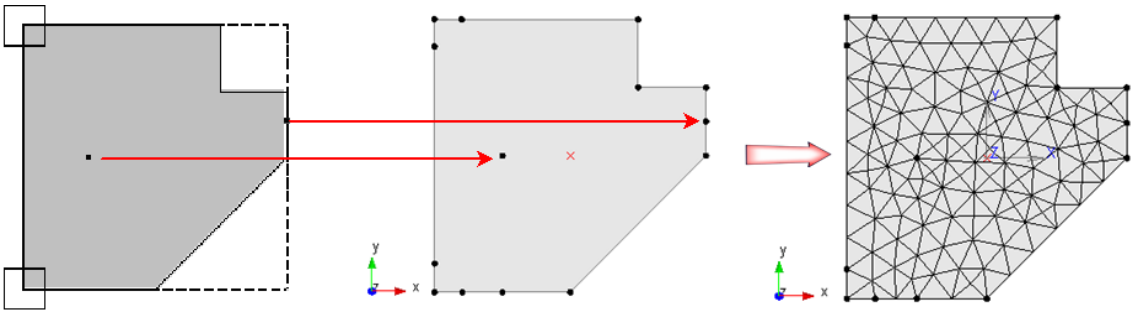


Fig. 1.13 Plate object with internal nodes

Contour creation ways

Import from ***.dxf, *.agr**	Drawing tools *APM Graph*	Block *APM Graph*: - current drawing block; - block library.	Parametric model *APM Graph:* - from file; - from library.

Contour selection ways (commands)

Contour / Simple contour **Contour / User defined contour**	**Draw / Hatch / Simple Hatch** **Draw / Hatch / User Defined**

Plate object editing

APM Structure3D	*APM Graph*
All commands which are used for editing of plates are accessible: *Editing of existing plates:* **Edit / Edit Object**, **Draw / Node / By Coordinates**, **Tools / Node Alignment**, **Tools / Rotate**, etc. *Creation/Addition of new plates:* **Tools / Copy** + **Tools / Paste**, **Tools / Loft**, **Tools / Mirror**, **Tools / Polar Array**, etc.	It is possible to edit only one or several plates, laying in one plane. Use **Draw / Plate Object / Edit Plate** command selecting one or several plates previously. *Editing of existing plates:* **Modify / Edit**, **Modify / Modify**, **Modify / Move**, **Modify / Rotate**, **Modify / Scale**, etc. *Creation/Addition of new plates:* **Modify / Copy**, **Modify / Mirror**, **Modify / Rectangular Array**, **Modify / Polar Array**, **Draw / Block / Insert Block**, **Draw / Block / Insert Object from DB**, etc.
Restrictions at creation and editing of plates	
– Editing is impossible when plate nodes displace outside plane (in this case plates will not change and new nodes will be created. – When editing group of plates created on the basis of the block or parametrical model the system gives warning.	– It is possible to create several plates by one operation. – If contour breaks or hatch deletes at plate editing it is necessary to set it anew. – When modification of plate group created on the basis of parametrical models with identical variables it is necessary to check *Recalc. all variables* option. – Editing of plate group created on the basis of the block or parametrical model is possible both in group and alone. – To finish editing activate **File / Ready** command.

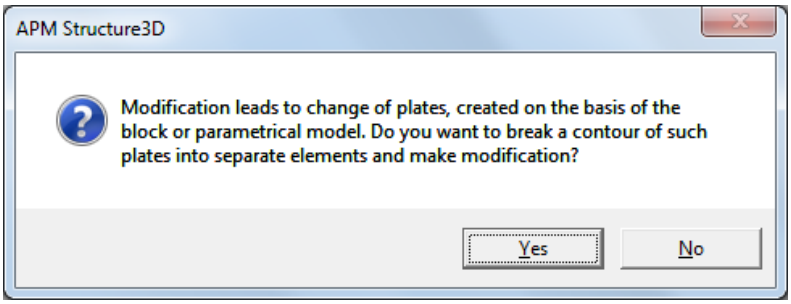


Fig. 1.14 Warning about plate's modification

To generate plate object mesh it is necessary to select it, set maximum side length using **Draw / Plate Object / Mesh Options** command and then press **Create Mesh** button. Thus in all nodes located inside plate contour internal plate nodes will be created. To display plate FE mesh press **Plate Object Mesh** button on *Filters* toolbar.

It is possible to set various meshing parameters of different plates. In case of poor meshing quality (very large or, contrary, very small mesh) it is necessary to set for the selected plates other meshing parameters and generate FE mesh anew.

If meshing parameters are not set for all or some plates FE meshing will be executed automatically according to default parameters before calculation.

To delete previously generated mesh select plate and press **Delete Mesh** command in the *Mesh Parameters* dialog box invoked by **Draw / Plate Object / Mesh Options** command.

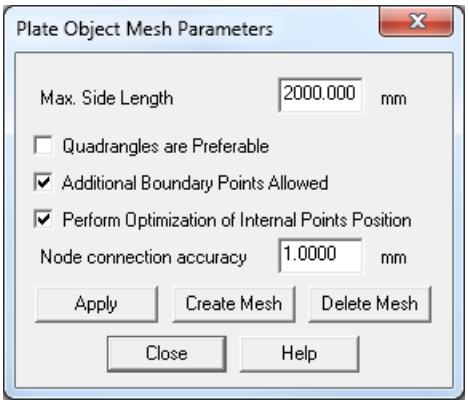


Fig. 1.15 Plate Object Mesh Parameters dialog box

To edit plate object it is necessary to select plate and to activate **Draw / Plate Object / Edit Plate** command. Thus selected plate will be opened in *APM Graph* editor.

Previously created mesh on plate object can be divided into plate finite elements with the help of **Draw / Plate Object / Explode Plate** command.

If during modeling meshing of rods intersected with plate was carried out new FE mesh can be generated in associative mode.

To do this select a plate and press **Draw / Plate Object / Edit Plate** command. Editing in *APM Graph* allows to take into account all nodes inside plate contour providing associative connection between finite elements and plate objects. Select **File / Ready** command to switch *APM Structure3D.*

Now at calculation the mesh will be generated with new nodes. Thus, at FE meshing for the plate two conditions are considered simultaneously:

- Internal nodes and contour nodes.
- Plate meshing parameters.

During modeling deletion of intermediate nodes is possible also. In this case associative meshing will be carried out accounting existing nodes.

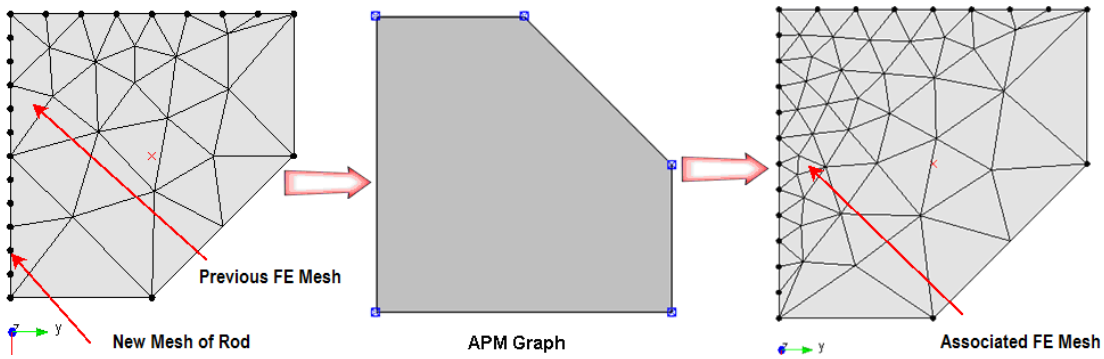
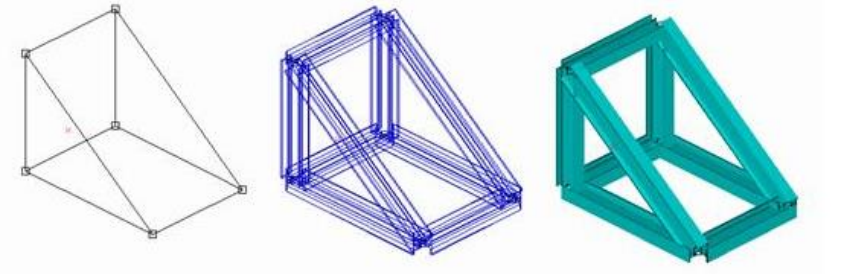


Fig. 1.16 Plate editing with associative meshing

View Filters

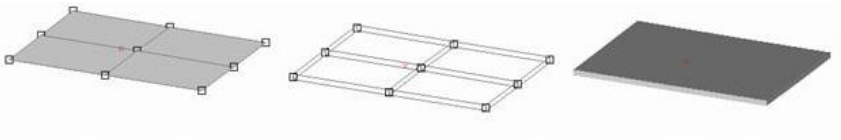
In editor, the construction can be represented in many ways. While working with editor, one is often faced with the necessity to represent only elements of a certain class, for example, only nodes and loads applied to them, and to hide other elements. Besides, some elements can be represented in many ways. To control the visualization level special tools, named *Filters*, are used in editor. Filters define whether a certain element is shown or not, and on what level. Rods can be shown on three visualization levels:



Wire-frame model | Wire-frame cross-section model | Solid cross-section model

Fig. 1.17 Rods show

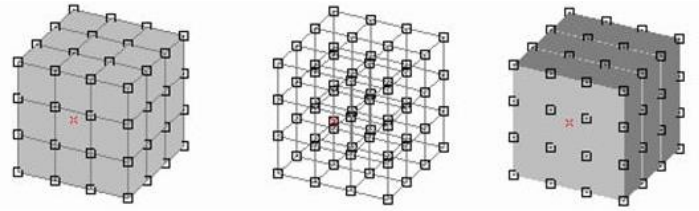
Plates are also represented with the help of three visualization levels:



Flat model | Wire-frame model | Solid model

Fig. 1.18 Plates show

Solid elements:



Edge model | Wire-frame model | Solid model with lighting

Fig. 1.19 Solids show

All view filters are available in the *View filters* (fig. 1.20) and *Extra view filters* (fig. 1.21) toolbars.

Fig. 1.20 View filters toolbar

View Filters / Nodes

Command: show/hide nodes
Shortcut:

View Filters / Node Numbers

Command: show/hide nodes numbers
Shortcut:

View Filters / Node Local CS

Command: show/hide nodes local coordinate systems

Shortcut:

View Filters / Wireframe Rods

Command: show/hide rods as wireframe models
Shortcut:

View Filters / Wireframe Cross-sections

Command: show/hide rods cross-sections as wireframe models with sections
Shortcut:

View Filters / Solid Cross-sections

Command: show/hide rods cross-sections as solid models with sections
Shortcut:

View Filters / Only Current Cross-section

Command: show/hide only current section (which is active on *Current parameters* toolbar)
Shortcut:

View Filters / Rod (Beam/Truss/Cable) Local CS

Command: show/hide local coordinate system of rod elements
Shortcut:

View Filters / Vertical Rods

Command: show/hide vertical rods (slope angle to Z axis is no more 15°)
Shortcut:

View Filters / Inclined Rods

Command: show/hide inclined rods (slope angle is in range 15°–75° to Z axis)
Shortcut:

View Filters / Horizontal Rods

Command: show/hide horizontal rods (slope angle to Z axis is no less 75°)
Shortcut:

View Filters / Flat Plates

Command: show/hide flat plates as flat polygons
Shortcut:

View Filters / Wireframe Plates

Command: show/hide wireframe plates in thickness view
Shortcut:

View Filters / Solid Plates

Command: show/hide solid plates in thickness view
Shortcut:

View Filters / Plate Local CS

Command: show/hide plates coordinate systems
Shortcut: 

View Filters / Plate Normals

Command: show/hide plate normals
Shortcut: 

View Filters / Plate Object Mesh

Command: show/hide plate object mesh
Shortcut: 

View Filters / Solid Elements

Command: show/hide solid elements sides as flat polygons
Shortcut: 

View Filters / Wireframe Solid Elements

Command: show/hide solid elements as wireframe models
Shortcut: 

View Filters / Solid Elements with Lighting

Command: show/hide solid elements with lighting
Shortcut: 

View Filters / Solid Local CS

Command: show/hide solid local coordinate system
Shortcut: 

View Filters / Elastic Links

Command: show/hide elastic links
Shortcut: 

View Filters / Couplings

Command: show/hide couplings
Shortcut: 

View Filters / Rigid/elastic Supports

Command: show/hide supports
Shortcut: 

View Filters / One-Dir Rigid Supports

Command: show/hide one-direction rigid supports
Shortcut: 

View Filters / Node Loads

Command: show/hide nodal loads
Shortcut: 

View Filters / Rod Loads

Command: show/hide rod loads
Shortcut: 

View Filters / Plate Loads

Command: show/hide plate loads
Shortcut: 

View Filters / Solid Loads

Command: show/hide solid loads
Shortcut: 

View Filters / Snow Loads

Command: show/hide snow load
Shortcut: 

View Filters / Wind Loads

Command: show/hide wind load
Shortcut: 

View Filters / Node Masses

Command: show/hide node masses as concentrated masses
Shortcut: 

View Filters / Contact Elements

Command: show/hide contact elements
Shortcut: 

View Filters / Contact Elements Local CS

Command: show/hide local coordinate system of contact elements
Shortcut: 

View Filters / Target Elements

Command: show/hide target elements
Shortcut: 

View Filters / Target Elements Local CS

Command: show/hide local coordinate system of target elements
Shortcut: 

View Filters / Center of Mass

Command: show/hide center of mass
Shortcut:

View Filters / Only Selected Elements

Command: show/hide only selected elements
Shortcut:

View Filters / Zero Level

Command: show/hide Z zero level (for wind load setting)
Shortcut:

View Filters / Dimension Scale

Command: show/hide dimension scale
Shortcut:

Fig. 1.21 Extra view filters toolbar

Extra View Filters / Node Load Values

Command: show/hide values of loads on nodes
Shortcut:

Extra View Filters / Plate Thickness Map

Command: show/hide plate thickness in view of thickness map
Shortcut:

Extra View Filters / Plate Pressure Map

Command: show/hide plate pressure in view of pressure map
Shortcut:

Extra View Filters / Plate Pressure Values

Command: show/hide values of pressure on plates
Shortcut:

Extra View Filters / Plate Snow Pressure Map

Command: show/hide plate snow pressure in view of pressure map
Shortcut:

Extra View Filters / Plate Wind Pressure Map

Command: show/hide plate wind pressure in view of pressure map
Shortcut:

Extra View Filters / Contact Elements Normal Stiffness Map

Command: show/hide normal stiffness of contact elements in view of stiffness map. It is necessary to set result range previously in dialog box (min and max values)

Shortcut: N

Extra View Filters / Contact Elements Tangent Stiffness Map

Command: show/hide tangent stiffness of contact elements in view of stiffness map. It is necessary to set result range previously in dialog box (min and max values)

Shortcut: T

Extra View Filters / Fictive Contact Elements and Normal Stiffness Map

Command: show/hide fictive contact elements and normal stiffness in view of stiffness map. It is necessary to set result range previously in dialog box (min and max values)

Shortcut:

Extra View Filters / Fictive Contact Elements and Tangent Stiffness Map

Command: show/hide fictive contact elements and tangent stiffness in view of stiffness map. It is necessary to set result range previously in dialog box (min and max values)

Shortcut:

Extra View Filters / Value Range

Command allows to set value range for extra view filters maps.

Shortcut:

Extra View Filters / Capture Image

Command saves current model view into image *.bmp or *.jpg. After command activation type in image file name

Shortcut:

Extra View Filters / Design Element Names

Command: show/hide names of design elements

Shortcut:

Layers

When designing a construction of high complexity, user has to operate with a large number of elements. To make it more efficient, elements are stored in groups called layers. By default, all elements are stored in one layer. User can create new layers, assign names to them, select an active layer, switch them on/off, and remove them. Editor always has one active layer, in which currently created elements are placed. Elements that belong to a layer that is switched off are not shown on the screen and are not accessible for selection and modification. Selected elements can be moved from the layers they belong to, to the active layer. When the layer that contains some elements is deleted the latter are moved to the active layer. Operating with layers manager is initiated via **Tools / Layers** command. To place selected elements into the active layer **Tools / Add to Current Layer** command is used. Option "on/off" layer and deletion of empty layers are provided.

Operations with elements

Select element or group of elements

To perform various operations with the construction or some part of the construction it is necessary to select the elements involved into the operations beforehand. Elements can be selected one by one or in a group. For this purpose use **Edit / Select Object**, **Edit / Select Group** or **Edit / Complex Selection** menu commands, respectively. Commands **Edit / Complex Selection by Box** and **Edit / Complex Selection by Circle** work in two modes: selection of object or selection of object group by box frame or circle.

Move

Any part of the construction can be moved to arbitrary position or joined with another part. This operation works with selected elements. An example of using this operation is shown below. This operation is initiated with **Edit / Edit Object** command. For detailed description of this command see *Chapter 2 Command Reference*.

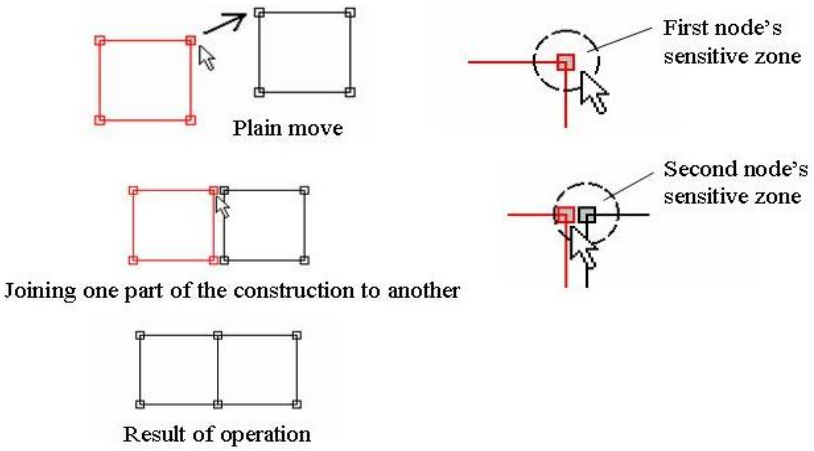


Fig. 1.22 Move operations

Copy / Paste

This pair of operations allows user to create a copy of a selected part of a construction in memory buffer and then paste the copied elements from memory into the construction editor. When pasted, the copy is selected and can be moved to the desired position. The copied elements can be pasted repeatedly.

See **Tools / Copy** and **Tools / Paste** commands from *Chapter 2 Command Reference*.

Multiply Structure

This tool allows user to create multi-sectional constructions. The parameters of the operation are intersectional distance vector and number of sections. Above that, the operation allows one to create solid elements from plates. When creating solid elements with multiplication tool it is necessary to remember that besides solid elements, plates are also created by copying the initial one on the multiplication vector. The operation is initiated with **Tools / Loft** command. See *Chapter 2 Command Reference*.

Examples of Multiply command application for different constructions with different parameters are shown below.

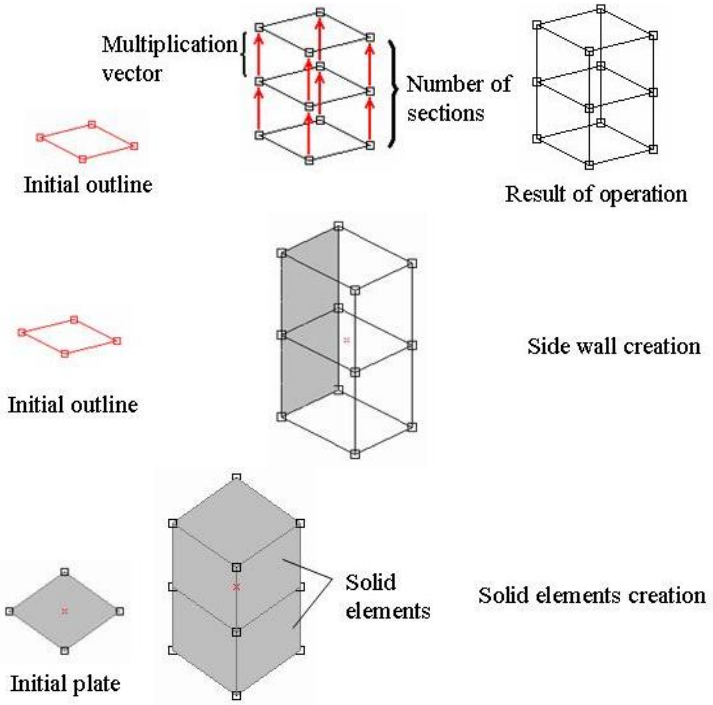


Fig. 1.23 Multiply operation

Loft

Loft operation can be considered as modified *Multiply Structure* operation. It creates multi-sectional constructions with linear dimensions alteration and sections rotation. The parameters of the operation are extrusion vector of one section, number of sections, dimension alteration factor and sections rotation angle. Operation works with selected elements. Multiplication vector is set for single section, therefore, for N sections the aggregate multiplication vector will be N times bigger. Rotation angle and dimension alteration factor are set for the total number of sections, therefore, for each section the rotation angle will be divided into N and dimensions will be altered linearly. When creating solid elements out of plates, one should remember that besides solid elements plates are also created by copying the initial one on extrusion vector. For more detailed see **Tools / Loft** command *Chapter 2 Command Reference*. Examples of structures built using this operation are shown below.

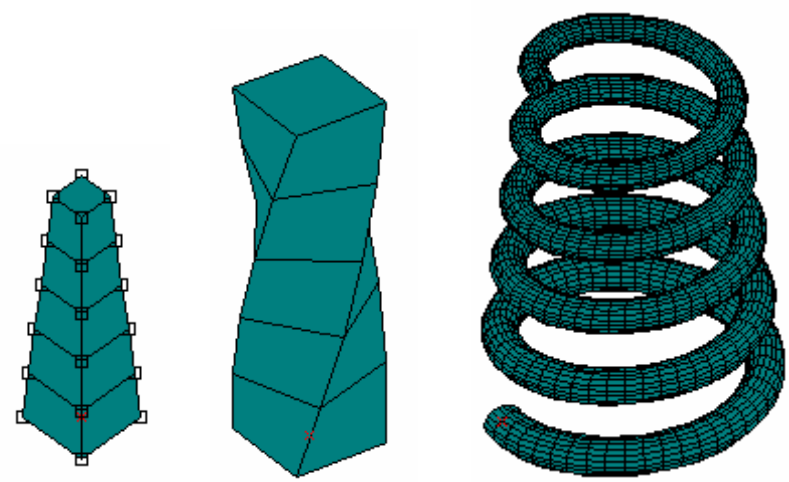


Fig. 1.24 Results of Loft operation

Rotate

Rotate operation allows user to rotate elements in the view plane, i.e. around the vector perpendicular to viewplane, therefore it is necessary to switch into the right view before performing rotation. Rotation is performed around the center of rotation. The operation works with selected elements. See **Tools /** **Rotate** command *Chapter 2 Command Reference*.

Mirror

Mirror operation allows user to create a mirror (symmetric) copy of a construction or part of a construction. Operation works with selected elements. Symmetry is built with respect to a symmetry plane, perpendicular to the view. To set the symmetry plane it is necessary to draw a line – symmetry plane trace in the viewplane. For **Tools /** **Mirror** command see *Chapter 2 Command Reference*.

Polar array

Polar array operation allows user to create a polar array of elements of the construction or a part of the construction. Operation works with selected elements. The parameters of polar array are rotation vector and full rotation angle. The operation can not only to copy but also to join successive copies of the constructions with rods, plates and solid elements (see *Multiply* command). Rotation angle is set for the total number of sections; therefore with N copies for each section rotation angle will be divided into N. An example of *Polar Array* operation execution, for arc of rod elements and creation of lateral elements, is shown below. For more details on **Tools /** **Polar Array** command see *Chapter 2 Command Reference*.

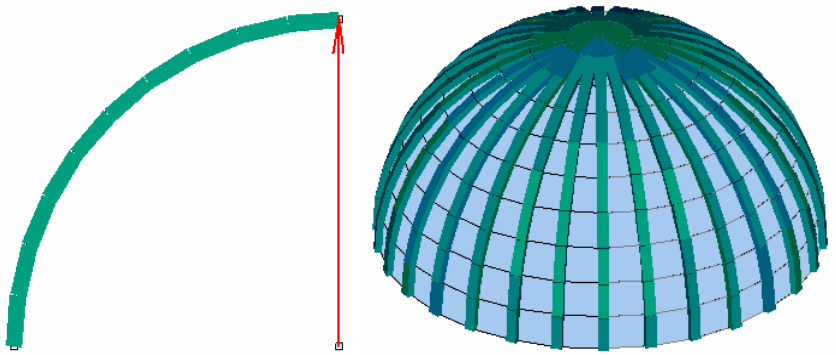


Fig. 1.25 Polar array operation

Nodes alignment

Nodes alignment operation allows user to align selected nodes by the coordinates of the base node. With the help of this tool one can “project” nodes onto a plate that goes through the basic node and is parallel to one of coordinate planes, or onto a plane that goes through the basic node and is parallel to one of coordinate axes. For more details on **Tools /** **Node Alignment** command see *Chapter 2 Command Reference*.

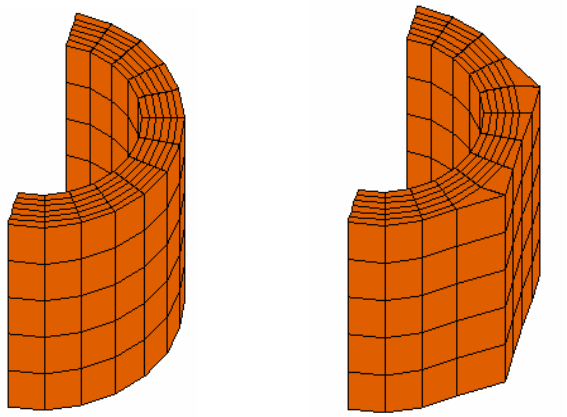


Fig. 1.26 Nodes alignment operation

Supports

The editor operates with different types of supports. Support in editor is regarded as a parameter of a node. Support is defined by the following parameters: restrictions on node displacement along X, Y, Z axes and restriction on node rotation around X, Y, Z axes. In an arbitrary node of a construction, displacement can be restricted in all or only in some directions. In general, it is possible to restrict 6 displacements: 3 linear and 3 rotational. Setting supports is about selecting restricted degrees of freedom (motion) of a node. Freedom degrees' directions are set in the node coordinate system. By default, coordinate system of a node coincides with global coordinate system. It is possible to restrict node displacement in any direction by appropriate setting of a local coordinate system in it.

Draw / Support / **Rigid Support** and **Draw / Support /** **One-Dir Rigid Support** commands set a support placement mode.

Besides, elastic supports can be set in a node. Elastic supports are characterized by stiffness for corresponding degree of freedom (motion). For translational motion, it is a force in case of a unit displacement of a node in a given direction, for rotational motion, it is a torque at one-degree rotation around a given axis. Elastic support is set for the local coordinate system of a node by **Draw / Support /** **Elastic Support** command.

Loads

Loads that act on a construction can be applied as nodal loads, rod element loads, and plate loads. Moreover, a load can be applied to a construction through supports displacement or as its dead weight.

Node loads

Concentrated forces and moments can be applied to a node. Nodal load is set in a global coordinate system. Nodal loads are set using **Loads / Force on Node**, **Loads / Moment on Node** menu commands.

Rod loads

The following types of loads can be applied to a bar element: concentrated forces and moments, trapezoid and uniformly distributed forces and moments, temperature loads, prestrain. Concentrated forces and moments can be applied in an arbitrary point of a rod and are set in a coordinate system of the rod. Distributed forces can be set both in the coordinate system of a rod and in the global coordinate system. Distributed moments are set in the coordinate system of a rod. Loads are applied to a rod using **Loads / Local Load on Rod, Loads / Global Load on Rod, Loads / Rod Temperature** and **Loads / Rod Prestrain** menu commands.

Rod temperature

Temperature load can be applied to rod elements using **Loads / Rod Temperature** menu command. Temperature values are defined as shown in the Fig. below – 8 points in general case.

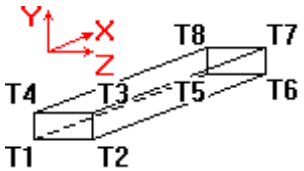


Fig. 1.27 Rod temperature in points

Rod Prestrain

For a rod element, it is possible to set prestrain in the form of: lengths of the non-deformed element, relative deformation, force operating in an element, or strain. Prestrain is set using **Loads / Rod Prestrain** command.

Plate and solid loads

Plate load is represented as uniform load on a plate using **Loads / Plate Distributed Load** and **Loads / Plate Linear Distributed Load** menu commands, or on a solid element using **Loads / Pressure on Solid.**

Support displacements

A load can be applied to a structure through of supports' displacements. Displacements can be translational and rotational. In calculations, only the displacements in direction of the fixed degrees of

freedom are taken into account. Supports' displacements are set in a local coordinate system of a node using **Loads /** **Support Displacements** menu command.

Dead weight

Dead weight is applied to the whole model and can be a part of any load case with an arbitrary proportionality factor. Dead weight can be specified using **Loads / Load Cases** menu command.

Acceleration

Acceleration is applied to the whole structure can be a part of any load case. It is possible to set linear and angular accelerations by **Loads / Acceleration / Linear Acceleration** and **Loads / Acceleration / Angular Acceleration** respectively.

Concentrated mass

Concentrated mass can be placed in any structure node using **Draw / Node / Mass** command. This mass is taken into account when calculating eigen frequencies and forced oscillations. Before assigning mass to a group of nodes select them using **Edit / Select Object, Edit / Select Group** or **Edit / Complex Selection by Box** and **Edit / Complex Selection by Circle** commands. Mass values are set in the node local coordinate system.

Snow load

Snow load can be applied to structure in automatic mode in accordance with building regulations[1]. Snow load can be applied to plate/shell elements only. In order to apply this load to rod elements one should cover the area in question with plates with zero stiffness (**Properties / Enable Plate Stiffness** menu command) so that their only function should be load transfer. Snow load can be defined using **Loads / Plate Snow Load** menu command. See *Chapter 2 Command reference* for further information.

Wind load

Wind load is a pressure load varying with height of the structure. Average component of this load can be defined in automatic mode in accordance with building regulations[2]. Following parameters need to be specified: aerodynamic factor, standardized wind pressure (by selecting wind region, or manually) and region type.

Wind load can be applied to plate/shell elements only. In order to apply this load to rod elements one should cover the area in question with plates with zero stiffness (**Properties / Enable Plate Stiffness** menu command) so that their only function should be load transfer. Wind load can be defined using **Loads / Plate Wind Load** menu command. See *Chapter 2 Command reference* for further information.

Temperature load on plates

Temperature can be applied either as gradient load varying with thickness or as varying over plate surface. These loads are applied using **Loads / Plate Temperature** and **Loads / Plate Linear Temperature** menu commands. Generally, temperature is defined by two values at each node, one at surface with positive z coordinate (along Z axis in the local coordinate system) and one on the opposite surface.

---

[1] SNiP 2.01.07-85* w/modif. 2003 (Loads and effects) as in p. 5.1*.
[2] SNiP 2.01.07-85* w/modif. 2003 (Loads and effects) as in p. 6.3*.

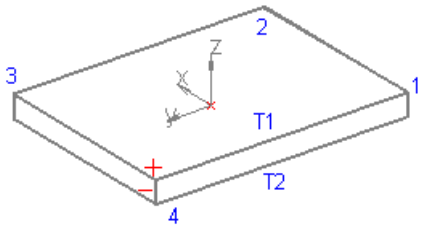


Fig. 1.28 Plate temperature in points

Temperature at node

In order to make thermal calculations one has to define boundary conditions - temperature in structural nodes. It can be done by **Loads /** **Node Temperature** menu command. Results from thermal analysis include temperature distribution over the model in question, which - in turn - can be used in further static analysis.

✍ *Note: thermal analysis must be done before or at the same time with static analysis in order to take temperature distribution into account.*

Importing structure model

Structure model can be imported into **APM Structure3D** from *.DXF files, from *.FRM files created in **APM Studio** or from *.DAT and *.BDF files (NASTRAN Bulk Data File). An example of model imported from APM Studio and analyzed in APM Structure3D is shown below.

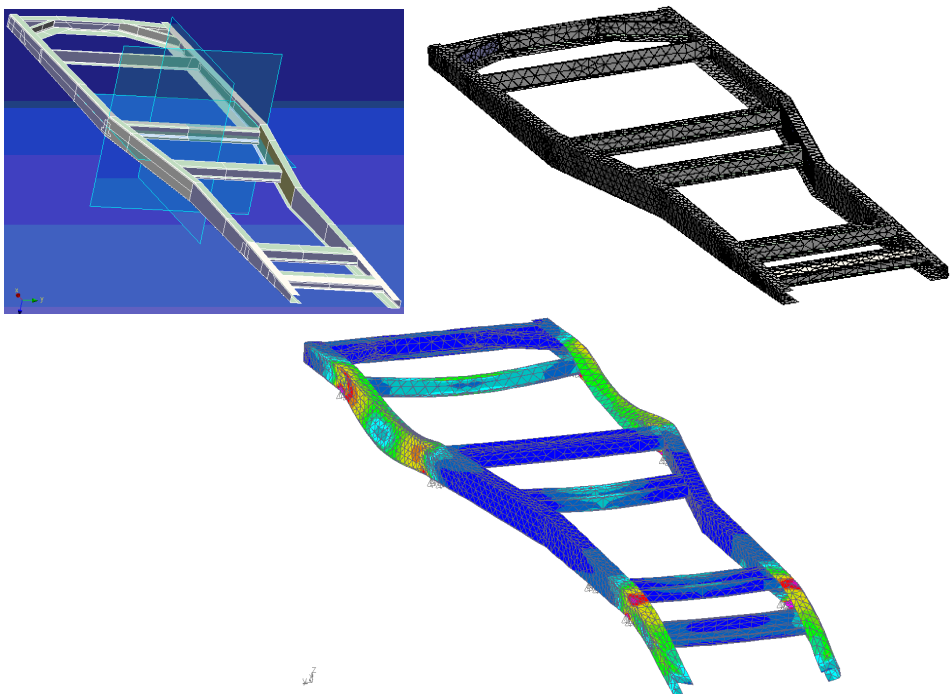


Fig. 1.29 Model imported from APM Studio

The design model created in *Autodesk Revit Structure* can be transferred to *APM Structure3D*. Conformity of transferred elements is presented in the table below.

Table 1.2 – Conformity of transferred elements

Autodesk Revit Structure	APM Structure 3D
Beam	Rod
Column	Rod
Brace	Rod

Truss	Truss
Wall	Plate
Floor	Plate
Slab	Plate
Boundary Conditions	Support
Point Load	Concentrated force
Line Load / Line Load with Host	Local load on rod / normal distributed force

Besides the elements listed in table 1.2 material parameters, rod sections and plate thickness are transferred for all elements from Autodesk Revit Structure to APM Structure 3D.
For correct model conversion it is necessary, that the following information has been presented in revit.ini file:
[ExternalCommands]
ECCount=1
ECName1=Save Model as Structure3D
ECClassName1=APMRevitToStructure.RevitStruct3D
ECDescription1=The user can select several of the views and when OK is clicked a new sheet is generated.
To transfer model use Tool / External Tools / Save Model as Structure3D command of Autodesk Revit Structure. After the end of converting process save the received file in APM Structure3D *.frm format. Example of transferred steel structure model is presented on Fig. below.

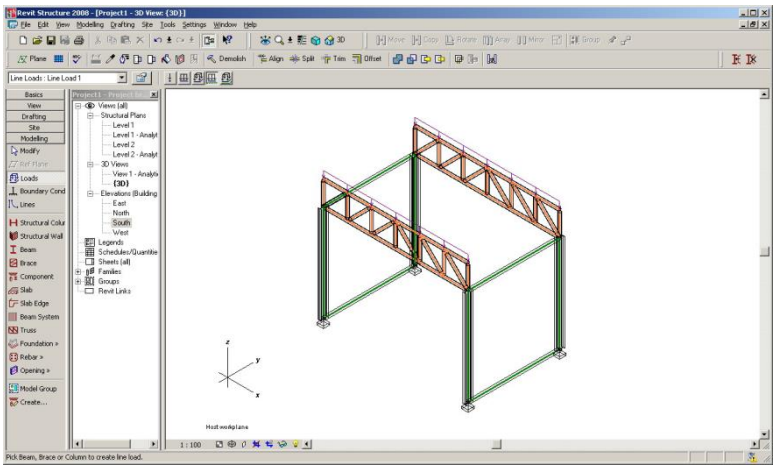


Fig. 1.30 Steel structure model in Autodesk Revit Structure 2008

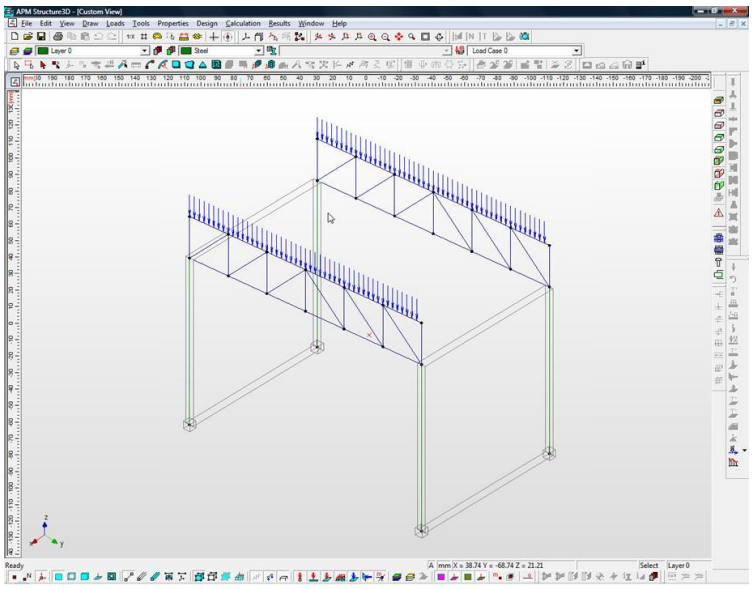


Fig. 1.31 Steel structure model in APM Structure3D

APM Structure3D supports Import/Export of Lira text files. Conformity of transferred elements is presented in the table below.

Table 1.3 – Conformity of transferred elements

Lira	APM Structure3D	Comments
1. Nodes		
Constraint	Support	rigid constraint
Release	Release	
	Hinge at rod end	
Rigid links	Rigid links	
2. Rods		
FE type: 2, 3, 5, 10, 205, 210, 309, 410	Rod-Beam	
FE type: 1, 4	Rod-Truss	
FE type: 310	Rod-Cable	
Stiffness		
Local CS	Local CS	for all sections, except the standard: angle, cross, nonsymmetrical T-section
Section type	Section type	for standard sections
Sections: standard	Section + Material	1. with 3D visualization; 2. section geometric parameters are calculated in APM Structure3D
Sections: steel from database	Section + Material	1. without 3D visualization; 2. section geometric parameters – for axisymmetric sections only, material properties
Sections: arbitrary geometry	Section + Material	1. without 3D visualization; 2. section geometric parameters and material properties
3. Plates		
FE type: 11, 12, 19, 21, 22, 23, 24, 27, 30, 41, 42, 44, 221, 222, 223, 224, 227, 230, 241, 242, 244, 281, 291, 292, 294, 341, 342, 344, 441, 442, 444, 521, 522, 523, 524, 527, 530.	DKT plate with fictitious stiffness	8-noded FE 27, 30, 227, 230, 527, 530 as 4-noded
FE type: 45, 46, 47	MITC plate with fictitious stiffness	
Stiffness	Thickness + Material	
4. Loads to node in Local CS and Global CS		
Force	Force at node	Global CS
Moment	Moment at node	Global CS
5. Loads to rod in Local CS and Global CS		
Concentrated force	Concentrated force	Local CS
Concentrated moment	Concentrated moment	Local CS
Distributed force	Distributed force	Local CS
Distributed moment	Distributed moment	Local CS
Trapezoidal load	Trapezoidal load	non-uniform distributed force and moment in Local CS
6. Plate load in LCS and GCS		
Uniform distributed load	Plate load	
Non-uniform distributed load		

7. Load cases		
Dead weight (distributed load)	Distributed load	for rods: linear distributed load for plates: area distributed load
User defined	User defined	load case name and loads
8. Materials		see rod and plate stiffness

APM Structure3D allows to import rod model from started *Kompas3D V11 – Steel Structures 3D* (developer "Ascon") using **File / Import from Kompas3D** menu command. Transfer of nodes, rods, materials and sections taking into account their orientation is supported. For the subsequent analysis it is necessary to set supports and loads in APM Structure3D.

Load cases

Load case can involve combination of loads of any kinds and is characterized by name and two states: on/off and active/inactive. Behavior of a construction can be calculated for any load case or a combination of load cases. Operating with load cases is similar to that with layers. If a load case is switched off, its loads will not be displayed on the screen. If a load case is active, a new load will be placed into the active load case by default.

For creating a new load case or editing an existing load case **Loads /** **Load Case** command is used. This command calls a *Load Case* dialog shown below.

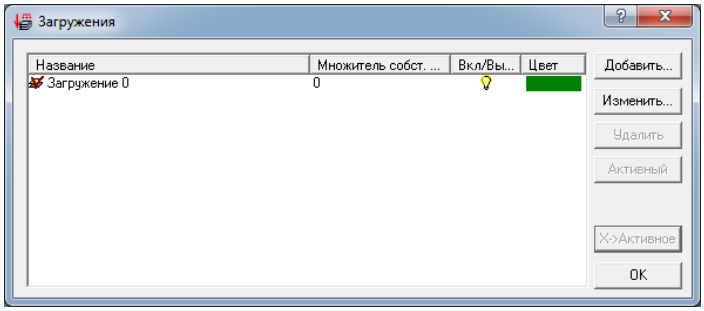


Fig. 1.32 Load Case dialog box

To create a new load case click on **Add** button.
To change an old load case click on it in the list and push **Modify** button.
To remove a load case, select it in the list and push **Delete** button.
Push **Set Active** button to make the selected load case active.
To move all loads from any load case to the active one, select the required load case in the list and click X -> Active button.

Dynamic load cases

APM Structure3D allows automatic application of seismic and dynamic wind loadings in accordance with building regulations[3].

All dynamic forces are defined using dynamic load case. All operations with dynamic load case are similar to those with static one, described above. It is recommended to use separate dynamic cases for each loading with the subsequent creation of load and code combinations.

✍ *Note: Dynamic wind load is a component of static wind load. To perform calculation for simultaneous static and dynamic wind action it is necessary to set static and dynamic wind in separate load cases and make calculation from its load combination.*

Use **Loads / Dynamic Load Case** menu command to create, modify or delete a load case. Dialog window is shown below.

[3] SNiP II-7-81 from 01.01.1996, SNiP II-7-81* from 01.01.2000 and SNiP 2.01.07-85.

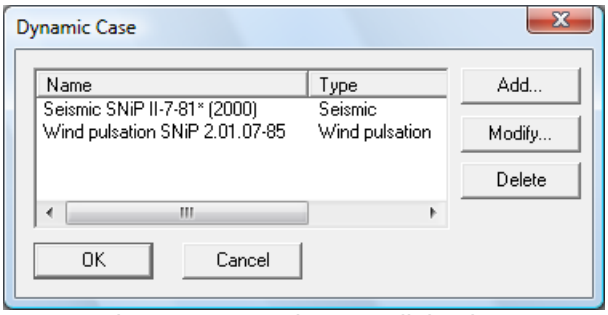


Fig. 1.33 Dynamic Case dialog box

Press **Add** button to create a new dynamic load case and then select the required type in the dialog window shown below.

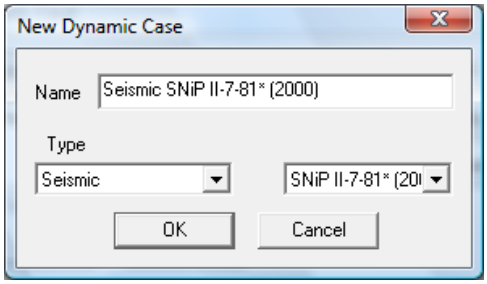


Fig. 1.34 New Dynamic Case dialog box

After pressing **OK** button new dialog window with seismic loading parameters will appear.

Seismic load direction is defined in *direction* group by projections on global Cartesian axes. Depending on the desired accuracy, certain *number of eigen shapes* can be taken into account. *Soil category*, *seismic type* and other parameters are selected according to building regulations.

Below you can find the description of controls of the dialog box in question.

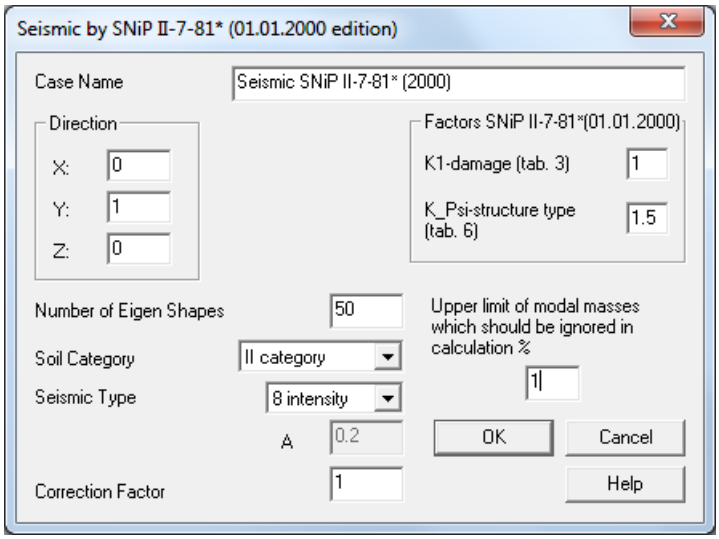


Fig. 1.35 Seismic SNiP II-7-81 dialog box*

Direction of seismic loading is set by cosines values of angles with respect to axes of the global coordinate system in *Direction* boxes.

Box *Number of eigen shapes* defines number of eigen shapes which will be used in seismic calculation.

Soil category and *Seismic type* are selected in accordance with building regulations.

In *Factors* boxes, factors K_1 *- damage* and K_ψ *- structure type* are set.

Correction factor is set for initial data correction. This factor can take any positive value, and is used as multiplication factor with the results of inertial forces calculation from seismic influence.

Upper limit of modal masses which should be ignored in calculation - frequencies will be ignored in seismic calculation if their modal masses are less than specified.

When calculating seismic effect, the design loads are accounted for by setting concentrated masses in corresponding nodes (see *Chapter 1 section Concentrated masses*).

The description of *Wind pulsation* dialog box is presented in Fig. below.

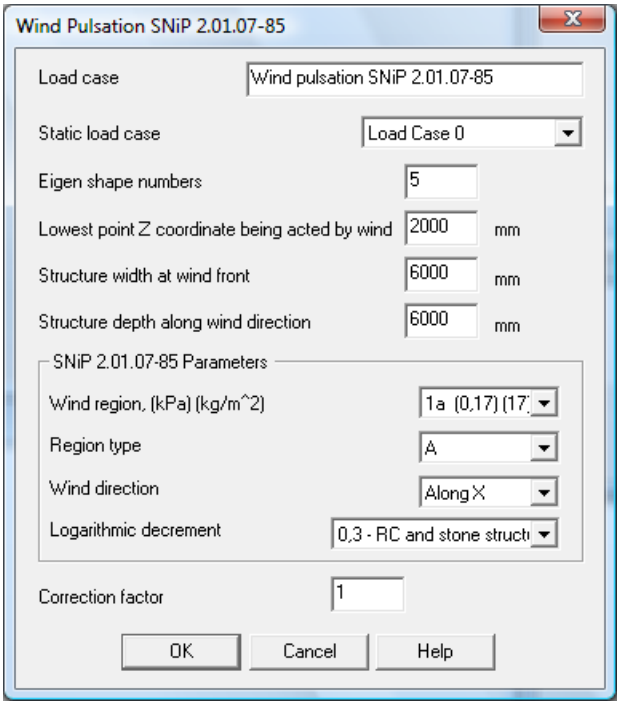


Fig. 1.36 Wind Pulsation dialog box

Load case edit box – the name of dynamic load case in the case list.

Static load case – name of load case with static wind load.

Eigen shape numbers – number of considered eigenvalues (4…6 recommended).

Lowest point Z coordinate being acted by wind – can differs from coordinate of the lowest structure point.

Structure width at wind front – structure width normal to wind direction.

Structure depth along wind direction – length of structure along wind direction.

SNiP 2.01.07-85 Parameters – wind region, *region type*, *wind direction*, *logarithmic decrement*.

Correction factor is set for initial data correction. This factor can be arbitrary positive value and multiply the results of inertial forces calculation.

Load combination

Load combination represents a linear combination of load cases. For creation of a load combination **Loads / Load Combination** command is used. This command calls a *Load Combination* dialog box shown below. To add load case into a combination it is necessary to select it in a load case list box, enter a factor for it and press **Add** button. To change a load case factor select the desired load case from the load case list box or factors list box, enter a new value in the *Factor* box and press the **Modify** button. To remove a load case from a combination, select the former in the load case list box or factors list box and press the **Delete** button.

It is provided to create several load combinations. Press the **New** button to create combination and type in combination name in the appeared dialog.

To add load case in combination select the combination name in the list.

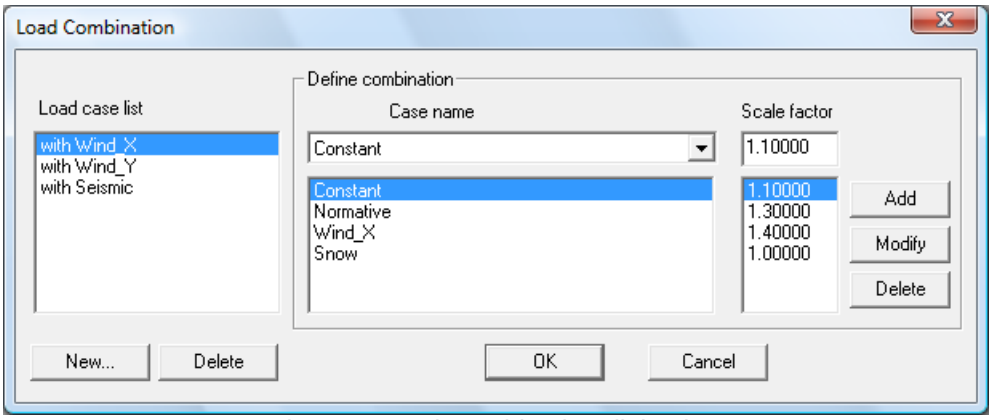


Fig. 1.37 Load Combination dialog box

Code combination

Calculation of the most dangerous code combination for rods is made on the basis of extreme values of several groups of parameters, namely: normal and tangent stresses in characteristic points of sections, longitudinal and lateral forces. Selection principles of load combinations and their factors are stated in building regulations[4].

To design code combination it is necessary to specify all the load cases involved. For this purpose, firstly, select load case from the list *Case name* where all available load cases are presented. Secondly, from the list *Case Type*, select the desired type. Types that are presented in this list, namely "Constant", "Temporary long", "Temporary short", "Specific", - comply with building regulations. "Static wind" load case is set as a separate type only one wind load can be used in code combination calculation. In other respects static wind load case contributes to the code combination as a usual temporary short load.

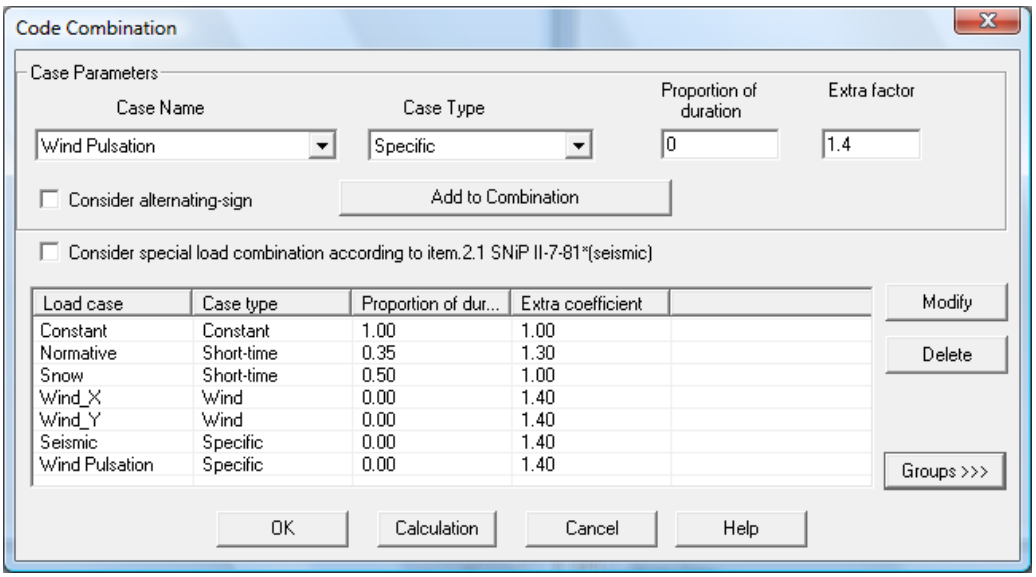


Fig. 1.38 Code Combination dialog box

Proportion of duration defines the value of load component (in proportion of unit) that is considered as temporary long in a given load case. Remaining part is considered as short-term. For constant and long-term load cases the duration is equal to 1; for short-term – 0.

Extra coefficient - additional multiplier for factors with which efforts from every load case are presented in the rated load combination (these factors are selected according to building regulations).

If a checkmark is placed in the *Consider alternating-sign* box, the corresponding load case participates in the code combination two times (both, as it is and with the opposite sign). In particular, it is used to specify seismic load. Sign-alternation can be set for *Specific* load case type only.

[4] SNiP 2.01.07-85.

After all parameters are specified, the load case is added to the code combination by pressing **Add to Combination** button. To change load case parameters select the load case, change the required parameters and press **Modify** button.

By pressing **Group>>>** button there is a list of load groups in additional part of a dialog box. **Create** button invokes dialog box for creation of loading groups: alternative, associated, coacting for accounting of additional conditions of code combination.

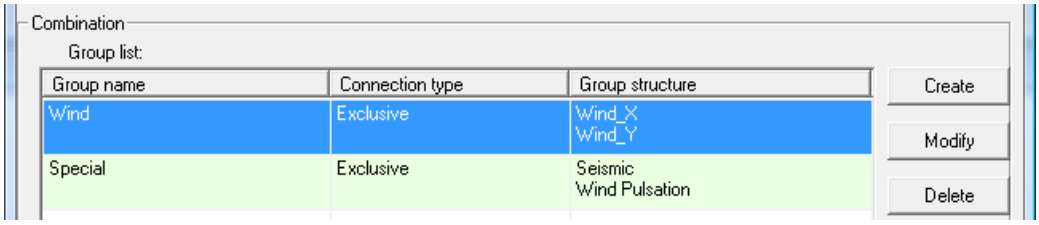


Fig. 1.39 Additional part of Code Combination dialog box

If *Consider special load combination according to item 2.1 SNiP II-7-81* (seismic)* box is checked, code combination coefficients are used from table 2 of item 2.1 SNiP II-7-81*. If this option is unchecked, code combination coefficients are used according to item 1.12 SNiP 2.01.07-85*.

When generating code combinations the following rules are used:

- Each load case or load combination can enter only once into a code combination except for those for which the duration is non-equal to 0 and 1. The duration shows what time loading is to be considered as long-term. Remaining part of time loading is considered as short-term.
- Only one special load case can enter into a special code combination.
- Only one wind load case can enter into a special code combination.
- Load cases of groups included into code combinations takes into account group type: alternative, associated, coacting.

Clicking **Calculation** button initiates the algorithm that calculates the most severe load combinations.

Current parameters

To accelerate selection of a current layer, material, section or load case *Current parameters* toolbar with drop-down menu are used.

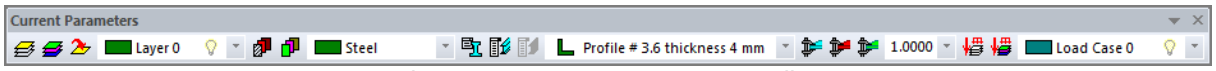
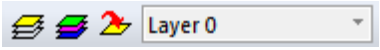


Fig. 1.40 Current Parameters toolbar

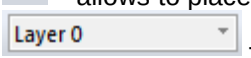


– *Layers* group

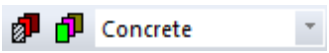

– invokes *Layers* dialog box


– shows/hides colored layers


– allows to place selected objects to an active layer



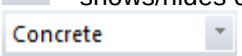
– drop-down menu for current layer selection



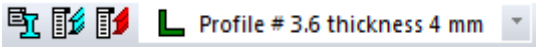
– *Material* group


– invokes *Material* dialog box


– shows/hides color materials



– drop-down menu for current material selection



– *Rod cross-section* group


– invokes *Cross-section manager* dialog box


– allows to assign cross-section to all rods


– allows to assign cross-section to selected rods

– drop-down menu for current section selection

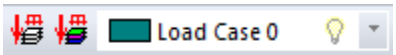
– *Plate thickness* group


– allows to set thickness to all plates


– allows to set thickness to selected plates


– allows to show only plates of current thickness

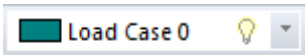
– drop-down menu for plates current thickness setting



– *Load cases* group


– invokes *Load cases* dialog box


– shows/hides color load cases



– drop-down menu for current load case selection

Chapter 2. Command reference

This chapter includes a complete description of all menu commands and dialog box options in the **APM Structure3D** program.

File menu

Commands of this menu allow user to create a new construction or a cross-section and to work with files and printing.

New / Structure

The command creates a new frame construction.
Shortcuts: or Ctrl+N

New / Cross-Section

The command creates a new cross-section.

New / Joint

The command invokes *APM Joint* for threaded, riveted or welded joint calculation.

Model from Template

The command calls the dialog window shown below, and allows to create a rod model using standard templates.

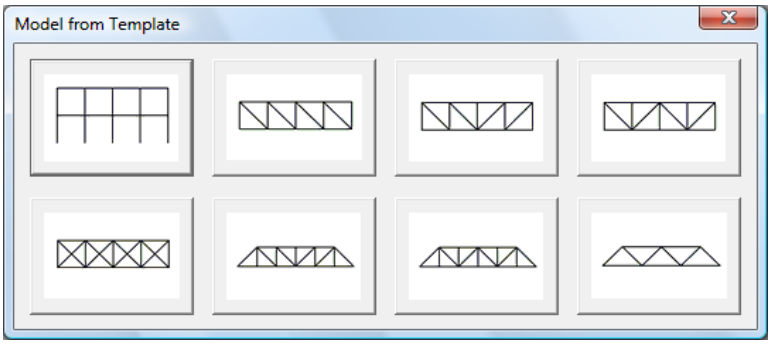


Fig. 2.1 Models from template

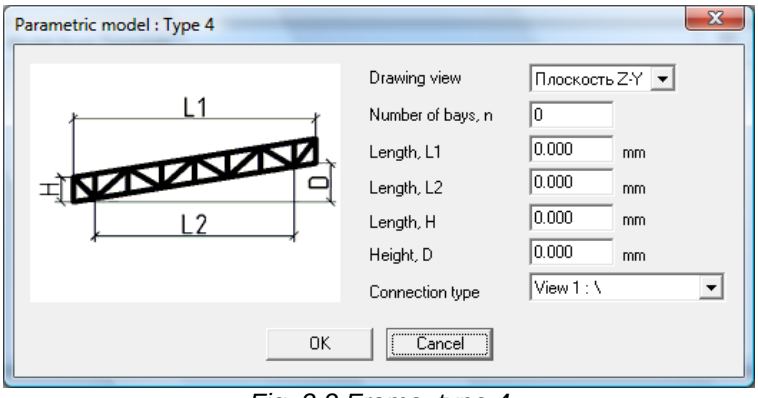


Fig. 2.2 Frame, type 4

After pressing the button with the Fig. of the required model there appears a dialog in which it is possible to set the dimensions and other parameters of the pattern.

Extra Models from Template

The command calls the dialog window shown below, and allows to create a rod model using extra templates.

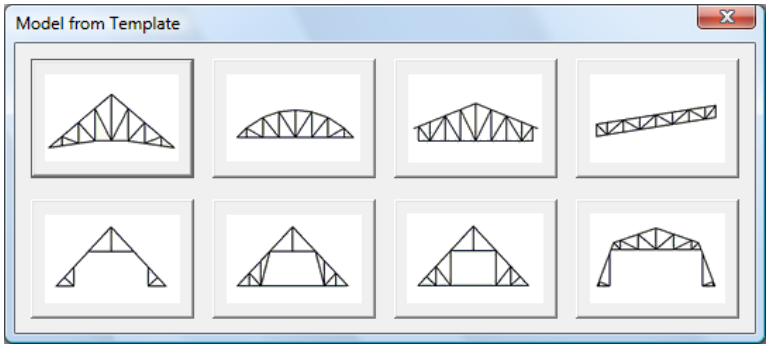


Fig. 2.3 Models from template

Open

the command loads a previously saved file. The following extensions are supported:

• APM Structure3D Files (* .FRM). button  ;

• APM Structure3D calculation result file (* .FRM), button  ;

A standard dialog box appears on the screen after the command is called. You can view the model saved in this file starting with v.15 in the preview window. If a model with calculation results saved in the same directory and with the same name as the model opens, the model will be loaded with the calculation results.

Shortcut: 

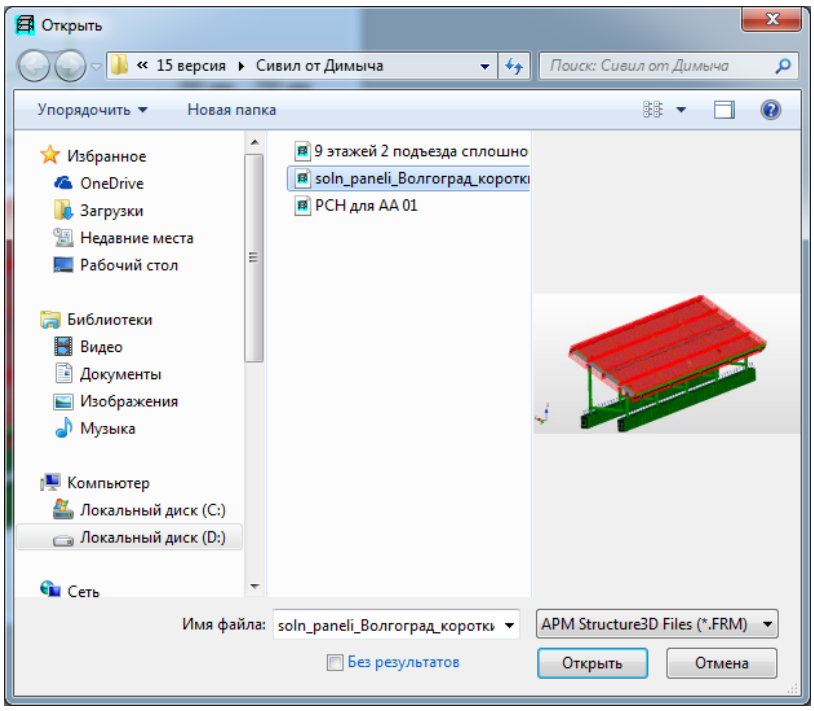


Fig. 2.4 Standard Open dialog box for loading previously saved files

Close

The command closes the active document. If the active document has been modified since the last save, the dialog box will appear on the screen and offer you to save the changes.

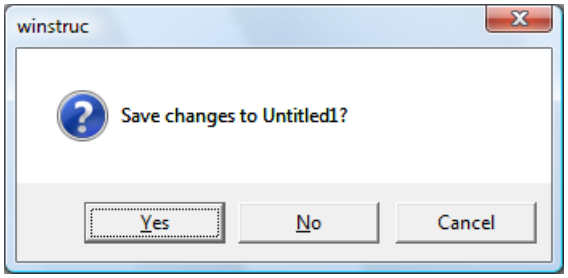


Fig. 2.4 Save changes dialog box

Save (Save As ...)

the command saves the active document to a file. If the document was not saved before or the **Save as** option is selected, the standard Windows dialog box shown in the figure below appears on the screen. The file can be saved without calculation results which will significantly reduce its size. The option can be used if the file is to be sent by e-mail.The calculation must be repeated in order to obtain the results.

One can switch the auto-repair feature in the dialog box that is done by the command **File | Settings** ...

Shortcut: 

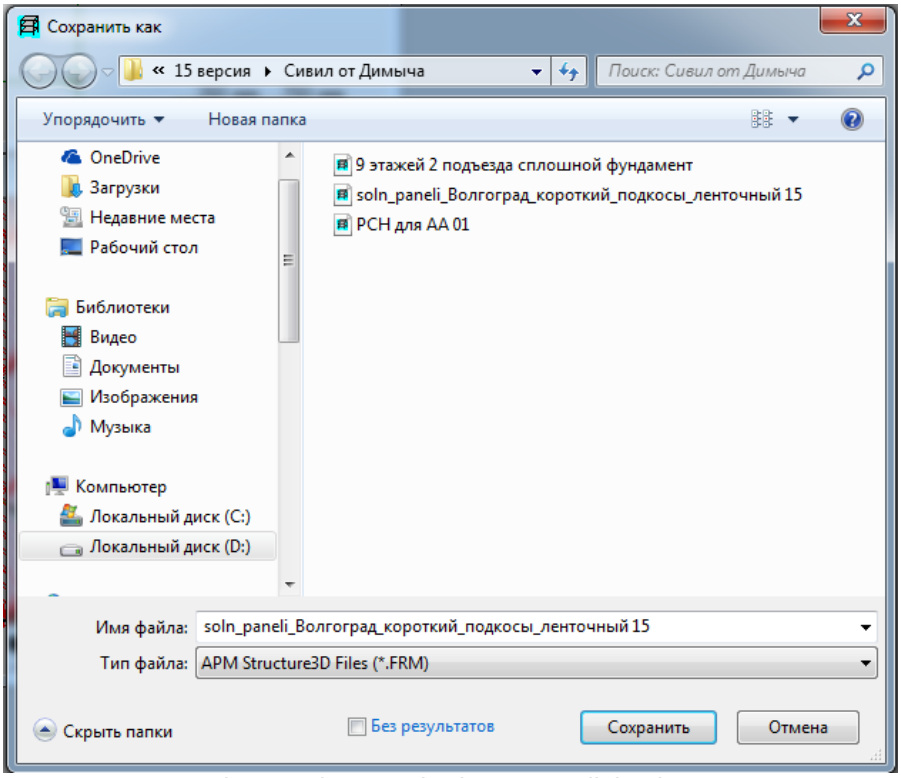


Fig. 2.5 The Standard Save As dialog box

Save / load results

The command allows to save the model calculation results in a separate file or load the calculation results from a separate file with the extension (* .FRMres),

Shortcut:  / 

Import

The command allows to import documents from files:

- DXF (*.dxf);
- DAT/BDF (NASTRAN Bulk Data File) (*.dat, *.bdf);
- Lira files (*.txt);
- MS Access Data Exchange (*.mdb)

The command imports DXF drawings, DAT / BDF (NASTRAN Bulk Data File) and SFM files (finite elements mesh generated in APM Studio) into the active document. For proper functioning you have to explode the construction into composing elements (AutoCAD command **explode**). The following DFX file objects are transformed into models of construction: LINE, POLYLINE, 3DFACE.

Export

The command allows to export models to:

- DXF 2D files (*.dxf);
- DXF 3D files (*.dxf);
- DAT/BDF (NASTRAN Bulk Data File) (*.dat, *.bdf);
- APM Graph Document (*.agr);
- Lira files (*.txt)

The command exports finite elements model to a file of format *.DAT / *.BDF (NASTRAN Bulk Data File). It can save frame model in DXF file format. All rods are converted to LINE objects and plates are converted to 3DFACE objects.

Import from Kompas3D

The command allows to import rod model from *Kompas3D V11 – Steel Structures 3D* (developer "Ascon"). Transfer of nodes, rods, materials and sections taking into account their orientation is supported.

Properties

The command invokes message box which contains information about current file version and program version.

Shortcut: **i** .

Settings

The command invokes *Settings* dialog box, where you can set *APM Structure3D* global parameters.

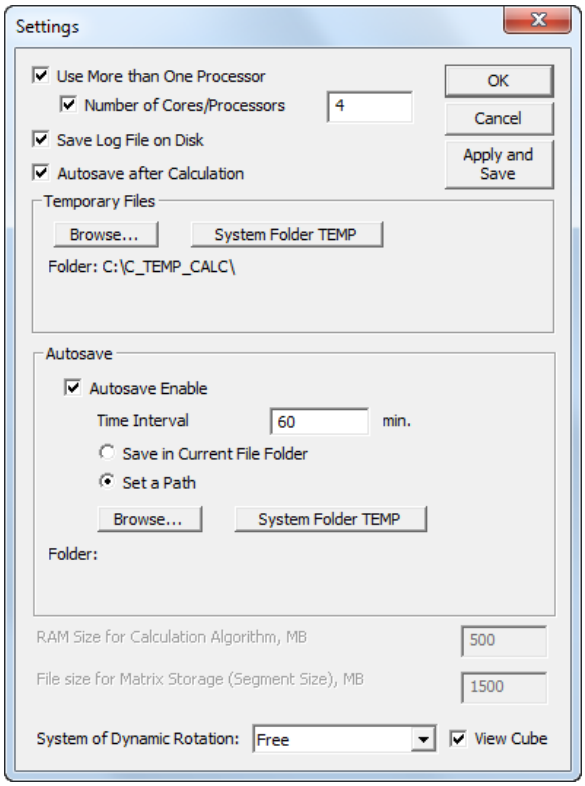


Fig. 2.5 Settings dialog box

- use more than one processor;
- save log file on disk;
- autosave after calculation;
- autosave with defined period during work;
- besides those autosave modes automatic restart of the program ("restart manager") is provided in case of incorrect shutdown. The model for restart saves each 2 hours;
- set a path to temporary file folder;
- select system of view rotation and display rotational cube.
- use the graphics processor in conjunction with CUDA (in particular available in solving the problem of topology optimization);
- switch (by a check mark) of an alternative results map;
- Set the transparency in the range of 0 - 1.0;
- Selecting the color scheme for the interface design of the APM Structure3D module

Shortcut: .

Print

The command allows you to print construction data and calculation results. This command calls the dialog box shown below. Set checkmark next to the information you want to have printed.

Shortcuts: or Ctrl+P

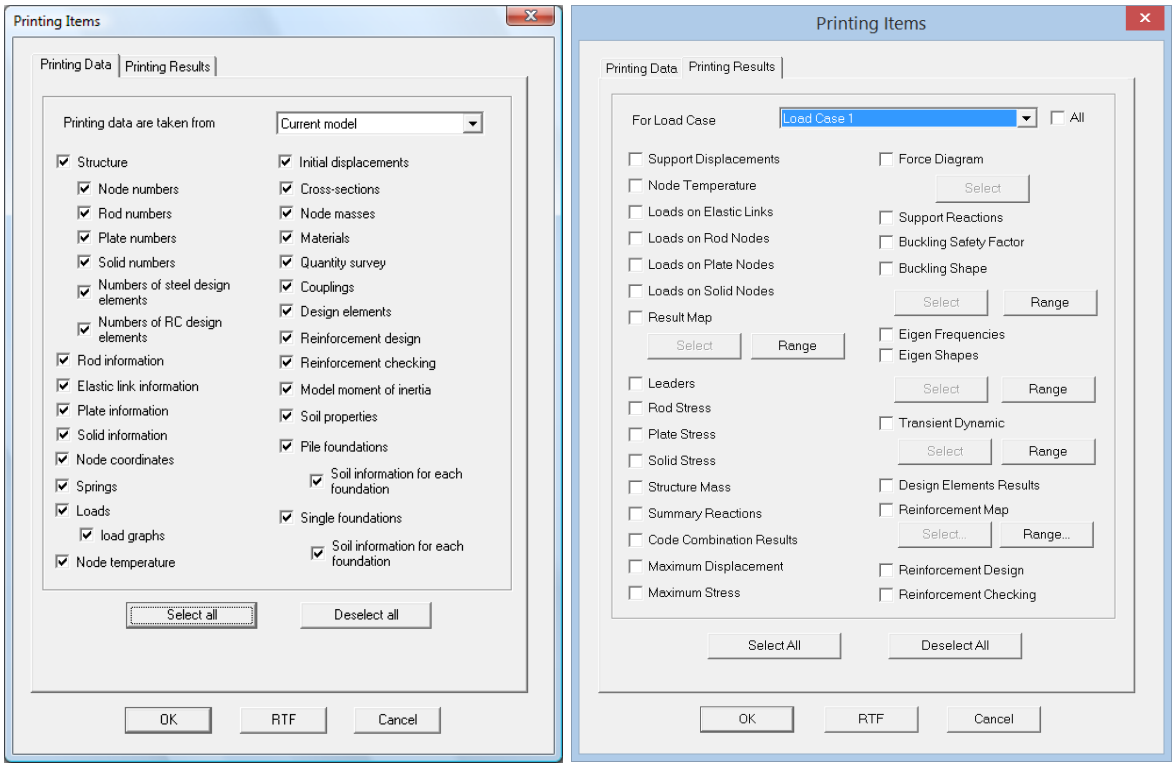


Fig. 2.6 Printing Items dialog box (printing data and results tabs)

The option **All**, located next to the drop-down list *Results for load case*, allows you to print results for all load cases.

Button **Select** under *Stress map* checkbox is used for stress component selection. For more detailed explanation about stress components see description of Calculation menu and Chapter 5.

Button **Select** under *Force Diagram* checkbox allows you to select forces for frame force diagram calculation. For more detailed description see description of Results menu and Chapter 5.

Button **Select** under *Forced oscillation* checkbox invokes following dialog box. This dialog box allows you to select time moments for base reactions and stress map.

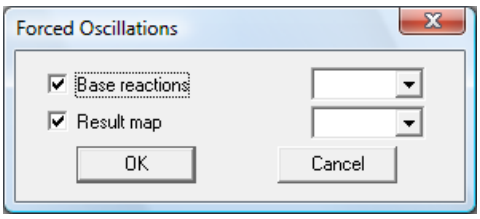


Fig. 2.7 Forced oscillation selection dialog box

Printer Setup

The command allows you to setup a printing device.

Most Recently Used Files

The command opens most recently used file. Command name corresponds to the file name. Menu can contain up to four commands of this kind.

Exit

The command closes current file and quits the program.

Edit menu

Commands of this menu allow you to select objects, invert selection and undo/redo one of last issued command.

All commands of select/unselect – are universal. In case of a single mouse click on an object the command works in the mode of a single select / cancel selection By pressed Shift button the unselection of earlier selected objects doesn't act.

Complex Selection by Box

This command is universal selector of editor. Clicking once command works in select object mode. Holding left button selection makes by box frame as in select group mode.

To switch editor in complex selection mode press Esc key.

Shortcut: 

Complex Deselection by Box

Holding left button deselection makes by box frame as in select group mode.

Shortcut: 

Complex Selection by Circle

This command is universal selector of editor. Clicking once command works in select object mode. Holding left button selection makes by circle frame as in select group mode.

Shortcut: 

Complex Deselection by Circle

Holding left button deselection makes by circle frame as in select group mode.

Shortcut: 

Select Object

The command enables object selection mode – you can select rods, nodes, plates, and solid elements.

Select Group

The command turns on the selection mode for a group of elements. To select a group of elements user has to an auxiliary rectangle in any view so that it should cover elements he wants to select. The first mouse click sets the first corner of the rectangle; the second mouse click set the opposite one. Right mouse click cancels operation.

Making selection with this command from left to right all elements which completely gets to a frame will be selected. Making selection from right to left all elements which are crossed by a frame will be selected and if one node of element at least lies inside a frame.

Edit Object

The command switches editor into the mode which allows selecting elements – nodes, rods, plates etc. To select an element, click close to it. The selected element is highlighted in a different color. When you select another element, the previous element is deselected. But it is possible to select several elements simultaneously. To do that hold SHIFT key while selecting. To deselect individual element click the right mouse button.

This mode allows you also to move selected elements. To do this, place cursor over the selected elements, press left button and holding it drag the selection across the screen. After you release the mouse button the new position will be locked.

You can also use attachment mode to attach selected elements to the specific node. To use attachment mode left-click in the sensitivity zone of the node that you want to attach and then drag the selection in the viewplane to get the node into the sensitivity zone of the node you want to attach to, and release mouse button. As a result, the whole selection moves to connect two nodes. Al coinciding nodes are connected at the same time.

You should remember that if some nodes coincide in the viewplane, then the one nearest to the viewplane will be selected. To select a particular node, it is necessary to move the viewplane closer to the desired element or perform selection in a different viewplane where node projections do not coincide.

Shortcut:

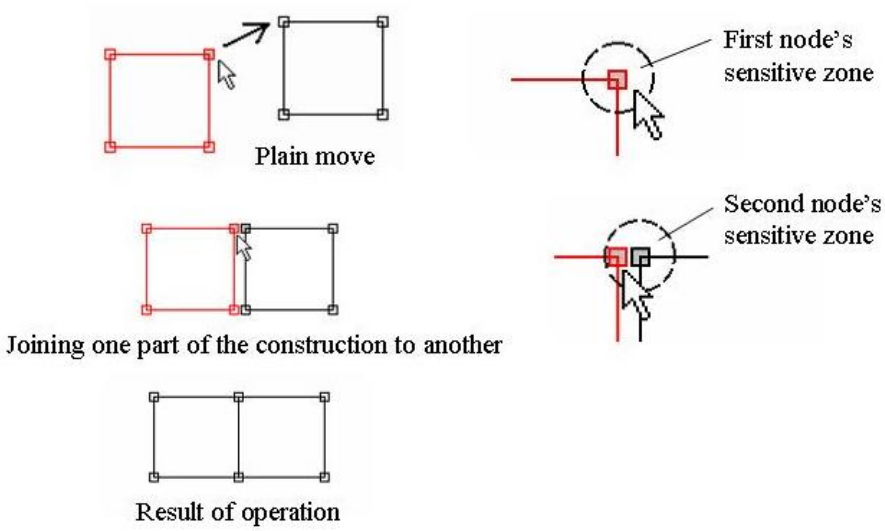


Fig. 2.8 Example of movement operation

Invert Selection

The command inverts construction selection.

Select All

This command allows to select all objects of the editor.

Undo

The command cancels last command performed.

Shortcut:

Redo

The command repeats last command cancelled.

Shortcut:

Undo Enable

The command enables/disables undo/redo support.

View menu

Commands of this menu allow you to change view settings.

✍ *Note: before selecting most of these commands it is necessary to activate the view you want to change.*

Status Bar

The command toggles status bar on and off.

Plane Position

The command invokes dialog box that allows you to change view plane position. This can be done in two ways: changing viewplane *depth* or changing viewplane position vector. *Method* radio-buttons allow you to select any method.

Shortcut:

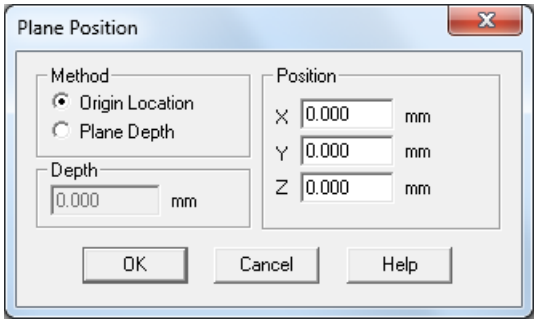


Fig. 2.9 View Position dialog box

Set Depth

The command switches on the mode that allows moving viewplane in the direction, perpendicular to the viewplane itself. This mode is useful when you want to work with different parallel planes. This mode changes the depth only for the *main* views by moving the plane trace on the screen. For example, you can set depth of the *Up* view in the *Front* or *Left* views. After selecting this command the trace of the chosen viewplane is shown in *main* views. Depth is set by clicking left mouse button in the view. This mode allows using node attachment.

Shortcut:

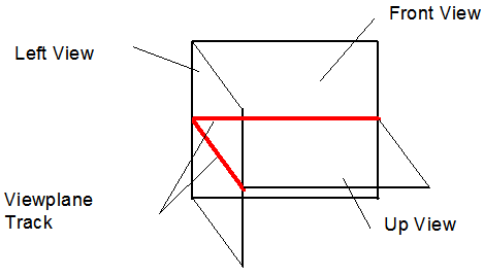


Fig. 2.10 View depth explanation

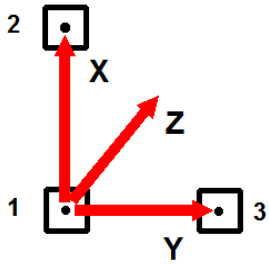


Fig. 2.11 View depth explanation

Set View Plane by 3 Nodes

The command switches on the mode that allows you to define a viewplane using three selected nodes. After calling the command select sequentially three nodes in any view. As a result, viewplane will be rotated and moved so that its origin will be located in the first node and axes will have the configuration shown in the Fig. above.

Shortcut:

Show Dimension Scale

This command shows dimension scale in active views of editor.

Shortcut:

Dynamic Rotation Mode

The command enables the mode of dynamic view rotation. The first left mouse click in the view starts rotation, mouse movements along vertical and horizontal axes rotate the view at θ and φ angles, respectively. The second mouse click stops rotation. Right mouse click cancels operation and returns the view to initial rotation.

Shortcut:

Use Local Coordinates

The command enables and disables usage of local coordinates in a view. Local coordinates are 2D coordinates related to the viewplane.

Grid

This command allows changing auxiliary grid settings. A dialog box appears on the screen as shown below.

Shortcut:

Cursor Step

This command allows you to set cursor linear and angular steps and radius of the sensitivity zone of the node. A dialog box appears on the screen, as shown below.

Shortcut:

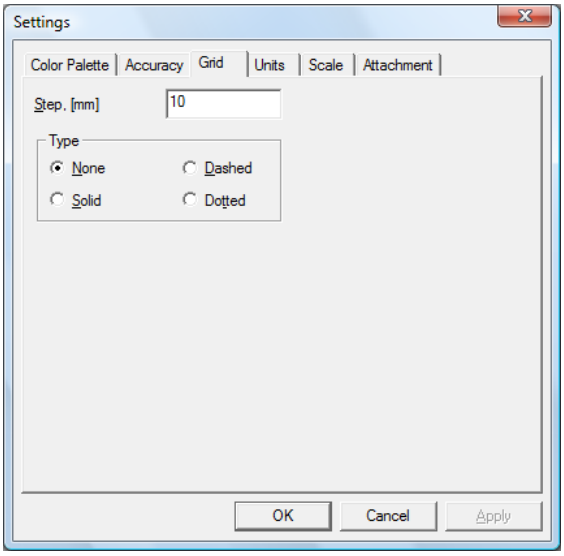


Fig. 2.12 Grid tab

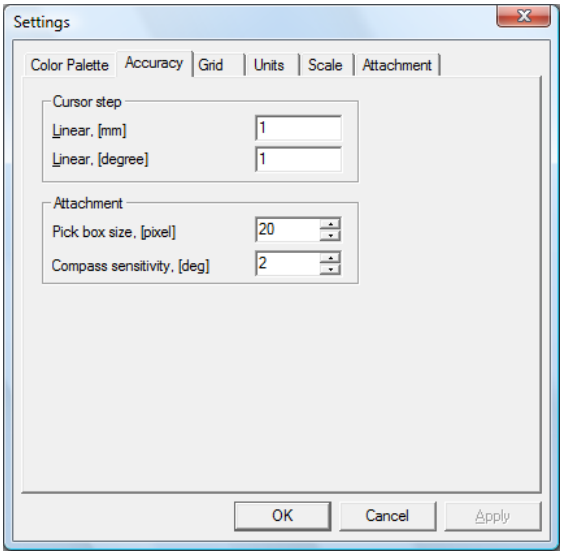


Fig. 2.13 Cursor property tab

Palette

This command allows you to change colors of all construction editor elements.

Shortcut:

Units

This command allows you to set units for entering initial data and viewing calculation results of current structure document. Defined units save in *APM Structure3D* configuration file and will be used for the next created file. A dialog box appears on the screen, as shown below.

Shortcut:

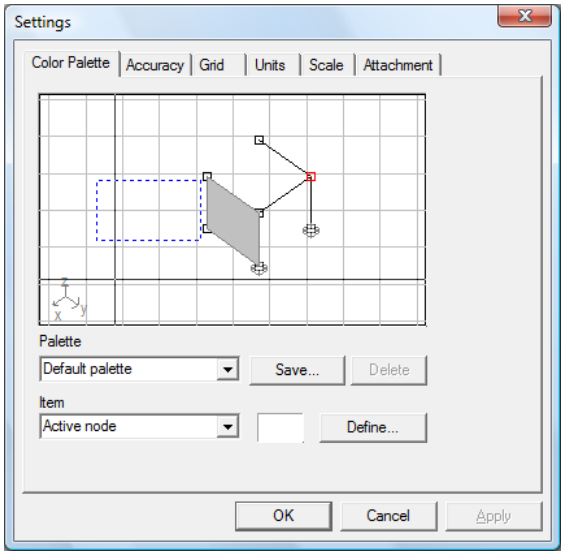


Fig. 2.14 Palette tab

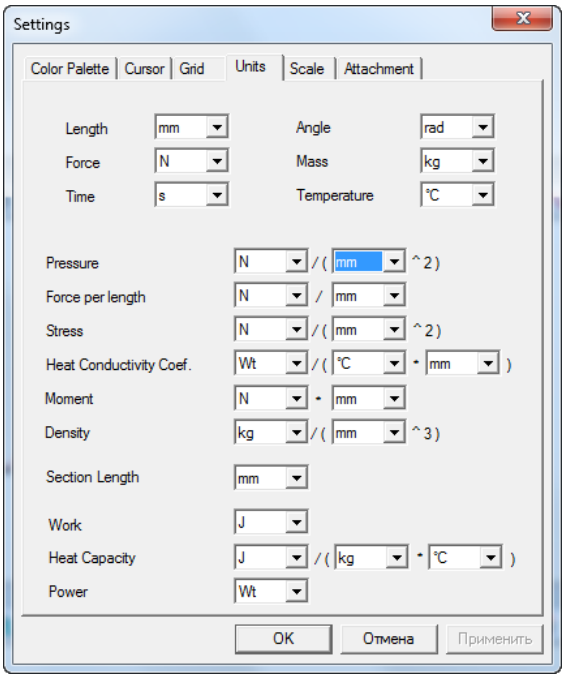


Fig. 2.15 Units tab

Scale

This command allows you to change image scale for the active view. A dialog box appears on the screen, as shown below. *Reduce* buttons allow user to decrease image scale, *Enlarge* buttons – increase it. Input box allows you to enter scale value manually.

Shortcut: 1:X

Attachment

This command allows you to choose one of the two snapping modes.

Shortcut:

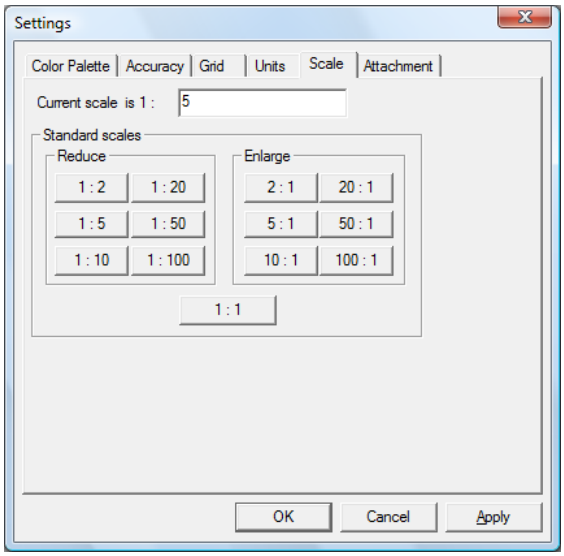


Fig. 2.16 Scale tab

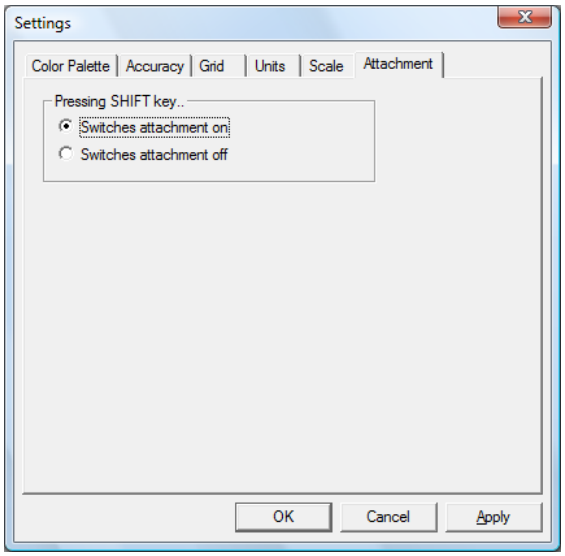


Fig. 2.17 Attachment tab

Pane View

The command enables the mode of view scrolling.

Shortcut: 

Zoom

The command enables the mode of enlarging rectangle region of a view to the whole window.
Shortcut: 

T-square On/Of

The command enables / disables T-square tool.
Shortcut: 

Zoom In

The command increases image scale.
Shortcut: 

Zoom Out

The command decreases image scale.
Shortcut: 

Show All

The command changes the view scale so that the entire model fits in the window.
Shortcut: 

Settings to All Views

The flag applies all property sheets settings to all views simultaneously.
Shortcut: 

View Set / Standard

Default view set with predefined Global CS of each view.

Arbitrary View

Toggles on/off arbitrary view page on/off

Left View

Toggles on/off left view page

Up View

Toggles on/off up view page

Front View

Toggles on/off front view page

Draw menu

This menu commands allow you to create and modify frame constructions.

Node / By Coordinates

The command switches editor into the mode for drawing nodes. To create a new node click in the view in the point with desired coordinates. Right mouse click near the node calls the dialog box which allows you to edit node coordinates.

Shortcut: 

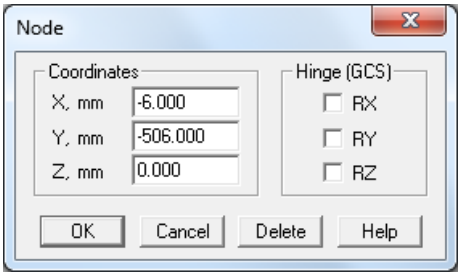


Fig. 2.18 Node dialog box

Node / On Rod

The command sets the mode which allows user to create nodes on rod or on the rod extension. Click left mouse button to select rod, then move the node that appears, using mouse, along the rod axis. Second mouse click locks node position and calls the dialog box which lets you edit node position on rod. Node position can be defined in two ways: with absolute coordinate, counting from one of the rod ends, or with the ratio between full rod length and length of one of its parts. Radio-buttons *Node Position* allows you to choose one of the two methods. Radio-buttons *Counting from* allow to choose one of two nodes as frame. While setting node position using absolute coordinates, this coordinate will be counted from the selected node. While using relative coordinates, the used in the relation will be the one to which the selected node belongs. The selected node is highlighted in all views. *Place new objects in* option allows to select layer in which new objects will be placed.

Shortcut: 

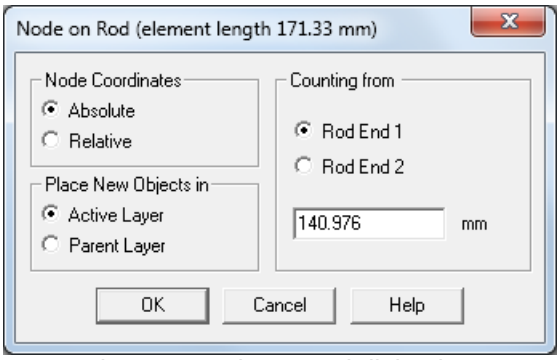


Fig. 2.19 Node on Rod dialog box

Node / Local Coordinate System

The command switches editor into the mode that sets local coordinate system for selected nodes. In the node-based coordinate system, fastenings, elastic supports, displacements in supports (movements in the directions of fixed degrees of freedom) are set. To set a coordinate system in one or several nodes, you are to switch on this mode and select the necessary node or one of the selected nodes by clicking on it. Then a dialog box of coordinate system orientation setting will appear on the screen. A local coordinate system is set by three successive rotations of initial coordinate system that coincides with global coordinate system around Z, Y’ and X’’ axes. Y’ and X’’ markings are used instead of Y and X to show that Y and X axes change their orientation after rotation. You can also

define a new position of local coordinate system by pressing **Specify** button and then gradually selecting two points. New X axis will pass through the node and first picked point. Second point defines new XY plane and direction of new Y axis. Z axis is created so that XYZ forms a right-hand system. Push **Delete** button to delete a local coordinate system in selected nodes and use global coordinate system.

Shortcut: 

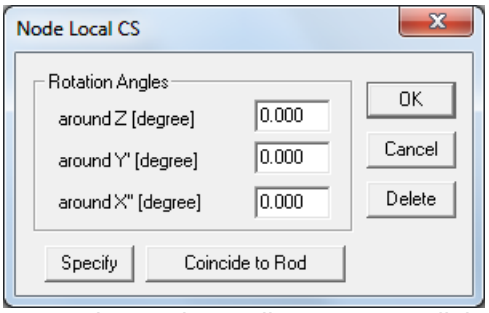


Fig. 2.20 Node Local Coordinate System dialog box

Node / Mass

The command switches editor into concentrated mass placing mode. To place mass in a node or a group of nodes click left mouse button on the required node or one of the selected nodes. *Node mass* dialog box will appear as a result. Nodes are selected using **Complex Selection** command. Mass values are in the local coordinate system of node.

Mass values in particular directions are specified in corresponding edit boxes.

If you want to add mass values to existing ones in a particular node, select radio button *Add to existing*. Otherwise, if you want *to replace existing mass* by new values select radio button *Replace existing*. To delete masses in selected nodes select the **Delete** button.

Shortcut: 

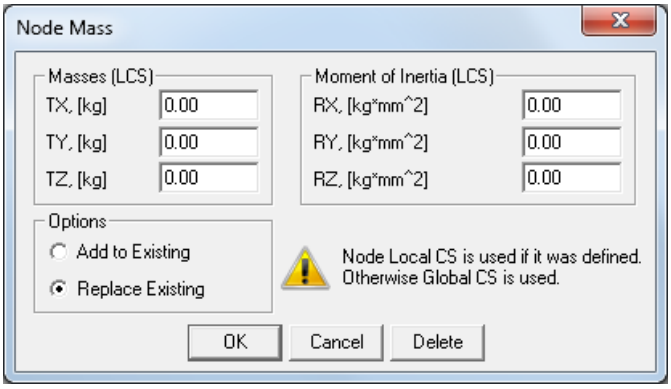


Fig. 2.21 Node Mass dialog box

Rod / By Coordinates

The command switches on the mode of drawing rods. In this mode, existing rods can be joined and new ones can be created. First click sets the first node, second click – the second one. If you have selected or created the first node you can undo this command by clicking right mouse button. The command uses attachment mode while joining existing nodes. If you want to create a node instead of selecting an existing one (for example while nodes belong to different planes, but their projections coincide in the viewplane), you should cancel *attachment*.

Shortcut: 

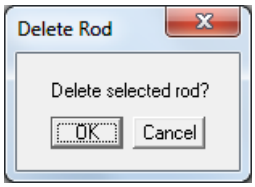


Fig. 2.22 Delete Rod dialog box

Rod / By Length and Angle

The command switches on the mode for drawing rod based on its length and direction. A rod can be created only from existing node. Angle can be counted from horizontal or from another rod. To have angle counted from horizontal select node. To have angle counted from rod, select rod first and then select one of its nodes. After that moving mouse set desired angle. Next mouse click locks the direction. After that moving mouse you can set only rod length. Last mouse click creates rod. Right mouse click calls a dialog box, which allows you to correct angle and length values, or cancel operation.

Shortcut: 

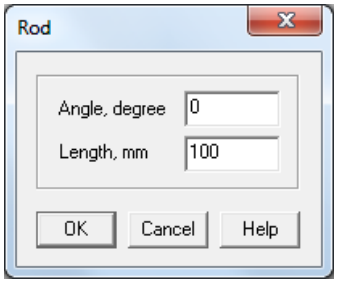


Fig. 2.23 Rod dialog box

Rod | Divide Rod into N Rods

The command switches editor into the mode which allows you to split one rod into a number of elements of the equal length. This command invokes a dialog box which allows you to enter the number of rods or the length of rods after dividing. *Place new objects in* option allows to select layer in which new objects will be placed.

Shortcut: 

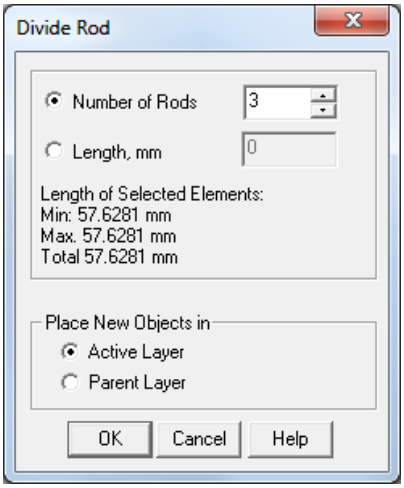


Fig. 2.24 Divide Rod dialog box

Rod | Hoop

The command allows you to strengthen the existing core element in a hoop having specified thickness. To do the command it is necessary to select core elements with a specified section, after the activation the Section dialog box appears where the cross section is selected.

Pipes| Straight Pipe

The command allows to create straight segment of a pipe with the cross section by a type "ring". Straight segment of a pipe is being created based on the available nodes.

It is possible to convert existing rods to pipelines. For that you need to set the properties "Pipe Segment" to these rods.

Shortcut: 

Pipes| Tee pipe

The command allows to create a straight segment of the pipeline at any place of whose one can draw the tap at the right angle.

It is possible to convert existing rods to pipelines. For that you need to set the properties "Pipe Segment" to these rods.

Shortcut: 

Pipes| Pipe Bend by 3 Points

The command allows to create curved pipe (a part of a pipeline curved with constant radius) for connection of two straight segments between each other. The curved pipe by three points is also being created as the arc:

- First, the center point of the arc is specified;
- Then, the node of the first connecting pipeline;
- The node of the second pipeline.

We will note, the curved pipe can be created between the straight segments of the pipeline where in the places of the tap connection the pipelines will be tangents to an elbow arc. Therefore the center of the curve, based on these considerations, must be chosen. In all the rest of the cases a message that it's impossible to connect chosen segments with a curved pipe.

Shortcut:: 

Pipes| Pipe Bend by 2 Rod Ends

The command allows to create a curved pipe (a part of a pipeline curved with constant radius) for connection of two straight segments between each other. Only the nodes of those pipelines which are separating to equal lengths from a point of their intersection can be connected by this command.

Fulfillment of the command needs next:

- click by LMB on the first pipeline that is connected close to the end to which the tap must be join;
- Click by LMB on the second pipeline that is connected close to the end with which the tap being connected must be join.

Shortcut: 

Pipes| Pipe Bend by 2 Rods and Radius

The command allows to create a curved pipe (a part of a pipeline curved with constant radius) for connection of two straight segments between each other. This command creates an elbow of two pipes (the ones that are being crossed or are not crossed) by the type of creation of rounding between segments. In this case the «extra» parts of pipelines will disappear.

Fulfillment of the command needs next:

- Click alternately by LMB on the first pipeline and then on the second pipeline that is connected;
- In the dialog window "Elbow Radius" that appeared, one must specify the value of radius with which the tap between bars will be mounted.

The connected straight segments of pipes will extend if a value of radius requiring lengthening of pipelines for their rounding with specified radius is entered and in entering will be connected by the tap with specified radius.

Shortcut: 

Plate / Rectangular 4-noded

The command enables a mode for creating rectangular plates. Plate is set by four nodes. In this mode existing nodes can be joined and new ones can be created. First mouse click sets the first node; second mouse click sets the second one and so on. If you have already selected or created the first node right mouse click cancels operation. The command uses attachment mode while selecting existing nodes.

Shortcut: 

Plate / Arbitrary 4-noded

The command enables a mode for creating arbitrary 4-noded plates. Present command is similar to the previous one; the difference being that the created plate can represent an arbitrary quadrangle.

Shortcut: 

Plate / 3-noded

The command enables a mode of creating 3-node plates. Plate is set by three nodes. In this mode existing nodes can be joined and new ones can be created. First mouse click sets the first node; second mouse click sets the second one and so on. If you have already selected or created the first node right mouse click cancels operation. The command uses attachment mode while selecting existing nodes.

Shortcut: 

Plate / Arbitrary with Mesh

The command enables mesh generation mode. This mode allows you to split a complex straight-line contour down to finite elements – plates. Every mouse click adds one node to a circuit, to close a circuit it is necessary to select its first node. It is also possible to add individual nodes belonging to a circuit. Double click on a node set's individual node that will be included into breaking mesh. ENTER key completes contour creation and generates mesh.

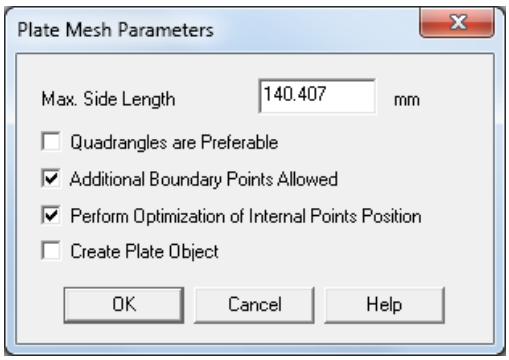


Fig. 2.25 Plate Mesh Parameters dialog box

Quadrangles are preferable checkbox – if the box is unchecked, triangular FE mesh creates by default. *Additional boundary points allowed* checkbox – in addition, those contour sides are meshed which lengths exceed the specified *maximum side length*. *Perform optimization of internal points position* checkbox – if the box is unchecked, angles of plate FE can be inside the required range 30˚–150˚. *Create Plate Object* checkbox allows to create plate object that characterized by the same properties as a plate (thickness, material etc.), but consisting of several plates.

Shortcut: 

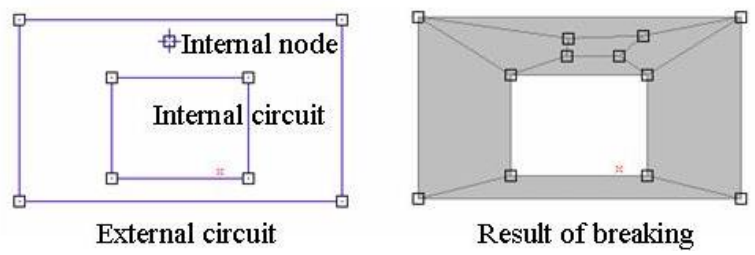


Fig. 2.26 FE meshing

Plate / Divide Plate

The command enables a mode that allows you to split a plate into smaller plates. The mode allows splitting single plate or a selected group of plates. *Number of elements* defines the number of elements along each of the two directions of the plate coordinate system. *Element type* sets the type of elements. Initial plates are deleted. *Thickness for new plates* checkbox allows to set thickness for plates created by meshing. If this option is unchecked, newly created plates will be the same thickness as initial plate. *Place new objects in* option allows to select layer in which new objects will be placed.

Shortcut: 

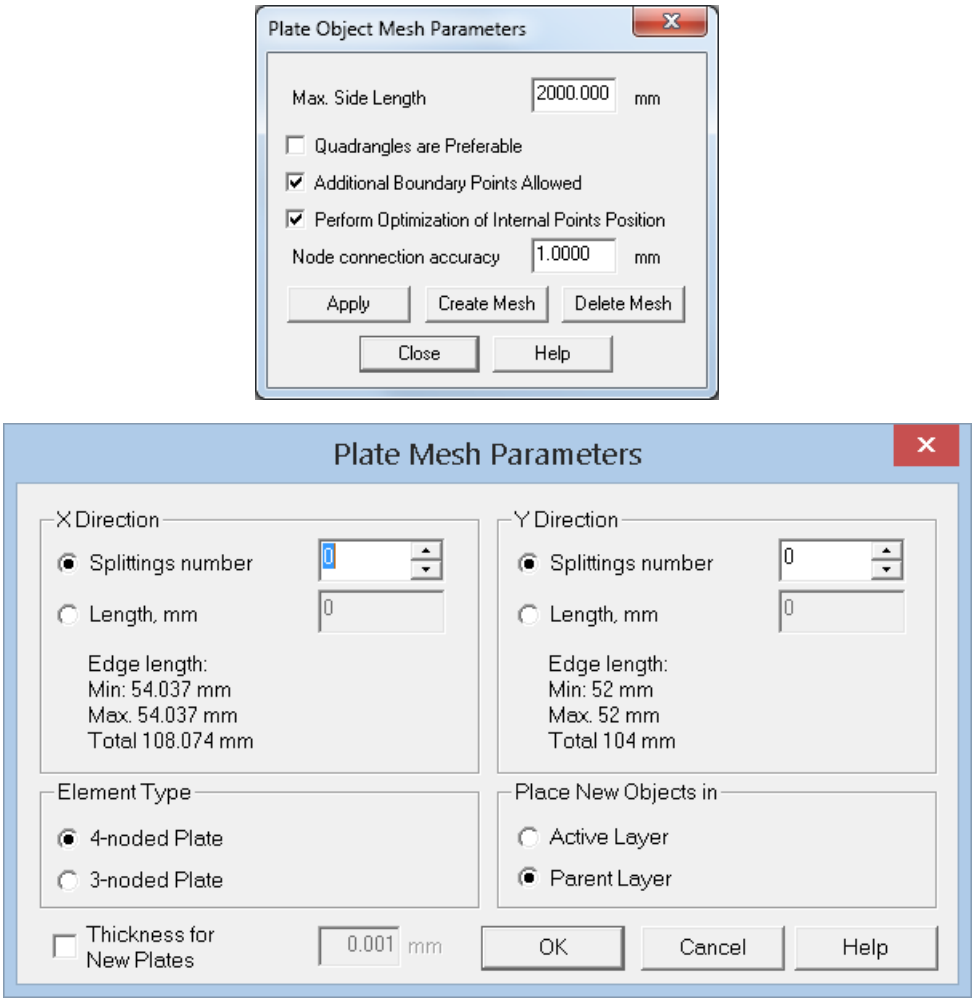


Fig. 2.27 Plate Mesh Parameters dialog box

Plate / On Free Faces of Solid Elements

The command allows to create plates on free faces of solids. Such necessity can arise at creation of combined structures consisting of solids and shells.

Shortcut: 

Plate Object / Create Plate Object

This command invokes *APM Graph* where you can create plate contour. For detailed information see *Chapter 1.*

Plate Object / Edit Plate Object

This command invokes *APM Graph* where you can modify earlier created plate contour.

Plate Object / Mesh Options

This command invokes dialog box where you can set mesh parameters for selected plate objects.

To view plate mesh use **Filters toolbar /**  **Plate Object Mesh** command.

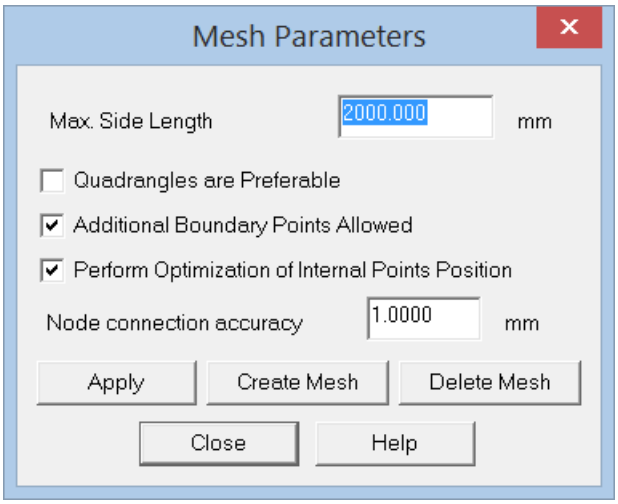


Fig. 2.28 Plate Object Mesh Parameters dialog box

Plate Object / Delete Mesh

This command deletes FEM mesh for selected plates.

Plate Object / Explode Plate Object

This command deletes selected plate objects and creates plate elements based on FEM mesh of plate objects.

Plate| Sphere of 4-Noded Shells

The command creates a sphere of triangular plate elements since simultaneous generation by the FE mesh. The user specifies the spherical center by a click of the left mouse (LMB) button on a selected point of one of the windows and then specifies by dislocation of a mouse the value of radius of the sphere that is created. After the completion of the sphere creation the dialog window of the sphere creation opens, where one can specify parameters of an object that is created, and of the FE mesh.

The value of maximum size equal by default the FE is one tenth out of the sphere radius that is created, but the radius value can be changed by the user.

Shortcut: 

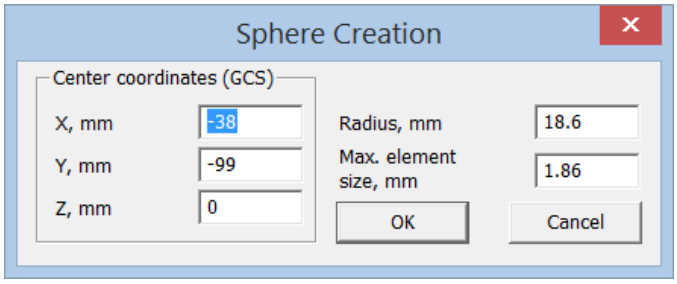


Fig. 2.34 Dialog window The sphere creation.

Plate | Cylinder of 4-Noded Shells /

Plate | Cylinder of 3-Noded Shells

The command creates a cylindrical surface of quadrangular/triangular plate elements. The user specifies the cylinder center by a click of the left mouse (LMB) button on a selected point of one of the windows and then specifies, by dislocation of a mouse, the value of the cylinder radius and its height. After the completion of a sphere creation the dialog window *of a cylinder creation* opens (Fig. 2.36), where one can specify parameters of the object that is created and the FE mesh.

The cylindrical surface seemingly can be created without ends as well as with different ends.

The value of maximum size equal the FE by default is one tenth out of the sphere radius that is created, but the radius value can be changed by the user.

Shortcut: 

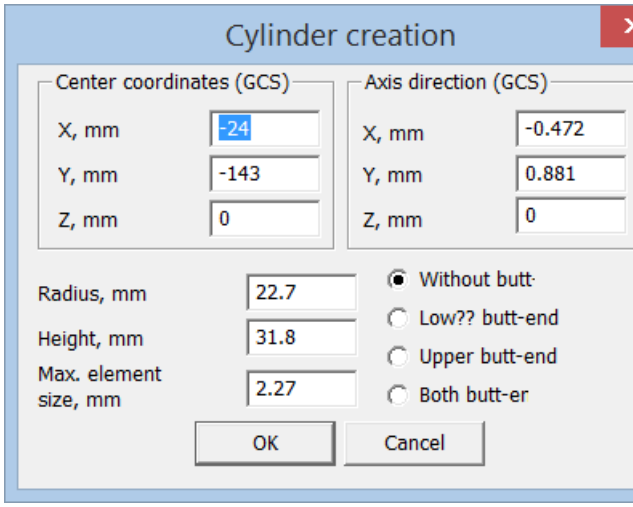


Fig. 2.36 Dialog window of a cylinder creation.

Plate| Torus of 3-Noded Shells

The command creates the tours surface from triangular plate elements. The user defines the center of the torus, the point defining torus axis, the center of the forming circumference and its radius by sequential clicks of the left mouse (LMB) button on the selected point of one of the windows. After the completion of a sphere creation the dialog window *of a torus creation* opens (Fig. 2.37) where one can specify parameters of an object that is created and the FE mesh.

A value of a maximum size equal to the FE, by default is one tenth out of the second radius of a torus that is created but the value of radius may be changed by the user.

Shortcut: 

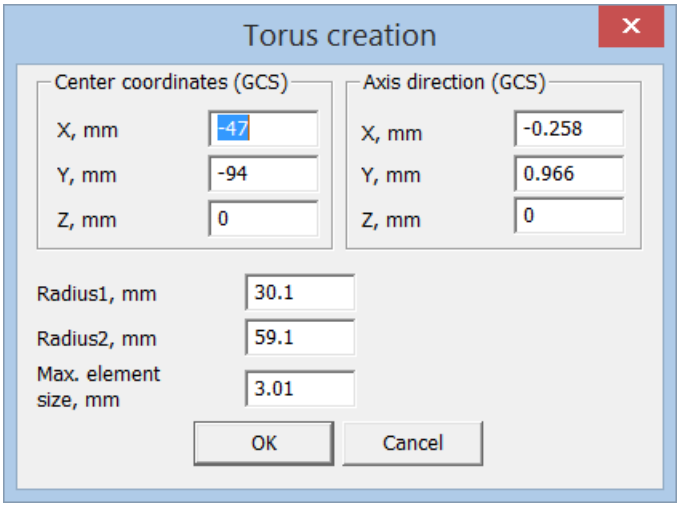


Fig. 2.37 Dialog window of a torus creation.

Plate | Torus of 4-Noded Shells

The command creates the torus surface of quadrangular plate elements. The user defines the center of the torus, the point defining torus axis, the center of the forming circumference and its radius by sequential clicks of the left mouse (LMB) button on the selected point of one of the windows. After the completion of the sphere creation, the dialog window *of the torus creation* opens (Fig. 2.38), where one can specify parameters of an object that is created and the FE mesh.

The value of the FE maximum size is equal by default to one tenth out of the second crating torus radius, but the value of radius can be changed by the user.

Shortcut: 

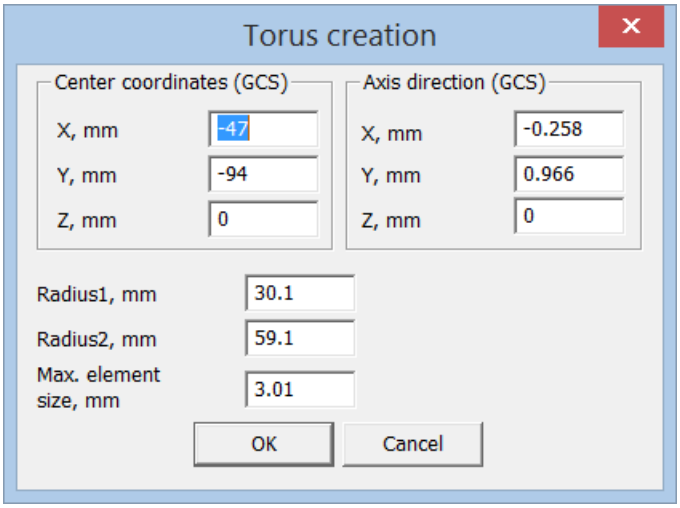


Fig. 2.38 Dialog window of a torus creation.

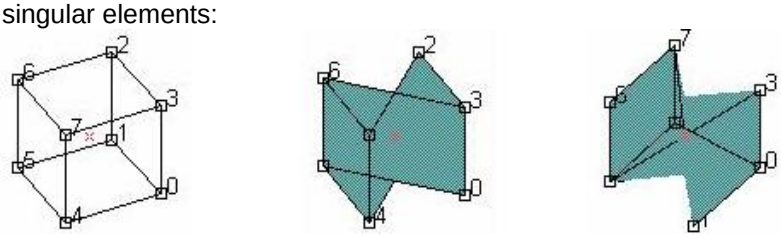
Solid / 8-noded Solid

The command enables a mode of creating 8-node solid elements. An element is defined by 8 nodes. In this mode, the existing nodes can be unified and new ones can be created. First mouse click sets the first node; second mouse click sets the second one and so on. If you have already selected or created the first node right mouse click cancels the operation. The command uses attachment mode while selecting among existing nodes.

While entering FE nodes it is necessary to observe local node numeration. The required order of nodes input is the following: 0-1-2-3-4-5-6-7, 0-3-2-1-4-7-6-5, 0-1-5-4-3-2-6-7, 0-4-5-1-3-7-6-2 and so on.

A degenerate element will be created in case of incorrect numeration order.

Examples of singular elements:



correct incorrect

Fig. 2.29 8-noded solid element creation

Shortcut: 

Solid / Divide 8-noded Solid

The command enables a mode of splitting 8-noded element into smaller elements. The splitting mode allows you to split a single element or a group of selected elements. *Number of Elements* sets the number of fragmentations along the ribs of a solid element. After fragmentation, the initial element is deleted. *Place new objects in* option allows to select layer in which new objects will be placed.

Shortcut: 

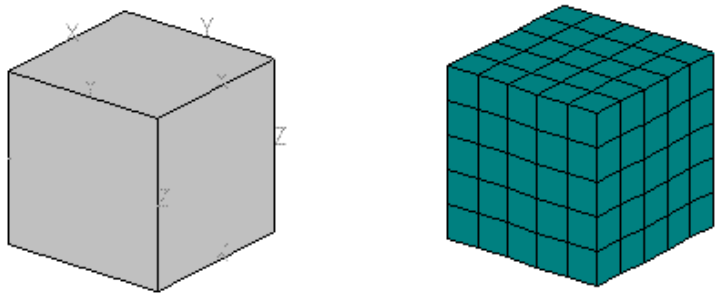


Fig. 2.30 Solid FE meshing

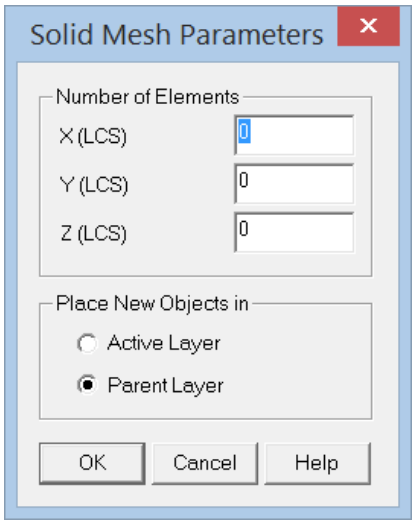


Fig. 2.31 Solid Mesh Parameters dialog box

Solid / 6-noded Solid

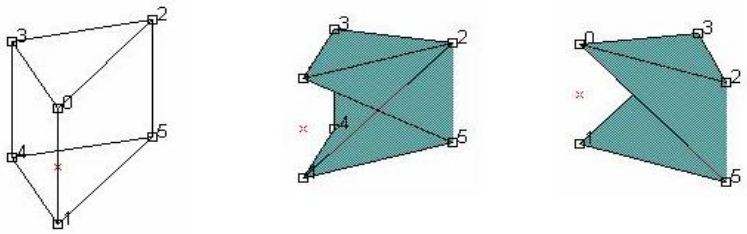
The command enables a mode of drawing 6-node solid elements. An element is defined by six nodes. In this mode, the existing nodes can be unified and new nodes can be created. First mouse click sets the first node; second mouse click sets the second one and so on. If you have already selected or created the first node, right mouse click cancels the operation. The command uses attachment mode while selecting among existing nodes.

While entering FE nodes it is necessary to observe local node numeration.

The required order of node input is the following: 0-2-3-1-5-4, 0-3-2-1-4-5, 1-4-5-0-3-2, 1-5-4-0-2-3, etc.

A degenerate element will be created in case of incorrect numeration order.

Examples of singular elements:



correct incorrect

Fig. 2.32 6-noded solid element creation

Shortcut 

Dimensional elements | 5-noded element

of volumetric elements. The existing nodes can be unified in this mode as well as new ones can be created. The first node is created with the first click, the second node with the second click etc. If you have already selected or created the first node, you can cancel the operation by clicking the right mouse button. The command uses an attachment mode when choosing the previously created nodes.

Shortcut: 

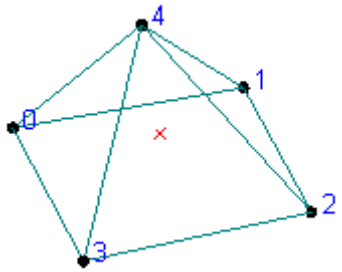


Fig. 2.33 Example of node numbering in 5-noded volumetric element

Dimensional elements | 20-noded element at 8 points

The command sets the mode to create 20-noded dimensional elements. The existing nodes can be unified in this mode and the new ones can be created. The first click creates the first node, the second click creates the second node etc. If you have already selected or created the first node, you can cancel the operation by making a right-click. The command uses an attachment mode when choosing the previously created nodes.

When using this command only the nodes numbered in *Fig.2.49* can be selected, intermediate nodes will be created automatically.

Shortcut: 

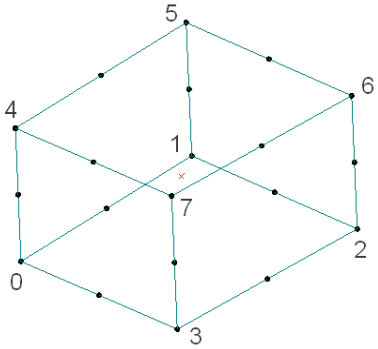


Fig. 2.34 Example of node numbering in 20-noded volumetric element

Dimensional elements | 15-noded element

The command sets the mode to create 15-noded dimensional elements. The existing nodes can be unified in this mode and the new ones can be created. The first click creates the first node, the second click creates the second node etc. If you have already selected or created the first node, you can cancel the operation by making a right-click. The command uses an attachment mode when choosing the previously created nodes.

Shortcut: 

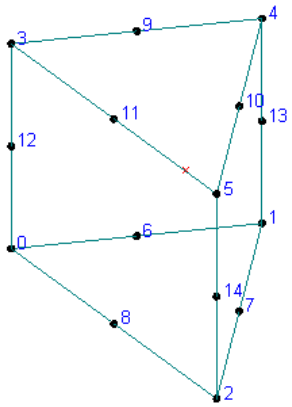


Fig. 2.35 Example of node numbering in 20-noded volumetric element

Dimensional elements | 15-noded element at 6 points

The command sets the mode to create 15-noded dimensional elements. The existing nodes can be unified in this mode and the new ones can be created. The first click creates the first node, the second click creates the second node etc. If you have already selected or created the first node, you can cancel the operation by making a right-click. The command uses an attachment mode when choosing the previously created nodes.

When using this command only the nodes numbered in *Fig.2.51* can be selected, intermediate nodes will be created automatically.

Shortcut: 

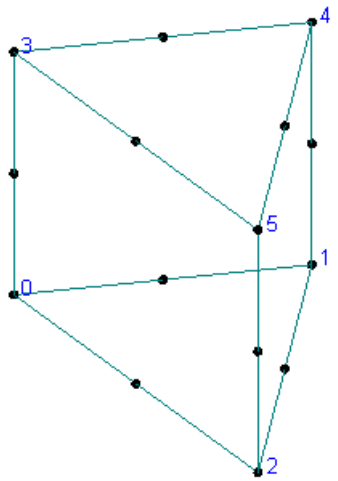


Fig. 2.36 Example of node numbering in 15-noded volumetric element

Dimensional elements | 13-noded element

The command sets the mode to create 13-noded dimensional elements. The existing nodes can be unified in this mode and the new ones can be created. The first click creates the first node, the second click creates the second node etc. If you have already selected or created the first node, you can cancel the operation by making a right-click. The command uses an attachment mode when choosing the previously created nodes.

Shortcut: 

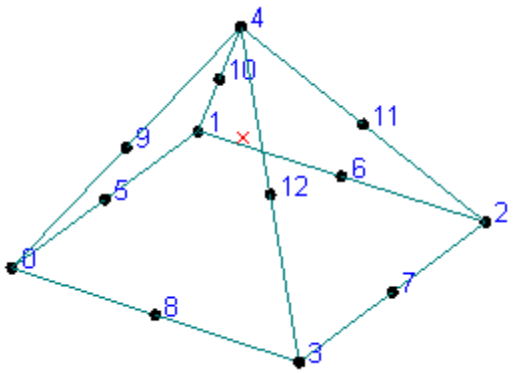


Fig. 2.37 Example of node numbering in 13-noded volumetric element

Dimensional elements | 13-noded element at 5 points

The command sets the mode to create 13-noded dimensional elements. The existing nodes can be unified in this mode and the new ones can be created. The first click creates the first node, the second click creates the second node etc. If you have already selected or created the first node, you

can cancel the operation by making a right-click. The command uses an attachment mode when choosing the previously created nodes.

When using this command only nodes numbered in *Fig.2.53* can be selected, intermediate nodes will be created automatically.

Shortcut: 

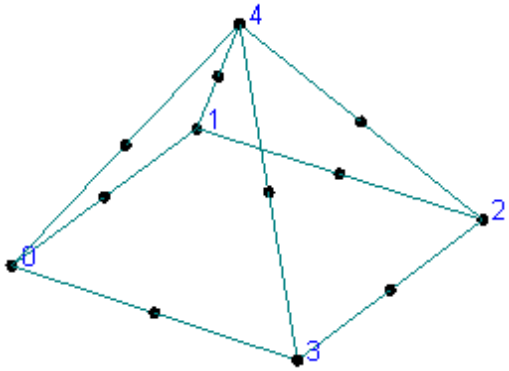


Fig. 2.38 Example of node numbering in 13-noded volumetric element

Dimensional elements | 10-noded tetrahedral element

The command sets the mode to create 10-noded dimensional tetrahedral elements. The existing nodes can be unified in this mode and the new ones can be created. The first click creates the first node, the second click creates the second node etc. If you have already selected or created the first node, you can cancel the operation by making a right-click. The command uses an attachment mode when choosing the previously created nodes.

Fig. below shows the numbers of the nodes that have to be consistently got around (in ascending order) by clicking LMB on each node.

In the process of creating a 10-node dimensional tetrahedron the cursor will move the edge which is to be bound to an intermediate node, in the same way as in the process of creating a 20-node solidus.

Shortcut: 

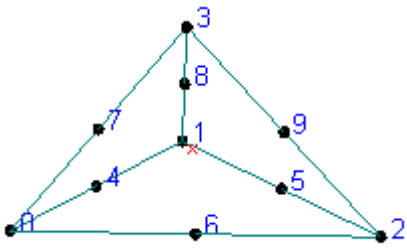


Fig. 2.39 Example of node numbering in 10-noded tetrahedral volumetric element

Dimensional elements | 10-noded tetrahedral element at 4 points

The command sets the mode to create 13-noded dimensional elements. The existing nodes can be unified in this mode and the new ones can be created. The first click creates the first node, the second click creates the second node etc. If you have already selected or created the first node, you can cancel the operation by making a right-click. The command uses an attachment mode when choosing the previously created nodes.

Fig below shows the numbers of the nodes that have to be consistently got around (in ascending order) by clicking LMB on each node. Intermediate nodes will be created automatically.

Shortcut: 

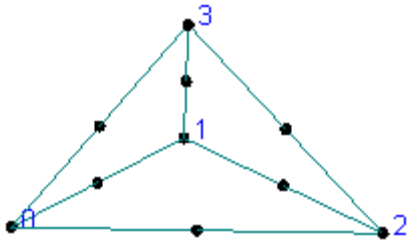


Fig. 2.40 Example of node numbering in 10-noded tetrahedral volumetric element

Solid / 4-noded Solid

The command enables a mode of drawing 4-node solid elements. In this mode, the existing nodes can be unified and new nodes can be created. First mouse click sets the first node; second mouse click sets the second one and so on. If you have already selected or created the first node, right mouse click cancels the operation. The command uses attachment mode while selecting among existing nodes.

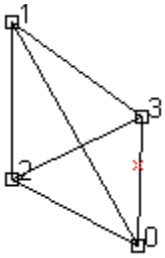


Fig. 2.41 4-noded solid element creation

Shortcut: 

Three-dimensional elements| 20-node element octahedral

The command sets a mode of 20-node three-dimensional octahedral elements creation. In this mode existing nodes can be connected or new ones can be created. The first click of a mouse specifies the first node; the second click specifies the second node, and so on. If you have already chosen or created the first node, pressing of the right mouse button you can cancel the command. The command uses a binding mode when it selects already created nodes.

Fig. 2.46 shows the numbers of nodes that need to be consistently get around (in order of increasing numbers) by clicking LMB on each of these nodes

During creation of a 20-node three-dimensional hexahedron the edge of binding in it in this step to an intermediate node will be created behind the cursor as a prompt (see Fig. 2. 47).

An accelerated choice: 

Three-dimensional elements| 10-node tetrahedral element

The command sets a mode of 10-node three-dimensional tetrahedral elements creation. In this mode existing nodes can be connected or new ones can be created. The first click of a mouse specifies the first node; the second click specifies the second node, and so on. If you have already chosen or created the first node, pressing of the right mouse button you can cancel the command. The command uses a binding mode when it selects already created nodes.

Fig. 2.48 shows the numbers of nodes that need to be consistently get around (in order of increasing numbers) by clicking LMB on each of these nodes

During creation of a 20-node three-dimensional hexahedron, the edge where in this step one will have to tie to an intermediate node also as to like during creation of 20-node co-lead will be created behind the cursor as a prompt.

An accelerated choice: 

Three-dimensional elements| Create a tetrahedron ball

The command creates a ball of tetrahedron three-dimensional elements since simultaneous generation by the FE mesh. The user specifies the center of the ball by a click of the left mouse (LMB) button on a selected point of one of the windows and then specifies by dislocation of a mouse the value of radius of the ball that is created. After the completion of the sphere creation, the dialog of *Creation of a sphere* opens where one can specify parameters of an object that is created and the FE mesh.

The value of the FE maximum size is equal by default one tenth out of radius of a ball that is created, but value of radius can be changed by the user.

An accelerated choice: 

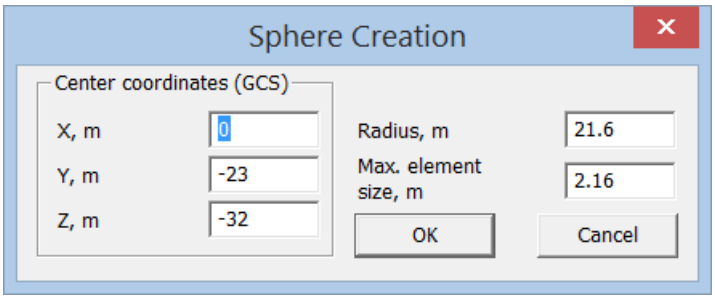


Fig. 2.42 Dialog window of a sphere creation.

Three-dimensional elements| Create a hexahedron ball

The command creates a ball of 8-node solid octahedral elements with simultaneous generation of FE mesh. The user specifies the center of the ball by a click of the left mouse (LMB) button on a selected point of one of the windows and then specifies by dislocation of a mouse the value of radius of the ball that is created. After the completion of the sphere creation the dialog window *Creation of a sphere* opens where one can specify parameters of an object that is created and the FE mesh.

The value of the FE maximum size is equal by default one tenth out of radius of a ball that is created, but value of radius can be changed by the user.

An accelerated choice: 

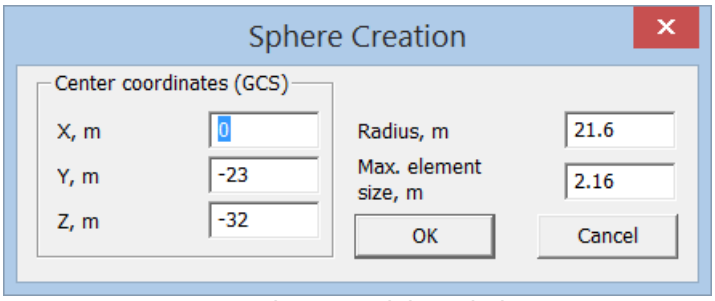


Fig. 2.43 Dialog window of a sphere creation.

Three-dimensional elements| Create a tetrahedron cylinder

The command creates a cylinder of 4 node three-dimensional tetrahedron elements since simultaneous generation by FE mesh. The user sets the center of the cylinder bottom end by a click of the left mouse (LMB) button on the selected point of one of the windows and then specifies by dislocation of a mouse the position of a cylinder axes and the value of a cylinder radius that is created. After the completion of the cylinder creation the dialog window *Creation of a cylinder* (Fig. 2.54) opens where one can specify parameters of an object that is created and the FE mash.

The value of maximum size equal the FE by default is one tenth out of the sphere radius that is created, but the radius value can be changed by the user.

An accelerated choice: 

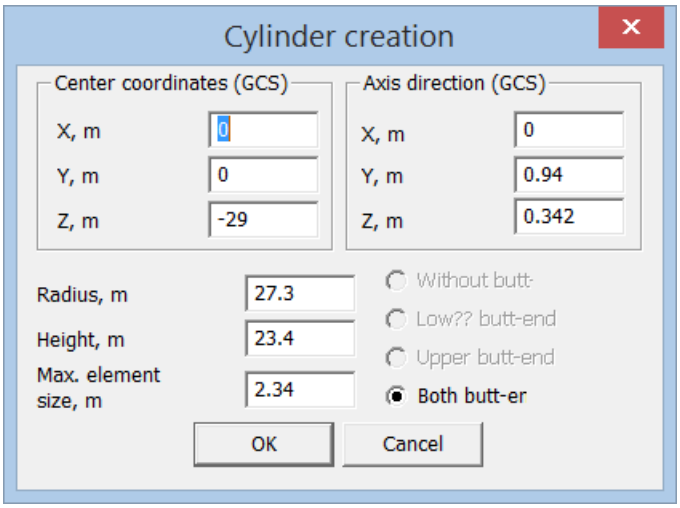


Fig. 2.44 Dialog window of a cylinder creation.

Solid / Rectangular Parallelepiped

The command enables a mode of drawing rectangular parallelepiped that consists of 8-noded finite elements, its sides parallel to global coordinates. *Number of Elements* sets the number of fragmentations by along each rib of parallelepiped. In this mode the existing nodes can be used and new nodes can be created. First mouse click sets insertion point; second mouse click sets point that defines the body diagonal of parallelepiped. If you have already selected or created the first node, right mouse click cancels the operation. The command uses attachment mode while selecting among existing nodes.

Shortcut: 

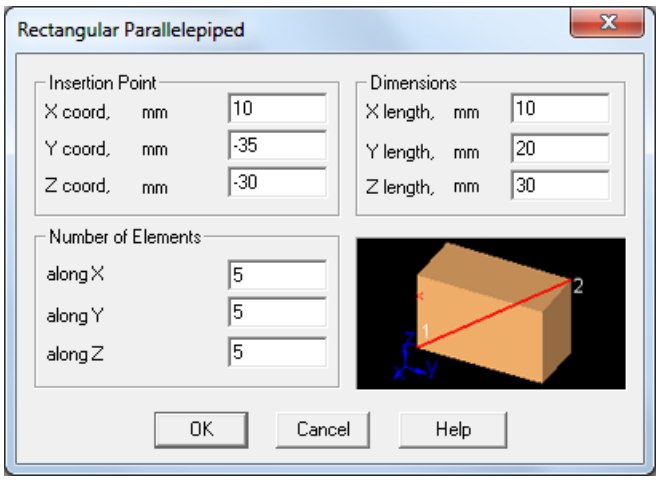


Fig. 2.45 Rectangular Parallelepiped dialog box

Solid / Solid Pipe

The command enables a mode of drawing a heavy-gauge pipe that consists of 8-noded finite elements and is arbitrarily oriented in space. A pipe can be closed or open in circular dimension. In this mode existing nodes can be used and new nodes can be created. If you have already selected or created the first node right mouse click cancels the operation. The command uses attachment mode while selecting among existing nodes. First mouse click sets insertion point, second mouse click sets point that defines length and direction of the pipe, third mouse click sets the point that defines starting angle and inner radius of a pipe (radius is calculated as point-to-axis of a pipe distance), fourth mouse click sets the point that defines end angle (this angle is measured according to right-hand screw rule, so that if you look in the direction of pipe axis vector, the angle is measured clockwise) and outer radius of the pipe.

Shortcut: 

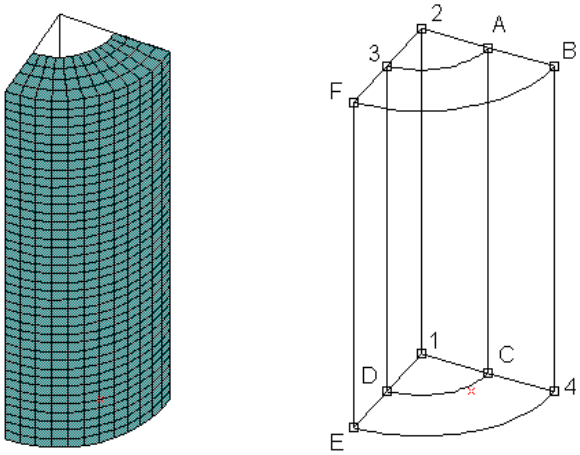


Fig. 2.46 Solid pipe creation

To draw a pipe section 3ABFDC4E that is open in circular dimension it is necessary to set 4 points: point 1 and 2 set the pipe axis, point 3 that defines starting angle and inner diameter of the pipe can be situated anywhere D3 straight line, point 4 that defines end angle and outer diameter can be situated anywhere along B4 straight line.

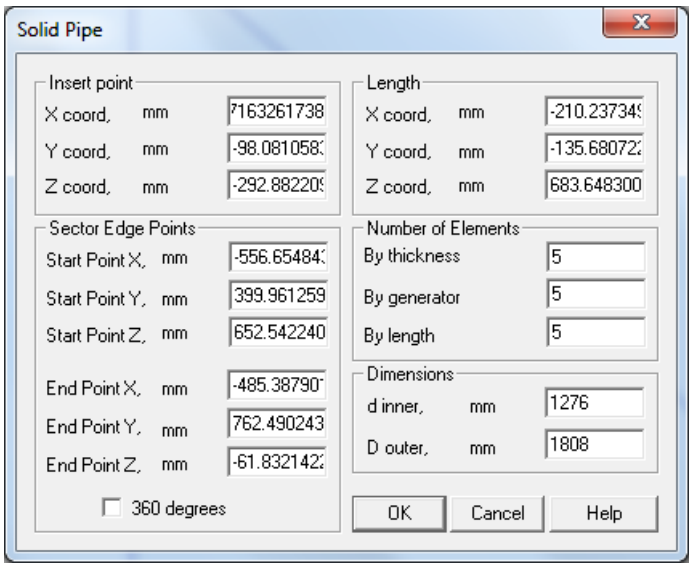


Fig. 2.47 Solid Pipe dialog box

In this dialog window, it is possible to edit any previously entered parameters. *Number of Elements* sets the number of radial, circular and axial fragmentations of the pipe. To draw a pipe that is closed in circular direction place the tick-mark at *360 degree* check box. In this case the coordinates of end angle point are ignored.

Shortcut: 

Solid | Ball of Tetrahedrons

The command creates a ball of tetrahedron three-dimensional elements since simultaneous generation by the FE mesh. The user specifies the center of the ball by a click of the left mouse (LMB) button on a selected point of one of the windows and then specifies by dislocation of a mouse the value of radius of the ball that is created. After the completion of the sphere creation, the dialog of *Creation of a sphere* opens where one can specify parameters of an object that is created and the FE mesh.

The value of the FE maximum size is equal by default one tenth out of radius of a ball that is created, but value of radius can be changed by the user.

Shortcut: 

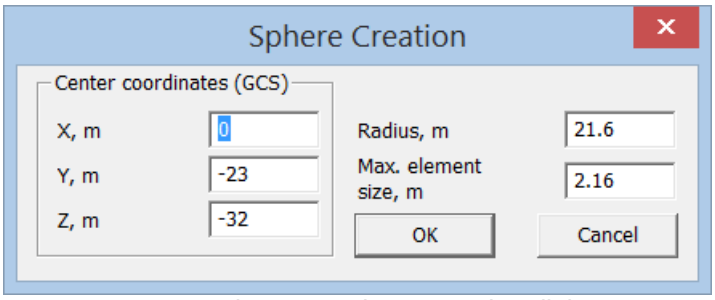


Fig. 2.48 Sphere Creation dialog.

Solid | Ball of Hexahedrons

The command creates a ball of 8-node solid octahedral elements with simultaneous generation of FE mesh. The user specifies the center of the ball by a click of the left mouse (LMB) button on a selected point of one of the windows and then specifies by dislocation of a mouse the value of radius of the ball that is created. After the completion of the sphere creation the dialog window *Creation of a sphere* opens where one can specify parameters of an object that is created and the FE mesh.

The value of the FE maximum size is equal by default one tenth out of radius of a ball that is created, but value of radius can be changed by the user.

Shortcut: 

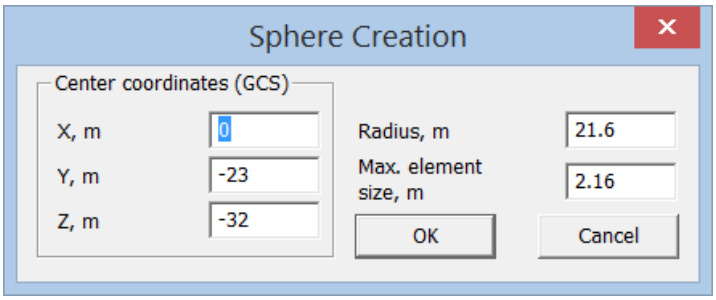


Fig. 2.49 Sphere Creation dialog.

Solid | Cylinder of Tetrahedrons

The command creates a cylinder of 4 node three-dimensional tetrahedron elements since simultaneous generation by FE mesh. The user sets the center of the cylinder bottom end by a click of the left mouse (LMB) button on the selected point of one of the windows and then specifies by dislocation of a mouse the position of a cylinder axes and the value of a cylinder radius that is created. After the completion of the cylinder creation the dialog window *Creation of a cylinder* opens where one can specify parameters of an object that is created and the FE mash.

The value of maximum size equal the FE by default is one tenth out of the sphere radius that is created, but the radius value can be changed by the user.

Shortcut: 

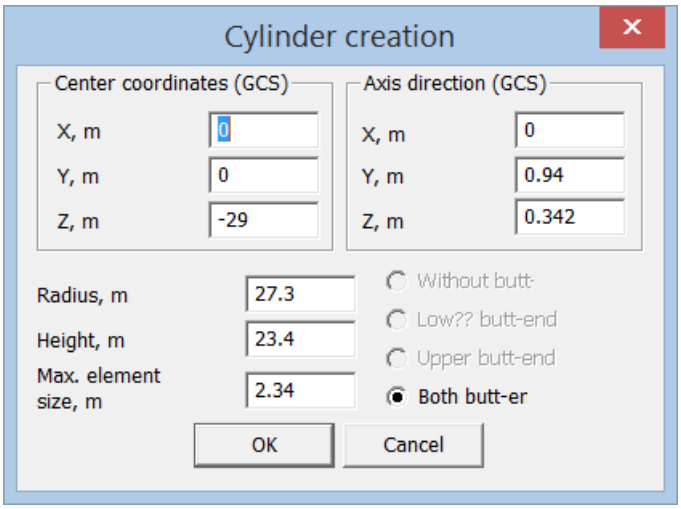


Fig. 2.50 Cylinder creation dialog.

Arc

The command enables a mode for drawing arc elements. An arc element is created in the viewplane and is approximated with straight-line rods with user defined accuracy. First mouse click defines arc center. Second mouse click sets arc starting angle. Third mouse click sets arc radius. Fourth mouse click sets arc ending angle and invokes a dialog box which allows you to enter the number of straight-line rods of which arc will consist, or cancel the operation. You can use attachment at any stage of this operation. Right mouse click cancels the operation. *Place new objects in* option allows to select layer in which new objects will be placed.

Shortcut: 

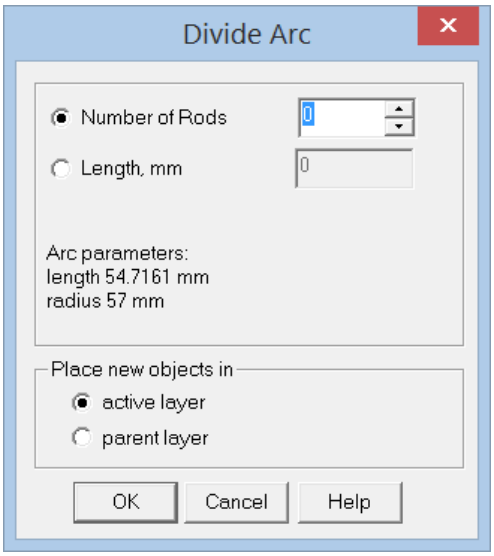


Fig. 2.51 Divide Arc dialog box

Circle

The command enables a circle drawing mode. Just like the arc, a circle is created in the viewplane and is approximated with straight-line rods. First mouse click sets the circle center. Second mouse click sets the circle radius and calls a dialog box which allows you to enter the number of straight-line rods which constitute the circle. The first rod begins in zero grad point. Using attachment at second mouse click (to define radius) will create the first node at the point lying in the cut formed by circle center and the attached node. Right mouse click cancels the operation. *Place new objects in* option allows to select layer in which new objects will be placed.

Shortcut: 

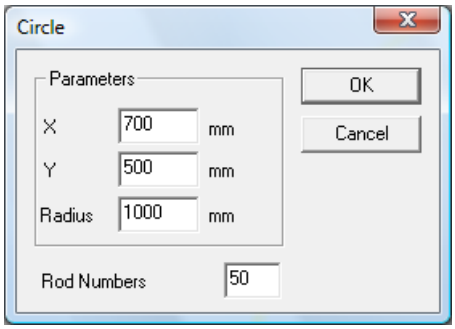


Fig. 2.52 Circle dialog box

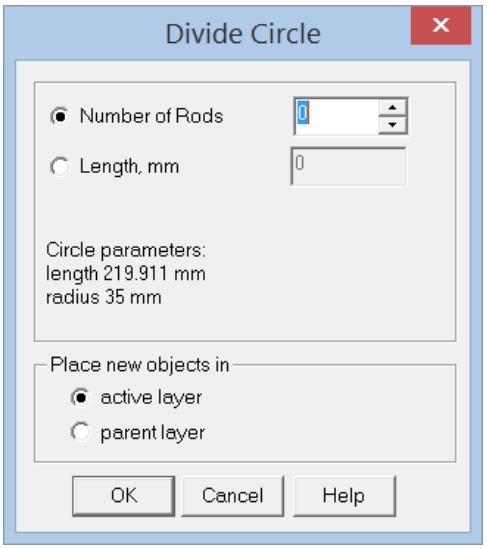


Fig. 2.53 Divide Circle dialog box

Support / Rigid Support

The command switches editor into a mode of setting bases (supports) for existing nodes. Rigid supports are set in the node coordinate system. To set or delete a support you are to select a node. Then a dialog box appears on the screen, as shown below. Buttons of *Support Type* group are used to define support as restraint or hinge. Check boxes *Lock Translation* can restrict node motion along X, Y, Z direction. Check boxes *Lock Rotation* can restrict node rotation around X, Y, Z axes. It is possible to set rigid support also on face of plate object that can be used in particular for modeling of the foundation mat on the elastic soil base.

Shortcut: 

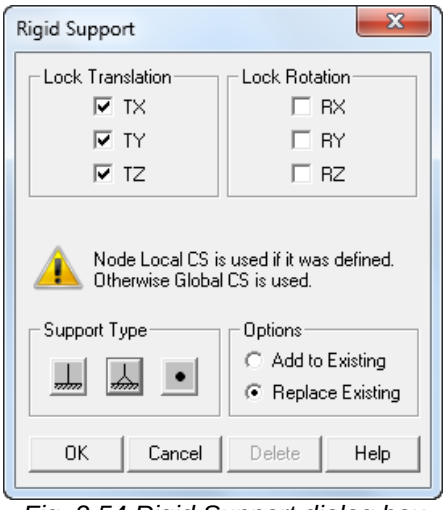


Fig. 2.54 Rigid Support dialog box

Support / One-Dir Rigid Support

The command switches editor into a mode of one-dir. rigid supports for existing nodes. Rigid supports are set in the node coordinate system. If not specified the node coordinate system coincides with the Global Coordinate System.

Unlike the command Support/ rigid Support the command allows to lock translation and/or rotations only in one direction either positive or negative one.

The command enables a mode to set one-sided elastic supports with displacement. A foundation that can be lifted off the ground is an example of such a support.

Having chosen the command it is necessary to ensure the construction to be immovable, otherwise the calculation can be incorrect.

Shortcut: 

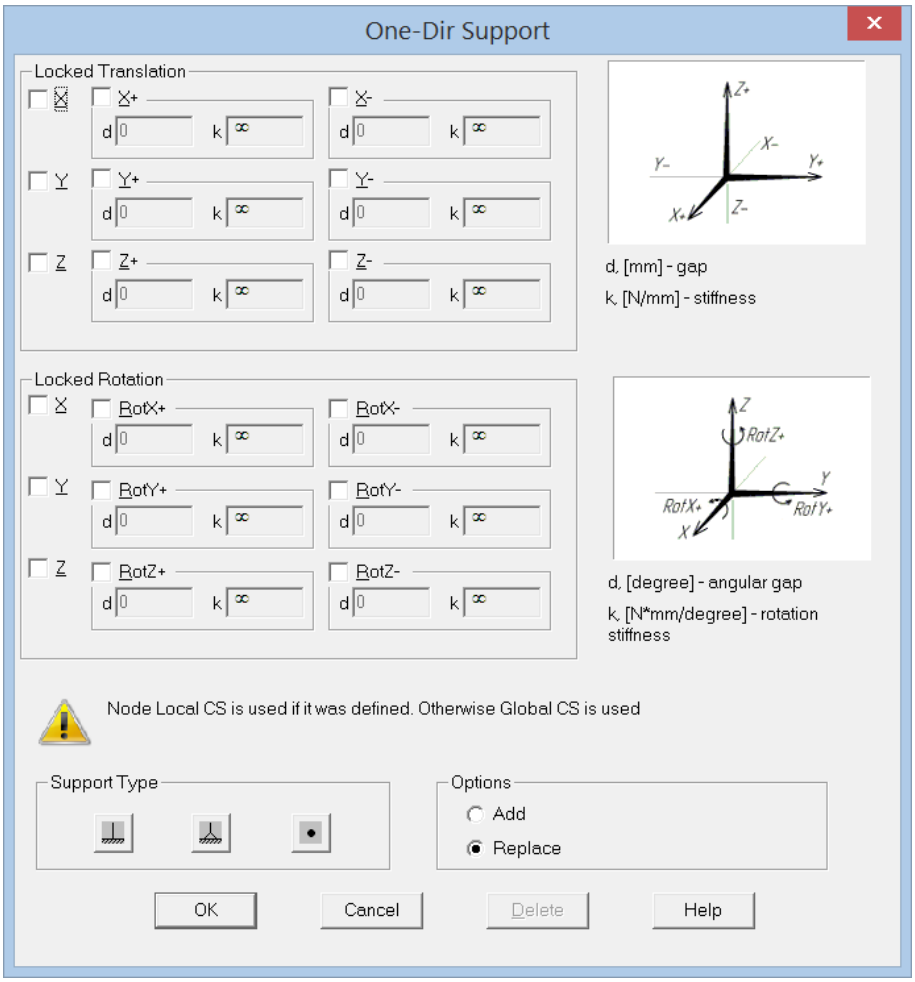


Fig. 2.55 One-Dir Rigid Support dialog box

Support / Elastic Support

The command enables a mode for entering elastic supports in existing nodes. Elastic supports are set in the node coordinate system. In order to define elastic supports in existing nodes, it is necessary to select desired nodes using **Edit / Complex Selection** command.

To define spring support in a node or a group of nodes, select the desired node or one of the selected nodes by clicking on, which will call *Elastic Support* dialog box.

In *Support stiffness* entry boxes, stiffness components along coordinate axes are set. Radio button *Add to existing* is used for entering additional values to the existing ones for the node. Otherwise, *Replace existing* radio button is used to replace existing supports with new values. To delete support in selected nodes press **Delete** button.

Shortcut:

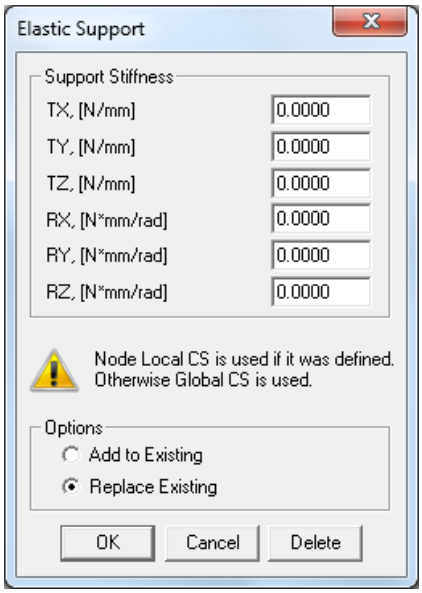


Fig. 2.56 Elastic Support dialog box

Support / Elastic Supports for Foundations

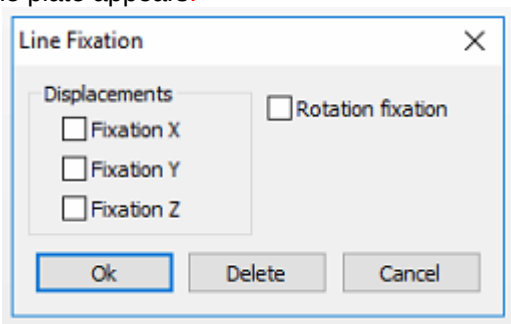
Commands allow to set and calculate foundation parameters in design elements mode.

	Elastic soil base for post foundation (activates after selection of rod-column and its base node)
	Elastic soil base for strip foundation (activates after selection of rod-girder);
	Elastic soil base for mat foundation (activates after selection of plate object);
	Elastic soil base for pile foundation (activates after selection of rod-column and its base node);

The detailed description of the commands is stated in section «Soil bases calculation».

Support / Line Fixation

This command is designed to fix the sides of the plates. After selecting this command and clicking on any edge of the plate a dialog box in which you can specify the types of supports that will be installed along this edge of the plate appears.



Line Fixation dialog box

Rigid Link

The command allows user to set rigid connection of nodes of different elements. This operation allows modeling of unilateral couplings, sliders and special type connections. To define rigid link, it is necessary to specify minimum two nodes and define required degrees of freedom for this link.

Shortcut:

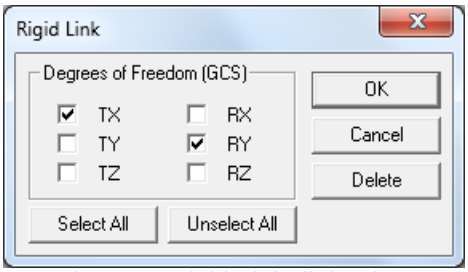


Fig. 2.57 Rigid Link dialog box

Elastic Link

The command enables a mode for introducing special element, called elastic link between two existing nodes.

In this mode, user can connect already existing nodes or create new ones. First mouse left click sets the first node, second click - second. By pressing the right mouse button you can cancel the command. The command uses attachment mode when connecting already existing nodes. If you wish to create a new node, rather than using one of the existing ones, for example, when nodes lie in different planes, but coincide in the viewplane, attachment mode should be disabled.

It is possible to set an elastic link damper mode by clicking *Damping* button. A damper element is to be considered when solving forced oscillations problem.

Shortcut: 

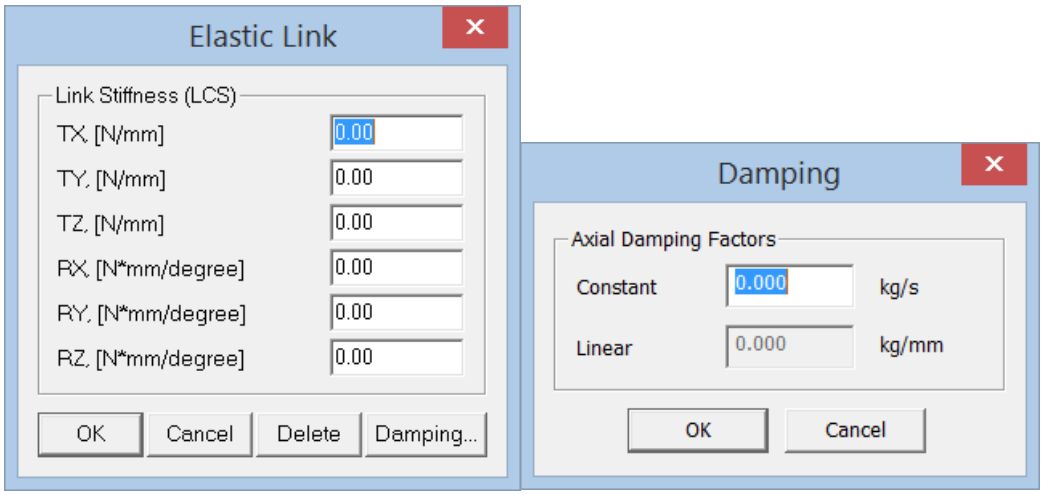


Fig. 2.58 Elastic Link dialog box

Hinge / At Node / Create for All

The command places hinges at all nodes, thus converting frame constructions to truss construction. This command calls dialog box shown below. Checkboxes allow you to permit rotation around X, Y, Z axis.

Shortcut: 

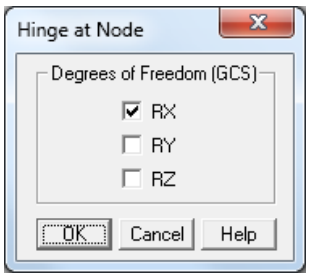


Fig. 2.59 Hinge at Node dialog box

Hinge / At Node / Create for Selected

The command creates hinges at selected nodes in global coordinate system. The command calls a dialog box shown above.

Hinge / At Rod End

The command switches editor into a mode which allows to place a hinge at rod end. Click near end of the rod to place hinge. That will invoke dialog box shown below. Use checkboxes to permit rotation around X,Y,Z axis.

Shortcut: 

Fig. 2.60 Hinge at Rod End dialog box

Rod Release

The command creates degree of freedom release for rod elements. It can be helpful when modeling one direction connections, sliders and special-type connections. To define release select one or more rod elements and in dialog window shown below select *degrees of freedom* for both ends of each rod.

Shortcut: 

Fig. 2.61 Rod Releases dialog box

Plate Release

The command creates degree of freedom release for plate elements. It can be helpful when modeling special-type connections. To define release select one or more plate elements and in dialog window shown below select *degrees of freedom* for required nodes of plates.

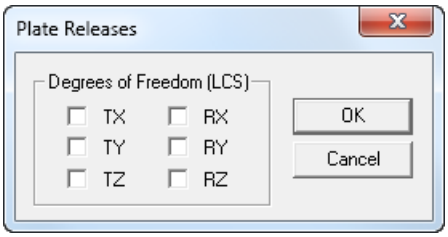


Fig. 2.62 Plate Releases dialog box

Delete Selected

The command deletes all selected elements.
Shortcut:

Delete All

The command deletes the entire construction. This command calls a message box.

Multiple Nodes

The command enables/disables a mode that allows user to create several nodes in one point of space. This mode is meant to solve problems on contact interaction.

Shortcut:

Loads menu

This menu commands allow you to apply loads to nodes and elements.

Force on Node

The command enables a mode for applying forces to existing nodes. After selecting a node, a corresponding dialog box appears on the screen. All operations with load (setting, change, removal) are fulfilled in a load case selected from the list of load cases. **Nodal load values** command on *Extra view filters* toolbar enables / disables mode for viewing load values of current load case.

Shortcut:

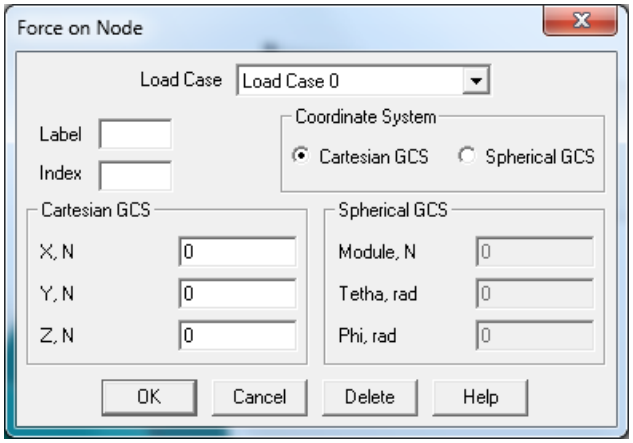


Fig. 2.63 Force no Node dialog box

Moment on Node

The command enables a mode for applying moments to existing nodes. After selecting a node, a corresponding dialog box appears on the screen. All operations with load (setting, change, removal) are performed for a load case selected from the list of load cases.

Shortcut:

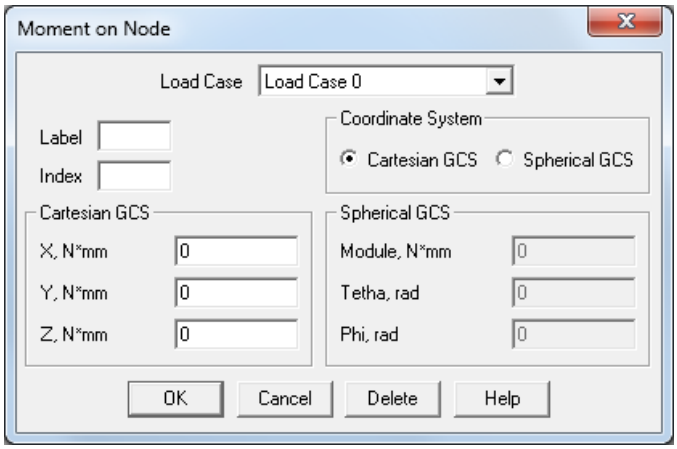


Fig. 2.64 Moment on Node dialog box

Support Displacements

The command enables a mode for assigning supports' displacement along fixed degrees of freedom (settlement of support). Displacements are set in the node coordinate system. Since nodes are to be selected for this operation, a dialog box shown below appears. If you want to enter additional values to existing displacements in a node, select radio button *Add to existing*. Otherwise, if you want to replace existing displacements with new values, select *Replace existing* radio button.

To delete displacements in selected nodes, click **Delete** button.

All operations with load (setting, change, removal) are performed for load case selected from the list of load cases.

Shortcut: 

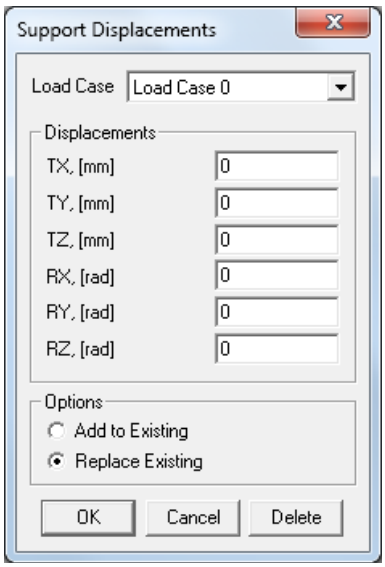


Fig. 2.65 Support Displacements dialog box

Temperature

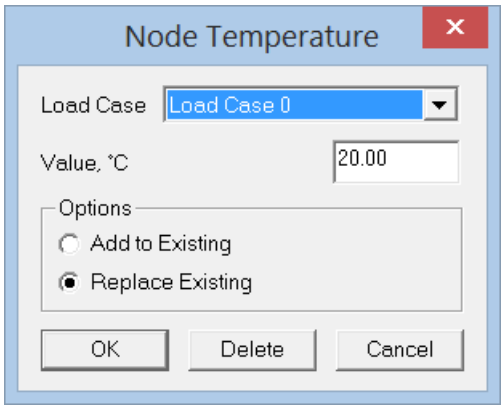


Fig. 2.66 Node Temperature dialog

The command switches editor into a mode specifying temperature at nodes which will be used at thermal calculation.

In this mode left-click on node or one of the selected nodes. As a result there will be a dialog box for temperature specifying in these nodes. By default temperature 20 C is accepted to have no thermal stresses. You can change this value by **Calculation | Calculation Options…** menu.

To take into account temperature field for stresses analysis it is necessary to carry out Steady-state heat transfer analysis with switched on *Temperature account* (Steady-state heat transfer*) option* for static calculation. Calculation of stationary heat conductivity is carried out for a single load case or load combination only.

Shortcut: 

Rod Prestrain

This command allows you to define various types of preload on rod elements using the dialog box shown below.

Shortcut:

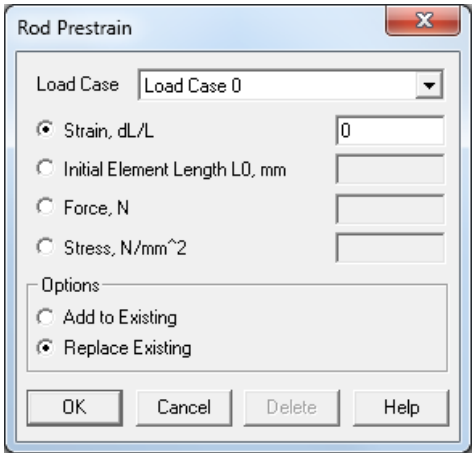


Fig. 2.67 Rod Prestrain dialog box

There are 4 different types of preload available for rod/beam or cable elements:

Strain, dL/L – ratio of difference between distance between nodes connected by this element and initial length of the element to the distance between the nodes;

Initial element length – initial length of element before deformation.

✍ *Note: valid only for cable elements.*

Force – internal axial force acting in element;

Stress – normal stress in element (axial direction).

Local Load On Rod

The command enables a mode for load application to a rod or a group of rods.

To apply load to a single rod, you are to select the rod in this mode by clicking on it. Then, a window of rod loads editor appears on the screen. Commands for switching into mode of setting a certain type of load from *Load Type* menu are available in this editor. Previously entered loads are

edited by right mouse click (in XY plane or axial direction) or by right mouse click together with SHIFT button (in XZ plane).

To set load for a group of rods, first select the necessary rods with the help of *Select* command and click on any rod in this mode. After that, the window of the rod loads editor appears on the screen. Then select one of the commands from *Load Type* menu for setting a corresponding load.

Shortcut: 

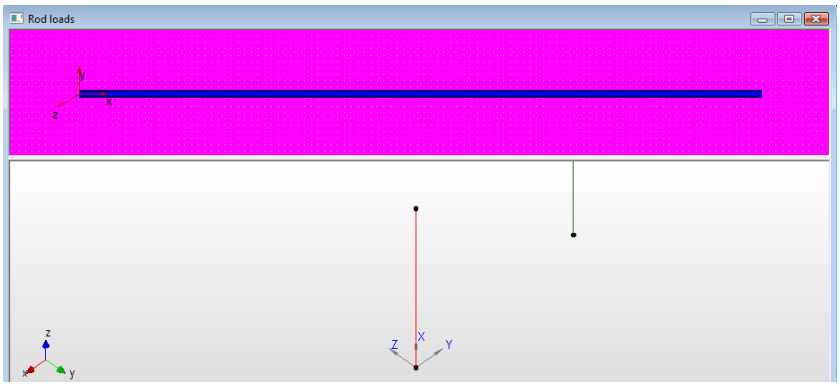


Fig. 2.68 Rod Loads editor window

Global Load on Rod

The command enables a mode of applying load to a rod or a group of rods.

To apply load to a single rod, you are to select rod in this mode by clicking on it. After that, the window of the rod loads editor appears on the screen.

Shortcut: 

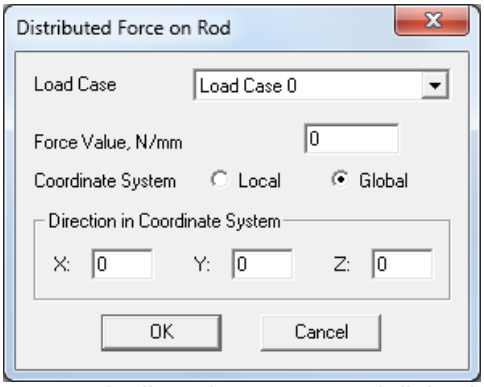


Fig. 2.69 Distributed Force on Rod dialog box

To set load for a group of rods, first select the necessary rods with the help of *Select* command and click on any rod in this mode. After that on the screen appears window of the rod loads editor. Forces direction is set by a vector in three-dimensional space. The coordinates of this vector are set in *Direction In Local or Global Coordinate System* entry field. For example if it is necessary to set load of 2N/mm in the direction reverse to Z axis, you can enter 2 in *Force Value* entry box and 0, 0, -1 in *Direction* boxes or enter -2 in *Force Value* entry box and 0, 0, -1 in *Direction* box. The load is added to a load case from the load case list.

Rod Temperature

This command applies temperature to a single rod or a group of selected rod elements. Click on one or group of selected elements with left mouse button, and *Temperature Load on Rod* dialog box will appear. Various types of temperature distribution can be selected using *Load type* radio buttons.

Shortcut: 

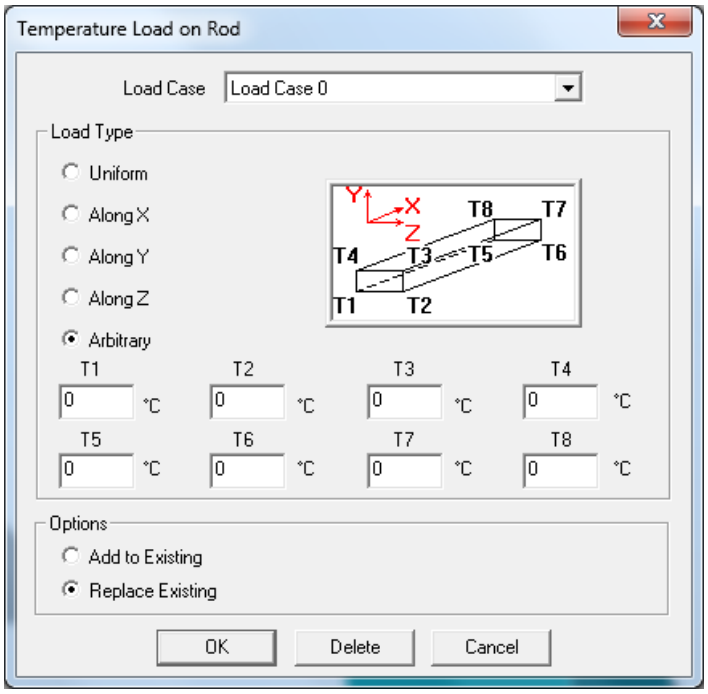


Fig. 2.70 Temperature Load on Rod dialog box

Delete Rod Loads

The command deletes all loads applied to selected rods.
The commands that are available only in rod loads editor are reviewed below.
Shortcut: 

Rod Load Type / Axial Force

The command allows you to apply axial concentrated axial force to a rod. Selecting this command switches editor into the mode of setting an axial load or editing an axial load for a single rod. To enter a new load click left mouse button in the force allocation point. Enter the values in the dialog that appears and push **OK**. To edit or delete a load, click right mouse button. All operations with load (setting, change, removal) are performed for load cases selected from the list of load cases.
Shortcut: 

Rod Load Type / Lateral Force

The command allows you to apply concentrated radial force to a rod. Selecting this command switches editor into the mode for setting or editing a radial force to rod. To enter a new load, click left mouse button in the force application point. Enter the values in the dialog box that appears, and click **OK**. To edit or delete a load in XY plane, click right mouse button, or right mouse button together with SHIFT button for XZ plane. All operations with load (setting, change, removal) are performed for load cases selected from the list of load cases.
Shortcut: 

Rod Load Type / Torsional Moment

The command allows you to apply concentrated moments of torsion to a rod or a group of rods. Command operation is completely similar to that of **Rod Load Type / Axial Force** command. All operations with load (setting, change, removal) are performed for load cases selected from the list of load cases.
Shortcut: 

Rod Load Type / Bending Moment

The command allows you to apply concentrated bending moments to a rod or a group of rods. Command operation is completely similar to that of **Rod Load Type / Radial Force** command. All operations with load (setting, change, removal) are performed for load cases selected from the list of load cases.

Shortcut: 

Rod Load Type / Distributed Axial Force

The command allows you to apply axial distributed forces to a rod or a group of rods. Selecting this command switches editor into the mode for setting an axial load or editing the axial load for single rod or right away calls force setting dialog for a group of rods if they were selected by *Select* command at first. To enter a new load, click in the starting and ending points of force application section in setting mode for a single rod. Enter the values in appeared dialog and then click **OK**. To edit or delete a load, click right mouse button. All operations with load (setting, change, removal) are performed for load cases chosen from the list of load cases.

Shortcut: 

Rod Load Type / Distributed Lateral Force

The command enables a mode which allows you to apply radial distributed forces to a rod or a group of rods. Selecting this command switches editor into mode of setting or editing an axial load for a single rod or immediately calls distributed the force setting dialog box for a group of rods if they were selected by *Select* command at first. To enter a new load, click to specify the starting and ending points of the zone of force application in the load setting mode for a single rod. Enter your values in the dialog box that appears and click **OK**. To edit or delete a load, click right mouse button in XY plane or right mouse button together with SHIFT button for XZ plane. All operations with load (setting, change, removal) are performed for load cases chosen from the list of load cases.

Shortcut: 

Rod Load Type / Distributed Torsional Moment

The command allows you to apply distributed torsion moments to a rod or a group of rods. Command operation is completely similar to that of **Rod Load Type / Axial Distributed Force** command. All operations with load (setting, change, removal) are performed for a load case chosen from the list of load cases.

Shortcut: 

Rod Load Type / Distributed Bending Moment

The command allows you to apply distributed bending moments to a rod or a group of rods. Command operation is completely similar to that of **Rod Load Type / Radial Distributed Force** command. All operations with load (setting, change, removal) are performed for load cases selected from the list of load cases.

Shortcut: 

Wind load on rod

the command sets the mode of input of wind load on the rods. The load is applied to the selected rods. In this mode you click one of the required rods with the left mouse button and after that the load dialog box appears.

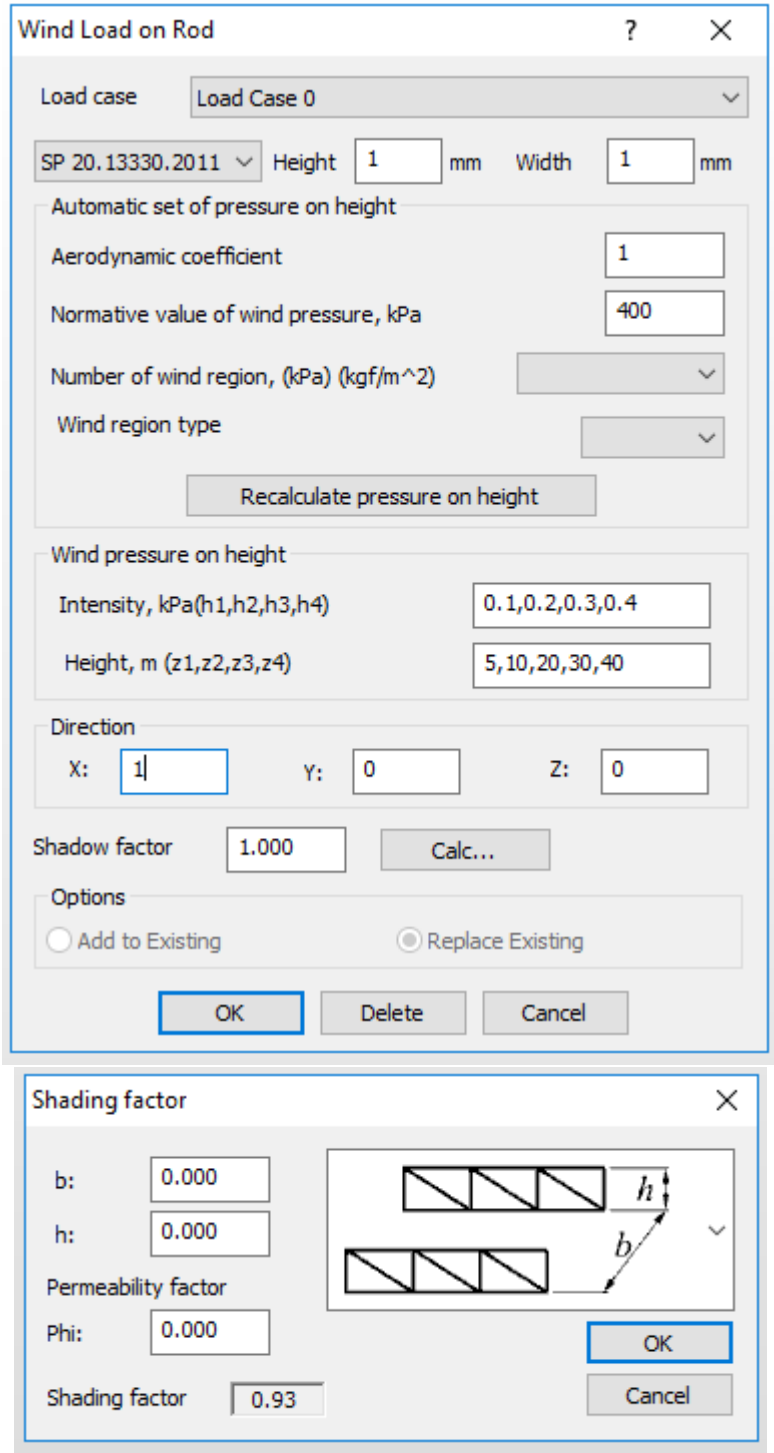
The load is set by the dependence of the wind pressure on the height. The pressure values are entered in a comma-separated field in the **Intensity** field, relevant heights are entered being separated by commas in the **Height** field. The direction of the wind load is given by the direction cosines to the global coordinate system in the group of input fields **Direction.**

The average component of the wind load in the form of this dependence can be formed in an automatic mode. This opportunity is done for SNiP 2.01.07-85 * (SP 20.13330.2011) for the "Load and impact") in accordance with clause 6.3. To do this, the group Automatic pressure setting from height is used. In the dialog you need to specify the height and width of the building, the aerodynamic coefficient, the standard value of the wind pressure (manually or by selecting the wind zone) and the

type of terrain. The load reliability factor is taken into account by multiplying the standard value of the wind pressure by it or by multiplying directly the wind pressure value by setting the dependence of the wind pressure on the height. After setting the above parameters, click the button Recalculate pressure from height. As a result, the fields Intensity and Height are filled automatically.

If there are several rods and they "shade" each other in the direction of the wind, then it can be taken into account by specifying the shading factor. To specify the parameters for its calculation, in the Wind load on the rod window, click the Calculate ... button and the Shadow factor dialog box opens.

To calculate it, it is necessary to select the scheme of co-operating rods, set the required dimensions, the permeability factor Phi, and the shading factor will be immediately calculated and its value will be shown in the corresponding field. Pressing the button Ok transfers the calculated value of the Shadow factor to the corresponding field of the Wind load on the rod.



Wind Load on Rod and Shading Factor dialog boxes

Plate Distributed Load

The command enables a mode which allows you to apply normal load to plates. Load is applied to plates marked with the help of *Select* command. Click left mouse button on one of desired plates in this mode. After that, a load setting dialog box appears on the screen.

Shortcut:

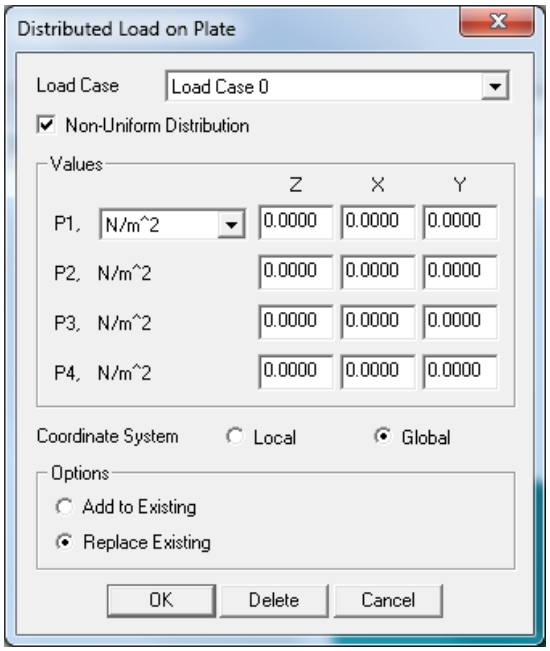


Fig. 2.71 Distributed Load on Plate dialog box

To delete a normal load from selected plates push **Delete** button in this dialog box. All operations with load (setting, change, removal) are performed for load cases selected from the list of load cases.

Plate Load command on *Extra view filters* toolbar enables / disables mode for viewing load map of current load case.

Plate Load Values command on *Extra view filters* toolbar enables / disables mode for viewing load values of current load case.

Plate Linear Distributed Load

This command enables a mode which allows you to apply linear distributed load to plates.

After command activation select plates using Complex Selection mode and press Enter/Space key. Specify 3 nodes in order as in Fig. 2.60 a. After selection of 3-d node the dialog box appears on the screen (Fig. 2.60 c). In the bottom part of dialog box there are node coordinates and in the upper part - corresponding values of pressure.

Plate Load command on *Extra view filters* toolbar enables / disables mode for viewing load map of current load case.

Plate Load Values command on *Extra view filters* toolbar enables / disables mode for viewing load values of current load case.

To delete load on selected plates from current load case press **Delete** button of dialog box.

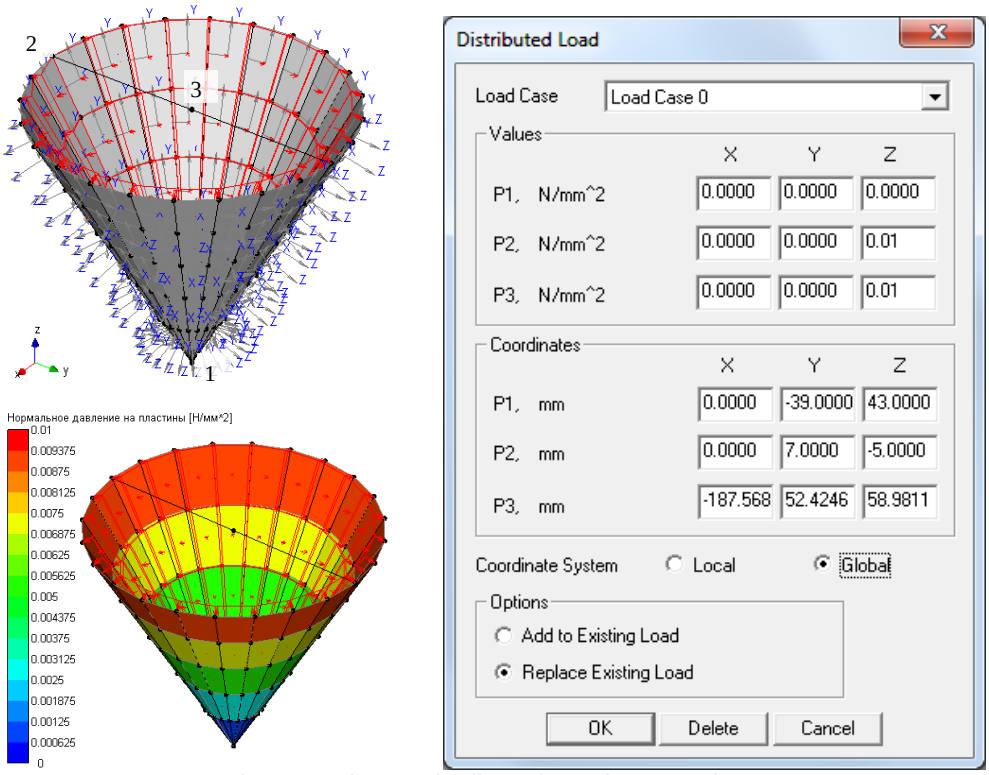


Fig. 2.72 Linear Distributed Load set on plates

Plate Snow Load

The command enables a mode, which allows you to apply snow load to a single or a selected plate. Load is applied to plates marked with the help of *Select* command. In this mode, click left mouse button on one of the desired plates. After that, a load setting dialog box appears on the screen. All operations with load (setting, change, removal) are performed for load cases chosen from the list of load cases.

Shortcut: 

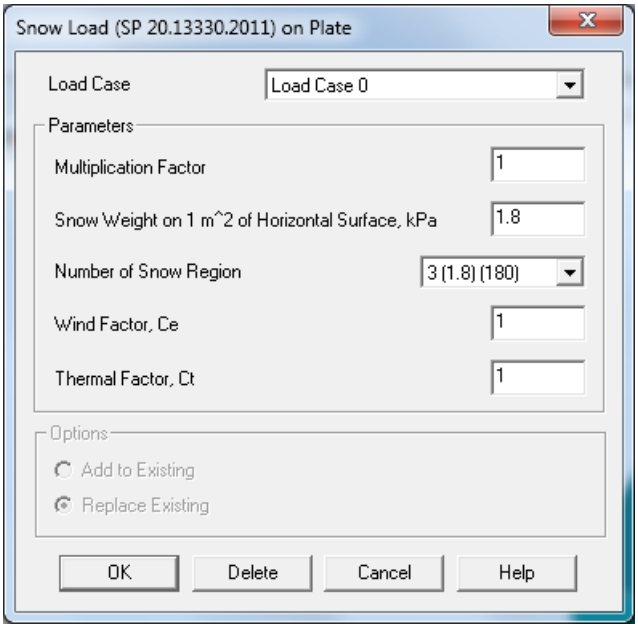


Fig. 2.73 Snow Load on Plate dialog box

Snow Load command on *Extra view filters* toolbar enables / disables mode for viewing snow load map of current load case.

Plate Wind Load

The command enables a mode which allows you to apply wind load to plates. A mouse click on selected plates calls the dialog box shown below. In this mode, click left mouse button on one of necessary plates. After that, a load setting dialog box appears on the screen.

Shortcut:

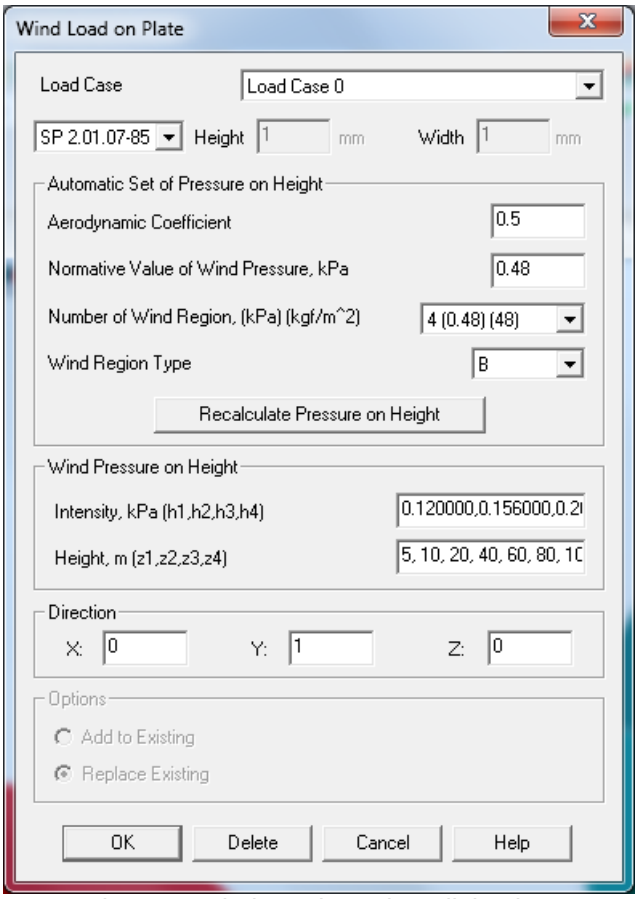


Fig. 2.74 Wind Load on Plate dialog box

Wind Load command on *Extra view filters* toolbar enables / disables mode for viewing wind load map of current load case.

Plate Temperature

The command enables a plate temperature definition mode. You can apply temperature load to one or a group of plates selected with *Select* command. In this mode, click left mouse button on the desired plate to call the *Temperature Load* dialog box shown below. All operations with this type of load are performed for selected load case.

Shortcut:

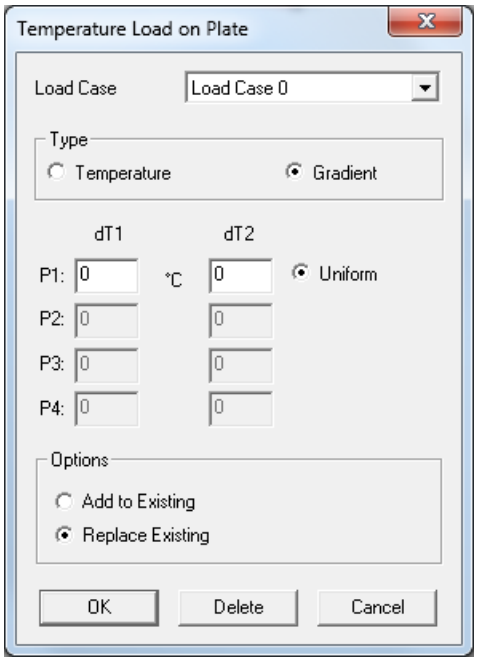


Fig. 2.75 Temperature Load on Plate dialog box

There are two types of temperature distribution available: *Temperature* – applies uniform temperature to a whole plate and *Gradient* – uniform temperature distribution through plate thickness. *dT1* – temperature change for surface with positive z coordinate (local plate coordinate system) and *dT2* - temperature change for reverse surface.

Plate Linear Temperature

The command enables a mode, which allows you to apply temperature load that changes linearly along a certain direction. This operation can be performed on one or a group of plates selected with *Select* command. In this mode, you define the beginning by clicking left mouse button and with next mouse click you select the end of direction vector.

Shortcut: 

✍ *Note: in this mode you can select existing nodes as well. Numerical values are entered in the dialog box shown below. All operations with this type of load are performed for selected load cases.*

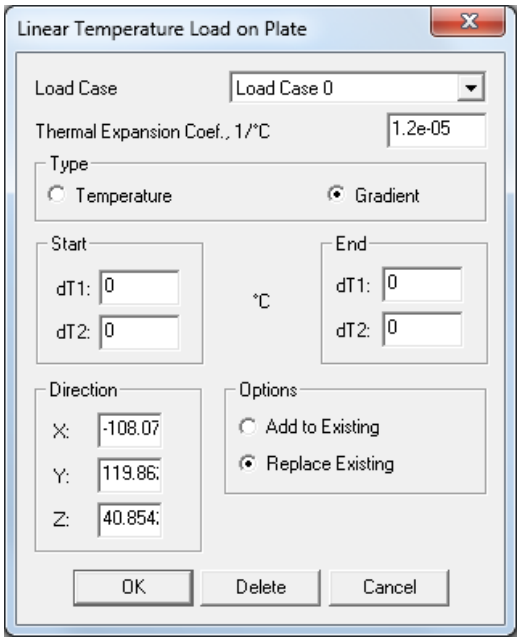


Fig. 2.76 Linear Temperature Load on Plate dialog box

There are two types of load distribution available: *Temperature* – defines temperature change along the surface, *Gradient* – defines temperature distribution through thickness. Groups *Begin* and *End* designate values for direction starting and ending points of the vector respectively. *dT1* – temperature change for surface with positive z coordinate (local plate coordinate system) and *dT2* - temperature change for opposite surface. Group *Direction* allows you to edit the direction vector.

Pressure on Solid

The command allows you to apply pressure to solid element faces. It calls a dialog box where numerical values and creation options are entered.

Shortcut: 

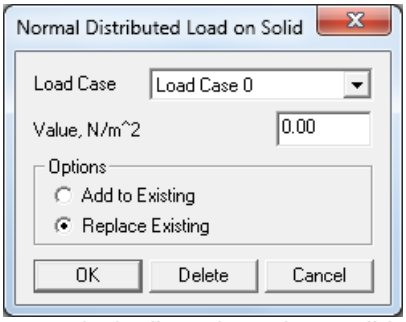


Fig. 2.77 Normal Distributed Load on Solid dialog box

Acceleration / Linear Acceleration

The command calls a dialog box in which you can define linear acceleration acting upon the whole model by entering a direction vector and its numerical value in [mm/s^2].

Shortcut: 

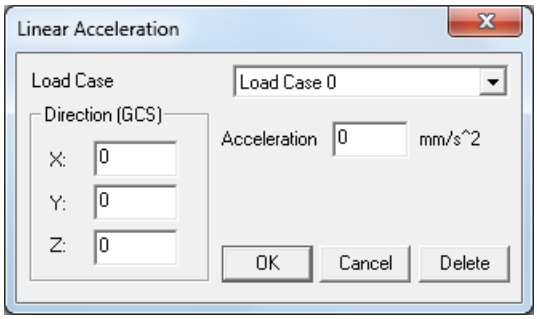


Fig. 2.78 Linear Acceleration dialog box

Acceleration / Angular Acceleration

The command calls a dialog box in which you can apply angular velocity and acceleration by defining their numerical values, center of rotation and rotation vector (acceleration and velocity vectors are considered collinear).

Shortcut: 

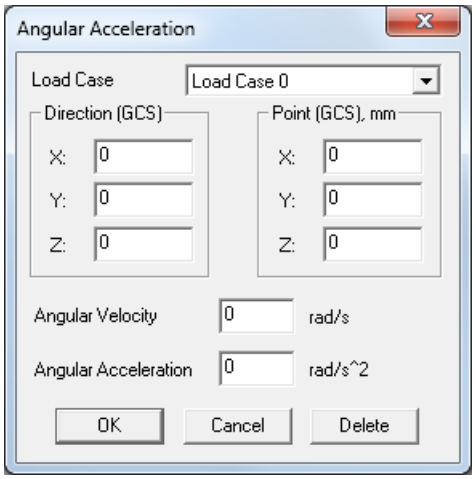


Fig. 2.79 Angular Acceleration dialog box

Load Case

Load case can involve a combination of loads of any types and is characterized by name and two states: on/off and active/inactive. State of construction can be calculated for any load case or load cases combination. Work with load cases is similar to work with layers. If a load case is switched off, it loads will not be represented on the screen. If load a case is active, then by default it will be suggested to place a new load in the active load case.

Shortcut: 

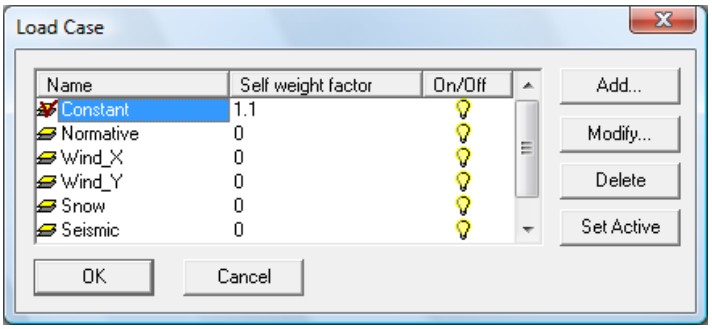


Fig. 2.80 Load Case dialog box

To create a new load case press **Add** button.
To change an old load case click on it in the list and press **Modify** button.
To remove a load case, select it in a list and press **Delete** button.
Press **Set Active** button to make active a load case selected from the load case list.

Dynamic Load Case

APM Structure3D allows automatic application of seismic loading in accordance with building regulations.

All dynamic forces are defined using dynamic load cases. All operations with dynamic load cases are similar to those for ordinary (static) ones, described above.

For detailed description see *Chapter 1*.

Load Combinations

Load combination represents a linear combination of load cases. This command calls a *Load Combination* dialog shown below. It is possible to create several load combinations. To add a load case into a combination, it is necessary to select it in a load case list box, enter a factor for it and press **Add** button. To change a load case factor, select a necessary load case from a load case list box or factors list box, enter a new value in a *Factor* field and push a **Modify** button. To remove load case from a combination, select it in a load case list box or factors list box and push **Delete** button.

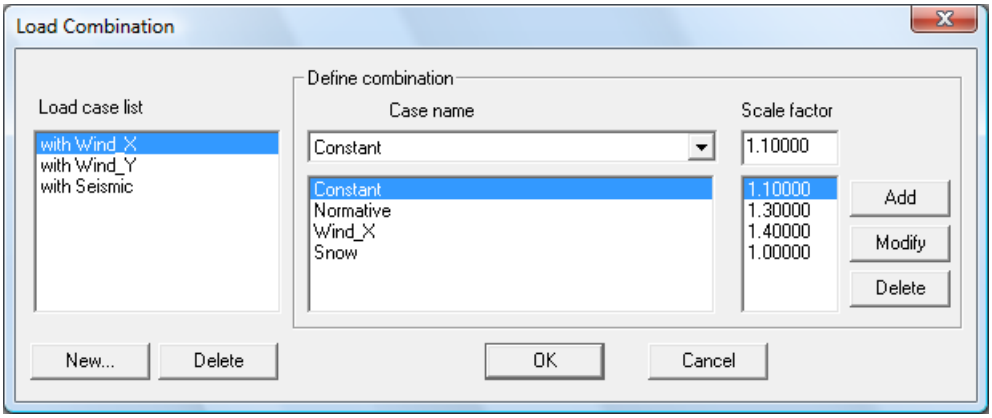


Fig. 2.81 Load Combination dialog box

Static Load to Mass

Static loads to mass conversion allows to consider not model dead weight only, but also the masses which set as a loads. It allows to perform calculations of eigen frequencies, seismicity and wind pulsations competently. Command calls dialog box shown below.

Load to mass conversion order:

1. In the *Add in* drop-down list select one of dynamic load case in which masses will be added. The *Dynamic mass* means that load case will be added in mass matrix at calculation of eigen frequencies. If dynamic load cases were created earlier (seismic or wind pulsations) there is possible to select one of them.
2. Select load case for conversion in the drop-down list, enter factor and press **Add** button. Loads components along Global Z axis will be converted to node masses.

✍ *Note: The model dead weight is considered once in mass matrix and there is no need to convert load case with structure weight multiplier.*

The list of static and dynamic load cases for conversion is presented in the bottom part of dialog box. Load case can be deleted from the list if necessary.

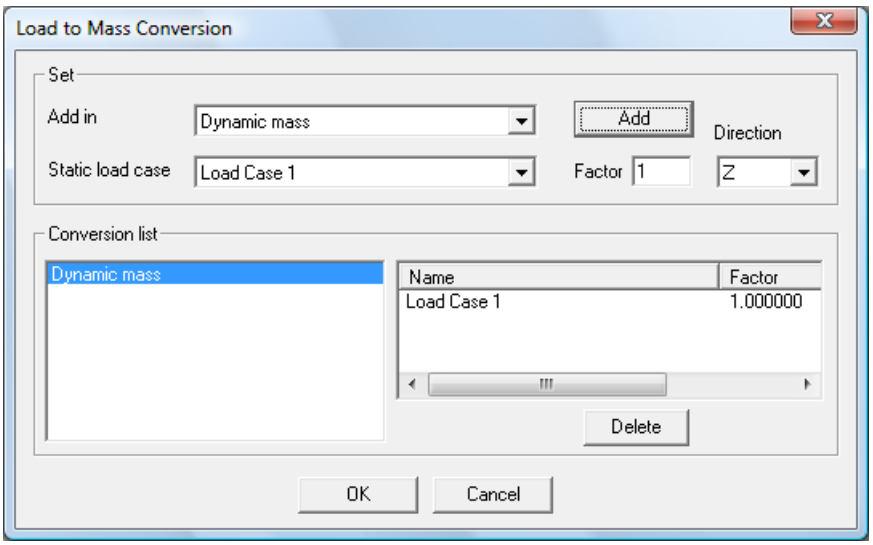


Fig. 2.82 Load to Mass Conversion dialog box

Stochastic Load Case

calls the window Fatigue Stochastic Loading for Load Cases. It is possible to set the fatigue stochastic loading for all loads or for individual ones (Figure 2.107).

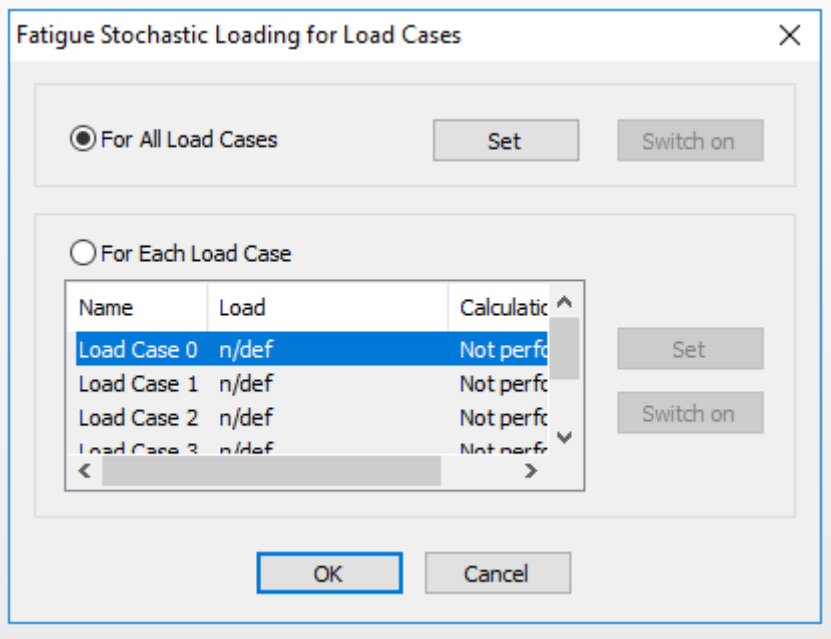


Fig.2.107 Fatigue Stochastic Loading for Load Cases dialog box

By clicking the **Set** button either for all loads or for one of the selected ones the window *Fatigue multistage stochastic load - the name of the load* appears (see Figure 2.108).

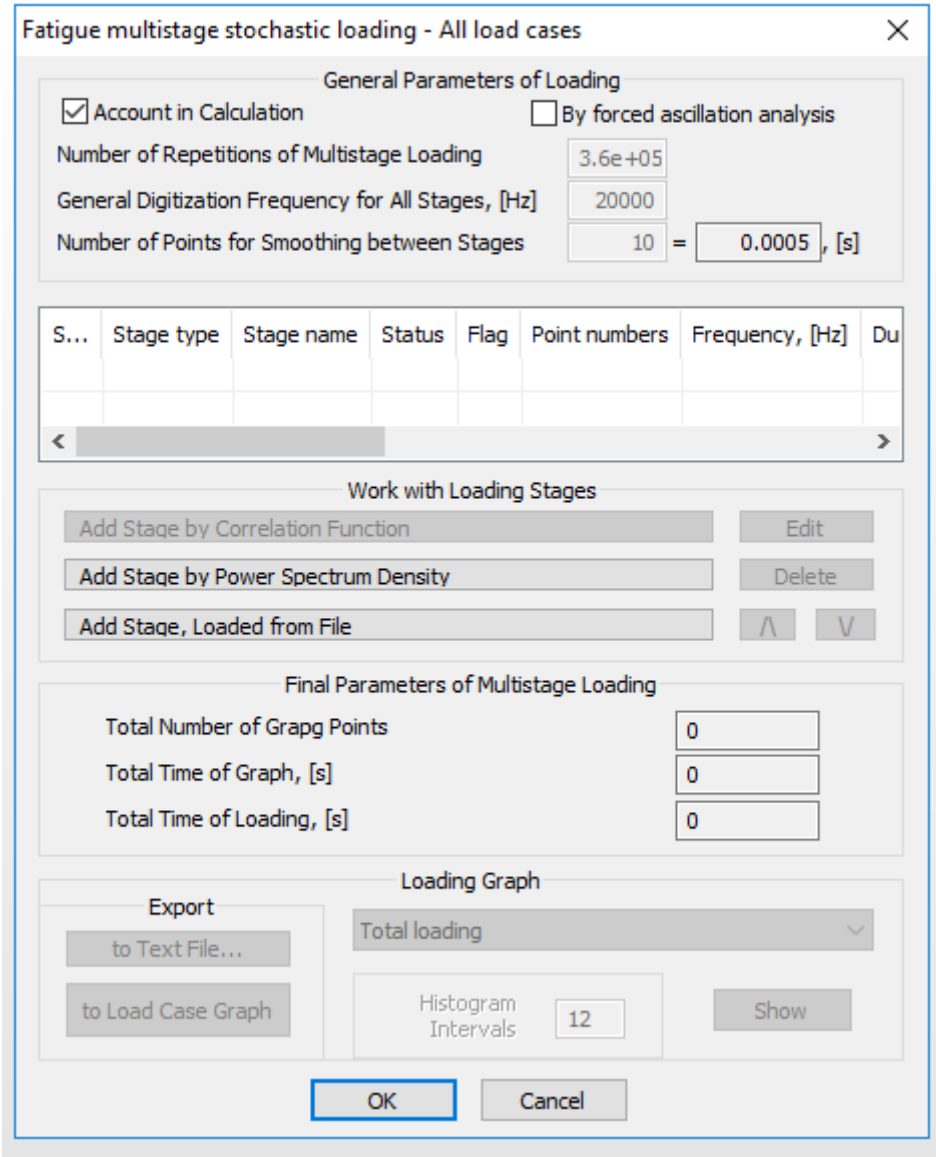


Fig.2.108 Fatigue multistage stochastic loading – All load cases dialog box

The stochastic loading consists of stages following one after another. Let's consider the creation of one of the stochastic loading stages.

The check mark *Account in calculation* switches the previously set fatigue stochastic loading for calculation. It is possible when the loading was set but is not included in the calculation. For more details about fatigue calculation see Chapter 10.

The **Add Stage by Power Spectrum Density** button opens the dialog box *Loading Stage –All Load cases* (Figure 2.109). In this window having the default parameters of the power spectral density and frequency parameters by pressing the **Calculate** button you will get a stochastic graph of the fatigue load modelling.

To view this graph click the **Show** button, the graph display window opens in which stochastic realization of the fatigue load stage is made (Figure 2.110).

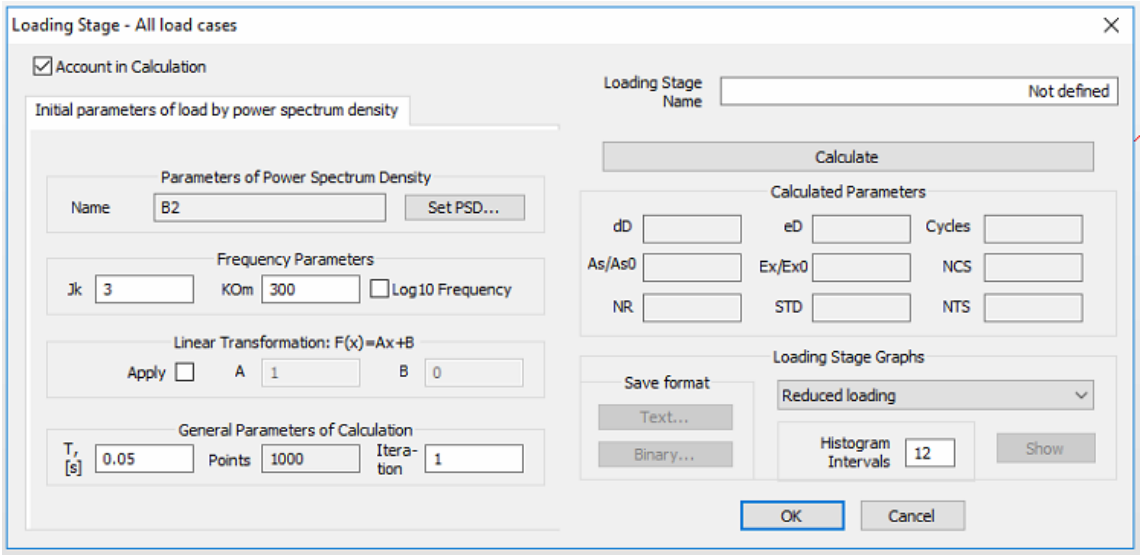


Fig.2.109 Loading Stage - All Load Cases dialog box

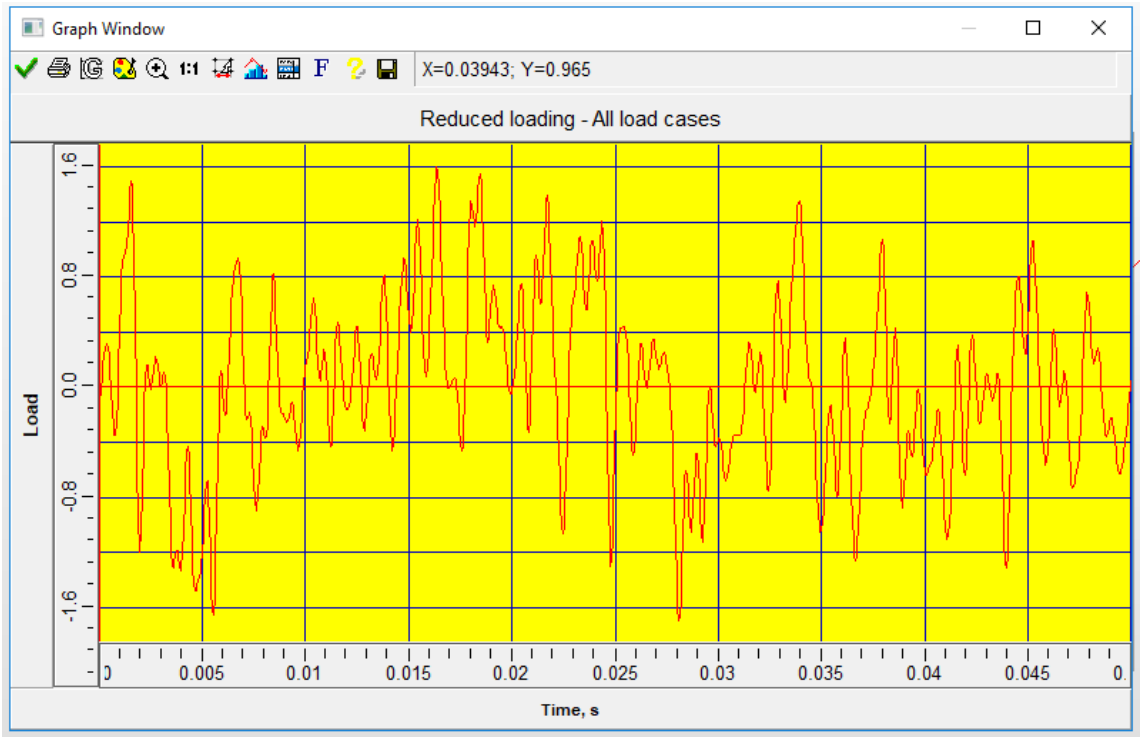


Fig. 2.110 Graphs window Reduced Loading dialog box

The implementation of stochastic fatigue loading stage can be saved in a format of *Text* or with extensions * .**prn** or * .**csv**, as well as a *Binary* file with the extension * .**pss** (see Figure 2.109), and also be read from the files of these formats with the **Add Stage, Loaded from File** button (see Figure 2.108).

Graph of Dynamic Load

The command calls an editor in which you can define graph of load changing in time for dynamic analysis. Here you define variation law for a proportionality coefficient, by which static load is multiplied, to obtain load value at certain time moments. All loads applied to the structure are variable in time according to the defined graph.

Shortcut: 

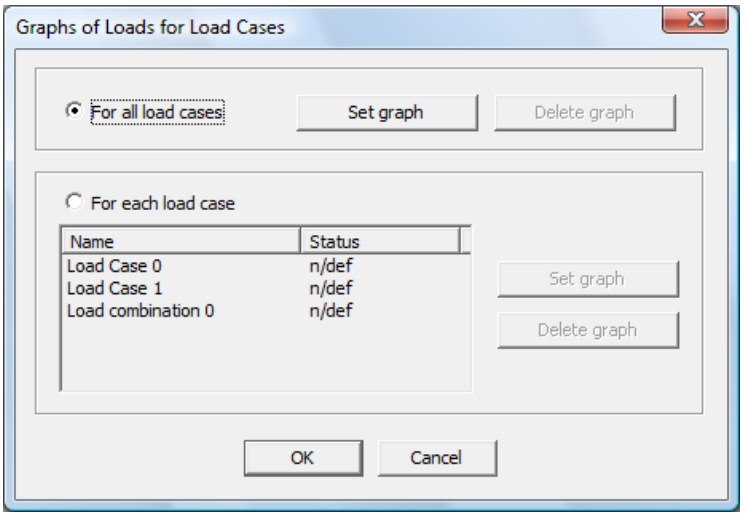


Fig. 2.83 Graphs of Loads for Load Cases dialog box

Selfweight on selected elements

This option allows you to take into account the weight of separate parts of the structure. To activate this option click the button Load case of a part structure in the dialog box Calculation Parameters, then the load multiplier will zeroise. After that you can set selfweight load in the Load Case menue. Following that a Selfweight on selected elements dialog box (Fig.2.102) appears showing load for selfweight on selected elements. The button "Unspecified" allows you to select elements for which selfweight is not set.

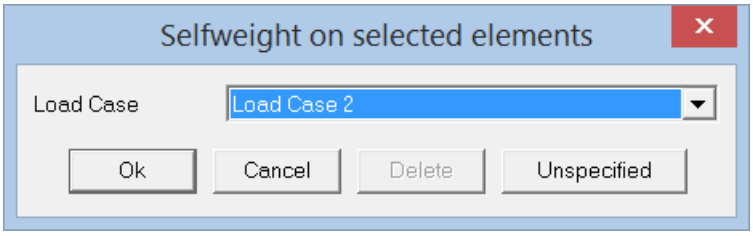


Fig. 2.1 Selfweight on Selected elements dialog box.

Tools menu

This menu contains commands that allow you to perform different operations with a structure to create more complicated geometrical models.

Copy

The command creates copies of selected elements into clipboard. To paste a created copy of elements into editor it is necessary to call **Tools / Paste** command.

Shortcuts: or Ctrl+C

Paste

The command pastes selected elements from clipboard into editor. Upon this, the pasted copy is selected and can then be moved to a desired position. When you select this command, a copy remains in the clipboard and can be pasted many times. The command is enabled after you select **Tools / Copy** command.

Shortcuts: or Ctrl+V

In the appeared dialog box it is necessary to select previously in what layers the copied objects will be inserted.

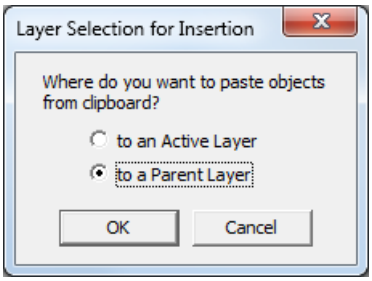


Fig. 2.84 Layer selection for insertion

Loft

This command enables *Loft* tool which allows you to create constructions that consists of repeated sections. These constructions are characterized by a single section vector and a number of sections. At first, it is necessary to select a node, a rod, a plate or a group of arbitrary combination of these elements, to make the command available. Then you are to set the lofting vector and the number of sections. Use first mouse click to set its starting point and second mouse click to set its ending point. Then, a dialog box shown below appears on screen which prompts you to enter sections number or edit the lofting vector. While creating solid elements out of plates with the help of lofting tool, it is necessary to remember that besides solid elements plates are also created by lofting the initial one. *Place new objects in* option allows to select layer in which new objects will be placed. *Copy* option allows to copy objects with all properties (loads, hinges, etc.)

Shortcut: 

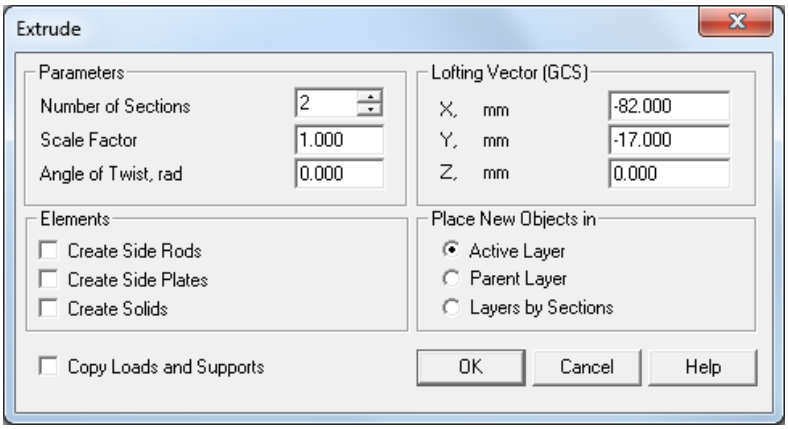


Fig. 2.85 Loft Construction dialog box

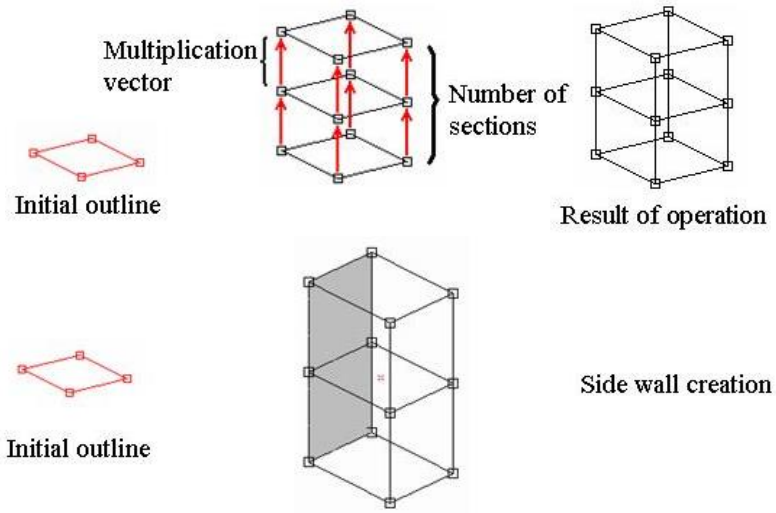


Fig. 2.86 Explanation for Loft tool

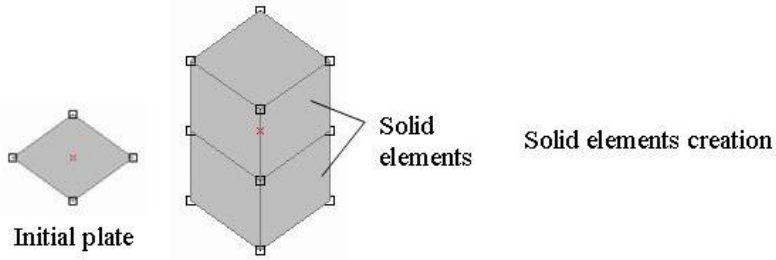


Fig. 2.87 Explanation for Loft tool (solid elements)

Modified *Loft* tool allows creating multi-sectional structures with linear dimensions change and sections rotation. These structures are characterized by displacement vector of a single section, number of sections, dimensions change factor and rotation angle of sections. At first, it is necessary to select a node, a rod, a plate or a group of arbitrary combination of these elements to make the command available. Then you are to set the vector that characterizes single section. First mouse click defines its starting point; the origin of the vector should be an existing node. This node becomes base for dimensions change, and rotation of sections will be made around this node in a plane perpendicular to displacement vector. Second mouse click defines the ending point of the vector. Then a dialog box appears that allows you to edit the vector value, set the number of sections, full rotation angle of a section and dimensions change factor. Lofting vector is set for a single section, so for N sections the aggregate lofting vector will be N times bigger. Rotation angle and dimensions change factors are set for the total amount of sections, so for single section rotation angle will be divided into N, and dimensions will vary according to the linear law. While creating solid elements out of plates with the help of extruding tool it is necessary to remember that besides solid elements plates are also created by copying the initial one by extruding vector.

Extrusion is performed in several stages:

- Copying selected nodes and nodes that belong to selected elements by extruding vector;
- Turning the copied nodes around the base one;
- Ranging the nodes with respect to the base one;
- Creating rods, plates and solid elements.

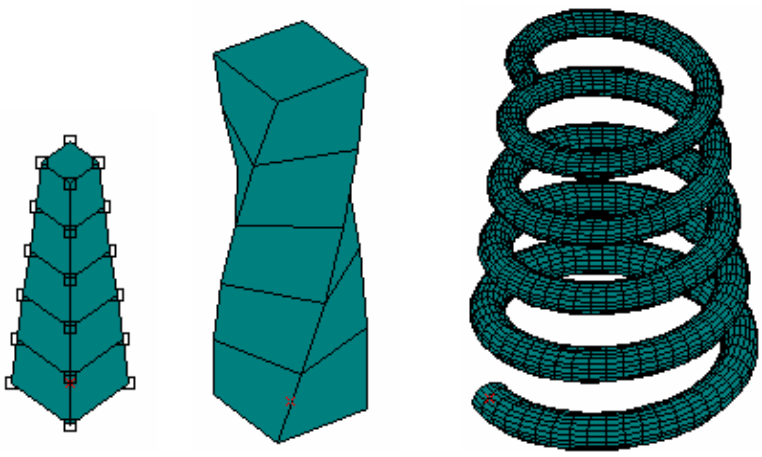


Fig. 2.88 Explanation for Loft tool (solid elements)

Rotate

The command enables a mode which allows you to rotate selected elements. The mode allows rotation of selected elements in the viewplane i.e. around a vector perpendicular to the viewplane. Rotation is done around the rotation center. Therefore you must choose correct viewplane first. After you have selected elements you need to rotate, click on them in the viewplane and rotate with mouse around the rotation center to reach the desired rotation angle, and then click with mouse again to lock the new position of the elements. Toolbar shows you the current rotation angle. The angle is increased or decreased according to *angular step*. Right mouse click cancels the operation.

Shortcut: 

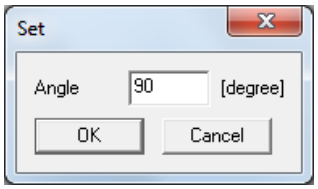


Fig. 2.89 Rotation Angle dialog box

Mirror

The command enables a mode which allows you to create a mirror copy (symmetry) of the selected elements. Symmetry is done with respect to a symmetry plane which is perpendicular to the view plane. To set a symmetry plane it is necessary to draw the symmetry plane track in the view. First mouse click sets the first point of the track; second mouse click sets the second point and creates the mirror copy. This mode allows you to use *attachment* to nodes while drawing the track line. Right mouse click cancels the operation. *Place new objects in* option allows to select layer in which new objects will be placed. *Copy* option allows to copy objects with all properties (loads, hinges, etc.)

Shortcut: 

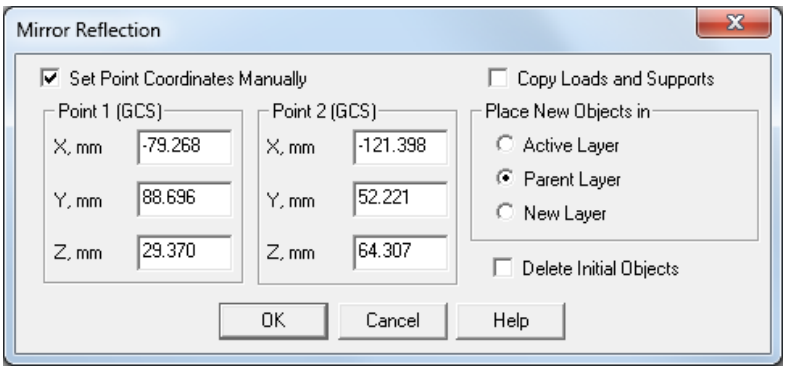


Fig. 2.90 Mirror Reflection dialog box

Polar Array

The command enables a mode of creating polar array of selected elements. Array is characterized by a rotation vector and a full rotation angle. The command copies selected elements around the rotation vector. This tool has a possibility not only to copy, but also to join consecutive copies with rods, plates and solid elements (see *Loft* command). Rotation angle is set for the total amount of sections, so if the total number of copies is N, then the rotation angle for each section will be divided into N. You should select a node, a rod, a plate, a solid element or a group of arbitrary combination of these elements at first to make command available. Then it's necessary to set a vector – the axis of rotation. First mouse click sets the starting point of this vector; it must be an existing node. The axis of rotation will run through this node. Second mouse click defines the ending point. After that, a dialog box appears on the screen where other parameters: number of sections, angle are set and the mode of joining copies with rods, plates and solid elements is switched on. *Place new objects in* option allows to select layer in which new objects will be placed. *Copy* option allows to copy objects with all properties (loads, hinges, etc.)

Shortcut: 

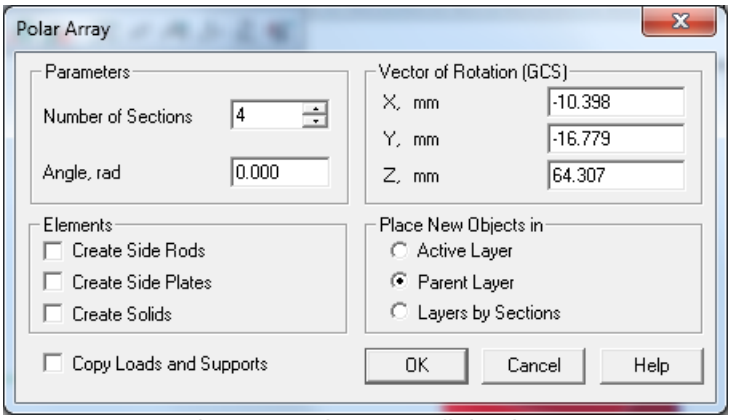


Fig. 2.91 Polar Array dialog box

Node Alignment

The command allows you to align a selected node by base node coordinates. With the help of this tool, it is possible to “project” nodes on the plane that runs through the base node and is parallel to one of the coordinate planes, or on the straight line that runs through the base node and is parallel to one of the coordinate axes. To project nodes on the plane that is parallel to XY plane of the global coordinate system, it is necessary to select a node that lies in this plane and check off *Align By Z Axis* in the dialog box. If you check off *Align By X Axis* and *Align By Y Axis*, then the nodes will project on the straight line parallel to Z axis.

Shortcut: 

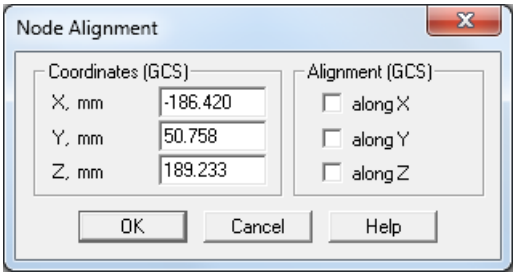


Fig. 2.92 Node Alignment dialog box

Move Nodes

This command allows to move nodes. In the appeared dialog box you can set offsets along X,Y,Z directions.

Shortcut: 

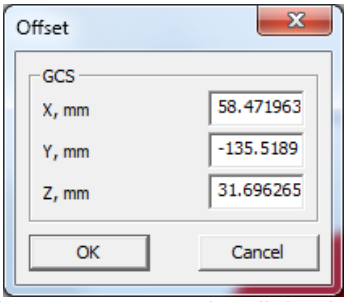


Fig. 2.93 Move Nodes dialog box

Spring

The command creates springs. This command calls a dialog box shown below that allows you to enter basic spring parameters: radius, step, coils number and also the number of straight-line rods per one coil. This parameter defines the degree of accuracy of spring model. Spring is placed in (0, 0, 0) point. Its axis is a vertical line.

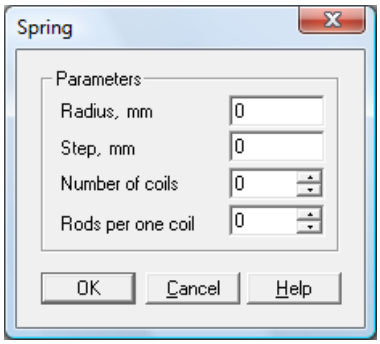


Fig. 2.94 Spring dialog

Pattern Grid

Command activates special mode that allows you to create rods with a single mouse click using pattern grid which appears after command calling in *active* state. Grid consists of horizontal, vertical and inclined segments. Pattern grid is set by three parameters: horizontal step, vertical step and rotation angle of the whole grid. Click on the necessary segment to create rod. Created rod will duplicate the selected segment. To delete rod it is enough to click on it again. Pattern grid creates rod lying in the viewplane.

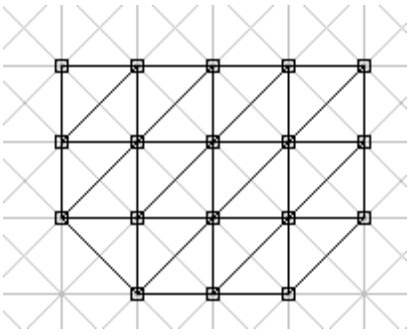


Fig. 2.95 Pattern grid

Pattern Settings

Command allows setting parameters of pattern grid shown above. Command invokes dialog box shown below.

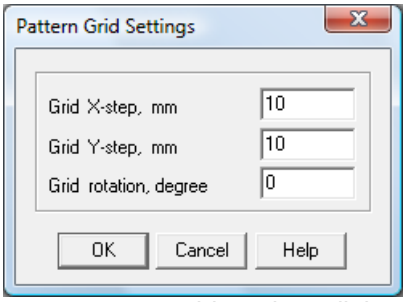


Fig. 2.96 Pattern Grid Settings dialog box

Layers

The command calls Grid Settings dialog box

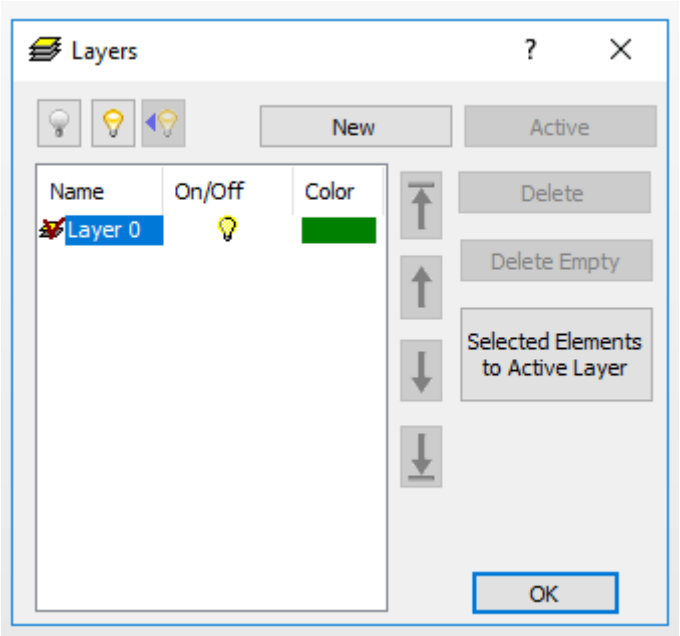


Fig.2.125 Layers dialog box

Disable all - the command switches off showing of all layers (the lights of all layers are damped).

Enable all - the command switches on showing of all layers (the lights of all layers are switced on).

Return to the previous state - the command restores the previous display state of the layers which was before the current moment.

On/Off On / Off - the command switches on / off showing all layers (all layers buttons get enabled/disabled)

New New - the command creates a new layer called "Layer n" where n is the current number of the new layer following the number of existing layers.

Active Active - the command makes the status of the selected layer active. Active is the layer in which the newly created elements of the construction will be placed. A check mark appears to the left of the name of the active layer. The name of the active layer is shown in the "Current Options" toolbar.

Selected Elements to Active Layer Selected Elements to Active Layer - the command places the selected elements in the active layer . - To Select Elements to Active Layer the same function is performed by the button on the "Current Options" toolbar.

Move up the list - the command moves the selected layer to the first line of the list of layers.

Move up one position - the command moves the selected layer one line higher in the list of layers.

Move up one position - the command moves the selected layer one line below in the list of layers.

Move to the bottom of the list - the command moves the selected layer to the last side of the list of layers.

Delete

Delete - deletes the selected layer. When deleting a layer that contains elements, the latter are moved to the active layer.

Delete Empty

Delete Empty - deletes those layers in which there are no construction elements.

Clicking the LMB on the color bar allows you to set its own color in each layer.

Color layers - the button in the "Current settings" toolbar, allows you to "colorize" elements belonging to different layers in the color that relates to them.

Semitransparent shell elements - the button on the "Current parameters" toolbar, allows you to "paint" the shaded elements in different shades in the shaded layers in order to determine their position in the structure.

A user can create new layers, give names to them, assign active ones, disable / enable or delete them, and assign color for visual control. An editor always has one active layer in which the elements are placed when creating. Items that are in the off layer are not shown on the screen and are not selectable. The selected elements can be moved from those layers in which they are into the active layer.

Shortcut:

Add to Current Layer

The command transfers all selected elements into the active layer.
Shortcut:

Check / Model Connection

The command checks the structure for connections between elements. All separate elements are selected automatically.

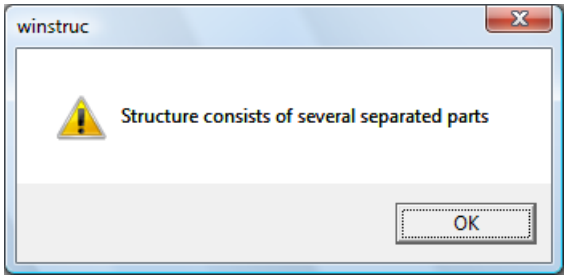


Fig. 2.97 Elements connection message box

Check / Materials

The command checks materials of model elements. For example, laminate property of material can be set for plate element only.

Check / Rod Cross-Section

The command determines whether any rod has an undefined cross-sections. All rods with undefined cross-section are selected automatically.

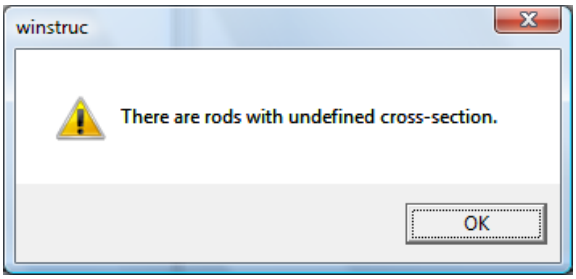


Fig. 2.98 Undefined cross-section message

Check / Plate Angles

The command checks whether plate angles fall in a given range. Minimum and maximum angles are set in the window shown below.

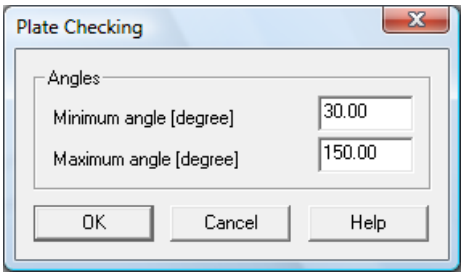


Fig. 2.99 Plate angles range dialog box for checking

Check / Solid Elements

The command invokes dialog box with the information about solids. After pressing **Calculate** button the following criteria of solid elements are calculated: *Volume*, *Jacobian*, *Aspect ratio*, *Collapse*.

The criterion selection is carried out in the drop-down list. The maximum, average and minimum values of model are read out in the upper part of dialog box for the selected criterion after calculation.

Further the upper and lower limits are set for checking the selected criterion.

After pressing **Check** button system will allocate solids, checking for which is not carried out.

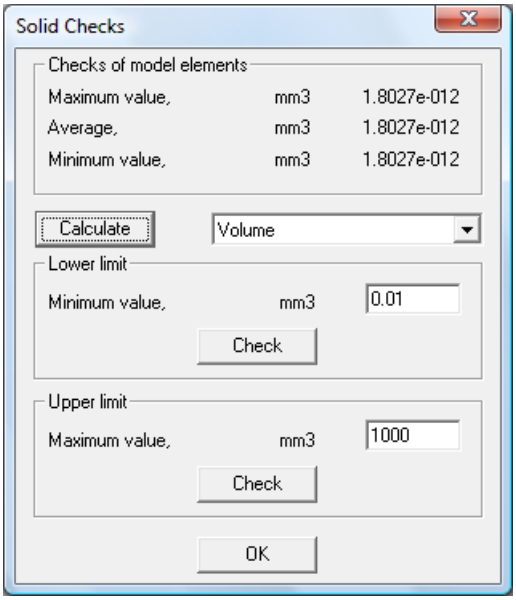


Fig. 2.100 Solid Checks dialog box

Check / On Duplicate Rods, Plates, Solids

This command allows to check coincidence of elements. The warning message appears on the screen if check don't meet the conditions.

Check / Aspect Ratio

This command displays color map *aspect ratio* that is the ratio of an element side (edge) maximum length to a minimum. Ratio value should not exceed 5:1.

Check / Tapering

This command displays color map *tapering*. Tapering is equal to zero for 3-noded plates or 4-noded tetrahedrons. The value range is 0...1. 0 is the ideal value.

Check / Jacobian

This command displays color map *jacobian*. The value range is 0...1. 1 is the ideal value. 0.7 is acceptable value.

Check / Warping

This command displays color map *warping*. Warping is equal to zero for 3-noded plates or 4-noded tetrahedrons. 5 degrees is acceptable value.

Connect Nodes

The command joins the nodes lying closer than entered in dialog box shown below.

If you specify a cluster of nodes and do the same operation, then the operation will be applied to the nodes of the specified part of a structure.

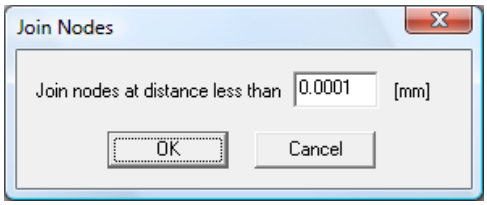


Fig. 2.101 Join Nodes dialog box

Mesh operations/Increase the order of **volume elements**

The command allows converting solid finite elements of a low order (4, 5, 6, 8-noded) into solid finite elements of a high order (10, 13, 15, 20-noded) which allows to increase the accuracy of calculations, while reducing requirements to the quality of the initial grid.

Mesh Operations/Separate volume elements

The command allows to separate the solid-elements of one detail from solid-elements of another detail. For example, you are to separate two details, which were designed as a single finite element model and have common nodes. To do this, place the solid-elements of every detail in separate layers, select solid-elements placed in different layers and having common nodes, then activate the Tools / Mesh operations /Separate volume elements. Two separate nodes will be created in each junction node (one node belongs to one detail (a layer) and the second node belongs to another detail or a layer). If any layer is switched off, the junction nodes will be still displayed on the model.

Mesh Operations /Mesh refinement (tetrahedrons 4)

This tool allows to refine the mesh of finite elements in models made of solid finite elements (4-noded tetrahedrons). Having activated the tool **Mesh refinement a** dialog box appears (see below) where you can choose the parameters of mesh refinement and a node around which the mesh is to be refined.

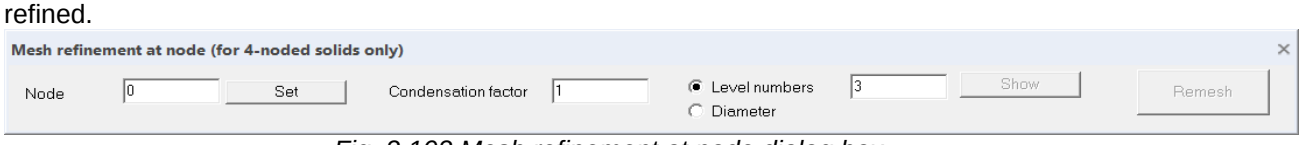


Fig. 2.102 Mesh refinement at node dialog box

Mesh operations/ Create free edges

This function simplifies the process of visual search for any incoherence in a model. Having activated this function, free edges will be highlighted during a dynamic rotation of the model.

Fig. 2.103 "Create Free Edges" function

Mesh Creation/Volume Fraction

The Volume Fraction dialog box (Figure 2.135) is used for preparation of topological optimization and subsequent postprocessing.

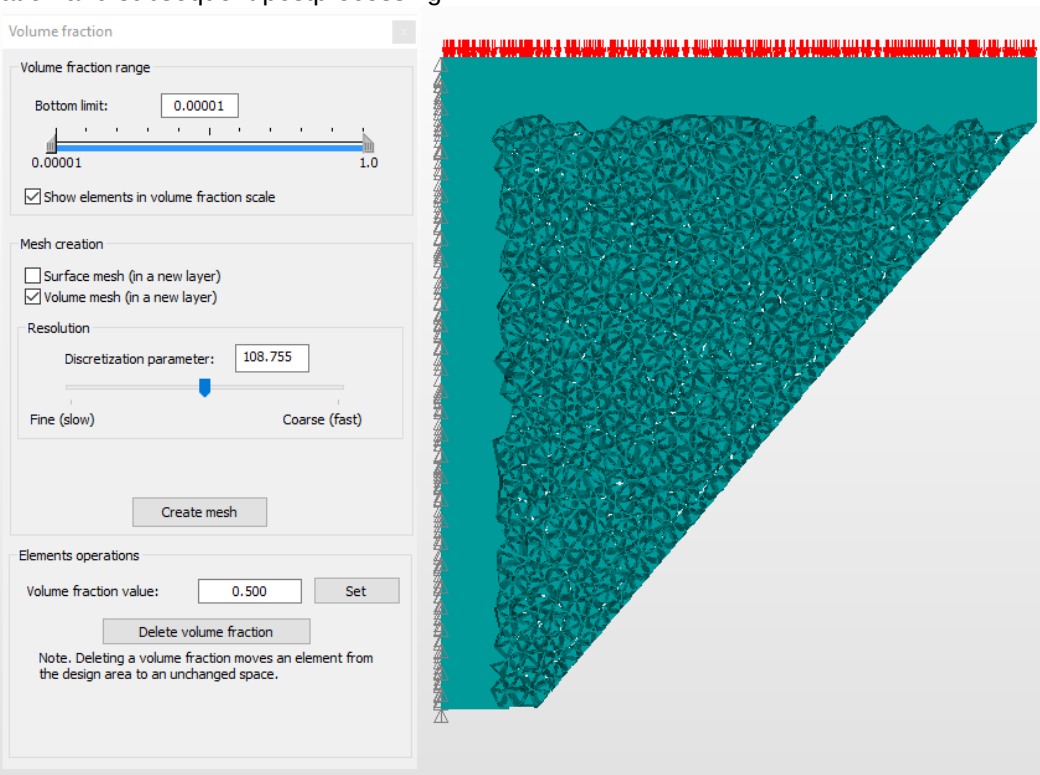


Fig. 2.135 Volume fraction dialog box and the construction in which the volume fraction is specified.

The nodes display will automatically turn off when you open the dialog box. To see the nodes (and the loads applied to them), enable the node display by clicking the Show nodes button on the *View Filters* toolbar.

To specify the area in which topological optimization will be performed, it is necessary to select the volume elements of the structure, set them the value of the volume fraction (by default 0.5) and click the **Set** button.

After that, a sparse structure will appear on the selected part of the structure (Figure 2.135 on the right), in which optimal structures will be created when calculating topological optimization.

By using the **Delete volume fraction** button, it is possible to delete the previously specified volume fraction, or a part of it, indicating that in the remote areas of the volumetric zone no topological optimization will be performed

Create Contact Elements

That tool is intended for contact zone creation (contact elements on free sides of the allocated solid elements). Before performance of that command, it is necessary for the model of contacting

details to arrange in different layers and to allocate solid elements, which, presumably, will participate in contact interaction. If more than one contact area is expected, the command should be executed consistently for each prospective area is supposed. The command will create contact finite elements on one part, and target contact finite elements on the other part.

Create Super Elements

The command allows to separate whole model to set of super elements (SE). This makes it possible to perform static analysis for models of larger DOF number.

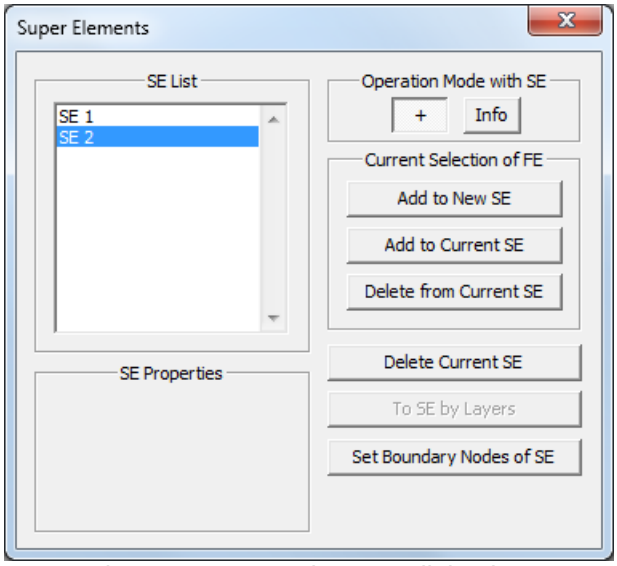


Fig. 2.104 Super Elements dialog box

Restrictions:
- all of the nodes and elements of each SE must be connected (linked);
- set of SE must contain all of the nodes and elements of whole model;
- static analysis can be performed only for set of SE.

+ button switches to SE operation mode (add/delete SE and etc.).
Info button switches to SE information mode.
Calculation results will be available for whole model as well as for each SE.

Element Groups

This tool is designed to unite different types of finite elements into a single group in order to perform fatigue calculation under stochastic external loading and to calculate topological optimization.

Features of work with GE:

- at least one group of elements in constructions must be created, for which the calculation will be carried out;

List GE is a list of groups of model elements.

Operating mode of a PC:

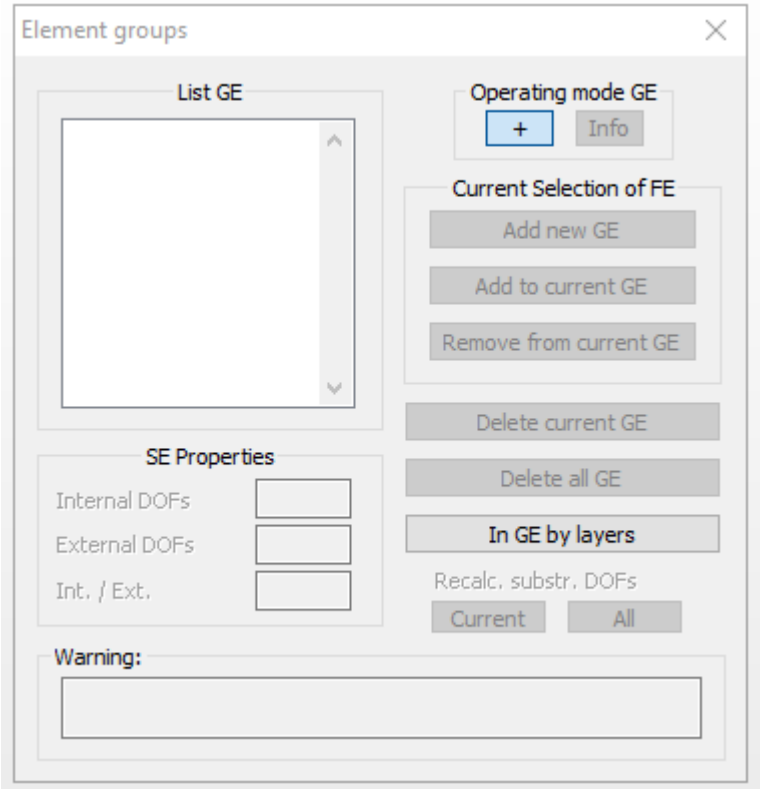
"+" - adding / removing elements of a construction in the GE. When selecting construction elements, the *Current GE selection* buttons become active, which allows:

- add to the new GE;
- add to the current GE;
- delete from the current GE.

"Info" is an information mode about the GE. When you click an element of a structure, it is highlighted in the GE list that contains the selected item and this is highlighted red on the model.

The buttons **Delete current GE** and **Delete all GE** delete, respectively, or highlighted GE or all GE.

The button **In GE by layers** allows to create a GE on the principle of belonging to different layers, however, to do this there should not be any GE in the GE List.



Element groups Dialog Box

Node –> Group of Nodes Connection

This tool is intended for creation of the several elastic links connected to one node. At first it is necessary to select group of nodes, after command activation specify nod to which elastic links should be attached.

In the appeared dialog box it is necessary to select what elements will connect nodes: rods or elastic links. At elastic link selection set its stiffness.

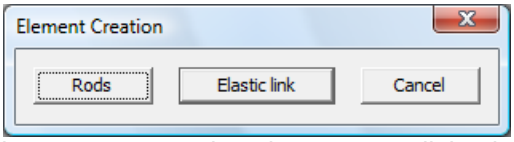


Fig. 2.105 Connection element type dialog box

Node Group –> Group of Nodes Connection

This tool is intended for creation of the several elastic links connected by pairs. After command activation it is necessary to select first group of nodes (they will be highlighted by blue color), second group of nodes (they will be highlighted by green color) and then press ENTER key. In the appeared dialog specify type of connection.

Separate Solids

The command allows to separate solid elements of one detail from another. For example, for contact problem solution, it is necessary to disjoin two details which are represented as FE model with the united nodes. For this purpose place solid elements of each detail in separate layers, select the solids located in different layers and having united nodes and activate **Tools / Separate Solids** command. Thus in each join node 2 disconnected nodes will be created that are corresponds to first detail (layer) and the second one. Join nodes will be shown on model at hiding of any one layer.

Additional Features / Intersection of 2 Rods

This command is used to intersect two rods that are in one plane. As a result there will be new node.

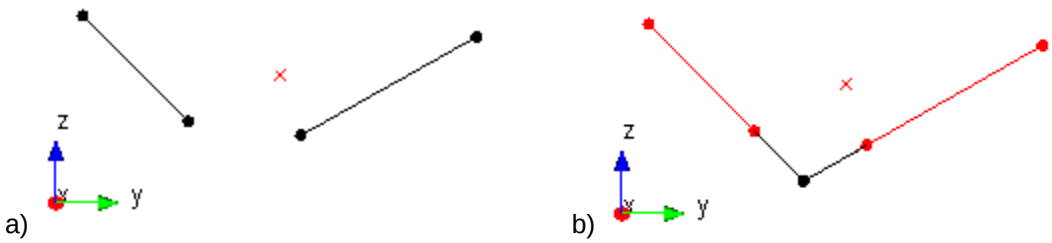


Fig. 2.106 Intersection of two rods

Additional Features / Minimum Distance between 2 Intersecting Rods

This command is used to connect two rods that are not in same plane. As a result there will be a new rod with two end nodes which connects two initial rods by minimum distance.

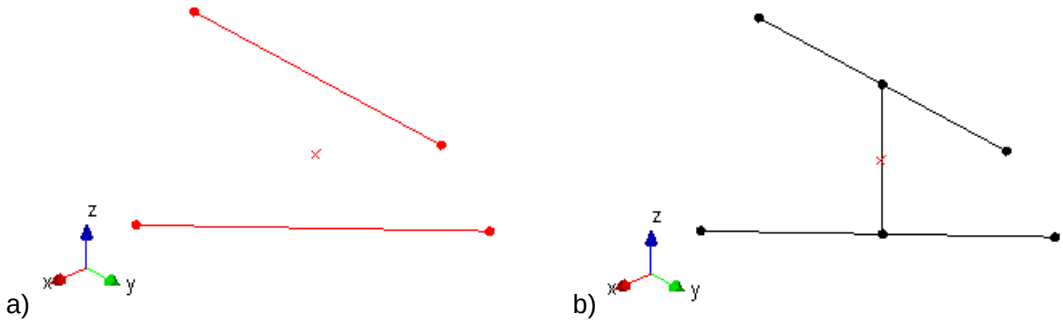


Fig. 2.107 Minimum distance between 2 intersecting rods

Additional Features / Intersection of Rod and Plate

Select intersected rod and plate that are not in the same plane and activate command. As a result there will be a new node in the intersection zone.

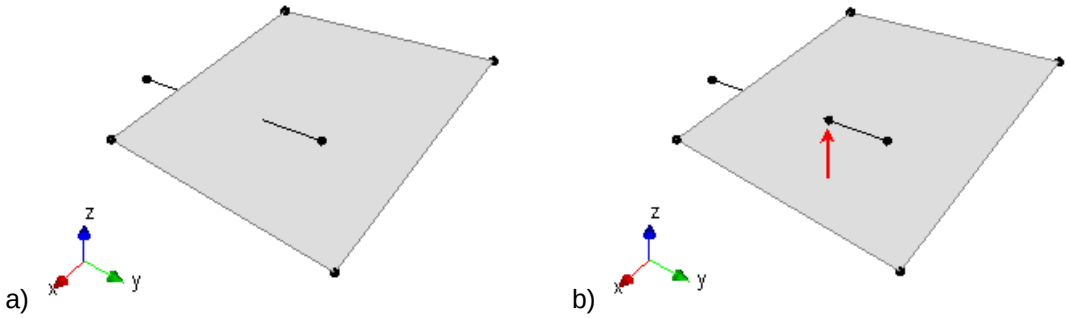


Fig. 2.108 Intersection of Rod and Plate

Additional Features / Intersection of 2 Plates

Select two intersected plates that are not in the same plane and activate command. As a result there will be nodes, which defines intersection zone.

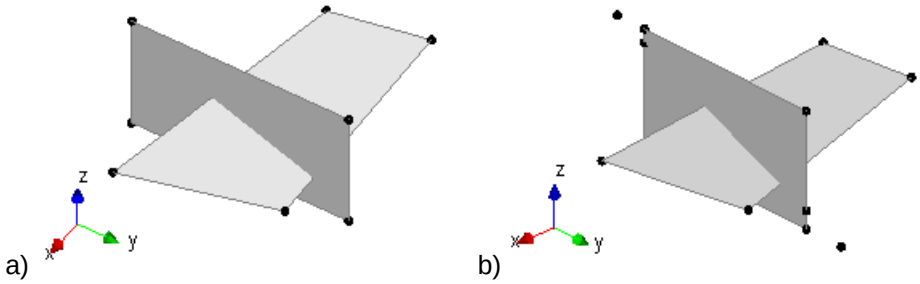


Fig. 2.109 Intersection of two plates

Additional Features / Angle between 2 Rods

Select two rods and after command activation there will be dialog box with angle between in degrees.

Depending on rods LCS the real angle or adjacent angle value will be shown.

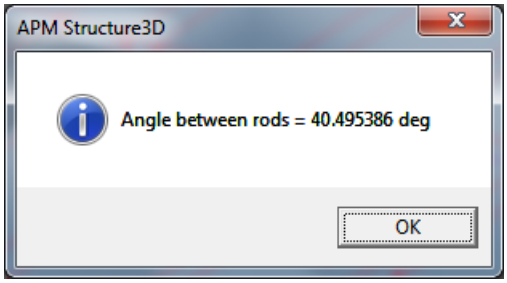


Fig. 2.110 Angle between two rods

Additional Features / Angle between 2 Plates

Select two plates and after command activation there will be dialog box with angle between in degrees.

Depending on plates LCS the real angle or adjacent angle value will be shown.

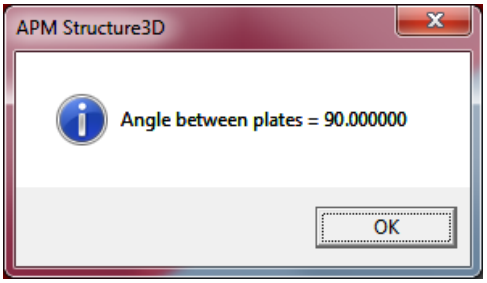


Fig. 2.111 Angle between two plates

Additional Features / Angle between Rod and Plate

Select intersected rod and plate and after command activation there will be dialog box with angle between in degrees.

Depending on the elements LCS the real angle or adjacent angle value will be shown.

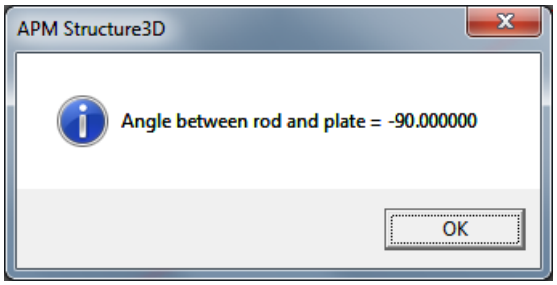


Fig. 2.112 Angle between rod and plate

Additional Features / Nodes on Rod Projection

Select one rod and required nodes and after command activation there will be new nodes on rod-directed line.

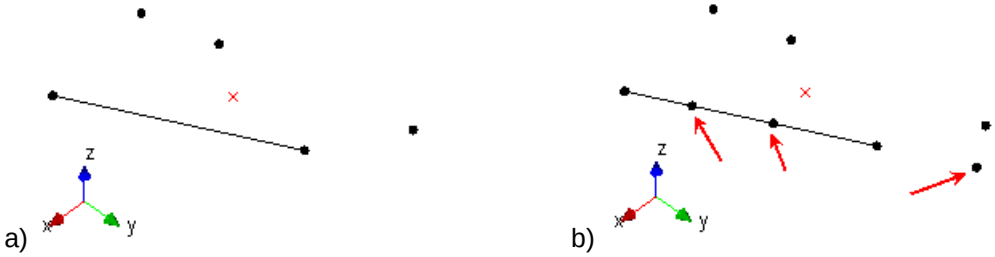


Fig. 2.113 Nodes on rod projection

Additional Features / Nodes on Plate Projection

Select one plate and required nodes and after command activation there will be new nodes lying in plane of plate.

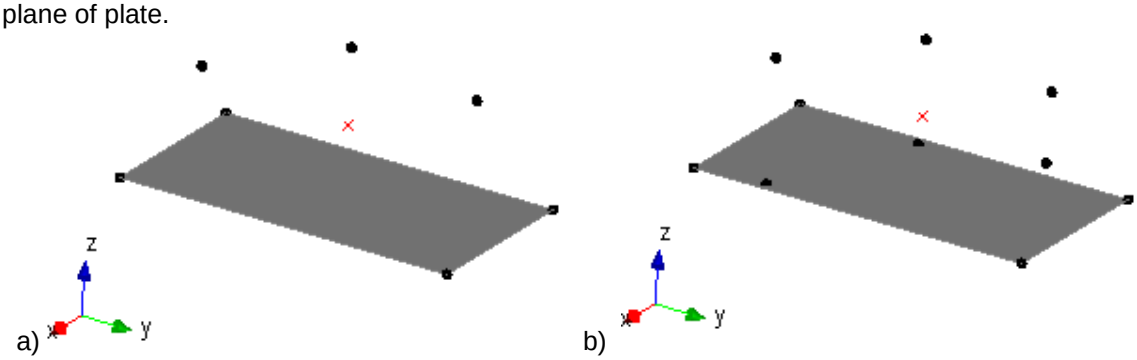


Fig. 2.114 Nodes on plate projection

Measure Distance between Nodes

After command activation specify two nodes and there will be distance between them in current units and it projections to X,Y,Z global axes in the appeared dialog box.

Shortcut: 

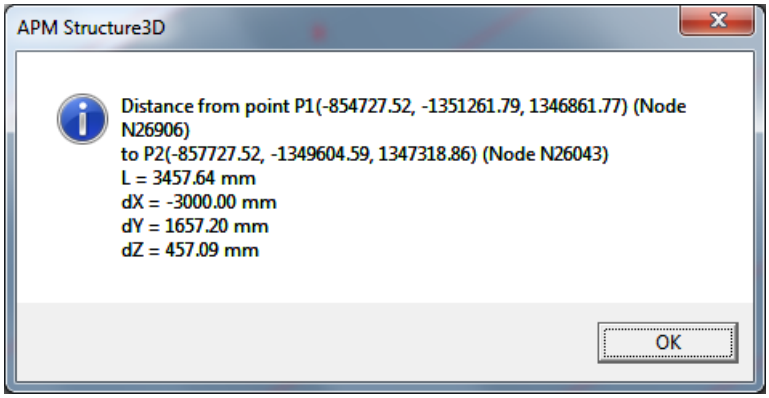
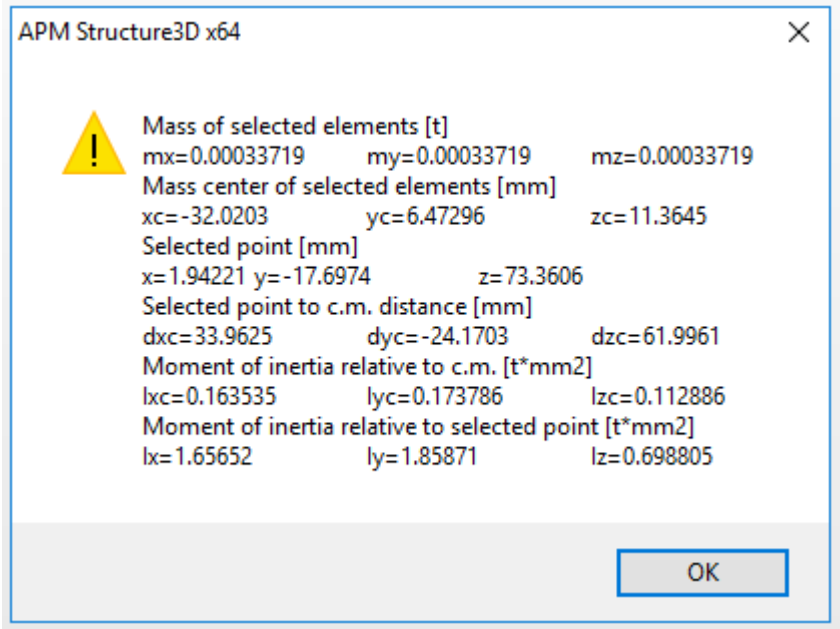


Fig. 2.115 Distance between nodes dialog box

Inertia of selected elements

The command allows to calculate inertia characteristics of a group of selected elements relative to the selected point (node).



Inertia Characteristics of selected elements

After selecting the elements for which it is required to determine inertia characteristics, one should specify the node for which the inertia characteristics will be determined and the corresponding window with results will be displayed.

FlowVision / TCP / IP exchange server ...

The command allows to create a TCP / IP exchange server for the joint solution of fluid and gas flow problems (FGA) and stress strain state (FSI).

FlowVision / Import of loads from FV

The command allows to import force factors acting on the model from the text files FlowVision. This is necessary for the joint solution of the problems of fluid and gas flow (FGA) and the problems of calculating the stress-strain state (FSI).

Properties menu

The commands of this menu allow you to set cross-sections and material parameters for rods.

Cross-Sections

The command calls a dialog box for cross-sections management. You can choose a current section that will be assigned to all newly created rods.

Shortcut: 

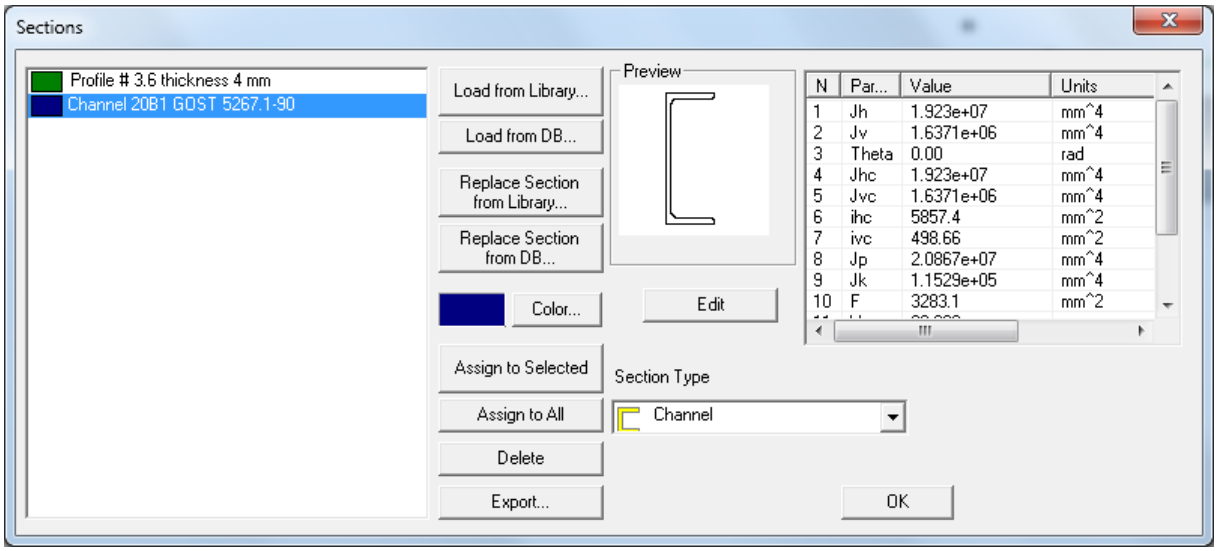


Fig. 2.116 Sections dialog box

With the help of **Color** button it is possible to set color for each section which will be used for rod visualization. Besides, it is possible to set cross-section for all or only selected rods with the help of **Assign to All** and **Assign to Selected** buttons. **Replace Section** buttons allow to replace completely in a design the selected section with another one.

Load from Library button invokes dialog box for selecting sections from libraries. Libraries are located in program installation folder.

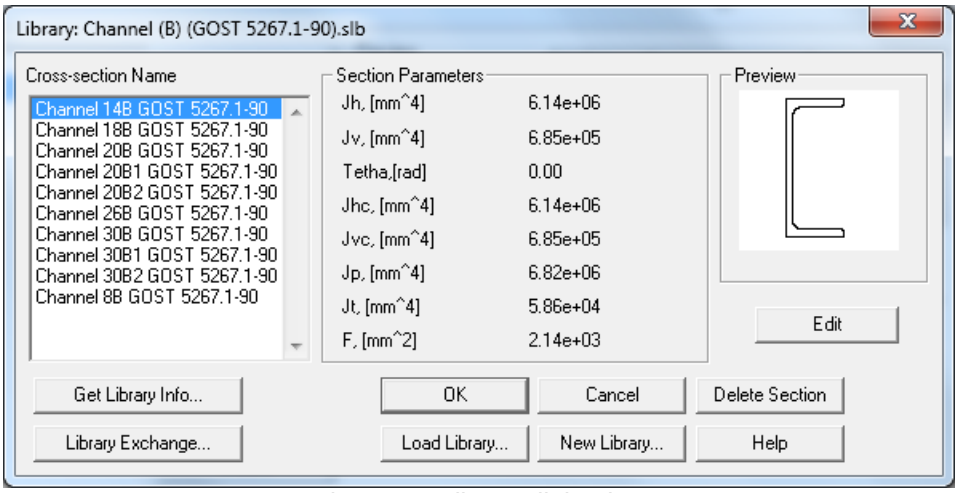


Fig. 2.117 Library dialog box

Edit button calls a dialog box where you can change geometrical properties for current section.

Get Library Info button calls a dialog box with library information.

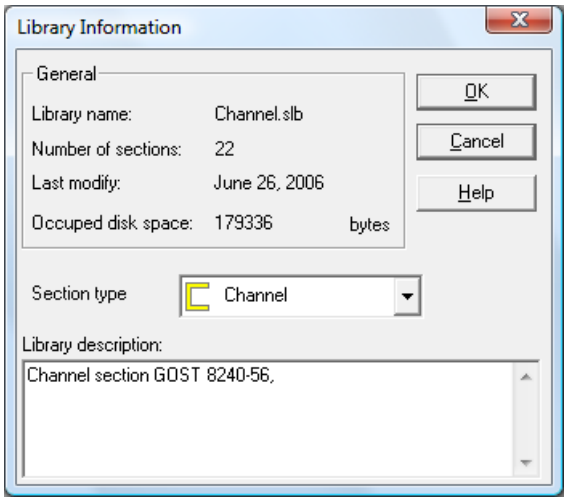


Fig. 2.118 Library Information dialog box

Library Exchange button allows you to import and export sections between two libraries.

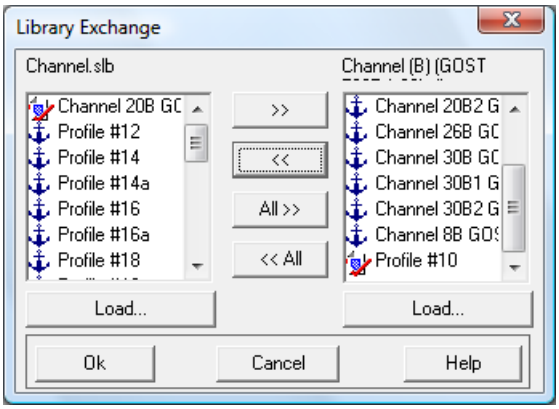


Fig. 2.119 Library Exchange dialog box

This icon means that section belongs to that library.

This icon means that section was imported from another library.

Load Library button opens another library.
Delete section button deletes selected cross-section from the library.
New Library button creates a new library.

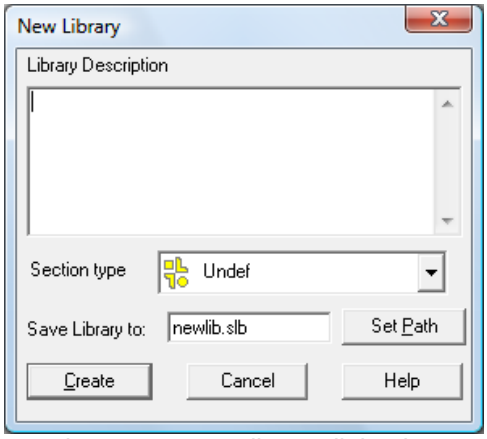


Fig. 2.120 New Library dialog box

Load from DB button allows to add section in model from database of parametrical sections, passing libraries. To call database manager make right click in a database tree.

Open in a tree interesting section type by the left double click, then select required section parameters in the list and press **OK** button. You can change section parameters in *Variables* dialog box if necessary. Do not change the standard section dimensions needlessly.

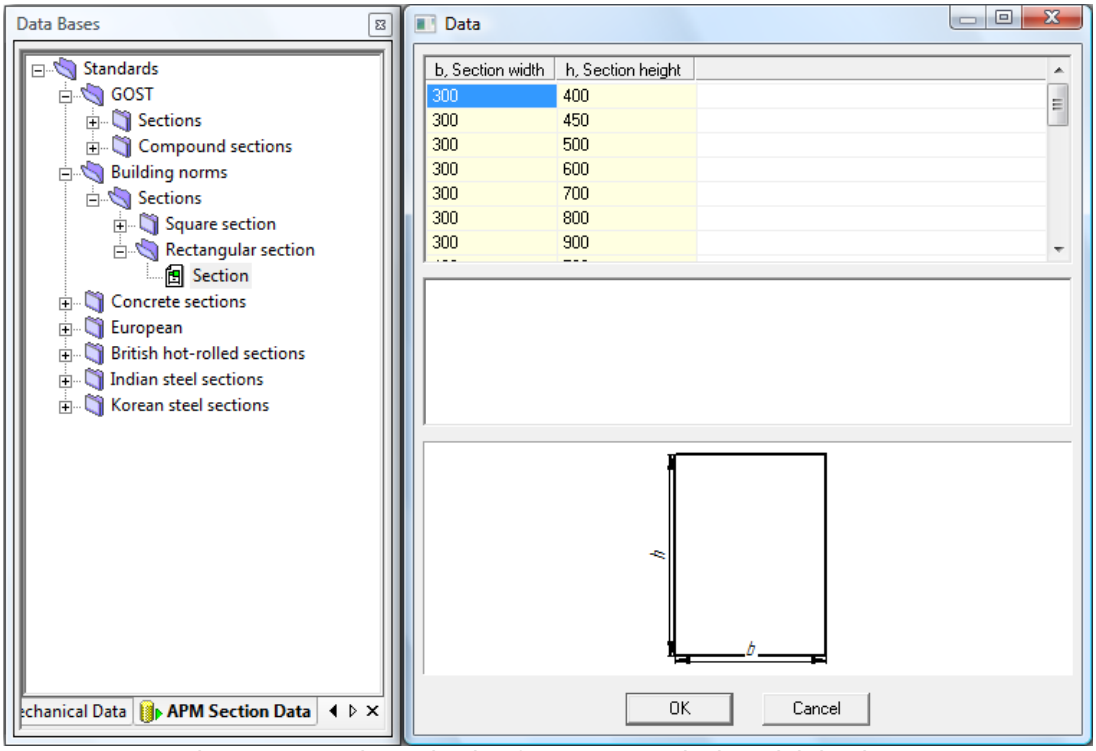


Fig. 2.121 Section selection from parametrical model database

✍ *Note: For correct reinforcing design it is necessary to specify section type: -1 – not defined, 0 – equal flange I-section, 1 – non-equal flange I-section, 2 – T-section, 3 – channel, 4 – angle, 5 – square pipe, 6 – rectangle, 7 – round, 8 – circle.*

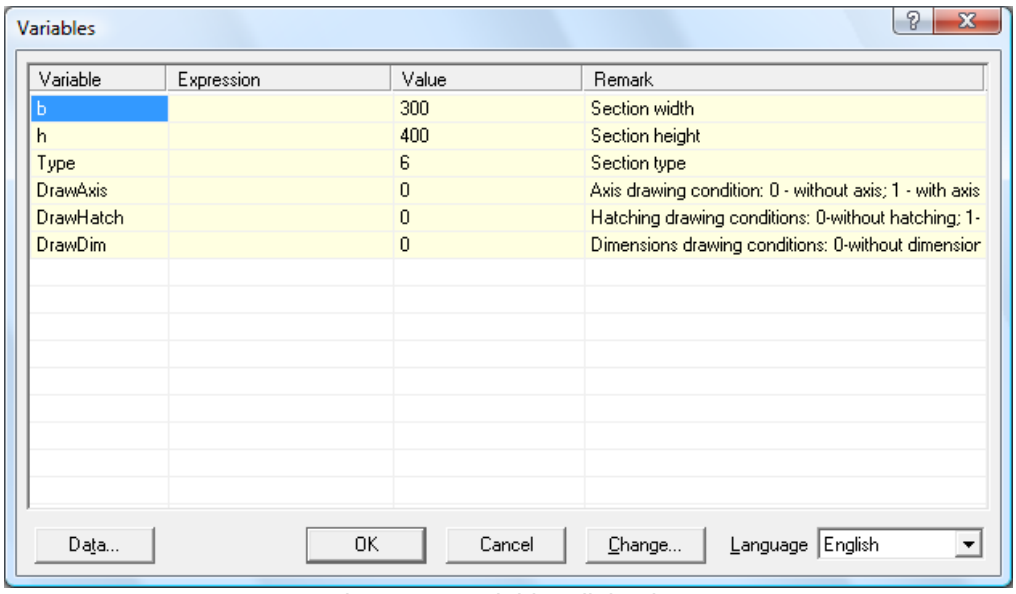


Fig. 2.122 Variables dialog box

Further the system will automatically define section dimensions suggest to enter section name. It is recommended to use section dimensions in its name. After that the section will be added in the list of sections of the current document.

Export button transfers the section to section library. Thus there will be a dialog in which it is necessary to specify a path to library and enter the name of added section.

Edit button calls a dialog box where you can change geometrical properties for current section.

In the right part of a dialog you can view image and geometrical characteristics of section. Section type and values of geometrical characteristics can be changed the subsequent calculation of design elements.

Cross-Section to Selected Rods

The command assigns section to selected rods. This command calls a *Library* dialog box. *Cross-section Name* list shows cross-sections contained in the library, *Section Parameters* group shows the main geometric characteristics of selected section, *Preview* group shows you cross-section reduced view.

Shortcut: 

Cross-Section to All Rods

The command allows you to assign cross-section to all rods.

Shortcut: 

Cross-Section Orientation

The command enables a mode which allows you to review cross-section orientation with respect to rod axis and to rotate cross-section about its axis. To do that, select a rod first. Then, the rod will be highlighted with another color and its coordinate system will be displayed as well as cross-section attached to this system (if it has been assigned). To rotate the coordinate system, click on it again to enable rotation mode. Moving mouse left or right sets the desired rotation angle. Angular cursor step is set using **View / Cursor Step**.

This mode allows you to direct Y axis of the rod system toward any node. That means that you can get rod axis, Y axis and the line between rod end and the selected node lie in the same plane. To do that, click in node sensitivity zone while in rotation mode. Here is an example. In the Fig. below we want to rotate cross-section of the selected rod. We will orient the rod system to have *Y* axis lying in the plane set by *X* axis and *bp* line. In rotation mode we click at *p* point. The result of the operation is shown below.

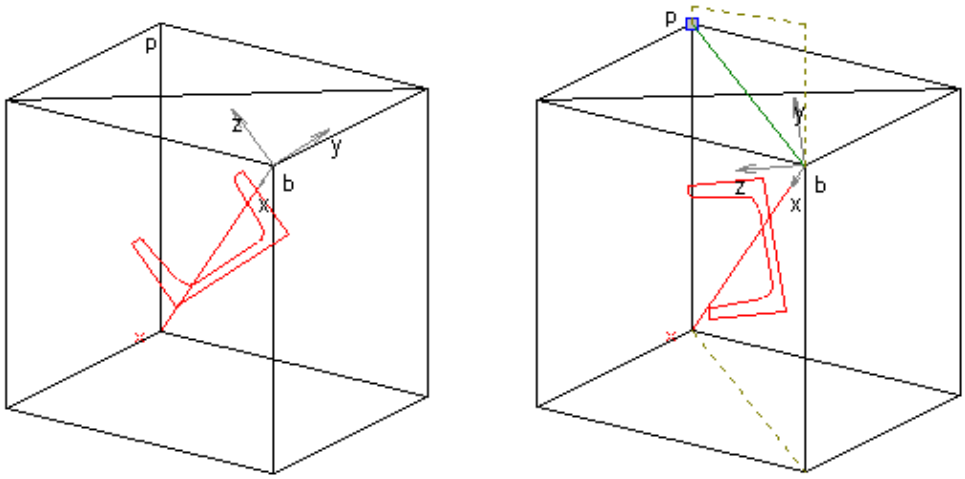


Fig. 2.123 Explanation of cross-section orientation

By default, the coordinate in the system of a rod X axis lies along rod's axis from the first node towards the second one. Y axis lies in the X axis plane of a rod and global Y axis. Z axis adds up to X and Y axes to form a right-hand triple.

You can use grey keys "+" and "-" to zoom cross-section in and out.

Shortcut: 

Rod Info

The command enables a mode that helps you review information about the rod. After you select a rod, a dialog box appears on the screen. *Rods* list in the left contains all construction rods and allows you to select rod to review. You can also use this list to change the name of any rod. Double click on

the rod name in the list to call a dialog box which allows you to enter a new name. *Section* group contains information about the cross-section with its preview. *Material* group contains information about material. You can change type of selected rod element using the list box under *Rods* list to any of BEAM, TRUSS or CABLE.

Shortcut: 

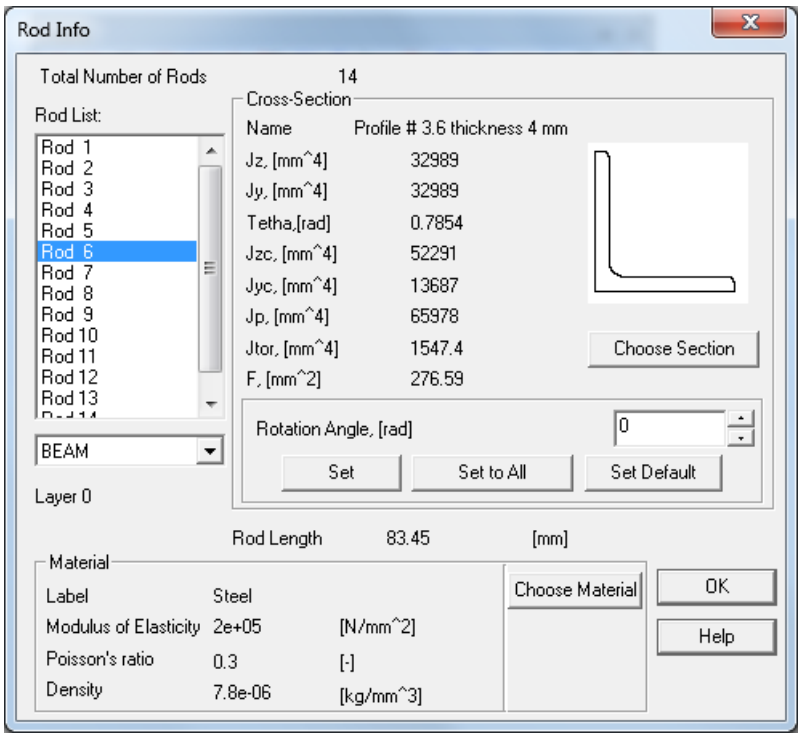


Fig. 2.124 Rod Info dialog box

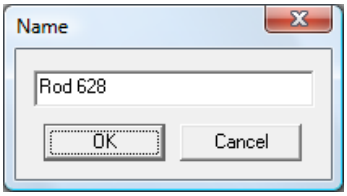


Fig. 2.125 Rod Name dialog box

Cross-Section Alignment Point

This command allows you to set alignment point for rod elements. By default rod elements are connected at center points of cross section. In the dialog box shown below you can select one of predefined alignment points as well as define a point yourself entering offsets from cross section center in *Offset* group.

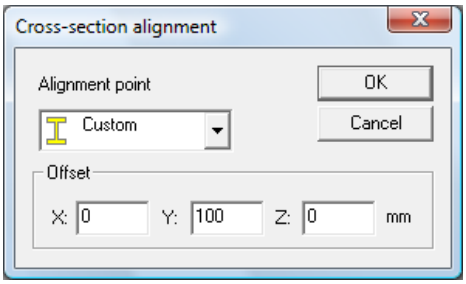


Fig. 2.126 Cross-section alignment dialog box

Invert Rod Local CS

The command replaces rod Local CS origin to other rod node. X axis directs along rod length, but already contrariwise; Y axis direction is not change; Z axis direction makes opposite and Local CS remains right.

Length of Selected Elements

The command allows user to see the length of the allocated rod elements (BEAM / TRUSS / CABLE).

Rod Element Type

The command allows you to set current and change rod element type (BEAM / TRUSS / CABLE).

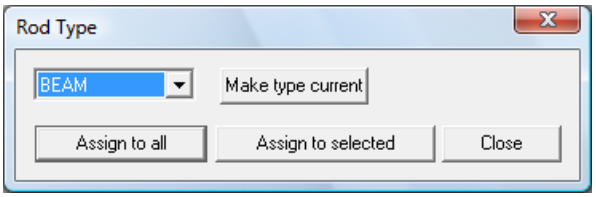


Fig. 2.127 Rod Element Type dialog box

Pipeline elements properties...

With help of this command the dialog window *"Modeling of pipelines"* is called.

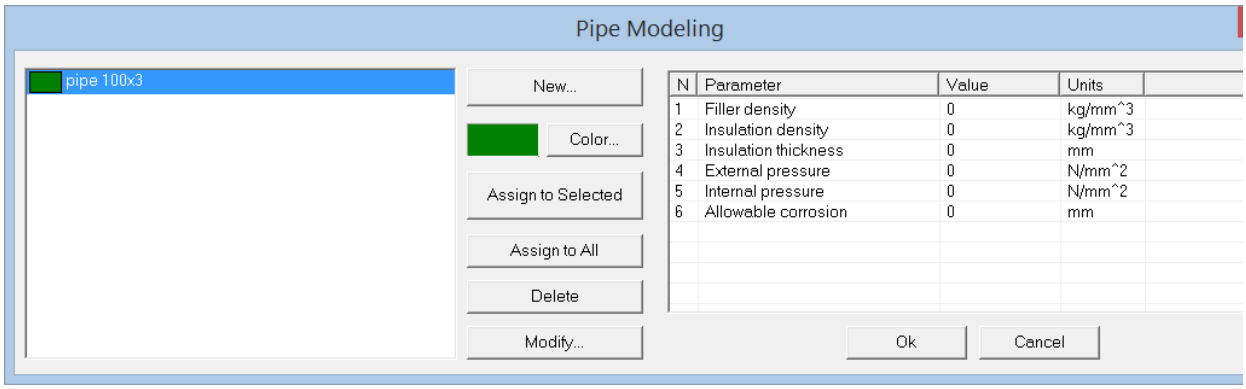


Fig. 2.128 Pipe Modeling dialog.

The defined name is being assigned to the group of bar elements with a ring section (the pipeline sections) and identical parameters of load are specified to the elements of this pipeline section.

With help of this window buttons one can specify for the selected pipeline section:

- **Color** – to set certain color to a selected pipeline section;
- **Assign to Selected** or **Assign to All** – to add either selected pipeline elements or all elements into the selected pipeline section;
- **Delete** – is removing previously set properties of a selected pipeline section (there remain the pipeline elements of a construction themselves);
- **Modify…** previously set properties of a pipeline section opens dialog window *Pipeline elements property* (Fig. 2. 138) for the selected section.

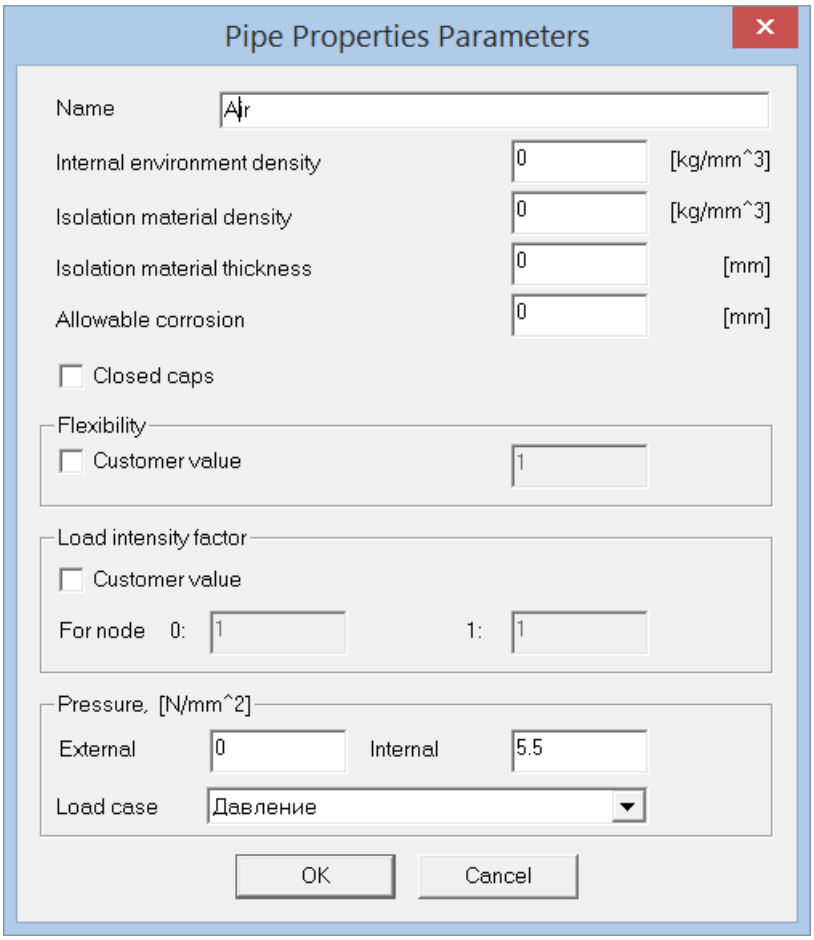


Fig. 2.129 Pipe Properties Parameters dialog.

- **"New" –** opens *Pipe Properties Parameters* dialog window for the selected section.
  - **Name** – specifies the section name;
  - **Internal environment density** – determines the density of internal environment, filling a pipeline;
  - **Isolation material density** and **thickness** – parameters of a pipe insulation material are set if it exists;
  - **Acceptable corrosion** – acceptable corrosion of a pipeline wall is set (its thinning);
  - **Closed caps** – defines if the ends are opened or closed
  - **Flexibility** is a value affecting a moment of pipe bending inertia. By default it is equal to 1, but the user can define it himself, above 1 by number.
  - **The factor of load intensity** shows how many times the stress at the end of a rectilinear area of a pipeline will be more than stress in its middle from external/internal pressure. The factor is specified for two nodes – the beginning of a segment (0) – and its end (1). By default these factors are equal in a one, but the user can specify his own values.
  - **External** and **internal** pressure on pipeline elements, respectively outside and from inside;
  - **Load case** – defines the load case for the internal/external pressure.

Shortcut: 

With help of the button  *"Color Properties of* the Pipeline Elements" can be switched on/switched off pipeline sections disabling – color assigned to these sections.

Thickness to Selected Plates

The command allows you to set thickness for selected plates. This command calls a dialog box shown below.

Shortcut:

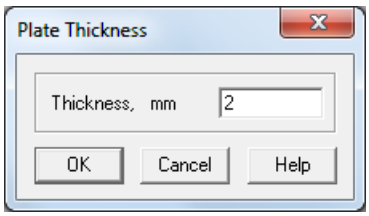


Fig. 2.130 Plane Thickness dialog box

Show Plate Thickness command on *Extra view filters* toolbar enables / disables mode for viewing plate thickness map.

Thickness to All Plates

The command allows you to set thickness for all plates. This command calls a dialog box shown above.

Shortcut:

Enable Plate Stiffness

The command allows user to enable/disable attribute of plate stiffness. Plates without stiffness only transfer loads and do not add the stiffness to the system. The command is applied to the allocated plates and when initiated, it calls a dialog window. To make plates without stiffness, check on *Without stiffness* in the dialog window. To return to plate stiffness switch off flag *Without stiffness.*

Plate Info

The command enables a mode that allows previewing plate info. After you select a plate, a dialog box shown below appears on the screen. *Plates* list in the left contains all construction plates and allows selecting plates for review. You can also use this list to change the name of any plate that is named by default. Double click on the rod name in the list to call a dialog box, which will allow you to enter a new name.

Shortcut:

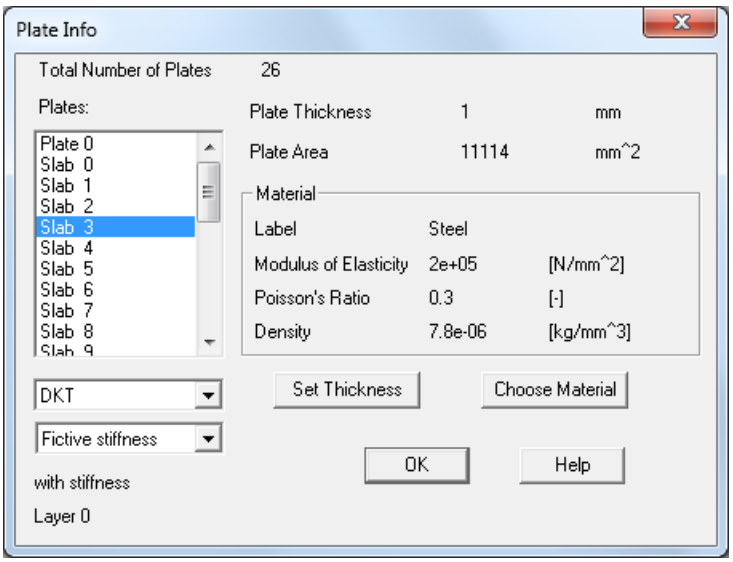


Fig. 2.131 Plate Info dialog box

.

Plates Offset

By using this command, you can set the reference point of the plate relative to its middle plane. The nodes on the plate lie in its middle plane by default. The command calls the dialog box shown in Figure 2.167.

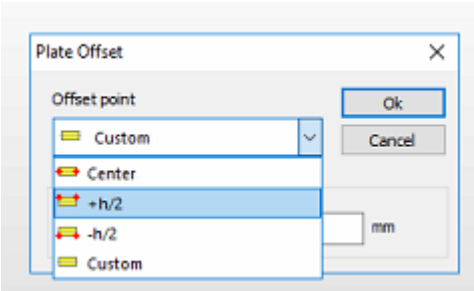


Plate Offset dialog box

In the Plate Offset list select the desired section position relative to its median plane. If the *User-defined* option is selected, the offset of the reference point relative to the median plane can be set manually in the *Offset by LCS*.

Note that the appearance of additional force factors in the form of moments resulting from the displacement of the plate offset will be taken into account when carrying out static and other types of calculation.

Shortcut: 

Nonsolid Plate

The command allows you to set / remove the solidity attribute of the plate. Nonsolid plates only transfer loads and do not add their solidity to the system. Plates without solidity are not split and nods of nonsolid plates should occur either on nods of solid plates, or on nods of cores, or on support.

The command is applied to the selected plates and after its call, a dialog box appears on the screen. To make the nonsolid plates, turn on the *Non Stiffness* mark in the dialog box. To return the plate solidity, turn off the mark *Non Stiffness*.

Shortcut: 

Invert Plate Local CS

The command changes the direction of selected plate's coordinate system normal to opposite.

Shortcut: 

Orientate Local CS of All Plates

The command invokes dialog box for changing plates LCS orientation. Group of plates LCS orientation can be helpful for viewing of load values, stress states etc.

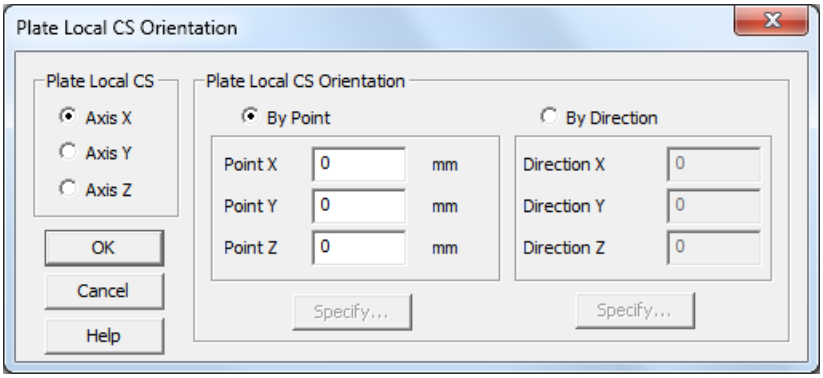


Fig. 2.132 Plate Local CS Orientation

In the left part of dialog box it is necessary to select plate local axis, which direction you wish to change.

In the right part of dialog box the direction of the selected axis can be set by point coordinates (it is possible also to specify point by mouse click using snap). Thus the LCS axis of each plate will be directed as projection of LCS origin-specified point vector. Since X and Y axes are always lying in a plate plane, Z axis will be directed normally to that half-space where the specified point are located, X and Y axes will be directed in accordance with point to plate plane projection.

The second way of plate LCS orientation is use of Global CS. Thus selected axis of plate LCS will be codirected with specified axis of GCS.

✍ *Notes:*

- *Axes of newly created LCS remain right orthogonal.*
- *X and Y axes are always located in a plate plane.*

Orientate Local CS of Selected Plates

The command is similar to the previous command, but applicable to selected plates.

Plate LCS by Default

The command sets default local coordinate system for all plates.

Area of Selected Elements

The command allows user to see area of allocated plates.

Plate Element Type

The command allows to set and edit plate type (DKT or MITC) and considers torsion stiffness (Fictive stiffness or Allman stiffness).

DKT (Discrete Kirchhoff Theory) – thin shell element, which is described by the theory based on Kirchhoff hypothesis: the section of an element remains plane in the deformed state, and a thickness is not less its maximum linear dimension than ten times. DKT shells are used by default in *APM Structure3D*.

MITC (Mixed Interpolation of Tensorial Components) – thick shell element, which thickness is not less than 1/5 their maximum linear dimension. For these finite elements not only shearing forces, but also internal lateral forces are considered.

Allman stiffness accounting should be selected when it is necessary to consider torsion moment in plate plane. In other cases use the *fictive stiffness* accepted by default.

Fig. 2.133 Type of Plate Elements dialog box

Solid Info

The command enables a mode that helps you review information about the solid element. After you select an element, a dialog box appears on the screen. *Solid Elements* list in the left contains all construction solid elements and allows selecting them for review. You can also use this list to change the name of any element. Double click on the element name in list calls a dialog box, which allows you to enter a new name.

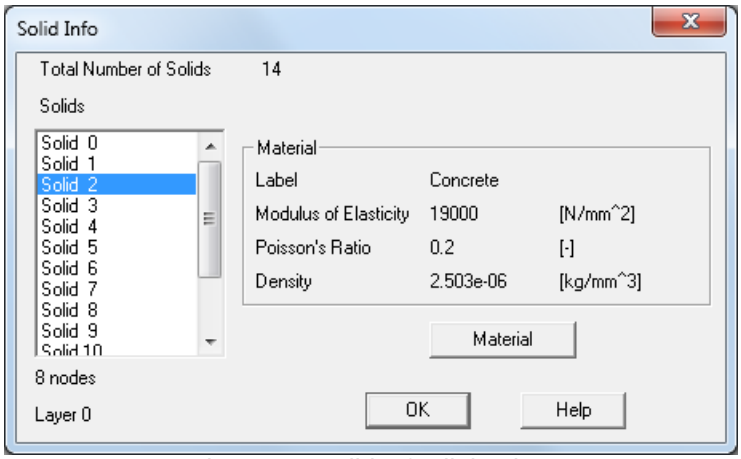


Fig. 2.134 Solid Info dialog box

Volume of Selected Elements

The command allows user to see volume of selected solid elements.

Orientate LCS of All Solid Elements

This command allows to orient local coordinate systems of all solid elements (if no solid selected) or selected solids (if some solids were selected before the command). It can be used for structural analysis of an orthotropic/anisotropic material, or displaying stress in given direction.

After command activation specify the first node (Local CS origin), the second node defines X axis direction, third node - Y axis and Z axis automatically completes the right-hand system.

Orientate LCS of Selected Solid Elements

The command is similar to the previous command, but applicable to selected solids.

Contact Elements Info

The command enables a mode, allowing to see information and to change properties of contact areas and elements. For this purpose, it is necessary to click the left mouse button on contact/target element, which will call the dialog box on the screen.

There is a list of existing contact areas in model in the left upper part of the dialog box. In the right upper part of dialog box, there is information about contact and target elements in the current area, and also buttons that allows you to invert coordinate systems. For correct performance of calculation algorithm, it is necessary that axis Z of local coordinate system of contact elements should be directed towards target elements, and axis Z of local coordinate system of target elements be directed towards contact elements. It is more preferable to have contact elements on more massive, less movable detail with a rougher grid. **Interchange** button interchanges the position of contact and target elements. In the lower part of dialog box, there are parameters of a current contact area that can be changed by pressing **Change properties** button.

Using **Delete** button you can delete the selected contact area from the contact area list.

In the lower part of dialog there are parameters of the current contact zone which can be changed by pressing **Apply** and **Set for all zones** button. Pressing of the button Ok allows to accept changes of parameters of the chosen contact zone and to close the "Contact Elements Information" dialog box.

Normal stiffness and *Tangent stiffness* are characteristics of the fictitious elements connecting details in contact. It is preferable to choose rigidity equal to that of a surface layer of contacting parts if the clearance between details is absent, and several orders less if the clearance is present.

Radius - parameter used for initial contact area determination. If the distance between a contact and a target element from one area is less than a given parameter, it is supposed that this pair of elements participates in contact at the initial stage.

Maximum allowable penetration - the accuracy parameter that specifies maximum allowable penetration of one detail into another.

Extra stiffness - the parameter used at calculation of efforts in contact area. It is preferable to choose stiffness equal to stiffness of a surface layer of contacting details.

Search factor is considered for correction of convergence process (if derivative is used).

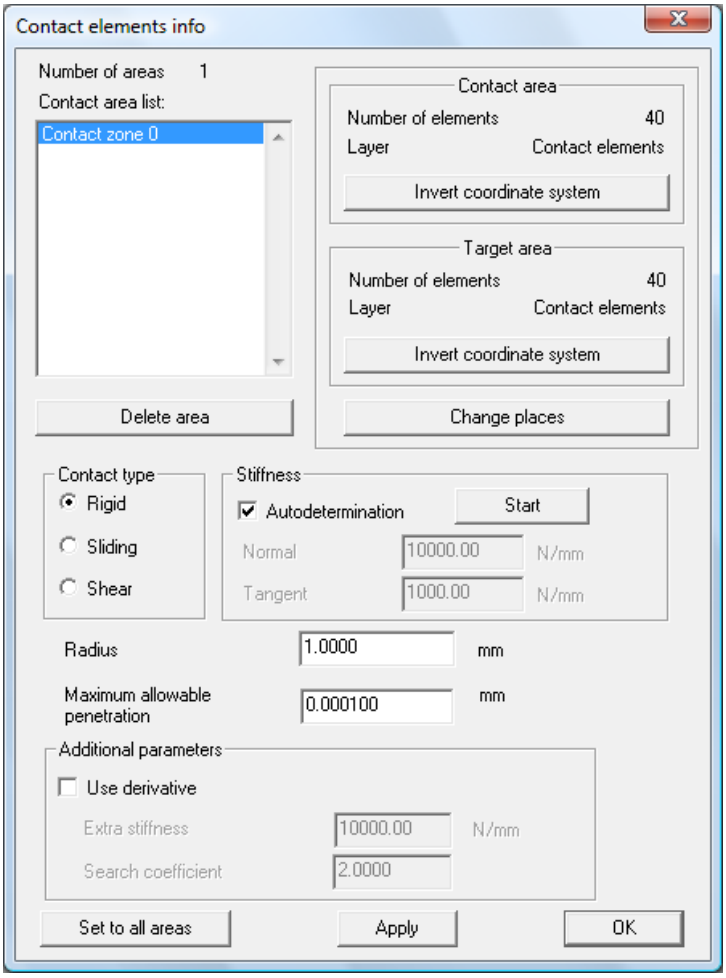


Fig. 2.135 Contact Elements Info dialog box

Soils Info

The command invokes *Soil List* dialog box for creation, editing and deleting of soils. Dialog contains soil list and image of the selected soil.

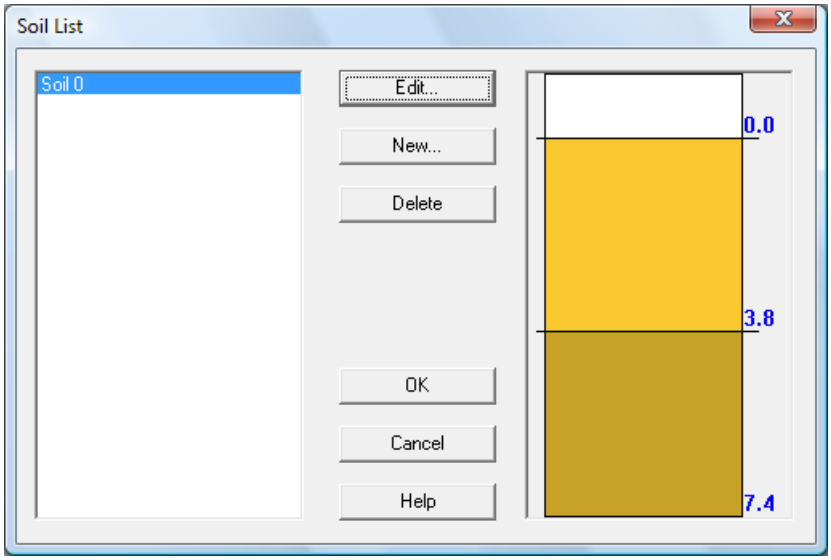


Fig. 2.136 Soil List dialog box

Edit button invokes *Soil Layers* dialog box for selected soil where you can change soil structure.

New button invokes *Soil Name* dialog box for new soil creation. After that it will be accessible in *Foundations* dialog.

Delete button allows to delete selected soil. The ground cannot be deleted to those while it is used in calculation of any foundation.

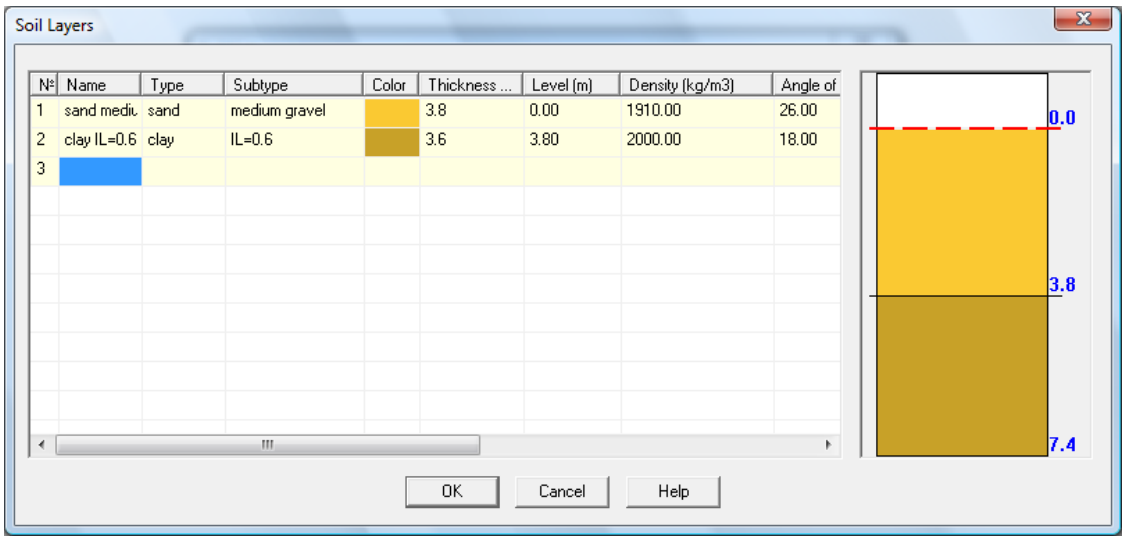


Fig. 2.137 Soil Layers dialog box

The soil list is presented in the left part of dialog. You can set soil structure according to engineering-geological estimation. To set soil layer it is necessary to select its type from the drop-down list. It is possible to use predetermined layer types: sand, clay with known physical characteristics, and also type in soil parameters manually: thickness, density, etc. All edit fields can be changed by mouse double click after selection of predetermined variant.

Materials

The command calls a dialog box shown below for materials management. This dialog contains the materials that are directly used in the model or suggested for use.

Shortcut: 

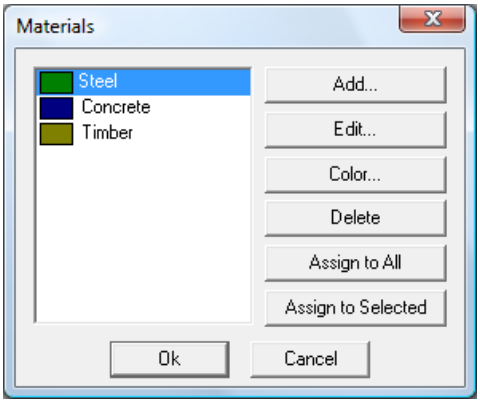


Fig. 2.138 Materials dialog box

Add button allows to add new material to the dialog list. In the appeared dialog box you can select the *basic material* type (*steel*, *concrete*, *masonry*) with predefined characteristic properties.

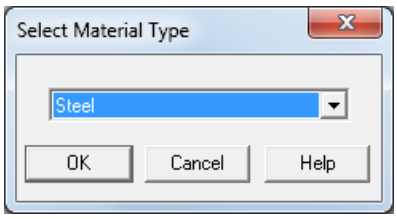


Fig. 2.139 Basic Material Types dialog box

Edit button allows you to change the properties of already existing material. It calls the *Groups of Material Properties* dialog.

Assign to All assigns current material to all elements.

Assign to Selected assigns current material to selected elements.

Color button allows to select color for current material.

Delete button deletes current material. If the deleted material is used in model, system will display warning.

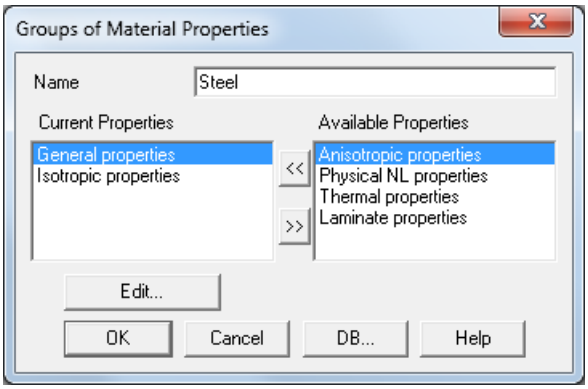


Fig. 2.140 Groups of Material Properties dialog box

After selecting the type of *basic material* in the dialog box, you can specify the name of the material and define a set of its characteristic properties, which are separated in different groups by the various properties.

Groups of material *properties*:
- *General*,
- *Isotropic*,
- *Anisotropic*,
- Физ. нелин. (т. течения изо),
- Физ. нелин. (т. течения Д. – П.)
- SMA,
- Термический материал.
- Течение,
- Удельная электрическая проводимость,
- Относительная диэлектрическая проницаемость,
- Относительная магнитная проницаемость,
- Слоистый композит,
- Демпфирование,
- Усталость,
- Свойство д/мех разр.
- Phys. Nelly
- Phys. Nelly.
- SMA,
- Thermal material.
- Current,
- Specific electric conductivity,
- Relative electric permittivity,
- Relative electric permeability,
- Laminated composite,
- Damping,
- Fatigue,
- Property

For example, such general mechanical properties of the material as the *Yield Point* and *Ultimate Strength* are presented in *General Properties* group; physical properties such as *Modulus of Elasticity*, *Poisson's Ratio*, etc. are presented in *Isotropic Properties* group.

<< button allows to add groups of properties from *available* list to *current*.

>> button allows to remove groups of properties from *current* list.

Edit button allows to edit properties of selected group in *Current Properties* list. For example, dialog box as shown below appears when editing the material properties from the group of *Isotropic Properties*.

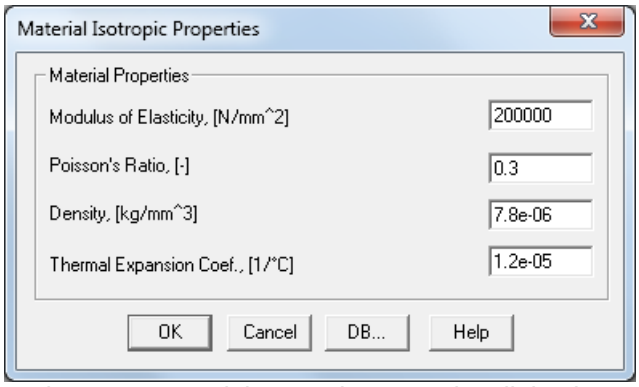
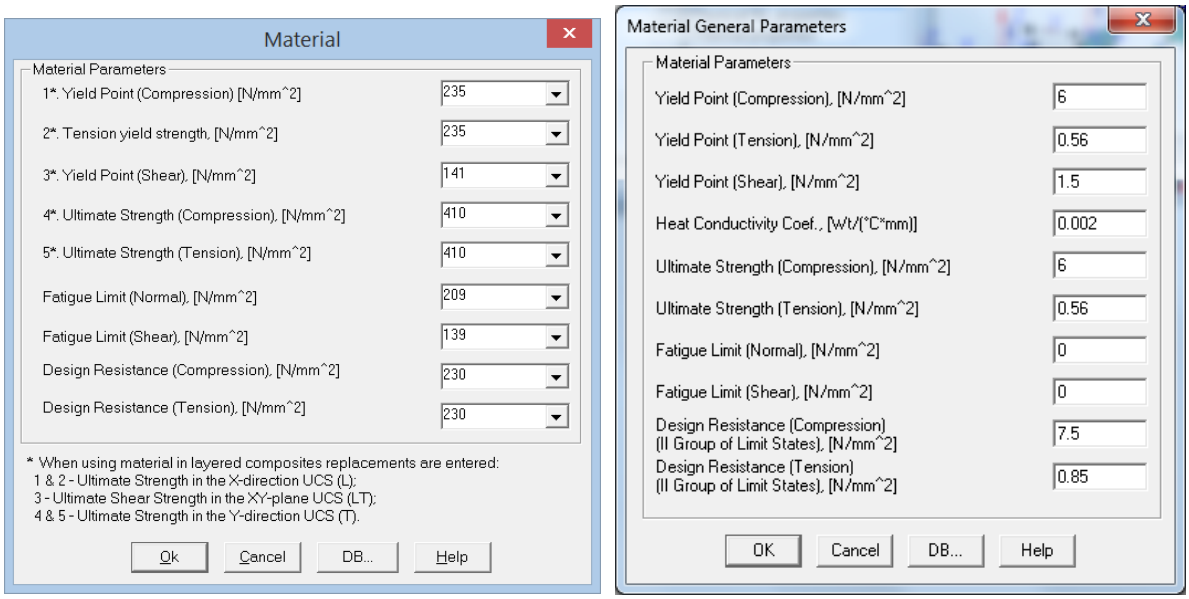


Fig. 2.141 Material Isotropic Properties dialog box

But dialog box content may depend on *basic material type* as shown below when editing the material properties from the group of *General Properties*.



a) for basic type *Steel* *b)* for basic type *Concrete*

Fig. 2.142 Material General Parameters dialog box

DB button allows you to select the material properties from the database. Thus if the button was pressed in *Groups of Material Properties* dialog box, when selecting material, all the properties in the current groups of properties will be replaced with the properties that characterized material from the database. But when you press the same buttons in dialog boxes shown above and select material from database, only those properties will be edited that are available in these windows.

By clicking on a popup menu in the Material dialog box (Fig. 2.161), you can set the dependence of a corresponding physical characteristics of the material on the temperature. The dependence can be set as a graph, a table or a function.

When selecting dependence in a form of a graph, the functions editor opens (Fig. 2.163), which helps to set a corresponding dependence function graph.

When selecting dependence in a form of a table a dialog box opens (Fig. 2.165), which helps to load an existing table or create a new one with a specified number of lines.

When selecting dependence in a form of a function the expression editor opens which defines an analysis function.

The Graph/Table/Expressions Editor is described in Chapter 8.

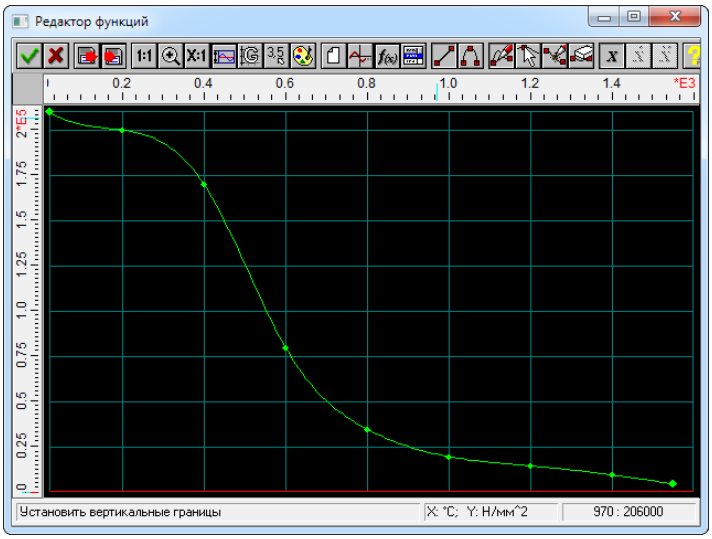


Fig. 2.143 Function editor

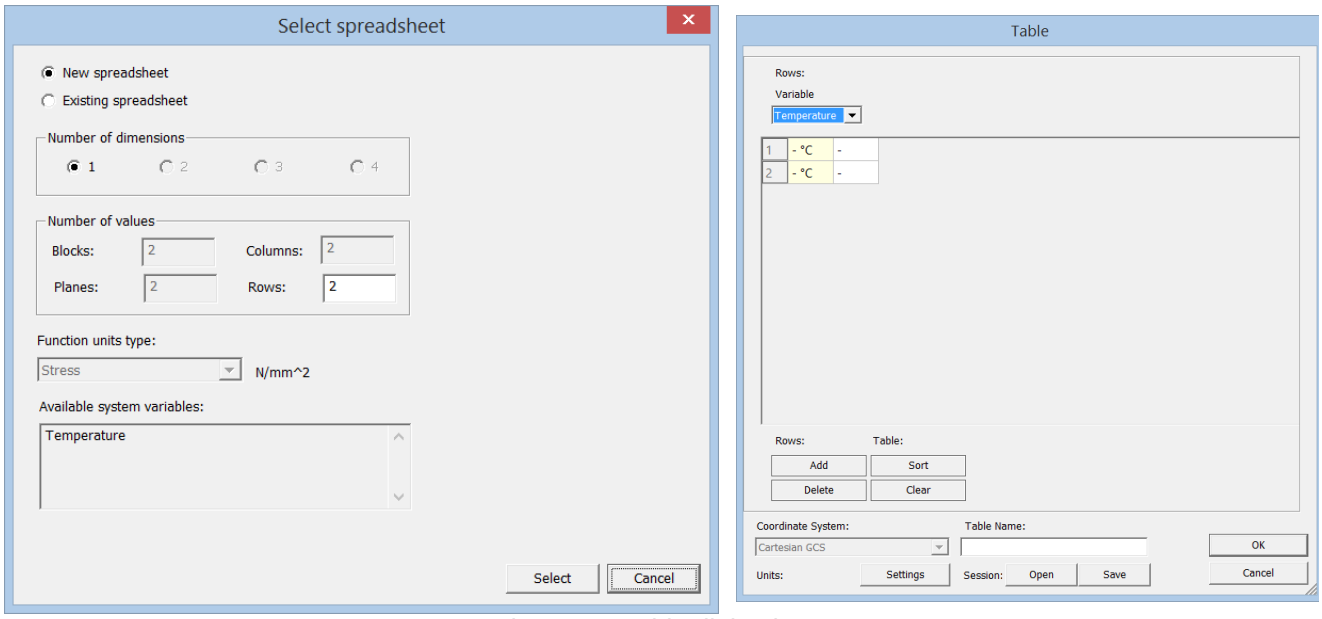


Fig. 2.144 Table dialog box

Anisotropic Properties

Anisotropic material can be set for plate and solid finite elements. Accounting of anisotropic properties for materials is carried out for all calculation types but with condition that local coordinate system of model finite elements must be codirectional.

Modulus of Elasticity, *Poisson's Ratio* and *Shear Modulus* can be set in local coordinate system of finite elements for *orthotropic* materials.

The *stiffness* matrix is set in local coordinate system of FE for *anisotropic* materials. The *flexibility* matrix can be set alternatively.

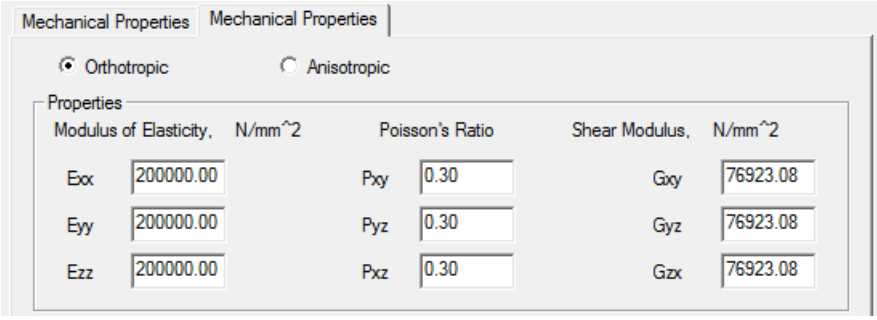


Fig. 2.145 Additional tab for orthotropic properties

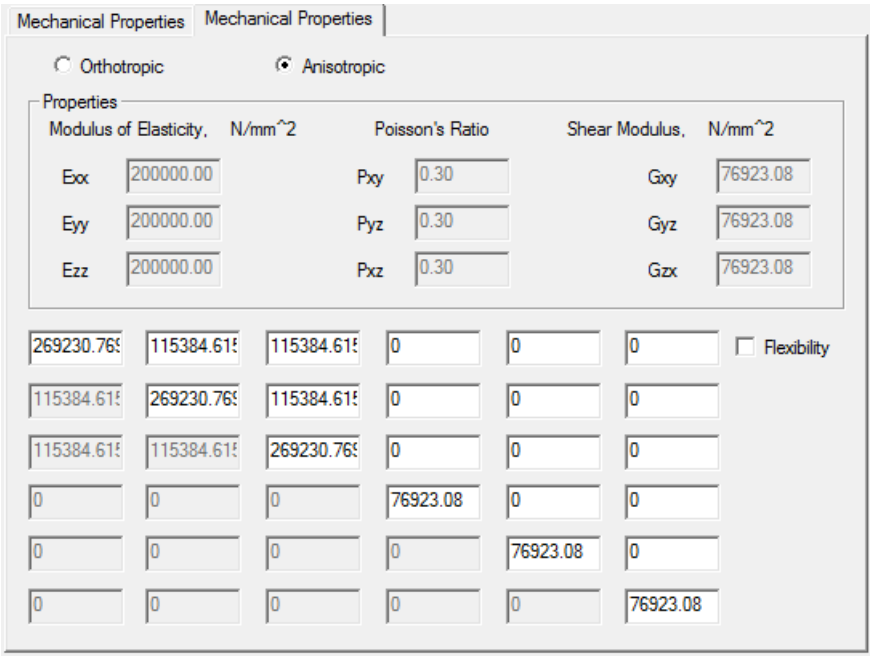


Fig. 2.146 Additional tab for anisotropic properties

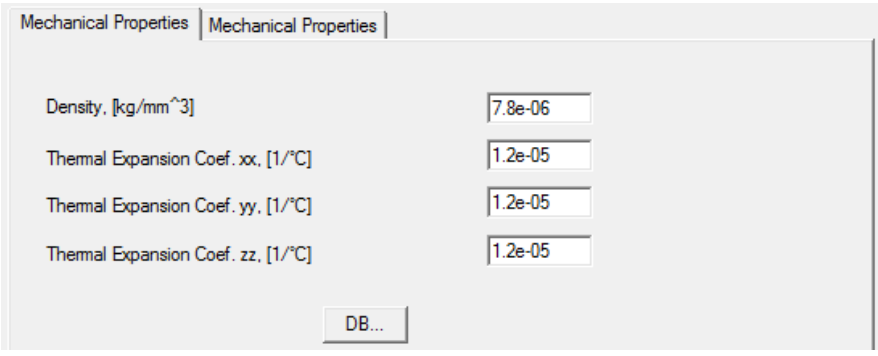


Fig. 2.147 Additional tab for anisotropic properties

Physical Nonlinear Properties

Addition of this group of properties to *current* invokes dialog window for stress-strain curve definition for nonlinear materials.

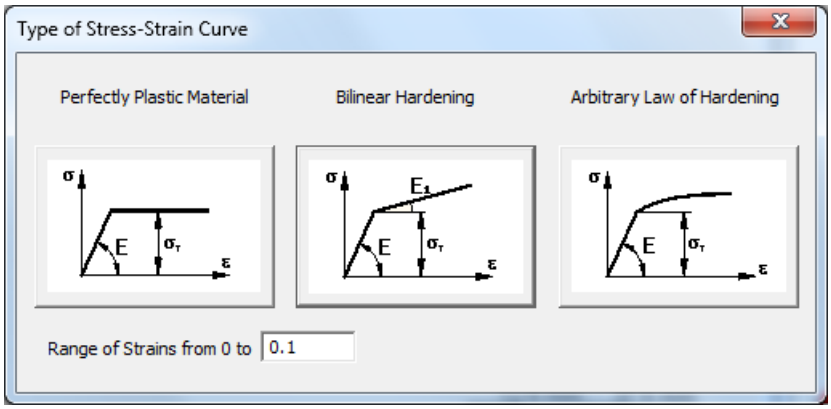


Fig. 2.148 Type of Stress-Strain Curve dialog box

Function editor is launched to define material nonlinearity graph after selection of nonlinearity type: *Perfectly Plastic* or *Arbitrary Law*.

✍ *Note**!** The plasticity account is possible for plate and solid elements at performance of physically nonlinear analysis.*

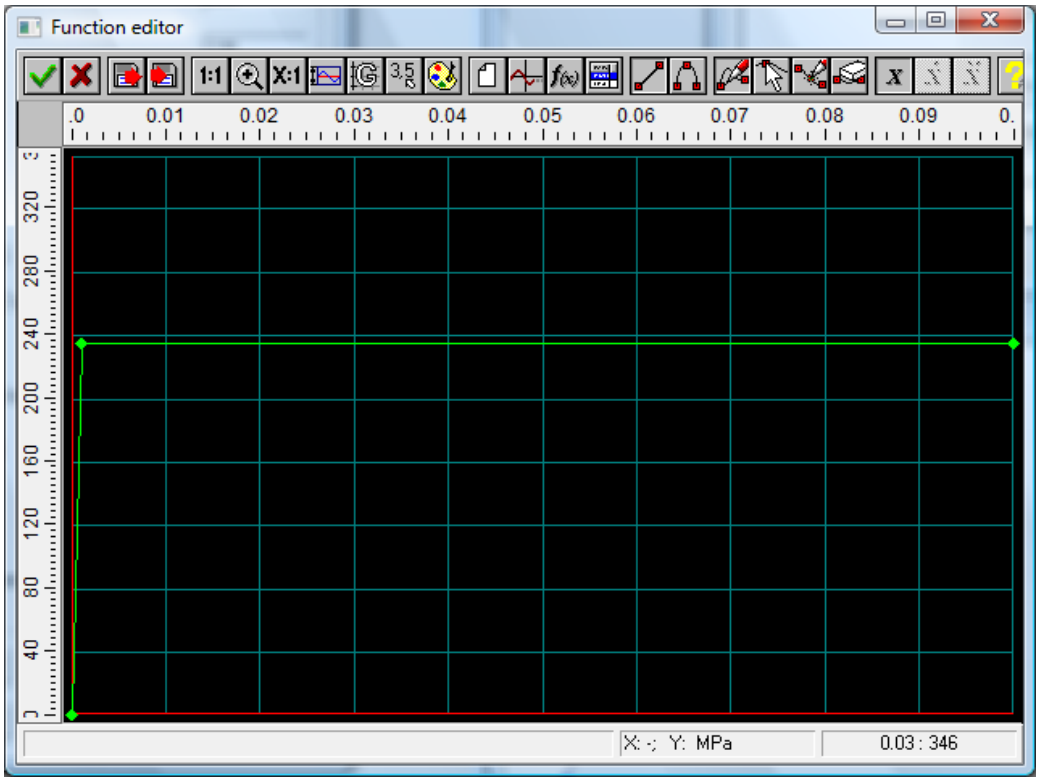


Fig. 2.149 Function editor window

Shear Modulus of Elasticity dialog box appears on the screen when select *Bilinear Hardening*.

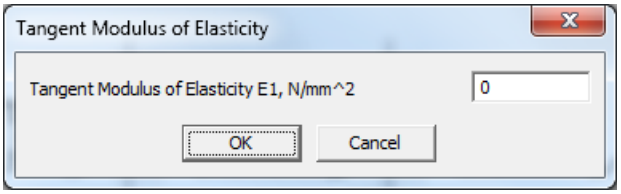


Fig. 2.150 Tangent Modulus of Elasticity dialog box

Thermal Properties

It is necessary to set material thermal properties for structure elements to perform steady-state or transient heat transfer analysis.

Thermal Properties of material can be set as constant values, graphs, tables or functions in *Thermal Properties* dialog box. This dialog contains *Anisotropic Material* option using which you can set *Heat Conduction* on X, Y and Z directions.

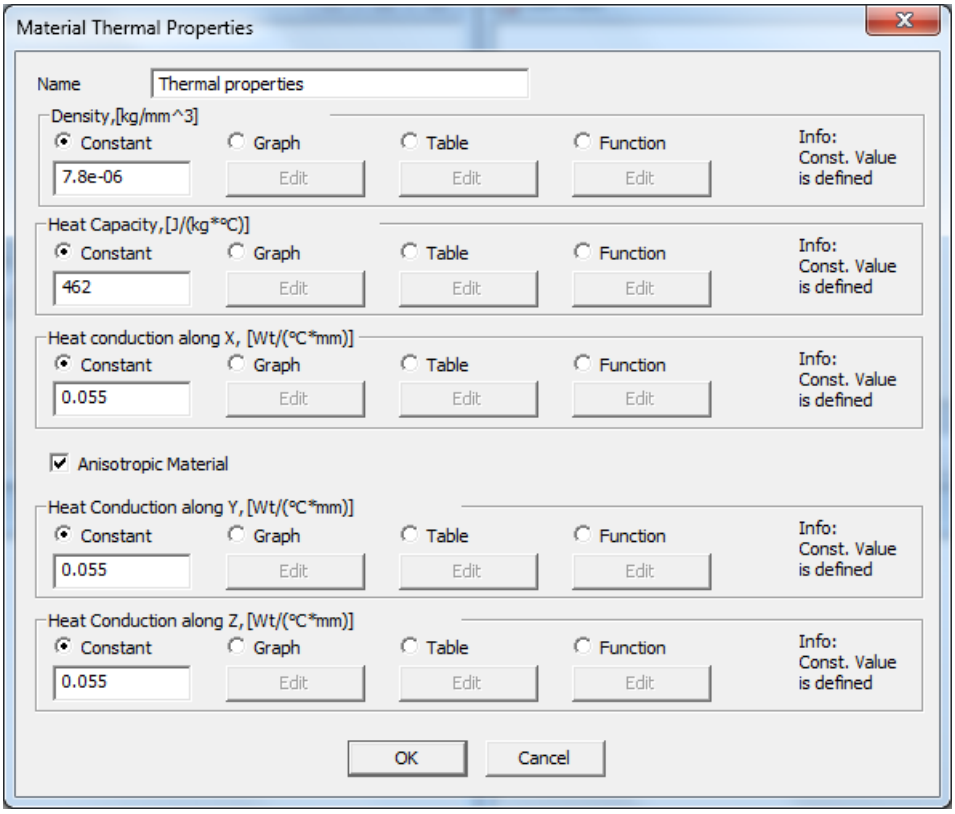


Fig. 2.151 Thermal Properties dialog box

Laminate Properties

This group contains layered composite (laminate) properties. Laminate is a plate with a thickness which is small compared with other two dimensions. The plane of the laminate is usually XY, Z axis is perpendicular to this plane. Laminate consists of thin layers of an orthotropic material, each layer is laid in a plane at predetermined angle.

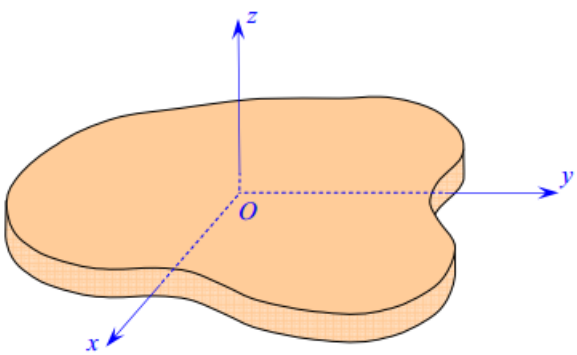


Fig. 2.152 Laminate local coordinate system

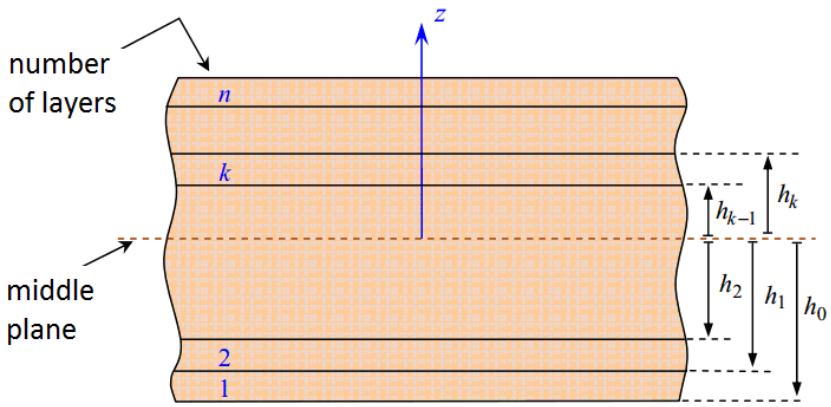


Fig. 2.153 Laminate layers

In the dialog box shown below you can create a laminate with layers each of which can have own characteristics and parameters such as material, thickness and fiber direction.

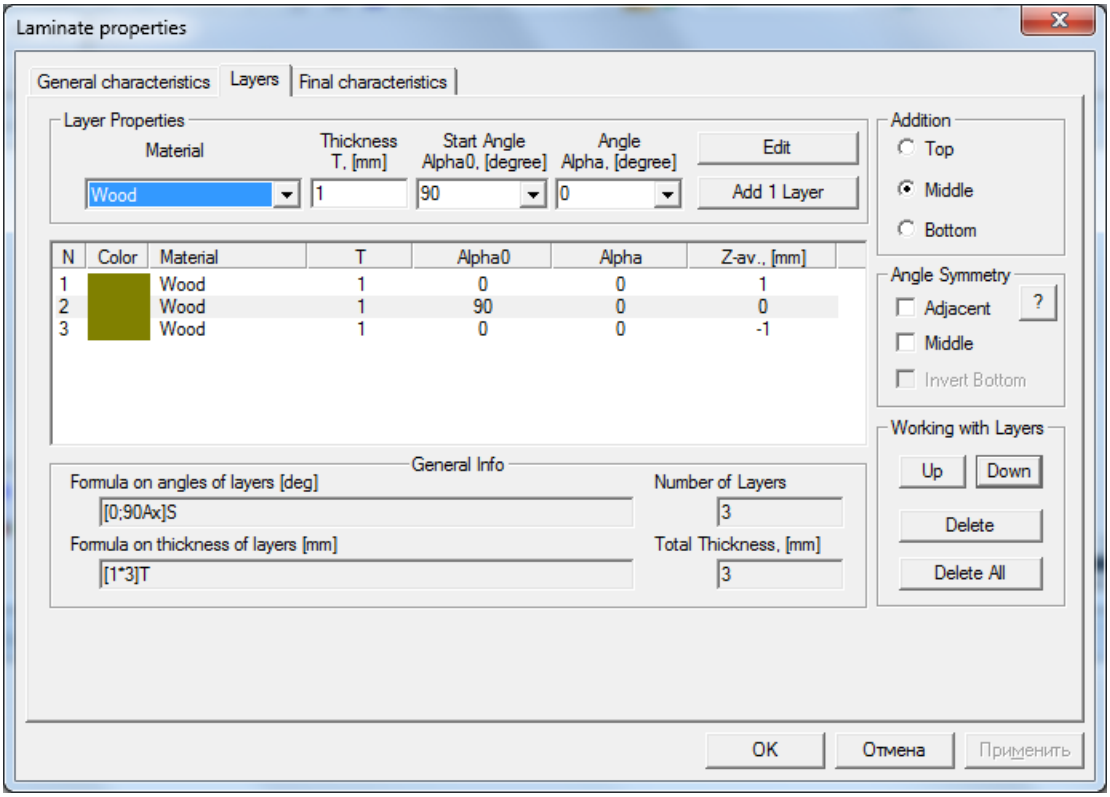


Fig. 2.154 Laminate Properties dialog box (Layers page)

Add button allows to add layer to a set of laminate.
Edit button allows to edit selected layer parameters.
There are various options in the right part of the page that can help in work with layers.

The final characteristics of a laminate are presented in *Final Characteristics* page of the dialog.

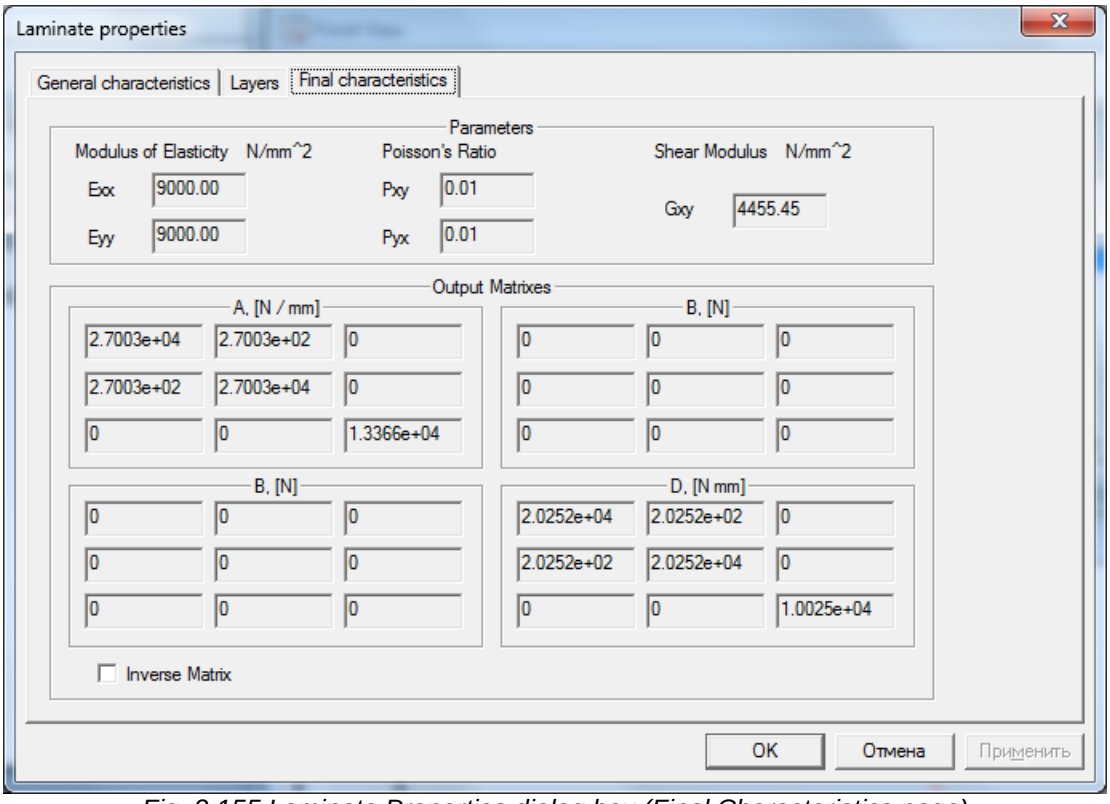


Fig. 2.155 Laminate Properties dialog box (Final Characteristics page)

Basic material type Masonry

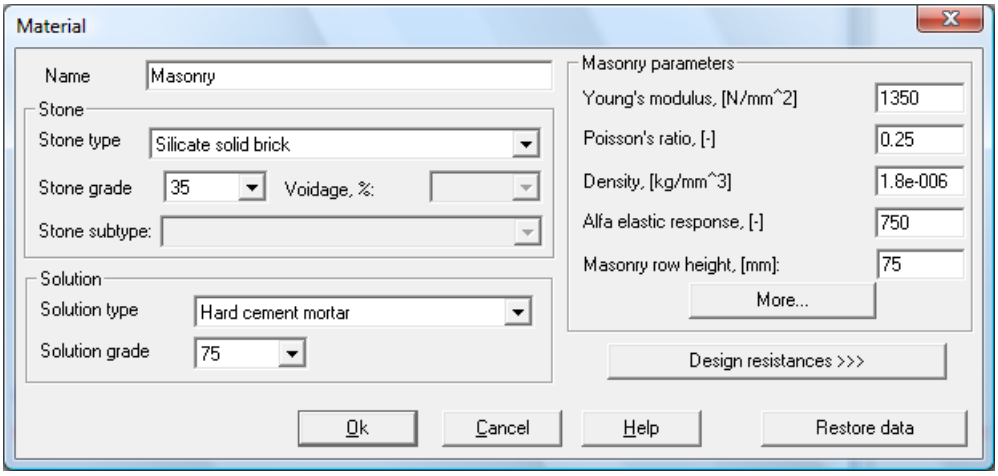


Fig. 2.156 Material Masonry dialog box

Stone type – drop-down list of stone types.

Stone subtype – drop-down list of stone subtypes, for the set of additional properties of separate stone types.

Voidage – drop-down list for the set of stone voidage. Voidage is set in percentage of material volume. There will be a choice between light and heavy stones for natural stones in the list.

Masonry row height edit field is intended for row height setting.

More button invokes dialog box where can be set additional parameters of material.

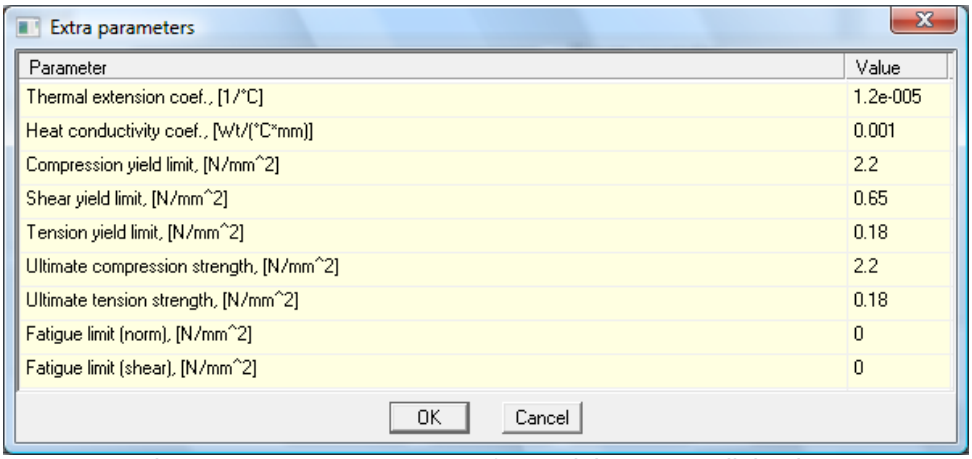


Fig. 2.157 Extra parameters of Material Masonry dialog box

Design resistances button opens additional part of dialog which is used for setting of additional design resistances of masonry.

Restore data button sets material parameters by default.

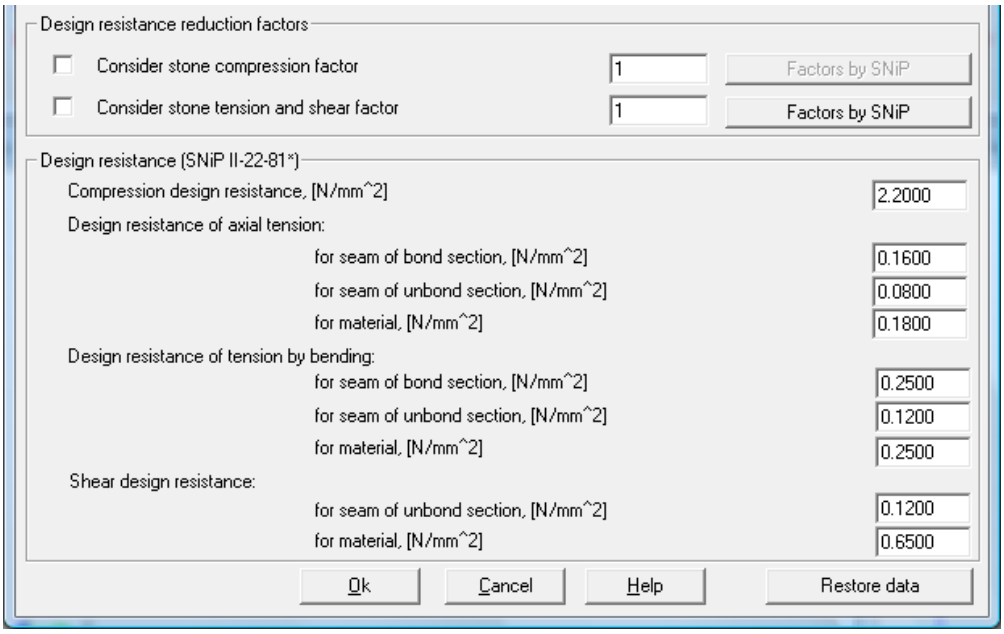


Fig. 2.158 Additional part of Material Masonry dialog box

Values of design resistances are taken from SNiP II-22-81* by default. Those values can be displaying taking into account reduction factors of design resistances. For this purpose it is necessary to mark option opposite to the corresponding factor.

Factors by SNiP button pressing gives information dialog about factors.

Values of design resistances are highlighted with different colors: black - value by default, dark blue - the user value, red - incorrect value.

Selected Elements Moment of Inertia

The command calls a dialog window with the information on weight, coordinates of the center of mass and the moments of inertia concerning axes of global coordinates of the selected elements.

Model Moment of Inertia

The command calls a dialog window with the information on weight, coordinates of the center of mass and the moments of inertia concerning axes of global coordinates of all model.

Model Dimensions

The command invokes window with the information about model dimensions and the maximum deviations relative to global coordinates.

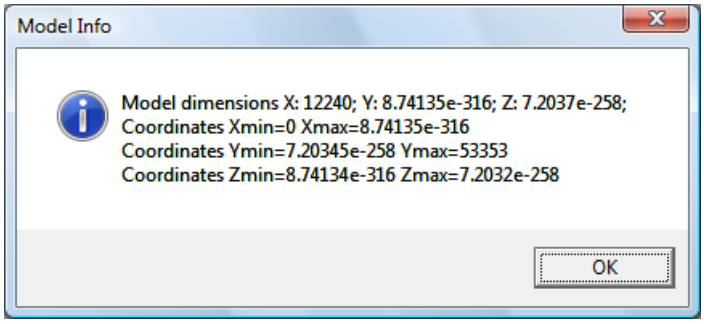


Fig. 2.159 Model Info dialog box

Model Info

The command calls a dialog window with the information about model. The information about number of nodes, elements, etc. is presented in window.

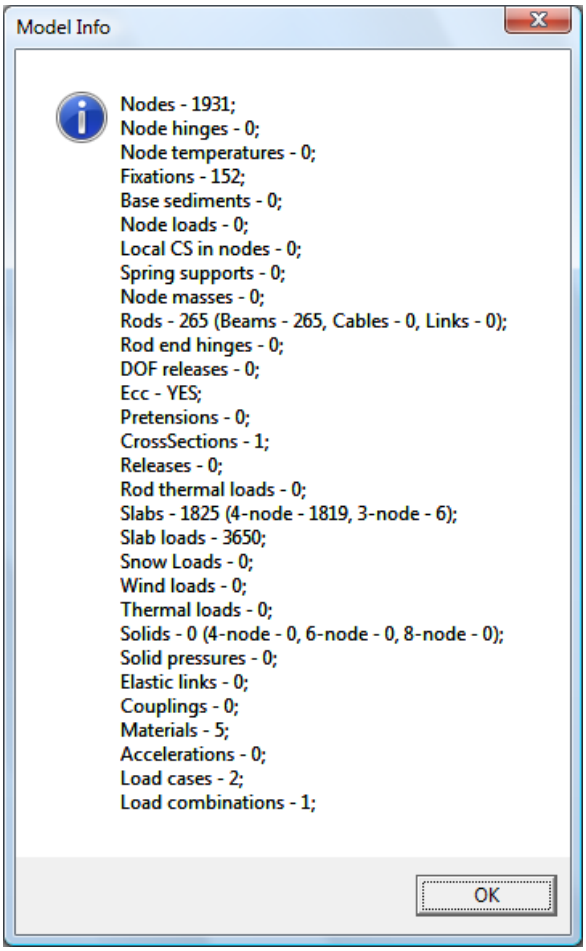


Fig. 2.160 Model Info dialog box

Additional Model Info

The command invokes dialog box with the expanded information about model characterizing hardware requirements for performing calculation. To display information about current model it is necessary to press **Calculate** button.

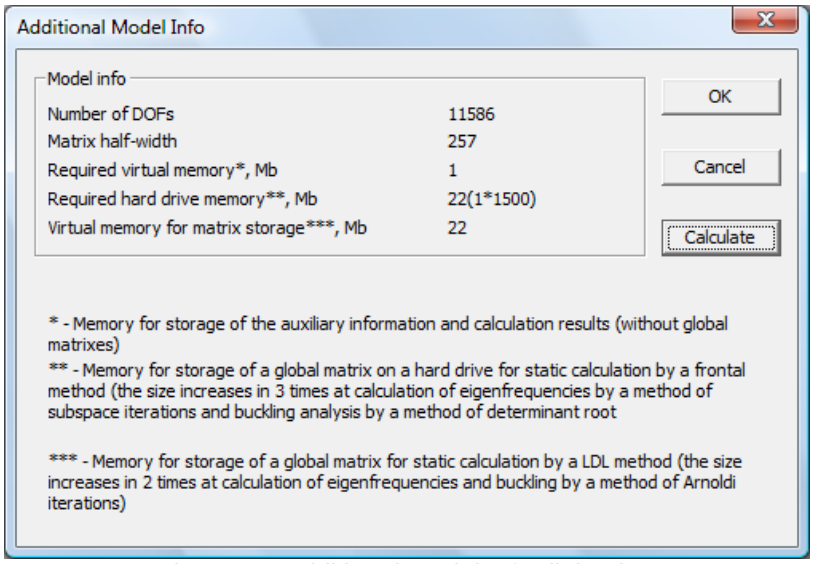


Fig. 2.161 Additional Model Info dialog box

Design menu

Design Element Type

These commands allow to select type of design elements: steel, reinforced (concrete and masonry) or wood. The detailed description of this menu commands and work with them is presented in chapter 5.

Design Elements

The command invokes dialog box where it is possible to work with design elements, and also look through calculation results. External interface of a dialog depends of selected design element type.

Shortcut: 

Selected Objects to Design Element

The command places the selected elements in ONE RC design element. Warning appears on the screen if impossible to create design element. The reasons, according to which elements cannot compose a design element, are checked in dialog.

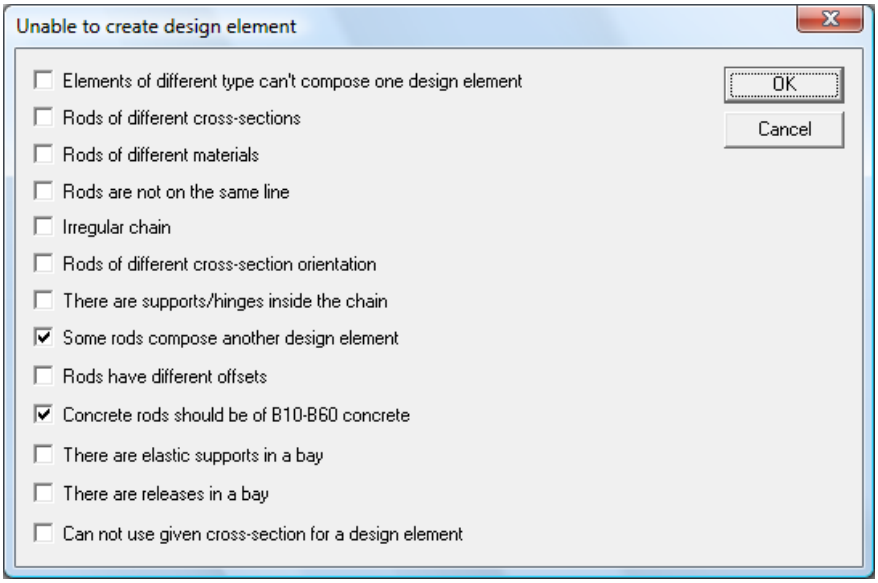


Fig. 2.162 Unable to create rod design element warning

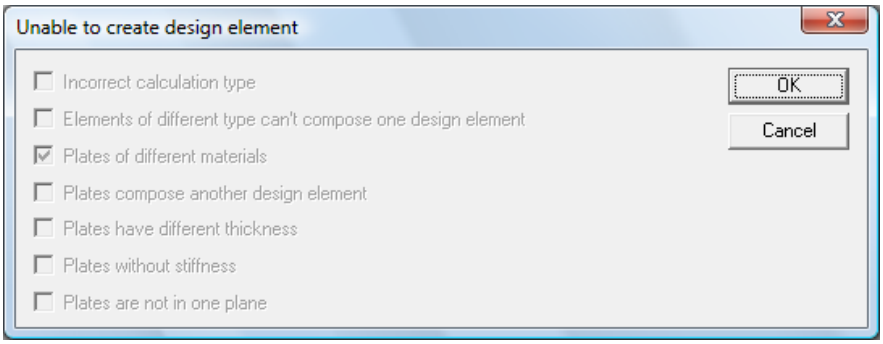


Fig. 2.163 Unable to create plate design element warning

Shortcut:

Selected Objects to Separate Design Elements

The command places each selected element (finite element) to separate RC design element.
Shortcut:

Selected Objects to RM Design Element

The command places the selected elements in ONE RM design element. Warning appears on the screen if impossible to create design element. The reasons, according to which elements cannot compose a design element, are checked in dialog.

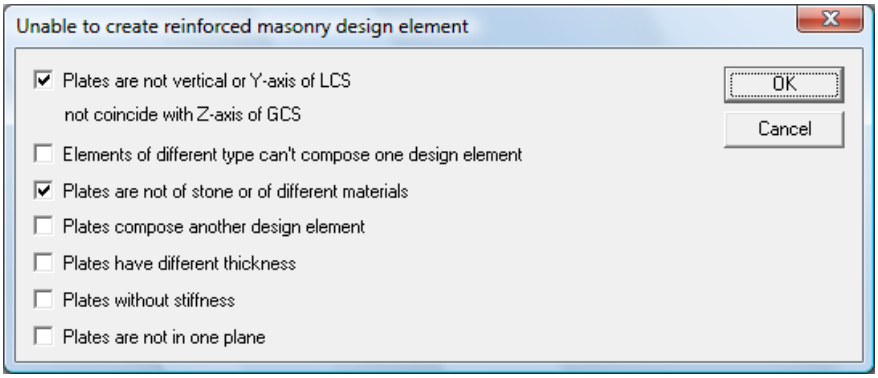


Fig. 2.164 Unable to create plate design element warning

Shortcut:

Selected Objects to Separate RM Design Elements

The command places each selected element (finite element) to separate RM design element.
Shortcut:

Delete from Design Element

The command deletes selected objects from design elements.
Shortcut:

Updating of Punching List

The command updates punching list of design elements – RC shells.
Shortcut:

Connections of Steel Structures

These menu commands allow to create parametrical models of steel joints automatically in *APM Graph*.

Calculation menu

The commands of this menu allow calculation and calculation parameters setting.

Calculation

The command executes a calculation of the construction. After a command call, on the screen appears a dialog window requiring a type of calculation that is performed and its parameters.

The calculation parameters are being duplicated with the command **Calculation | Calculation Parameters**.

See in detail about the calculations and results of them in Chapters 4 and 6.

Static calculation is performed for the selected available load cases and a combination of load cases listed in the *download* field.

If you expand the drop-down list *for loadcase*, then you can use the checkmarks to select those loads and combinations of load cases for which the static calculation is to be performed (Figure 2.204). Clicking the PCM on one of the loads allows to remove / install the checkmarks on all downloads.

Calculations of stability, nonlinear, steady-state and transient heat transfer, forced oscillations, eigen frequencies with preloading are performed only for the selected load case.

Moreover, if, for example, the results of a static calculation are required to calculate stability then only those loads for which a static calculation has been performed will be available, Fig. 2. 205.

The calculation of the contact interaction is performed when performing a nonlinear calculation.

If you select one (or several at the same time) of the calculation item, additional parameters of the selected calculation type are opened. When carrying out a static calculation at the same time as calculating the stationary or non-stationary thermal conductivity in the construction of thermal stresses, it is necessary to set the checkmarks "To take into account the temperature (from stationary or non-stationary heat conductivity") when specifying the static calculation parameters.

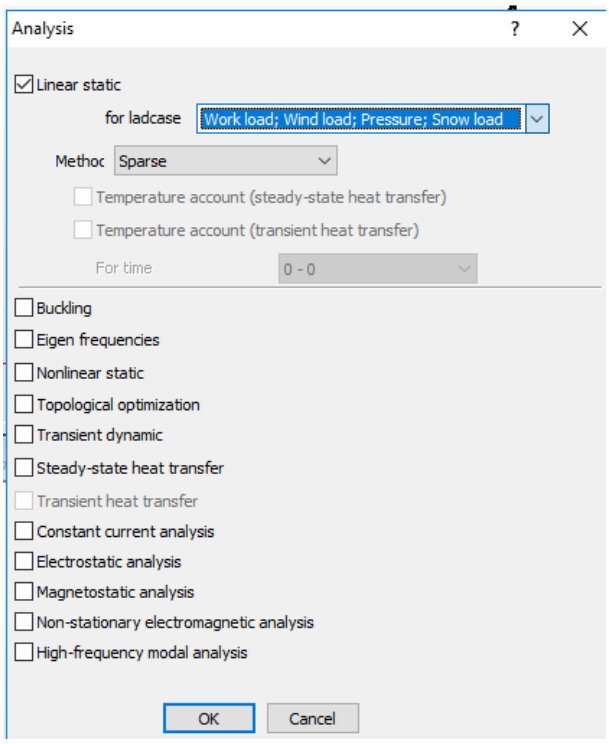


Fig.2.203 Analysis dialog box

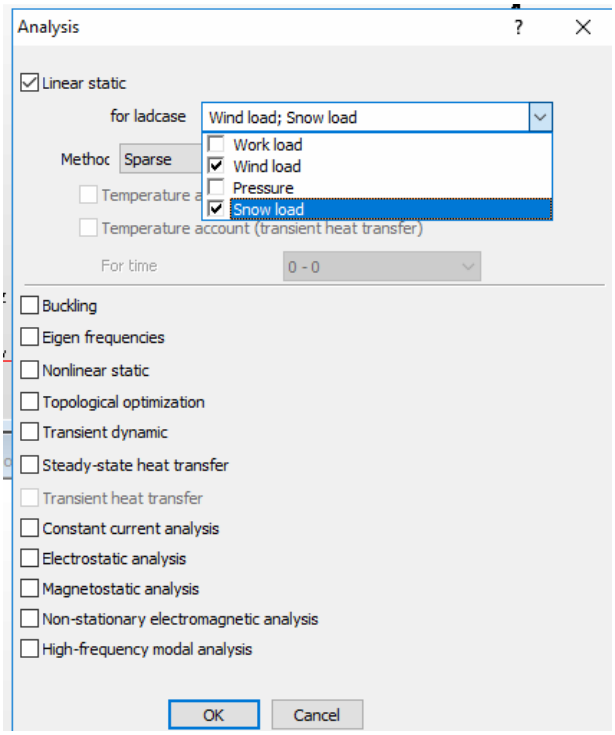


Fig. 2.204 The window Analysis with the expanded list of downloads selected for calculation.

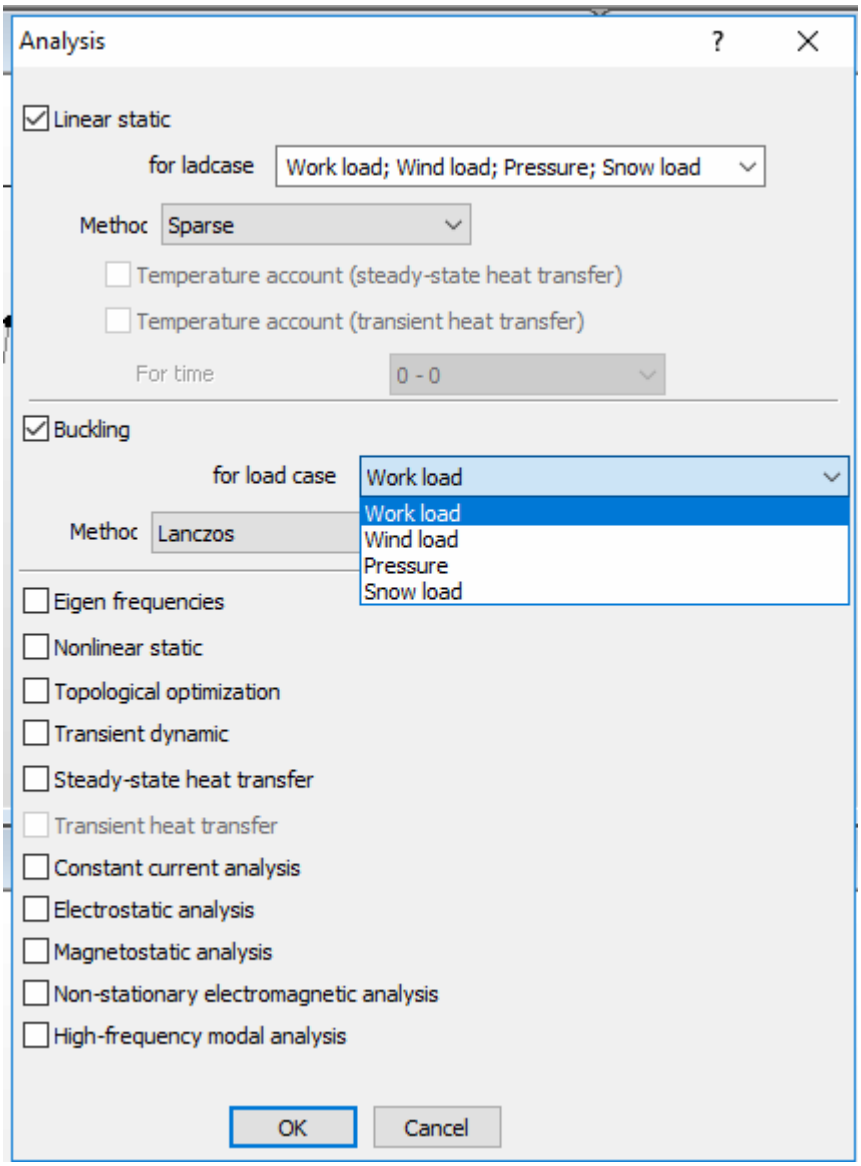


Fig. 2.205 Available load cases for stability calculation

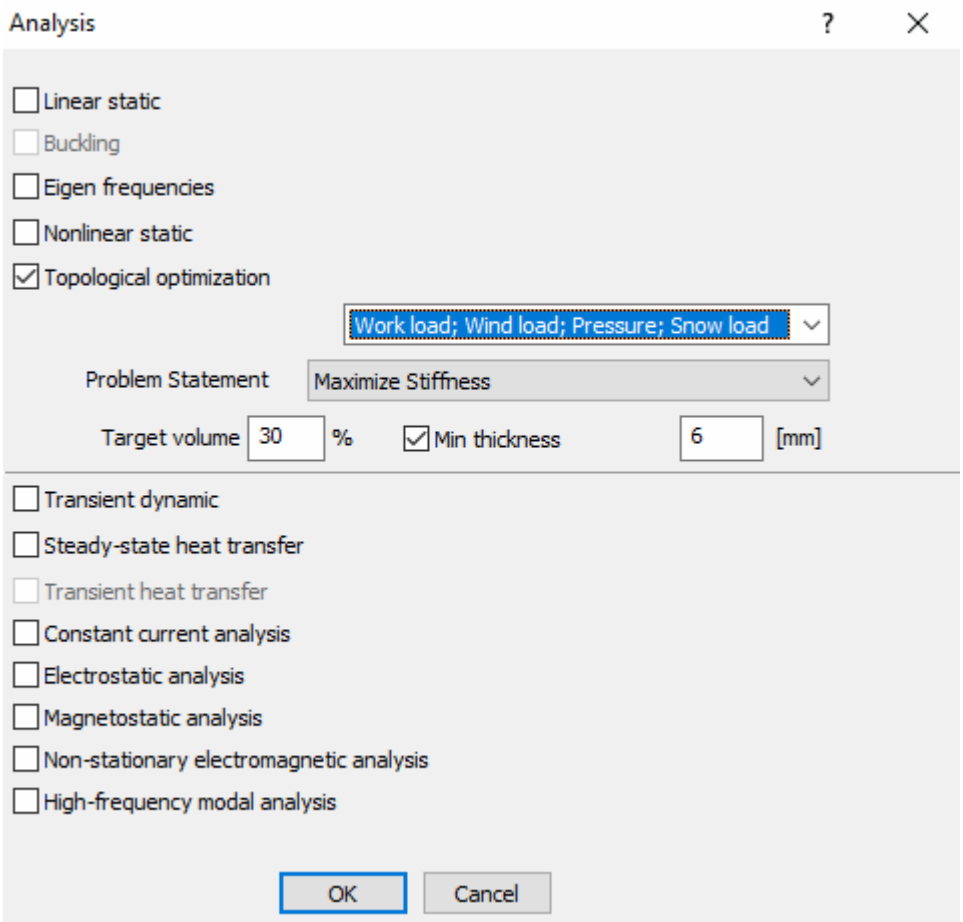


Fig. 2.206 Topological optimization calculation parameters

To calculate Topological optimization, you must select the load case for which the calculation will be performed, the *Volume fraction* is set by a user before the calculation, the *Filter type* is selected from the drop-down list and the calculation time depends on it.

Keep every __ iteration in the calculation process - each n-th iteration will be saved. When writing here zero, no iterations will not be saved.

Min. radius defines the minimum thickness of the edge, wall, etc. It is not allowed to make the walls, edges, thrusts of the optimized model less than that.

If the *Forced Oscillations* option is selected, then, after the checkmark is placed, the corresponding field for selecting dynamic calculation parameters will be opened, which is shown in Fig.2.207.

If at the same time a linear static calculation is performed, then when installing a checkmark in the option *Use N.U. from the load* at the starting point, the results from the selected load for which the static calculation was performed will be entered.

From the drop-down list, the method for calculating forced oscillations can be chosen.

Self-form decomposition or Direct integration.

In the input field *Number of registered own forms* the number of own forms of structural oscillations is indicated for calculation by the method of decomposition of oscillations in its own forms.

In the *Logarithmic decrement* input field, the value of the corresponding decrement is entered, which characterizes the Logarithmic decrement. This parameter is the same for all natural frequencies, because the system is calculated from the assumption that the decrement is the same for all frequencies.

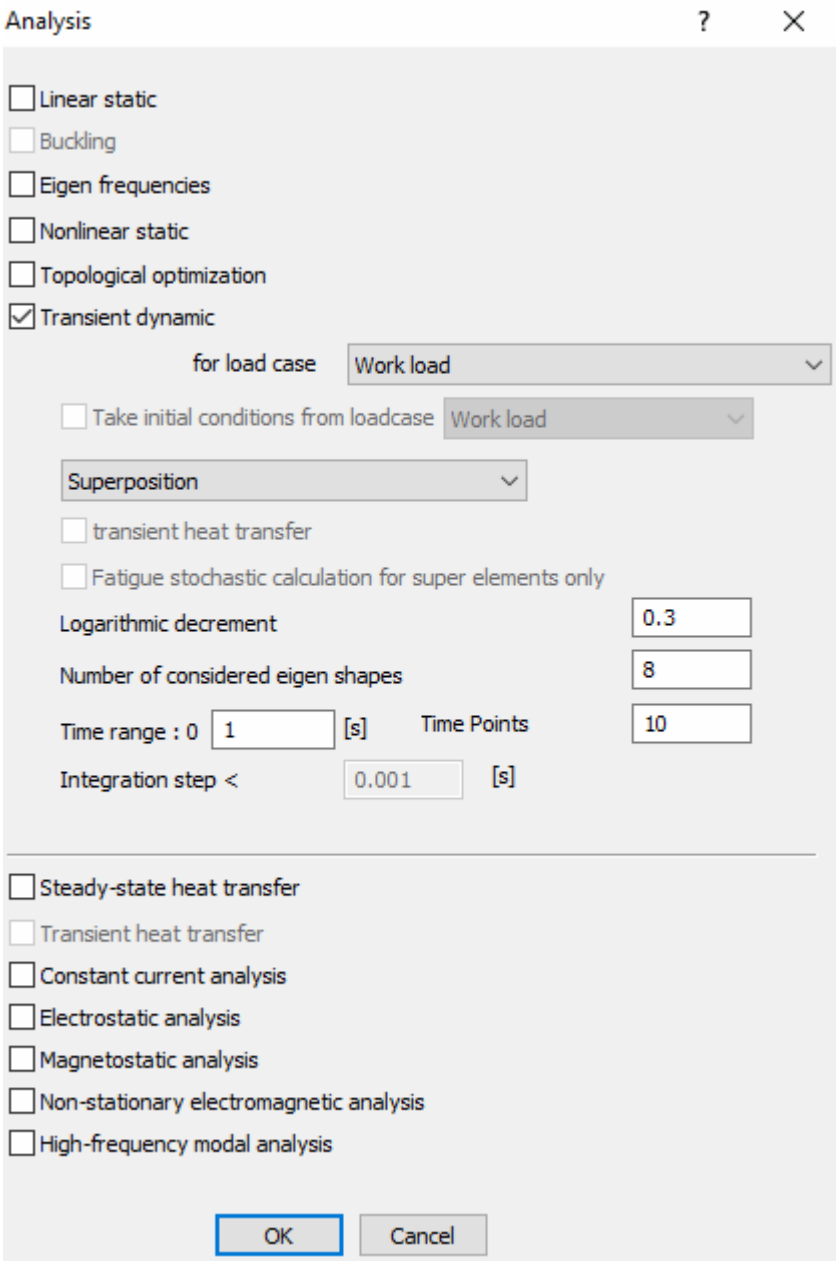


Fig. 2.207 Calculation parameters Forced oscillations

In the *Time interval* the time interval for the calculation is specified.

The number of calculated time points determines at how many points the behavior of the system will be calculated.

When choosing the calculation method the *Direct Integration,* attenuation can be specified through the parameters for elastic links for individual links and the integration step. When the calculation time *Interval* and the number of *Time Points* are set, then the integration step for which the calculation is performed is set. However, if the *Integration step* is set to be less than that obtained through the time interval and the number of intervals, the explicit *Integration step* will be taken for calculation.

A checkmark in the option *Take into account the preload* allows you to calculate own frequencies with the preloading of the structure.

The calculations of direct currents, the electrostatic calculation, and the magnetostatic calculations, as well as the stationary electromagnetic calculation and high frequency modal analysis are carried out for specially prepared models with corresponding loads and boundary conditions three-dimensional 4-node, 6-node and 8-node elements as well.

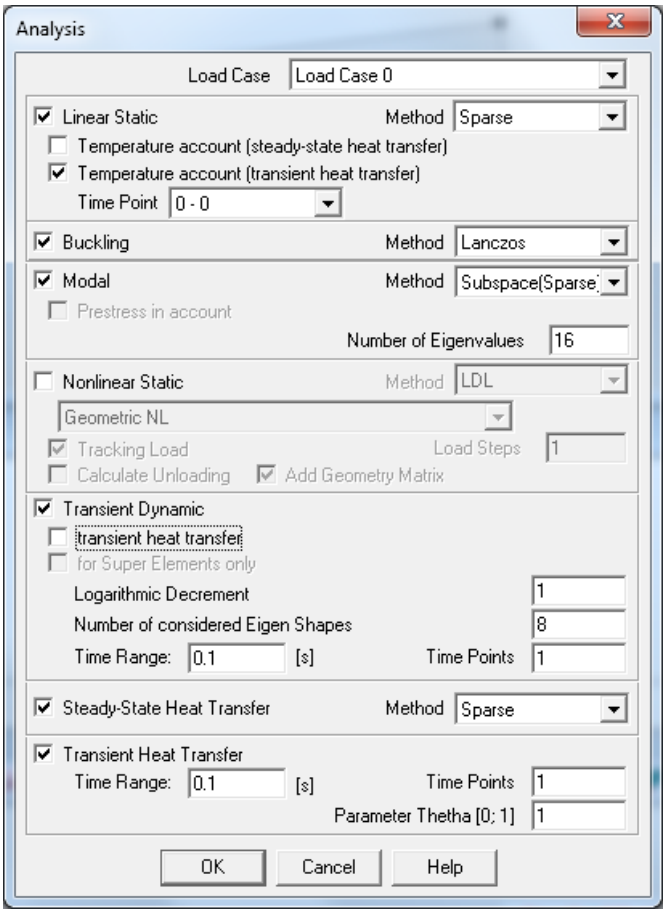


Fig. 2.165 Analysis Types dialog box

Clicking **OK** button calls the following dialog box for selecting parameters of dynamic calculation (if checkbox *Transient Dynamic* is selected):

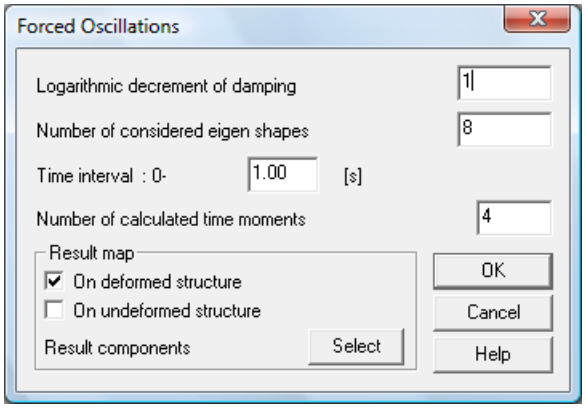


Fig. 2.166 Forced oscillation dialog box

Damping characteristics of the structure is defined by *Logarithmic Decrement of damping*. This decrement is considered to be frequency independent.

You can define *number of considered eigen shapes* in the title field. The calculation uses natural modes decomposition method.

You can define desired time interval in the field *Time interval*, and number of desired time moments in the next input field.

In *Result map* group you can select whether to use a deformed shape of structure or not to show the result parameter map. You can also select parameters to be displayed by pressing **Select** button.

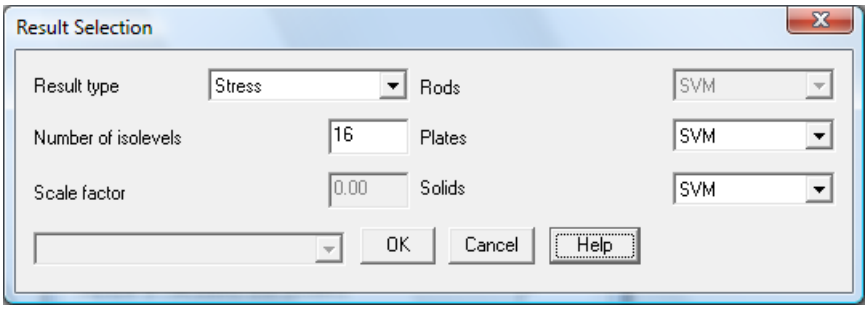


Fig. 2.167 Result Selection dialog box

During the calculation, a dialog box that reflects calculation stage is shown on the screen.

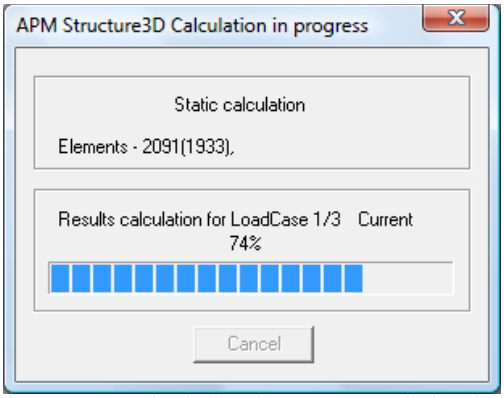


Fig. 2.168 Calculation in progress dialog box

Calculation of Super Elements

This command starts static calculation of super elements.

Fatigue Calculation for Stochastic Load Cases

This command starts fatigue calculation for previously defined stochastic load cases.

Design

The command performs bearing capacity check of design elements. It demands preliminary performance of static calculation since it uses values of stresses and loads. All parameters of calculations and properties of design elements can be set in **Results / Design Elements**. See also Chapter 1 Design elements.

Design of Reinforced Elements

The command starts reinforcing calculation for all design elements. For reinforcing design it is necessary to perform static calculation and load or code combination calculation. Results are presented in *Design Elements* dialog box.

Checking of Reinforced Elements

The command starts reinforcing checking calculation for all design elements. For reinforcing checking it is necessary to perform static calculation and load or code combination calculation. Results are presented in *Design Elements* dialog box.

Code Combinations

the command calls the Calculated Load Combinations dialog box, in which you can set the load case table for calculation of Load Design Combination Calculation (LDC).

Calculation of the most dangerous design combinations of forces (LDC) for rods is based on the extreme values of several groups of quantities, namely normal and tangential stresses at characteristic points of sections, longitudinal and shear forces. The principles of combination selections and their coefficients correspond to those in Clause 1 of SNiP 2.01.07-85 (Clause 6 of SP 20.13330.2011).

The Reliability Factor is selected from the corresponding drop-down list (Figure 2.209). It determines the reliability factor for the entire structure and can be *Increased, Normal, Reduced, Large-span buildings and buildings more than 250 m tall and a User value.*

Limit states group - the first and second.

The first group of limit states is the main one since it determines the loss of the bearing capacity of the structure. Calculations are made for strength and stability within this group.

The second group of limit states is associated with the invalidity of its elements for a normal operation. It includes large deflections of beams, crossbars, excessive sedimentation of foundations, significant crack opening of reinforced elements etc.

The *Load type* along with the *Load case type* is determined by the normative document *Load Reliability Factor* for each of the selected loads (see Figure 2.209).

Load type depends on the duration of the loads and is divided into *permanent and temporary (long, short-term and special) loads.*

To perform the LDC, one must specify all the loads considered. To do this, select the desired one from *the Loads* list, where all the loads are available. Then the desired type is selected from *the Load Typ*e list. The types presented in this list, namely "permanent", "long", "short-term", "special", correspond to paragraphs 1.4-1.9 SNiP 2.01.07-85 * (paragraphs 6.3-6.4 of SP 20.13330.2011). A separate type of "wind" load is allocated additionally. This is caused by the fact that the calculation of the LDC in each combination can involve no more than one wind load. With the exception of this property, wind loading is included in the LDC as an ordinary short-term load.

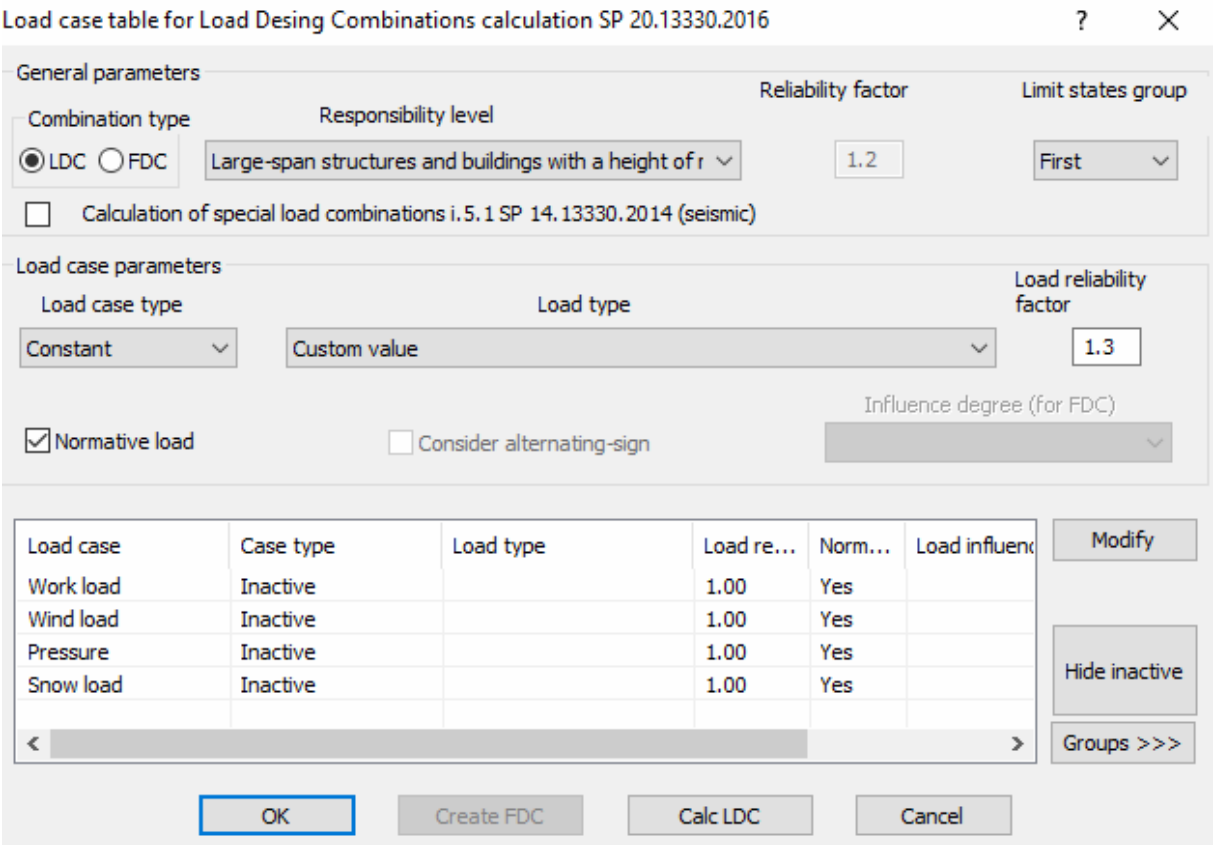


Fig.2.209 Load case table for Load Design Combinations calculation SP 20.13330.2011 dialog box

Duration share - the value determines which part of the time load (in fractions of a unit) is accepted in this load case as a long-acting one. The rest is considered to be a short-term one. For constant and long loads, the fraction of the duration is 1. For short-term loads, it is 0.

When the checkmark is ticked in the *Normative Load* box for a specific loading in the calculation of the LDC its estimated value will be introduced, i.e. load multiplied by the load reliability factor. If there is no checkmark, it will be considered that the calculated values are given in this load.

When the checkmark is ticked in the *Consider alternating sign*, the corresponding loading enters the LDC twice, in itself and with the opposite sign. This is used to set the seismic load in particular. You can set the *alternating* only for the *special* loading.

To change the load parameters, you need to highlight it, change the required parameters and click the **Modify** button.

When you click the **Groups >>>** button, a list of loading groups appears in the additional part of the dialog box. The **Create** button calls a dialog box for creating load groups: mutually exclusive, concurrent, simultaneously acting to introduce additional conditions for a combination of loads.

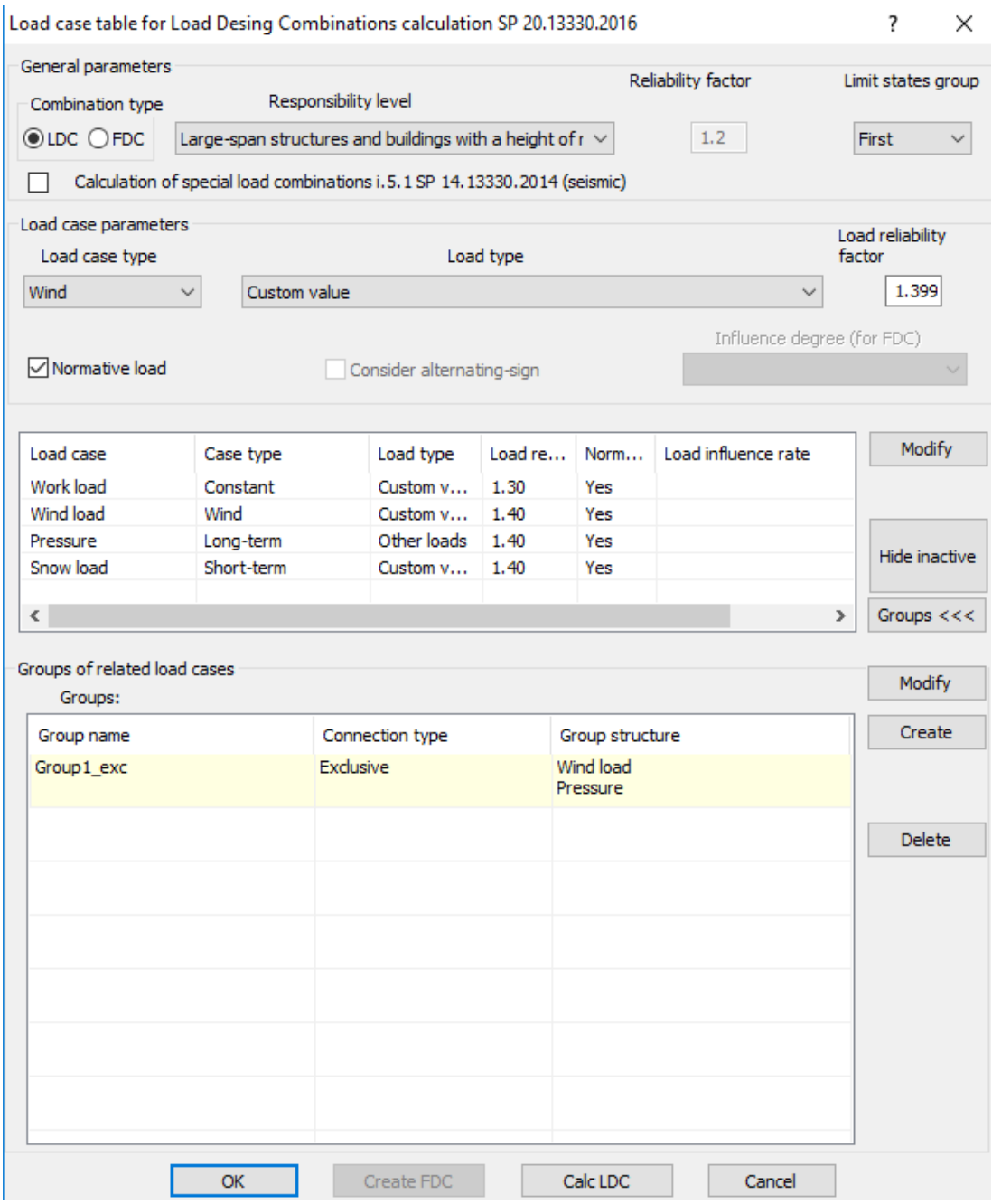


Fig.2.210 Groups of related load cases in Load case table

When a checkmark is set in the Calculation of special load combinations box in accordance with 2.1 of SNiP II-7-81 * (seismic), the combination coefficients for the LDC calculation are used from Table 2 of clause 2.1 of SNiP II-7-81 * (SP 14.13330.2014). If this option is NOT enabled, then the combination coefficients in the calculation of the LDC are done in accordance with clause 1.12 of SNiP 2.01.07-85 * (item 6 of SP 20.13330.2011).

The following rules are used when forming combinations:

1. Each load or combination of loads can be presented in the combination only once except those for which a specified fraction of the duration is different from 0 and 1. The proportion of the duration indicates how much a stochastic load is considered to be a permanent load. The remaining part of the stochastic load is considered as a short-term one.

2. A special combination can only include one of the special loads

3. The combination may include only one of the wind loads.

4. If groups are created, then the group loads are combined in the group type: mutex, cocurrent, concurrent.

The button "Calculation" starts calculation according to the criteria of the worst combinations of loads.

Commands for Joints Calculation

These commands transfer initial data from *APM Structure3D* to *APM Joint* for joint calculation. Select rods which have common node for joint calculation and press required button.

Shortcuts:

– Threaded Closed Joint
– Threaded Opened Joint
– Riveted Joint
– Single-Sided Welded Joint

In the appeared dialog box select load case with the joint nodal loads. *APM Sturcture3D* will transfer joint geometry according to the selected section and loads to *APM Joint*. Each rod and nodal load is placed to a separate layer in *APM Joint*.

The next stages of joint calculation are stated in *APM Joint* documentation.

Fatigue Calculation

The command calls a window with installations for fatigue calculation of a structure.

Initial data for fatigue calculation are stress-strain states corresponding to the maximum and minimum force influence on the structure at cyclic loads. It is supposed that the forces acting on the structure change following one law.

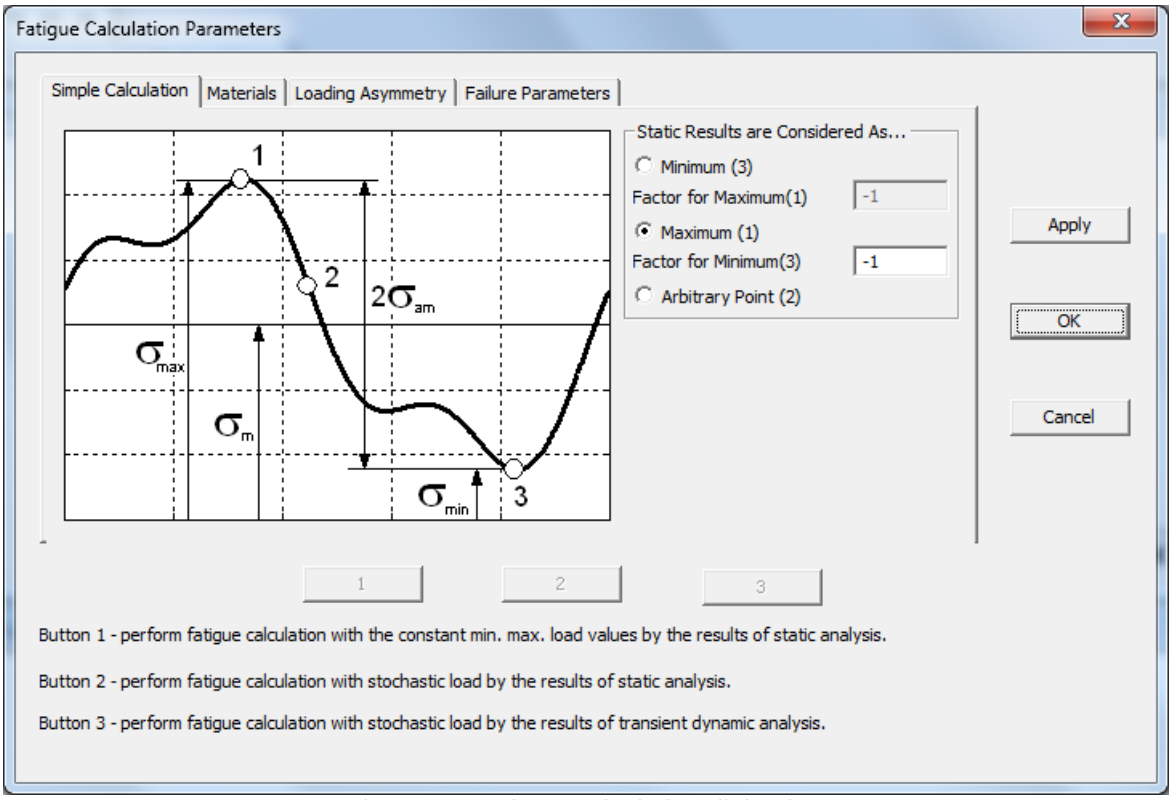


Fig. 2.169 Fatigue Calculation dialog box

The **Simple Calculation** tab assumes setting limits for changing external load based on the results of static calculation.

The group *Static results are considered as...* allows user to set the maximum and minimum values of the load acting on model of design. So, if static calculation has been performed for an average level of loading it is necessary to choose radio button *Arbitrary point (2)*, and then, in edit boxes *Factor for maximum (1) and Factor for minimum (3)* to enter dimensionless factors, by which it is necessary to increase the system of forces to have extreme load cases. If static calculation has been lead for a level of loading corresponding the maximal pressure it is necessary to choose radio button *Maximum (1)* and in edit box *Factor for minimum (3)* to specify dimensionless factor by which it is necessary to increase system of forces to have the level of loading corresponding the minimal pressure.

The table of the factors used at calculation is located in the bottom part of dialog. A certain set of factors can be set for each material.

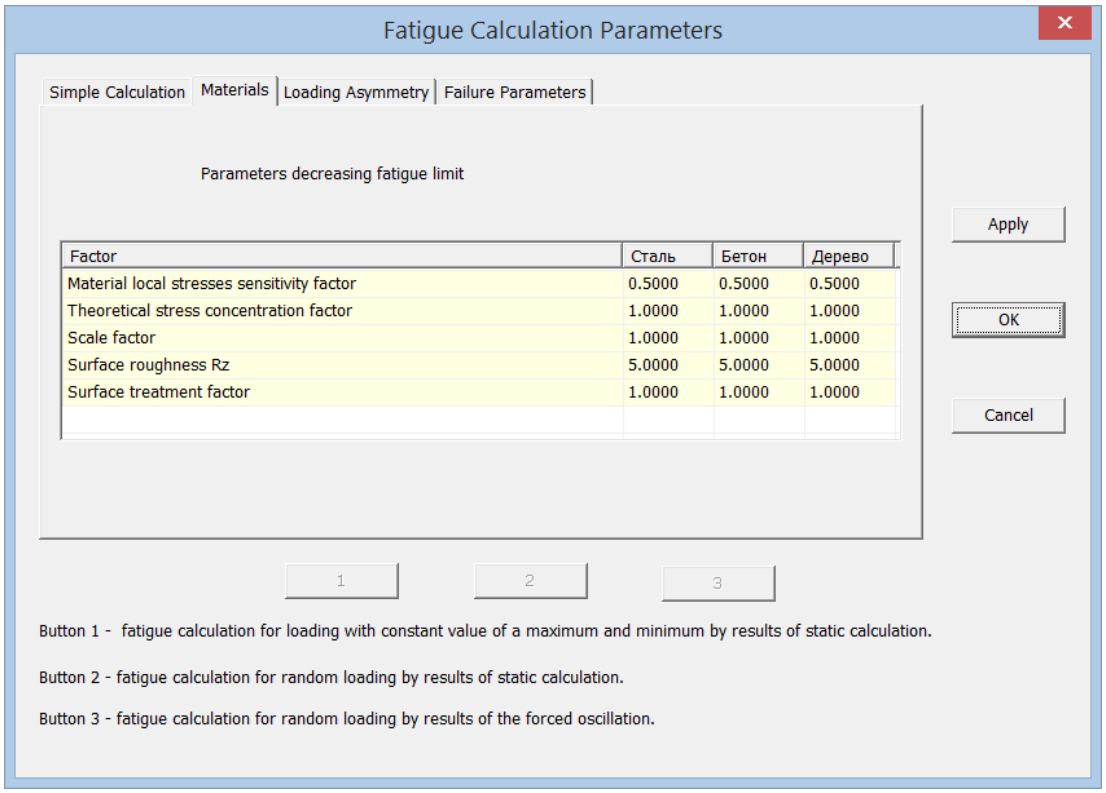


Fig. 2.170 Dialog window "Fatigue Calculation Parameters" tab "Materials".

The tab "Materials" of the dialog "Parameters of a Fatigue Calculation" contains a list of the fatigue limit of all materials reduction coefficients for an editable model.

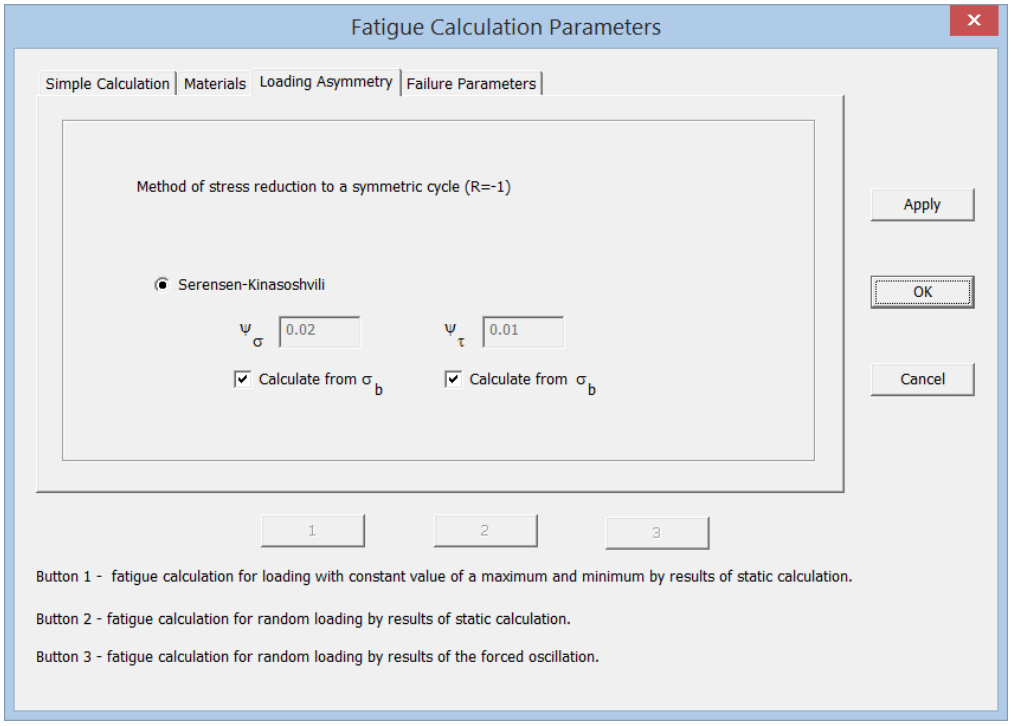


Fig. 2.171 Fatigue Calculation Parameters dialog tab "Asymmetry of Loading".

The tab "Asymmetry of Loading" is presented in Fig. 2.180. The content of the tab allows to carry out an asymmetric load to symmetric load, with the asymmetry index of R = -1.

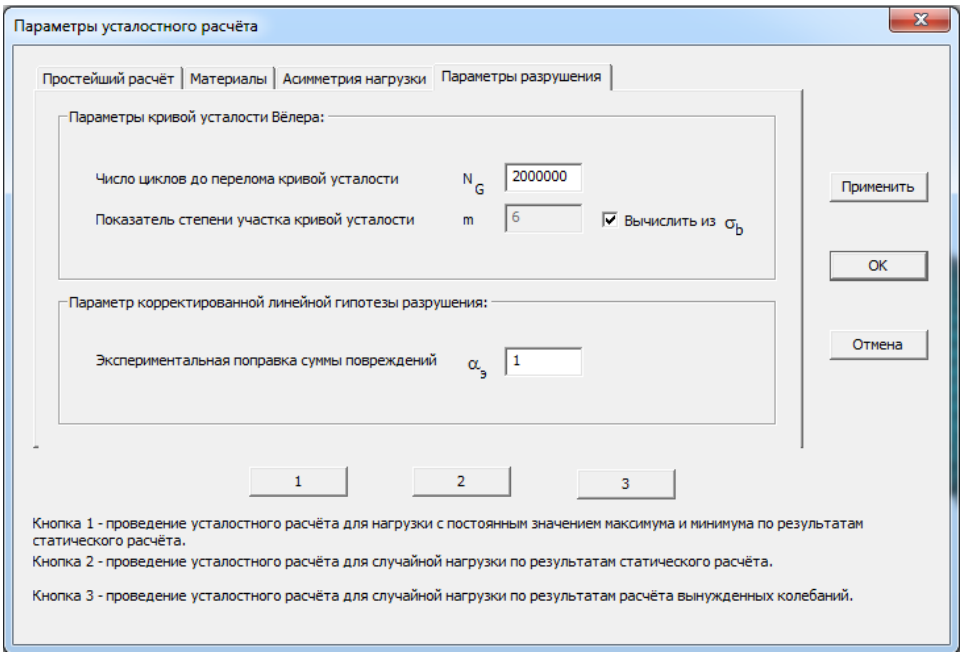


Fig. 2.172 Fatigue Calculation Parameters tab *Parameters of Destruction dialog.*

The tab "Parameters of Destruction" contains two groups of specified parameters, Fig. 2.181. The group "Wohler Parameters of a Fatigue Curve" allows to specify quantities defining a type of fatigue curve.

The group "Parameter of Adjusted Linear Density of Destruction" - allows to introduce the *Experimental correction of a sum of damages*.

Buttons of a calculation start of different fatigue calculations methods are located at the bottom of a dialog that is examined.

The push on "1" button carries out a start for a calculation on the simplest algorithm when load consists of alternating sequences with a constant value of maximum and minimum, see a scheme with the tab "Simplest calculation" of this dialog.

The push on «2» button leads to a start for a calculation of a fatigue algorithm for random load based on the results of a static calculation.

The push on «3» button leads to a start for a calculation of a fatigue algorithm for random load based on the calculation results of forced fluctuations for super-elements.

More detailed information about coefficients is *in Chapter 4 of the Fatigue Detachment.*

Calculation Options

The command calls a Calculation options dialog window with pages corresponding to each calculation type. Calculation options are saved to *.frm file with the model. After opening of model is no needed to set calculation options anew.

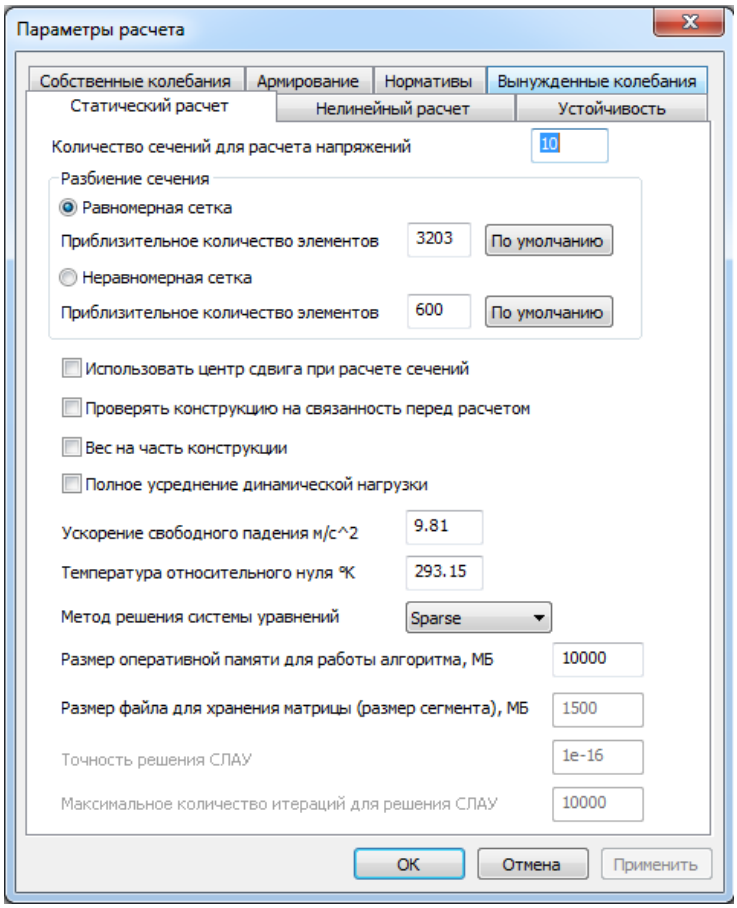


Fig. 2.173 Analysis Options (static analysis tab) dialog box

Stresses in rods are calculated in a discrete number of cross-sections. You can define the number of cross-sections in *Number of cross-sections for stress calculation* input field. In order to calculate stress distribution in the rod cross-section, the cross-section is meshed into small finite elements. There are two meshing algorithms: with regular and non-regular mesh. Regular algorithm is faster than the non-regular on the same number of elements, but requires more elements to keep sufficient precision. Here you can choose one of these algorithms and also define the approximate number of elements. Push **Default** button to restore program initial number of finite elements.

Checkbox *Check structure connection before calculation* allows you to turn off this option. You can perform this checking using **Tools / Check connection** menu command.

Checkbox *Use Shear Center for Cross-Section Calculation* allows you to take into account shear effects for rod calculation.

Using *Equation solve method* list you can select the most suitable method for static calculation.

In *LDL* method global stiffness matrix of the whole structure is factorized into the form of $[L]^T[D][L]$ and then solved using Gauss procedure. Using LDL method stiffness matrix is in RAM and if it is not enough Windows automatically creates temp files on a hard disk. The method has restriction on problem dimensions - 300 thousand DOFs approximately.

Frontal method is most suitable for models with great number of degrees of freedom. Its main feature is that global stiffness matrix is assembled implicitly and stored on disk and the system of linear equations is solved with some kind of front propagating through all DOFs.

The following parameters are valid only for frontal method:

Working RAM size – RAM size used for solver to store “front” and other frequently accessed data.

File size for matrix storage – selected according to the operating system installed and file system used.

MT_Frontal is the modified frontal method for multiprocessor computers and in many cases works much faster, than *LDL* and *Frontal*. However its efficiency strongly depends on half-width value of stiffness matrix.

Sparse is the improved method for work with the sparse matrixes, a providing increment of calculation speed. By *Sparse* method only nonzero elements are stored in a stiffness matrix, and in

temp files located on a hard disk. It is intended for large models with the high half-width of a stiffness matrix.

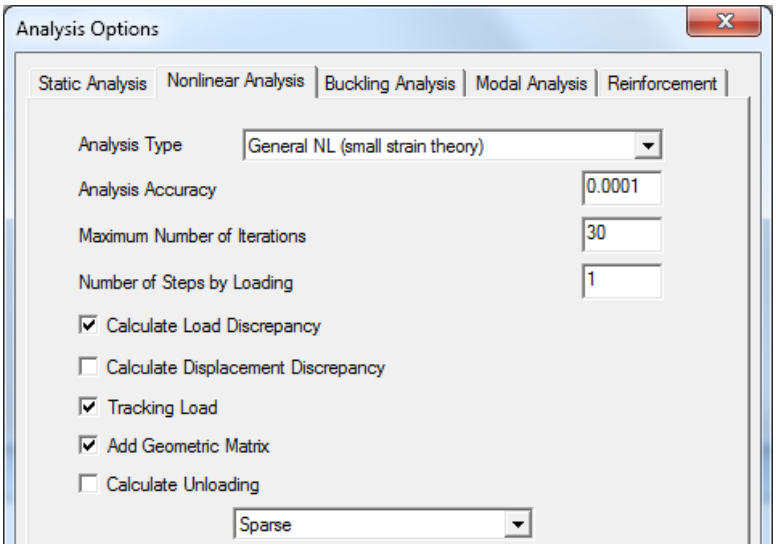


Fig. 2.174 Analysis Options (nonlinear analysis tab) dialog box

To perform nonlinear analysis you must define type of nonlinearity; it can be geometric or physical. Available *analysis types*:

- *Geometric NL*;
- *Analysis with one-dir supports*;
- *Physical NL (small strain theory)*;
- *General NL (small strain theory)*;
- *Physical NL (large strain theory) - flow theory*;
- *General NL (large strain theory) - flow theory*;

Account of general or physical nonlinearity is available for solid elements only.

You can define relative *nonlinear analysis accuracy* and *maximum number of iterations* for nonlinear analysis. *Calculate unloading* is the additional option for physically nonlinear problems. *LDL* or *Sparse* can be set as a solution method.

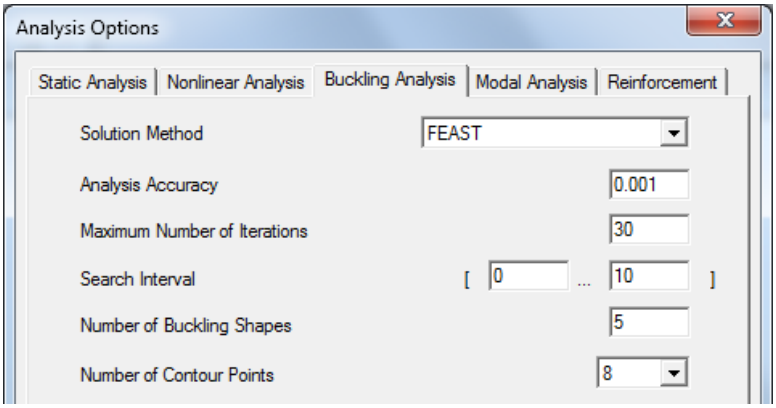


Fig. 2.175 Analysis Options (buckling analysis tab) dialog box

There are three available solution methods for buckling analysis:

The Arnoldi iterations is a method of a generalized eigenvalue problem solution, allowing to get the reserve coefficient with relatively low costs of processor time. But the method does not let to get a solution for systems with a great number of freedom degrees, therefore is available only to a version of x32 system. The method determines only one minimum resistance reserve coefficient.

Determinant or *Determinant (Sparse)* – procedure for models with high number of DOF. For this method you can define *maximum value of buckling safety factor* in corresponding field to narrow seek area. Parameters of *analysis accuracy* and *maximum number of iterations* are set for the methods. *Maximum value of buckling safety factor*, *memory size* for algorithm work, MB and *file size for matrix storage* (size of segment), MB are parameters only for *determinant* method.

✍ *Note: the general size of files on hard drive will depend on dimension and topology of the problem.*

Lanczos allows to calculate previously defined *number of buckling shapes* in the neighborhood of the *rough value of buckling safety factor*.

FEAST allows to calculate previously defined *number of buckling shapes* in the *search interval* of buckling safety factors.

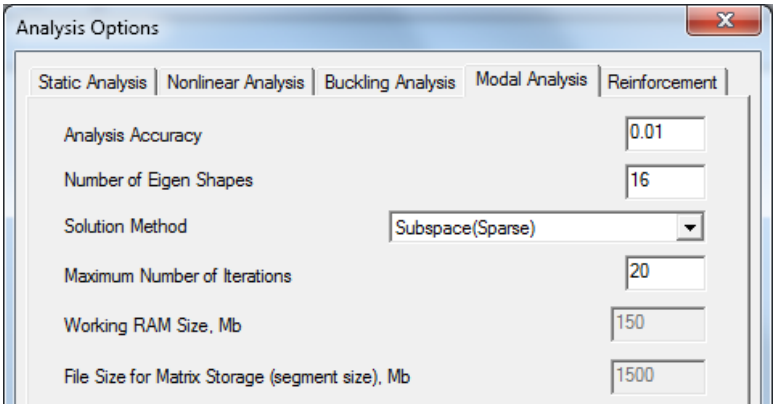


Fig. 2.176 Analysis Options (modal analysis tab) dialog box

For eigen frequencies calculation there are different solvers: Arnoldi's iterations, subspace Iterations, subspace Iterations (Sparse), subspace iterations (Sparse) without orthogonalization, *Lanczos iterations*.

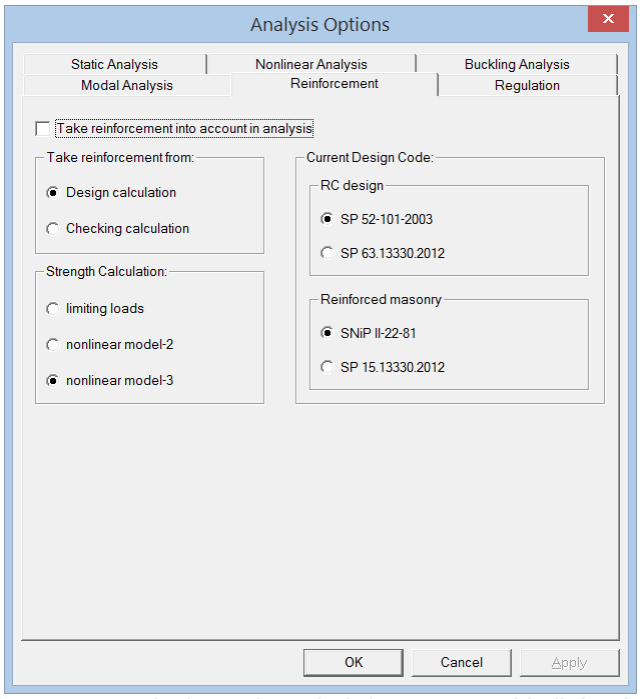


Fig. 2.177 Analysis Options (reinforcement tab) dialog box

For the reinforcement calculation the there is a choice of normative documents on which the calculation is executed for reinforced concrete structural elements (SP 52-101-2003 or SP 63.13330.2012) and reinforced masonry elements (SNiP II-22-81 or SP 15.13330.2012).

Structure model can be recalculated with accounting of reinforcement stiffness that can be takes from design or checking calculation.

Design and checking calculation of reinforced concrete elements are realized in this version as a nonlinear and linear (strength calculation is performed by limiting loads criteria) problems.

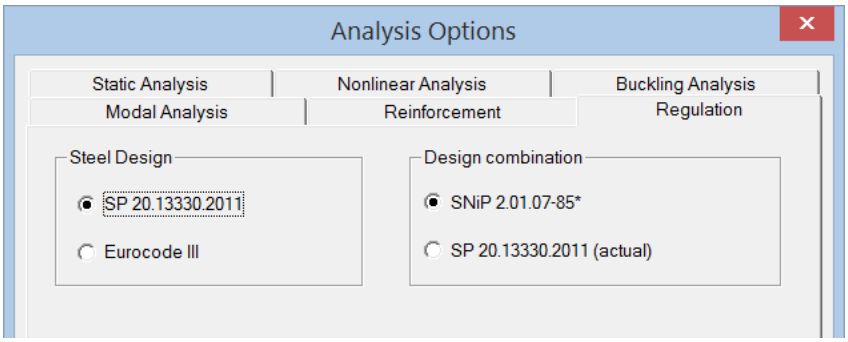


Fig. 2.178 Analysis Options (Regulation tab) dialog box

Regulation tab allows to set the code documents for steel structural elements design (SP 20.13330.2011 or Eurocode 3) and in the calculation of the RSU (SNiP 2.01.07-85 or SP 20.13330.2011).

Results menu

This menu contains commands that allow you to view and analyze calculation results.

Select Super Element to get Results

This command allows to select one of the super elements to get results or whole model. The command will be activated only when calculation of super elements will be performed.

Loads

The command calls a window, which allows you to view a set of numerical parameters of rods and plates: nodal loads, displacements, frame forces in rods and construction mass.

Result Map

The command calls a window to have a look at a number of design parameters. Besides, it allows user to set different result presentation options.

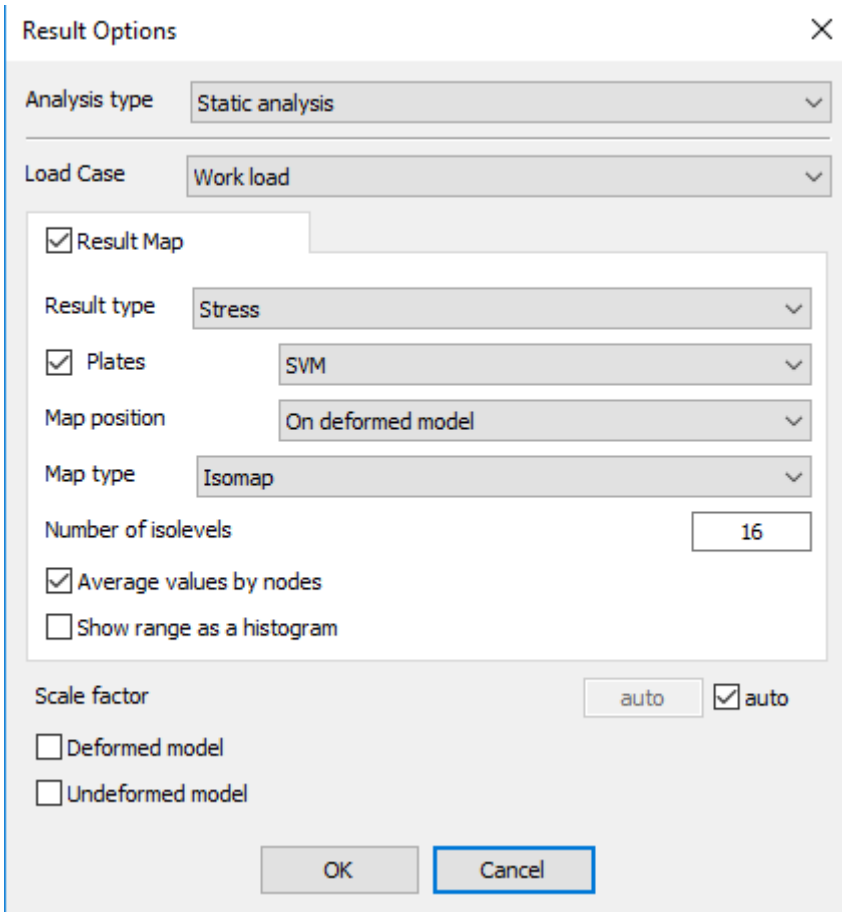


Fig. 2.179 Result Selection dialog box

The available results from the calculation types are selected from the Result type.

In the load case we select from the drop-down list one of those loads for which, in our case, Static calculation was performed.

A checkmark in the option Result Map indicates that a result map will be displayed.

In the Result Group, you can select Stress, Loads, Movements, Factor of safety, Major stresses, Fatigue, Deformations, Major stresses in a vector form.

In order to display the map of the selected result type for Rods, Plates, Volumetric elements are shown when there are checkmarks in the corresponding elements type. If there are no types of elements in the model, these elements are not available in the Output Results Display window.

The output parameters are selected for each type of finite element.

Below we give a description of some parameters.

- UX – displacement in X axis of global coordinate system.
- ROTX – rotation angle around X axis in global coordinate system.
- USUM – total displacement.
- ROTSUM – total rotation angle.
- SX – normal stress in X axis of element local coordinate system.
- SXY – tangent stress in surface with normal X and in Y direction of element local coordinate system.
- SVM – effective stress von Mises.
- SMAXTAU – equivalent stress by maximum shearing stress theory
- SMOHR – equivalent stress by Mohr theory

$$S_{\mathrm{MAXTAU}} = S_1 - S_3\,; \quad S_{\mathrm{MOHR}} = S_1 - kS_3\,,$$

S_1, S_3 – first and third principal stresses; *k* – relation of tensile yield stress to compressive yield stress for materials of plates.

Signs “+” and “–“ in designation of plate parameter mean the plate surface it calculated on. “+” means plate surface in Z axis (normal) direction of plate coordinate system. “-” means plate surface in

opposite Z axis (normal) direction of plate coordinate system. “max” – maximum absolute value by plate thickness.

Similar stress components with an index 0 (SX0, SXY0, SVM0 etc.) are corresponding to stress components for median plate surfaces. Normal and tangent stresses can be both positive, and negative unlike positive equivalent stresses. Positive value – tensile stress, and negative – compressive stress.

For more detail, finite element information see Chapter 7.

Scale Factor text field specifies the scale factor for drawing a deformed structure.

Averaged Node Values flag concerns parameter map drawing in isoareas form. If this flag is switch on the values of selected parameters in a node will be averaged over all elements with this node.

Meaning of other fields is obvious from their designation.

When result map is displayed, you can see numerical value of specified result by pressing left mouse button in Selection mode at area in question.

Stress in Cross-section

The command enables a mode, which allows you to view stress map in arbitrary cross-section of the rod. To view a stress map, select a rod then move the arrow that appears on the rod to reach desired position. After you have set the arrow at a desired position, click to call a stress map window. The command is available only in *Stress Map* and *Results* views.

Show Element Forces

The command calls a force selection dialog box that allows you to select force components for diagrams shown below.

The group of radio buttons is used to select force component. Select checkbox *Show values on Diagram* to see values on frame forces diagrams. See also chapter 4.

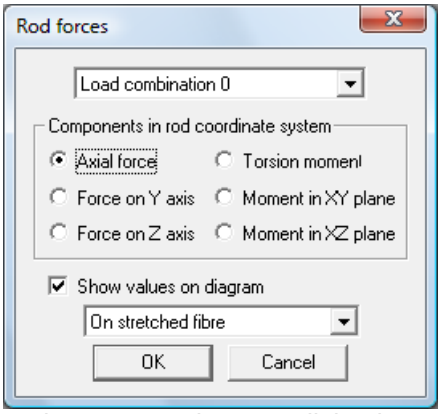


Fig. 2.180 Rod Forces dialog box

Support Reactions

The command calls a window with base reactions represented by a table of values. Each row in the column represents reactions in one node. When a row is selected, the corresponding node is highlighted in a different color. More detailed information about reactions can be found in chapter 6.

Composite cross-section

With the help of this command it is possible to see the stress and strain in the cross-section of the selected composite plate. First you need to open any result map and after that, in the Result menu, select the Composite section and then select the plate by clicking on the LMB to view the results in its section. After that, the Composite Cross Section dialog box opens, Fig. 2.224.

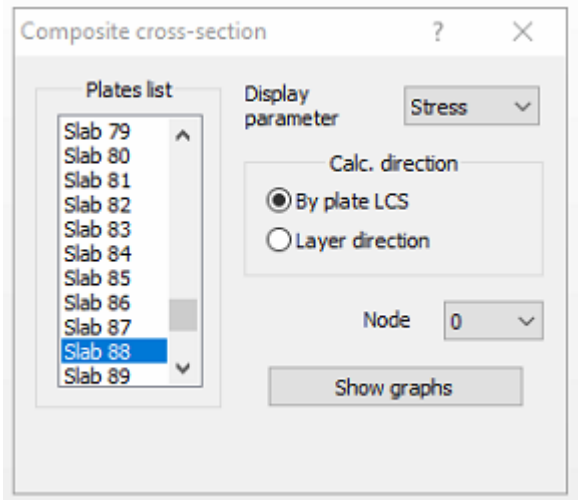


Fig. 2.224 The Composite Cross Section dialog box

In the Plates List window, the selected plate is highlighted, however, you can select any other plate to view the stress and strain in its section.

From the drop-down list, the Display option, we select either Stress or Strain that we want to view for this particular plate.

The selected parameter can be viewed in the Calculation direction: By LCS plate or in the layer direction.

Next, after selecting the Node (from the drop-down list), click the Show Graphs button.

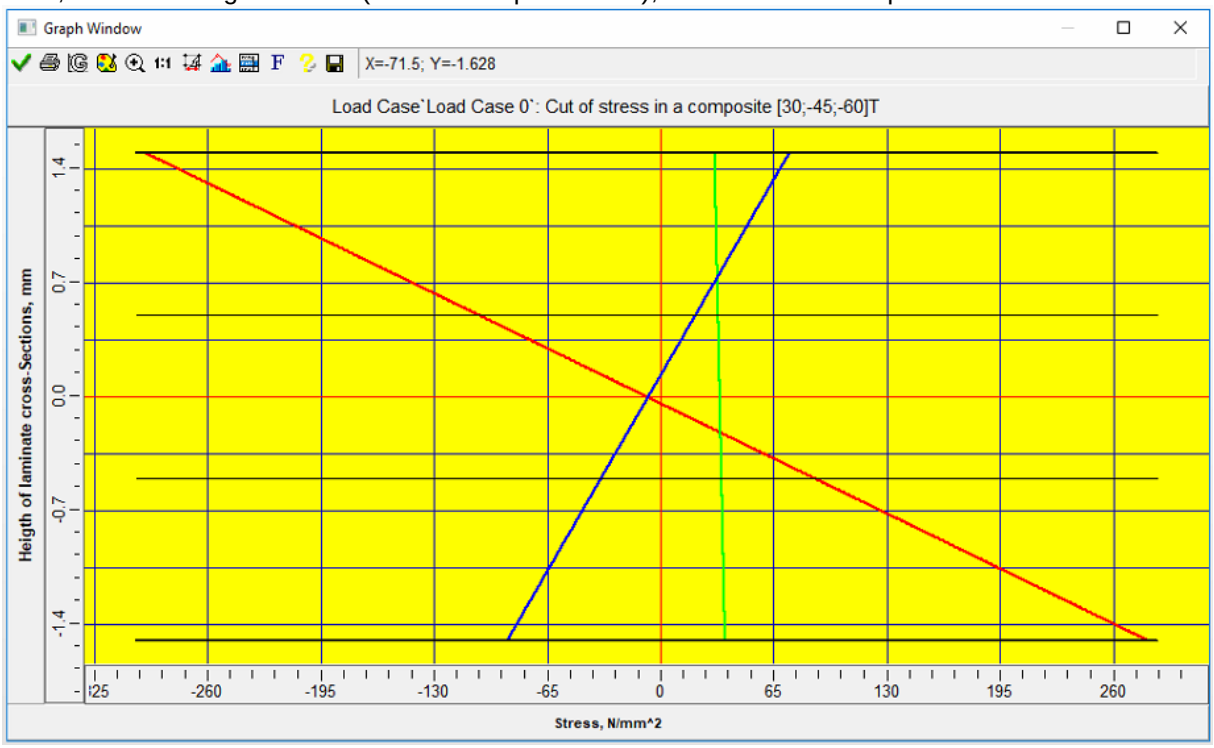
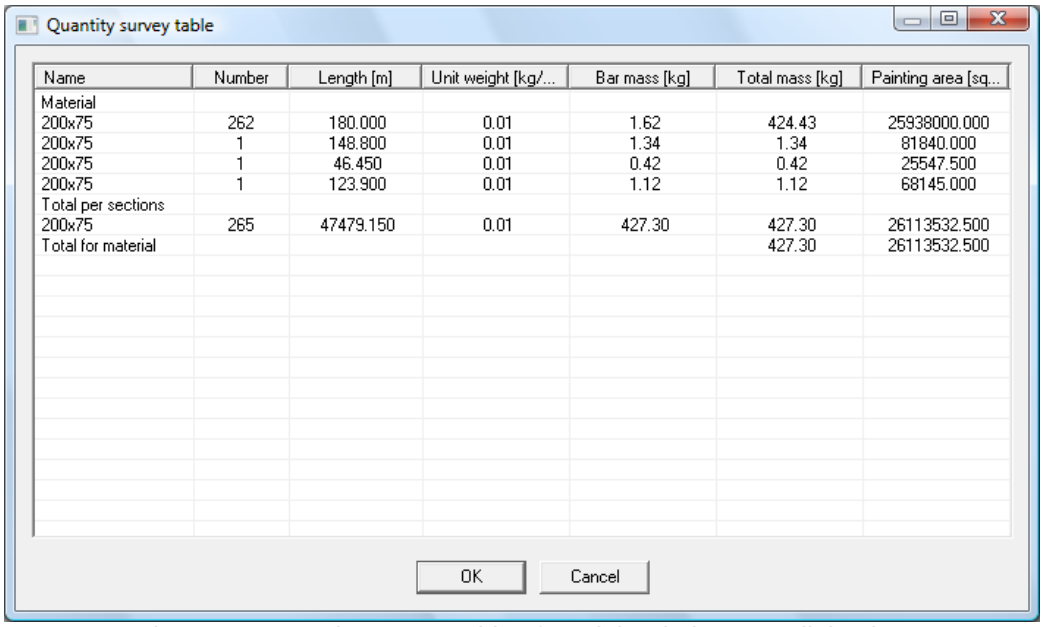


Fig. 2.225 The Graph Window Cut of stress in a composite

Quantity Survey

The command displays the summary table of rod elements.



Quantity survey table

Name	Number	Length [m]	Unit weight [kg/...	Bar mass [kg]	Total mass [kg]	Painting area [sq...
Material						
200x75	262	180.000	0.01	1.62	424.43	25938000.000
200x75	1	148.800	0.01	1.34	1.34	81840.000
200x75	1	46.450	0.01	0.42	0.42	25547.500
200x75	1	123.900	0.01	1.12	1.12	68145.000
Total per sections						
200x75	265	47479.150	0.01	427.30	427.30	26113532.500
Total for material					427.30	26113532.500

OK Cancel

Fig. 2.181 Quantity survey table of model rod elements dialog box

Code Combination Results

The command displays table with results of code combination calculation.

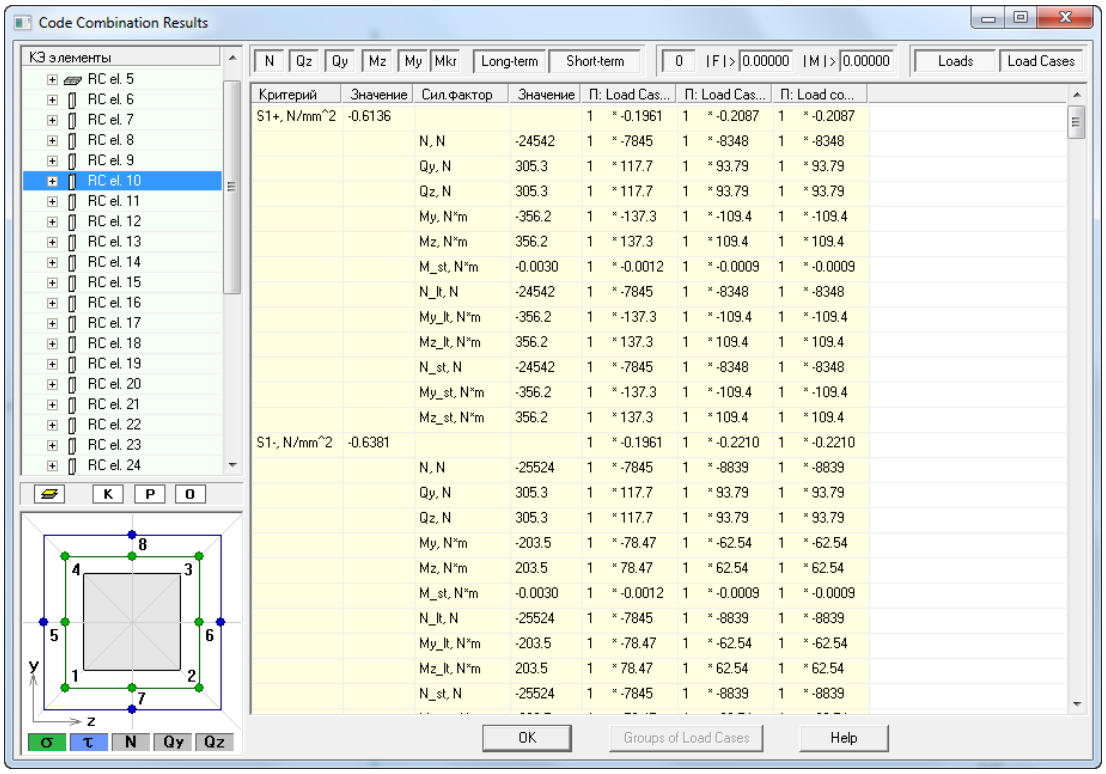


Fig. 2.182 Results of code combination for rod

Rods

At code combination calculation extreme values of normal and tangent stresses in section characteristic points are defined.

For normal stresses: $\sigma_i = \dfrac{N}{F} \pm \dfrac{M_y z_i}{I_y} \pm \dfrac{M_z z_i}{I_z}$, *i* – section point (*i = 1…8*).

If $y = \pm\dfrac{b}{2}$ и $z = \pm\dfrac{h}{2}$, it will be: $\sigma_i F = N \pm \dfrac{M_y}{l_{z,j}} \pm \dfrac{M_z}{l_{y,j}}$.

For shear stresses: $\tau_y F = \dfrac{Q_y}{2} \pm \dfrac{M_{ip}}{2(l_{y1} + l_{y2})}; \tau_z F = \dfrac{Q_z}{2} \pm \dfrac{M_{ip}}{2(l_{z1} + l_{z2})}$.

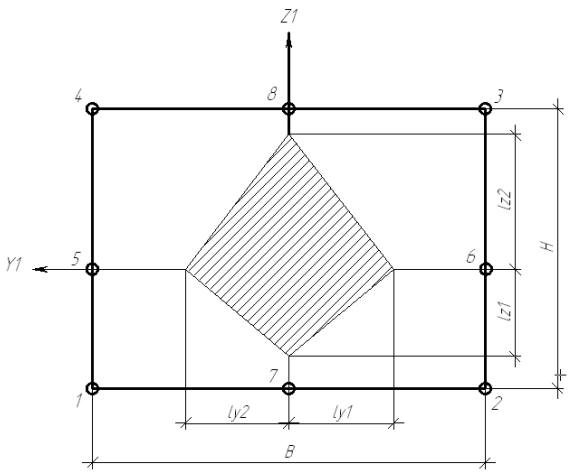


Fig. 2.183 Rod cross-section points

Table 2.1 – Designation of RSU criteria for rods

Symbol in RSU table	Symbol in equations	Symbol in RSU table	Symbol in equations
S1+	σ_{1+}	T5+	τ_{5+}
S2+	σ_{2+}	T6+	τ_{6+}
S3+	σ_{3+}	T7+	τ_{7+}
S4+	σ_{4+}	T8+	τ_{8+}
S5+	σ_{5+}	T5-	τ_{5-}
S6+	σ_{6+}	T6-	τ_{6-}
S7+	σ_{7+}	T7-	τ_{7-}
S8+	σ_{8+}	T8-	τ_{8-}
S1-	σ_{1-}	N+	N^{+}
S2-	σ_{2-}	N-	N^{-}
S3-	σ_{3-}	Qy+	Q_{y+}
S4-	σ_{4-}	Qy-	Q_{y-}
S5-	σ_{5-}	Qz+	Q_{z+}
S6-	σ_{6-}	Qz-	Q_{z-}
S7-	σ_{7-}		
S8-	σ_{8-}		

Plates

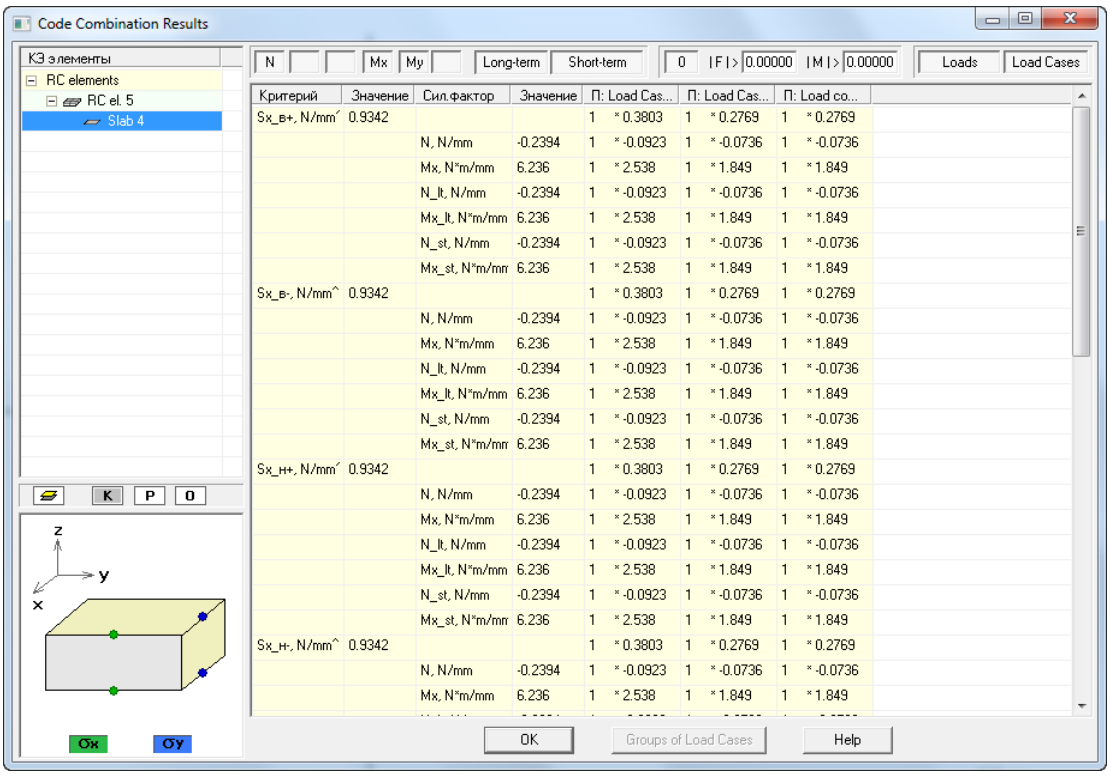


Fig. 2.184 Results of code combination for plate

Stresses are calculated in upper and lower faces taking into account bending on following dependences:

$$\sigma_X^{H(B)} = N_X \pm \frac{6M_X}{h^2}; \qquad \sigma_Y^{H(B)} = N_Y \pm \frac{6M_Y}{h^2}; \qquad \tau^{H(B)} = T_{XY} \pm \frac{6M_{XY}}{h^2};$$

where h – plate thickness, B and H – indexes of upper and lower plate faces. Angle changes with step $\alpha = 22{,}5^\circ$.

Table 2.2 – Designation of RSU criteria for plates

Angle, grad.	Symbol in RSU table	Symbol in equations	Symbol in RSU table	Symbol in equations
90	Sh90+	σ_+^B	Sl90+	σ_+^H
90	Sh90-	σ_-^B	Sl90-	σ_-^H
67,5	Sh67_5+	σ_+^B	Sl67_5+	σ_+^H
67,5	Sh67_5-	σ_-^B	Sl67_5-	σ_-^H
45	Sh45+	σ_+^B	Sl45+	σ_+^H
45	Sh45-	σ_-^B	Sl45-	σ_-^H
22,5	Sh22_5+	σ_+^B	Sl22_5+	σ_+^H
22,5	Sh22_5-	σ_-^B	Sl22_5-	σ_-^H
0	Sh0+	σ_+^B	Sl0+	σ_+^H
0	Sh0-	σ_-^B	Sl0-	σ_-^H
-22,5	Sh_22_5+	σ_+^B	Sl_22_5+	σ_+^H
-22,5	Sh_22_5-	σ_-^B	Sl_22_5-	σ_-^H
-45	Sh_45+	σ_+^B	Sl_45+	σ_+^H
-45	Sh_45-	σ_-^B	Sl_45-	σ_-^H
-67,5	Sh_67_5+	σ_+^B	Sl_67_5+	σ_+^H
-67,5	Sh_67_5-	σ_-^B	Sl_67_5-	σ_-^H
90	Th90+	τ_+^B	Tl90+	τ_+^H
90	Th90-	τ_-^B	Tl90-	τ_-^H
45	Th45+	τ_+^B	Tl45+	τ_+^H
45	Th45-	τ_-^B	Tl45-	τ_-^H
0	Th0+	τ_+^B	Tl0+	τ_+^H
0	Th0-	τ_-^B	Tl0-	τ_-^H
–	Qx+	Q_{X+}		
–	Qx-	Q_{X-}		
–	Qy+	Q_{Y+}		
–	Qy-	Q_{Y-}		

Buckling

the command calls a window with the buckling safety factors resulting from the calculation of stability and deformation.

To view the forms of a buckling loss, select the line in the table and click the Shape button. See Chapter 6 for details.

This command reflects the previous way of outputting the buckling calculation results.

Below are the results of calculating buckling in a new way.

Shortcut: .

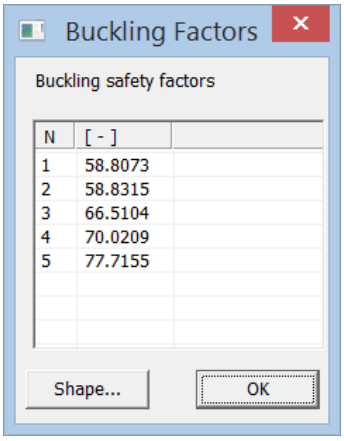


Fig. 2.231 Results of calculation of Buckling

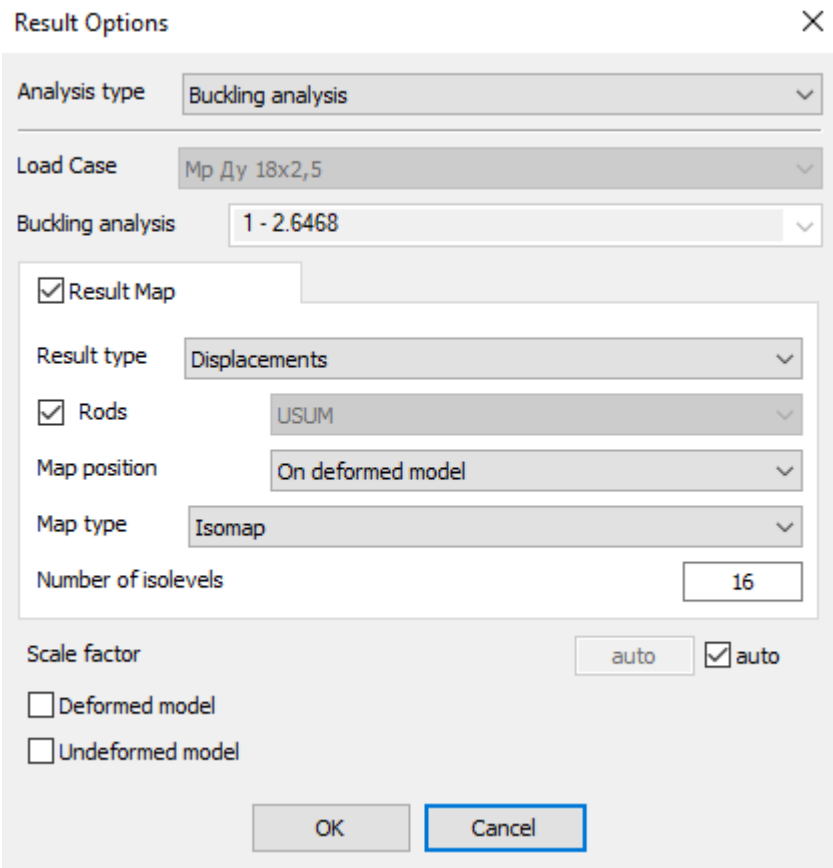


Fig. 2.232 The Results Options dialog box when viewing the buckling calculation results

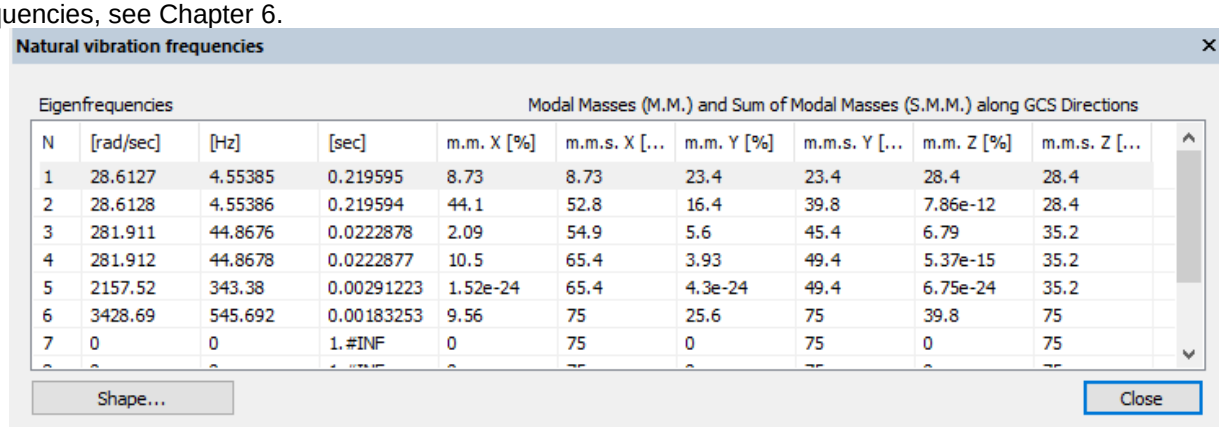
To view the forms of buckling loss, select the Calculation type - Sustainability, from the Buckling drop-down list, select the line with the number of the resistance loss form and its corresponding stability factor and click the OK button. A map of buckling loss for the selected form will open.

See Chapter 6 for details.

Natural Frequencies

the command calls a window with natural vibration frequencies of the construction and the sums of the modal masses along the directions in the global coordinate system. Press the Shape button to

view the waveform for the selected frequency. For more information on calculating the natural frequencies, see Chapter 6.



N	[rad/sec]	[Hz]	[sec]	m.m. X [%]	m.m.s. X [...	m.m. Y [%]	m.m.s. Y [...	m.m. Z [%]	m.m.s. Z [...
1	28.6127	4.55385	0.219595	8.73	8.73	23.4	23.4	28.4	28.4
2	28.6128	4.55386	0.219594	44.1	52.8	16.4	39.8	7.86e-12	28.4
3	281.911	44.8676	0.0222878	2.09	54.9	5.6	45.4	6.79	35.2
4	281.912	44.8678	0.0222877	10.5	65.4	3.93	49.4	5.37e-15	35.2
5	2157.52	343.38	0.00291223	1.52e-24	65.4	4.3e-24	49.4	6.75e-24	35.2
6	3428.69	545.692	0.00183253	9.56	75	25.6	75	39.8	75
7	0	0	1.#INF	0	75	0	75	0	75

Fig. 2.233 Dialog box with the results of calculating natural vibration frequencies.

Seismic norms of many countries (Eurocode 8, UBC-97, seismic norms of Ukraine, etc.) assume that the sum of modal masses for each of the seismic action directions should be no less than the established limits. 85-90% is usually assumed for the horizontal component of the seismic action, for the vertical component it is 70-90%.

This command reflects the previous way of displaying the results of calculating the natural frequencies.

Below is the conclusion of the results of the calculation of natural frequencies by a new method.

Shortcut:  .

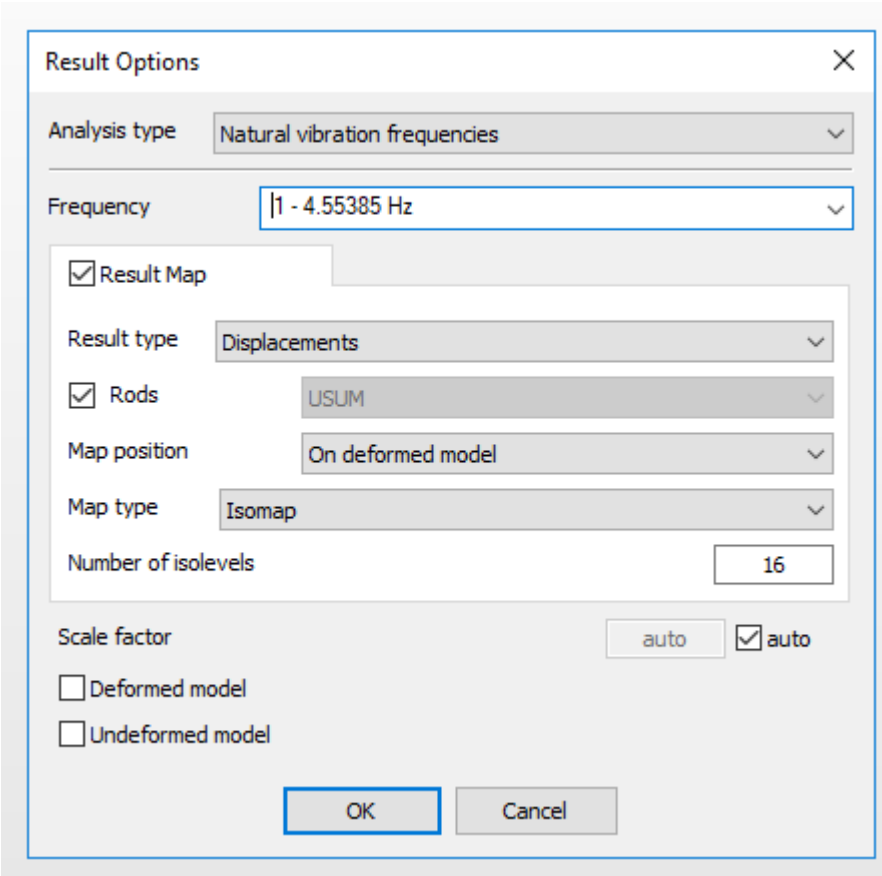


Fig. 2.234 Dialog box Result Options when viewing the results of the calculation of natural frequencies

To view the value of the natural frequency and its own form, select the Calculation type - Natural frequencies in the Frequency drop-down list, select the line with the number of the natural frequency and its own form and the corresponding modal mass at those coordinates in the GCS for which the i-

frequencies and the sum of modal masses in different coordinates from the 1st to the i-th natural frequency. After clicking the OK button, the map of its own form opens for the selected natural frequency.

See Chapter 6 for details.

Animation

This command allows to display maps of results (stress, strain, displacements, eigen shapes, etc.) in animation mode.

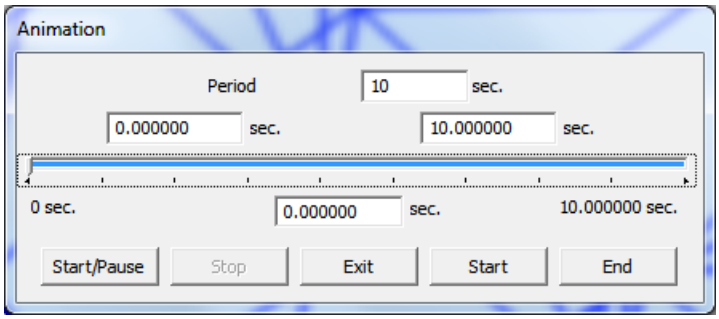


Fig. 2.185 Animation parameters dialog box

Forced Oscillations

The command shows animated result map for dynamic response of the structure based on forced oscillations calculation results.

Graph of Displacements

The command allows you to see graphs of displacements in time in global axes for arbitrary node.

Graph of Stress

The command allows you to see stress-time diagram for arbitrary rod cross-section.

Fatigue Longevity for Stochastic Load Case

The command displays the results of fatigue calculation.

Result Animation of Heat Transfer Analysis

The command invokes animation window with heat transfer results.

Result Map of Heat Transfer Analysis

The command invokes dialog box with available heat transfer results.

Result Options of Heat Transfer Analysis

The command invokes dialog box for setting options of heat transfer results map.

Reinforcement Map

The command invokes a window for viewing the results of reinforced elements. Besides, it allows to set various options for representation of reinforcing map according to table.

Table 2.3 – Reinforcing parameters

Results	Element type	Parameters	Comments
Design/Checking status	Rods	Status	1 – reinforcing is designed 0 – reinforcing was not designed -1 – reinforcing is not designed
	Plates	Status	

Reinforcement ratio	Rods	Reinforcement ratio	Reinforcement ratio in area extent
	Plates	Summary reinforcement ratio Reinforcement ratio along X axis Reinforcement ratio along Y axis	Reinforcement ratio in area extent
Rod reinforcement	Rods	ASSUM Upper longitudinal reinforcement Bottom longitudinal reinforcement Upper lateral longitudinal reinforcement Bottom lateral longitudinal reinforcement (for T and I sections) Transverse reinforcement along Z axis Transverse reinforcement along Y axis	Reinforcement area, mm^2
Plate reinforcement	Plates	Along X axis on both sides Along Y axis on both sides Along X axis on top Along X axis on bottom Along Y axis on top Along Y axis on bottom	Reinforcing intensity, mm^2/ mm
Reinforcement utilization factor of rods	Rods	Longitudinal reinforcement along Z axis Longitudinal reinforcement along Y axis By time point of crack occurrence along Z axis By time point of crack occurrence along Y axis By skew bending By concrete strip between oblique plane along Z axis By oblique plane on shear force action along Z axis By oblique plane on moment action along Z axis By concrete strip between oblique plane along Y axis By oblique plane on shear force action along Y axis By oblique plane on moment action along Y axis By strength of an element between dimensional sections By strength of dimensional sections By combined action of torsion and bending moments By combined action of torsion moment and shear force Maximum	From 0 to 1
Reinforcement utilization factor of plates	Plates	along X axis along Y axis	From 0 to 1
Crack opening width	Rods	Short-term Long-term	Actual width of crack opening, mm
	Plates	Short-term along X axis Short-term along Y axis Long-term along X axis Long-term along Y axis	

Reinforcement Result Options

The command invokes a window for setting options of reinforcement map.

Result Options

The command invokes a window for setting options of results map.

Result Range

The command allows user to define values range for result map drawing. It calls a dialog box shown below.

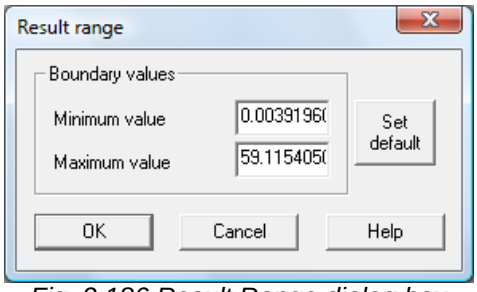


Fig. 2.186 Result Range dialog box

Window menu

This menu contains commands that allow you to activate and arrange windows.

Cascade

This command arranges all child windows in a cascade format.

Tile

This command arranges all child windows in a tiled format.

Arrange Icons

This command arranges icon windows.

View 1, View 2 …

The command activates view.

Help menu

Content
shows a window with the contents of the help system for the APM Structure3D program.

The site of STC "APM"
the command sends in the current browser to the site of the Scientific and Technical Center "APM" at http://apm.ru/

Project Gallery of Scientific and Technical Center "APM"
the command sends in the current browser to the project site of STC "APM" at http://cae.apm.ru/, where the gallery of completed projects is presented.

Video lessons
the command sends in the current browser to the youtube channel of STC "APM" company, which presents examples of various calculation tasks performed including in APM Structure3D.

Customer Support Center
the command sends in the current browser at http://helpdesk.apm.ru/ - the center of calls and support for registered users to receive prompt technical support and to take into account the wishes for finalizing the software products of the NTC "APM" company.

About the module ...
the command calls a dialog box containing information about the module.

Chapter 3. Cross-section Editor

APM Graph is used as the cross-section editor. Difference between the cross-section editor and *APM Graph* are additional functions for working with sections. The description of the main menu and toolbar commands can be found in the *APM Graph* user's guide. Cross-section editor allows user to draw a cross-section and save it in single file or add to the cross-section library. Further cross-sections stored in library can be used to set rod cross-section in 3D frame construction editor. Cross-section editor is enabled by **File / New / Cross-Section**. Cross-section editor window is shown below.

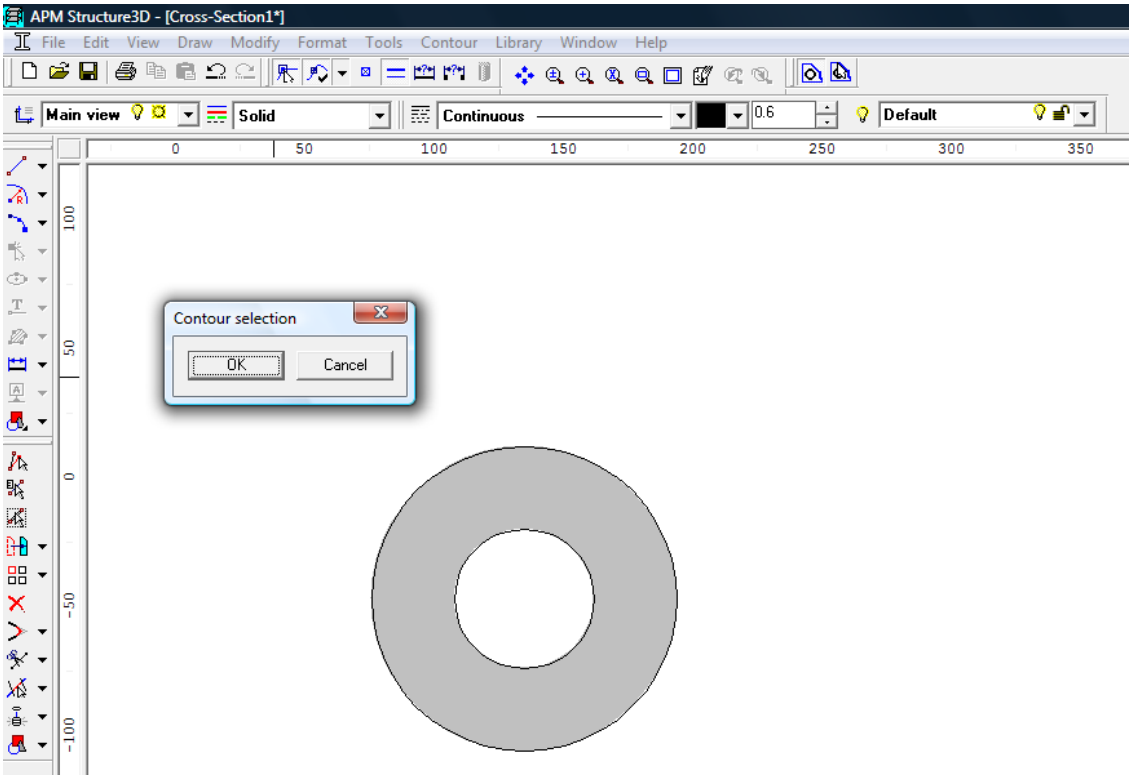


Fig. 3.1 Cross-section editor window

Contour menu

These menu commands give the opportunity of cross-section creation through simple, and user defined contour.

Fig. 3.2 Contour menu

Simple Contour

This command activates the mode of automatic contour selection. To select a contour click the mouse on any contour element. First you should specify external contour and after that, the internal ones, if any. The resulting contours are highlighted in dark blue color. Corresponding contour can be found only if it is closed. Press **OK** button in contour dialog box after contours selection. The area inside the selected contours will be filled with a color. It means that contour is defined. To remove contour click mouse inside the contour area in **Delete** mode .

Shortcut:

User Defined Contour

This command activates the mode of closed contour selection manually. It is used when contours cannot be defined automatically. For contour selection click mouse on contour elements one by one. To remove contour click mouse inside the contour area in **Delete** mode .

Shortcut:

Library menu

These menu commands allow you to work with libraries.

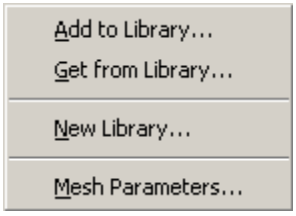


Fig. 3.3 Library menu

Add to Library

This command enables a dialog box, which allows you to add created section to cross-section library.

Get from Library

This command enables a dialog box, which allows you to load created section from cross-section library to editor.

New Library

This command enables a dialog box, which allows you to create new cross-section library.

Mesh Parameters

This command enables a dialog box, which allows you to set mesh type and number of section elements.

Cross-section creation

The cross-section can be created in four ways:

- Create in the cross-section editor.
- Open in cross-section editor earlier created file in *.agr format.
- Import to the cross-section editor through *.dxf file format.
- Insert from section database in the form of parametrical model.

Let's consider each ways in detail.

1. Process of section creation consists of several stages. Cross-section seems to be a two-dimensional area, and two-dimensional area is determined in editor as set of contours. The contour is the closed curve consisting of basic elements. First, it is necessary to draw section contours, using drawing tools such as line segments, arcs, etc. The next step is contours specifying by **Contour / Simple contour** and **Contour / User Defined Contour** commands. The resulting surface is filled with grey color. Explanatory example is given in a Fig. below.

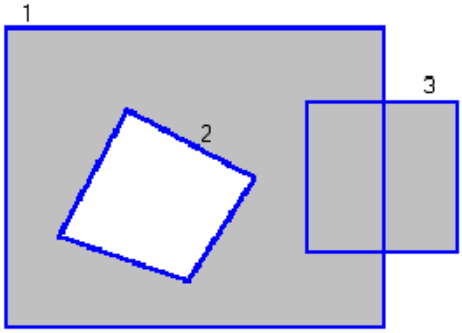


Fig. 3.4 Example of cross-section and contours forming it

You can save document at any time using **Save** (File menu, File toolbar or Ctrl+S) or **File / Save As** command. Cross-section file can be saved in *.wcr (cross-section file) or *.agr (APM Graph file).

2. **Open** button (*File* menu, *File* toolbar or Ctrl+O) allows to open a file saved before (*.wbc cross-section file or *.agr APM Graph file).

3. Section import from *.dxf file format is carried out by **File / Import** command.

4. At section creation, it is possible to load available parametrical models created in *APM Graph*. Databases are managed by *APM Base* kernel (databases management system). The standard *APM Graph* command **Draw / Block / Insert Object from Database** is used for access to databases. Parametrical sections are located in *APM Mechanical Data* and *APM Section Data* (`C: / Documents and Settings / All Users / Application Data / APM Winmachine / DataBase`).

It is possible to load parametric model from *.agp file. Use **Draw / Block / Insert Block** command.

It should be noted that any block or parametric model needs to be unblocked before contour selection. Use **Modify / Explode Block** command for unblocking.

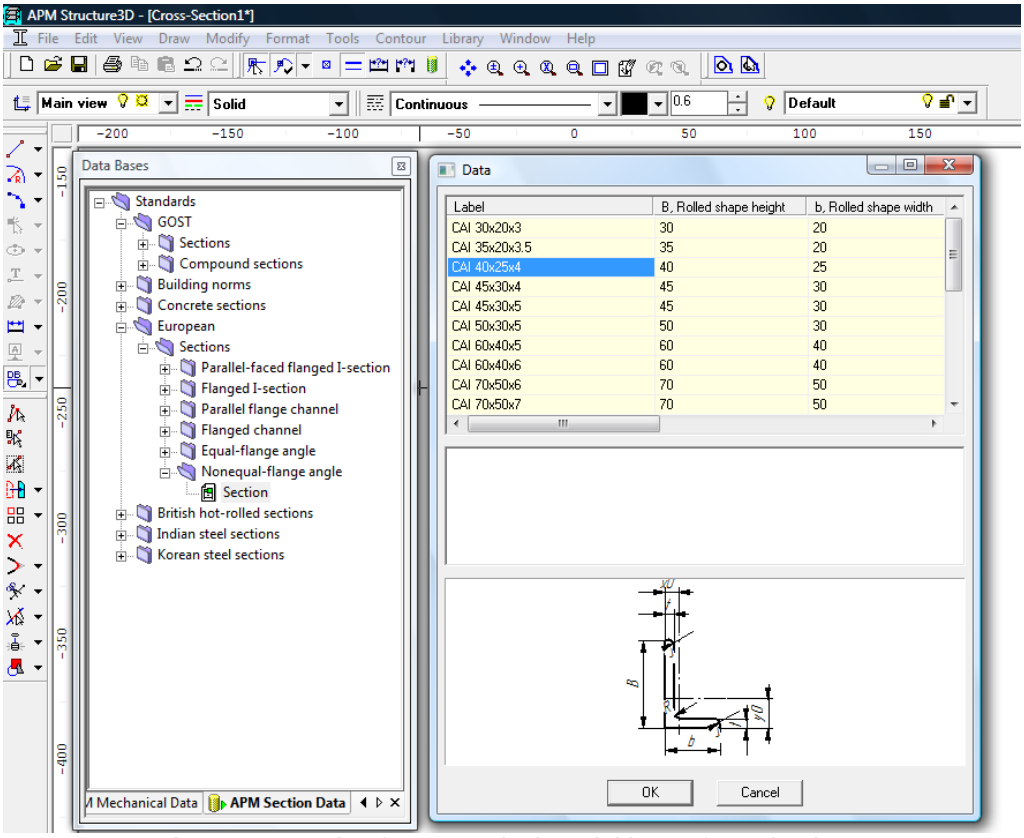


Fig. 3.5 Example of parametrical model insert from database

Cross-section library

Addition new section to library

APM WinMachine supply libraries of standard sections, which are enlarged and updated. Section libraries are installed to *APM WinMachine* directory (by default `C:\Program Files\APM WinMachine)`. Cross-section editor tools allow user to create user libraries that can be used for non-standard sections.

After a section is drawn, it is necessary to select **Library / Add to Library** command to add it to the library. On the screen there will be a dialog box shown below.

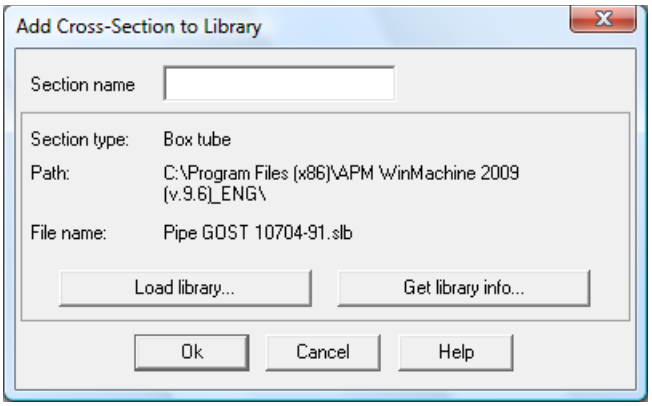


Fig. 3.6 Add Cross-Section to Library dialog box

Except for section drawing commands, commands for creation of cross-sections library are added in the editor.

Loading cross-section from library

For loading cross-section from a library, select **Library / Get from Library** command. Dialog box *Library* shown below will appear on the screen. This dialog box allows you to load cross-section from library, create a new library, delete a section from the library and make cross-sections exchange between two libraries.

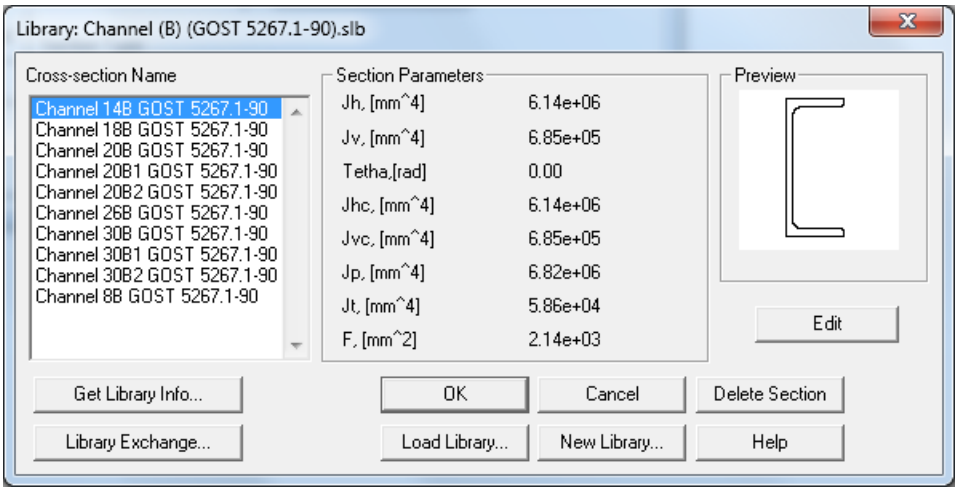


Fig. 3.7 Library dialog box

For loading cross-section, select its name in the list-box and press **OK**. *Section parameters* are shown in dialog box center, cross-section *preview* is shown on the right. For library selection, press **Load library** button. To delete cross-section, press **Delete section** button. **New Library** button allows you to create new library. This action is identical to **New Library** menu command. **Get Library Info** button calls a dialog box showing library information. **Library exchange** button allows you to import and export sections between two libraries.

Editing cross-section geometrical parameters

Necessary geometrical parameters of section are calculated automatically at section addition to library and can be edited. For editing section select **Library / Get from library** menu command. There will be *Library* dialog box on the screen. Press button **Edit** for modifying section parameters. Set demanded parameters in *Section Parameters* dialog box.

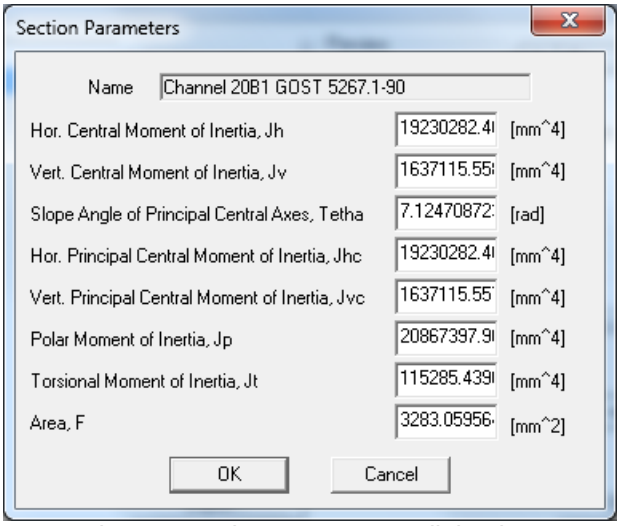


Fig. 3.8 Section Parameters dialog box

There is a triangular finite element meshing during new section geometrical parameters calculation. By default, the approximate number of final elements equals 3200 for *uniform mesh* and 600 for *irregular mesh*. User can change meshing parameters of section having selected **Library / Mesh Parameters** main menu command. This command enables a *Mesh Parameters* dialog box where you can enter the required values.

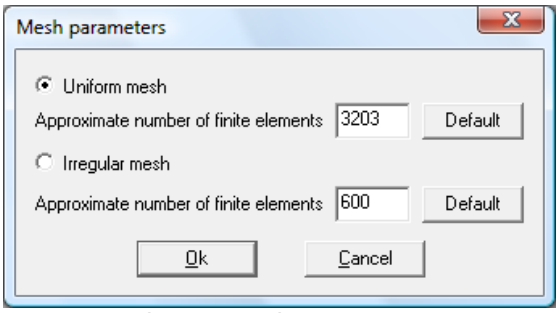


Fig. 3.9 Mesh parameters

The default meshing parameters correspond to a solved problem and do not need to be corrected in most cases.

Exchange between libraries

It is necessary to press **Library exchange** button for sections exchange between libraries. This command calls a dialog box shown below.

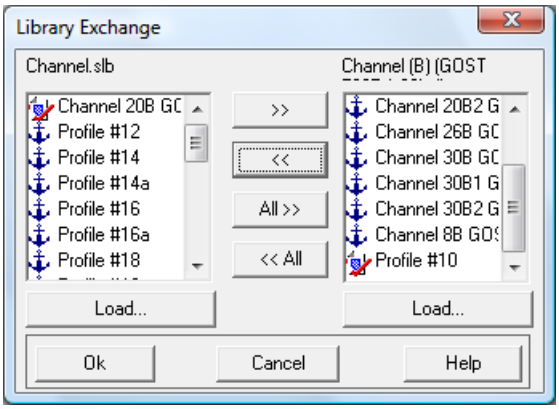


Fig. 3.10 Library Exchange dialog box

Load button allows to load library into a corresponding part of box (left or right).
>> button copies selected sections from left library to right.
<< button copies selected sections from right library to left.
All >> button copies all sections from left library to right.
<< All button copies all sections from right library to left.

 This icon means that section belongs to that library.
 This icon means that section was imported from another library.

Cross-section library creation

Different ways let you create new library:
1. **Library / New Library** menu command.
This command will call a *New Library* dialog box shown below.
2. **New library** button in *Library* dialog box.

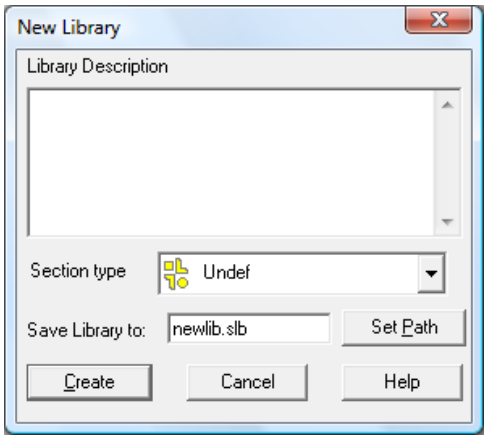


Fig. 3.11 New Library dialog box

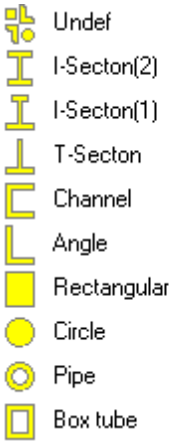


Fig. 3.12 Section types

Use *Save library to* edit box to enter library file name. **Set Path** button sets path to the file.
Library Description edit box allows you to write additional information about the library.

Chapter 4. Calculations

This Chapter describes different calculations performed in *APM Structure3D*.
APM Structure3D allows you to make the following types of calculation:

- Linear Static analysis
- Buckling analysis
- Modal analysis
- Transient Dynamic analysis
- Nonlinear analysis (geometric, physical and general with contact interaction)
- Steady-State and Transient Heat Transfer analysis
- Fatigue calculation
- Seismic calculation
- Code combination calculation
- Design calculation
- Transient heat transfer
- Constant current analysis.
- Electrostatic analysis.
- Magnetostatic analysis.
- Nonstationary electromagnetic analysis.
- High-frequency modal analysis.

Brief description of all types of calculation is given in this chapter. Detailed theoretical examination can be found in books on finite element method and buildings design.

Linear Static analysis

Static calculation is based on the matrix displacement method, in which unknown node displacements are obtained. The main equation for this method is the equilibrium equation $K \cdot x = F$, where *K* – stiffness matrix of the system, *F* – external loads vector, *x* – vector of unknown node displacements. Dimension of this system is equal to the number of degrees of freedom of the structure. In general case, every node has 6 degrees of freedom (3 linear and 3 angular). Solving this system, i.e. after we obtain all node displacements of the structure we find all other unknown parameters, such as deformations, stresses and internal forces.

In static calculation it is assumed that structure scheme is undistorted, and longitudinal loads acting in rods and plates does not affect bending moments.

You can obtain the following results after static calculation:

- Linear and angular node displacements
- Loads at rod ends, plate nodes and solid elements
- Stresses acting in rods, plates and solid elements
- Stress distribution in arbitrary cross-section of any rod
- Force diagrams for entire structure
- Specified parameters for separate beam such as: bending moments, torsion moments, lateral and axial forces, bending and torsion angles, stresses and strains along beams length. All these parameters are represented in the form of graphs and plotted in local rod coordinate system. You can obtain both relative deformation (displacements comparative to the line connecting two deformed edges of rod) and deformation in global coordinate system. In case of a structure consisting of only one rod, relative and total deformation are the same
- Reactions (forces and moments) acting in supports
- Total construction mass

Code combination

Definition of code combination is made for design elements only. Code combination purpose is definition for each design element of such combinations based on existing load combinations, under which certain parameters tend to their respective extremum. Thus, the group of 30 criteria including values of normal and tangent stresses in characteristic points, and also axial and radial forces are considered. Selection principles of combinations and factors, with which combinations enter in code combinations, comply with those stated in building regulations. Calculation represents a search for all load combinations for each design element, and definition of criteria values, resulting from these load combinations. Then, the extremum criteria value is selected which represent the result of the calculation, together with the respective code combination. Calculation results can be used in calculation of bearing capacity of design elements.

Buckling analysis

Buckling calculation (by Euler) is applicable for structures in which elements are in a straight state under given loading, that is they work in tension or compression. For every structure with a given loading diagram, there is fixed load value, under which initial equilibrium shape becomes unstable. Another stable – strained – state becomes possible. When the structure gets out of its initial equilibrium shape we can say that the structure has buckled by Euler. The corresponding load that allows new steady shape is called critical load.

During calculation, all loads are reduced to nodes. Nodal loads vector is represented as $P = p \cdot F$, where *p* is a scalar, called loading parameter, *F* – external loading vector. Thus we consider simple loading, when all loads increase proportional to loading parameter *p*.

Result of buckling calculation is buckling safety factor that shows how many times more one need to increase external load (all force factors simultaneously) to make the system buckle, and buckling shape. Buckling calculation is made together with static calculation as far as we need to know axial forces in rods and stresses in plates that are received during static calculation. Buckling calculation (by Euler) as well as static calculation is carried out regarding to strainless structure.

Mathematically buckling analysis is defined as eigen-value problem.

$$(K + \lambda \cdot L) \cdot \overline{\Delta} = 0$$

where K is stiffness matrix, L is geometry matrix, λ is loading bottleneck and Δ is eigen displacement vector.

Modal analysis

Eigen frequencies calculation is carried out using distributed mass matrix. Calculation is based on solving generalized eigenvalue problem.

$$(-\omega^2 \cdot M + K) \cdot \overline{\Delta} = 0$$

where ω - circular eigen frequency, M - mass matrix, K - stiffness matrix, Δ - eigen shape vector.

Nonlinear analysis

In linear static analysis we assumed displacements to be small, but in many cases this assumption leads to inaccurate or wrong results even for strains remaining in linear elastic region. It is necessary to include geometrical nonlinearity into calculations in order to determine accurate displacements. I.e. membrane stresses can significantly decrease displacements in plate structures under bending loading, although they are usually excluded from computations. In some other cases displacements could be bigger than those predicted by linear theory – important for load-carrying ability calculation.

For linear static calculation we used following relationship between strain and nodal displacements for element:

$$\{\varepsilon\} = [B_0] \cdot \{q\}$$

In nonlinear calculation we assume that strain depends nonlinearly on nodal displacements:

$$\{\varepsilon\} = ([B_0] + [B_{NL}(q)]) \cdot \{q\}$$

This nonlinear component occurs from full expression for strain tensor:

$$\varepsilon ij = (Ui,j + Uj,i + Uk,i*Uk,j)/2$$

Using this nonlinear relationship and equating external and internal forces for finite element assembly we obtain nonlinear matrix equation that can't be solved at once. To obtain solution we modify this matrix equation and apply iterative Newton-Raphson procedure. Final equation includes stress-stiffening (geometry) matrix used in buckling analysis and large displacement matrix.

For solution accuracy estimation we use maximum value of internal loads discrepancy. User can control solution process by defining accuracy of calculation and maximum number of iterations.

Transient Dynamic analysis

Forced oscillation calculation implies that structure is acted upon by loads that vary in time according to a certain law. Main equilibrium equation that describes system behavior is

$$M \cdot \ddot{\Delta} + C \cdot \dot{\Delta} + K \cdot \Delta = P(t)$$

where M - mass matrix, C - damping matrix, K - stiffness matrix, Δ - node displacements vector, $P(t)$ - time dependent external loading vector. Damping matrix C is approximated as linear combination of K and M matrices:

$$C = \beta_1 K + \beta_2 M$$

where coefficients β_1 and β_2 are chosen so as to make oscillation decrements at eigen frequencies constant. This provides frequency independent damping that is typical for most construction materials and structures.

According to equilibrium equation notation all loads vary according to the same law. This equation is solved using mode shape decomposition:

$$\Delta = \sum_{i=0}^{n} v_i q_i(t)$$

where v_i - vector of ith mode shape, q_i - generalized displacements of ith mode shape.

Base matrix equation is reduced to n independent equations:

$$\ddot{q}_i + \gamma \dot{q}_i + \omega_i^2 q_i = R_i(t)$$

where ω_i -eigen frequencies for i^{th} mode shape, γ - damping coefficient, that can be represented as $\gamma = \frac{\delta}{\pi}$, where δ - logarithmic decrement of oscillation, equal to the logarithm of the ratio of amplitude at some moment in time to amplitude after oscillation period $\delta = \ln \frac{A_t}{A_{t+1}}$.

$M_i = v_i^T M v_i$ - generalized mass for i^{th} mode shape

$Q_i = v_i^T P$ - generalized load for i^{th} mode shape

Calculation is done under the assumption of zero initial conditions with the help of Duhamel integral.

$$q_i = A_i e^{-n_i t} \sin(\omega_i t + \varphi_{0i}) + \frac{1}{M_i \omega_i} \int_0^t Q_i(t) e^{-n_i (t-\tau)} \sin(\omega_i (t-\tau)) d\tau ,$$

where A_i and φ_{0i} are determined out of initial conditions,

$$n_i = \frac{\pi \gamma}{T_i} = \frac{\gamma \omega_i}{2} = \frac{\delta \omega_i}{2\pi}$$

For most structures consideration of different shapes of calculation is decreased with increase of frequency number *i*. For practical calculations consideration of first 3-5 mode shapes is sufficient. Use command **Loads / Graph of Dynamic Load** to define load – time diagram.

To make calculation it is necessary to indicate the following parameters:

- Load-time diagram

- Logarithmic decrement for oscillations
- Number of mode shapes considered in calculation
- Period of calculation
- Number of time design moments
- Results of forced oscillation calculation are:
- Nodes displacements
- Stresses acting in rods, plates and solid elements
- Base reactions
- Eigen frequencies and mode shapes

The direct integration method
The direct integration assume**s** that all the matrices are linear not over the whole range but only over **short periods of time Δt.** And the equilibrium equation is considered only at discrete points in the time interval.

It is assumed that at time t = 0 all the vectors Δ , $\dot{\Delta}$ $\ddot{\Delta}$ are known, and to find a solution during the time interval T, this interval is divided into n equal time intervals Δt: Δt = T / n

When using the **Habolt method** you find a solution of the equation $M \cdot \ddot{\bar{\Delta}} + C \cdot \dot{\bar{\Delta}} + K \cdot \bar{\Delta} = P(t)$ for the moment of time t + Δt. So in order to use this method you must know the solution of the basic equation for the two moments of time t and t + Δt. We do not usually know this solution. Therefore, to find these two initial points **θ-Wilson method** is used.

The **θ-Wilson method** assumes a linear change in acceleration in the time interval from t to t + Δt. This method does not require a special starting procedure (as the **Habolt method**) because movement, velocity and acceleration at time t + Δt are expressed in the same rate as for the time moment t.

These methods provide absolute stability (convergence) in comparison with other methods.
Direct integration method advantages:

- Does not restrict a range of tasks;
- Various nonlinear tasks including contact elements with gaps etc. can be solved;
- Materials with different damping characteristics can be used;
- A damper can be easily installed at any node or in various directions;
- It does not need precalculations of natural frequency;
- The accuracy of forced oscillations calculation can be increased by reducing an integration step in this method.

Fatigue calculation

After static calculation, it is possible to lead fatigue calculation. Parameters are set in the dialog box called from **Calculation / Fatigue calculation / Definition of safety factor** menu.

To perform such calculation, we find ranges of limiting values of normal and tangent stress components in points of design, which are designated as:

$$\sigma_{x\max}, \sigma_{x\min}, \ \sigma_{y\max}, \sigma_{y\min}, \sigma_{z\max}, \sigma_{z\min}$$

$$\tau_{xy\max}, \tau_{xy\min} \ \tau_{xz\max}, \tau_{xz\min}, \ \tau_{yz\max}, \tau_{yz\min}$$

Calculations of equivalent stresses:

$$\sigma_{ex}^{*} = \sigma_{amx} K + \psi_{\sigma} \sigma_{mx}$$

Long-term strength value on normal stresses:

$$\sigma_{-1} = (0.55 - 0.0001\sigma_b)\sigma_b$$

Long-term strength value on tangent stresses

$$\tau_{-1} = (0.5 \div 0.6)\sigma_{-1}$$

Correction factor of geometry, material and processing:

$$K = \left(\frac{K_\sigma}{K_{d\sigma}} + \frac{1}{K_{F\sigma}} - 1 \right) \frac{1}{K_V}$$

Busy hour-to-day ratio

$$K_\sigma = 1 + q(\alpha_{c\sigma} - 1)$$

$q = 0.15$ - cast iron,

$q = 0.5$ - low-carbon и low-alloy steels,

$q = 0.65$ - alloy steels,

$q = 0.9$ - high-alloy steel

here *q* - sensitivity coefficient of material to local stresses and $\alpha_{c\sigma}$ - theoretical stress concentration factor. These parameters are set in *Fatigue calculation* dialog.

$$K_{F\sigma} = 1 - 0.22 \lg Rz \left(\lg(\frac{\sigma_b}{20} - 1) \right)$$

here Rz - surface roughness Rz, is set in *Fatigue calculation* dialog.

$$K_{F\tau} = 0.575 K_{F\sigma} + 0.425$$

$K_{d\sigma} = 1$ - scale factor, is set in *Fatigue calculation* dialog.

$K_V = 1$ - strengthening factor is set in *Fatigue calculation* dialog.

Strength condition

$$\sigma_e = \frac{s}{\sqrt{2}} \sqrt{\left(\sigma^*_{ex} - \sigma^*_{ey}\right)^2 + \left(\sigma^*_{ey} - \sigma^*_{ez}\right)^2 + \left(\sigma^*_{ez} - \sigma^*_{ex}\right)^2 + 2\frac{\sigma_{-1}}{\tau_{-1}}\left(\tau^{*2}_{exy} + \tau^{*2}_{eyz} + \tau^{*2}_{ezx}\right)} = \sigma_{-1}$$

Safety factor of long-term strength

$$s = \frac{\sqrt{2}\sigma_{-1}}{\sqrt{\left(\sigma^*_{ex} - \sigma^*_{ey}\right)^2 + \left(\sigma^*_{ey} - \sigma^*_{ez}\right)^2 + \left(\sigma^*_{ez} - \sigma^*_{ex}\right)^2 + 2\frac{\sigma_{-1}}{\tau_{-1}}\left(\tau^{*2}_{exy} + \tau^{*2}_{eyz} + \tau^{*2}_{ezx}\right)}}$$

Cycle number definition

In addition time of one cycle of load t_c is to be entered

$m = 5 + \left(\frac{\sigma_b}{80} \right)$ - exponent of curve endurance

$N_G = 2\ 10^6$ - base number of load cycles

Admissible number of load cycles at level of external stresses σ_e

If $\sigma_e \le \sigma_{-1}$ - operating time is not limited

$N = N_G \left(\frac{\sigma_{-1}}{\sigma_e} \right)^m$ if $\sigma_e \ge \sigma_{-1}$ in this case operating time $t = t_c \cdot N$

Results of fatigue calculation are accessible after static calculation at viewing *Results Map*.

σ_{-1} $\sigma_{-1} = (0{,}55 - 0{,}000\,1 \cdot \sigma_{\text{b}}) \cdot \sigma_{\text{b}}$, $\tau_{-1} = 0{,}8 \cdot \sigma_{-1}$, $N_G \sigma_{-1} 2 \cdot 10^6 10^8 \sigma_{-1} N$ $\tau_{-1} = 0{,}55 \cdot \sigma_{-1}$.

$= \begin{cases} N_G \cdot \sigma_{-1}^m / \sigma^m, & \sigma > \sigma_{-1} \\ \infty, & \sigma \le \sigma_{-1} \end{cases} \begin{cases} \sigma^{\text{m}} \cdot N = \text{const}, & 10^3 < N \le N_G \\ \sigma = \sigma_{-1} = \text{const}, & N > N_G \end{cases}$ m

$= \frac{1}{K}$

$\cdot \left(5 + \frac{\sigma_{\text{b}}}{80}\right) \sigma_{\text{b}} N_G$ $\left.\begin{matrix} \sigma_{\text{м}} = \frac{1}{2}(\sigma_{\text{н}} + \sigma_{\text{к}}), & \tau_{\text{м}} = \frac{1}{2}(\tau_{\text{н}} + \tau_{\text{к}}) \\ \sigma_{\text{ам}} = \frac{1}{2}(\sigma_{\text{н}} - \sigma_{\text{к}}), & \tau_{\text{ам}} = \frac{1}{2}(\tau_{\text{н}} - \tau_{\text{к}}) \end{matrix}\right) \left.\begin{matrix} \sigma_{\text{ам}} = \sigma_{-1} - \psi_\sigma \cdot \sigma_{\text{м}} \\ \tau_{\text{ам}} = \tau_{-1} - \psi_\tau \cdot \tau_{\text{м}} \end{matrix}\right\} \psi_\sigma \psi_\tau \sigma_{\text{м}} \sigma_{\text{b}}$ $\psi_\sigma = 0{,}02 + 0{,}000\,2 \cdot \sigma_{\text{b}}$, ψ_σ $\psi_\tau = 0{,}5 \cdot \psi_\sigma$,

$= 0{,}48 + 0{,}000\,55 \cdot \sigma_{\text{b}} \sigma_{\text{b}} \bar{x} = \frac{1}{N} \sum_i^N x_i$, $D = \frac{1}{N-1} \sum_i^N (x_i - \bar{x})^2$, $STD = \sqrt[2]{D}$ As

$= \frac{1}{N} \cdot \frac{1}{\left(\sqrt{D}\right)^3} \cdot \sum_i^N (x_i - \bar{x})^3$, As0 $= \sqrt{\frac{6 \cdot (N-1)}{(N+1) \cdot (N+3)}}$, Ex $= -3 + \frac{1}{N} \cdot \frac{1}{D^2} \cdot \sum_i^N (x_i - \bar{x})^4$, Ex0

$= \sqrt{\frac{24 \cdot N \cdot (N-2) \cdot (N-3)}{(N-1)^2 \cdot (N+3) \cdot (N+5)}}$. $\sigma^{\text{R}} = \text{K} \cdot \sigma_{\text{ам}} + \psi_\sigma \cdot \sigma_{\text{м}}$ $\psi_\sigma \psi_\tau \sigma_{\text{э}1}$ $\tau^{\text{R}} = \text{K} \cdot \tau_{\text{ам}} + \psi_\tau \cdot \tau_{\text{м}}$

$= \sqrt{\frac{\left(\sigma_x^{\text{R}} - \sigma_y^{\text{R}}\right)^2 + \left(\sigma_y^{\text{R}} - \sigma_z^{\text{R}}\right)^2 + (\sigma_z^{\text{R}} - \sigma_x^{\text{R}})^2}{2} + \left(\frac{\sigma_{-1}}{\tau_{-1}}\right)^2 \cdot \left((\tau_{xy}^{\text{R}})^2 + (\tau_{yz}^{\text{R}})^2 + (\tau_{zx}^{\text{R}})^2\right)} + \cdots$ $\sigma_{-1} \tau_{-1} \sigma_{\text{э}1} \sigma_{\text{э}1} \sigma_{\text{э}1}^{\max} \text{n}_{\text{э sin}}$ n_i / N_{0i} $\text{n}_i N_{0i}$ $\sum_{\substack{i=1 \\ K \cdot \sigma_i \ge \sigma_{-1}}}^{N_h} \frac{\text{n}_i}{N_{0i}} = \alpha_{\text{э}}$, $K \cdot \sigma_i$

$\ge \sigma_{-1} 10^3 < N_i < N_G N_{0i} = \frac{\sigma_{-1}^{\text{m}} \cdot N_G}{\sigma_i^{\text{m}}}$. $\sum_{\substack{i=1 \\ K \cdot \sigma_i \ge \sigma_{-1}}}^{k} \sigma_i^{\text{m}} \cdot \text{n}_i = \sigma_{-1}^{\text{m}} \cdot N_G \cdot \alpha_{\text{э}} = (\sigma_{\text{э}1}^{\max})^{\text{m}} \cdot \text{n}_{\text{э sin}} \cdot \alpha_{\text{э}}$, $\text{n}_{\text{э sin}}$

$= \frac{1}{\alpha_{\text{э}}} \sum_{\substack{i=1 \\ K \cdot \sigma_i \ge \sigma_{-1}}}^{k} \text{n}_i \cdot \left(\frac{\sigma_i}{\sigma_{\text{э}1}^{\max}}\right)^{\text{m}}$ (7). $\sigma_{\text{э}1}^{\max} N_{0\text{э}1\max} = \frac{\sigma_{-1}^{\text{m}} \cdot N_G}{\sigma_{\text{э}1}^{\max}}$. $N_{0\text{э}1\max} \sigma_{\text{э}1}^{\max}$

$< \sigma_{-1}/K\, \sigma_{\text{э}1}^{\max} 10^3 < N_i < N_G n_N = \frac{N_{0\text{э}1\max}}{\text{n}_{\text{э sin}}}$. $\text{n}_{\text{э sin}} \le 1 \sigma_{\text{э}1}^{\max} \le \sigma_{-1}/K\, \sigma_{\text{э}1} \text{n}_{\text{э sin}} \sigma_{\text{э}1}$

Setting of fatigue parameters

For setting of fatigue material properties the point "Calculations" -> "Parameters of a fatigue calculation..." must be selected in the main menu. In the open dialog with the same name bookmarks for a reference to corresponding fatigue property fields, are available, Fig. 4.41.

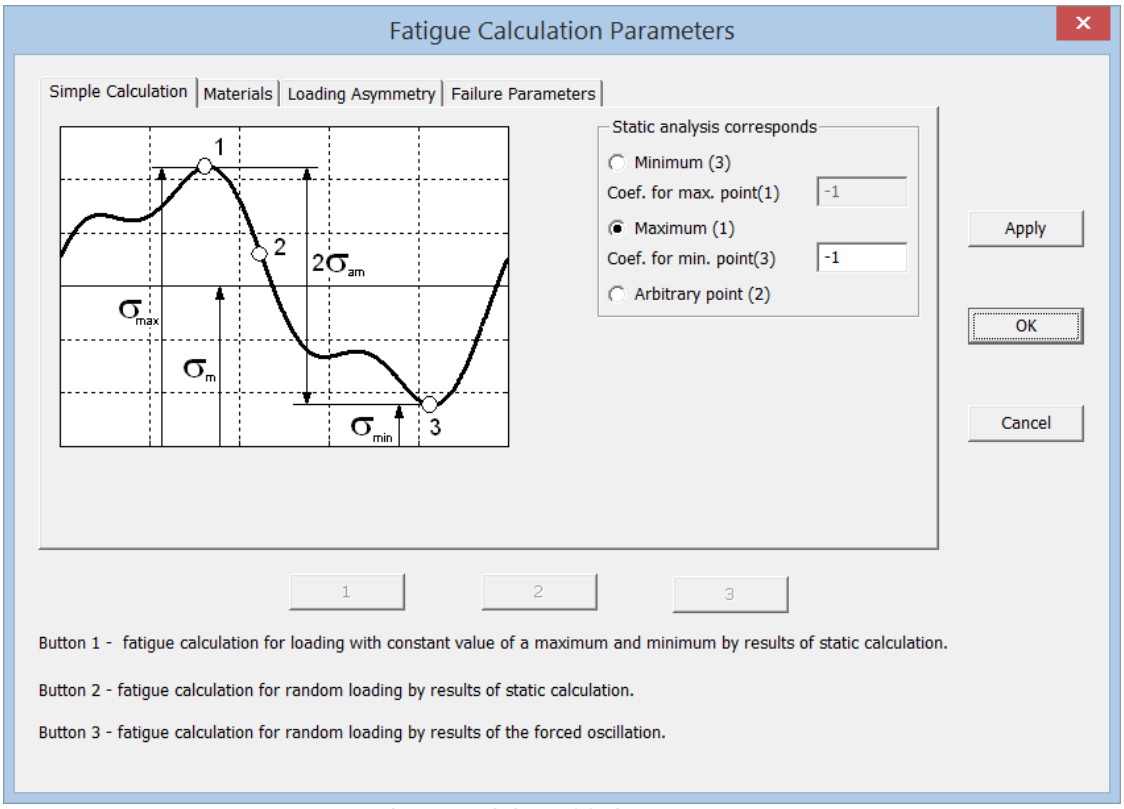


Fig. 4.1 *Dialog of fatigue parameters setup.*

The tab "Simple Calculation" contains an explanatory scheme for definition of parameters in the group "Static Calculation Corresponds". All the fatigue methods of a calculation are based on the fact of the model's being assumed to work in an elastic zone. If during the calculation of stress in the node of one of the finite elements it exceeds the value of the limit of variability, a warning, Fig.s 4.2, will be given.

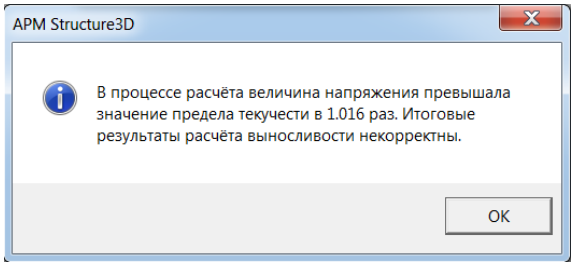


Fig. 4.2 *the Warning about exceeding in the calculation of the yield stress.*

The results of the «Simple Method» calculation are available after a static calculation in the point "Fatigue" of the dialog "Results Options", Fig. 4.3.

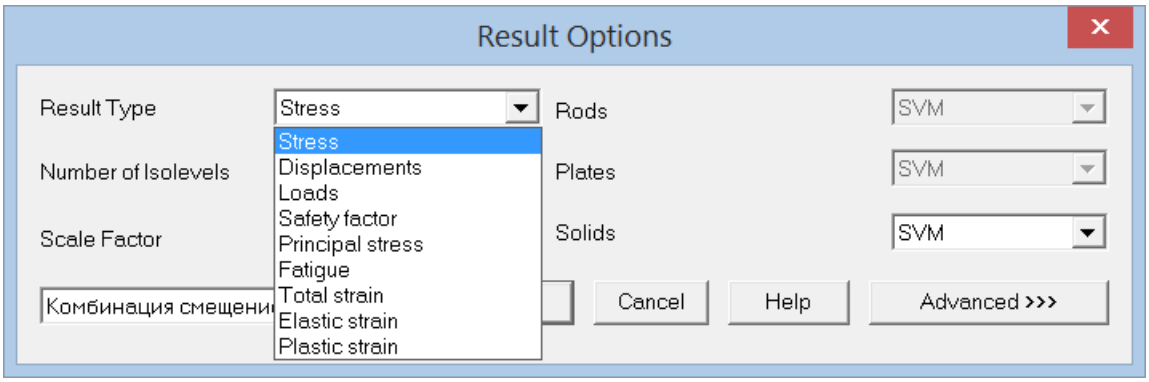


Fig. 4.3 *Choice of simple fatigue calculation results.*

Among the available values calculated after calculating the statics for "Simple Calculation of Fatigue" - is "The Safety Factor" and "Number of Cycles ". The safety factor is defined as the ratio of the fatigue limit, $\sigma_{-1}\sigma_{-1}$to the calculated (the equivalent of the damaging effects of symmetrically loaded) stress in the elements of the construction.

Setting of fatigue strength endurance limit for normal (n) stresses and tangents (k) stresses is specified not in the dialog "Parameters of a Fatigue Calculation" but in the material parameter setup dialog, Fig. 4. 44. If the given fatigue limit values turn smaller than 0,001 MPa, calculation is carried out by the statistical formulae applicable for carbon and low alloyed steels:

$$\sigma_{-1} = (0{,}55 - 0{,}000\,1 \cdot \sigma_{\mathrm{b}}) \cdot \sigma_{\mathrm{b}},$$
$$\tau_{-1} = 0{,}55 \cdot \sigma_{-1}.$$

For brittle materials (high carbon steel, cast iron) the statistical formula is another:

$$\tau_{-1} = 0{,}8 \cdot \sigma_{-1},$$

Therefore in this case computed values must be specified manually and one must set them in corresponding fields of a material properties setup dialog.

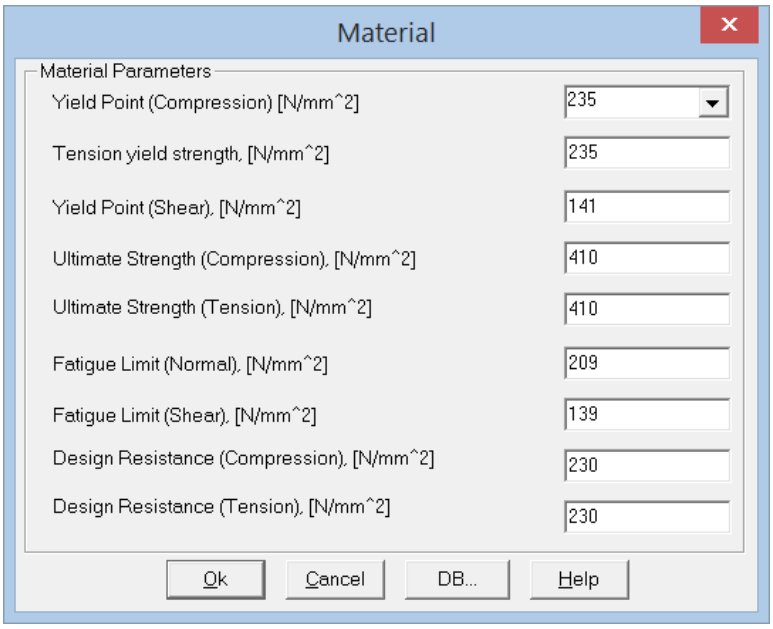


Fig. 4.4 *Dialog of general properties of the material.*

For materials with fatigue curve, reflected on Fig.4.5, the basic number of cycles $N_G N_G$is corresponding to the physical endurance limit$\sigma_{-1}\sigma_{-1}$.

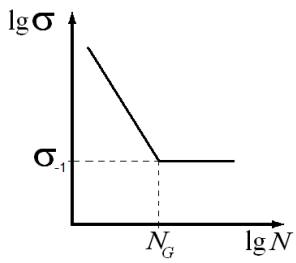


Fig. 4.5 *Wohler's Curve for carbon and middle alloyed steel.*

The base number of N_G cycles is accepted in determination of the fatigue limit for ferrous metals as$2 \cdot 10^6 2 \cdot 10^6$ [1,2,7]. For the materials near which the limit of a horizontal Wohler section of a fatigue curve is not detected, the concept «of the conditional fatigue limit» for which the base number is defined as for loading$10^8 10^8$ cycles is used.

The zone of unlimited fatigue$\sigma_{-1}\sigma_{-1}$ corresponds to the lower stresses and the zone of limited fatigue – to the ***high*** ones. Wohler's curve defines the number of cycles, probabilities of destruction corresponding to 50% for matching stress, definition – symmetric load and the uniaxial intense one of state. One of the analytical descriptions of a curve of stamina:

$$N = \begin{cases} N_G \cdot \sigma_{-1}^m / \sigma^m, & \sigma > \sigma_{-1} \\ \infty, & \sigma \le \sigma_{-1} \end{cases} \quad N = \begin{cases} N_G \cdot \sigma_{-1}^m / \sigma^m, & \sigma > \sigma_{-1} \\ \infty, & \sigma \le \sigma_{-1} \end{cases}$$ Or

$$\begin{cases} \sigma^m \cdot N = const, & 10^3 < N \le N_G \\ \sigma = \sigma_{-1} = const, & N > N_G \end{cases} \begin{cases} \sigma^m \cdot N = const, & 10^3 < N \le N_G \\ \sigma = \sigma_{-1} = const, & N > N_G \end{cases} \quad (1),$$

where m is an exponent depending on material, the quality of machining process and thermo processing [1,2,7] its:

$$m = \frac{1}{K} \cdot \left(5 + \frac{\sigma_b}{80}\right) m = \frac{1}{K} \cdot \left(5 + \frac{\sigma_b}{80}\right) \quad (2),$$

Where *K* is the coefficient of reduction of the fatigue limit, the tensile$\sigma_b \sigma_b$ strength in the MPa.

If the reserve in fatigue coefficient is larger than the unit which corresponds to the area of unlimited stamina, the derived number of cycles is limited with a value specified in the field "Number of Cycles Before a Change of a Fatigue Curve$N_G N_G$" in the tab "Parameters of Destruction" (it is described below by the text). This is done in order not to display the infinite quantities in the charts.

The tab "Materials" of the dialog "Parameters of a Fatigue Calculation" contains a list on the coefficients of the fatigue limit reduction for all materials of an editable model.

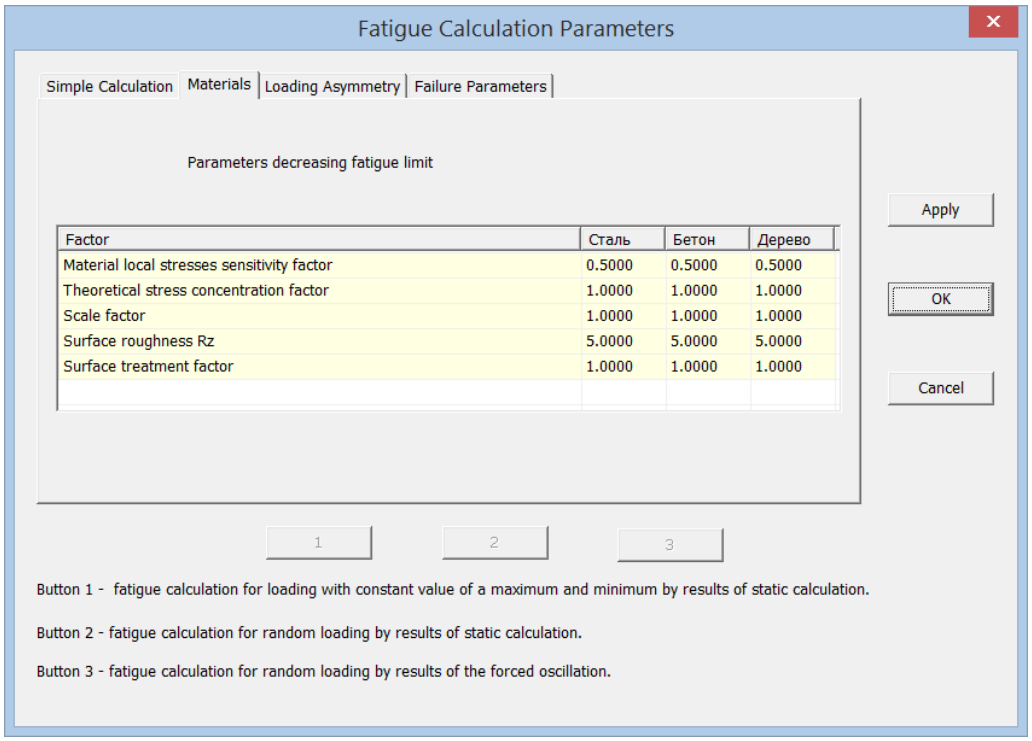


Fig. 4.6 *the Tab of parameters decreasing fatigue limit setup.*

Description of used values and calculation for a final coefficient of reduction of the fatigue limit are described in [1,2].

The content of the tab "Asymmetry of Loading" allows to use a symmetric loading with the indicator of an asymmetry of R = -1 to carry out provision of asymmetric loading.

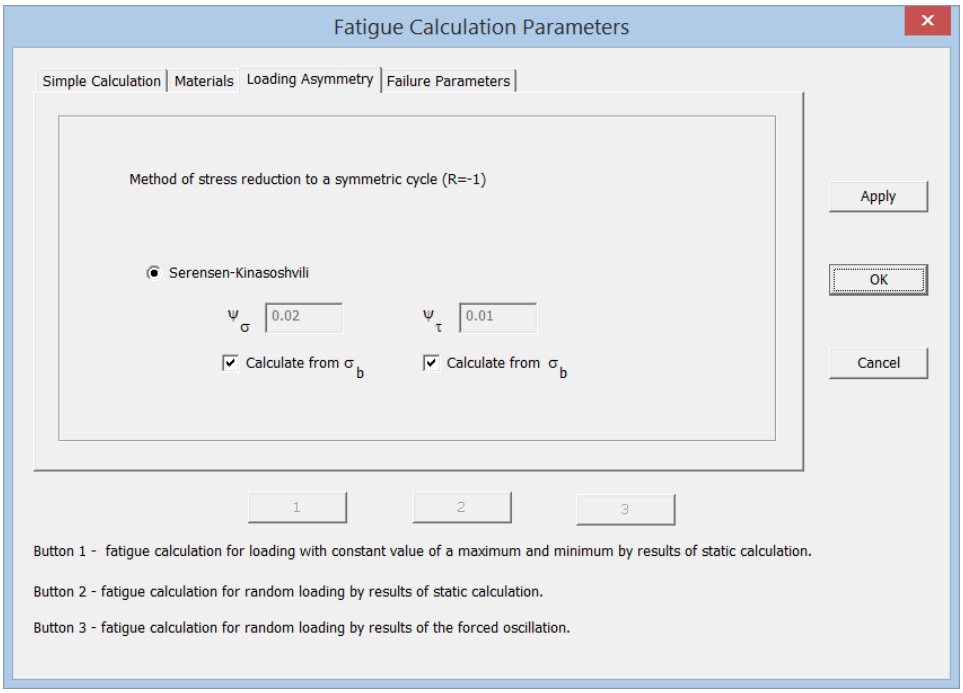


Fig. 4.7 *the Tab of properties of cast to symmetric loading cycle.*

On basis of numerous experiments with samples of defined material R = -1 with use of statistical methods [3] the diagram of the limit of amplitudes limit of stresses is being revealed in coordinates of the average and half-span of stresses of cycle for specified durability.

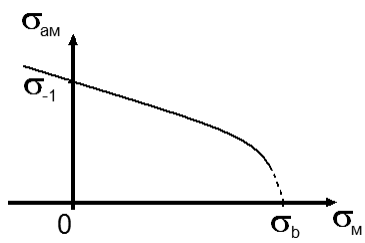


Fig. 4.8 *Diagram of the stresses limit for specified durability.*

See for the formulae of calculation of a stresses mean value (the subscript "м") and amplitude or half-span (the «am» subscript) by**Ошибка! Источник ссылки не найден.**:

$$\left.\begin{aligned}\sigma_м &= \frac{1}{2}(\sigma_н + \sigma_к), \quad \tau_м = \frac{1}{2}(\tau_н + \tau_к)\\ \sigma_{ам} &= \frac{1}{2}(\sigma_н\text{-}\sigma_к), \quad \tau_{ам} = \frac{1}{2}(\tau_н\text{-}\tau_к)\end{aligned}\right\}\left.\begin{aligned}\sigma_м &= \frac{1}{2}(\sigma_н + \sigma_к), \quad \tau_м = \frac{1}{2}(\tau_н + \tau_к\\ \sigma_{ам} &= \frac{1}{2}(\sigma_н\text{-}\sigma_к), \quad \tau_{ам} = \frac{1}{2}(\tau_н\text{-}\tau_к)\end{aligned}\right. \quad (3),$$

Where $_{\Sigma n}$ and $_{\tau n}$ are starting values of stresses, $_{\sigma k}$, and $_{tk}$ – final values obtained in semi-cycle.

For approximation of a diagram of the stresses limit different dependencies one of which – the correlation of Serensen – Kinasoshvili is used:

$$\left.\begin{aligned}\sigma_{ам} &= \sigma_{\text{-}1}\text{-}\psi_\sigma \cdot \sigma_м\\ \tau_{ам} &= \tau_{\text{-}1}\text{-}\psi_\tau \cdot \tau_м\end{aligned}\right\}\left.\begin{aligned}\sigma_{ам} &= \sigma_{\text{-}1}\text{-}\psi_\sigma \cdot \sigma_м\\ \tau_{ам} &= \tau_{\text{-}1}\text{-}\psi_\tau \cdot \tau_м\end{aligned}\right\} \quad (4).$$

The coefficients of sensitivity to an asymmetry of stresses cycle [1]$\psi_\sigma\psi_\tau\psi_\sigma\psi_\tau$ approximate the diagram of the stress limit only in that part of a diagram where the mean values of stresses are$\sigma_м\sigma_м$ far from the tensile strength. A linear dependence of the stress limit is assumed to be upset precisely in the range of fluidity. As has already been described above, a warning about the fact of excess over the limit of variability of calculated stresses equivalent in Mieses in an element of a model, Fig. 4.42 is issued in the program during the calculation.

The tab "Asymmetry of Loading" exhibited switches for coefficients of sensitivity to an asymmetry of cycle «Calculate from$\sigma_b\sigma_b$» allows to make calculations on the following equations:

$$\psi_\sigma = 0{,}02 + 0{,}000\,2 \cdot \sigma_b,$$
$$\psi_\tau = 0{,}5 \cdot \psi_\sigma,$$

Where σ_b is the tensile strength in the MPa.

But this formula is applicable only for steels and for light alloys that are deformed, the value calculated by the formula must be manually specified:

$$\psi_\sigma = 0{,}48 + 0{,}000\,55 \cdot \sigma_b \psi_\sigma = 0{,}48 + 0{,}000\,55 \cdot \sigma_b.$$

The tab "Parameters of Destruction" contains two groups of defined parameters; the Group "Wohler Parameters of a Fatigue Curve" allows fig. 4.49 to define quantities defining a type of the fatigue curve which in the logarithmic coordinates consists of two segments, Fig. 4.45.

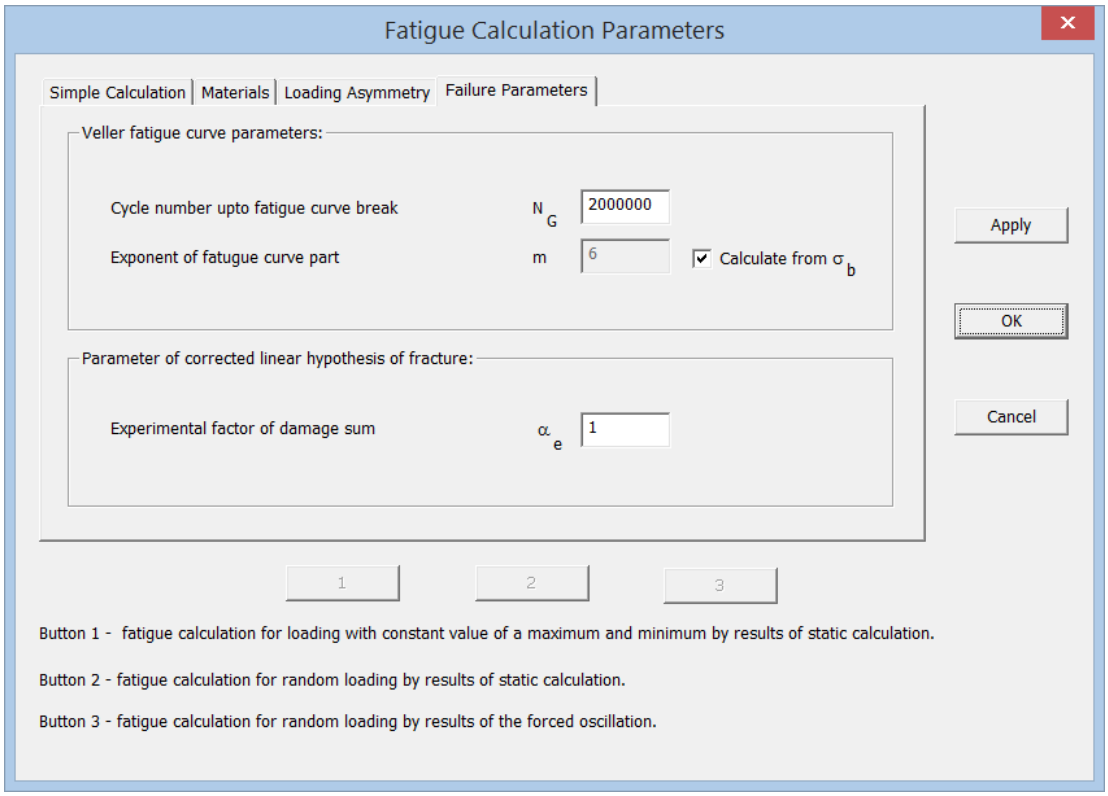


Fig. 4.9 the Tab of failure parameters.

A switch near the field "Exponent of a Segment of the Fatigue Curve" with the name «Calculate from$\sigma_b\sigma_b$» is allowed to specify an exponent by the statistically revealed formula (2).

If the exponent is approximated by the other formula, these calculations must be manually made and specify in a matching field, becoming available after deactivation of a switch that was specified above.

Besides tabs, in the dialog "Parameters of fatigue calculation" some more important buttons are present. Pressing the button "OK" to save of all fatigue parameters specified in the dialog and closing of the dialog. Pressing the button "Apply" leads to the fact that, besides data save, (as for the "OK" button) data received in all methods of the fatigue calculation will be cleared.

Buttons of a calculation start of different fatigue calculations methods are located at the bottom of a dialog that is examined. The "1" and "2" buttons turn out inaccessible for the press before that point until all the following conditions are satisfied:

- windows of results of cards display are absent;
- finite elements type of plate and 8-node three-dimensional element are presented;
- Implemented calculation of statics.

The "3" button will be unavailable, until the same conditions are fulfilled except the latter one, instead of which, it is owing the calculation of forced fluctuations for super-elements (SE) described below is carried out.

Before to press of the "1», «2» and «3» buttons new parameters for fatigue calculations must be saved by a press of an "Apply" button if they were. After a press of one of three buttons the dialog will be closed and a calculation will be made on a chosen algorithm.

The push on "1" button carries out a start for a calculation on the simplest algorithm when load consists of alternating sequences with a constant value of maximum and minimum, see a scheme with the tab "Simplest calculation" of this dialog.

The push on «2» button leads to a start for a calculation of a fatigue algorithm for random load based on the results of a static calculation.

The push on «3» button leads to a start for a calculation of a fatigue algorithm for random load based on the calculation results of forced fluctuations for super-elements.

Definition of random load

Random load is specified for dedicated loading or simultaneously for all loadings. The mechanism is the same as at definition of loading graphs. For a call to the dialog "Fatigue Stochastic Loadings for Load Cases" the menu item "Stochastic Loading...» Fig. 4.50 must be selected in "Loads" submenu.

The status of fatigue load can be as follows:

– "Not specified" when there is no load;;
– «Switched on» when load is set and turned on for a calculation;
– «Switched off» when load is specified, but disconnected for a calculation.

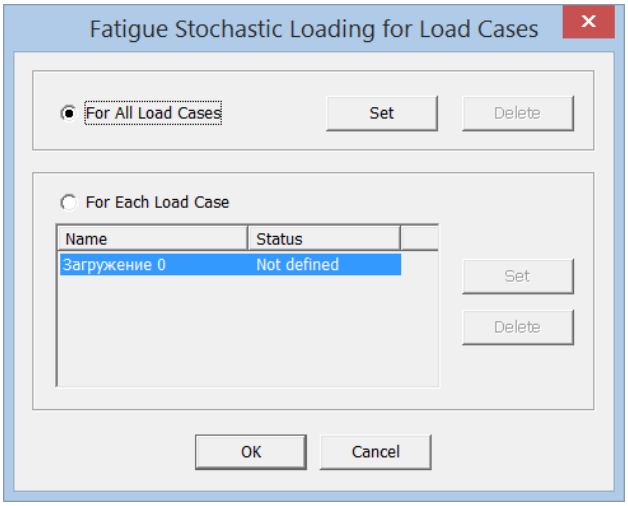


Fig. 4.10 *Dialogue installation of random load for loadings.*

Only with the status "switched on" the block of load loading will be used in a random fatigue calculation. If load is absent, for its setting one must press the "Specify" button for a call to the dialog "Fatigue Multistage Random Loading"; if load has already been specified, the same button will the name of "Change".

If setting of random load is selected «For all loadings», precisely this load will be used in a calculation at the choice of any loading in a calculation dialog, but only under a condition that a graph of load is set in a point «For all Loadings» in the dialog "Graphs of Loads for Loadings". If a calculation must be made with random load specified for particular loading, a graph of load must be removed in the dialog "Graphs of Loads for Loadings" in the point «For all loadings».

The dialog "Fatigue Multistage Random Loading" allows combining several stages of loading each of which possesses its statistical parameters.

Group "General Parameters for a Calculation" fig. 4.51, is allowed to carry out determination of general for all stages parameters. The switch "Switch on for a Calculation" allows specifying from a dialog the status of load for the further calculation.

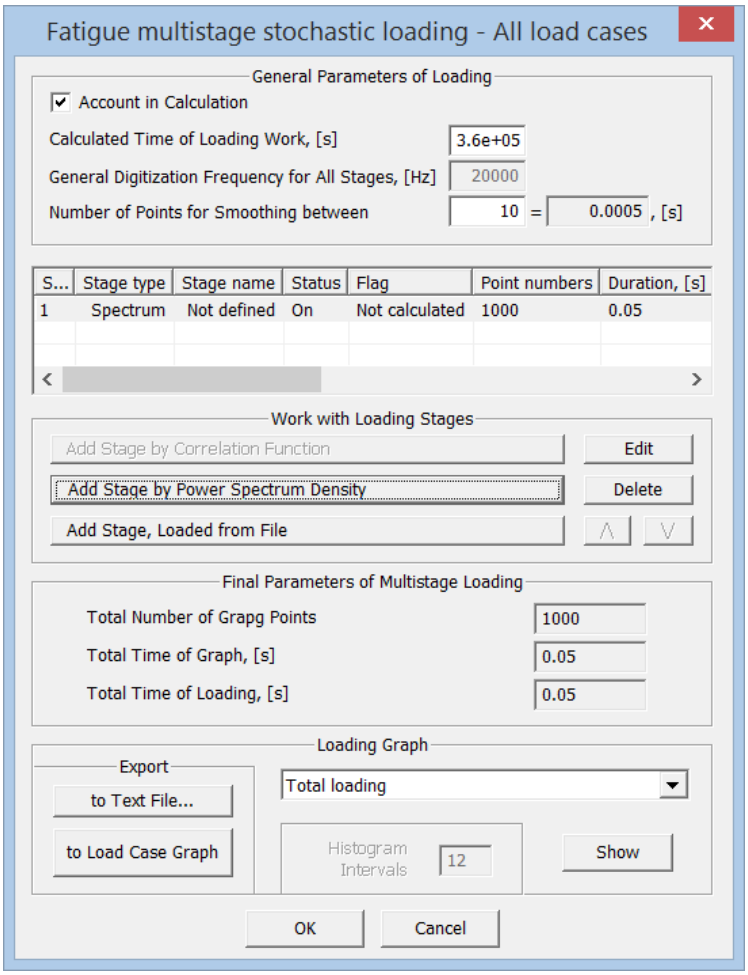


Fig. 4.11 Dialog of definition by a multistage random loading of all loadings; see the name of the dialog.

The parameter "Estimated Time of Loading Operation" responds in necessary time of multistage loading (for example, calculation of vibration). All the random fatigue calculation bases on an assumption about ergodicity of random impact that is synthesized. This property allows to switch from a calculation of a random function in all the wondered interval of time to the realization much lower in time, but possessing the same statistical characteristics. Such a necessity is caused mainly by a desire of calculation time reduction. Therefore the ratio of necessary calculation time to time of specified loading determines the number of repetitions which will subsequently be taken into consideration during calculation of reserve in durability.

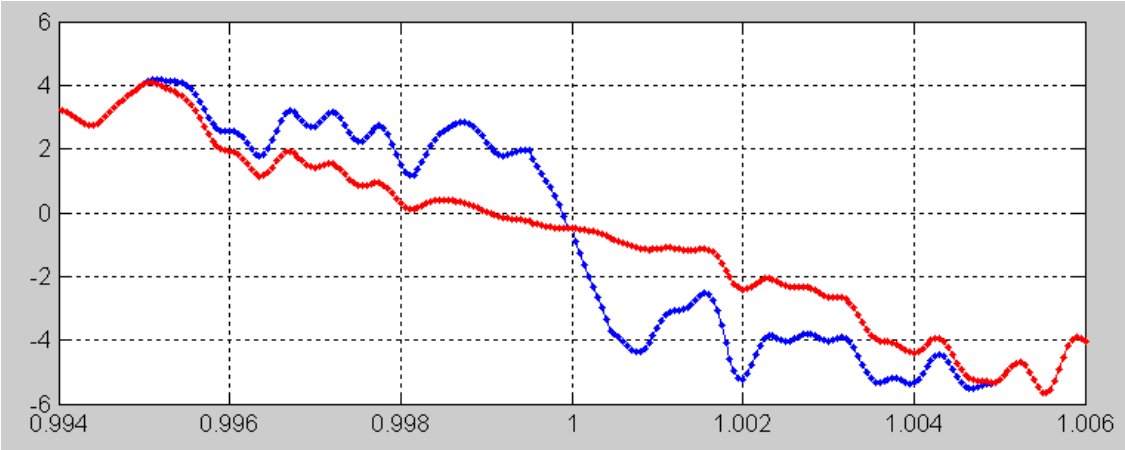
In determination of stresses its maximum magnitude is the defining parameter of load, therefore for correct description of perturbing effect specified frequency it is recommended to specify digitization frequency as exceeding this quantity of frequency by a notch. If the spectrum of perturbing effect is specifying, in the field "General Digitization Frequency" a characteristic increased value of maximum significant frequency this will be specified for one of the loading stages. This parameter is defined before definition of the very first stage and is unavailable if at least one stage is already specified.

The parameter "Number of Points of an Inter-stages Smoothing" allows to specify the time interval where smoothing the joints will be facing each other involved in the calculation of the loading stages. If the stadium is one only, this parameter value is ignored.

The smoothing of joints is being drawn in two stages.

1) The smoothing of the ends (or only one end if the stage is initial or final) of each stage to a zero value with a linear function.

2) An inter-blocks smoothing, at the fact that already smoothed loading ends participate in a linear transition from the level of mathematical expectation (ME) of the previous stage to the ME level of the next stage, Fig. 4.52.



With blue color the number of points of a smoothing is 10, with red color the number of points of a smoothing is 100.
Fig. 4.12 Fragment of the two stages graph with the different number of smoothing points. The first stadium with ME = 3, the second stage with ME = -4.

The smoothing between stages with the different ME level makes sense only at setting of the value of a power factor or movement, when setting the value of acceleration and speed in knots CE, MO value should be zero.

The list of random loading stages allows to clearly carry out comparison of determined statistical characteristics between each other on the below described parameters.

1. The number of a stage.
2. A type of the stage which can be read from a file, specified based on the correlation function and specified through spectral power density.
3. The name of the stage is a field defined by a user, allowing to identify the stages in a list between themselves.
4. The status of the stage speaks of the fact of its being able to be «switched on» and being able to be «switched off» for a calculation. If the status is specified as «switched on», it does not take part in a calculation.
5. The flag of a stage speaks of whether a stage has a resulting graph which is already smoothed at ends.
6. The number of points specifies size of a resulting graph of a stage.
7. Duration in seconds is determined by a ratio of the number of stage points to digitization frequency.
8. The number of repetitions reports about the number of repetitions of a stage of loading graph during a calculation process.
9. Rationing, "Switch on", or "Switch off".
10. KA and KB are introduced rationing coefficients.
11. Statistical parameters that were determined, if synthesis of a graph on a stage of loading was implemented, but if the synthesis was not implemented, the corresponding table columns will be blank.

For the current stage, the control buttons are available for including operations such as editing, deleting and reordering stages, .working with prescribed stages. For setting of a new stage of random loading one must press one of the buttons the name of which starts as "Add a stage...» Three possibilities of setting are implemented:

- A stage specified by the correlation function;
- A stage specified by spectral power density;
- A stage being loaded from a file created by a third-party application in text format.

Consider setting the properties dialog of the new stage, given by power spectral density accelerations Fig. 4.53.

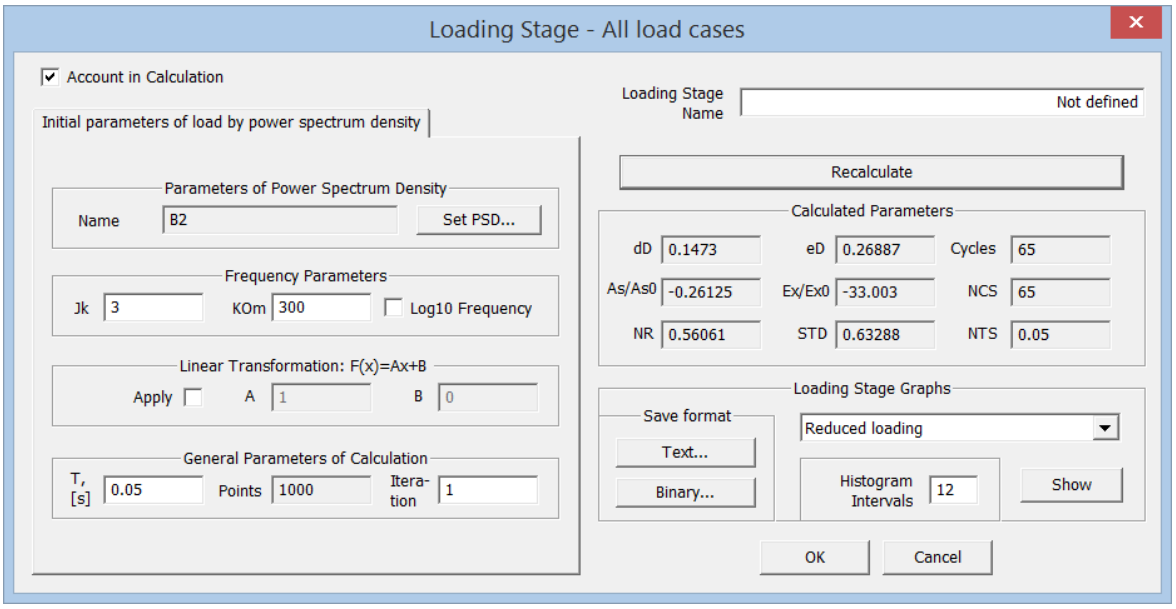


Fig. 4.13 *Dialog loading stage, given by power spectral density.*

The best orientation in a roster of stages from Fig. 4.51 needs the field "Name of a Loading Stage".

The group "Calculated Parameters" will contain blank fields until their calculation is made by a push on the button "Compute". Four below described groups of parameters are input data for a calculation.

The first group "Parameters of Spectral Power Density" contains a button for a call of the dialog which allows to select among the predefined in a program on [5] the spectra which must be implemented for a current stage of loading in the group "PSD Template", Fig. 4.54. There is accessibly in the group "Current Point Value" is able to change the current situation in terms of power spectral density graphics acceleration. In the fields of the group "Point" are accessibly removal of a current point or addition of a nova one before or after a current point. Resulting values characterizing editable PSD are collected in the group "Calculated Parameters". The push on the group buttons "PSD Table" allows to execute save or load of an in advance saved spectrum.

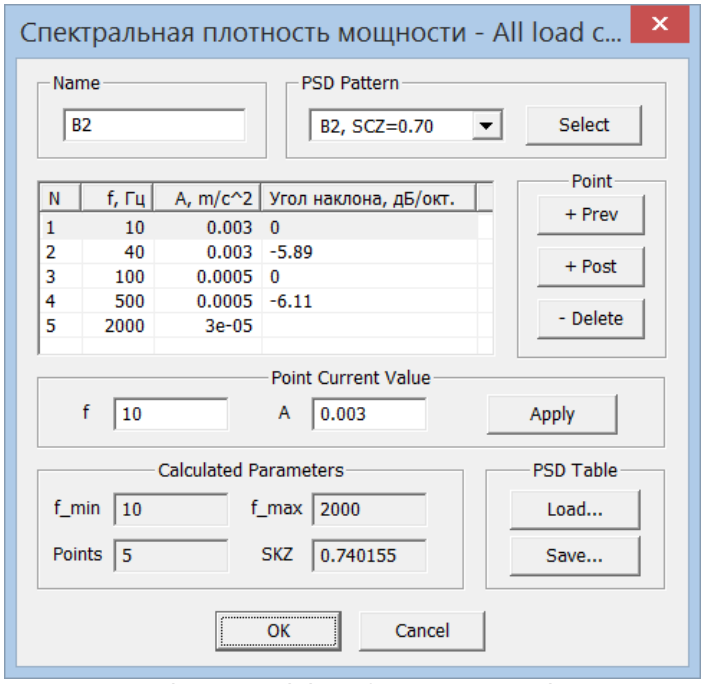


Fig. 4.14 *Dialog of power spectral density setup.*

.The second group of "Frequency Options" dialogue with Fig.4.54 has three options. The "Jk" field affects randomness of a distribution that is synthesized. The range of possible integer values of 0 … 4e+9, in addition, the zero value means each recount will give a new distribution. All the non-zero values allow to reproduce a defined random distribution every time.

The "KOm" field defines a frequency number, participating in the synthesis of a random disturbance. If this value is equal to one, it will eventually be given not random and fixed effect of the sinusoidal root mean square value (rms) equal to the value obtained in the "RMS" dialogue, **Ошибка! Источник ссылки не найден.**

The switch «Log10 Frequency» allows to set a logarithmic distribution involved in the synthesis of a random distribution of frequencies. If the switch is deactivated, there will be a distribution based

on the frequencies from minimum frequency to maximum frequency (see dialog fields «f_min» and «f_max» of the group "Calculated Parameters" dialog, **Ошибка! Источник ссылки не найден.**) will be linear. If the switch is deactivated, the distribution will be logarithmic when in the area of low frequencies in comparison to high frequencies greater unloading occurs between frequencies participating in the synthesis of random effect.

The third «Linear Transformation" group: F(x) = Ax+B» allows to pass shift of a mean value (mathematical expectation) of a distribution and scaling, which leads to change of a final value of RMS. The parameters of this group do not influence a synthesis of a distribution itself, after participation of these parameters the distribution is only converted, therefore there is no need to make recalculation. If the "Apply" switch is turned off, standard quantities of transformation A=1, B=0, regardless of the values which are specified in corresponding fields, will be used.

The fourth group "General Calculation Parameters" for a current stage of random loading contains a field of "T, s", this is time for the synthesis of a stage graph. For this time it is recommended to set to not less than order of a larger reciprocal from a value of minimum frequency «f_min» in RMS from Fig. 4. 54. Then the synthesized graph will satisfy specified spectral power density and the calculated statistical parameters will be consistent. The "Points" field is not intended for editing; it is informative and contains the number of points which fell on the preset time of a graph with consideration for the parameter "General Digitization Frequency for all Stages, Hz" from Fig. 4.51.

The "Replays" field of the group "General Calculation Parameters" allows to use the property of ergodicity of a random distribution stage that is synthesized. If the time schedule is more than an order of magnitude greater than the inverse of the minimum frequency «f_min», then using this setting one can change the ratio between the contributions from each of the stages in a total random load.

The group "Calculated Parameters", as was already mentioned, contains resulting statistical values of a loading stage synthesized random distribution. For the synthesis of a graph and calculation of parameters one must push the button "Compute", **Ошибка! Источник ссылки не найден.**. Unless the calculation is made, but if the dialog is closed by the button "OK", corresponding fields will not be displayed in the roster of stages from Fig. 4.51, but during the calculation of static character or forced fluctuations the synthesis of random loading will be carried forcibly.

The "dD" and "eD" field values determine an absolute and relative error in calculated dispersion of a synthesized stage where the value of variance equal to a square of a computed value of "RMS" of a RMS's setup dialog is taken as base, **Ошибка! Источник ссылки не найден.**. The relative error is rationed to 1. If the relative error is more than 10%, it is recommended to increase the number of frequencies participating in synthesis of random effect stage in a field of "KOm", fig. 4.53.

The "STD" field corresponds to a mean square value determined by synthesized random effect in the following formulae from mathematical expectation sample values (average) and variance:

$$\bar{x} = \frac{1}{N}\sum_{i}^{N} x_i,$$

$$\mathrm{D} = \frac{1}{\mathrm{N\text{-}1}}\sum_{\mathrm{i}}^{\mathrm{N}}\left(\mathrm{x_i\text{-}\bar{x}}\right)^2, \mathrm{D} = \frac{1}{\mathrm{N\text{-}1}}\sum_{\mathrm{i}}^{\mathrm{N}}\left(\mathrm{x_i\text{-}\bar{x}}\right)^2,$$

$$\mathrm{STD} = \sqrt[2]{\mathrm{D}}\mathrm{STD} = \sqrt[2]{\mathrm{D}},$$

Where x_i – values of random load in i – a i-moment of time, N are the general number of (count) points of a distribution.

The "As/As0" and "Ex/Ex0" fields are serving for evaluation of deviation measure of a synthesized loading stage from a normal distribution. On the absolute value they report the smaller values of three ones about the fact that the deviations from a normal synthesized distribution are not significant. If the received values turned higher based on the modulus than three, for reduction of these magnitudes it is recommended to increase time for a graph (the number of points of ***N***). Formulae for a calculation of an asymmetry and excess of a random distribution, as well as quadratic means of an error of dissymmetry coefficients and excess:

$$\mathrm{As} = \frac{1}{N}\cdot\frac{1}{\left(\sqrt{D}\right)^3}\cdot\sum_{i}^{N}(x_i - \bar{x})^3,$$

$$\mathrm{As0} = \sqrt{\frac{6 \cdot (N-1)}{(N+1) \cdot (N+3)}},$$

$$\mathrm{Ex} = -3 + \frac{1}{N} \cdot \frac{1}{D^2} \cdot \sum_{i}^{N} (x_i - \bar{x})^4,$$

$$\mathrm{Ex0} = \sqrt{\frac{24 \cdot N \cdot (N-2) \cdot (N-3)}{(N-1)^2 \cdot (N+3) \cdot (N+5)}}.$$

Field «NR» of reference character, determines a coefficient of an irregularity equal to a ratio of the number of transitions of a distribution barring zeroes to the number of extremes [4]. The closer this value is to unity, the distribution is more like a sine wave and a narrow band. Small values correspond to the composition of the complex frequency random process and meet the broadband process.

The "Cycles" field is being calculated on basis of the « falling rain» method [4]. It is bisecting to a "Cycles" field received as a result of a calculation of a semi-cycle number for display. The resulting calculation of the number of half cycles is halved to display in the "Cycles" field.

The "NCS" field displays the resulting number of cycles with consideration for repetitions from the "Replays" field, see fig. 4.53.

The "NTS" field displays total time of a distribution (but not graphics!) with consideration for repetitions from the "Replays" field.

We will consider the final group "Graphs of the Loading Stage" from Fig. 4.53". The field "Interval Histogram" is only used when displaying graphs of distributions the recommended N_h value is not less than 6. The drop-down list allows to select a type of chart displayed in a separate window, Fig. 4.55.

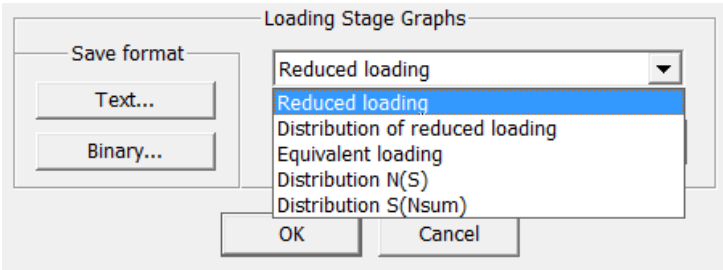


Fig. 4.15 *Drop-down dialog of "Setting of a Loading Stage" list of a group "Graphs of Loading Stage".*

The selection of a chart type as "Given Loading" allows to output a graph that is synthesized in a separate window with a rationing consideration.

The type "Distribution of Given Loading" allows to output the graph of a distribution which must be like the Gaussian bell.

It allows "Equivalent Loading" to display some equivalent load when defined quantities correspond to stresses and every point is calculated as equivalent in damaged ability to symmetric load. There is no great physical meaning in this graph, but distributions of N(S) and S(NSum) are forming on its basis. The distribution of N(S) is the number of cycles depending on load and S(NSum) is a dependence on the number of corresponding load cycles. It is on the basis of the schedule S(NSum) an equivalent to the value of the cycles number, which impairs the damaging ability of equivalent symmetric sinusoidal load with the same half spread value as the maximum in the distribution of S (NSum) exists.

The Group "Format of Conservation" of the same group "Charts of Loading Stage" contains the buttons "Text" and "Binary» that allows to store the synthesized stage in different format. Text format to save can be used to carry out in a third-party application the calculation of the additional desired values, or to correct reporting stage steps. The further load of a corrected text file is possible only in case of creation in the dialog "Fatigue Multistage Random Loading" from Fig. 4.51 of a stage being

loaded from a file. But before display of the dialog a file must be selected from the matching storage folder. File formats are supported in the “Export/Import”:

1) PRN, where the separator is a space and/or a character of a tab;
2) The CSV where the separator is a comma between numbers. Attention! Invalid is the number the comma of which corresponds to a decimal point for this file format!

The appearance of a dialog for a case of loading being loaded from a stage file will be almost the same as in Fig. 4.53 as soon as the group "Parameters of Spectral Power Density" and "Frequency Parameters" is absent. The button "Compute" and the field of "T, s" in the group "Generic Calculation Parameters" will be unavailable; the field "dD" and "eD" will be vacuous in the group "Calculated Parameters"; the group "Save Format" will be called "Load Format" with corresponding functionality.

We will return to the consideration of the dialog "Fatigue Multistage Random Loading" at Fig. 4.51. The group "Resulting Parameters of Multistage Loading" contains three fields which in themselves are accumulating corresponding values for included stages. But the first two are without consideration of the number of repetitions, (the number of points and time) answer for a graph itself. And in the field "Total Time of Set Loading, s" times from each stage sum, taking into consideration the number of "Replays" from Fig. 4.53, in other words, calculated "NTS" parameters sum.

The same group "Graph of Loading" is exactly as in a dialog with Fig. 4.53, but only one type of chart – "Total Loading", on graphs being agreed by the way of stages succession in a list with consideration for smoothing.

The group "Export" contains two buttons: "Into a Text File..." and "Into a Graph for loading". Export of all loading stages will be carried out to the text file in a procedure of their being followed. The push on the button "Into a Graph for Loading" will lead to export of received final multistage load into a graph of load for associated loading. The name of this loading is specified in the name of a dialog, see Fig. 4.51. If export was not made, than for loading the graph of load would remain former or, in general, it will remain not defined.

The calculation methodology

Stresses are calculated over the computed values of stresses amplitudes and their average values (3) were led to symmetric loading by (4), respective R = -1 [1,2,6,7]:

$$\sigma^{R} = K \cdot \sigma_{aм} + \psi_{\sigma} \cdot \sigma_{м} \sigma^{R} = K \cdot \sigma_{aм} + \psi_{\sigma} \cdot \sigma_{м}$$
$$\tau^{R} = K \cdot \tau_{aм} + \psi_{\tau} \cdot \tau_{м} \ \tau^{R} = K \cdot \tau_{aм} + \psi_{\tau} \cdot \tau_{м},$$

Where K is the coefficient of the fatigue limit reduction and$\psi_{\tau}\psi_{\tau}$ ψ_{σ} ψ_{σ}coefficients of influence of an asymmetry of cycle on the amplitude limit.

After calculating the reduced values of stresses in finite elements It calculates stresses equivalent one-axial tension, using the formula:

$$\sigma_{э1} = \sqrt{\frac{\left(\sigma_x^R - \sigma_y^R\right)^2 + \left(\sigma_y^R - \sigma_z^R\right)^2 + \left(\sigma_z^R - \sigma_x^R\right)^2}{2} + \cdots + \left(\frac{\sigma_{-1}}{\tau_{-1}}\right)^2 \cdot \left((\tau_{xy}^R)^2 + (\tau_{yz}^R)^2 + (\tau_{zx}^R)^2\right)} \sigma_{э1} =$$
$$\sqrt{\frac{\left(\sigma_x^R - \sigma_y^R\right)^2 + \left(\sigma_y^R - \sigma_z^R\right)^2 + \left(\sigma_z^R - \sigma_x^R\right)^2}{2} + \cdots + \left(\frac{\sigma_{-1}}{\tau_{-1}}\right)^2 \cdot \left((\tau_{xy}^R)^2 + (\tau_{yz}^R)^2 + (\tau_{zx}^R)^2\right)} \quad (5),$$

Where *x*, *y*, *z* are the indexes of stresses in corresponding directions/planes – the fatigue

σ_{-1} σ_{-1} limit on normal stresses, the fatigue τ_{-1} τ_{-1} limit on tangential stresses.

As a result a distribution of stresses under the $\sigma_{э1}$ $\sigma_{э1}$ number of semi-cycles determined in the schematization stage is got. The example of a distribution of N(S) is shown in a Fig..

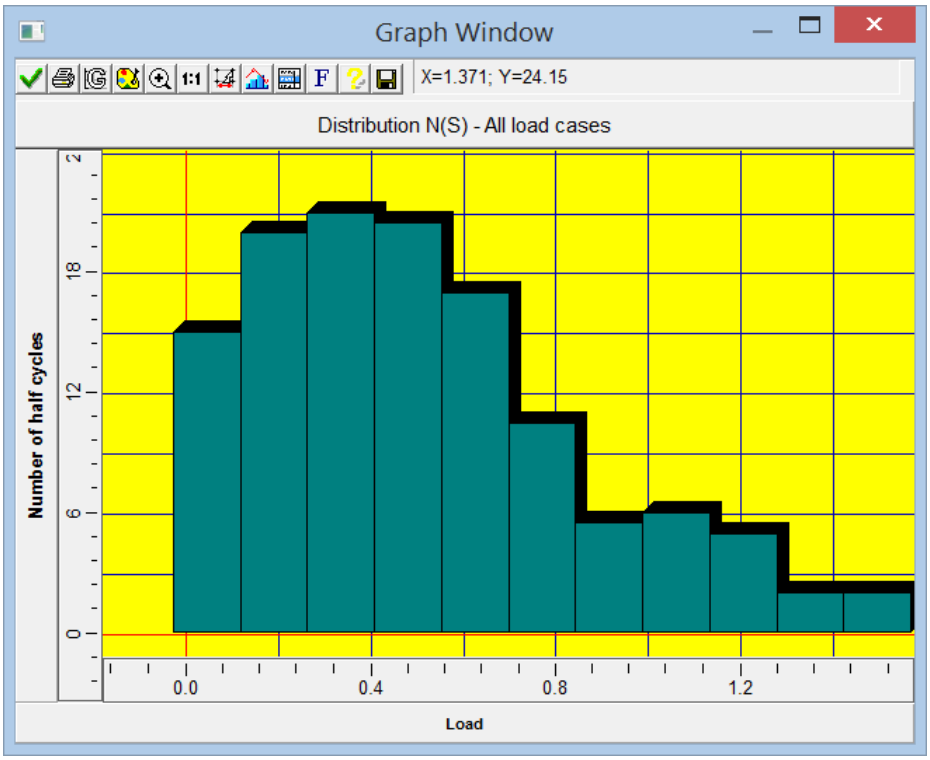


Fig. 4.16 *the Histogram of N(S) of a probability distribution based on the stresses for* $\sigma_{э1}$ $\sigma_{э1}$ *Nh = 24, where a total number of semicycles is equal to 128506.*

From a distribution of the semi-cycles number from stresses (see fig. 4.56) a distribution total in the number of semi-cycles is being constructed, see fig. 4.57:

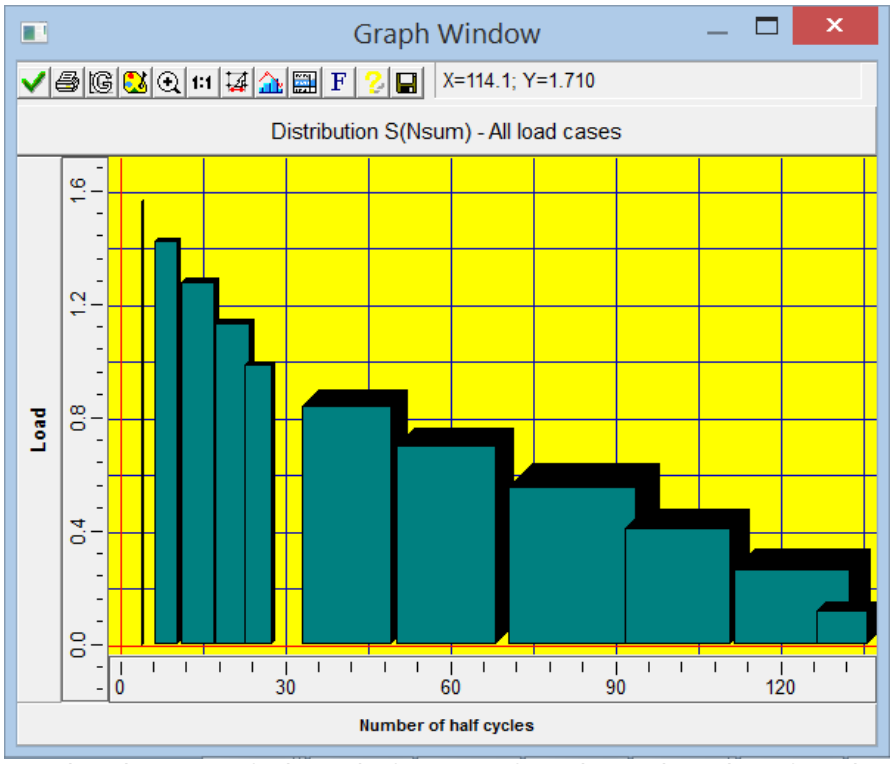


Fig. 4.17 *the Histogram of S(Nsum) of stresses from the total number of semi-cycles.*

For cast of this distribution to equivalent sinusoidal stress it is selected for calculated stress – the$\sigma_{э1}^{max}\sigma_{э1}^{max}$ highest of the achieved. Then the equivalent number of cycles$n_{э\,sin}n_{э\,sin}$ which at calculated stress possesses the same damaged ability as the synthesized stochastic number is being computed. For it one must use a **corrected** linear hypothesis of summation of fatigue damages during irregular loading [1,6,7].

In accordance with this hypothesis fatigue damage being caused by stress of σ_i constitutes a certain equal share of$^{n_i}/_{N_{0i}}$ $^{n_i}/_{N_{0i}}$ complete damage corresponded to an appearance of fatigue cracks and destruction. The value$n_i n_i$ corresponds to the number of cycles designed with this stress level. Value is$N_{0i}N_{0i}$ equal to the number of cycles which corresponds to 50% of probability of destruction of a part with the same σ_i stress level.

Destruction on this theory comes then since the condition was satisfied:

$$\sum_{\substack{i=1 \\ K\cdot\sigma_i \geq \sigma_{-1}}}^{N_h} \frac{n_i}{N_{0i}} = \alpha_э,$$

Where N_h – as a histogram number, – is equal to the number of the stress levels, $_{\alpha e}$ is an experimentally determined correction in a corrected linear hypothesis.

Attention must be turned to the condition under the sum sign: only the semi-cycles for which the condition is satisfied will produce damaging effect:

$$K \cdot \sigma_i \geq \sigma_{-1} K \cdot \sigma_i \geq \sigma_{-1} \qquad (6).$$

From the equation of the curve endurance (1) in the multi-cycle range of ($10^3 < N_i < N_G 10^3 < N_i < N_G$), the value of the cycles number corresponding to 50% of probability of destruction is derived:

$$N_{0i} = \frac{\sigma_{-1}^{m} \cdot N_G}{\sigma_i^{m}}.$$

The received quantity of the cycles number is being substituted to an equation of a fatigue curve:

$$\sum_{\substack{i=1 \\ K\cdot\sigma_i \geq \sigma_{-1}}}^{k} \sigma_i^{\mathrm{m}} \cdot \mathrm{n}_i = \sigma_{-1}^{\mathrm{m}} \cdot N_G \cdot \alpha_э = (\sigma_{э1}^{\max})^{\mathrm{m}} \cdot \mathrm{n}_{э\,\sin} \cdot \alpha_э,$$

from where derivation of the equivalent cycles number is carried out:

$$\mathrm{n}_{э\,\sin} = \frac{1}{\alpha_э} \sum_{\substack{i=1 \\ K\cdot\sigma_i \geq \sigma_{-1}}}^{k} \mathrm{n}_i \cdot \left(\frac{\sigma_i}{\sigma_{э1}^{\max}}\right)^{\mathrm{m}} \quad (7).$$

To get the reserve coefficient for the number of cycles, first the number of cycles corresponding to 50% of probability of destruction for the calculated are computed$\sigma_{э1}^{\max}\sigma_{э1}^{\max}$:

$$N_{0э1\max} = \frac{\sigma_{-1}^{\mathrm{m}} \cdot N_G}{\sigma_{э1}^{\max}}.$$

The value$\mathrm{N}_{0э1\max}\mathrm{N}_{0э1\max}$ is the basic value. The maximum value of this magnitude is limited by the amount of N_G = 2e6. Such values correspond to a case, when,$\sigma_{э1}^{\max} < \sigma_{-1}/\mathrm{K}\,\sigma_{э1}^{\max} < \sigma_{-1}/\mathrm{K}$ what corresponds to a case of unlimited stamina. Under high values the$\sigma_{э1}^{\max}\sigma_{э1}^{\max}$ base number of cycles polynomial, based on an equation of a curve of endurance (1) in multi-cycle ($10^3 < \mathrm{N_i} < \mathrm{N_G}10^3 < \mathrm{N_i} < \mathrm{N_G}$) area, decreases.

Then the coefficient of reserve in the number of cycles is determined by the formula:

$$n_N = \frac{N_{0э1\max}}{\mathrm{n}_{э\,\sin}}.$$

The derived values of the coefficient of reserve in the number of cycles are so limited above by a quantity of N_G = 2e6. Such a value of the coefficient of reserve can occur in two cases:

1) $\mathrm{n}_{э\,\sin} \leq 1\mathrm{n}_{э\,\sin} \leq 1$;
2) $\sigma_{э1}^{\max} \leq \sigma_{-1}/\mathrm{K}\,\sigma_{э1}^{\max} \leq \sigma_{-1}/\mathrm{K}$.

Calculation of random fatigue on statics

Fatigue calculation based on the results of static justified use in the case, If the natural frequencies lie on the frequency axis to the right of the frequency of the driving forces.

For spending of a fatigue calculation the following order of actions must be implemented:

- Create a model of calculation containing elements from the phylum plate and/or 8-nodes and the key three-dimensional finite element;
- Specify random fatigue load for selected loading or combinations of loadings;
- Specify necessary fatigue parameters;
- Implement a static calculation;
- Carry out a start of a fatigue calculation for random load based on the results of statics.

The results of this calculation will be available in the list "Choice of results" of the dialog "Parameters of results output".

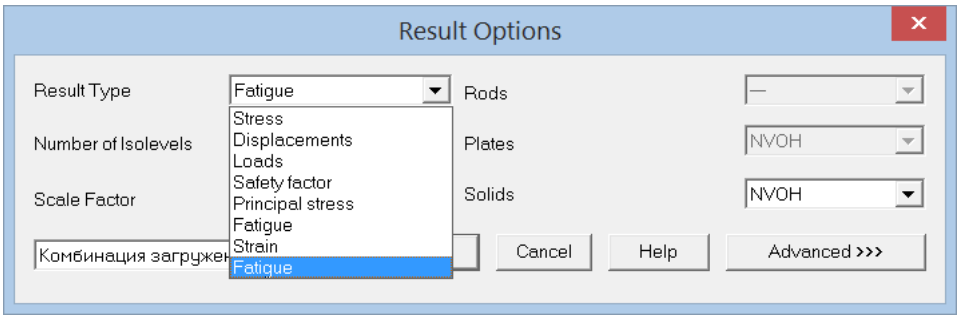
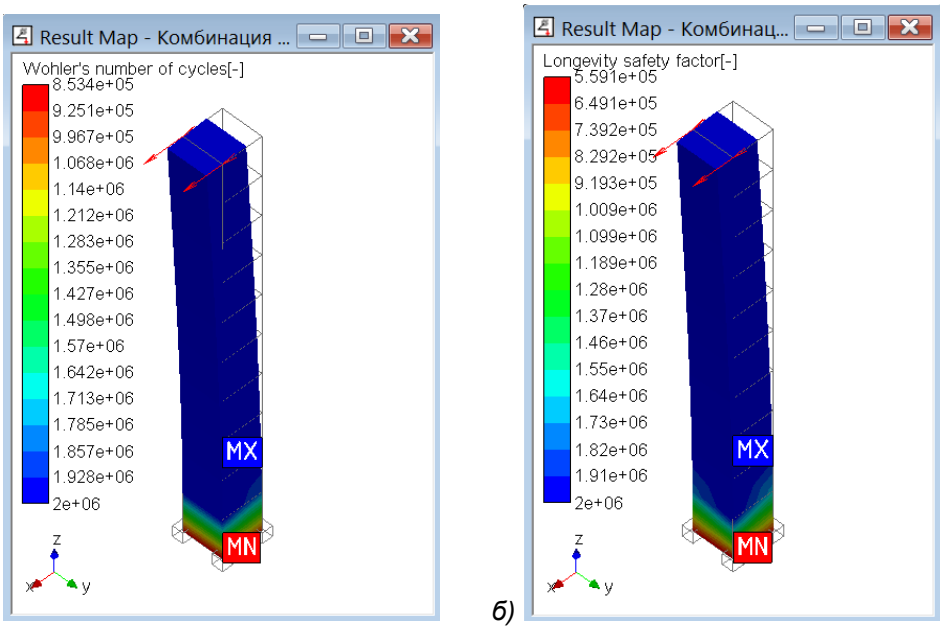


Fig. 4.18 *the Selection of results of the simplest fatigue calculation.*

In the list for plates and three-dimensional elements a list of two rows will become available: "NVOH" and "SAFN". Selecting the line «NAOH» from the list will lead to drawing the calculated values of the number of Wohler cycles, equivalent to the corresponding maximum load reached in the application of random load. This number of cycles corresponds to 50% of destruction probability. This value of cycles is a basis for calculation of the reserve coefficient across durability (the number of cycles). For display of estimated reserve for the number of cycles of a model a "SAFN" line must be selected in the list for plates and three-dimensional elements.

If after a static calculation passing the pattern was edited, before a fatigue calculation passing on statics a warning will be said. If after the static calculation model is edited, that prior to the calculation of static fatigue a warning will be issued.

On Fig.4.59 the results of the calculation of the safety factor for fatigue strength are presented a) and the safety factor in durability (b) for the bulk of finite elements.



а) *б)*

Fig. 4.19 *Results map for the number of cycles on Wohler (a) and the safety factor for fatigue durability (б).*

Because the resulting quantities of a fatigue calculation are being calculated based on the results of statics, the calculation is being done consistently for all loadings for which the fatigue load is specified. Therefore, if during the calculation of stress in the node of one of the final elements the value of the limit of variability is exceeded, the warning of Fig. which corresponds to the current loading that is calculated will be given.

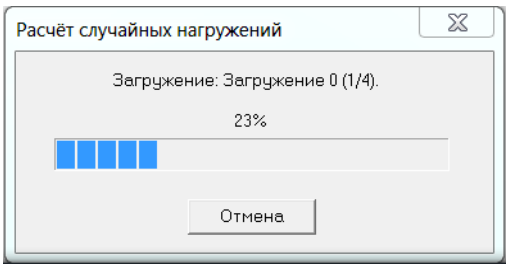


Fig. 4.20 *Dialog with the progress indicator for calculating the parameters of fatigue as a result of static*

Random fatigue calculation through forced fluctuations

The following order of actions must be implemented for a calculation process:

1. Create a calculated model containing elements from the phylum plate and/or 8-node and a three-dimensional finite element;
2. Create a super-element of selected elements from the phylum plate and/or 8-node and a three-dimensional finite element;
3. Specify random fatigue load and export it into a graph of loading for corresponding loading or a combination of loadings;
4. Set necessary fatigue parameters;
5. Carry out a start of a fatigue calculation for random load through a calculation of forced fluctuations for corresponding loading or a combination of loadings.

The fatigue calculation is used through calculation of forced fluctuations in case of an intersection in the frequency axis of the spectrum of perturbing effect and the natural frequencies of the model.

Naturally, than more terminal cells in a calculated model, that, necessary time is of calculation of each time count.

Call the dialog "Calculation" from the main menu "Calculations -> Calculation...» A part of a dialog is shown in Fig. 4.61. Then turn on a switch in the item "Calculation for Super Elements Only". The switch will be unavailable until at least one FE is created. Fill out the fields "Logarithmic Decrement of Fluctuations" and "Number of the Considered own forms". The field «Interval: 0-, is based on a specified graph of load, is filled out automatically in case of a calculation for FE. The field "Moments of Time" contains the number of points of a specified load graph. It is not recommended to specify a more multiple automatically specified number for a curve created by connection of points by lines.

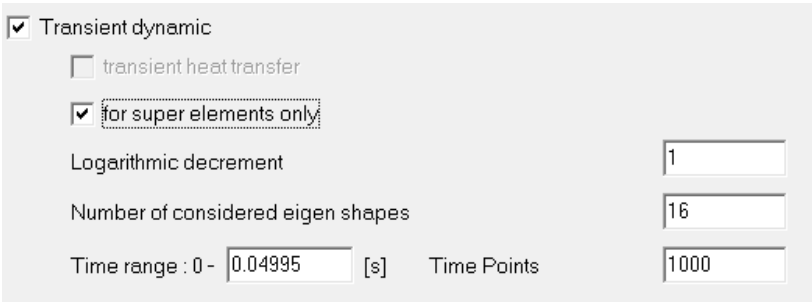
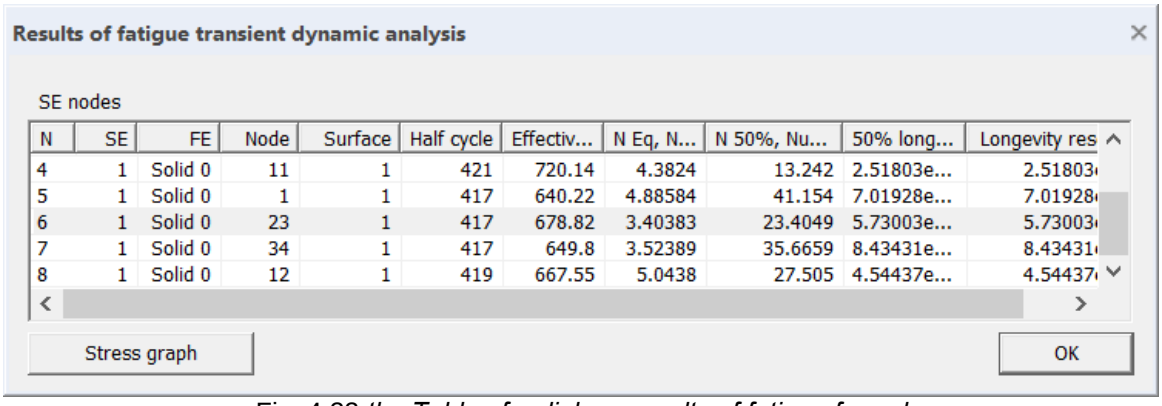


Fig. 4.21 *Part of the dialog "Calculation"*

After the calculation of forced fluctuations was made, call the dialog "Results of Forced Fatigue Fluctuations", having selected an item of the main menu "Results -> Durability During Random Fatigue Loading...". The dialog contains the list of all nodes of previously set FE.



N	SE	FE	Node	Surface	Half cycle	Effectiv...	N Eq, N...	N 50%, Nu...	50% long...	Longevity res
4	1	Solid 0	11	1	421	720.14	4.3824	13.242	2.51803e...	2.51803
5	1	Solid 0	1	1	417	640.22	4.88584	41.154	7.01928e...	7.01928
6	1	Solid 0	23	1	417	678.82	3.40383	23.4049	5.73003e...	5.73003
7	1	Solid 0	34	1	417	649.8	3.52389	35.6659	8.43431e...	8.43431
8	1	Solid 0	12	1	419	667.55	5.0438	27.505	4.54437e...	4.54437

Fig. 4.22 *the Table of a dialog «results of fatigue forced fluctuations» with final calculation values.*

The columns reflect information based on the SE, KE, nodes, and surface (only for plate KE) numbers. The columns reflect the information on the numbers of SE, CE, components and surfaces (for plate-TBE). The next columns of each node of FE from SE correspond to the found number of the semi-cycles by the "method of rain", equivalent stress$\sigma_{э1}\sigma_{э1}$ (5), the equivalent number of cycles$n_{э\,sin}n_{э\,sin}$ (7) in a column with a name of "N Eq".

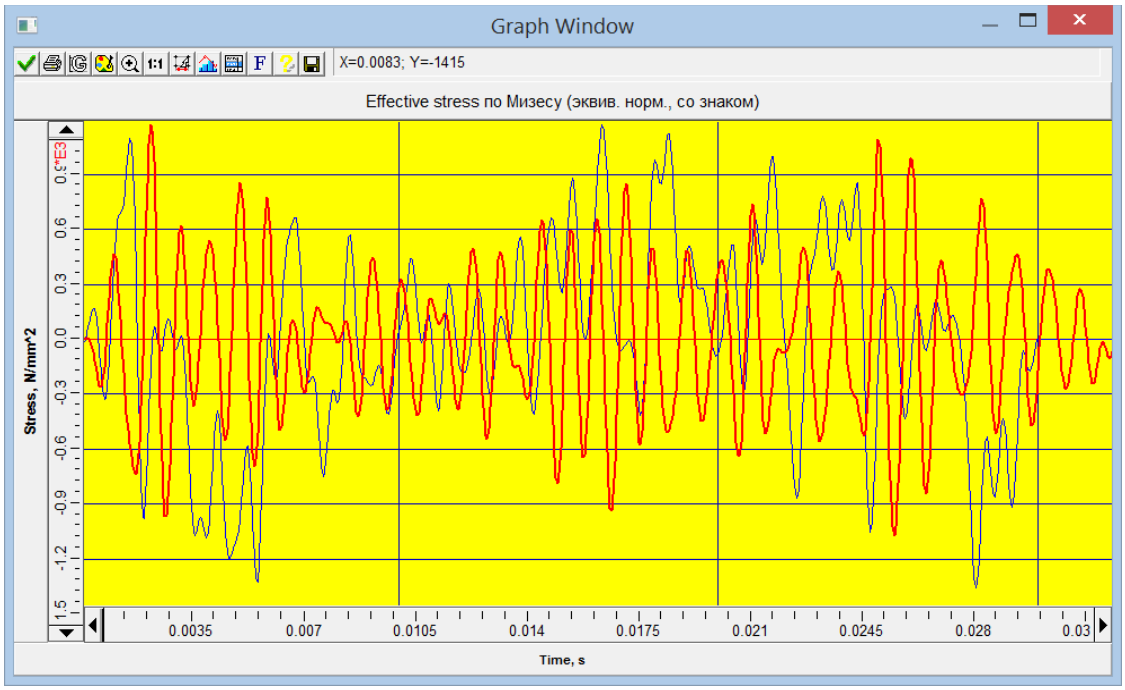
The column of "N of 50%" corresponds to the number of cycles for a computed value of equivalent stress based on Wohler's$\sigma_{э1}\sigma_{э1}$ curve. The values in a column of "50% durabilities", based on the equivalent number of cycles per a unit of time, reflect a value of a column of "N of 50%" converted by one ***hour***. This value is being calculated as a ratio of the equivalent number of cycles to total time of loading, see fig. 4.51, the considering number of "Retries" in each stage, fig. 4.53.

The column "Reserve in Durability" is resulting reserve across durability, is calculated as a ratio of values in the column "Durability of 50%" to the value "Estimated Time of Loading Operation", see fig. 4.51, expressed in hours.

As can be seen from a final result table, Fig. 4.52, values in a "N Eq" column can be at all absent and the values in a column of "N 50%" correspond to a $\boldsymbol{N_G}$ value specified in a dialog from Fig. 4.49. In these cases the equivalent reduced stresses do not obey an equation (6), which means satisfaction of the condition of unlimited endurance under which computation of reserve loses the sense of it.

One more of the indicators of achieved equivalent stress level is the sign of minus in values from the column "Equivalent Stress". It was done to denote those values of stresses equivalent in Mieses in a node of the finite elements which are larger than the limit of tensile strength. As has already been mentioned above by the text, in such a case a dialog with Fig. 4.42 is written after a calculation. If there was excess of the limit of yield strength, the general assumption of tensely deformed state can't be considered correct because the deformations ceased to be resilient and the deformations have already become plastic.

For display of a story of loading and equivalent values of stresses from time of a selected node press the "Graph of Stresses" button. The example of an effective stresses graph is given in fig. 4.63.



With red color – a rationed value of load.
With blue color – equivalent stresses in the node.
Fig. 4.23 the Graph of stresses of a selected node from time.

The name of a graph contains indication that effective stress is displayed with a sign. The sign of effective stress corresponds to the sign of the greatest principal stress module the minus sign corresponds to a predomination of compressive stresses, plus – of the tensile ones.

During modeling of fatigue behavior of plate final elements value «1» is specified in the "N of the Surface" column for the upper surface of plates and in "2" – for the bottom one. The position of the

plate top is determined by the position of a local coordinate system perpendicular. If a graph of stresses is compared for nodes distinguished only by the position of the surface, the rationed values of load may differ from each other only by a sign.

Thermal analysis

We consider steady-state thermal analysis. Boundary/initial conditions for this calculation are temperature values in nodes. Temperature distribution, obtained in this calculation can be used in static analysis to include thermoelasticity effects.

✍ *Note: thermal analysis must be done before or simultaneously with static one.*

Heat transfer transient analysis

The general principles of modeling and performing heat transfer transient analysis are similar to principles and stages for other calculations.

General stages of heat transfer transient analysis

1) Creation of finite-element model, definition of materials and application of thermal loads on structure;
2) Definition of calculation parameters and performance of analysis;
3) Viewing and analysis of results.

***Thermal Loads* toolbar**

The set of thermal loads is carried out by means of *Thermal Loads* toolbar.

Fig. 4.24 Thermal Loads toolbar

Thermal BC and IC on Nodes command (hereinafter BC – boundary conditions; IC – initial conditions) allows to set BC and IC on nodes. This button becomes active if at least one node of model is created.

Shortcut:

Thermal BC and IC on Rods command allows to set BC and IC on rod elements. This button becomes active if at least one rod of model is created.

Shortcut: 

Thermal BC and IC on Shells command allows to set BC and IC on shell elements. This button becomes active if at least one shell of model is created.

Shortcut: 

Thermal BC and IC on Solids command allows to set BC and IC on solid elements. This button becomes active if at least one solid of model is created.

Shortcut: 

Thermal BC and IC on nodes, Thermal BC and IC on rods, Thermal BC and IC on shells, Thermal BC and IC on solids

Groups of active buttons on the right of *Thermal Loads* toolbar are various, it depends on which button (**Thermal BC and IC on Nodes**, **Thermal BC and IC on Rods**, **Thermal BC and IC on Shells** and **Thermal BC and IC on Solids**) is pressed on the left and corresponds to BC and IC for various types of finite elements (node, rod, shell or solid). Let's consider various types of BC and IC which can be set.

Initial Temperature button activates IC mode and allows to set initial temperature on finite element. This IC can be set for all types of finite elements.

Shortcut: 

Temperature button activates BC mode and allows to set temperature on finite element. This BC can be set for all types of finite elements.

Shortcut: 

Heat Flow button activates BC mode and allows to set heat flow on finite element. This BC can be set for all types of finite elements.

Shortcut: 

Volume Heat Source button activates BC mode and allows to set heat source (power) on volume of finite element. This BC can be set for rod, shell and solid finite elements.

Shortcut: 

Heat Point Mass button activates BC mode and allows to set additional heat capacity in node. This BC can be set for nodes only.

Shortcut: 

Initial Temperature on Surface button activates IC mode and allows to set initial temperature on surface of finite element. This IC can be set for rod, shell and solid finite elements.

Shortcut: 

Temperature on Surface button activates BC mode and allows to set temperature on surface of finite element. This BC can be set for rod, shell and solid finite elements.

Shortcut: 

Heat Flux on Surface button activates BC mode and allows to set heat flux on surface of finite element. This BC can be set for rod, shell and solid finite elements.

Shortcut: 

Convection button activates BC mode and allows to set convective heat transfer on surface of finite element. This BC can be set for rod, shell and solid finite elements.

Shortcut: 

Convective heat transfer on surface (in each point of surface) of finite element is described by equation:

$$q_{conv} = \alpha(T - T_0),$$

where α – heat transfer factor in point on surface of FE; T – temperature in point on surface of FE; T_0 – ambient temperature, q_{conv} – heat flow in point on surface of FE.

Radiation button activates BC mode and allows to set radiation heat transfer on surface of finite element. This BC can be set for rod, shell and solid finite elements.

Shortcut: 

Radiation heat transfer on surface (in each point of surface) of finite element is described by equation:

$$q_{rad} = \sigma\varepsilon\left(T^4 - T_0^{\,4}\right),$$

where σ – Stefan-Boltzmann constant; ε – thermal emissivity factor in point on surface; T – temperature in point on surface; T_0 – ambient temperature, q_{rad} – heat flow in point on surface of FE.

Modes of thermal loads application

There are some modes to apply thermal loads.

1) One or several finite elements were selected and load set mode for any FE type was activated, thus the following options are possible:

a) Among the selected elements there are no elements of that type for which loads is set. To set load FE of corresponding type should be selected.

b) Among the selected elements there is at least one element of that type for which load is set.

2) Load set mode for any type of FE is activated, thus one or several elements of this type should be selected.

Let's consider variants which are possible in load set mode. User can make selection as in a single selection mode (it is necessary to hold Shift key for selection of several elements), so in a select mode by frame. There are two ways to finish selection:

a) «Space» or «Enter» key;

b) left button click.

Processing of the chosen set of elements is made as follows: the first created element of the corresponding type is looked for and checked.

If loads of corresponding type is set on element, there is a dialog box (work with this dialog is described below) with the BC or IC list of corresponding type which were set earlier on an element; an opposite case load set dialog appears on the screen (work with this dialog is described below).

List of BC or IC dialog

The name of this dialog changes depending on thermal load set mode.

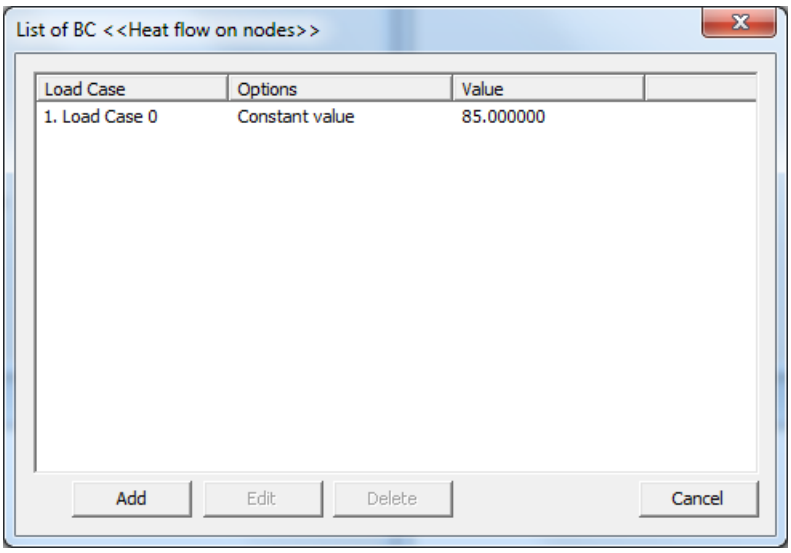


Fig. 4.25 List of BC or IC dialog

The list consists of three columns: *Load Case*, *Options* and *Value*. *Load Case* column shows load case name in which load is placed. In *Options* column can be two options: *Constant value* and *Graph* which displays how thermal load was set. *Value* column displays user value of load if load was set as constant value or word which points to independent variable if load was set as graph.

Add button invokes load set dialog in which load can be set for selected elements.

Edit button invokes load set dialog in which selected load can be changed.

Delete button allows to delete selected load.

Load set dialog

The name of this dialog changes depending on thermal load set mode.

Dialog contains load type and type of FE to which load is applied.

Loads of all types except "Convection" and "Radiation" are defined by only one parameter, and loads of two last types are defined by two parameters: heat transfer factor for convection or thermal emissivity factor for radiation and ambient temperature for both types. Besides loads can be set on all FE and on FE surface (it is impossible to set these loads on nodes).

Let's consider type of dialog for load with one parameter. In the top part of dialog there are *Load Case* drop-down list for saving load. With the help of radio buttons *Constant Value* and *Graph* load

can be set as constant or graph. It will be necessary to choose type of independent variable from the drop-down list to set load as graph. When independent variable "X coordinate", "Y coordinate" or "Z coordinate" is selected additional drop-down list appears where type of coordinate system must be defined. After that **Edit** button will be active using which graph of load can be edited.

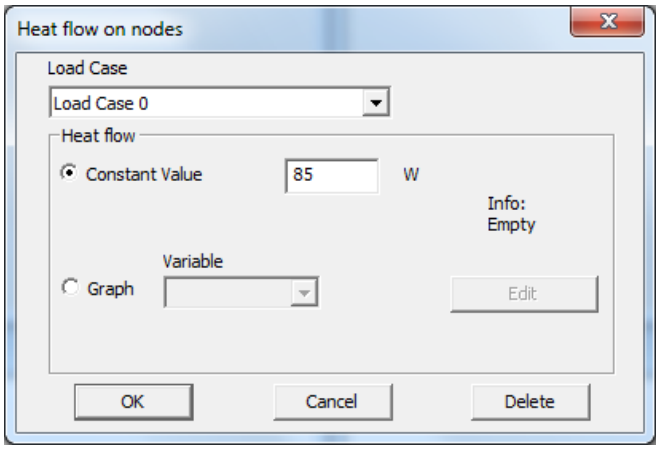


Fig. 4.26 Dialog for setting thermal load as constant

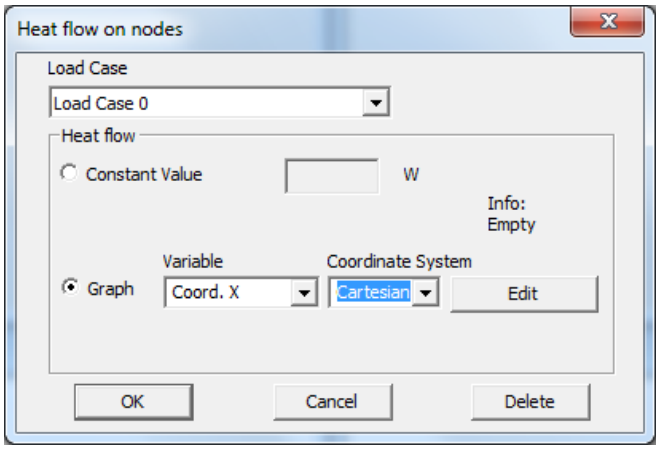


Fig. 4.27 Dialog for setting thermal load as graph

How load is set displays in *Info* text field.
For loads which set on FE surface three types of dialog boxes are possible.

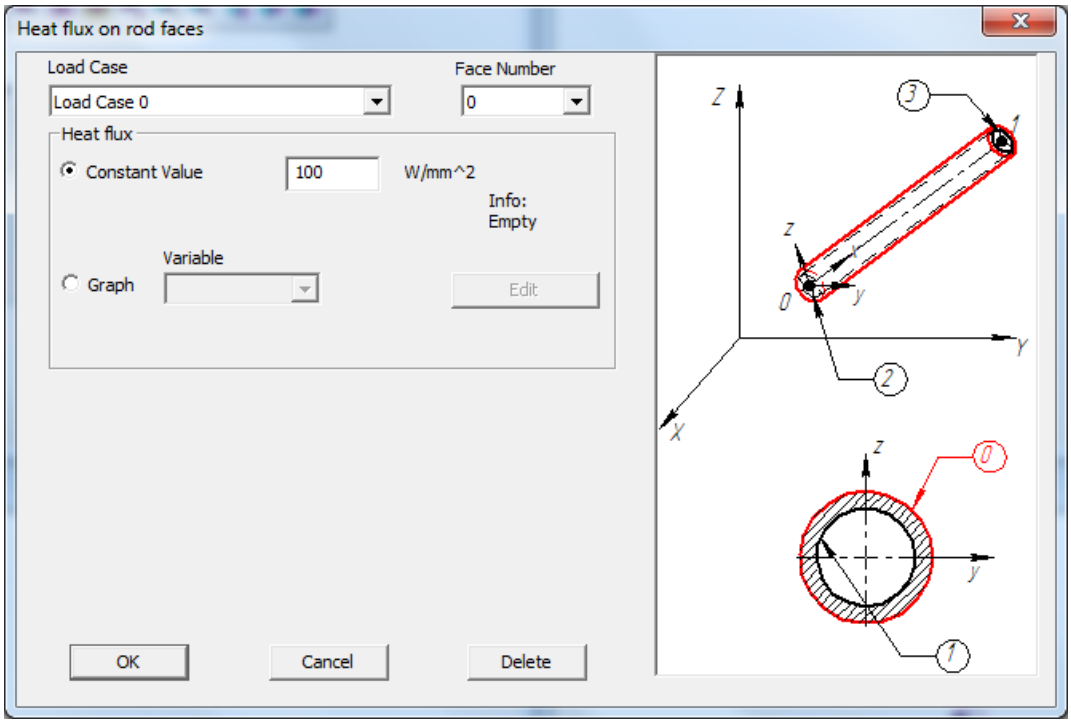


Fig. 4.28 Dialog for setting thermal load on rod element surface with one parameter

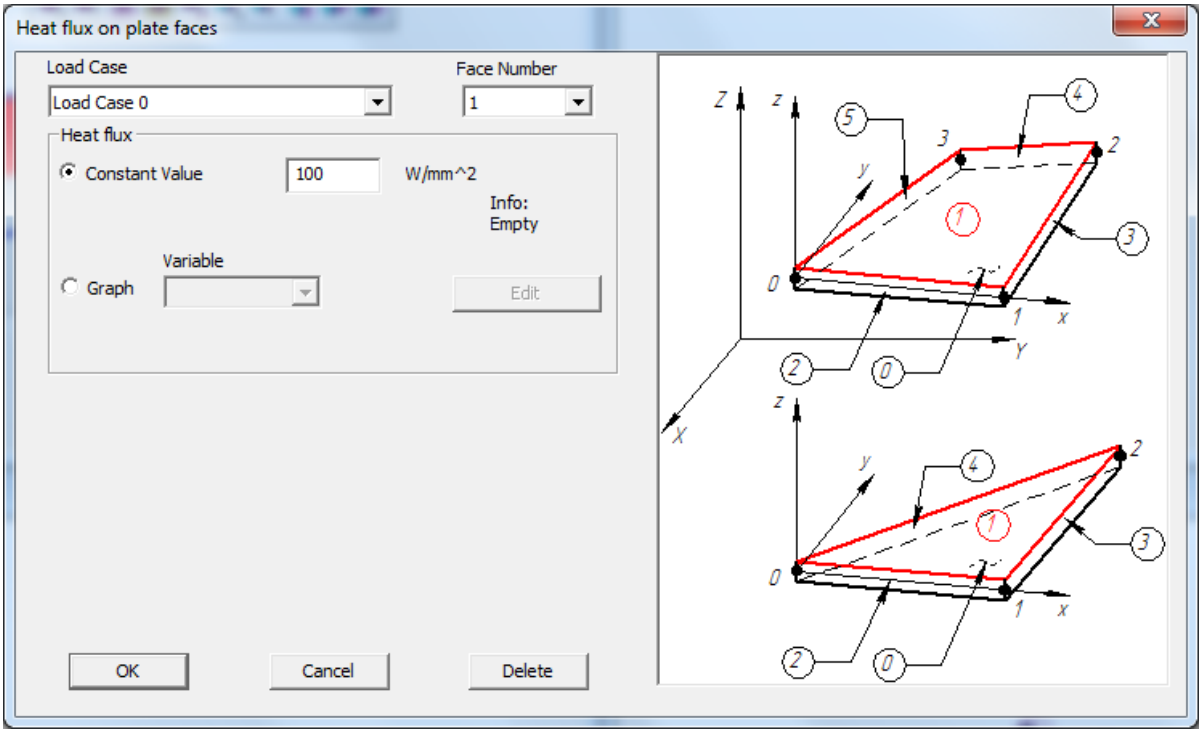


Fig. 4.29 Dialog for setting thermal load on shell element surface with one parameter

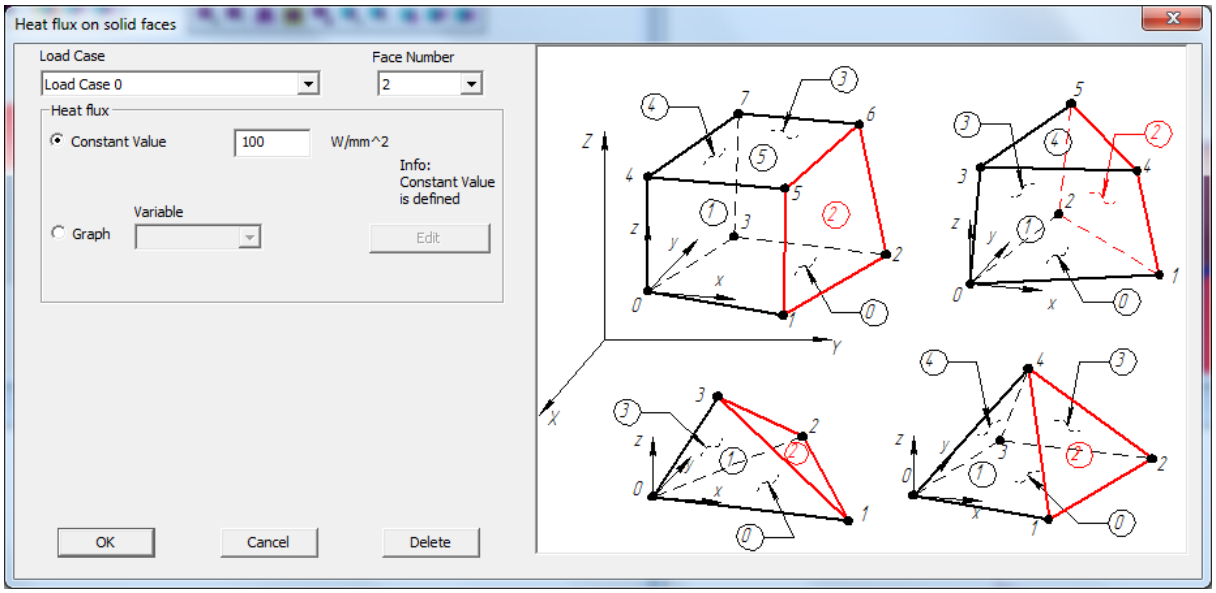


Fig. 4.30 Dialog for setting thermal load on solid element surface with one parameter

The *Face Number* drop-down list is added in this dialog for selection of face number on which load is set. It depends on corresponding type of FE. The selected face is highlighted by red color in drawing displayed in the right part of dialog.

For Convection and Radiation load types this dialog is presented below.

The second parameter of load *Ambient Temperature* is added in this dialog.

OK button creates or changes load parameters.

Cancel button allows to exit dialog without changes.

Delete button allows to delete load.

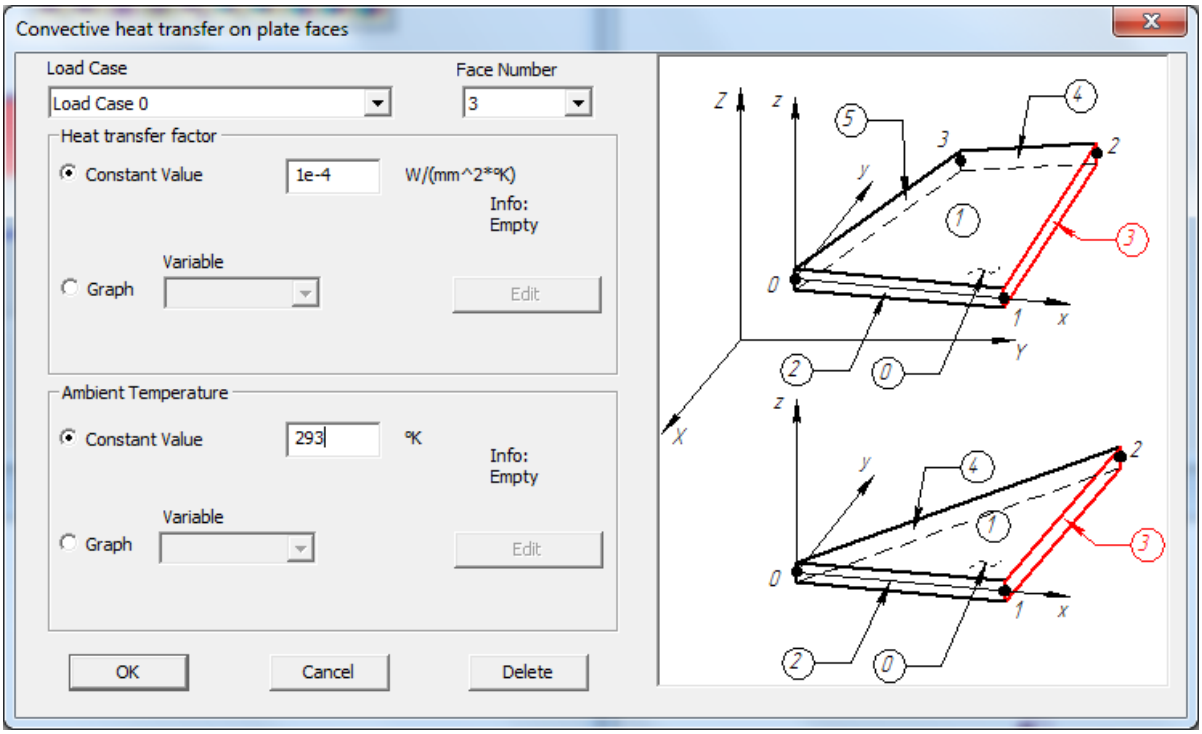


Fig. 4.31 Dialog for setting Convection and Radiation load

Filters of thermal loads toolbar

Buttons of this toolbar are used to show or hide corresponding thermal loads which are already defined.

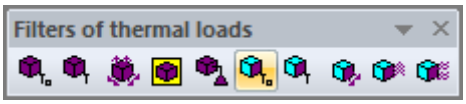


Fig. 4.32 Filters of thermal loads toolbar

Thermal material creation

To perform heat transfer transient analysis it is necessary to set *Thermal material* for all elements.

After pressing **OK** button the dialog box appears on the screen where material parameters should be set as constants or as graphs.

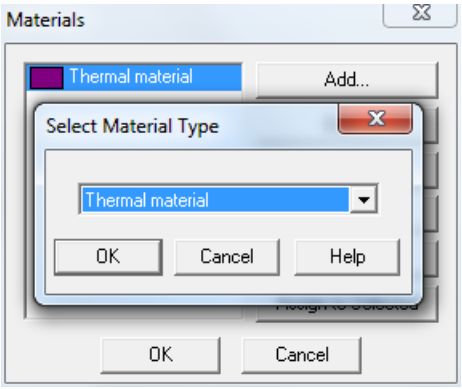


Fig. 4.33 Dialog for material type selection

In this dialog there are *Anisotropic Material* checkbox that allows to set additional *Heat Capacity* along *Y* and *Z directions*.

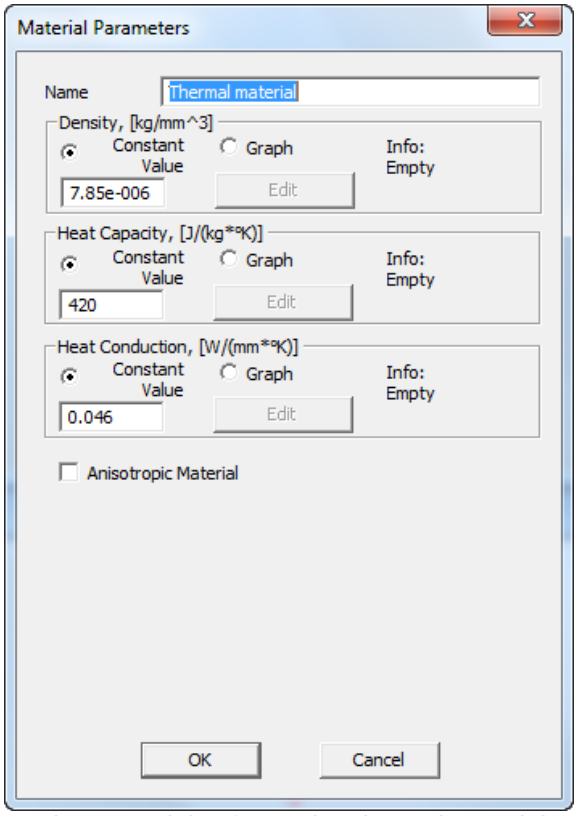


Fig. 4.34 Dialog for setting thermal material properties

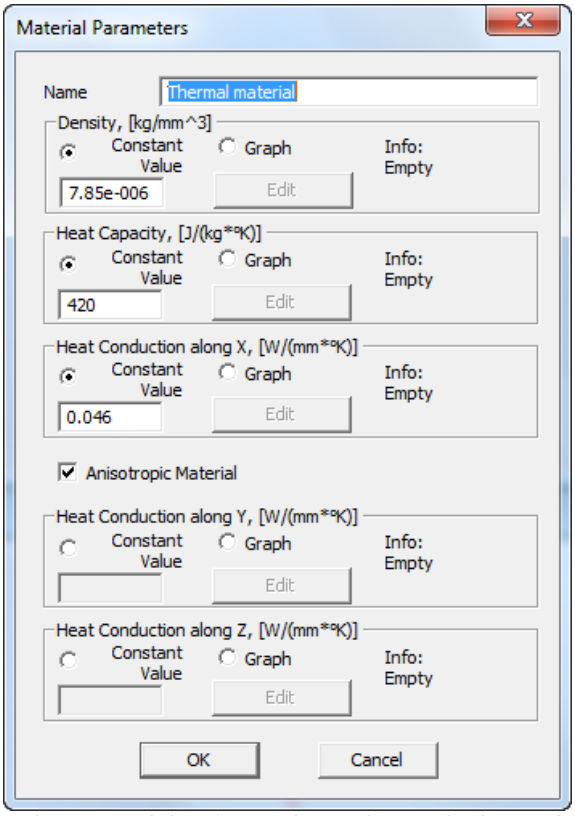


Fig. 4.35 Dialog for setting anisotropic thermal material properties

Calculation parameters

To perform heat transfer transient analysis select corresponding checkbox and required *Load Case* in *Calculation* dialog invoked by **Calculation/Calculation** main menu command.

Then there will be *Transient Heat Transfer* dialog where you can set calculation parameters.

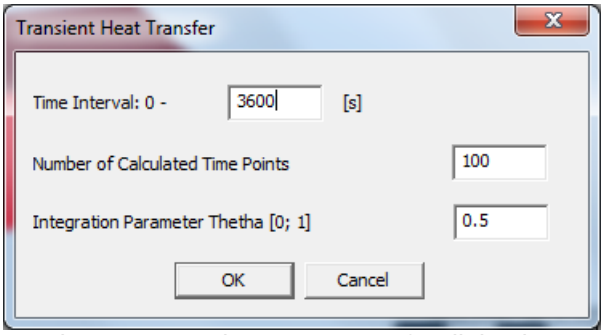


Fig. 4.36 Transient Heat Transfer dialog box

Viewing calculation results

Select **Results/Result Map of Heat Transfer Analysis…** main menu command to view calculation results and then there will be *Results of Transient Heat Transfer Analysis* dialog box.

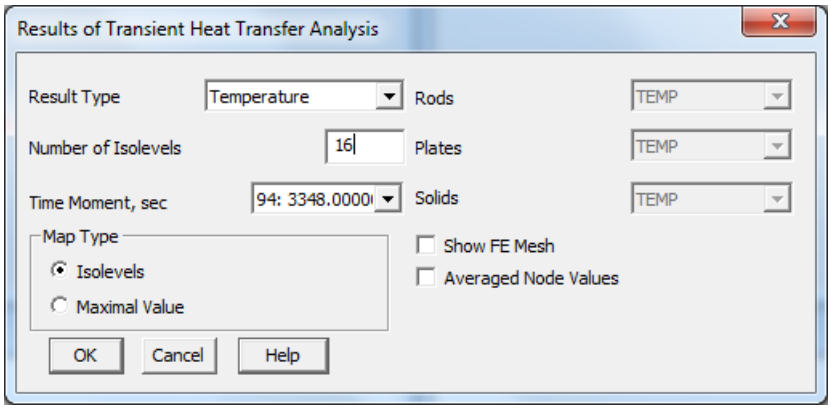


Fig. 4.37 Results of Transient Heat Transfer Analysis dialog box

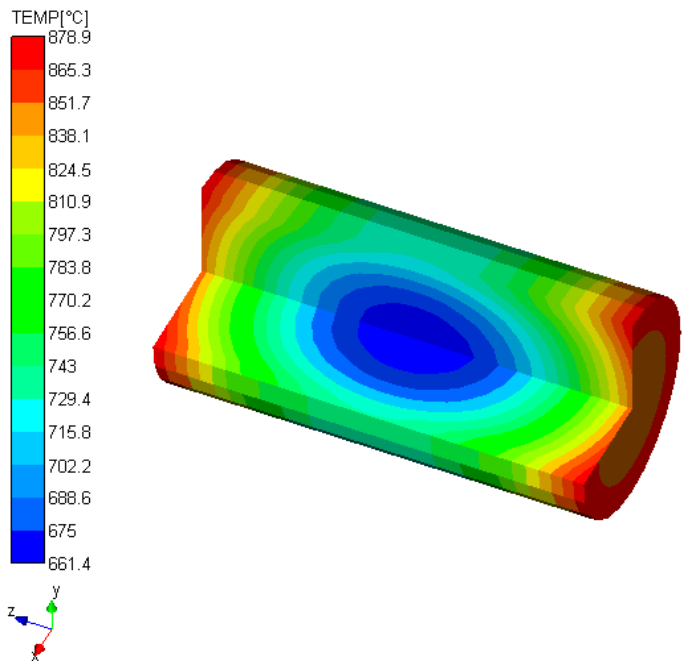


Fig. 4.38 Example of temperature map

In whole work with this dialog box is similar to work with the same dialog available after static calculation.

However when *Vector Heat Flow* option (TFLUX_X_LOC – heat flux along X axis of LCS) or *Vector Temperature Gradient* option (TGRAD_X_LOC – temperature gradient along X axis of LCS) is selected in *Result Type* drop-down list the *Vector Scaling* feature displays in dialog that allows to set scale for vectors in relative units [1; 100].

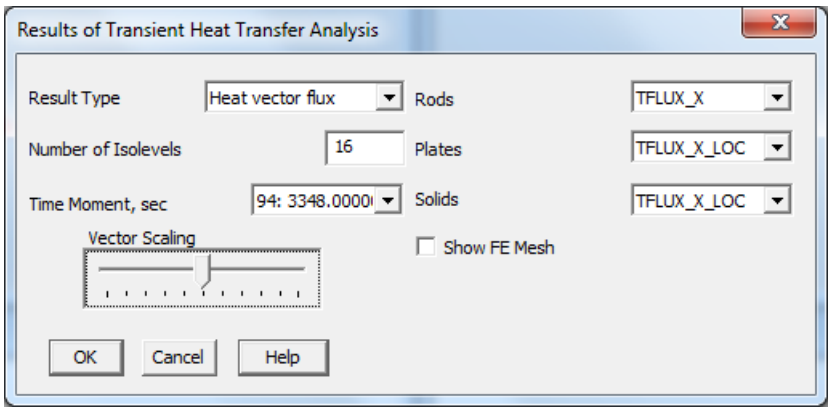


Fig. 4.39 Vector Results of Transient Heat Transfer Analysis dialog box

To change parameters of result map select **Results/Result Options of Heat Transfer Analysis…** main menu command after which activation you can change parameters of result map in the appeared dialog.

When viewing result map **Results/Result Animation of Heat Transfer Analysis…** main menu command is available which allows to animate current result map. This command is available when result map is closed as well. Thus animation window will display results of the previously opened map, if the result map wasn't invoked before temperature results will be animated.

Seismic calculation

This type of calculation is performed using eigen mode shapes decomposition in Duhamel integral form according to building regulations.

Initial equation of system motion for seismic calculation:

$$M \cdot \ddot{\overline{\Delta}}^t + C \cdot \dot{\overline{\Delta}} + K \cdot \overline{\Delta} = 0,$$

where: **t** index means the total displacement of the system.

$\overline{\Delta}^t = \overline{\Delta} + \overline{\Delta}_g$ - total displacement, where $\overline{\Delta}_g$ - base displacement

Substituting expression for total displacement to the motion equation we will receive effective seismic load

$$\overline{P}_{eff}(t) = -M\ddot{\overline{\Delta}}_g(t) \quad \text{or} \quad \overline{P}_{eff}(t) = -M\{\cos\}\ddot{\Delta}_g(t),$$

where $\{\cos\}$ - direction cosines vector of the corresponding freedom degrees with the direction of seismic load, $\ddot{\Delta}_g(t)$ - ground acceleration.

The solution is performed by eigen shape decomposition method using Duhamel's integral.

Seismic load calculation is performed according to response spectrum dependence as well as Russian Seismic Code - SNiP II-7-81* 2000 y. and SP 14.13330.2014.

By Russian seismic code the dynamic seismic load is replaced by effective static inertia forces S_{ik}, acting in direction of k-th DOF(mass) for i-oscillation shape.

$$S_{ik} = K_1 K_\psi Q_k A \beta(T_i) \eta_{ik} \cos\varphi_{ok},$$

where K_1 – damage factor by table 3, K_ψ — *factor* by table 6 and according to chapter 5.

Q_k –k-th mass weight;

A – seismicity factor (relative maximum accelerations) depending on earthquake intensity. The factor value should be taken 0.1, 0.2, 0.4, respectively, for the seismicity 7, 8, 9.

β – dynamic factor corresponding to the i-th tone of the natural vibrations (depending on the oscillation period of the i-th shape and soil category).

η_{ik} – factor depending on the structure deformation shape in its natural oscillations of the i-th tone (reduced acceleration).

φ_{ok} – angle between the direction of the seismic action and the displacement of the k-th degree of freedom.

If the seismicity of 8 points or more, soil category III factor 0.7 is added to the S_{ik} value according to the requirements of SNIP II-7-81 * (SP 14.13330.2014).

The responses X_i for each oscillation shape are determined from inertia loads, then the maximum one $X_\alpha = \max_i |X_i|$ is determined and the design value is calculated:

$$X = [X^2_\alpha \pm \sum_{i \neq \alpha} (X_i)^2]^{1/2}.$$

Design calculation

After static calculation is performed, it is possible to check design elements strength and buckling according to building regulations depending on the type of cross section used and loads applied.

Number of checks by building regulations is defined by the type of section of an element and the complete set of loads acting on it.

Rods are checked on:

- strength at action of longitudinal force N;
- buckling at compression in planes XOZ and XOY;
- strength at action of bending moment M_y or M_z;
- strength at action of lateral force V_z or V_y;
- strength at joint action N, M_y and M_z;
- buckling of the flat form of a bend at action of moment M_y;
- maximum flexibility.

At check by classical methods:

1) Strength by equivalent stresses (Mises):

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

use factor $k = \dfrac{\sigma_{экв}}{\sigma_T}$

$\sigma_x, \sigma_y, \sigma_z, \tau_x, \tau_y, \tau_z$ - normal and shear stresses

σ_T - yield stress of rod material

2) Buckling in reciprocally perpendicular planes

$$i_{1,2} = \sqrt{\frac{I_{x,y}}{A}} \qquad \overline{\lambda_{1,2}} = \frac{l_{efx,y}}{i_{1,2}} \qquad \lambda_{1,2} = \overline{\lambda_{1,2}} \cdot \sqrt{\frac{\sigma_T}{E}}$$

At $\lambda_{1,2} \leq 2.5 \quad \varphi_{1,2} = 1 - (0.073 - 5.53 \cdot \dfrac{\sigma_T}{E}) \cdot \lambda_{1,2} \cdot \sqrt{\lambda_{1,2}}$

Otherwise at $\lambda_{1,2} \leq 4.5$

$$\varphi_{1,2} = 1.47 - 13.0 \cdot \frac{\sigma_T}{E} - (0.371 - 27.3 \cdot \frac{\sigma_T}{E}) \cdot \lambda_{1,2} + (0.0275 - 5.53 \cdot \frac{\sigma_T}{E}) \cdot \lambda_{1,2}^2$$

At $\varphi_{1,2} = \dfrac{332}{\lambda_{1,2}^2 \cdot (51 - \lambda_{1,2})}$

use factor $k_{1,2} = \dfrac{N}{\varphi_{1,2} \cdot A \cdot \sigma_T}$

σ_T - yield stress

$i_{1,2}$ - radius of inertia

$\lambda_{1,2}$ - flexibility

N - normal force
A - cross-section area

$\varphi_{1,2}$ - downturn factor of stress

3) Maximal flexibility
See also Chapter 6, section Design elements results.

Calculation of pipeline segments

Straight elastic tube section

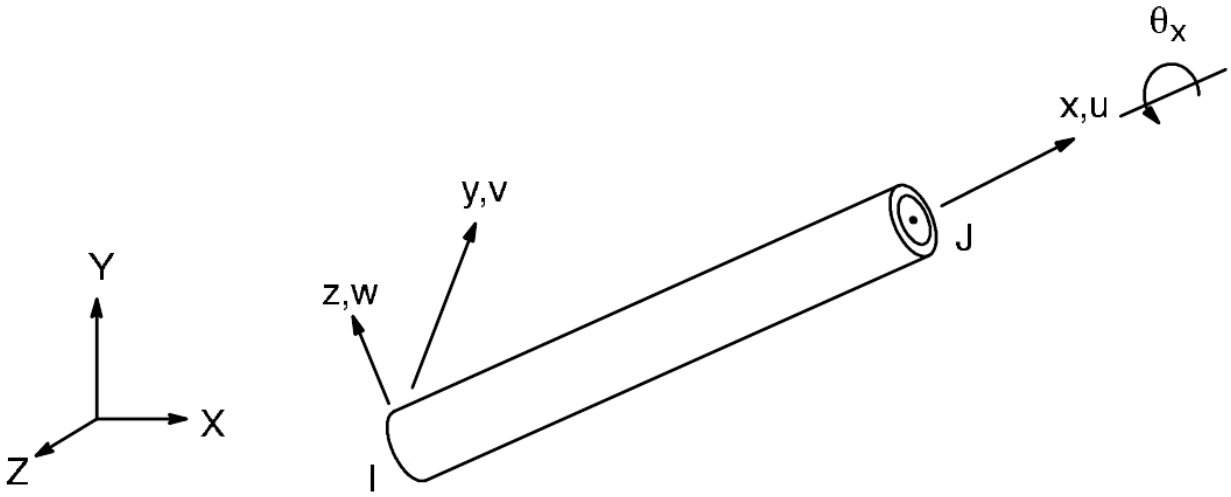


Fig. 4.40 Scheme of the straight segment of a pipeline

This element works in tension, compression, bending and torsion. It has 6 degrees of freedom in every of its two nods: Relocations to the direction of X, Y, and Z axes as well as turns around of X, Y, and Z axes. This element is assumed with the thin wall, it bears a ratio of $D_0/t \geq 20$.

. To a straight segment of a pipeline as by a rod finite element the following types of loads can be applied:

- Concentrated and distributed (lateral and axial) forces;
- Concentrated and distributed (bending and torsional) moments;

Table 4.1 The straight pipeline loads

Types of loads	Distribution to a section and length
Temperature on a pipe segment	It is possible to specify values on extreme points of horizontal diameter and vertical diameter and an element by the length. There is a linear distribution in the intermediate points.
Temperature on nodes	The constant in a cross-section and linear by length.
Pressure (internal and external)	The constant by a cross-section and length

The element mass matrix is completely similar to a mass matrix of a beam element, excluding:

$$A = A^w = \frac{\pi}{4}\left(D_o^2 - D_i^2\right)$$ - Sectional area of the tube

$$I_y = I_z = I = \frac{\pi}{64}\left(D_o^4 - D_i^4\right)\frac{1}{C_f}$$ - Moment of inertia

$$J = \frac{\pi}{32}\left(D_o^4 - D_i^4\right)$$ - Torsion moment of inertia

Where: D_0 - outer diameter;

D_i - internal diameter;

C_f - flexibility

t - tube wall thickness.

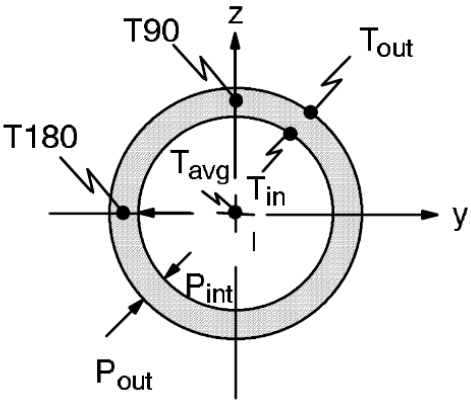


Fig. 4.41 Application of pressure and temperature load in the cross-section of a pipeline

Axial strain in a pipeline

$$\epsilon_x^{pr} = \frac{1}{E}(1-2\nu)\left(\frac{P_i D_i^2 - P_o D_o^2}{D_o^2 - D_i^2}\right),$$

Where: ν is the Poisson ratio;

P_i - internal pressure on pipeline wall;

P_o - external pressure on pipeline wall;

E - Young's modulus of pipeline material.

Stresses Calculation

The components of stresses in a pipeline are determined by the following dependences:

$$\sigma_{dir} = \frac{F_x + \frac{\pi}{4}\left(P_i D_i^2 - P_o D_o^2\right)}{a^w}$$ - Magnitude of axial stresses;

$$\sigma_{bend} = C_\sigma \frac{M_b r_o}{I_r}$$ - Stresses due to bending;

$$\sigma_{tor} = \frac{M_x r_o}{J}$$ - Stress due to torsion;

$$\sigma_h = \frac{2 P_i D_i^2 - P_o\left(D_o^2 + D_i^2\right)}{D_o^2 - D_i^2}$$ - Stresses due to pressure;

$$\sigma_{\ell f} = \frac{2 F_s}{A^w}$$ - Shear stresses from the lateral force.

Where: F_x is an axial force;

$a^w = A^W$ – area of the pipe cross-section;

F_S is $\sqrt{F_y^2 + F_x^2}\sqrt{F_y^2 + F_x^2}$ a lateral force;

C_σ is a stress concentration ratio;

M_b is a $\sqrt{M_y^2 + M_x^2}\sqrt{M_y^2 + M_x^2}$ bending moment;

M_x is a torque;

Curved pipe with constant radius

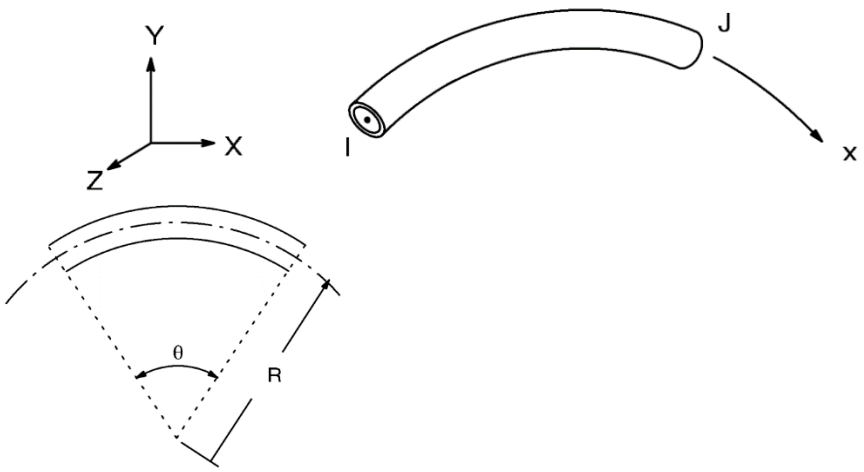


Fig. 4.42 The Scheme of a flat curved area of a pipeline

The following types of loads can be placed to the curved area of a pipeline:

- Concentrated and distributed forces (lateral and axial);
- Concentrated and distributed moments (bending and torsional);

Table 4.2 – Loads on a curved section a pipeline

Types of loads	Distribution by a section and length
Temperature on pipe segment	It is possible to specify values on extreme points of horizontal diameter and vertical diameter and an element by the length. There is a linear distribution in the intermediate points.
Temperature on nodes	The constant in a cross-section and linear by length.
Pressure (internal and external)	The constant by a cross-section and length

$$\sqrt{F_y^2 + F_x^2}\sqrt{M_y^2 + M_x^2}$$

Calculation of contact interaction

Calculation of contact interaction is lead within the scope of nonlinear calculation, under the assumption of small displacements and elastic strains. During calculation, the fictitious elements connecting contacting surfaces, and, depending on reciprocal displacements nodes of elements are

created, at each iteration forces in contact area are specified and static calculation is performed. Convergence criterion is the condition of the minimal interpenetration of objects.

Calculation results of contact interaction are all the components available after static calculation, and also distribution field of normal and tangential forces, interpenetration and a condition of contact elements in contact area.

Design and checking of reinforced concrete elements

After static and code combination calculation it is possible to design and check reinforcing of concrete elements according to building regulations[5].

✍ *Notes*

Calculations which are not performed:

- Shell elements on action of lateral forces and torsion moments.
- Rod elements of round and ring sections - in a case of eccentric tension on lateral forces action, torsion moments, and also on the second group of limiting states.
- Rod elements of T and I sections - on action of torsion moments.
- Rod elements of rectangular, T and I sections - in case of diagonal eccentric tension with the small compressed zone (<1.5*a, where a – concrete cover). In case of diagonal eccentric tension without the compressed zone calculation is performed in separate directions.

Design and checking of reinforced masonry elements

After static and code combination calculation it is possible to design and check reinforcing of masonry elements according to building regulations[6].

Batch calculations

APM Structure3D allows to perform batch calculations for files of *.frm format. First it can be used if necessary to calculate some problems, which demands considerable time for calculation. And secondly it allows to use operating time rationally when necessary to perform calculations for many various files.

To perform batch calculations launch **Start / Programs / APM WinMachine … / Batch_Structure3D.** After that Batch window appears on the screen. In the files list it will be necessary to set a path to *.frm files which calculation will be started in a batch mode with the help of **Add file folder** and **Add file** buttons.

✍ *Note: Only ***.frm** files containing in the specified folder will be added in the list. Entering folders will not open, as well as files of other types, being in the specified folder will be ignored.*

[5] SP 52-101-2003
[6] SNiP II-22–81*

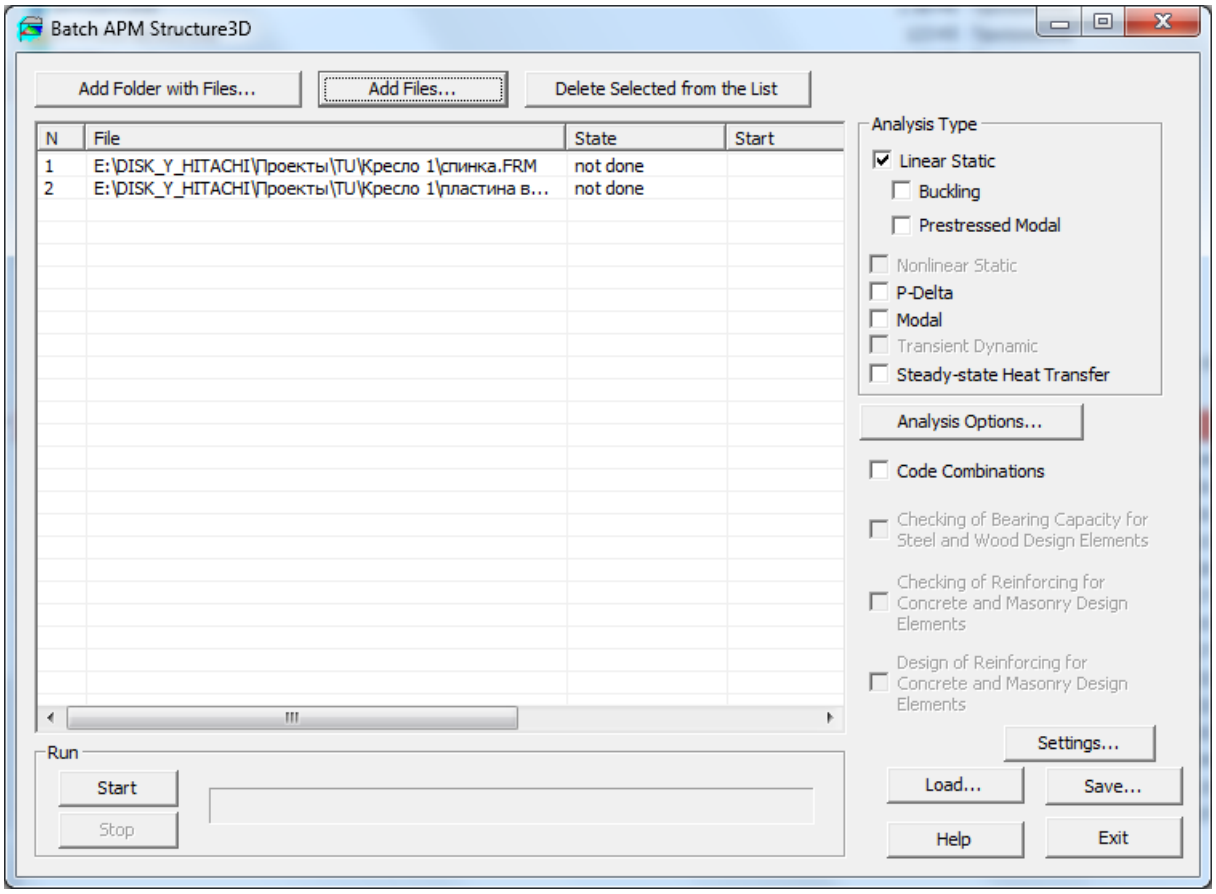


Fig. 4.43 Batch Structure3D window

The files added in the list are numbered as their addition and this numbering will be defining calculation order. The order of calculation procedure can be changed; drag file to the required place with holding left mouse button.

For the selected files it is necessary to choose in *Calculation type* frame what calculation will be performed. **Calculation Options** button invokes dialog box where calculation options can be specified.

Buckling, P-Delta and Prestressed Modal analyses will be performed for that load case which is active at the moment of file saving.

Settings button invokes dialog box in which all options correspond to *APM Structure3D* possibilities except *Autosaving*.

The list of files can be saved in a file with name extension *.batchfrm using **Save** button and with the help of **Open** button it can be loaded.

To perform calculations use **Start** button. After calculation files will be saved in the same directory with addition part of initial file name, for example ***.autosave.2009.07.08.13.42.25.FRM**. It is possible to stop calculation by means of **Stop** button.

Chapter 5. Design Elements, Soil Bases and Foundations

Design elements general information

The design element models physically homogeneous element of a design - column, girder, plate. The design element is one or more finite elements representing the model part located between two joints (columns, girders) which are considered as a unit at calculation according to building regulations[7].

The command **Design / Design elements** invokes dialog window for editing of properties, deletion of design elements, and also viewing of calculation results.

Design element is considered as continuous chain of rod or plate elements with defined properties. If the design element cannot be created, the system displays warning. In a warning window the list of all restrictions on design element creation is presented. The reasons are checked on which the given design element cannot be created.

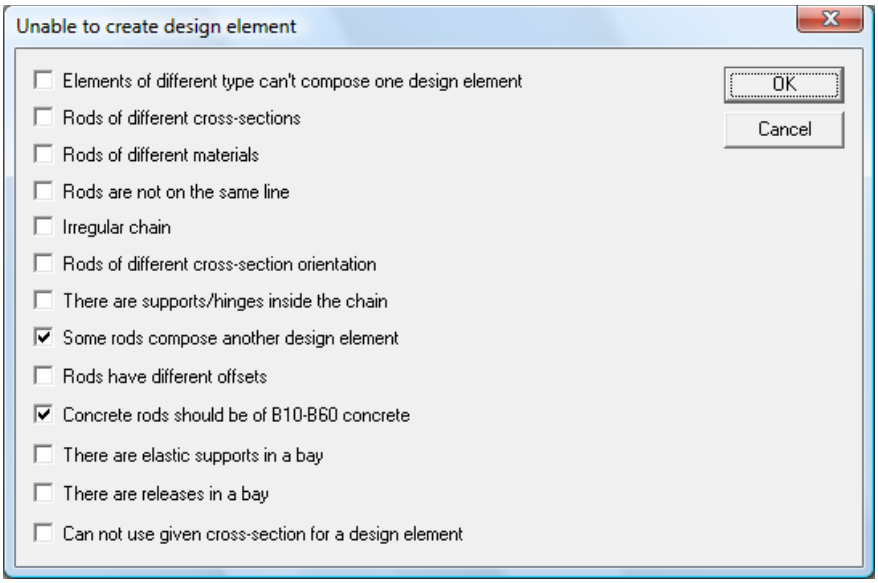


Fig. 5.1 Unable to create rod design element warning

System will display window and offer variants of continuation in case of model modification affecting design elements.

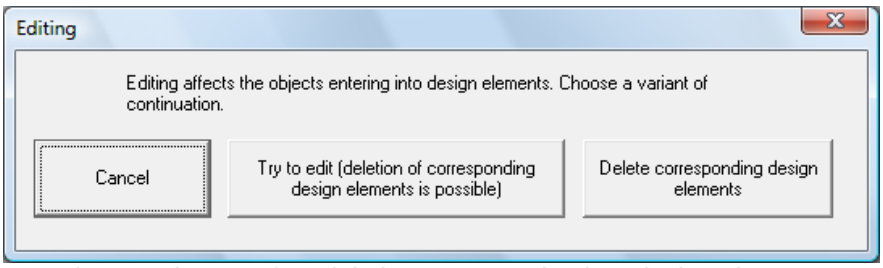


Fig. 5.2 Change of model elements entering into design elements

General principles of work with design elements coincide with ones of layers (see above).

[7] SNiP II-23-81*, STO 36554501-002-2006, SP 52-101-2003 or SNiP II-22-81*

Steel / Wood design elements

First, it is necessary to set type of design using **Design / Steel/Wood elements** menu command. Further required objects are selected and added in active design element by pressing **Selected Objects to Design Element** button. If there is no design elements the new design element is automatically created. After that check of bearing capacity can be performed.

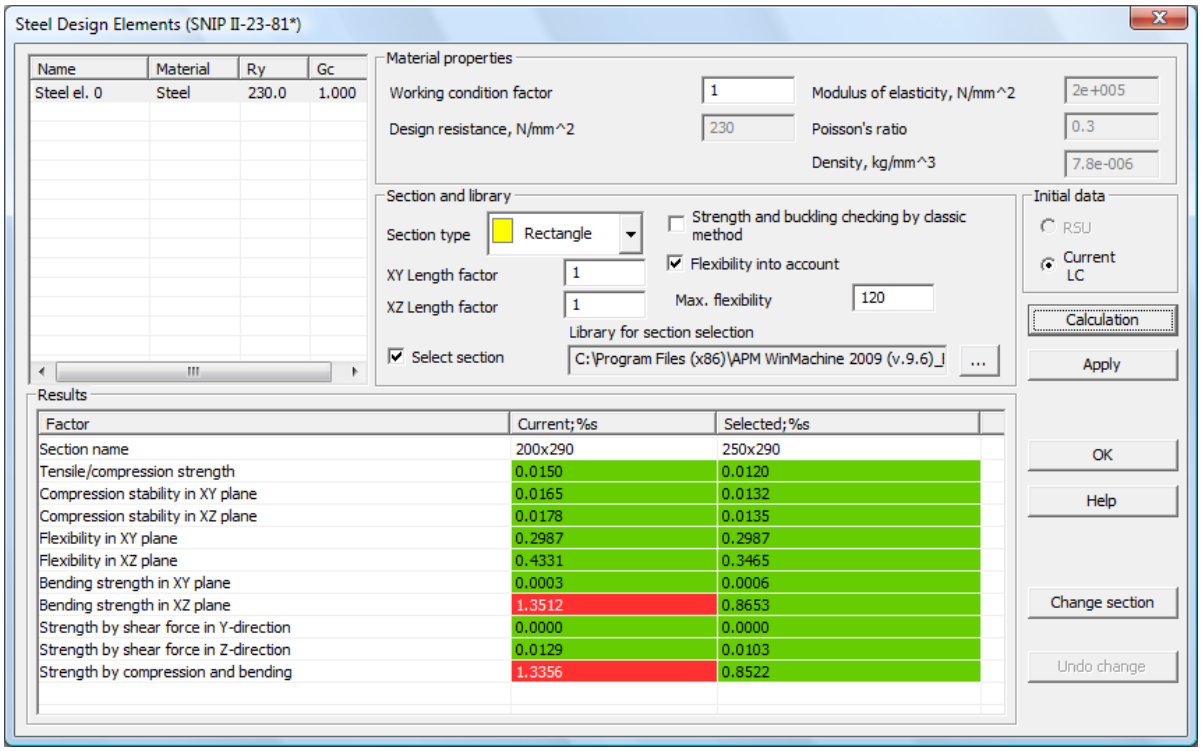


Fig. 5.3 Steel Design Elements dialog box

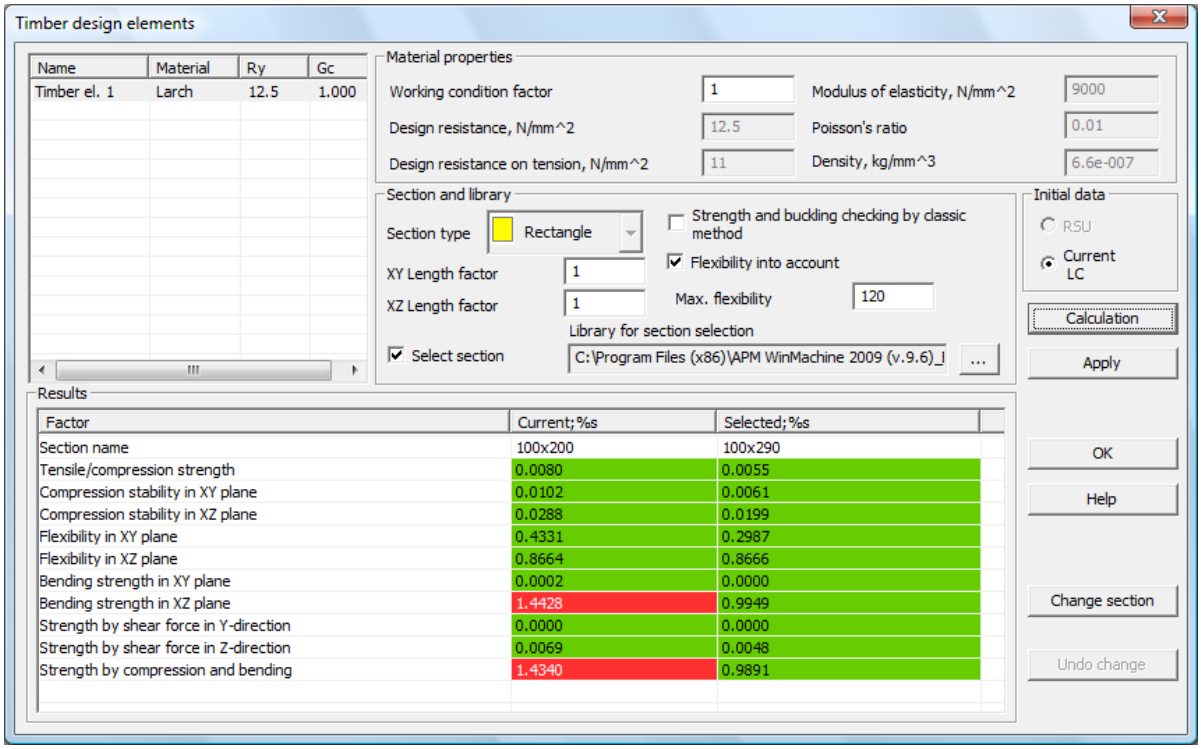


Fig. 5.4 Wood Design Elements dialog box

In the left upper part of dialog there is a list of design elements. The rods entering into the selected element are highlighted in a current model view.

For each design element it is necessary to set its properties such as material (*modulus of elasticity, Poisson's ratio* and *density*), section type, and also some parameters defined in SNiP.

Table 5.1 – Design element parameters

Parameters	***SNiP II-23-81****	***STO 36554501-002-2006***
Design resistance – relation of material yield stress to safety factor by material. It is inaccessible for edit.	Table 2, Table 51	Table 3, 4.
Working condition factor – depends on structure destination.	Table 6.	i. 3.2, 3.3, 3.4.
Length factor – relation of effective length of design element to real length.	i. 6.1-6.7, 5.13-5.16.	i. 4.21.
Flexibility limit	i.6.15, 6.16	Table 14.

Building regulations determine various checks for various sections of steel structures, therefore for each design element it is necessary to set *section type*. If the section is taken from library with the predetermined type the library will be selected automatically.

Strength and buckling checking by classic methods – check of bearing capacity will be made using classical methods without taking into account the existing building regulations, using classical formulas of strength of materials.

Flexibility into account – this mark will add the maximum flexibility to the list of criteria for section selection.

The *Initial Data* group is used for initial data choice for checking bearing capacity of design elements.

Besides, it will be necessary to specify the library with the required sections if it is planned to select a section for design element. The **Load library** button is used for this purpose.

To edit design element properties select it the *Design elements* list, set required parameters and press **Apply** or **Apply to All** (it properties need to be applied for all design elements) buttons.

After any parameter modification it is necessary to confirm change by pressing **Apply** button or **Apply** to All.

Reinforced design elements

First, it is necessary to set design element type using **Design / Design Element Type / Reinforced Elements** command. Further demanded objects are selected and added in automatically created design element by **Design /** **Selected Objects to Design Element** or **Design /** **Selected Objects to Reinforced Masonry Design Element** commands. To add object to separate design element use **Design /** **Selected Objects to Separate Design Elements** or **Design /** **Selected Objects to Separate Reinforced Masonry Design Elements** commands. The further work with design elements is made in *Design Elements* dialog box which is invoked by **Design /** **Design Elements** command.

Calculation types

Design calculation is performed to select reinforcing for the most adverse load combination. To select reinforcing for all design elements use **Calculation / Design of Reinforced Elements** command.

Checking calculation is performed, as a rule, for existing structures at changing of loads, service conditions and also when detects serious structure defects.

At transition from design to checking calculation the system suggests to use results of design calculation. Thus, checking calculation can be performed on the basis of the corrected design calculation results.

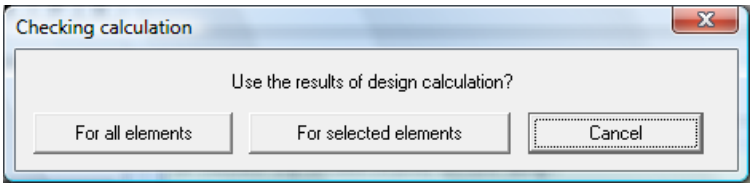


Fig. 5.5 Checking calculation dialog box

To check reinforcing for all design elements use **Calculation / Checking of Reinforced Elements** command.

Building regulations used at calculations of design elements:

- SP 51-101-03 Concrete and reinforced concrete structures without prestressed reinforcing.
- SNiP 52-01-03 Concrete and reinforced concrete structures. General norms.
- GOST 5781-82 Hot-rolled steel for reinforced concrete structures.
- SNiP II-22-81* Stone and reinforced stone structures.
- Stone and reinforced stone structures design manual (SNiP II-22-81*).
- SP 63.13330.2012 Concrete and reinforced concrete structures. Basics.

Design Elements dialog box

There is *calculation type* drop-down list in the left top corner of dialog box. Below it there is a list of design elements. The objects entering into the selected design element are highlighted in a current view of structure editor. To delete the selected design element from the list press DELETE key.

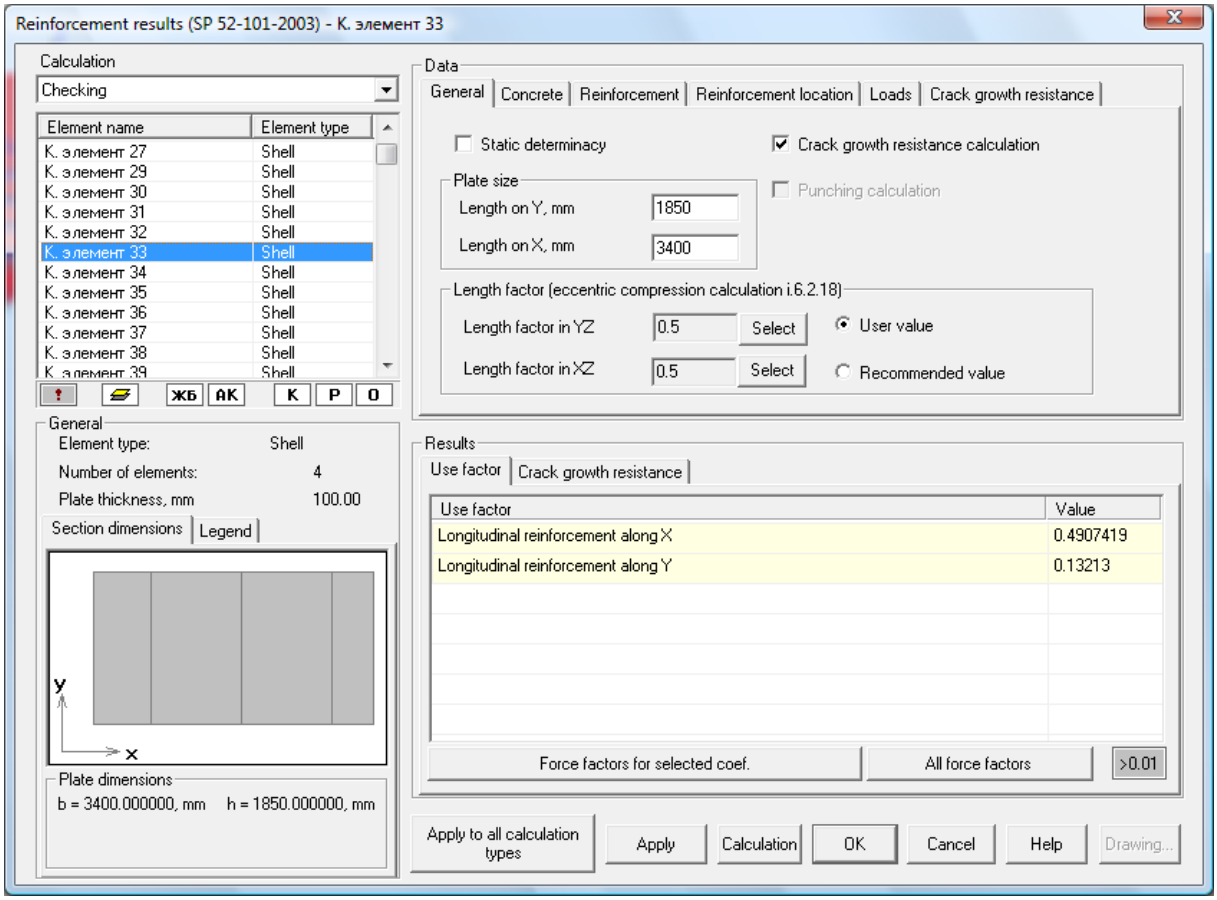


Fig. 5.6 Design Elements dialog box

View filters of Design Elements dialog box

View filters of this dialog are used for convenient viewing of different design elements in the list. White color of buttons - adjustment is on, grey color - adjustment is off.

 / Show elements with negative results (reinforcing is not design or don't meet the conditions by checking calculation) / Show all elements

Show only elements with the inability to calculate / show elements with any result

Show only elements without longitudinal reinforcement / Show all elements

Show elements from visible layers only / Show elements from all layers. This option is convenient for synchronization of results displaying with a reinforcing map

Show / Hide reinforced concrete elements

Show / Hide reinforced masonry elements

Show / Hide design elements – columns

Show / Hide design elements – girders

Show / Hide design elements – shells

Work with group of elements

Usually single reinforcing schemes are used for groups of elements. To change parameters of several design elements it is necessary to select their elements simultaneously. It is possible to select elements both on design model, and in the list of design elements of a dialog.

1. Elements selection on model (Design Elements is closed) is convenient, for example, for floor objects selection. To select several elements use **Edit / Select Element** command. To select elements group hold SHIFT key. If it is convenient to select elements with frame use **Edit / Select Group** command. The selected elements will be highlighted by red color and by grey color in *Design Elements* dialog box. Further it is possible to change parameters and press **Apply** or **Apply to all calculation types** buttons.

2. To select elements in the list of a dialog box specify interesting elements using mouse holding CTRL key. To deselect one element it is necessary to click on it again holding CTRL key.

Design Elements dialog box buttons

Apply to all calculation types button	saves changes for the selected design elements for all calculation types
Apply button	saves changes for the selected design elements for current calculation type
Calculation button	performs calculation for selected design elements
Drawing button	generates drawing of column or girder for selected design element

Reinforced concrete design elements (shells)

Data

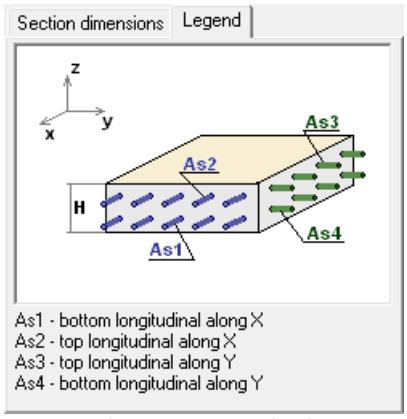
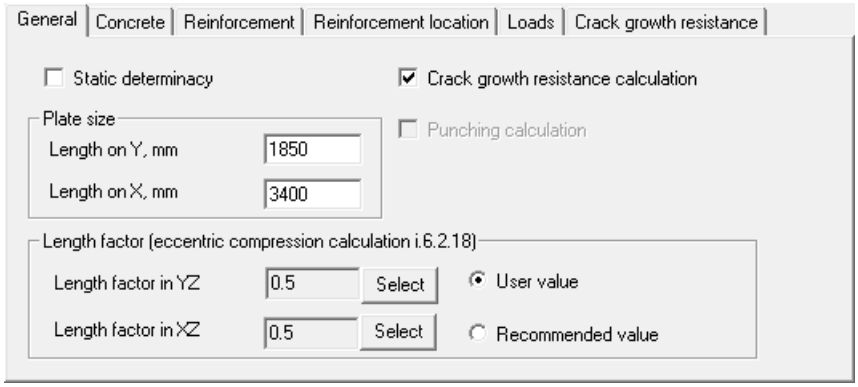


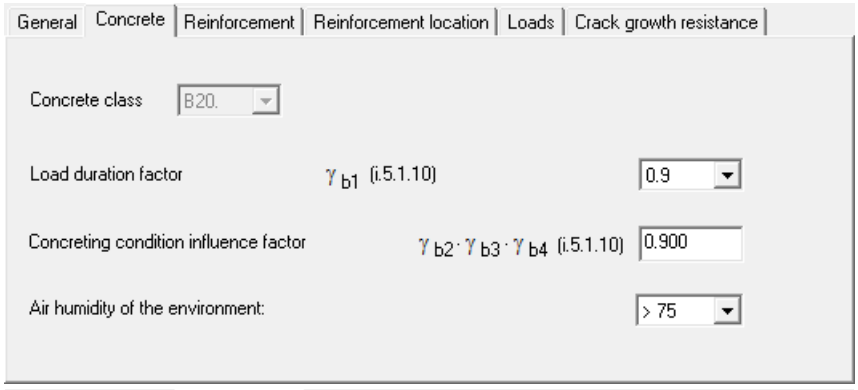
Fig. 5.7 Legend tab

Designation and color of reinforcing are presented in a legend.

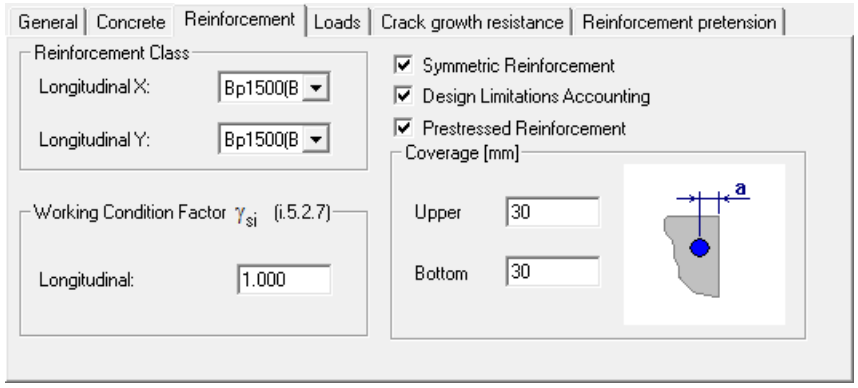
There are tabs in the dialog for setting initial data: *General*, *Concrete*, *Reinforcement*, *Reinforcement location*, *Loads* and *Crack growth resistance*. They are described in details below.



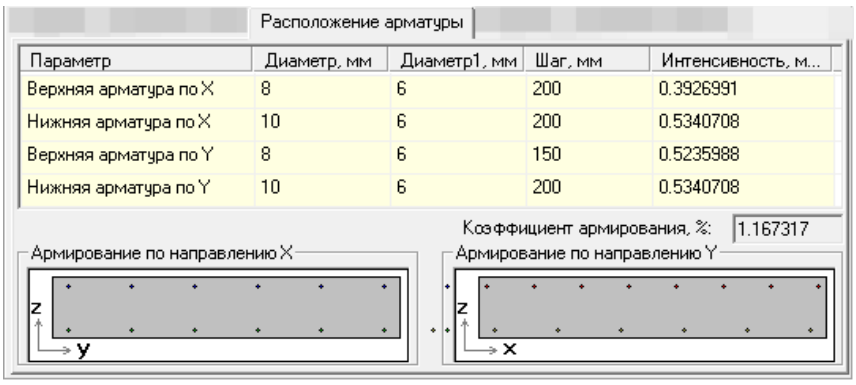
a)



b)



c)



Параметр	Диаметр, мм	Диаметр1, мм	Шаг, мм	Интенсивность, м...
Верхняя арматура по X	8	6	200	0.3926991
Нижняя арматура по X	10	6	200	0.5340708
Верхняя арматура по Y	8	6	150	0.5235988
Нижняя арматура по Y	10	6	200	0.5340708

d)

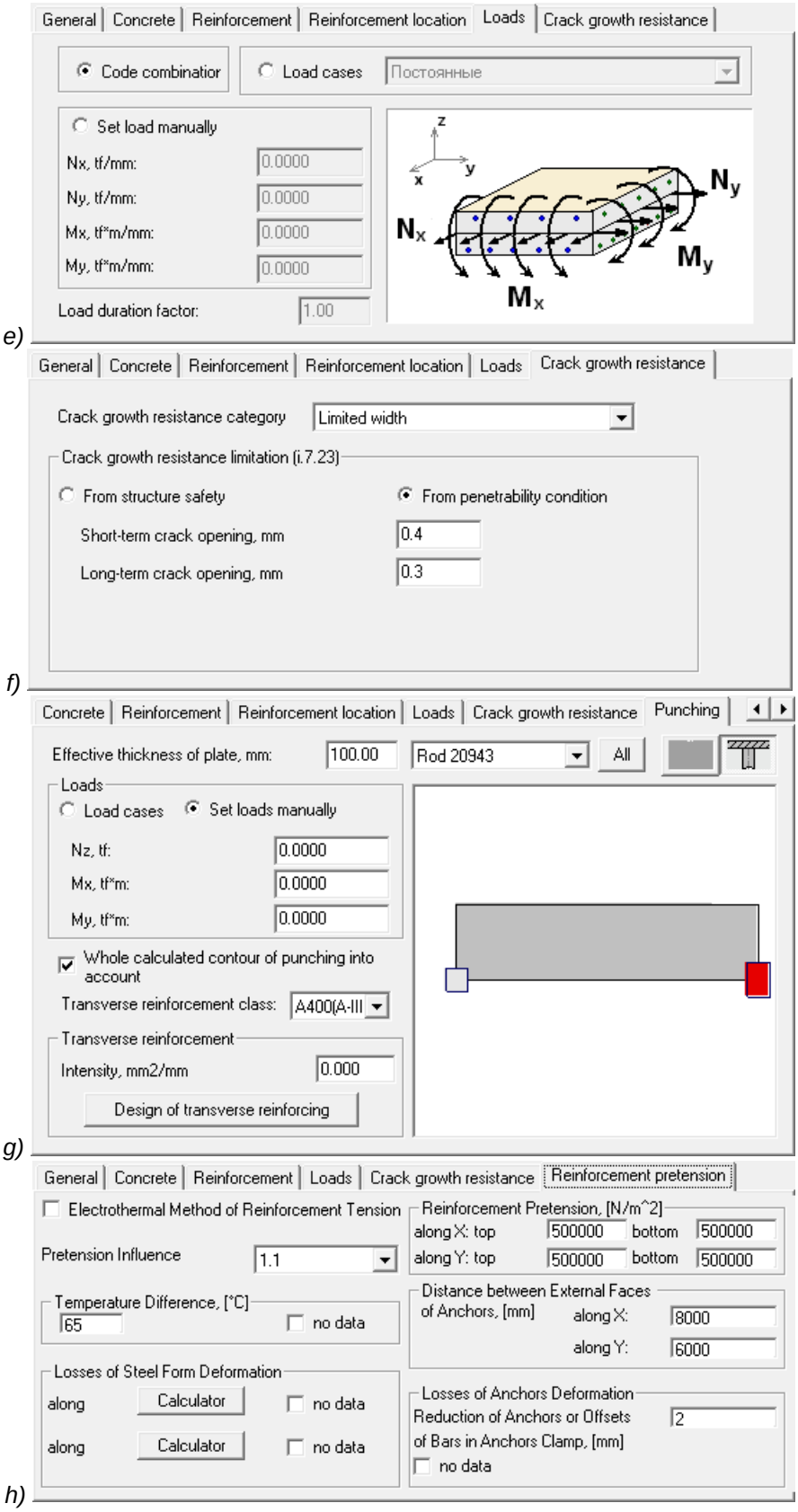


Fig. 5.8 Tabs of Data group

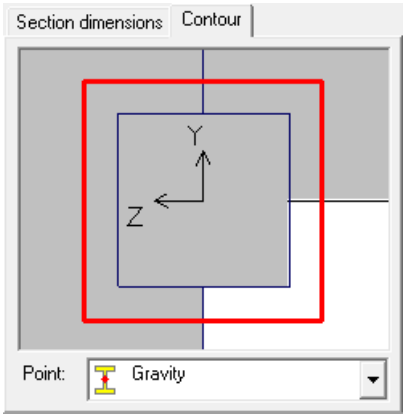


Fig. 5.9 Support type image

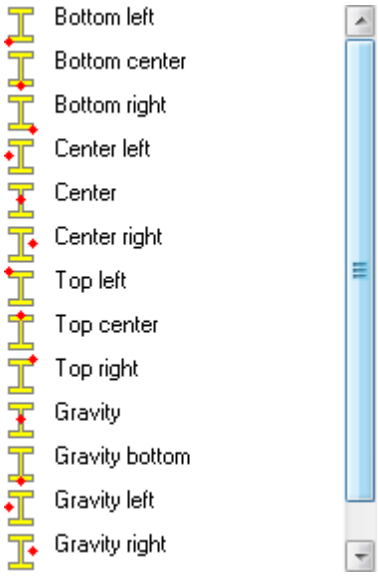


Fig. 5.10 Possible variants of columns and plates supporting

Table 5.2 – Data group parameters

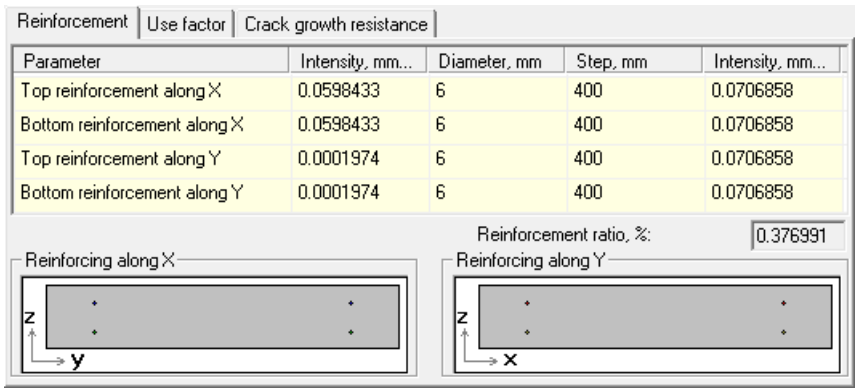
Tabs	Parameters	Comments
General	Static determinacy	Parameter must be unchecked for statically indeterminate structures. I. 4.2.6 SP 52-101-2003
	Length on YZ*, mm	I. 6.2.18 SP 52-101-2003 as l;
	Length factor in YZ	I. 6.2.18 SP 52-101-2003;
	Length on XZ*, mm	I. 6.2.18 SP 52-101-2003 as l
	Length factor in XZ	I. 6.2.18 SP 52-101-2003
	Crack growth resistance calculation	Calculations by II group of limiting states
Concrete	Concrete class**	I. 5.1.3 SP 52-101-2003, I. 5.1 SNiP 52-01-2003
	Load duration factor	I. 5.2 SNiP 52-01-2003, I. 5.1.10 SP 52-101-2003
	Concreting condition influence factor	
	Air humidity of the environment	T. 5.5-5.6 SP 52-101-2003
Reinforcement	Reinforcement class	I. 5.3 SNiP 52-01-2003, I. 5.2.3 SP 52-101-2003, I.1 GOST 5781-82, T. 5.7-5.8 SP 52-101-2003.
	Coverage	I. 7.3.2 SNiP 52-01-2003. I. 8.3.2 SP 52-101-2003.
	Working condition factor	I. 5.2.7 SP 52-101-2003.
	Symmetric reinforcement (for design calculation)	Design of symmetric reinforcement (upper and bottom)
	Design limitations accounting	1. Concrete cover – i.8.3.2 – >= 10 mm. or bar diameter 2. Distance between bars (longitudinal and transverse) i.8.3.3 – >=25 mm or bar diameter 3. Distance between longitudinal bars – i.8.3.6 – if h<=150 mm – <= 150 mm, if h>150 mm – 1.5*h or 400 mm. 4. Distance between transverse bars at punching – i.8.3.15 – <= 1/3*h0 or <= 300 mm.

Reinforcement location (for checking calculation only)	Diameter, mm	i.1.4 GOST 5781-82
	Diameter1, mm	Specifying alternate rebar
	Step, mm	i.1.4 GOST 5781-82
	Intensity, mm^2/mm	Determined automatically
	Reinforcement ratio, %	Reinforcement ratio on area
Loads	Code combination	Case variants
	Load case	
	Set loads manually	
	Load duration factor	i. 6.2.16 SP 52-101-2003
Crack growth resistance (if option is checked in General tab)	Crack growth resistance category	Crack disabled or Limited width
	Crack growth resistance limitation	By structure safety or Penetrability condition i. 7.2.3 SP 52-101-2003.
	Reinforcement diameter for crack growth resistance calculation (design calculation)	Reinforcement diameter for auto design
Reinforcement pretension	Electrothermal method of reinforcement pretension	Losses is not concidered (i. 2.2.3.5 SP 52-102-2004).
	Pretension influence	i. 2.1.2.3 SP 52-102-2004
	Temperature difference, [°C]	i. 2.2.3.4 SP 52-102-2004
	Losses of steel form deformation	i. 2.2.3.5 SP 52-102-2004, losses depends on: - number of nonsimultaneously pretensioned bars (group of bars); - approaching of anchors along reinforcement tension line by calculation of form deformation, mm.
	Reinforcement pretension, [N/m^2]	Pretension of bars (group of bars) according to i. 2.2.3.9 SP 52-102-2004
	Distance between external faces of anchors, [mm]	i. 2.2.3.5 SP 52-102-2004
	Losses of anchors deformation	i. 2.2.3.6 SP 52-102-2004 - reduction of anchors or offsets of bars in anchors clamp, [mm]

Notes:
* Length as overall dimension of design element by default.
** Corresponds to a plate material.

Results

There are three tabs for viewing results: *Reinforcement, Use factor and Crack growth resistance.* They are described in details below.



a)

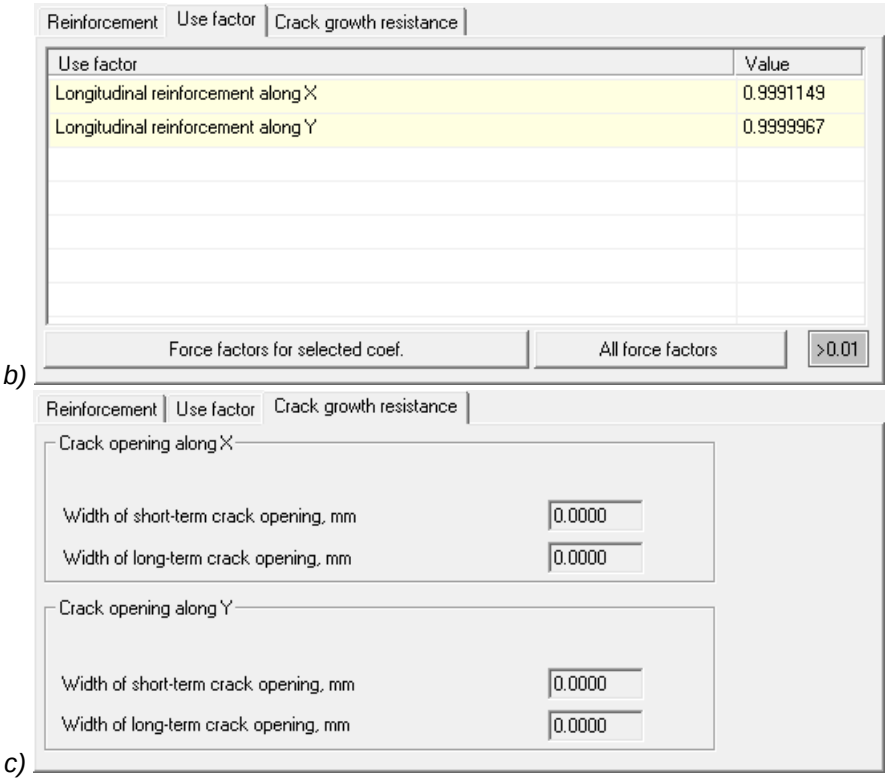


Fig. 5.11 Tabs of Results group

Table 5.3 – Results group parameters

Tabs	Parameters	Comments
Reinforcement (for design calculation only)	Calculated intensity, mm^2/mm	Required intensity of reinforcing along direction
	Diameter, mm	According to GOST 5781-82
	Step, mm	Selects from normal dimension series
	Real intensity, mm^2/mm	Intensity on the basis of the accepted reinforcing
Use factor	Use factor of reinforcement	Must be in range from 0 to 1
	All force factors	Invokes dialog box with force factors
	Force factors for selected coef.	Invokes dialog box with force factors for selected use factor
	>0.01 / >0.01	If button is pressed (>0.01) there are only use factors with values more than 0.01.
Crack growth resistance (if option is checked in General tab)	Reinforcement diameter, mm	GOST 5781-82
	Width of short-term crack opening, mm	Real width of crack opening
	Width of long-term crack opening, mm	

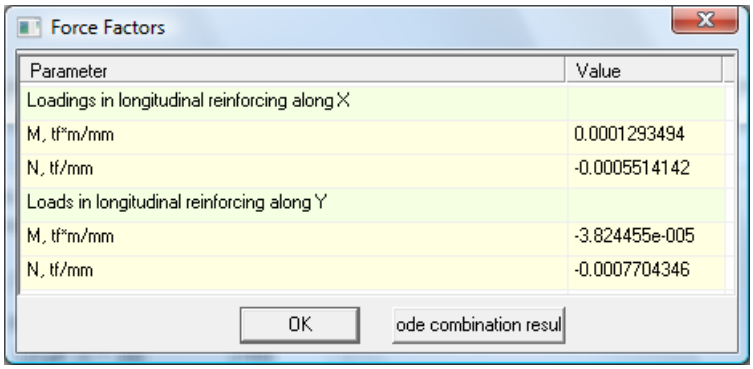


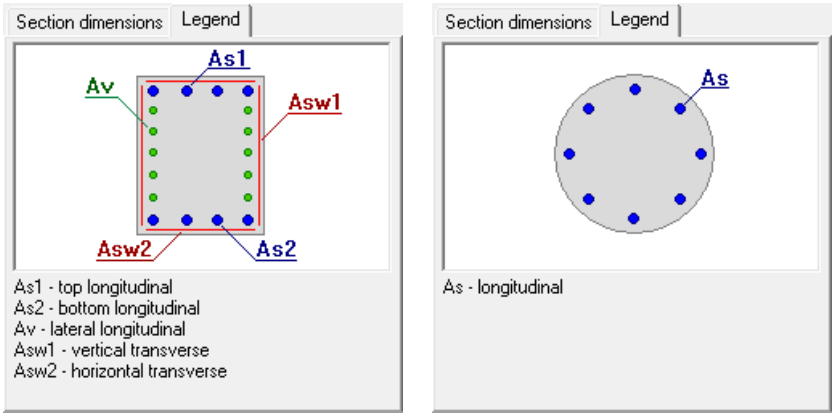
Fig. 5.12 Force factors dialog box

Reinforced concrete design elements (rods)

There are two types of rod reinforced concrete design elements: girder and column. Column is a vertical element, girder - horizontal. If the element has an angle with a horizontal plane more than 45 degrees - it is considered as column.

Each element has the local coordinate system, in which the X axis is directed along a rod, and Z and Y makes right-hand system.

The designation and color of reinforcing are presented in *Legend* tab.



a) *b)*

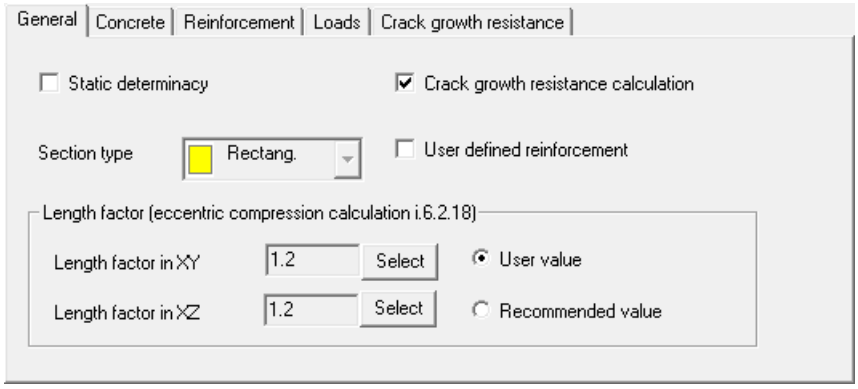
Fig. 5.13 Legend tab for various section types

Features of rod reinforcing design

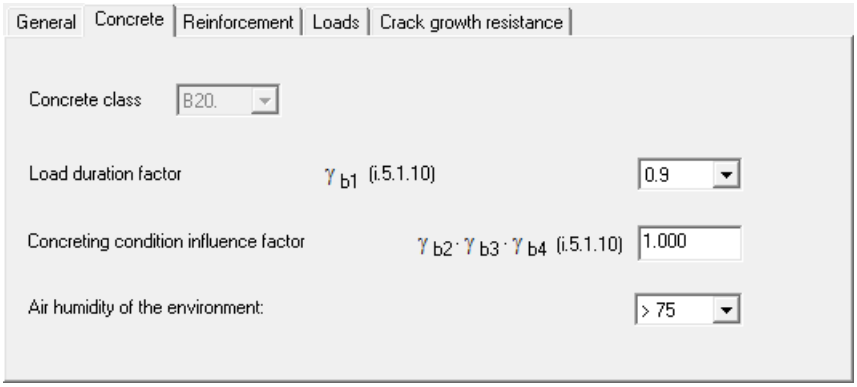
- It is impossible to design reinforcing for round and ring sections of girders.
- Minimum number of upper and bottom reinforcement – 2.
- Lateral reinforcement is always symmetric.

Data

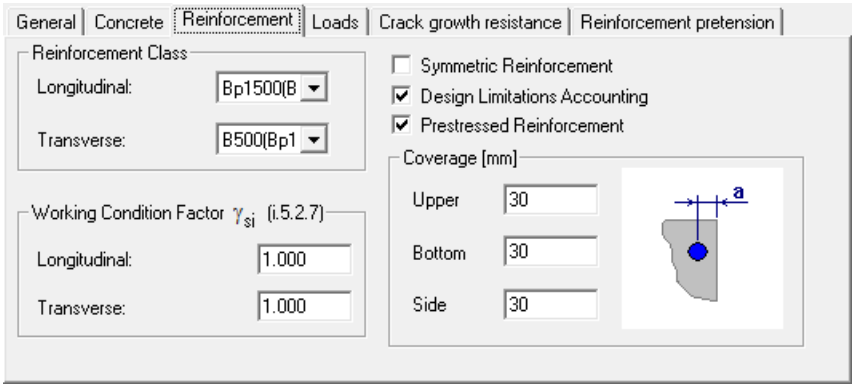
There are tabs in the dialog for setting initial data: *General*, *Concrete*, *Reinforcement*, *Reinforcement location*, *Loads* and *Crack growth resistance*. They are described in details below.



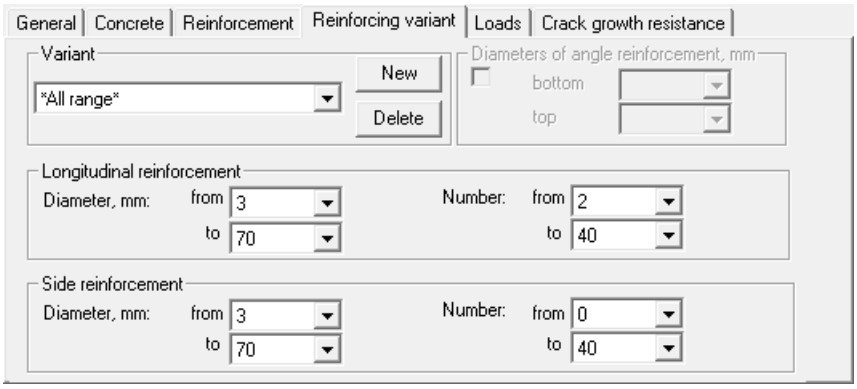
a)


General
Concrete
Reinforcement
Loads
Crack growth resistance
Concrete class
B20.
Load duration factor
γ b1 (i.5.1.10)
0.9
Concreting condition influence factor
γ b2 · γ b3 · γ b4 (i.5.1.10)
1.000
Air humidity of the environment:
> 75

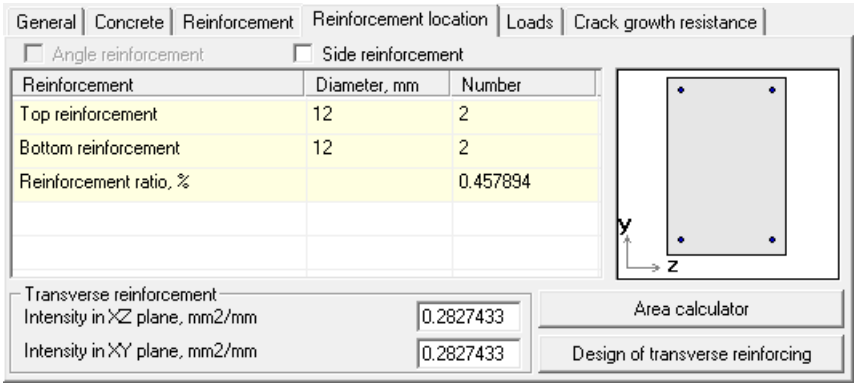
b)


General
Concrete
Reinforcement
Loads
Crack growth resistance
Reinforcement pretension
Reinforcement Class
Longitudinal:
Bp1500(B
Transverse:
B500(Bp1
Symmetric Reinforcement
Design Limitations Accounting
Prestressed Reinforcement
Coverage [mm]
Upper
30
Bottom
30
Side
30
a
Working Condition Factor γsi (i.5.2.7)
Longitudinal:
1.000
Transverse:
1.000

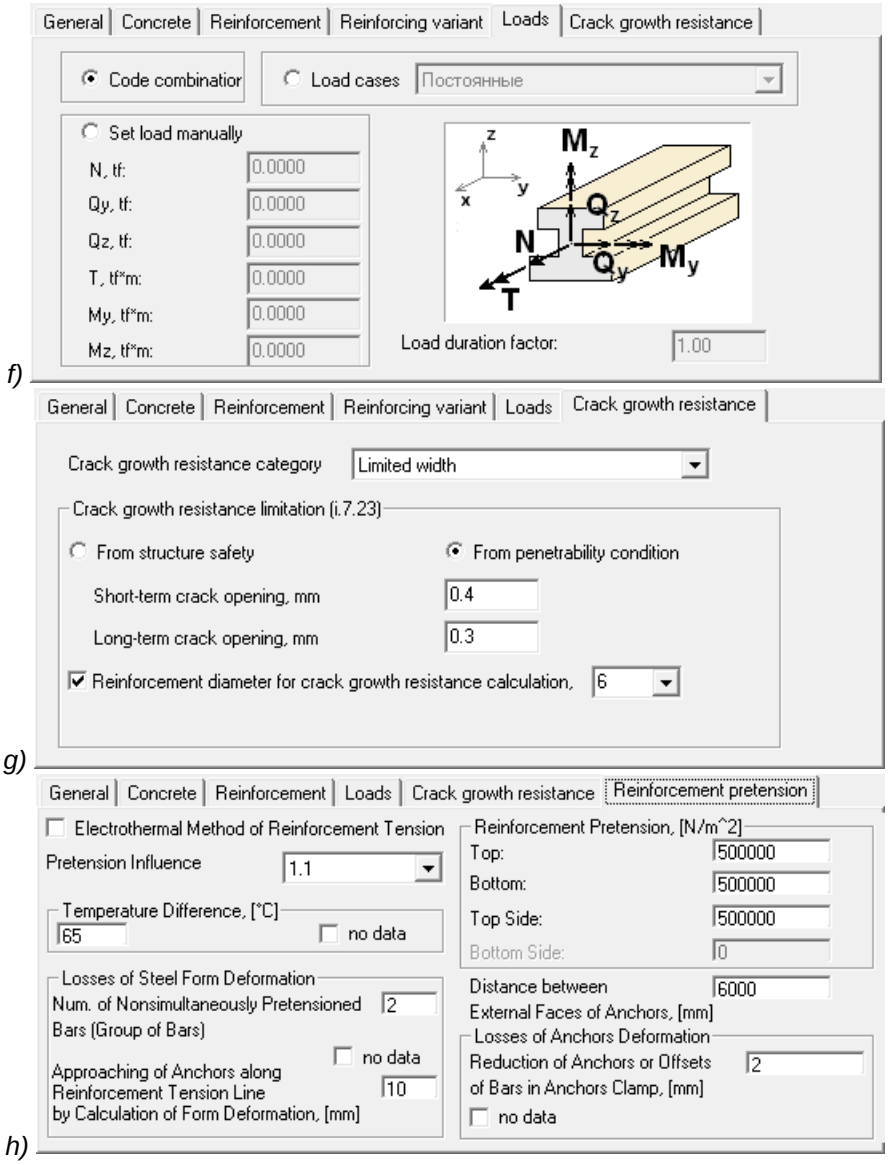
c)


General
Concrete
Reinforcement
Reinforcing variant
Loads
Crack growth resistance
Variant
All range
New
Delete
Diameters of angle reinforcement, mm
bottom
top
Longitudinal reinforcement
Diameter, mm:
from
3
to
70
Number:
from
2
to
40
Side reinforcement
Diameter, mm:
from
3
to
70
Number:
from
0
to
40

d)


General
Concrete
Reinforcement
Reinforcement location
Loads
Crack growth resistance
Angle reinforcement
Side reinforcement
Reinforcement
Diameter, mm
Number
Top reinforcement
12
2
Bottom reinforcement
12
2
Reinforcement ratio, %
0.457894
y
z
Transverse reinforcement
Intensity in XZ plane, mm2/mm
0.2827433
Intensity in XY plane, mm2/mm
0.2827433
Area calculator
Design of transverse reinforcing

e)



f)

g)

h)

Fig. 5.14 Tabs of Data group

Table 5.4 – Data group parameters

Tabs	Parameters	Comments
General	Static determinacy	Parameter must be unchecked for statically indeterminate structures. I. 4.2.6 SP 52-101-2003
	Section type	Determines automatically
	Length factor in XY	I. 6.2.18 SP 52-101-2003
	Length factor in XZ	I. 6.2.18 SP 52-101-2003
	Crack growth resistance calculation	Calculations by II group of limiting states
	User defined reinforcement	Allows to set user defined reinforcing variant
Concrete	Concrete class*	I. 5.1.3 SP 52-101-2003; I. 5.1 SNiP 52-01-2003
	Load duration factor	I. 5.2 SNiP 52-01-2003, I. 5.1.10 SP 52-101-2003
	Concreting condition influence factor	
	Air humidity of the environment	T 5.5-5.6 SP 52-101-2003

Reinforcement	Reinforcement class	I. 5.3 SNiP 52-01-2003, I. 5.2.3 SP 52-101-2003, I.1 GOST 5781-82, T 5.7-5.8 SP 52-101-2003
	Coverage	I. 7.3.2 SNiP 52-01-2003, I. 8.3.2 SP 52-101-2003
	Working condition factor	I. 5.2.7 SP 52-101-2003
	Symmetric reinforcement (for design calculation)	Design of symmetric reinforcement (upper and bottom)
	Design limitations accounting	1. Concrete cover – i.8.3.2 – >= 10 mm. or bar diameter 2. Distance between bars (longitudinal and transverse) i.8.3.3 – >=25 mm or bar diameter 3. Distance between longitudinal bars – i.8.3.6 – if h<=150 mm – <= 150 mm, if h>150 mm – 1.5*h or 400 mm.
Reinforcement location (for checking calculation only)	Diameter, mm	Longitudinal, transverse and lateral reinforcement
	Number	
	Intensity of transverse reinforcement	Intensity of transverse reinforcement along directions
	Area calculator	Design of longitudinal reinforcement area
	Design of transverse reinforcement	Design of longitudinal reinforcement intensity
Reinforcing variants (for design calculation only)	Variant	User defined reinforcing variants
	Diameter, mm	Design reinforcing
	Number	
Loads	Code combination	Case variants
	Load case	
	Set loads manually	
	Load duration factor	I. 6.2.16 SP 52-101-2003
Crack growth resistance (if option is checked in General tab)	Crack growth resistance category	Crack disabled or Limited width
	Crack growth resistance limitation	By structure safety or Penetrability condition i. 7.2.3 SP 52-101-2003.
	Reinforcement diameter for crack growth resistance calculation (design calculation)	Reinforcement diameter for auto design
Reinforcement pretension	Electrothermal method of reinforcement pretension	Losses is not concidered (i. 2.2.3.5 SP 52-102-2004).
	Pretension influence	i. 2.1.2.3 SP 52-102-2004
	Temperature difference, [°C]	i. 2.2.3.4 SP 52-102-2004
	Losses of steel form deformation	i. 2.2.3.5 SP 52-102-2004, losses depends on: - number of nonsimultaneously pretensioned bars (group of bars); - approaching of anchors along reinforcement tension line by calculation of form deformation, mm.
	Reinforcement pretension, [N/m^2]	Pretension of bars (group of bars) according to i. 2.2.3.9 SP 52-102-2004
	Distance between external faces of anchors, [mm]	i. 2.2.3.5 SP 52-102-2004
	Losses of anchors deformation	i. 2.2.3.6 SP 52-102-2004 - reduction of anchors or offsets of bars in anchors clamp, [mm]

Note:
* Length as overall dimension of design element by default.

Area calculator is intended to design diameter and number of longitudinal reinforcement by reinforcement area or contrary. It is probably to use different diameters. The summary area of reinforcement is presented in the bottom part of dialog.

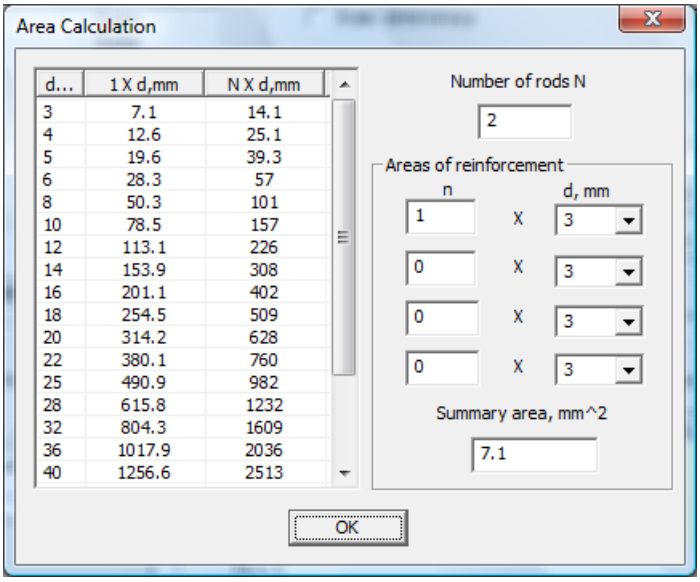


Fig. 5.15 Area calculator

Intensity and required transverse reinforcement number are calculated automatically in the *Transverse Reinforcing* dialog box.

After pressing **OK** button intensity will be set in *Reinforcement location* tab.

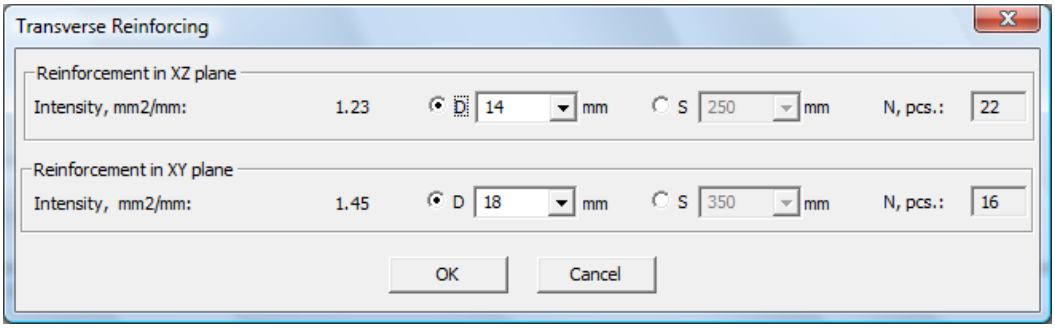
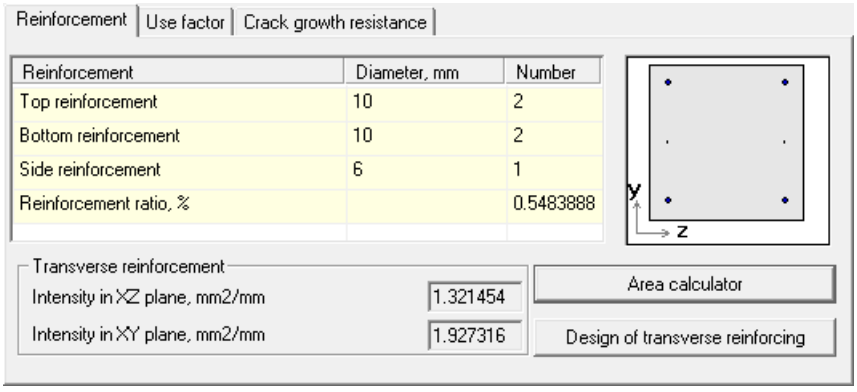


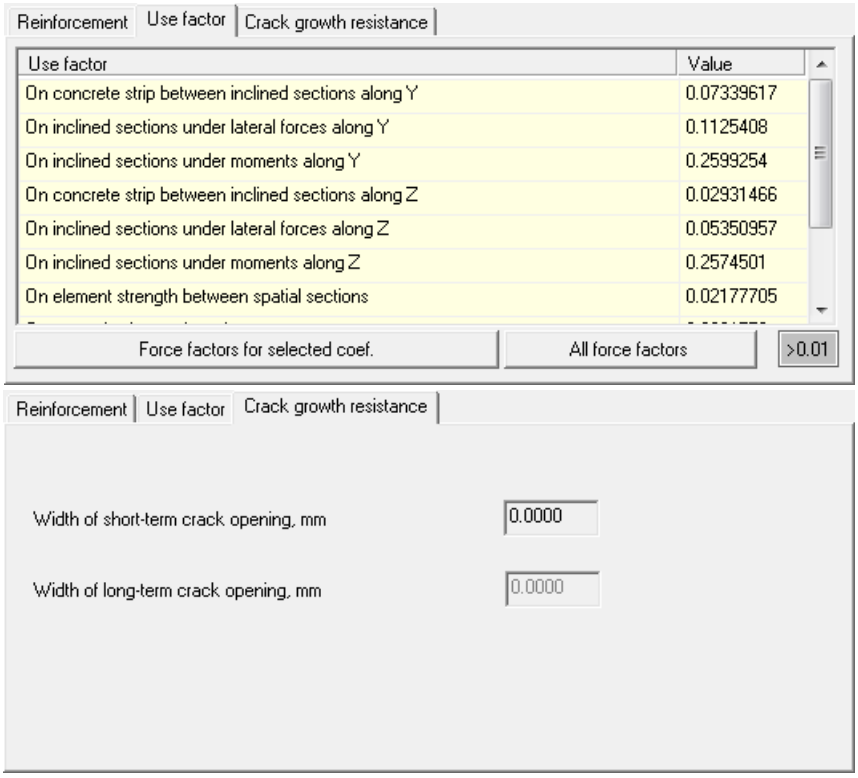
Fig. 5.16 Transverse Reinforcement design

Results

There are three tabs for viewing results: *Reinforcement, Use factor and Crack growth resistance.* They are described in details below.



a)



b)

c)

Fig. 5.17 Tabs of Results group

Table 5.5 – Results group parameters

Tabs	Parameters	Comments
Reinforcement (for design calculation only)	Diameter, mm	According to GOST 5781-82
	Number	Natural number, >= 2
	Reinforcement ratio	Reinforcement ratio by area
	Area calculation	Design of longitudinal reinforcement area
	Design of transverse reinforcement*	Design of longitudinal reinforcement intensity either by diameter or by step
Use factor	Use factor of reinforcement	Must be in range from 0 to 1
	All force factors	Invokes dialog box with force factors
	Force factors for selected coef.	Invokes dialog box with force factors for selected use factor
	>0.01 / >0.01	If button is pressed (>0.01) there are only use factors with values more than 0.01.
Crack growth resistance (if option is checked in General tab)	Width of short-term crack opening, mm	Real width of crack opening
	Width of long-term crack opening, mm	

Note:
** If transverse reinforcing is not required, given results are highlighted in grey color.*

The area calculator (fig. 5.15) is intended for selection of a diameter and number of longitudinal reinforcement by design area. Moreover, it is possible to use different diameters, for example, for corner and central rebars. The diameters selection is performed through an appropriate number of rebars. The total area of reinforcement provided in the bottom part of the calculator window.

Example: the System selected the longitudinal reinforcement d = 40 mm, n = 4 pcs. However, the user wants to use, for example, the rebar d = 20 mm. For this, first, enter in the appropriate edits n = 4 pieces, d = 40 mm and determine the required area of rebar 5026.4 mm2. Next on the calculator table find the area of a rebar d = 20 mm. It is 314.2 mm2. Finally, we determine the number of rebars

5026,4 / 314,2 = 16 pcs. To get the required area you can also changing the number of rebars of selected diameter. This approach is most effective if you use different diameters of rebar.

To get the calculated intensity of transverse reinforcement, you can specify diameter or pitch. The required number of rods of this design element will be calculated automatically in the right part of the dialog box.

Example: By results of design calculation required intensity of transverse reinforcing must be 2,31 mm2/mm. For reinforcing of column that length is 10050 mm using diameter 10 mm 402 pcs. of reinforcement with step 50 mm is required. Thus real reinforcement intensity has made 3,14 mm2/mm.

Fig. 5.18 Transverse reinforcement design

Reinforced masonry design elements

There are two types of reinforced masonry design elements: column and shell. Each element has the local coordinate system.

Data

Designations and color of reinforcing are presented in *Legend* tab.

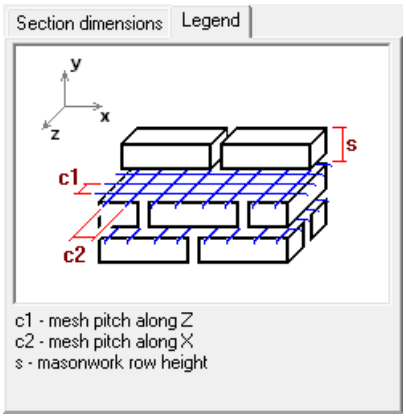


Fig. 5.19 Legend tab

There are tabs in the dialog for setting initial data: *General*, *Brickwork*, *Loads and Reinforcement* (for checking calculation). They are described in details below.

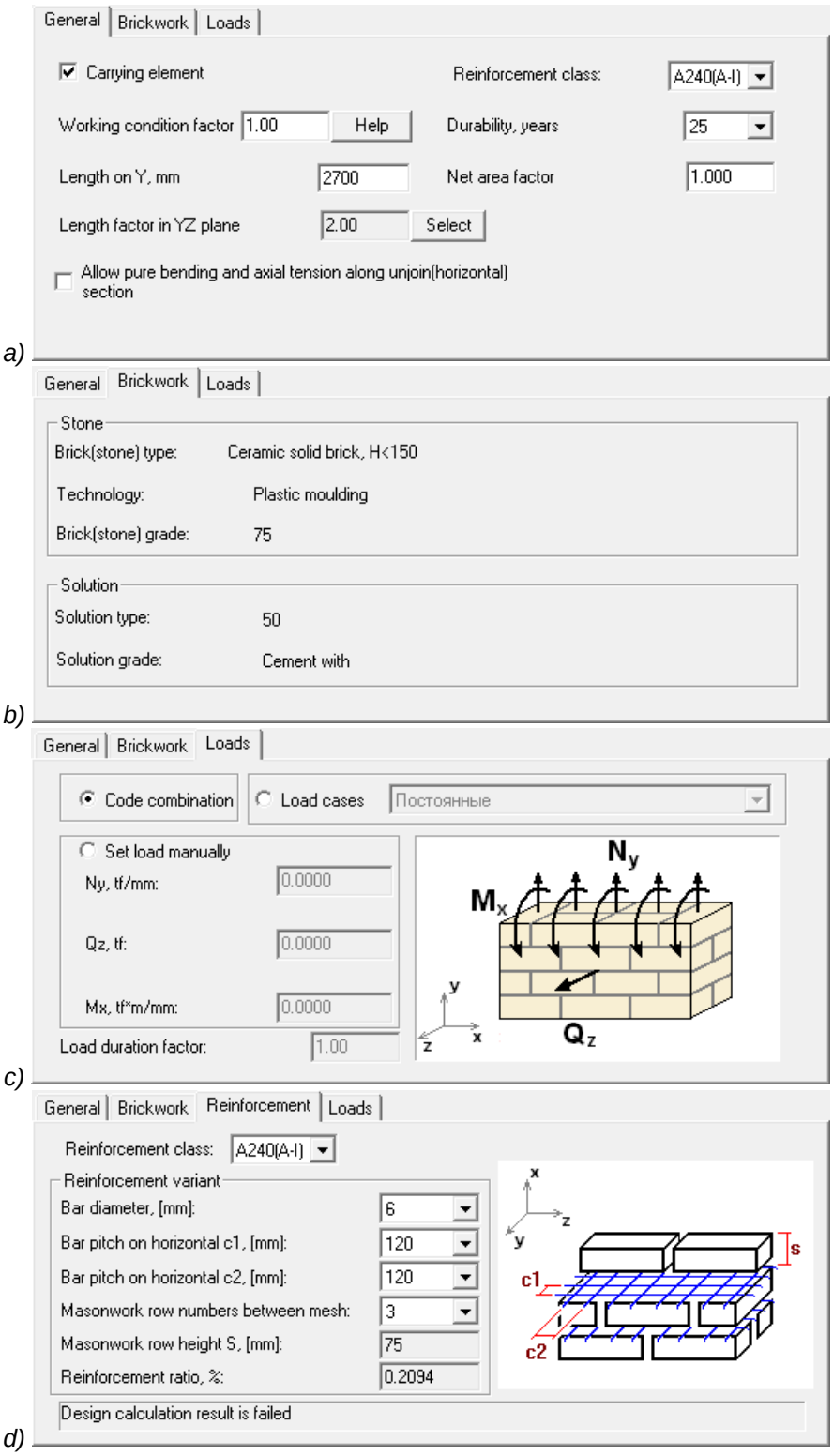


Fig. 5.20 Data group tabs

Table 5.6 – Data group parameters

Tabs	Parameters	Description
General	Carrying element	I. 4.9 SNiP II-22-81*
	Section type (for rod design element only)	Determines automatically
	Length factor in XY	I. 4.3 SNiP II-22-81*
	Length factor in XZ	I. 4.3 SNiP II-22-81*

	Working condition factor	I. 3.11* SNiP II-22-81*
	Allow pure bending and axial tension along unjoin (horizontal) section	Checking of this option allows to calculate elements at corresponding load types. It contradicts notes to I. 4.18 and I. 4.19 SNiP II-22-81 *
	Reinforcement class (for design calculation only)	I. 2.6 SNiP II-22-81*; T. 5.7-5.8 SP 52-101-2003
	Durability, years	I. 5.3 SNiP II-22-81*
	Net area factor	I. 4.19 SNiP II-22-81*
Brickwork*	Brick (Stone) type Technology Brick (Stone) grade Solution type Solution grade	I. 3. SNiP II-22-81*
Reinforcement (for checking calculation only)	Reinforcement class	I. 2.6 SNiP II-22-81*; T. 5.7-5.8 SP 52-101-2003
	Bar diameter, mm	I. 6.77 SNiP II-22-81*
	Bar pitch on horizontal c1, mm Bar pitch on horizontal c2, mm	Distances between bars of horizontal reinforcing mesh
	Masonwork row numbers between mesh	I. 6.76 SNiP II-22-81*
	Masonwork row height S, mm	Depends on element material
	Reinforcement ratio, %	I. 4.30 SNiP II-22-81*
	Design calculation status	If design calculation results are successful required reinforcement ratio is displayed.
Loads	• Code combination • Load case • Set loads manually	Load variants
	Load duration factor	I. 4.30 SNiP II-22-81*

Note:
** Depends on element material.*

Results

There are *Use factor* tab for viewing calculation results.

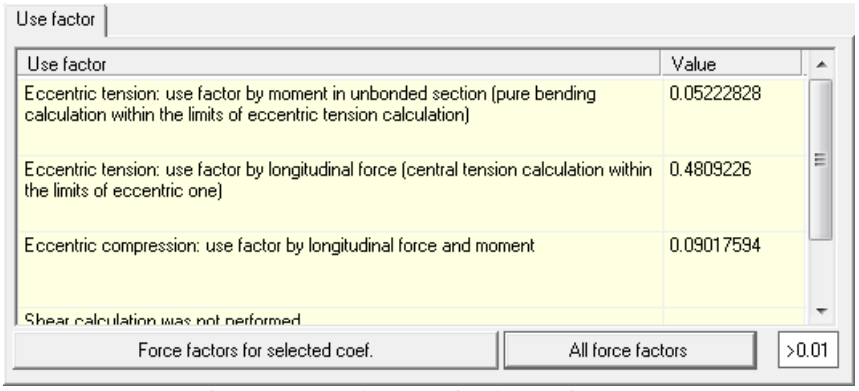


Fig. 5.21 Use factor tab of Results group

Table 5.7 – Results group parameters

Tabs	Parameters	Comments
Use factor	Required reinforcement ratio (for design calculation only)	If reinforcement is required
	Use factor	Must be in range from 0 to 1
	All force factors	Invokes dialog box with force factors
	Force factors for selected coef.	Invokes dialog box with force factors for selected use factor

	>0.01 / >0.01	If button is pressed (>0.01) there are only use factors with values more than 0.01

Soil bases and foundations calculation

Foundation calculation begins with a previously selection of the constructive decision and parameters, such as the base dimensions and depth of foundation.

Check of the accepted dimensions and foundation reinforcing is performed by soil strength condition. Deformation calculation of the soil bases is made by condition of structure and soil base combined action.

Calculation of soil base deformation under average pressure in foundation base, not exceeding soil base design resistance (i. 5.5.8 SP 50-101-2004) should be carried out using scheme in the form of linearly deformable half-space (i. 5.5.31) with conditional restriction on compressed soil depth (i. 5.5.41 SP 50-101-2004).

For modeling of the elastic soil base definition of proportionality constants named coefficients of soil reaction is required.

On the basis of engineering-geological researches *APM Structure3D* allows to set soil structure and define soil design resistance and coefficients of soil reaction.

Calculation of internal forces in "soil base-foundation-structure" system is supposed to be performed on the basis characterized by variable stiffness in the plan (coefficients of soil reaction). Coefficients of soil reaction depend on structure and soil physical properties and also soil base loads. These coefficients can be previously defined or by iteration procedure. Procedure includes following steps:

1) structure calculation on the rigid supports and definition of soil reaction coefficients initial distribution by soil settlement depth;

2) calculation of coupled displacements of structure, foundation and soil base with the accepted soil reaction distribution under action of the set loads;

3) definition of foundation settlements with use of the accepted soil base model and also the next iteration and recalculation of soil reaction coefficients;

4) repetition of steps 2) and 3) before convergence by control parameters (for example, coefficients of soil reaction).

To work with the elastic soil bases use *Elastic Foundations* toolbar commands.

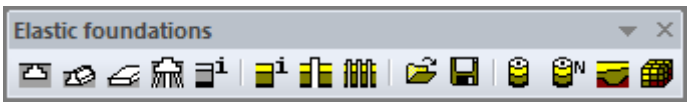


Fig. 5.22 Elastic Foundations toolbar

	Elastic soil base for post foundation (activates after selection of rod-column and its base node)
	Elastic soil base for strip foundation (activates after selection of rod-girder)
	Elastic soil base for mat foundation (activates after selection of plate object)
	Elastic soil base for single pile foundation (activates after selection of rod-column and its base node)
	Soils information (accessible if soils are defined)
	Engineering-geological elements
	Holes
	Hole list
	Load geological info
	Save geological info
	Show/Hide holes
	Show/Hide hole names
	Show/Hide stratification map

General principles of work with Foundation dialog box

Generally the structure can be based on foundations of various types. Uniform dialog window with the list of foundations is used for work with all bases. Tabs of this dialog depend on the selected foundation type. When foundation is selected in the dialog list corresponding element is highlighted by

red color in structure editor. And contrary, selected element on model will be highlighted in foundation list of the dialog box.

Apply button is intended to accept all changes made in dialog tabs. In case of the incorrect set of parameters the message is displayed.

Calculation of soil base for post foundation

Post foundation is set for column as a rule. Therefore for calculation of elastic base for post foundation it is necessary to create steel, reinforced concrete or reinforced masonry design element – column.

When column and nod with the set support is selected, command **Elastic base for post foundation** becomes accessible and after its activation, there will be *Foundations* dialog window. Further components of this dialog will be considered in detail.

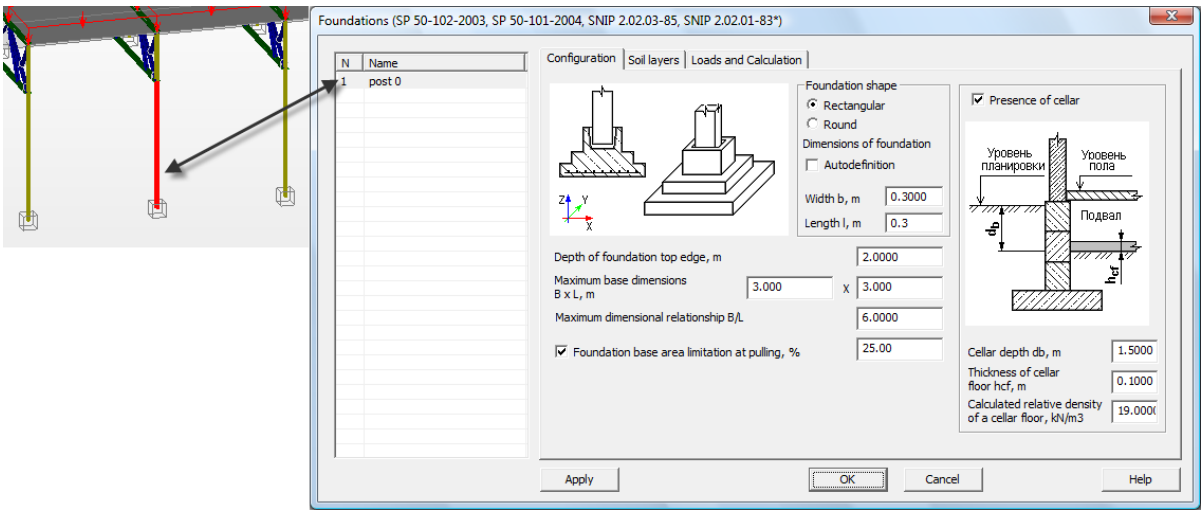


Fig. 5.23 Foundations dialog box

Configuration tab

Foundation shape corresponds to the column form by default.

Foundation dimensions – dimensions of the foundation offset coincide with overall dimensions of column section by default. Dimensions of the foundation top edge can be increased in comparison with column dimensions. It is possible to use round foundation for rectangular section column also.

Depth of foundation top edge – depth relative to zero level. Foundation depth cannot be lower than rocky soil.

Maximum dimensional relationship B/L allows to set additional design limitations to foundation base dimensions.

Foundation base area limitation at pulling, % – restriction by triangular diagram type under foundation base.

Working condition factor button (– not defined, – defined) – invokes dialog box for factor selection according to t. 5.2 SP 50-101-2004.

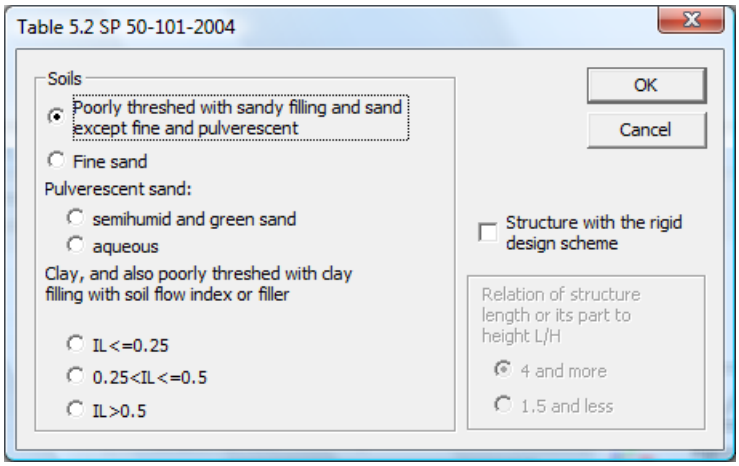


Fig. 5.24 Dialog box – Table 5.2 SP 50-101-2004.

Presence of cellar checkbox allows to set cellar parameters: depth, floor thickness, floor relative density, which are used for soil design resistance calculation.

Soil Layers tab

Only one soil can correspond to one soil base.

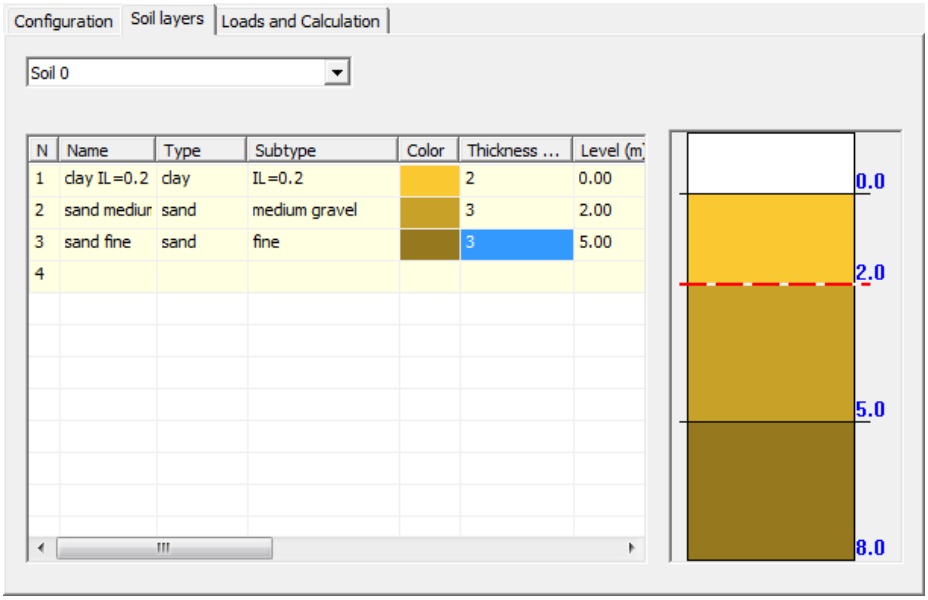


Fig. 5.25 Soil Layers tab

The soil list is presented in the left part of dialog. You can set soil structure according to engineering-geological estimation. To set soil layer it is necessary to select its type from the drop-down list. It is possible to use predetermined layer types: sand, clay with known physical characteristics, and also type in soil parameters manually: thickness, density, etc. All edit fields can be changed by mouse double click after selection of predetermined variant.

First, it is necessary to choose soil *type* (clay or sand). *Subtype* drop-down list content depends on soil type. It is possible to use predetermined layer types: sand, clay with known physical characteristics, and also type in soil parameters manually: thickness, density, etc. Detailed description of soil modeling is listed in this chapter below.

Loads and Calculation tab

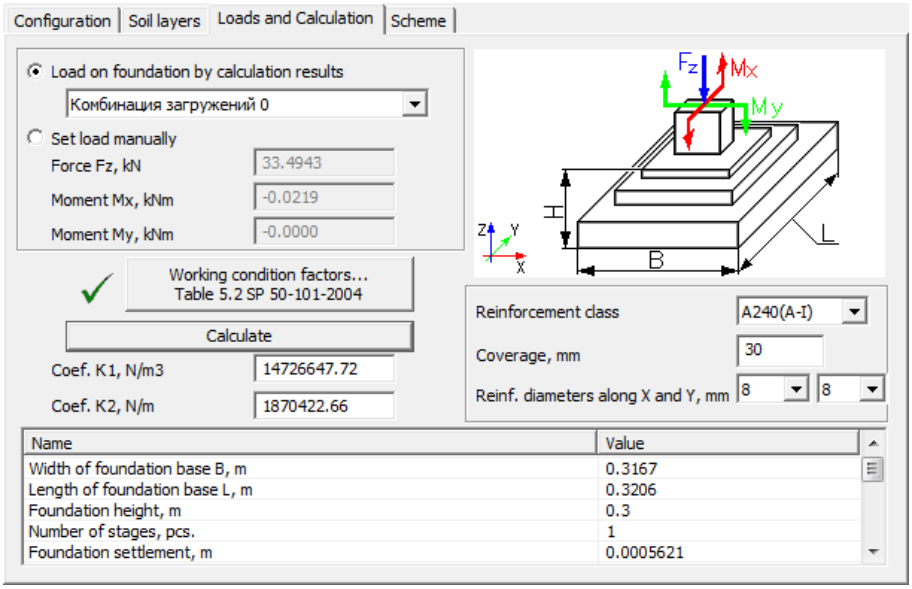


Fig. 5.26 Loads and Calculation tab

Values of loads acting on soil base can be set manually according to the scheme or can takes from results of static calculation.

Calculate button starts calculation. It is necessary to set reinforcement and concrete cover. After calculation rigid support is replaced by elastic support with stiffness which is equal to product of soil reaction coefficient and foundation base area.

Scheme tab

The scheme tab with the base geometrical dimensions and its location relative to soil layers becomes accessible after calculation.

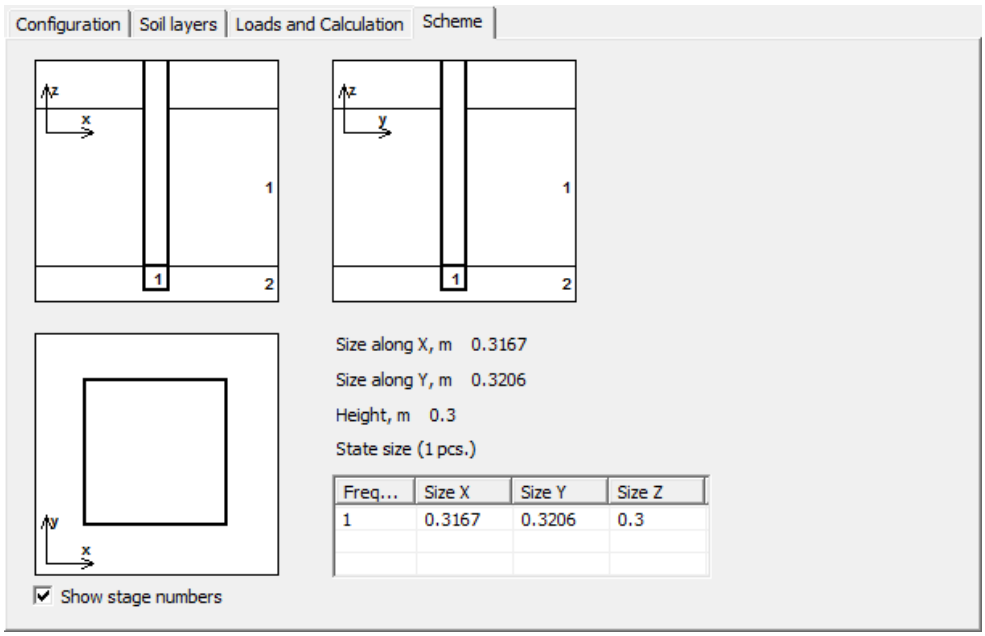


Fig. 5.27 Scheme tab

Calculation of soil base for strip foundation

The strip base represents a beam under wall or nearby columns. For elastic base calculation it is necessary to create reinforced concrete girder design element and then set supports in design element nodes.

The design elements of one section and located on one ground can enter into one foundation.

After selection of a girder or group of girders the command **Elastic base for strip foundation** becomes accessible which invokes *Foundation* dialog box.

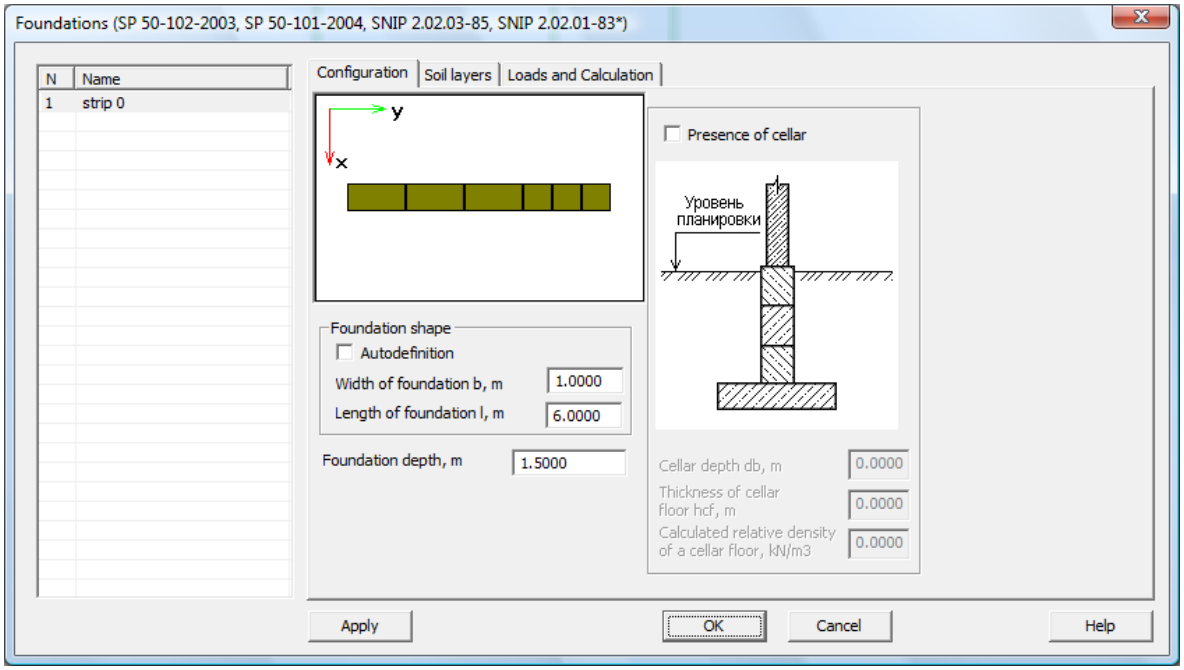


Fig. 5.28 Foundation dialog box (Configuration tab)

Configuration tab

The foundation image corresponds to the up view. Color of the base corresponds to color of section that allows to check that one basis has been created from elements of one section.

Foundation shape corresponds to the design elements dimensions by default.

Foundation depth – depth relative to zero level. Foundation depth cannot be lower than rocky soil.

Presence of cellar checkbox allows to set cellar parameters: depth, floor thickness, floor relative density, which are used for soil design resistance calculation.

Soil Layers tab

Only one soil can correspond to one soil base.

Generally the building site can be non-uniform. In this case it is necessary to set soil for each base.

You can set soil structure according to engineering-geological estimation. To set soil layer it is necessary to select its type from the drop-down list. It is possible to use predetermined layer types: sand, clay with known physical characteristics, and also type in soil parameters manually: thickness, density, etc. All edit fields can be changed by mouse double click after selection of predetermined variant.

First, it is necessary to choose soil *type* (clay or sand). *Subtype* drop-down list content depends on soil type. It is possible to use predetermined layer types: sand, clay with known physical characteristics, and also type in soil parameters manually: thickness, density, etc. Detailed description of soil modeling is listed in this chapter below.

Loads and Calculation tab

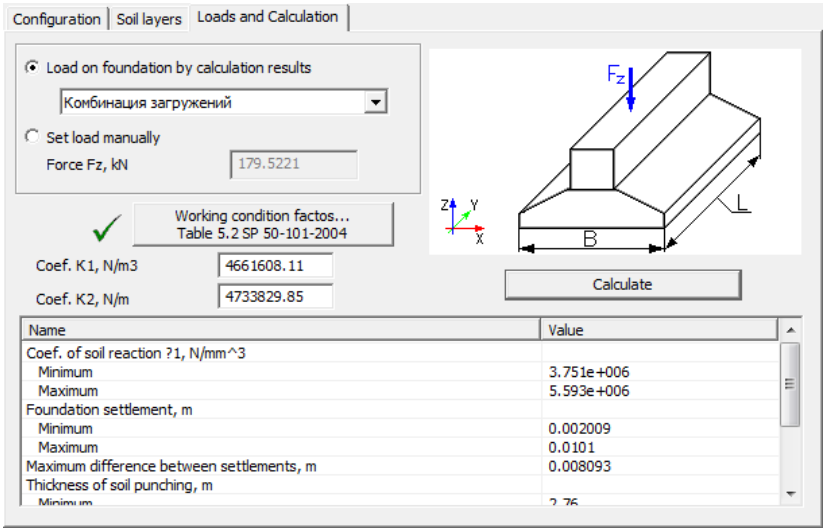


Fig. 5.29 Loads and Calculation tab

Values of loads acting on soil base can be set manually according to the scheme or can takes from results of static calculation.

Working condition factor button (– not defined, – defined) – invokes dialog box for factor selection according to t. 5.2 SP 50-101-2004.

Calculate button starts calculation. After calculation rigid support is replaced by elastic support with stiffness which is equal to product of soil reaction coefficient and foundation base area.

Further it is necessary to perform calculation of system "soil base-foundation-structure" on elastic support. After that reinforcing of foundation elements can be designed taking into account elastic base.

Calculation of soil base for mat foundation

The mat foundation represents a plate. For elastic base calculation it is necessary to create reinforced concrete shell design element and then set supports in design element nodes.

The design elements located on one ground can enter into one foundation.

After selection of a shell or group of shells the command **Elastic base for mat foundation** becomes accessible which invokes *Foundation* dialog box.

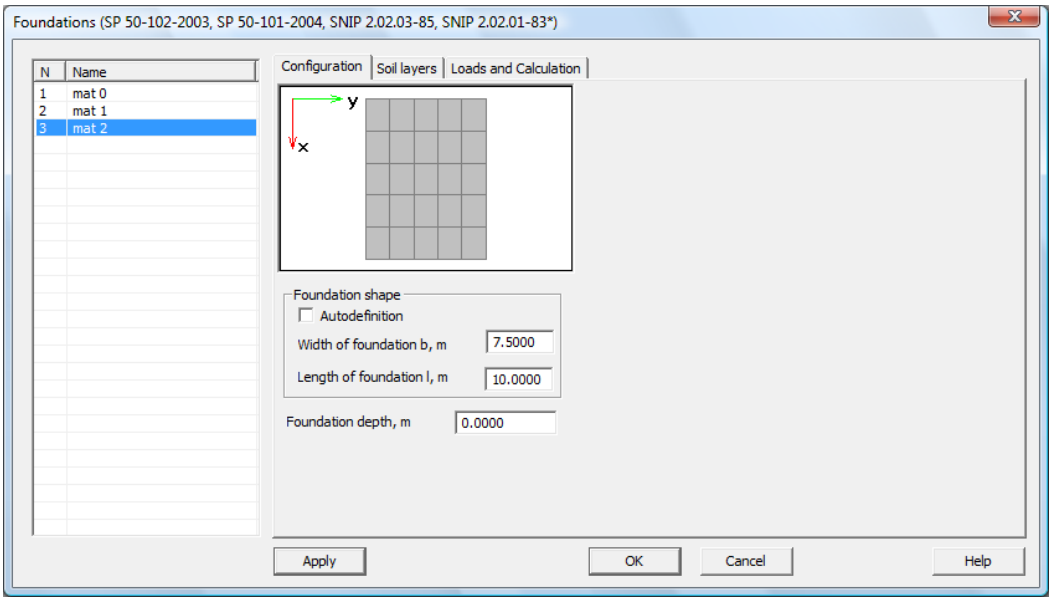


Fig. 5.30 Foundation dialog box (Configuration tab)

Configuration tab

The foundation image corresponds to the up view.

Foundation shape corresponds to the design elements dimensions by default.

Foundation depth – depth relative to zero level. Foundation depth cannot be lower than rocky soil.

Soil Layers tab

Only one soil can correspond to one soil base.

Generally the building site can be non-uniform. In this case it is necessary to set soil for each base.

You can set soil structure according to engineering-geological estimation. To set soil layer it is necessary to select its type from the drop-down list. It is possible to use predetermined layer types: sand, clay with known physical characteristics, and also type in soil parameters manually: thickness, density, etc. All edit fields can be changed by mouse double click after selection of predetermined variant.

First, it is necessary to choose soil *type* (clay or sand). *Subtype* drop-down list content depends on soil type. It is possible to use predetermined layer types: sand, clay with known physical characteristics, and also type in soil parameters manually: thickness, density, etc. Detailed description of soil modeling is listed in this chapter below.

Loads and Calculation tab

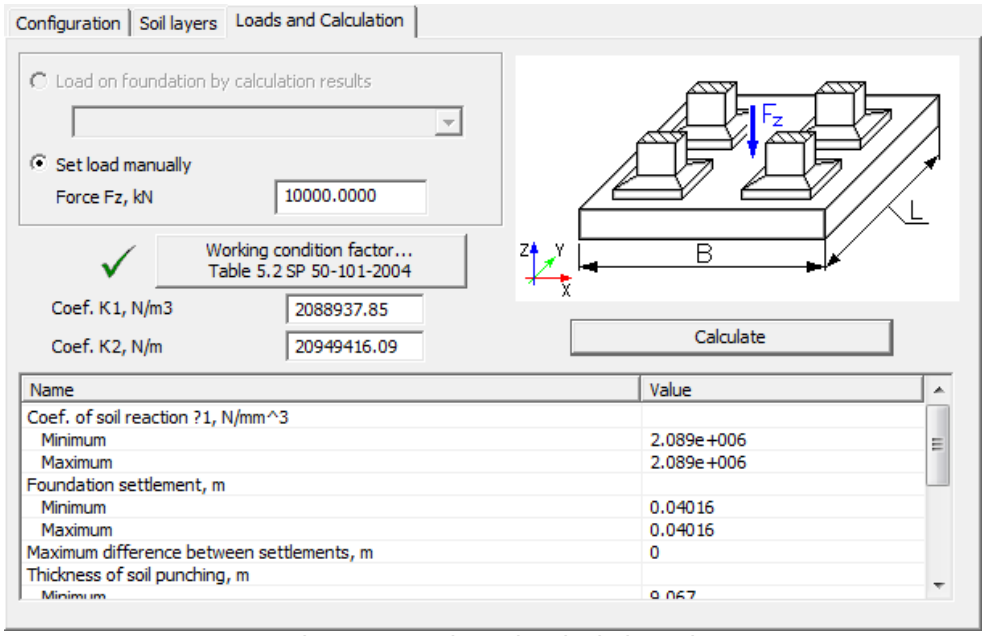


Fig. 5.31 Loads and Calculation tab

Values of loads acting on soil base can be set manually according to the scheme or can takes from results of static calculation.

Working condition factor button (– not defined, – defined) – invokes dialog box for factor selection according to t. 5.2 SP 50-101-2004.

Calculate button starts calculation. After calculation rigid support is replaced by elastic support with stiffness which is equal to product of soil reaction coefficient and foundation base area.

Further it is necessary to perform calculation of system "soil base-foundation-structure" on elastic support. After that reinforcing of foundation elements can be designed taking into account elastic base.

Calculation of soil base for pile foundation

Rod selection – steel, reinforced concrete or masonry design element – column and node with the support makes accessible command **Elastic base for pile foundation**. After command activation there will be Foundation dialog window. To set pile foundation it is necessary to select lowest column parts with nodes and press **Elastic base for pile foundation** button. Pile

foundation list located in the left part of appeared dialog. Select required item in the list to set parameters. Let's consider all tabs of this dialog in detail.

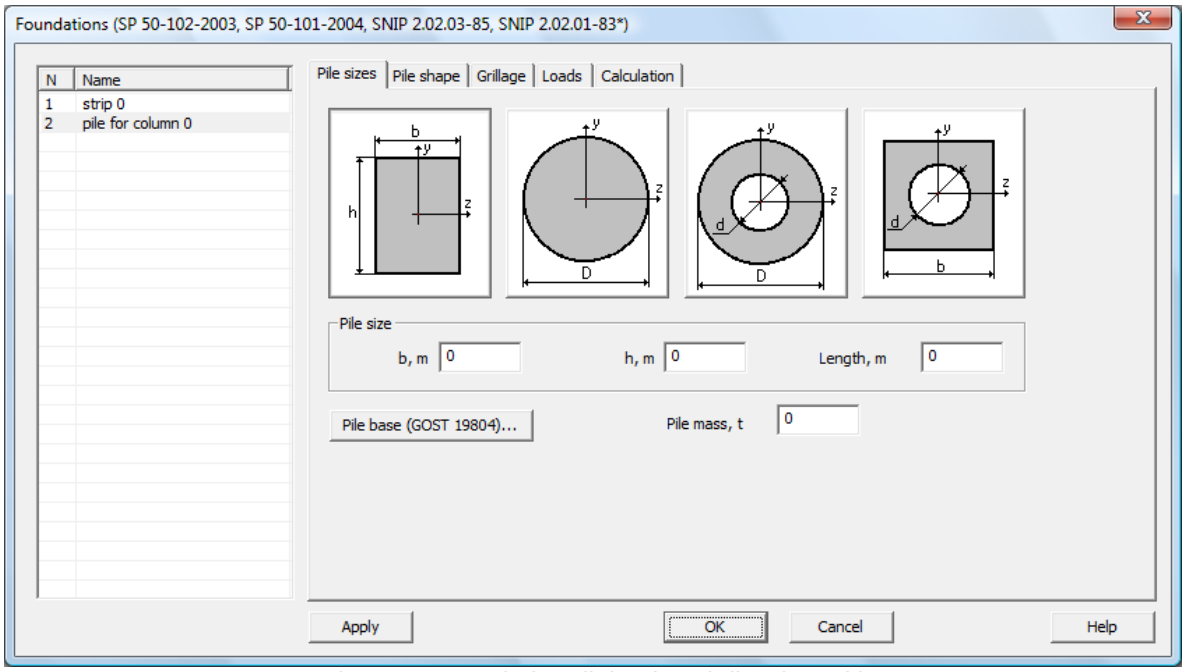
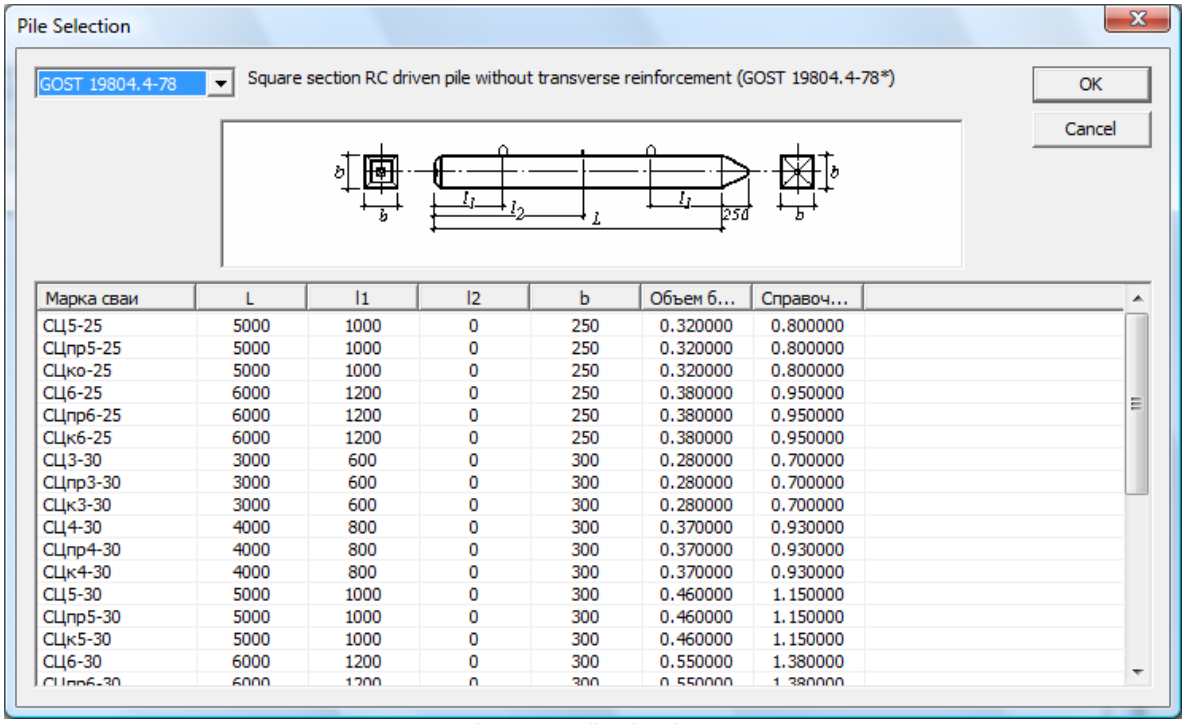


Fig. 5.32 Foundation dialog box (Pile Size tab)

Select standard pile from databases using Pile base button and tab edit fields will be filled automatically. If non-standard piles are used type in required parameters manually.



Марка сваи	L	l1	l2	b	Объем б...	Справоч...
СЦ5-25	5000	1000	0	250	0.320000	0.800000
СЦпр5-25	5000	1000	0	250	0.320000	0.800000
СЦко-25	5000	1000	0	250	0.320000	0.800000
СЦ6-25	6000	1200	0	250	0.380000	0.950000
СЦпр6-25	6000	1200	0	250	0.380000	0.950000
СЦк6-25	6000	1200	0	250	0.380000	0.950000
СЦ3-30	3000	600	0	300	0.280000	0.700000
СЦпр3-30	3000	600	0	300	0.280000	0.700000
СЦк3-30	3000	600	0	300	0.280000	0.700000
СЦ4-30	4000	800	0	300	0.370000	0.930000
СЦпр4-30	4000	800	0	300	0.370000	0.930000
СЦк4-30	4000	800	0	300	0.370000	0.930000
СЦ5-30	5000	1000	0	300	0.460000	1.150000
СЦпр5-30	5000	1000	0	300	0.460000	1.150000
СЦк5-30	5000	1000	0	300	0.460000	1.150000
СЦ6-30	6000	1200	0	300	0.550000	1.380000
СЦпр6-30	6000	1200	0	300	0.550000	1.380000

Fig. 5.33 Pile database

Pile Configuration tab is intended for selection pile type and set pile parameters which depend on its type according to table 5.8.

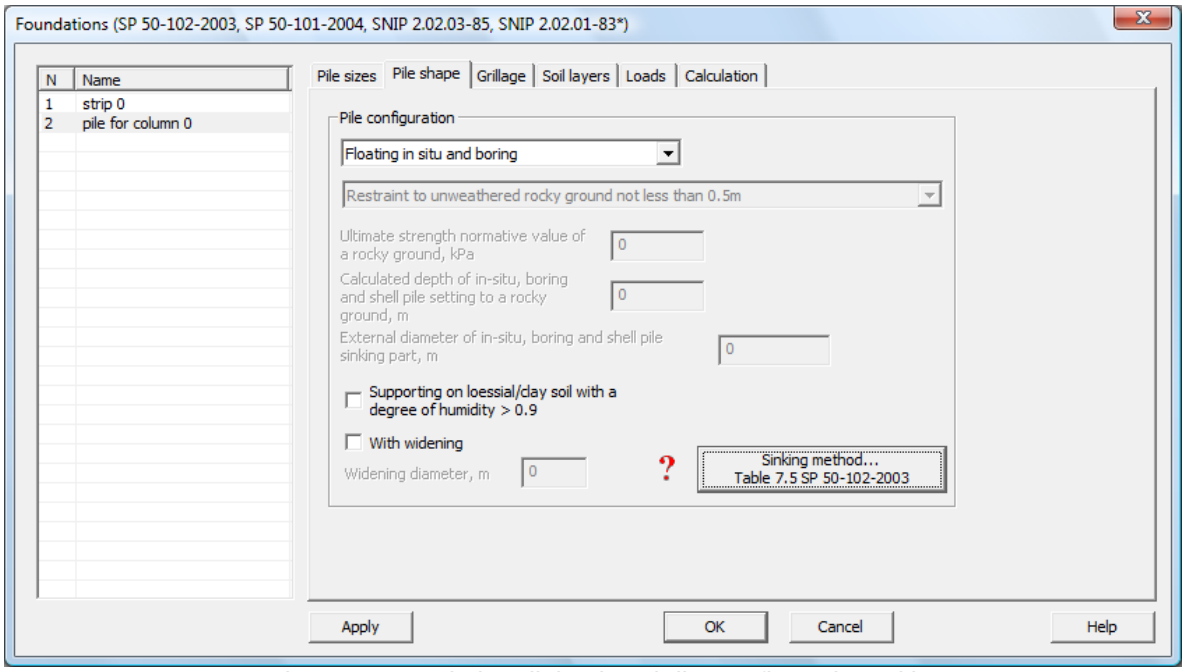


Fig. 5.34 Foundation dialog box (Pile Configuration tab)

Table 5.8 – Pile parameters

Pile type	Available parameters
Driven	–
Shell	• Support type • Ultimate strength normative value of rocky ground, kPa • External diameter of in situ, boring and shell pile sinking part, m
In situ and boring	• Ultimate strength normative value of rocky ground, kPa • Calculated depth of in situ, boring and shell pile setting to a rocky ground, m • External diameter of in situ, boring and shell pile sinking part, m
Floating driven Floating shell	• Sinking method (table 7.3 SP 50-102-2003).
Floating shell with concrete filling Floating in situ and boring	• Supporting on loessial/clay soil with a degree of humidity >0.9 • With widening • Sinking method (table 7.5 SP 50-102-2003)
Floating screw	• Soil parameters (table 7.8 SP 50-102-2003) • Diameter of blade
Floating bored screwed	• Sinking method (table 7.2.11 SP 50-102-2003)
Floating jacking	• Sinking method (table 7.3 SP 50-102-2003)

Sinking method button (? – not defined, ✓ – defined) –- invokes dialog box for pile sinking method or soil type selection according to tables 7.3, 7.5 or 7.8 SP 50-101-2004.

Grillage tab is intended for set grillage parameters and account of cellar.

Presence of cellar checkbox allows to set cellar parameters: depth, floor thickness, floor relative density, which are used for soil design resistance calculation.

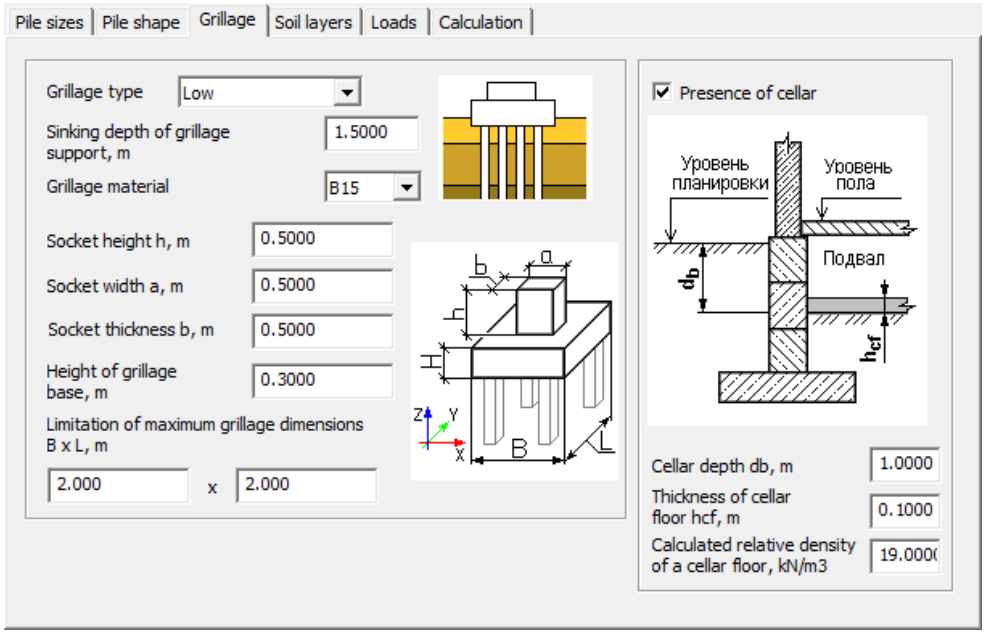


Fig. 5.35 Grillage tab

Soil layers tab is accessible for floating piles only. First, it is necessary to choose soil *type* (clay or sand). *Subtype* drop-down list content depends on soil type. It is possible to use predetermined layer types: sand, clay with known physical characteristics, and also type in soil parameters manually: thickness, density, etc. Detailed description of soil modeling is listed in this chapter below.

Loads tab. Values of loads acting on soil base can be set manually according to the scheme or can takes from results of static calculation. It is necessary to perform static calculation of rigid supported structure previously.

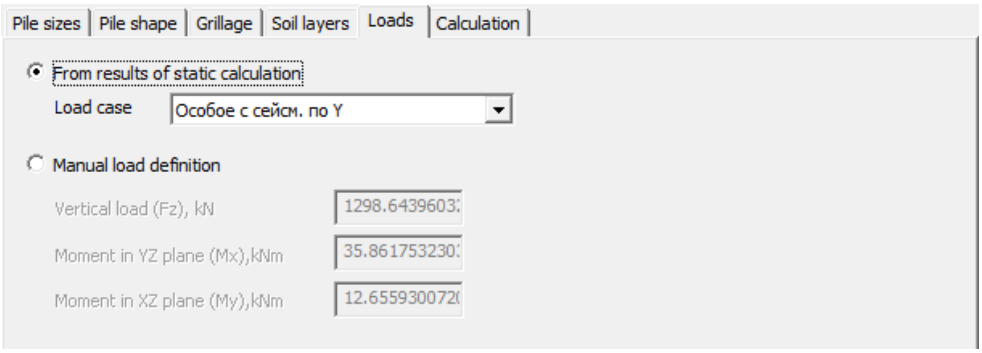


Fig. 5.36 Loads tab

In *Calculation* tab select calculation method of soil strength characteristics and safety factor.

Pile capacity by soil and *Pile capacity on pulling by soil* values is displayed after pressing **Calculate pile capacity** button.

It is possible to set pile capacity manually for example, if pile strength by material will be less than pile strength by soil.

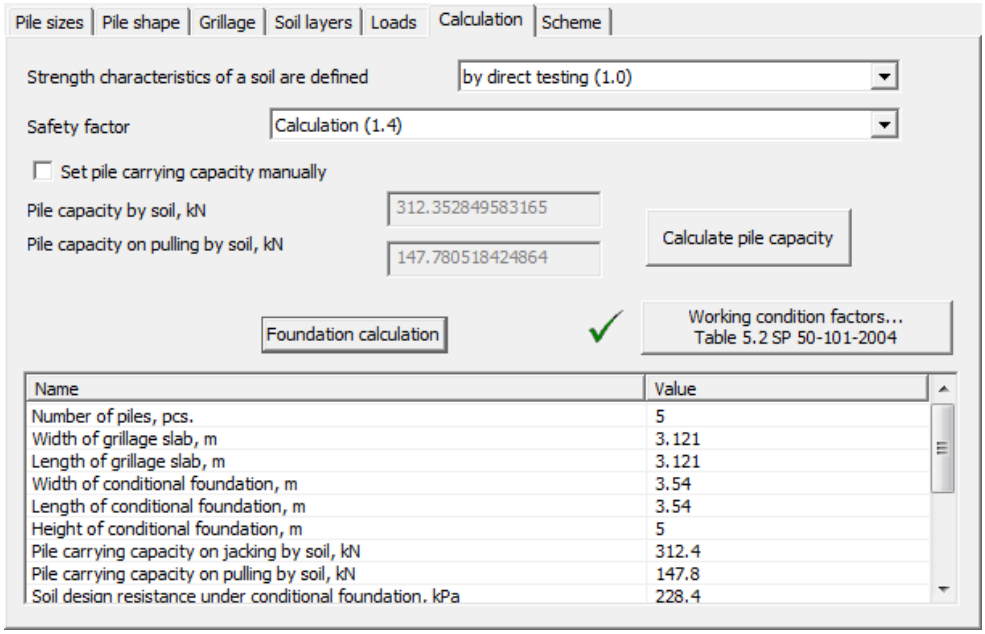


Fig. 5.37 Calculation tab

Working condition factor button (– not defined, – defined) – invokes dialog box for factor selection according to t. 5.2 SP 50-101-2004.

Foundation calculation button starts calculation of soil punching thickness taking into account loads on the basis, coefficients of soil reaction, heeling, characteristics of pile carrying capacity by soil, required number of piles, and also for floating piles: geometrical dimensions of a grillage slab, dimensions of conditional foundation, soil design resistance under conditional foundation. If the required number of piles is more than 20 – the message is displayed.

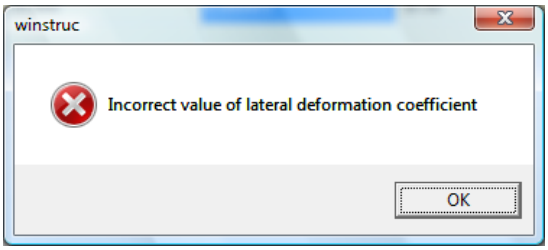


Fig. 5.38 Message

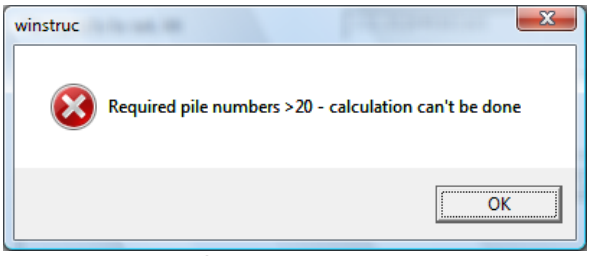


Fig. 5.39 Message

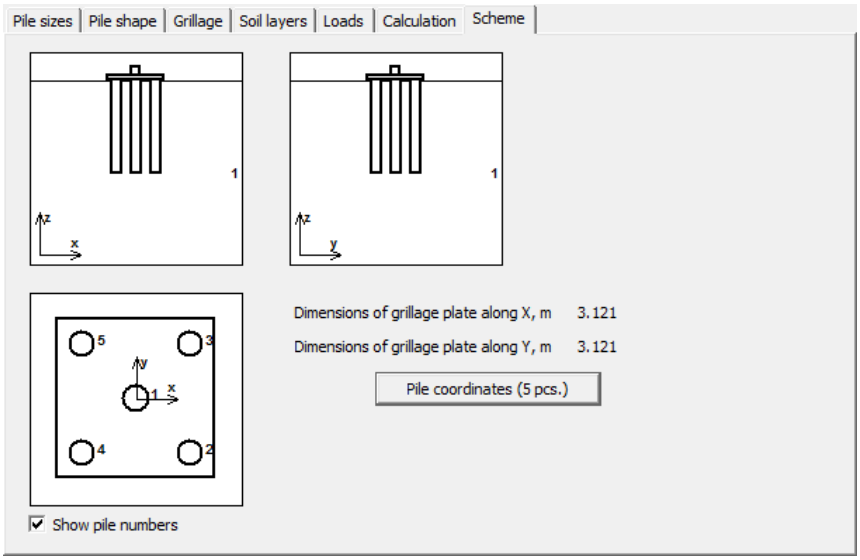


Fig. 5.40 Scheme tab

After calculation *Scheme* tab becomes accessible where piles location is shown. For all pile types except bearing piles the support will be replaced with an elastic support of corresponding stiffness along Z axis. To perform calculation of «soil-foundation-structure» system it is necessary to recalculate it taking into account elastic supports of the piles.

Soil modeling. General definitions

Engineering-geological element (EGE) – the engineering-geological body presented by one rock with statistically homogeneous properties: density, angle of internal friction, relative friction, coef. of lateral strain, modulus of elasticity. Own color corresponds to each EGE for displaying stratification map and ground.

Hole is characterized by its location and EGE layers. The thickness is set for each layer. Location of a hole in a plane (coordinates X, Y) can be set using mouse or keyboard. The hole location level (coordinate Z) is set by absolute mark. Besides building zero level can be set in addition relative to sea level. Soil stratification approximation is carried out according to holes location.

Soil is characterized by layers with thickness. Properties of each layer: density, angle of internal friction, relative friction, coef. of lateral strain, modulus of elasticity. Own color corresponds to each soil layer.

For one soil base of foundation it is possible to set only one soil. Soil characteristics can be set in a dialog window or take from soil stratification map.

Soil characteristics

There are 2 ways for setting of soil characteristics in *APM Structure 3D*:

1. Set the list of predetermined soils and then select sequentially one of it for soil base. Such approach is preferable if the building site is homogeneous (changes of soil characteristics or the number of soil bases are insignificant).

2. Set EGE with properties, make holes according to given engineering-geological researches, create soil stratification map. Such approach allows to receive soil characteristics in any point for a significant amount of the soil bases and non-uniform geological conditions automatically.

Let's consider each way in detail.

Work with soil list

Soil List command invokes *Soil List* dialog box for creation, editing and deleting of soils. Dialog contains soil list which presented in the left part of dialog and image of the selected soil on the right.

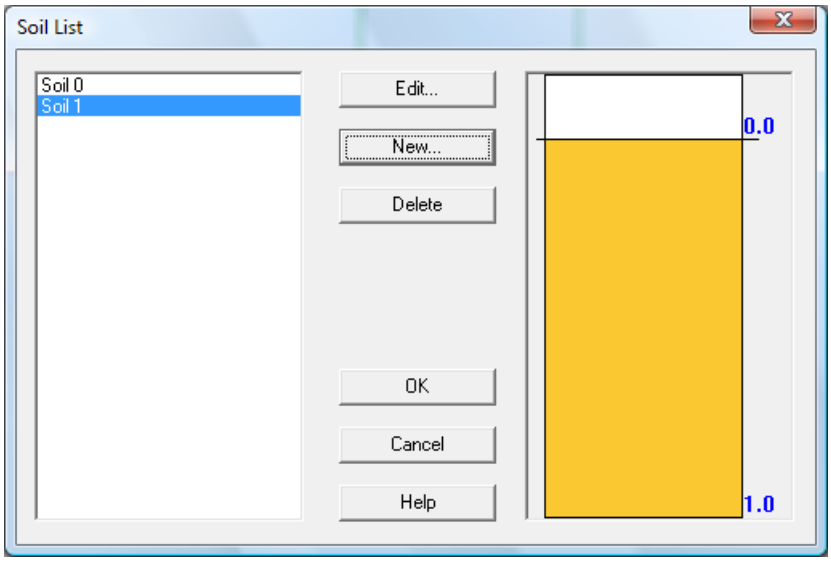


Fig. 5.41 Soil List dialog box

Edit button invokes *Soil Layers* dialog box where you can change soil structure.

New button invokes *Soil Name* dialog box in which you can type in soil name. After pressing **OK** button *Soil Layers* dialog box appears on the screen. After new soil is created it will be accessible in *Foundation* dialog box.

Delete button allows to delete soil. The soil cannot be deleted while it is used in calculation of any foundation.

Soils which were created by means of this command will be accessible to the subsequent selection in *Soil Layers* tab of *Foundation* dialog box.

Soil stratification map

Definition of soil characteristics for a non-uniform building site includes following stages:

1) Set engineering-geological elements (EGE) and their properties.
2) Set hole location and soil layers according to EGE.
3) Approximation by available holes for creation of soil stratification map.
4) Creation of separate soils for each foundation.

Let's consider soil stratification map creation in detail.

To set hole select **Engineering-geological elements** button on *Elastic foundations* toolbar. To set new EGE it is necessary to choose soil type (clay or sand). It is possible to use predetermined layer types: sand, clay with known physical characteristics, and also type in soil parameters manually: thickness, density, etc.

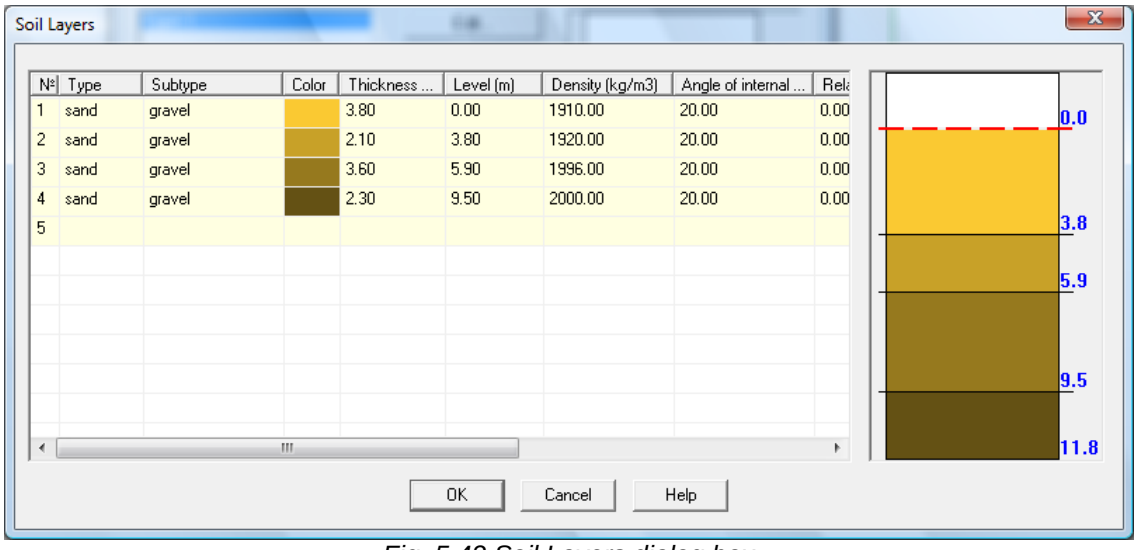


Fig. 5.42 Soil Layers dialog box

To set holes location use **Holes** command. The method of holes creation should be used if there are previously created nodes in holes location for exact cursor snap. After specifying hole location there will be *Engineering-geological data* dialog box. That dialog can be invoked by **Engineering-geological data** command also.

Area for soil stratification map in GCS - this adjustment allows to set soil stratification map area automatically according to holes location. Input of area boundary coordinates is provided also if automatic map does not cover all foundation. Last variant can be used, for example, for extended building in plane at holes location along line.

Holes List includes number, name and coordinates of holes which can be entered from the keyboard. For addition/deletion of holes the buttons located below are used.

The set of soil layers from earlier created EGE in the right part of a dialog can be assigned for each hole. Thus it is possible to set one of two parameters - *thickness* or *level*. Color of a layer corresponds to EGE color.

It is possible to display holes and their names on model using buttons: **Show/Hide Holes** and **Show/Hide Hole Names**.

All changes in dialog box are accessible to display in model after pressing **Apply** button.

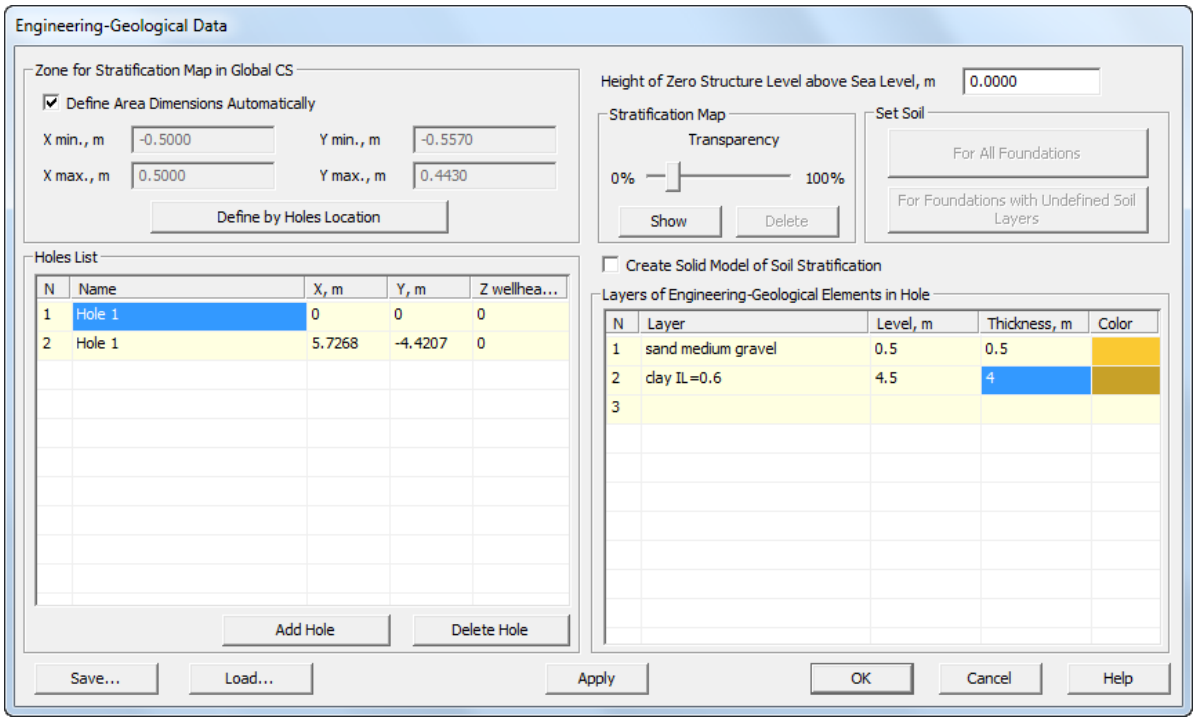


Fig. 5.43 Engineering-geological data

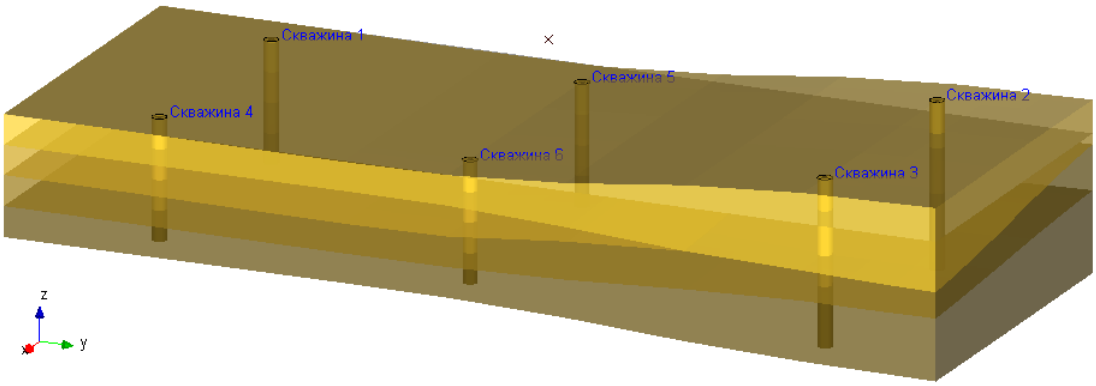


Fig. 5.44 Example of soil stratification map

Set soil group buttons allow to set a soil **For all foundations** or **For foundations with undefined soil** only. Last adjustment allows to combine both ways, for example when the building part is located on a homogeneous building site, and under other part of a building soils are non-homogeneous. After pressing one of the specified buttons soil stratification map is displayed on the basis of approximation (Displaying is possible only if the **Show/Hide Stratification Map** button is pressed).

Interpolation of soil properties is based on automatic creation of new soils under each base. This operation can be followed after foundation creation. Detailed description of soil base creation for foundations are presented above.

Soil stratification map adjustment allows to set a visualization transparency.

Engineering-geological data save together with *APM Structure3D* model, however it is possible to save EGE and holes in a separate file (*.soildata). **Save** and **Load** buttons are used for this purpose.

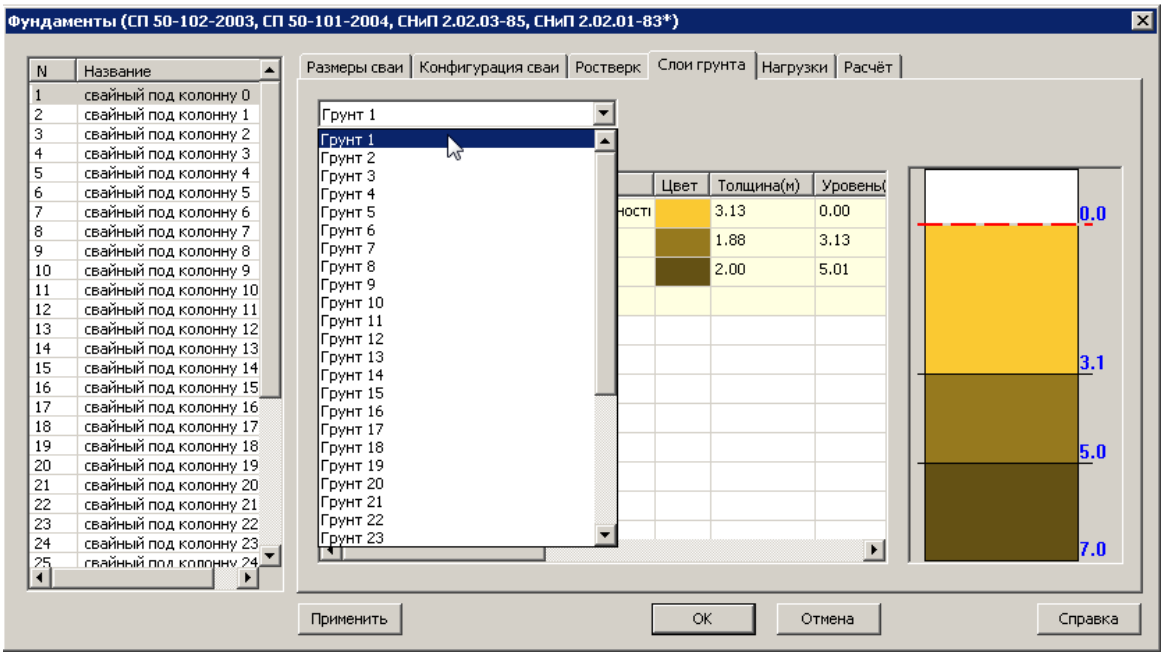


Рис. 5.45 Пример создания новых грунтов для свайного поля.

Chapter 6. Results

This chapter gives a brief description of results of all types of calculation carried out in APM Structure3D.

Static calculation results

Results of static calculation are:

- Linear and angular node displacements
- Loads at rod ends, plate and solid elements nodes
- Force diagrams for entire structure
- Force diagrams for entire structure
- Stresses acting in rods, plates and solid elements
- Stress distribution in arbitrary cross-section of any rod
- Force diagrams for entire structure
- Specified parameters for separate beam such as: bending moments, torsion moments, lateral and axial forces, bending and torsion angles, stresses and strains along beams length. All these parameters are represented in the form of graphs and plotted in local rod coordinate system. You can obtain both relational deformation (displacements comparative to the line connecting two deformed edges of rod) and deformation in global coordinate system. In case of structure consisting of only one rod comparative and total deformation are the same.
- Reactions (forces and moments) acting in supports
- Total construction mass

Node displacements and loads at rod ends and plate and solid nodes

Node displacements and rod end loads as well as plate and solid nodes are presented as a table.

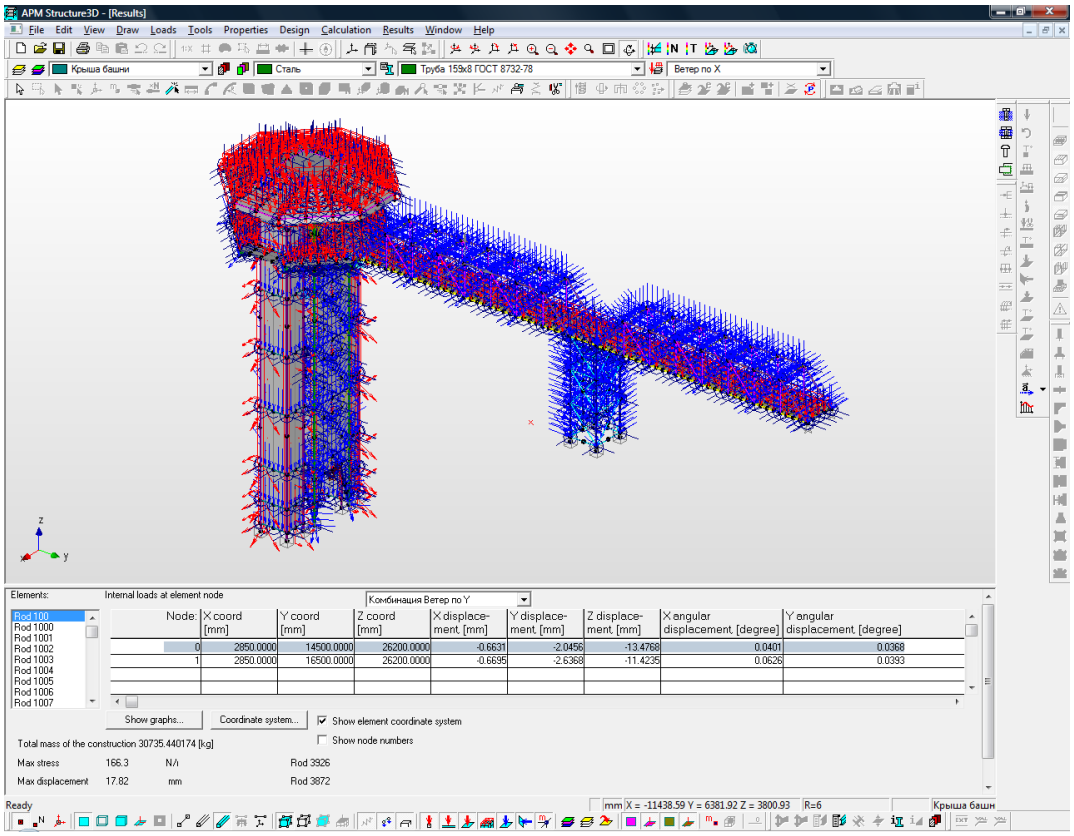


Fig. 6.1 Displacements and Loads at Nodes dialog box

Displacement values are presented in global coordinate system while loads at nodes are represented in element coordinate system (rod or plate). For solid elements, both displacements and loads are represented in global coordinate system because solid local coordinate system coincides with global one. To see these results select **Results / Loads** menu command that will call the window shown above.

In the upper part of the window, the current element is highlighted. To view the desired element you can select it either in the upper part of the window or in *Elements* list in its lower part. *Show Graphs* button allows user to view design parameters of a beam along its length. These parameters are shown above in the results list as item 4. This command invokes a dialog box shown below. The box contains buttons that allow you to look through corresponding graphs. Set *Show Full Deformation* checkmark to plot full deformations graph or remove this checkmark to view deformation relative to the line connecting two deformed edges (i.e. without taking edge displacement into account).

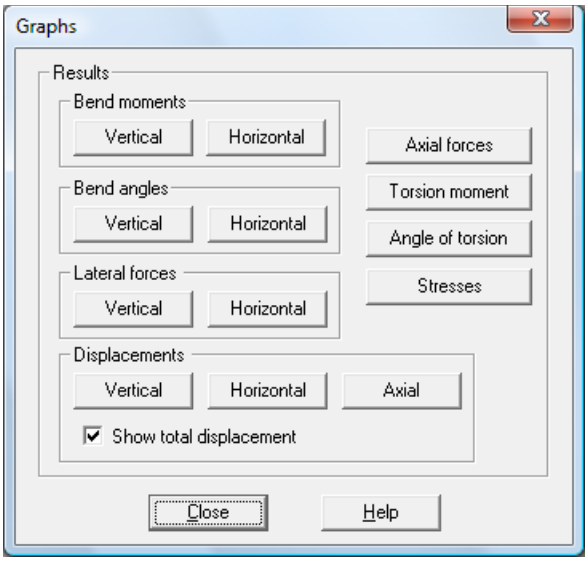


Fig. 6.2 Rod Coordinate System Dialog box

Force diagrams for entire structure

Force diagrams for entire structure are plotted with the help of **Results / Show Element Forces** command. Diagrams for elements belonging to ON layers are displayed in the window; diagrams for OFF layers are not plotted. For each separate rod, these diagrams can be seen in **Results / Loads** menu. An example of window representing diagrams on rods is shown below.

Diagrams in the element are displayed in a color map of the results. It is possible to place leaders on a result map using Leader command of the results toolbar. The following is an example of a window with diagram on bars.

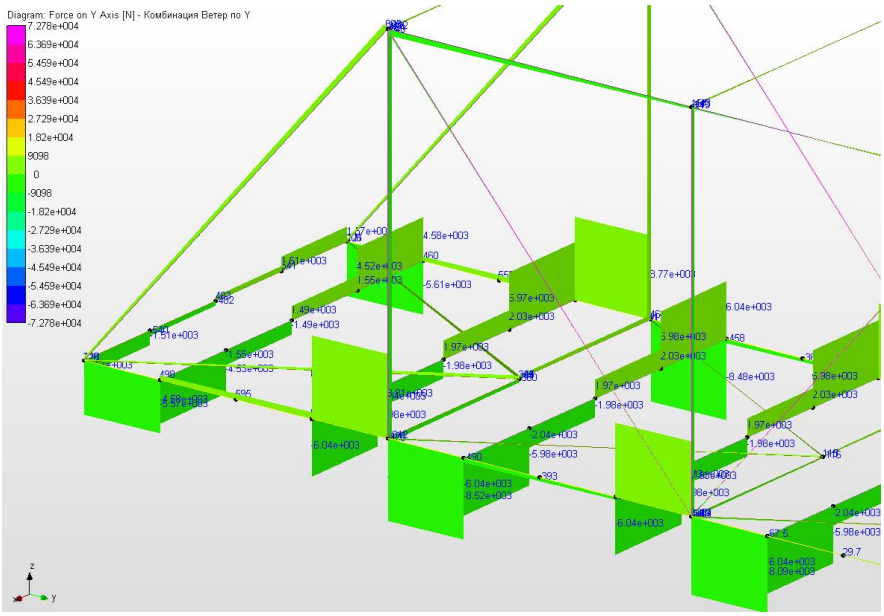


Fig. 6.3 Window representing forces acting on rods

Loads in rods, plates and solid elements

Loads in rods, plates and solid elements are displayed as stress map. Construction stress map is frame construction painted in colors according to stress values on the surface. Deformation map is a deformed construction view. Displacement values are shown scaled-up for intuitiveness. To see the map of stress and deformation select **Results / Stress Map** menu command.

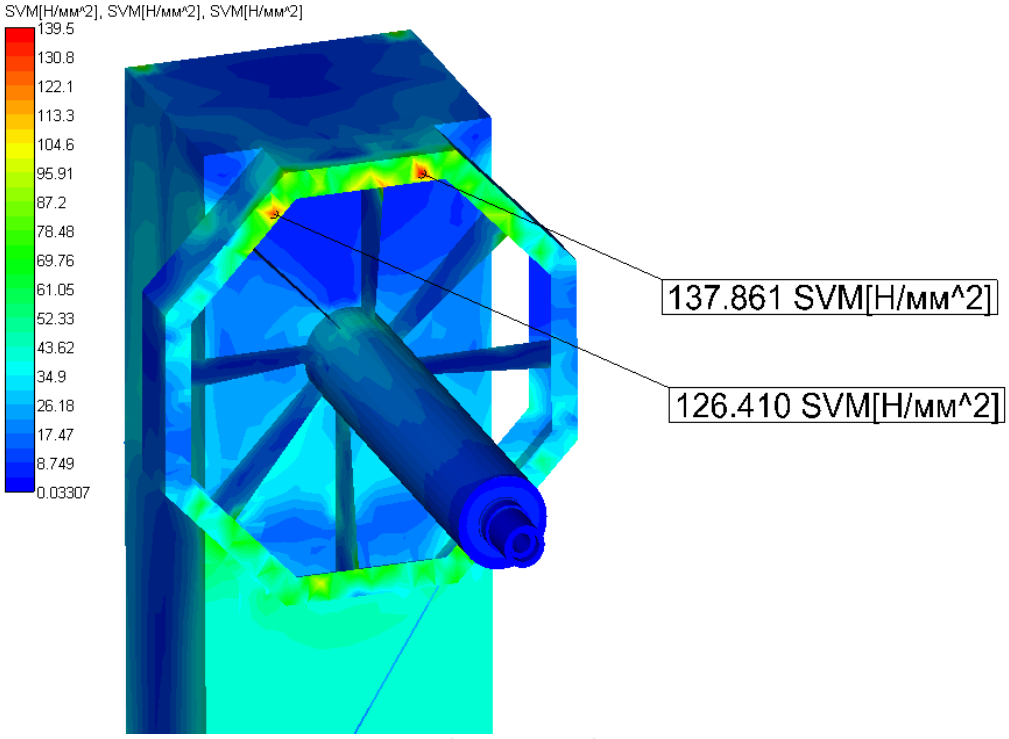


Fig. 6.4 Result map.

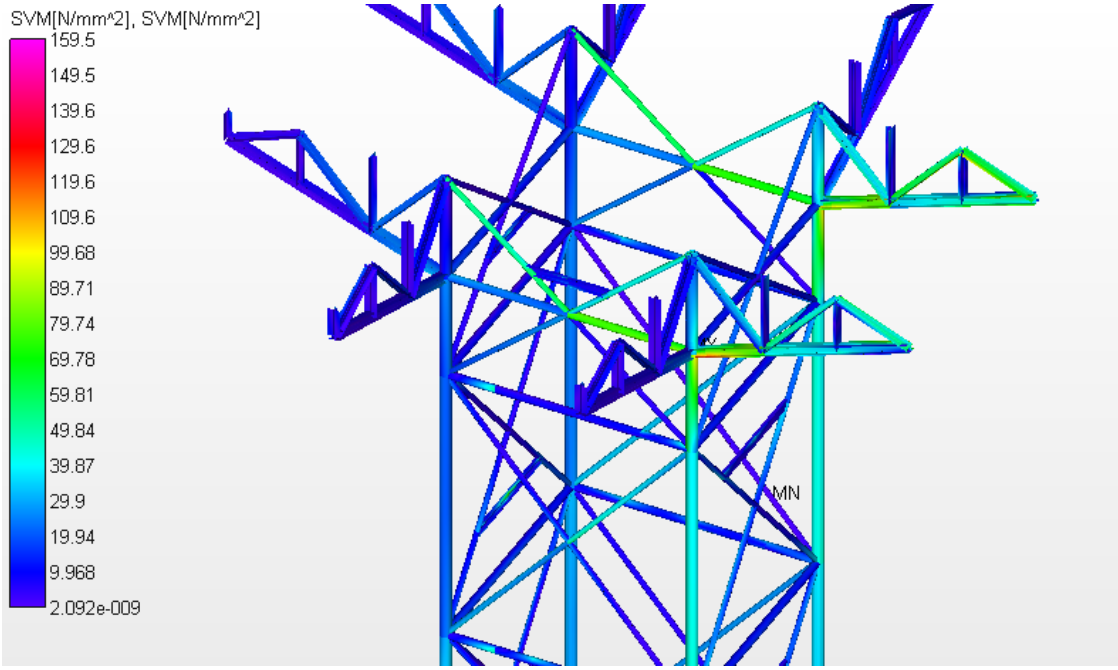


Fig. 6.5 Stresses on underformed structure in 3D vizualization

Map of the result distribution in rods, plates and volume elements (alternative)

Such results of calculation as stresses, displacements, forces, safety factor, etc. in rods, plates and volume elements can be displayed as an alternative result map. In order to display this map, in the File | Settings set a checkmark in the option Alternative Results Map.

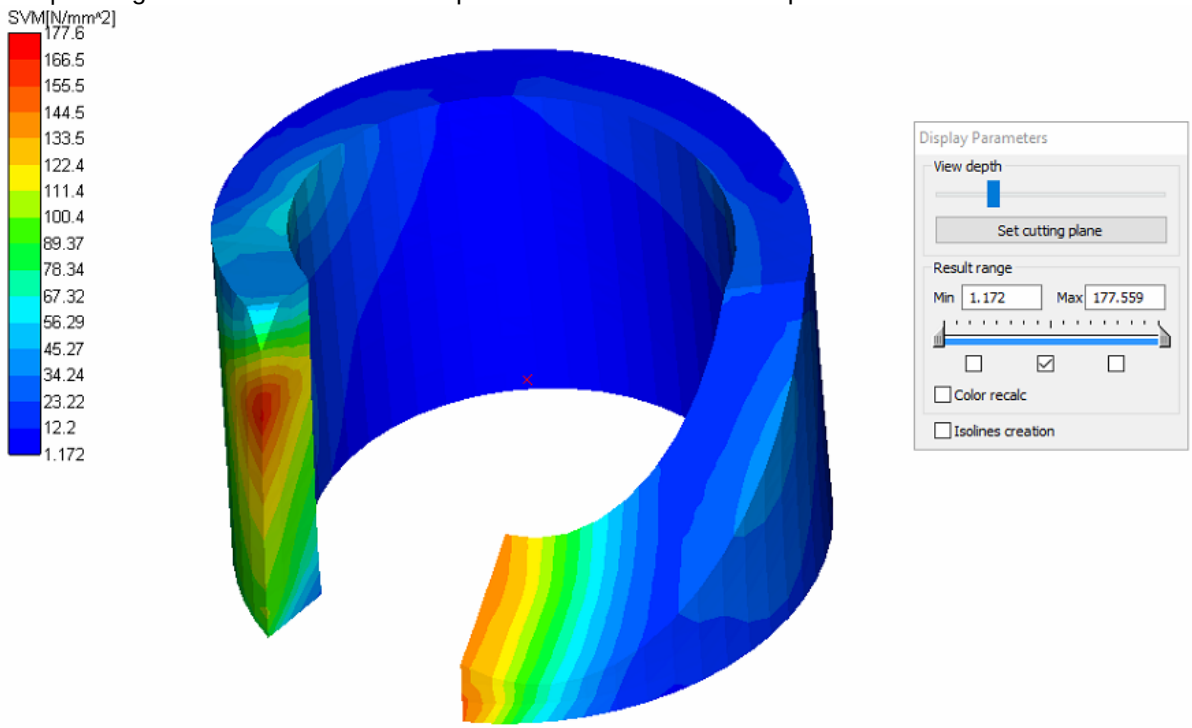


Fig. 6.6 Stress display on an alternative results map

Having opened an alternative results map, an additional Display Parameters window appears (see Figure 6.6.) In which a user can customize the view of the alternative results map.

The view depth allows to cut a part of the results map in the direction that matches the screen plane. In Fig. 6.6. after cutting off a part of the model, the model itself was rotated. The Set cutting plane button sets the depth of the cut according to the current position of a slider, but the plane of the cut is a plane parallel to the plane of the screen.

Using Min Max input fields or two result range limiters, the user can leave only a certain part of the results map for viewing, fading the part that goes to the specified range (Figure 6.7.).

In Fig. 6.7. only a part of the results map that lies in the specified range is shown. To be more specific, the view depth was returned to its initial state.

The entire range of the results with the displacement of the engines is now divided into three parts:

- From the beginning of the range to the left cursor,
- Between the cursors,
- From the right cursor to the end of the range.

Directly under the cursors there are three switches in which a user can set the checkboxes. These three checkboxes enable / disable the display of each of the above ranges of results.

The checkbox in the Color recalc option allows you to change the color range so that not the full range is displayed in it, but only the range of results that is currently displayed in the alternative results map.

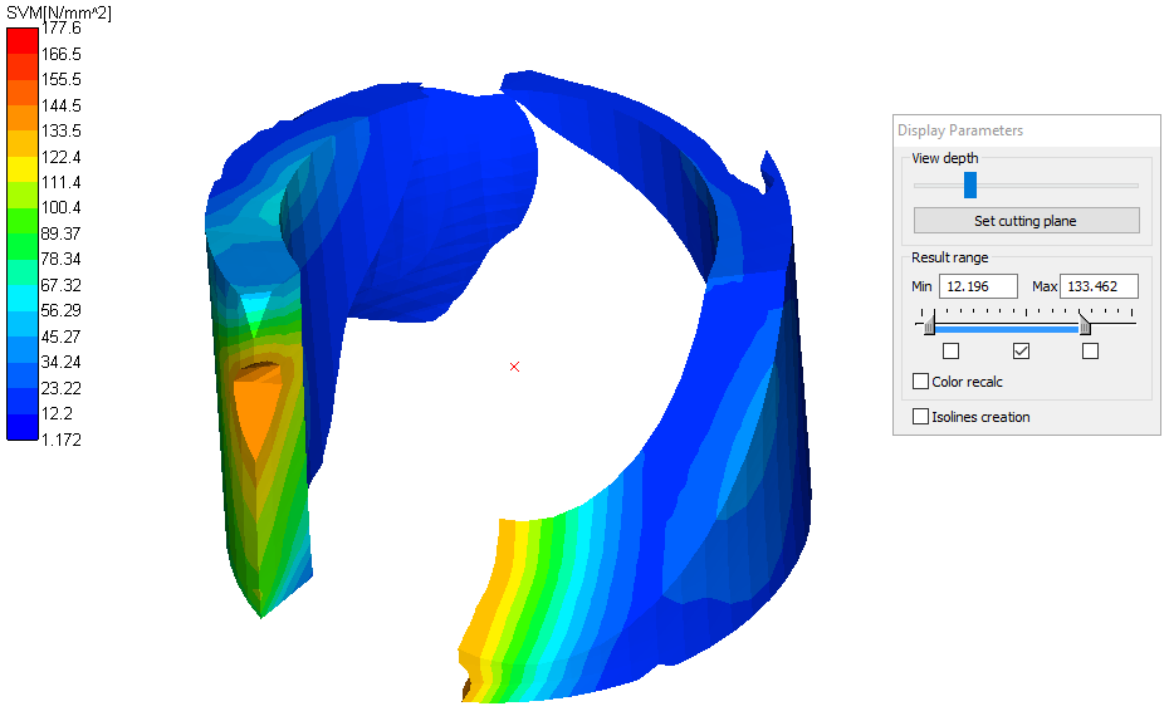


Fig. 6.7 Display part of the stress map with the selected range in an alternative results map

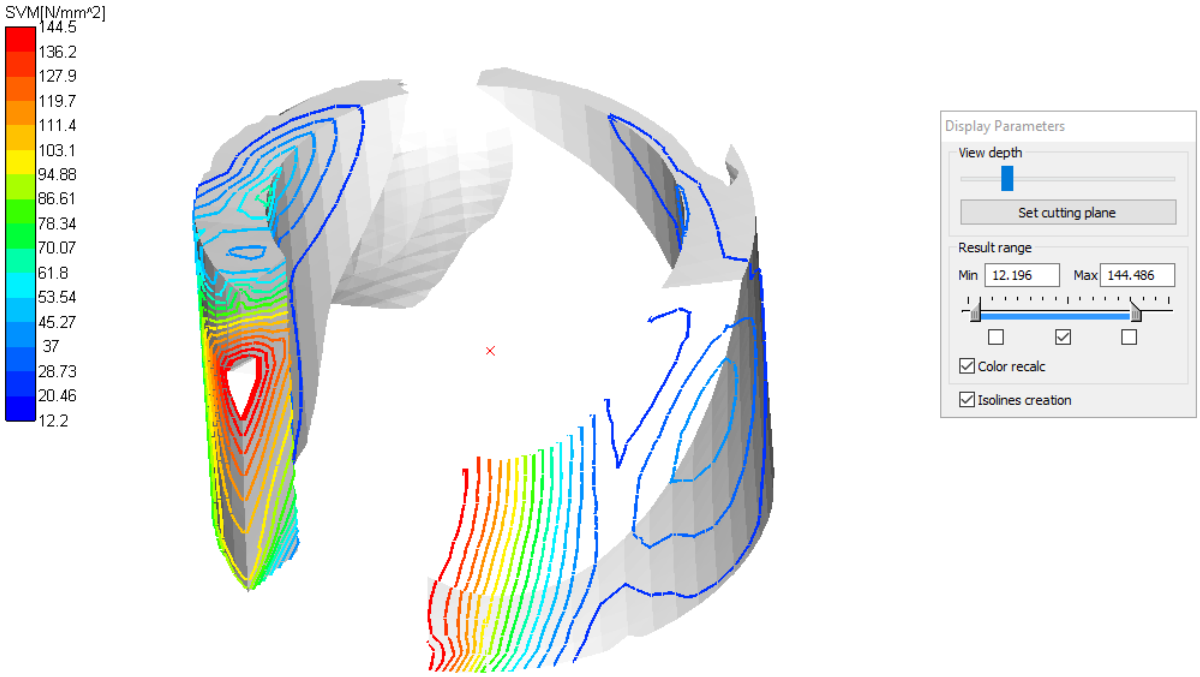


Fig. 6.8 Display part of the stress map using isolines in an alternative results map

When setting a checkmark in the option Isolines creation in the displayed part of the results map, the color lines will indicate the boundaries of the transition from one color tone on the result scale to another. By default, the results map is drawn with an isolevel number 16, and therefore, in the entire range of the results map, the number of isolines will also be 16.

Calculation results of topology optimization

To view the results of topology optimization in the Result Options window, select Topology optimization and the Static tab (Figure 6.9). In this tab one can see the results of the calculation for a specific load case obtained after the n-th iteration. The value of the distribution of the volume fraction

(Fig. 6.10), which determines the energy level on the elements of the model after carrying out the optimization calculation, is shown.

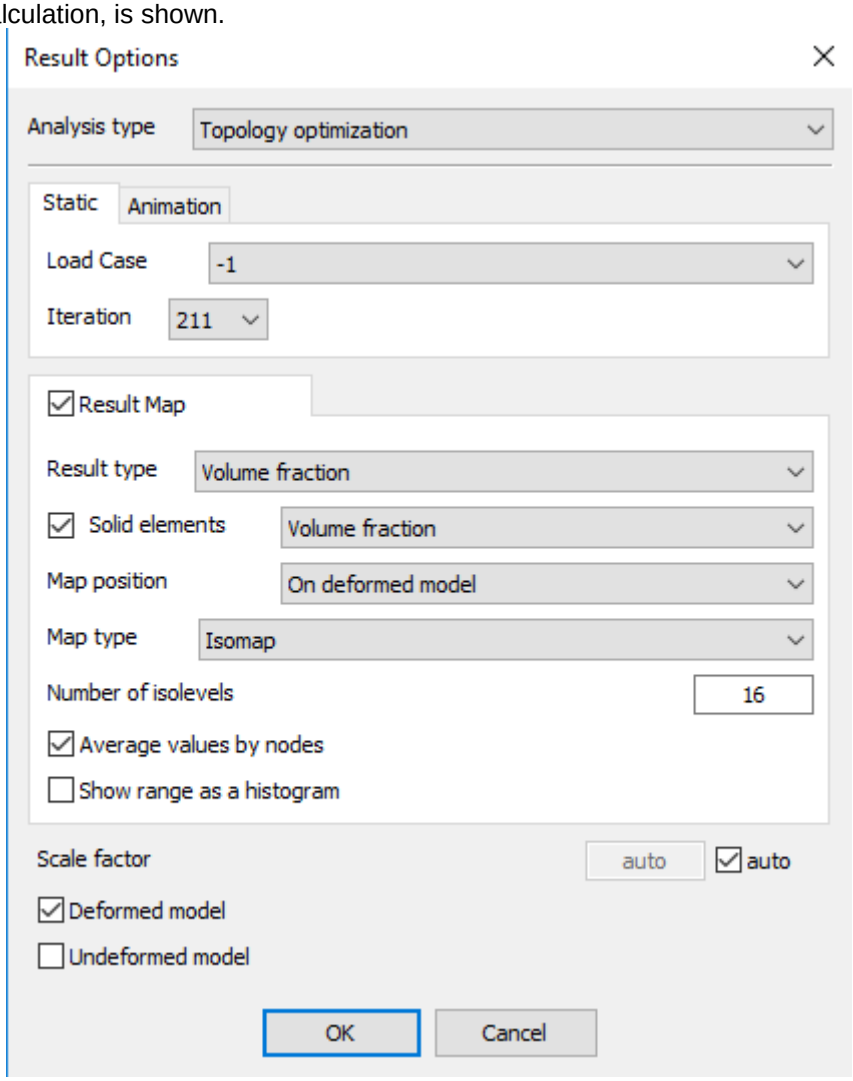


Fig. 6.9 Window Result Options with the settings for displaying the Volume fraction - the results of the calculation of Topology optimization

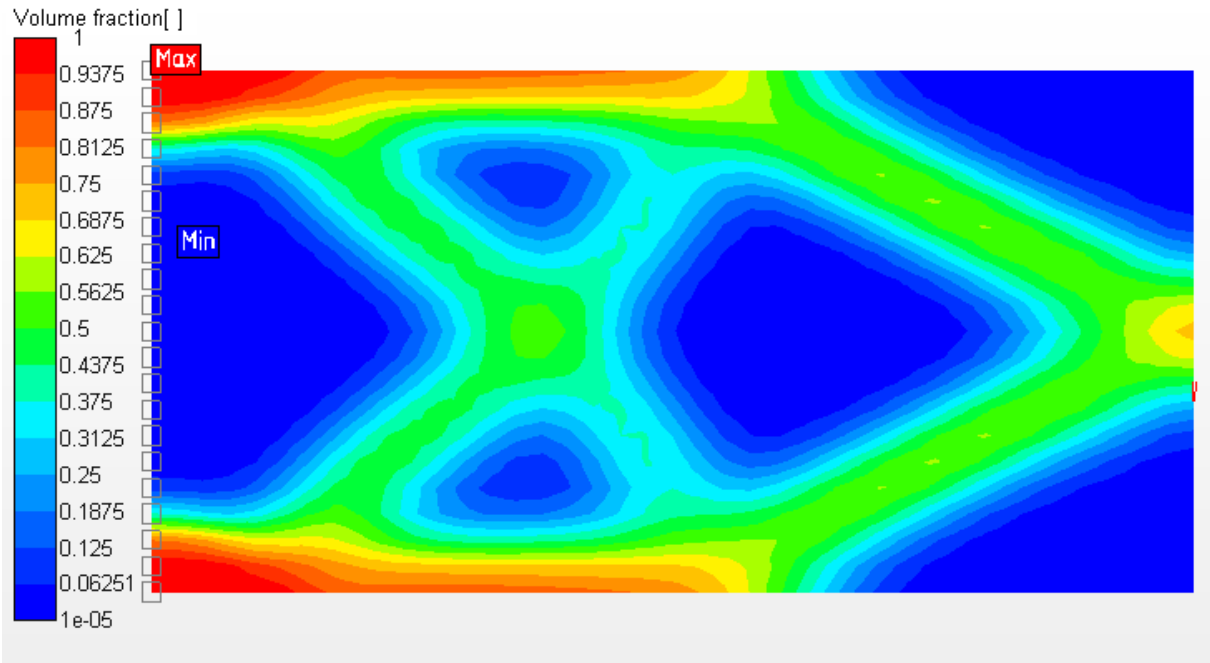


Fig.6.10 Volume fraction in the results map for the final iteration

When the Animation tab is selected, it is possible to animate the process of finding calculation results for different iterations.

The purpose of Topology Optimization calculation is to create a new mesh structure of the model having the material located in certain places.

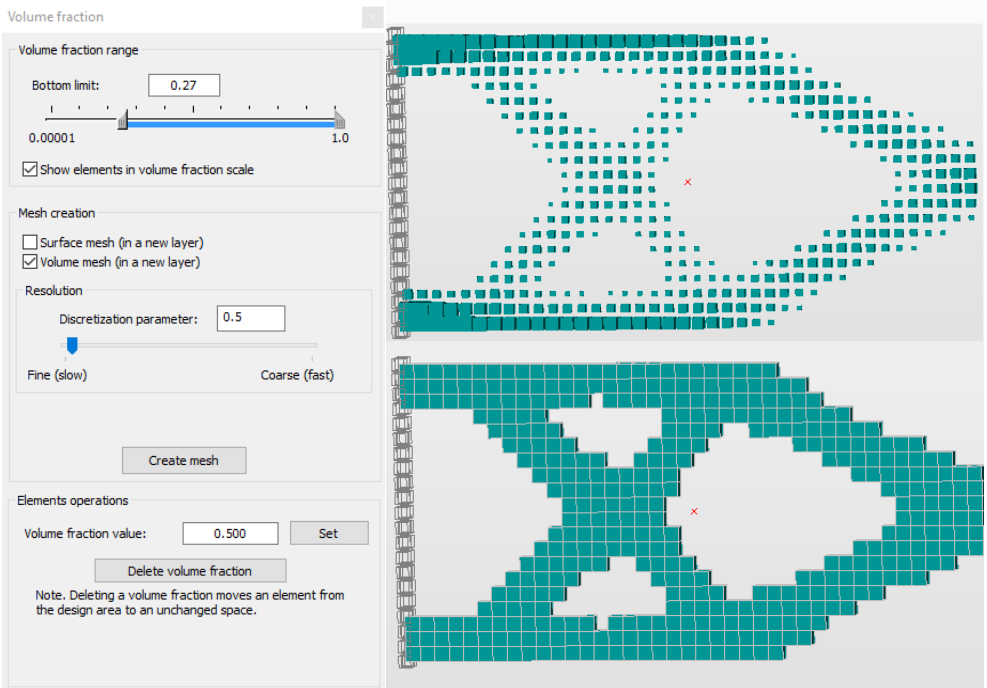


Fig. 6.11 Volume fraction range in the model through the window Volume fraction. From above - the display of elements in the scale of the volume fraction, from below - without the scale of the volume fraction

Figure 6.11 shows in the upper part an optimized model in which a checkmark is set in the option Show element in volume fraction scale - the dimensions of the finite elements are displayed in proportion to their volume fraction obtained as a result of calculation of topology optimization. The lower boundary of the volume fraction is set using the upper cursor of the volume fraction range (Fig. 6.11).

If a checkmark is not set in the Volume fraction range, then depending on the position of the cursor, the range of values of the volume fraction will show the elements that should be left in the model after Topological optimization.

The Discretization parameter determines the size of the finite elements in the generated mesh (meshes). At small values of the parameter, the spatial configuration of the grid will most fully reflect the result of the calculation, however, the small size of the elements and the large number of elements can make calculation difficult. At high values of the parameter, the mesh will consist of larger elements that produce a coarser approximation of the result of the optimized model, although the construction of such a mesh and subsequent calculations are performed significantly (1-2 orders) faster than with a fine mesh.

A new mesh of 4 nodal tetrahedral elements can be created in new layers after clicking the Create Mesh button. The result of creating a finer mesh of optimized structure in a new layer is shown in Fig. 6.12. Window The volume fraction with the parameters at which the new mesh was generated, is shown on the left.

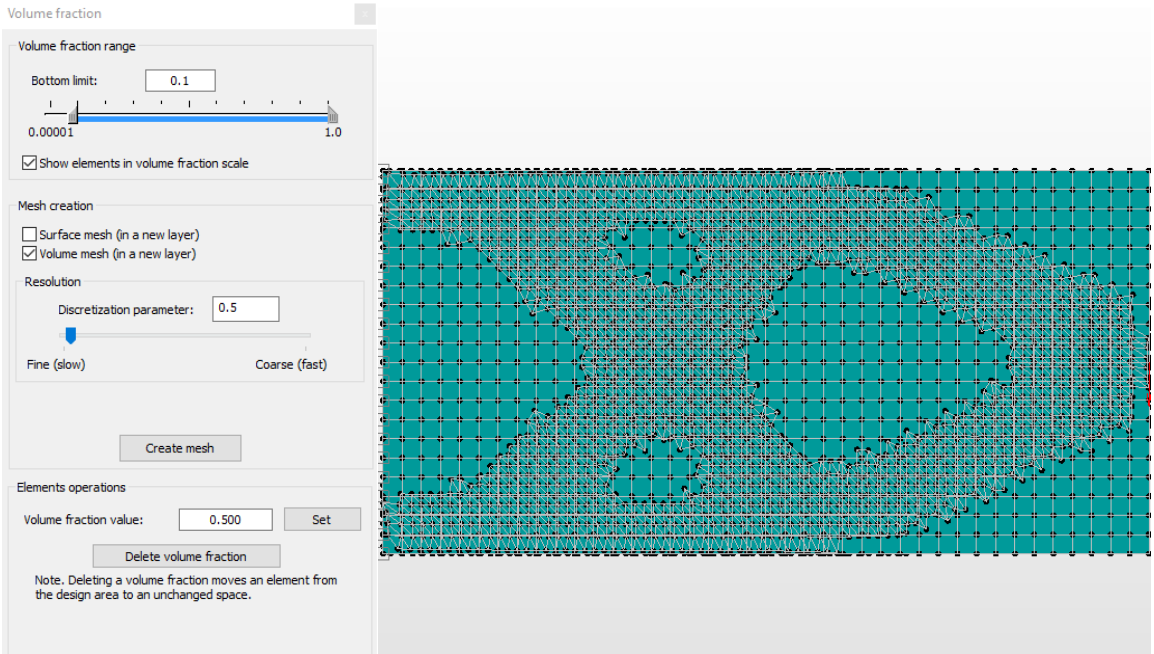


Fig. 6.12 Creation of a new mesh of optimized structure with a smaller step in comparison with the original one

Effective stress map in rod cross-section

User can see equivalent stress distribution in arbitrary section of the rod. To do that, select **Results / Stress in cross-section** menu command, which is available at viewing construction stress map. That will switch editor into stress-in-section review mode. Place cursor near required zone of the rod and click to call window with the stress map in the selected cross-section.

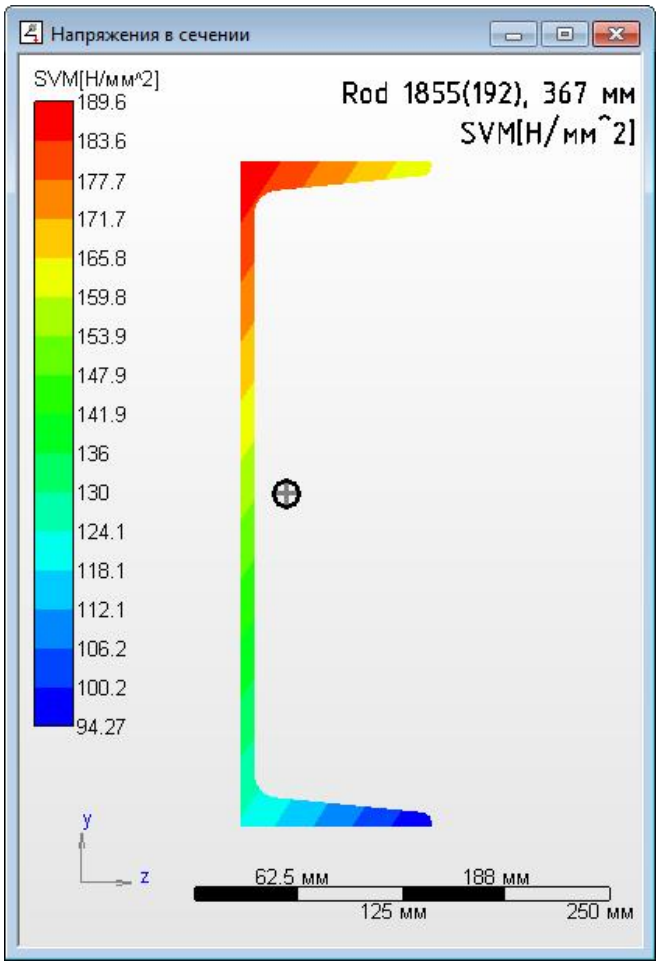


Fig. 6.6 Effective stress map in cross-section

Base reactions

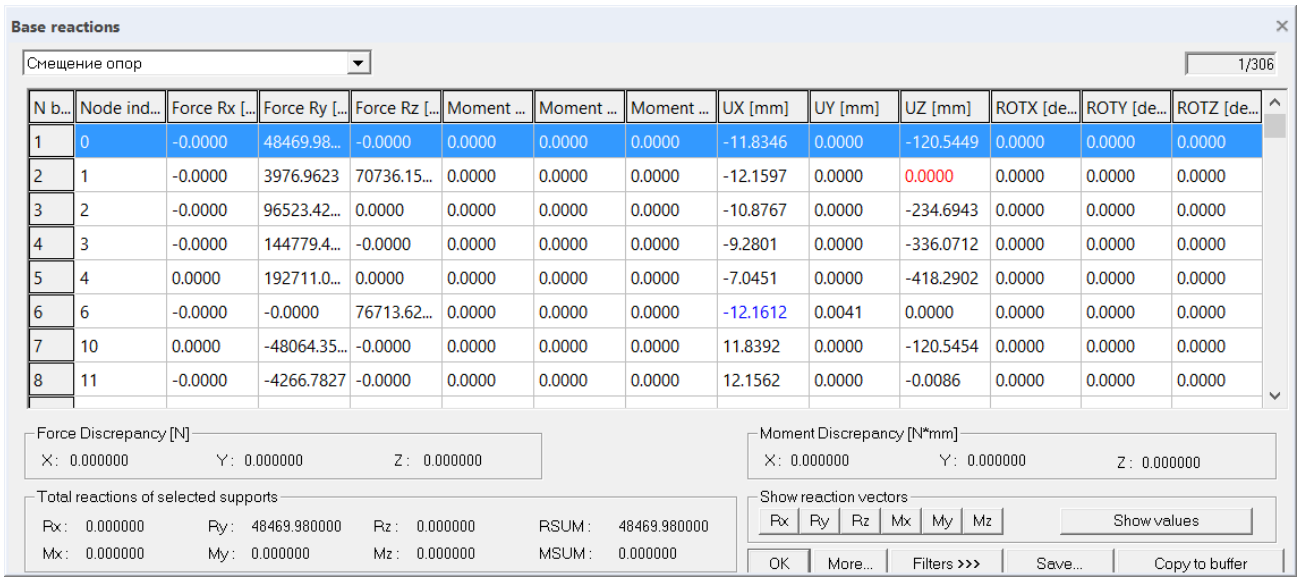
Base reactions are shown in global coordinate system as a table using command **Results / Base Reactions**. A selected node is highlighted in the table with a different color in structure editor windows. Clicking on the column header you can sort the bearings in ascending/descending order of the reaction in the selected column of the table.

The discrepancy of forces and moments (sum of reactions and external forces) appears in the global coordinate system. You can also print this table to a printer or to a file in RTF format.

Show reaction vectors– displaying vector and response values for the selected supports.

Filters allow you on/off supports displaying in certain directions of GCS and one-directional supports.

More button invokes dialog box with the information about the centre of gravity, total reactions in supports, etc.



Base reactions

Смещение опор

1/306

N b...	Node ind...	Force Rx [...	Force Ry [...	Force Rz [...	Moment ...	Moment ...	Moment ...	UX [mm]	UY [mm]	UZ [mm]	ROTX [de...	ROTY [de...	ROTZ [de...
1	0	-0.0000	48469.98...	-0.0000	0.0000	0.0000	0.0000	-11.8346	0.0000	-120.5449	0.0000	0.0000	0.0000
2	1	-0.0000	3976.9623	70736.15...	0.0000	0.0000	0.0000	-12.1597	0.0000	0.0000	0.0000	0.0000	0.0000
3	2	-0.0000	96523.42...	0.0000	0.0000	0.0000	0.0000	-10.8767	0.0000	-234.6943	0.0000	0.0000	0.0000
4	3	-0.0000	144779.4...	-0.0000	0.0000	0.0000	0.0000	-9.2801	0.0000	-336.0712	0.0000	0.0000	0.0000
5	4	0.0000	192711.0...	0.0000	0.0000	0.0000	0.0000	-7.0451	0.0000	-418.2902	0.0000	0.0000	0.0000
6	6	-0.0000	-0.0000	76713.62...	0.0000	0.0000	0.0000	-12.1612	0.0041	0.0000	0.0000	0.0000	0.0000
7	10	0.0000	-48064.35...	-0.0000	0.0000	0.0000	0.0000	11.8392	0.0000	-120.5454	0.0000	0.0000	0.0000
8	11	-0.0000	-4266.7827	-0.0000	0.0000	0.0000	0.0000	12.1562	0.0000	-0.0086	0.0000	0.0000	0.0000

Force Discrepancy [N]
X: 0.000000 Y: 0.000000 Z: 0.000000

Moment Discrepancy [N*mm]
X: 0.000000 Y: 0.000000 Z: 0.000000

Total reactions of selected supports
Rx: 0.000000 Ry: 48469.980000 Rz: 0.000000 RSUM: 48469.980000
Mx: 0.000000 My: 0.000000 Mz: 0.000000 MSUM: 0.000000

Show reaction vectors
Rx Ry Rz Mx My Mz Show values

OK More... Filters >>> Save... Copy to buffer

Fig. 6.7 Base Reactions dialog box

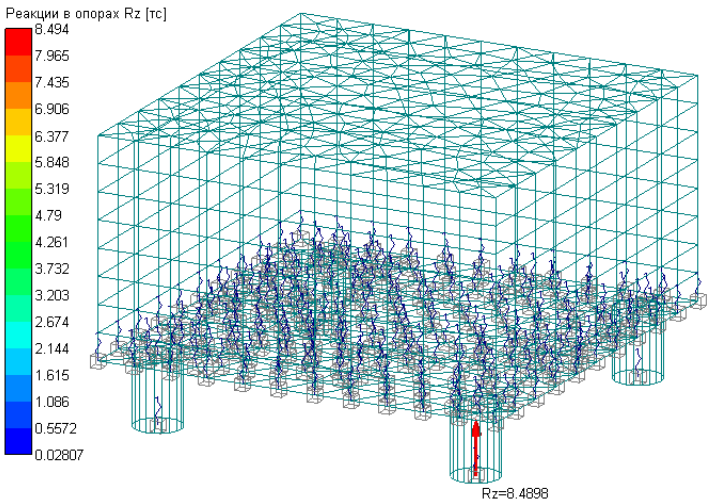


Fig. 6.8 Reaction vector Rz for the support selected in the table.

Composite Cross-section

this command makes is possible to see the stress and strain in the cross-section of the selected composite plate. First open any results map and after that, in the *Results* menu select the *Composite Cut* and then select the plate by clicking on the LMB to view the results in its section. After that, the *Composite Cross Section* dialog box opens, Fig. 6.15.

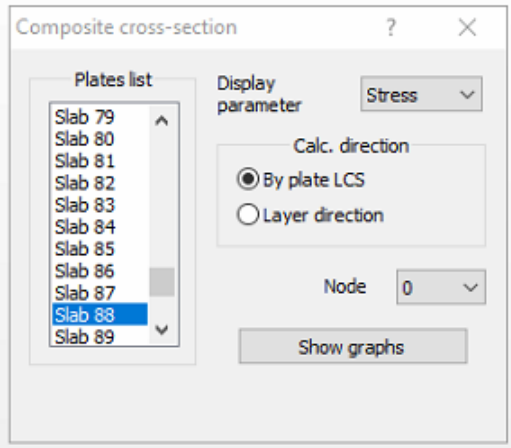


Fig. 6.15 The Composite Cross Section dialog box

In the *Plates List* window the selected plate is highlighted, although you can select any other plate to view the stress and strain in its section.
From the drop-down list, the *Display parameter,* it is selected what we want to view for this particular plate, *Stress* or *Strain*.
The selected parameter can be viewed in the Design direction *By plate LCS* or in the *Layer direction*.
Next, after selecting the *Node* (from the drop-down list), click the **Show Graphs** button.

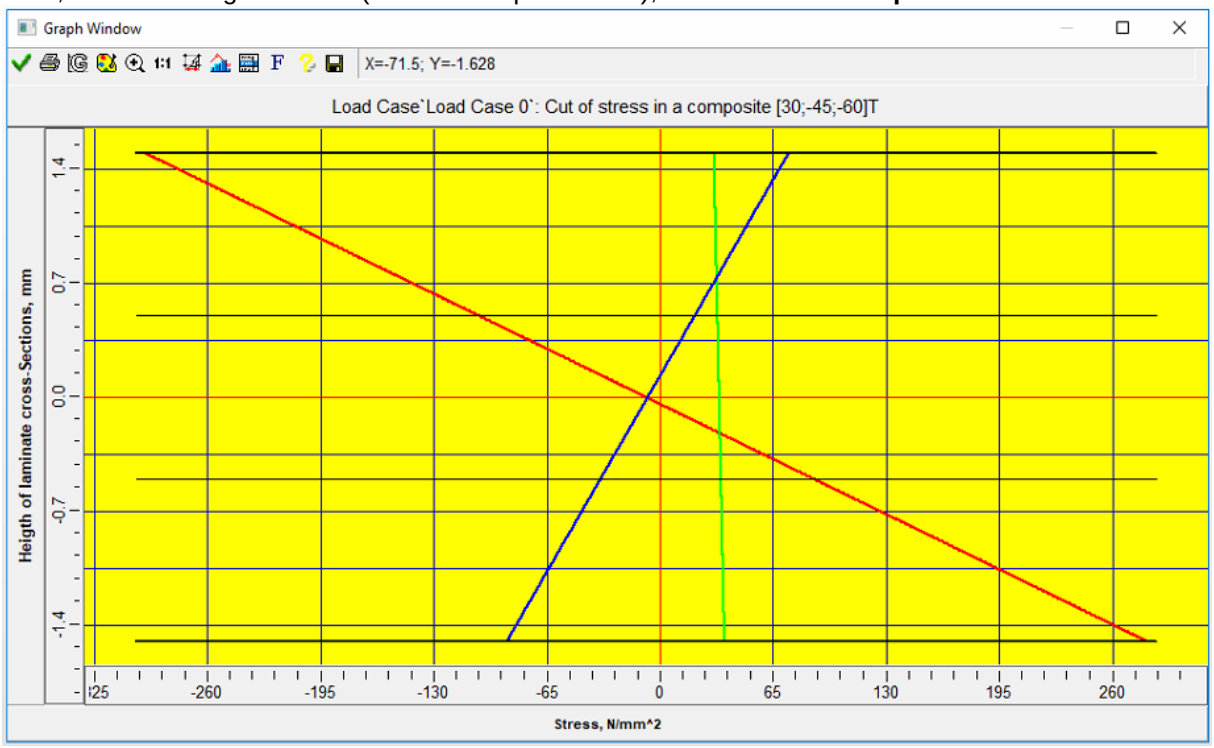


Fig. 6.16 The Graph Window with composite parameters by section

Buckling analysis results

In the 15th version the results of calculating buckling were made through the Results Map and the Results Options window (Figure 6.22). Also the old mechanism of viewing the results of calculating the stability was left (Figure 6.23)

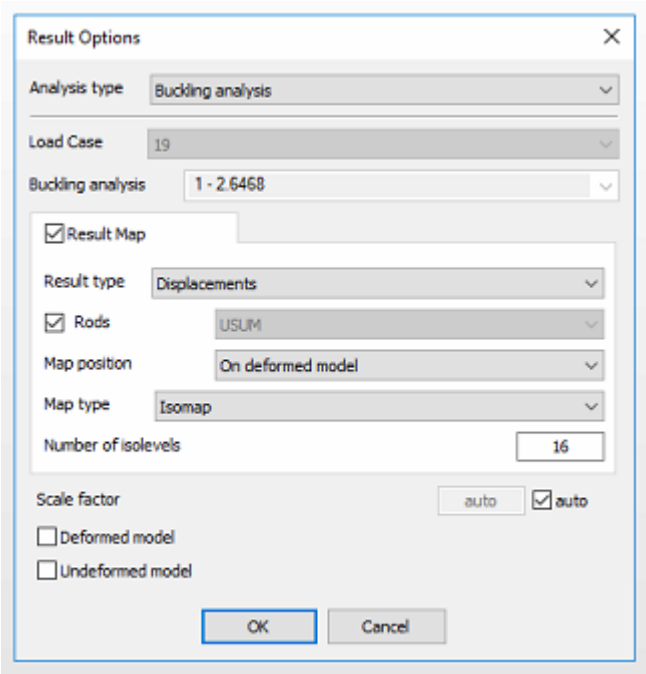


Fig. 6.22 The Result Option dialog box for displaying the Buckling Analysis results

Buckling results are:

- Buckling safety factors.
- Buckling shapes of a structure.

Results | Buckling command displays a table with the buckling factors. To view a buckling shape, select a row in the table and click "Shape".

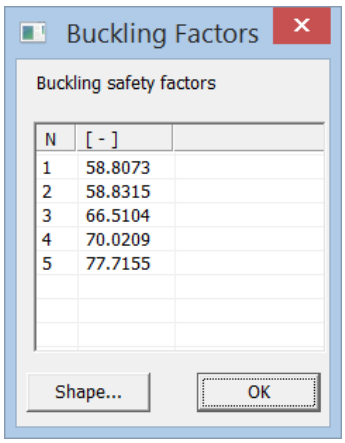


Fig. 6.9 Buckling safety factor dialog.

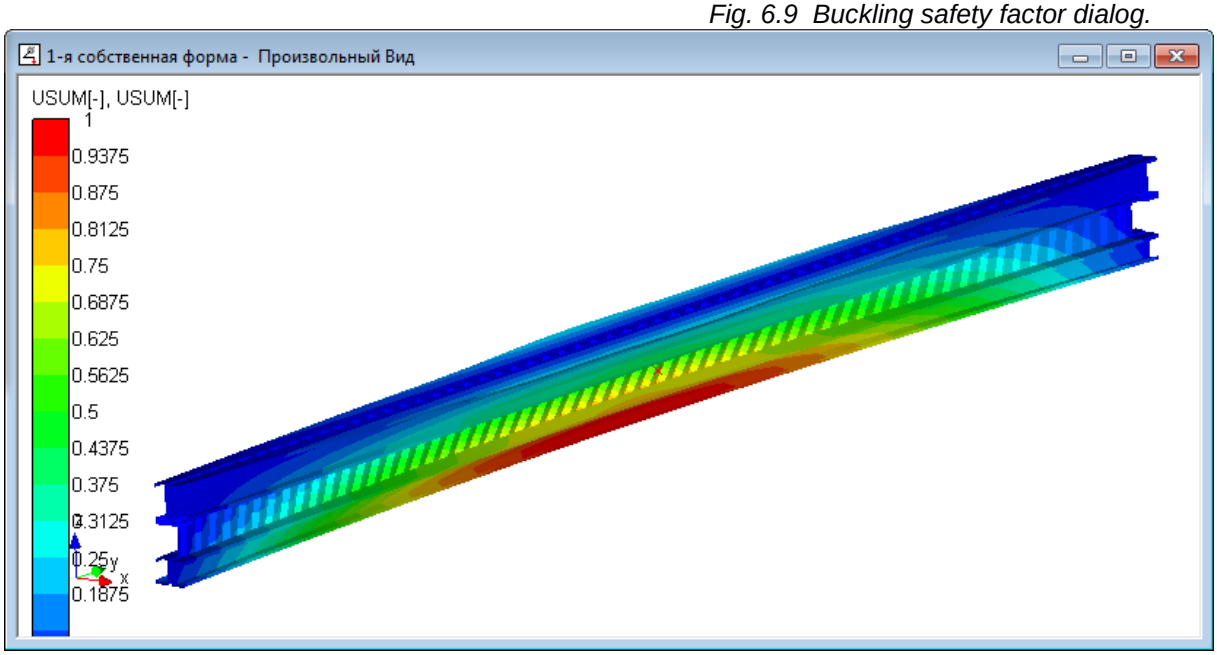


Fig. 6.10 1-buckling shape

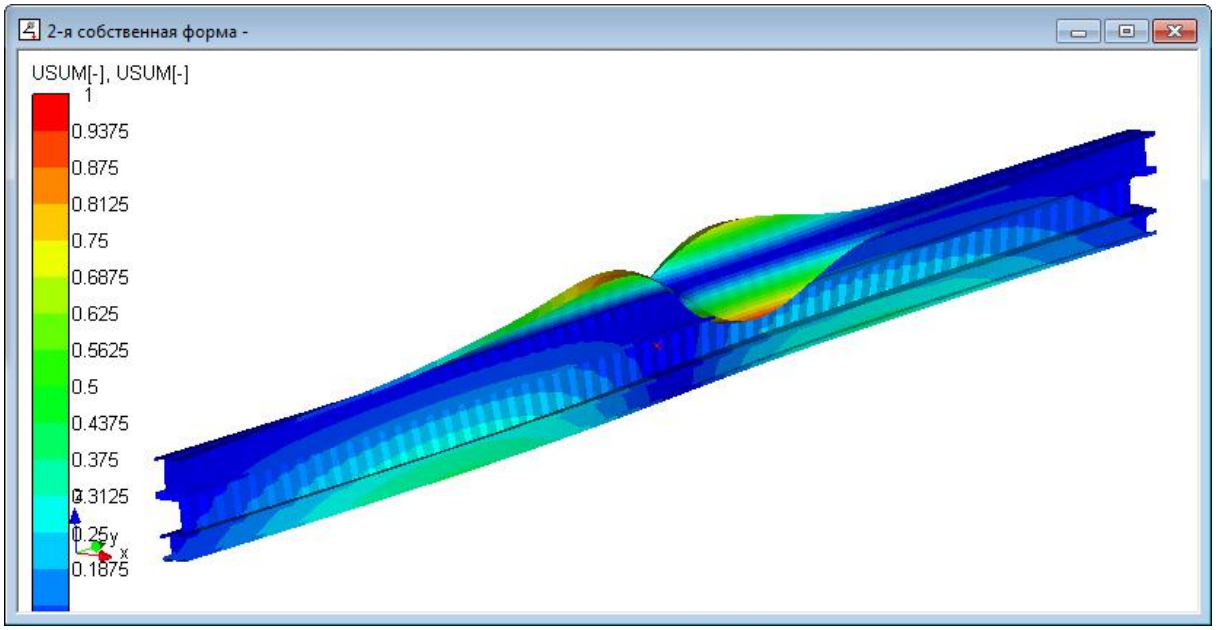


Fig. 6.11 2- buckling shape

Geometrical nonlinear analysis results

Results are the same items as for linear static calculation.

Calculation results of single-sided supports and ropes

The result of calculating one-sided supports and ropes is the display of "working" one-sided supports (non-working single-sided supports are crossed out). For the models with ropes, the results will be all those parameters that are characteristic of the results of static calculation, considering characteristics of the rod end element, i.e. the rope (flexible thread). Stresses, motion of the structure with ropes, as well as strength in the rope and sagging rope are available.

Sagging is the displacement of individual sections of the rope under the action of gravity relative to the straight line connecting the rope fixing points.

And one more feature. If the rope, as a rod element, is subjected to compression during the calculation, it will automatically go into a mode when it is under the action of gravity alone and, due to gravity, will exert "pulling" forces in the attachment points to the structure.

Results of physically nonlinear problem

Calculation of physically nonlinear problem is possible taking into account unloading (see description of **Calculation / Calculation Options** command).

At calculation with unloading two load cases are created instead of one: Load Case 0 - Loading and Load Case 1 - Unloading.

Besides results of linear static calculation maps of total relative strain are accessible for "Unloading" case and total relative strain, elastic strain and plastic strain for "Loading" case.

Components correspond linear (EPSX, EPSY, EPSZ), shear (EPSXY, EPSYZ, EPSZX) and intensity (EPSINT) of relative strain.

General nonlinear analysis results

Viewing of results is possible for two modes: Loading and Unloading.

General nonlinearity (geometric and physical at the same time) allows to take into account both geometrical and material nonlinearity simultaneously.

View of the results is available for two modes - Loading and Unloading.

Modal analysis results

In the 15th version, the results of calculating the natural frequencies were derived from the Results Map and the Results Options window (Figure 6.26). Also the old mechanism for viewing the results of the calculation of natural frequencies was left (Figure 6.27).

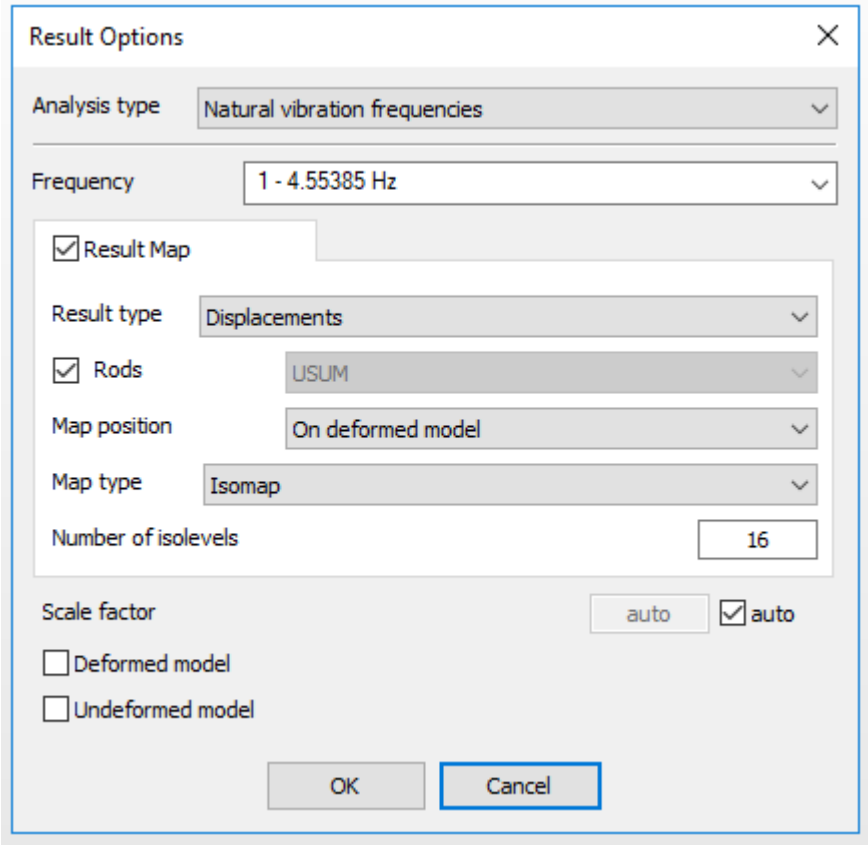
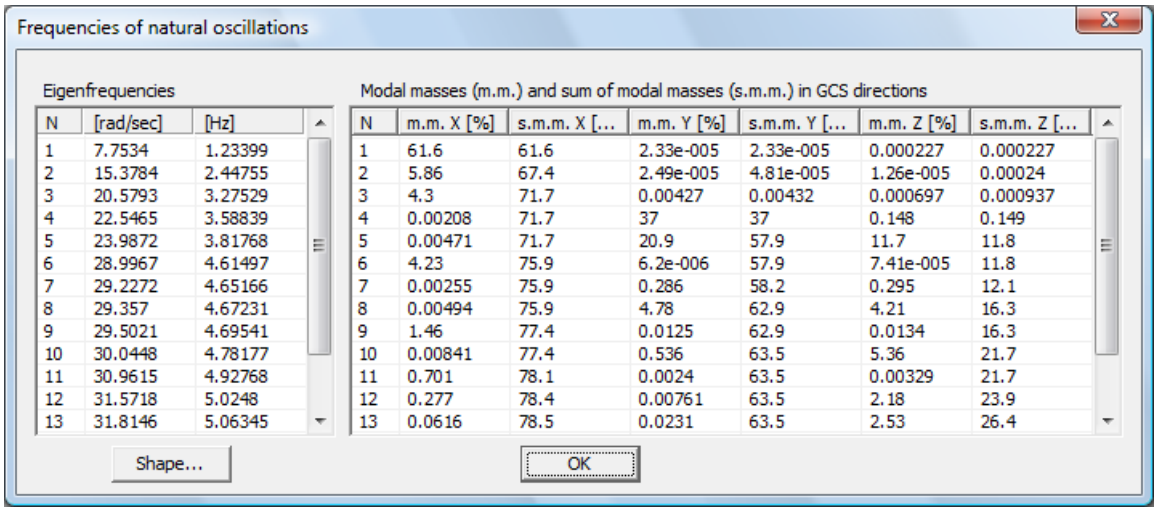


Fig. 6.26 Results Option dialog box for displaying the results of the calculation of Natural frequencies

Results for eigenfrequency calculation are:

- Eigenfrequencies for the structure
- Corresponding mode shapes for the structure
- Modal masses and sum of modal masses corresponding to each eigenfrequency. In the seismic code of many countries (Eurocode 8, UBC-97, Russian seismic code etc.), it is assumed that the sum of modal masses in each direction of the seismic load should be not less than the preset limit. Usually the horizontal component of the seismic action is assumed to be 85-90%, vertical - 70-90%.

Select command **Results /** **Natural frequencies** to see eigenfrequencies and mode shapes. Eigenfrequencies are shown as a table.



Frequencies of natural oscillations

Eigenfrequencies

N	[rad/sec]	[Hz]
1	7.7534	1.23399
2	15.3784	2.44755
3	20.5793	3.27529
4	22.5465	3.58839
5	23.9872	3.81768
6	28.9967	4.61497
7	29.2272	4.65166
8	29.357	4.67231
9	29.5021	4.69541
10	30.0448	4.78177
11	30.9615	4.92768
12	31.5718	5.0248
13	31.8146	5.06345

Modal masses (m.m.) and sum of modal masses (s.m.m.) in GCS directions

N	m.m. X [%]	s.m.m. X [...	m.m. Y [%]	s.m.m. Y [...	m.m. Z [%]	s.m.m. Z [...
1	61.6	61.6	2.33e-005	2.33e-005	0.000227	0.000227
2	5.86	67.4	2.49e-005	4.81e-005	1.26e-005	0.00024
3	4.3	71.7	0.00427	0.00432	0.000697	0.000937
4	0.00208	71.7	37	37	0.148	0.149
5	0.00471	71.7	20.9	57.9	11.7	11.8
6	4.23	75.9	6.2e-006	57.9	7.41e-005	11.8
7	0.00255	75.9	0.286	58.2	0.295	12.1
8	0.00494	75.9	4.78	62.9	4.21	16.3
9	1.46	77.4	0.0125	62.9	0.0134	16.3
10	0.00841	77.4	0.536	63.5	5.36	21.7
11	0.701	78.1	0.0024	63.5	0.00329	21.7
12	0.277	78.4	0.00761	63.5	2.18	23.9
13	0.0616	78.5	0.0231	63.5	2.53	26.4

Shape... OK

Fig. 6.12 Eigenfrequency values window

Example of mode shape window is shown below.

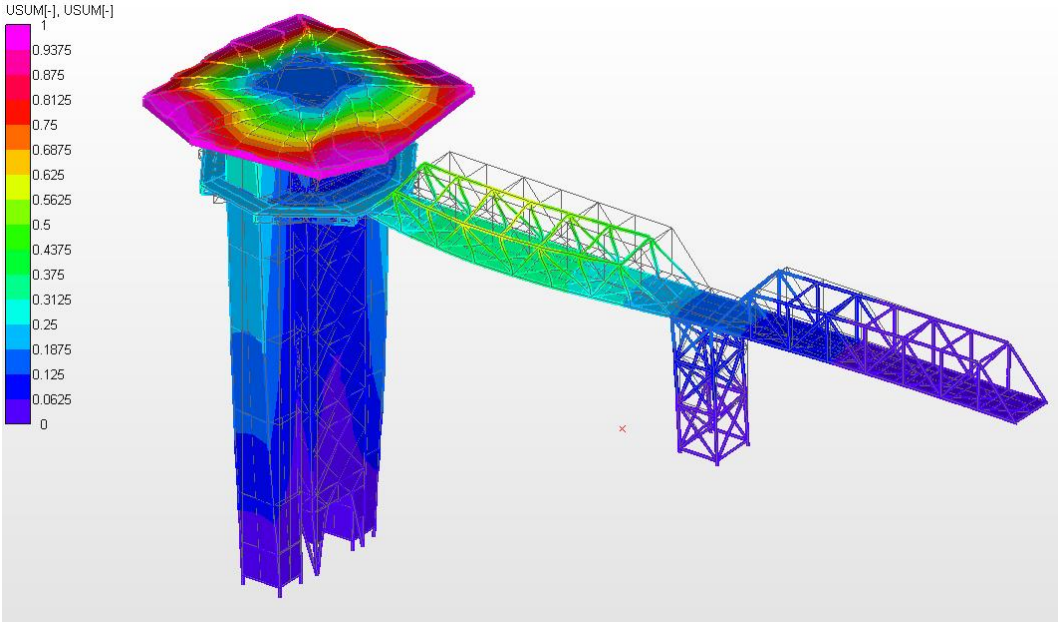


Fig. 6.13 Mode shape window

Results of forced oscillation calculation

In the 15th version the calculation results of forced oscillations were made through the Results Map and the Result Options window (Figure 6.29). In this window, on the *Static* tab for the selected time instant, you can view any of the static calculation parameters (Stresses, Movements, etc.).

On the *Animation* tab, for the selected option, view the animated image (Figure 6.30) from the time. After clicking the **OK** button, the Animation window appears (Figure 6.32) in which the animation settings are specified.

Also the old mechanism for viewing the results of calculating forced oscillations was also left.

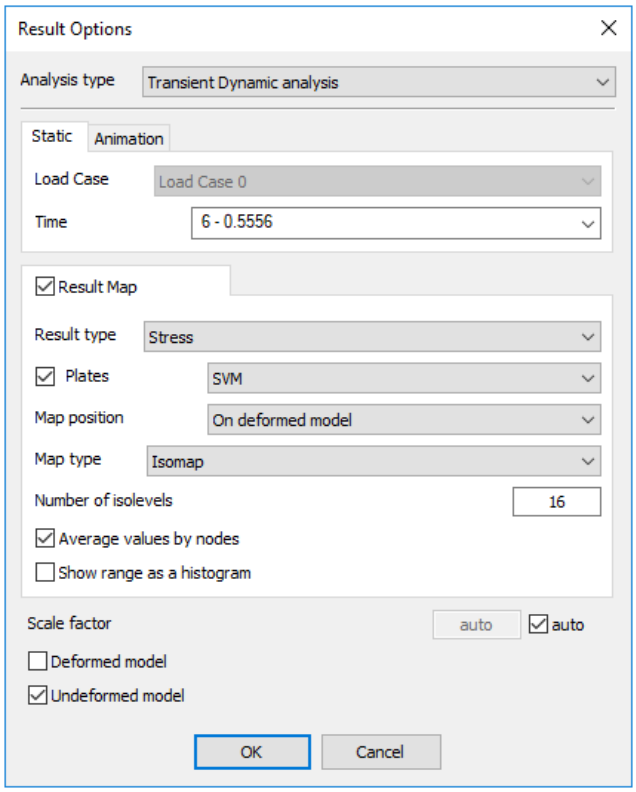


Fig. 6.29 Results Options Parameters dialog box for viewing static calculation results for Forced oscillations at the selected time point

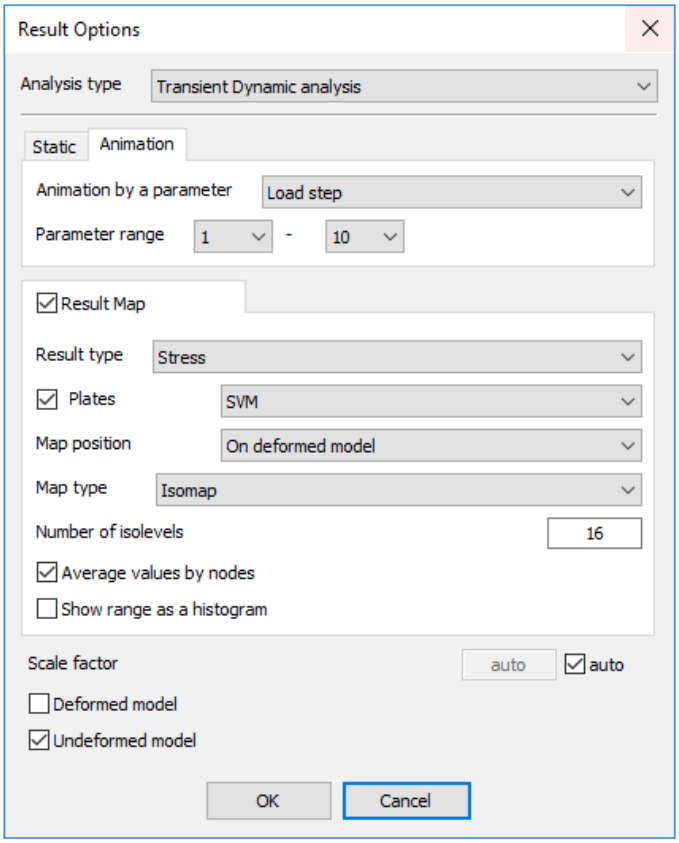


Fig. 6.30 Result Options dialog box for viewing the animated image of the calculation results of the selected parameter Forced oscillations by load steps

Results of forced oscillation analysis are:

- Node displacements
- Stresses in rods, plates and solid elements

- Base reactions
- Eigenfrequencies and mode shapes

Node displacements

You can see node displacements by selecting command **Results / Graph of Displacement**. An example of graph is shown below.

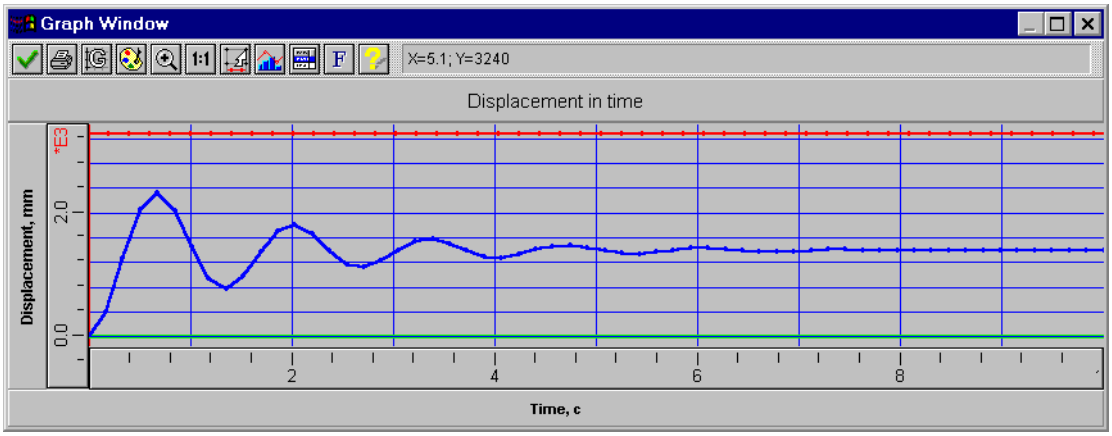


Fig. 6.14 Graph of node displacements window

Stresses, displacements, internal loads

Stresses acting in rods, plates and solid elements as well as displacements and internal loads are shown as color maps for each calculated moment in time, forming a slide show. An example of stress map window is shown below.

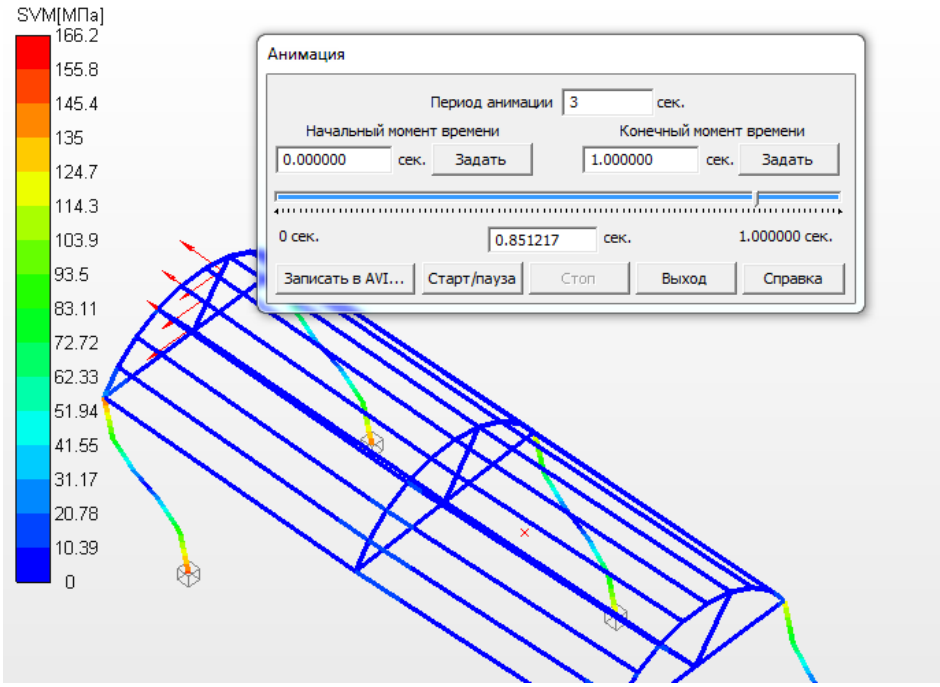


Fig. 6.15 Stress map animation window

To start animation, push the button *Start/Pause*. In an input field "Period of animation" set duration of time which the duration period of animation will last. A stress map (displacements) is drawn on a structure with a result range scale. With help of settings in the dialog window "Animation" one can view a not full oscillation period, but only some its part. After view of at least one animation loop the process itself can be written to a AVI file with help of a corresponding button.

You can also obtain graph of equivalent stress in arbitrary cross-section of any rod using command **Results / Graph of Stress**. Example of graph window is shown below.

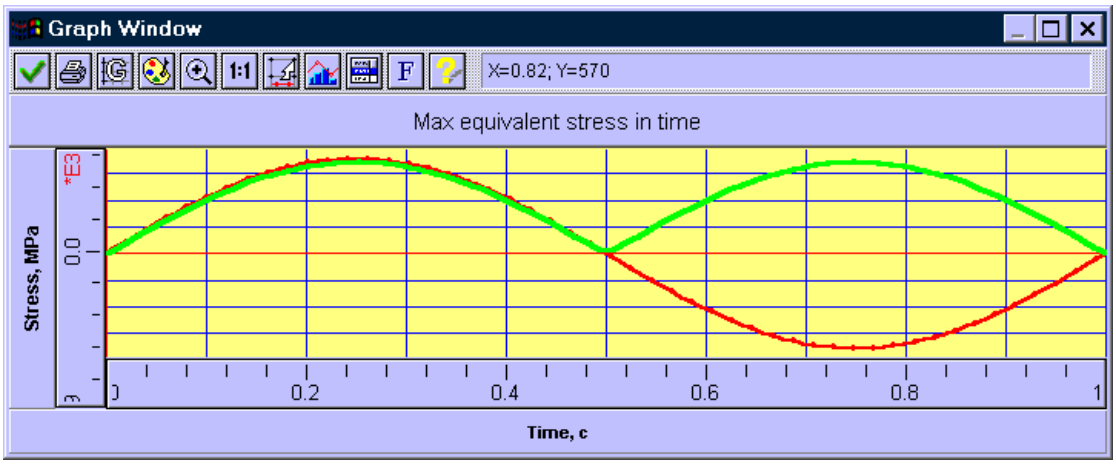


Fig. 6.16 Time-variable graph of stress in arbitrary cross-section window

Base reactions

Base reactions are shown in global coordinate system as a table for every calculated moment of time with the help of **Results / Base Reactions** command. See description of static calculation results for more detailed description.

Eigenfrequencies

You can see eigen frequencies and mode shapes by selecting **Results / Eigenfrequencies** command. See description of eigen frequency calculation.

Results of contact interaction calculation

Result of contact interaction calculation is the stress-strain state of structure (see results of static analysis), and also maps of normal and tangent forces distribution, interpenetration and condition of contact elements in contact area presented in the form of isoareas. After calculation it is possible to see condition of contact/target elements, estimate form and dimensions of contact zone by distribution of normal force, and also to check solution accuracy using interpenetration map. All results of contact interaction calculation are displayed separately from all structure, or on transparent model for viewing isoareas of contact and target elements.

Results of steel design elements

Bearing capacity calculation of metal structure rod elements is carried out for design elements and is implemented according to building regulations in this version of **APM Structure3D**. Strength / buckling check of rod elements can be executed by classical methods of strength of materials as well.

To perform this calculation, preliminary static or p-delta analysis is necessary, and as well as design elements creation. Calculation is initiated by **Calculation / Design** menu command or the **Calculation** button in dialog *Design elements*. **Calculation** button allows to inspect bearing capacity without closing the dialog window.

When the design element does not pass all of checks, there is an opportunity to select section automatically from cross-sections library. For this purpose at installation of design element properties, it is necessary to check *Select section* and to choose cross-sections library, having specified path to it.

For correct work of bearing capacity check algorithm and selection of sections, it is necessary to observe the type of the section setting to a design element. I.e. section of rods forming a design element and sections in the library chosen for selection should be of the same one type. For example, if the section of rods is the I-beam, only I-beam sections should be in the library for selection.

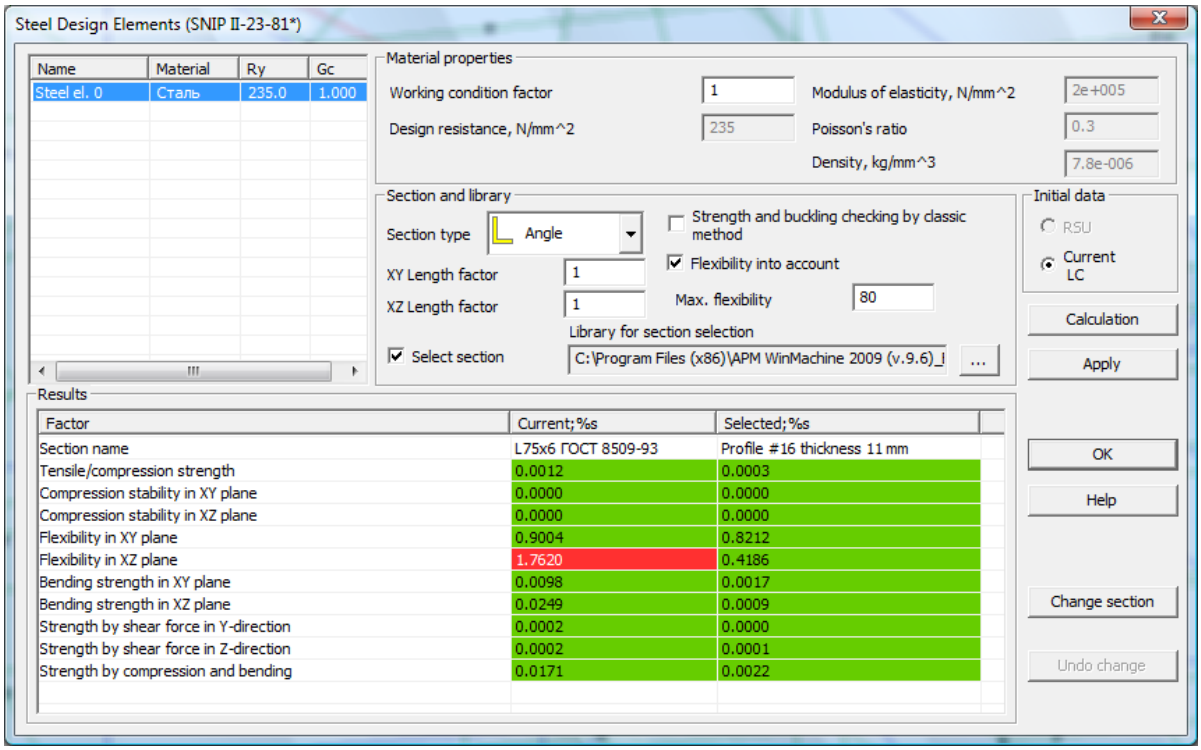


Fig. 6.17 Results dialog box

Check type is listed in the first column of the table, and operating ratios are given in the second and in the third column correspondingly. So, for example, use factor will be the attitude of an operating compressing / stretching stress to maximum allowed for check «tensile / compression strength». Use factor will be less then unit if the design element meets the criteria of that check type. One can see, that some checks have not passed (they are highlighted), and during calculation the section was chosen that meets all building regulations checks.

The **Change section** button performs automatic replacement of rods cross-section constituting a design element with the selected one. The **Undo change** button returns initial section to the rods that form a design element.

Results of reinforcing elements

Result of design calculation is definition of reinforcing intensity and selection of reinforcement diameter and step for columns, girders and plates. Result of checking calculation is the use factor which should be in range from 0 to 1.

To see reinforcing results for each design element use **Design / Design Elements** command. It is necessary to choose an interesting design element in the appeared dialog window in the list at the left, and reinforcing results according to the first and second groups of limiting states will be accessible for viewing in tabs of *Results* group. Results content will depends on type of selected design element.

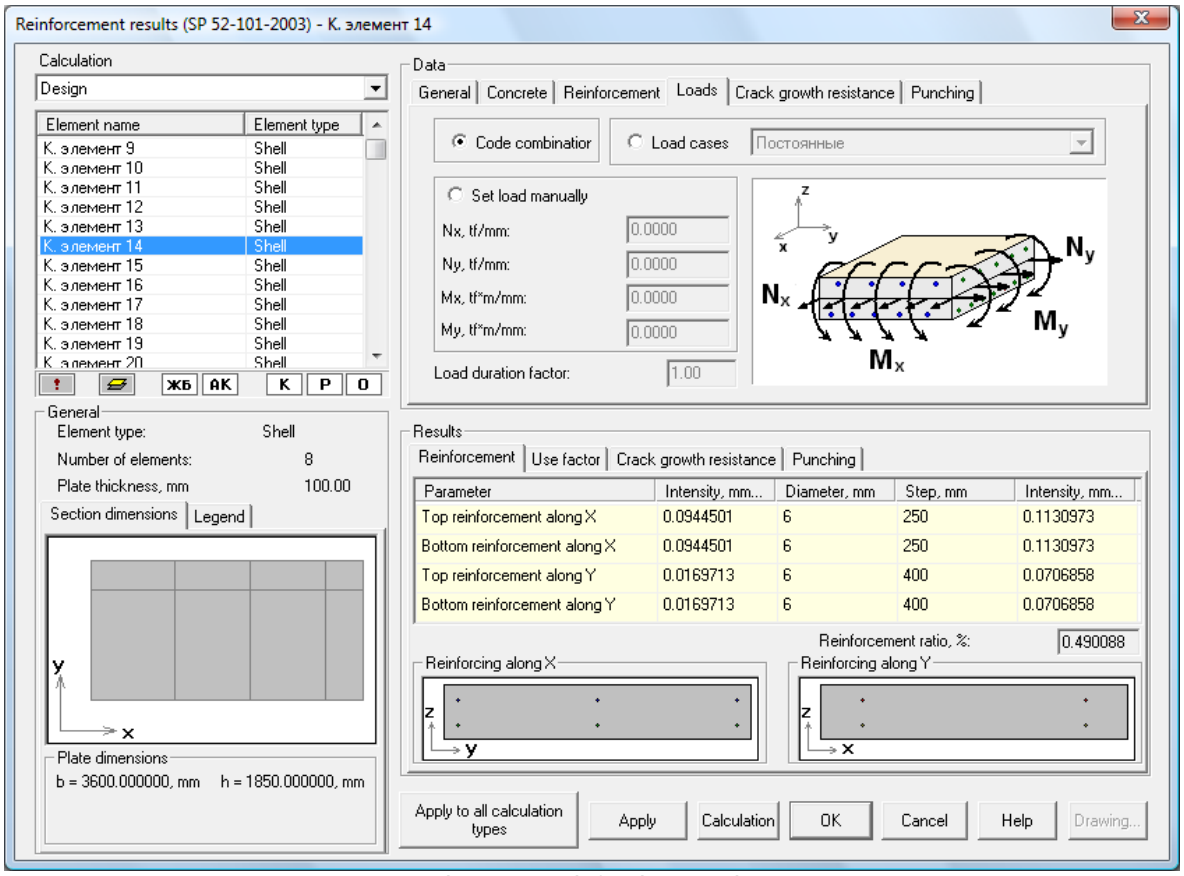


Fig. 6.18 Reinforcing results

Reinforcement results presentation

For expanded visual representation of reinforcing results commands of *Reinforcement Results* toolbar are intended. This toolbar is accessible if the **Design / Design Element Type / Reinforced Elements** option is checked. The description of commands is resulted below.

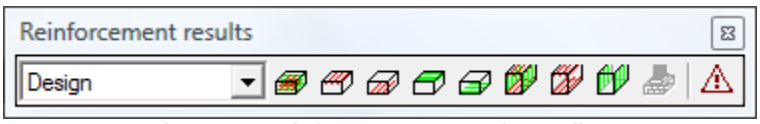


Fig. 6.19 Reinforcement Results toolbar

- Show plate reinforcement
- Show upper reinforcement along X axis
- Show bottom reinforcement along X axis
- Show upper reinforcement along Y axis
- Show bottom reinforcement along Y axis
- Show rod reinforcement
- Show rod longitudinal reinforcement
- Show rod transverse reinforcement
- Show post foundation
- Show elements with unsuccessful calculation
- show only elements with the impossibility of calculation
- Show only elements without longitudinal reinforcement

Chapter 7. Design of Structure Steel Joints

This chapter describes automatic creation of the drawing documentation for joints of metal constructions.

APM Structure3D allows user to create drawings of the following typical joints:

- Pinned column base.
- Fixed column base (2 hooks).
- Concrete column base.
- Beam to beam connection.
- Corner of frame connection.
- Column to beam connection.
- Beam to column connection (angled).
- Beam to beam connection (angled).
- Beam to column connection (flanged).
- Beam to beam connection (flanged).
- Plate-rod connection.
- Plate-internal node connection.
- Plate-zone node connection.
- Tube connection.

For drawing creation of typical joint, it is necessary to select connected rod elements and to press the button of the corresponding joint type on *Steel connection* toolbar. If the selected elements cannot form a certain connection type the corresponding button becomes inactive or the warning message appears on the screen.

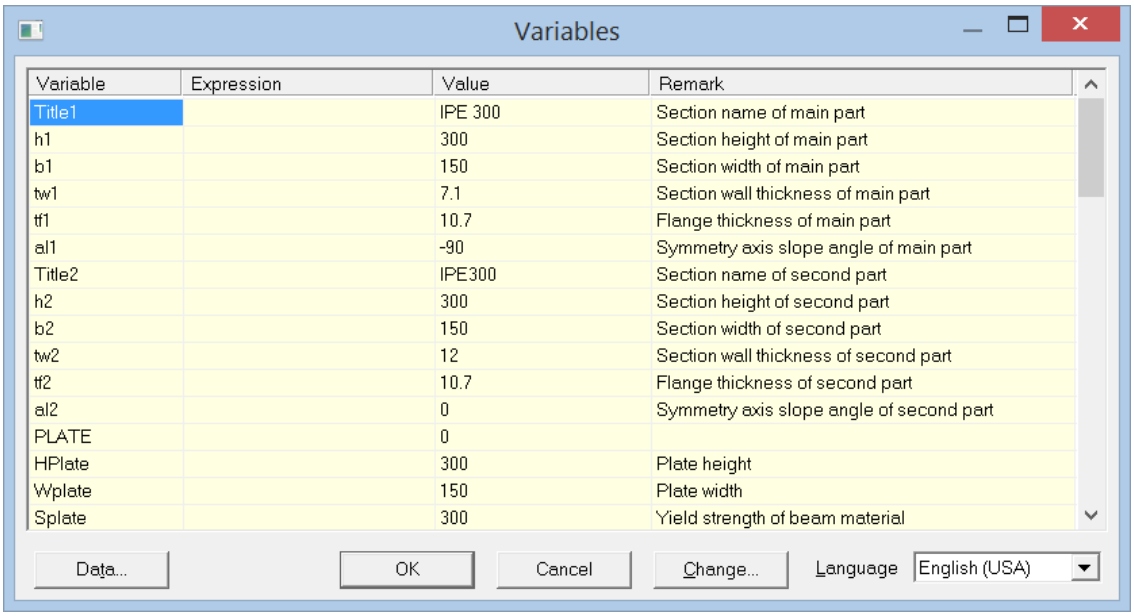
Fig. 7.1 Connection of steel elements toolbar

After activation of a command, a graphical editor *APM Graph* with a corresponding parametric model is run. The values of variables of a drawing model variables (type of section, geometric dimensions, material …), correspond to a calculated model. For editing of connection parametric models the *APM Graph* editor toolbar "Node model parameters" is serving. Editing of parameter models can be carried out in three ways:

– in a variable list;
– with use of dialog windows with icon dialog windows.
– printing and editing dimensions directly on the drawing.

– toolbar **"Parameters"** of a connection model.

 – the command calls a dialog with a variable list. The variable list depends on a parametric model. For editing a variable must be selected in a list and the button "Change" of a dialog window must be pushed.



Variable	Expression	Value	Remark
Title1		IPE 300	Section name of main part
h1		300	Section height of main part
b1		150	Section width of main part
tw1		7.1	Section wall thickness of main part
tf1		10.7	Flange thickness of main part
al1		-90	Symmetry axis slope angle of main part
Title2		IPE300	Section name of second part
h2		300	Section height of second part
b2		150	Section width of second part
tw2		12	Section wall thickness of second part
tf2		10.7	Flange thickness of second part
al2		0	Symmetry axis slope angle of second part
PLATE		0	
HPlate		300	Plate height
Wplate		150	Plate width
Splate		300	Yield strength of beam material

Fig. 7.2 An example of a variable edit dialog for the model "The Connection Column-Beam".

– the command calls an icon dialog window for the choice of connection parameters. General view of a dialog window and the tabs depend on a parameter model. The main dialog icon windows are presented for different connection types further.

There is editing of variables sizes and inscriptions possibly and directly on the model. The sizes and the inscriptions available for editing are selected for patterns in **blue color** and the dependent sizes not available for editing – in **black color**. For editing choose the command **"Edit/ Modification"** and then click size. After it one can change in the dialog box that appeared the value or the text of a variable.

Further, we shall consider each type of connection in detail.

Pinned column base

Shortcut:

Hook type:

Wedge type:

Type of strengthening:

Beam to plate connection type: welded , bolted
Number of bolts is unlimited, but distance between them is constant.
Bolts can be aligned with respect to beam axis.

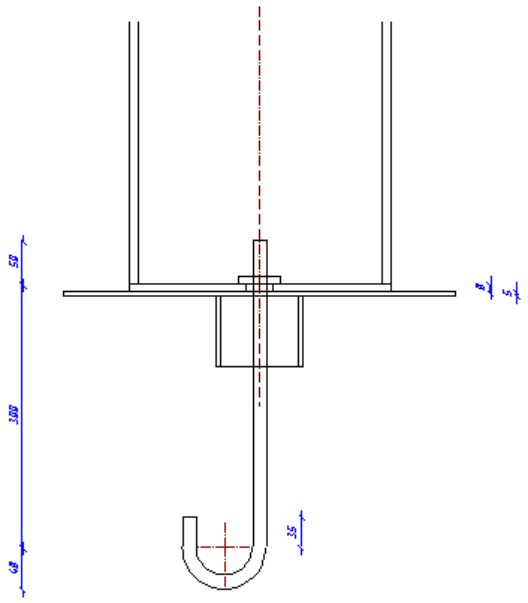
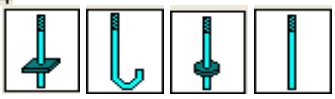


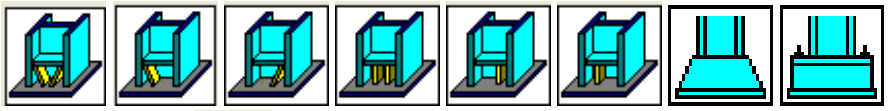
Fig. 7.3 Example of pinned column base

Fixed column base (2 hooks)

Shortcut: 

Hook type: 

Wedge type: 

Type of strengthening: 

Beam to plate connection type: welded 

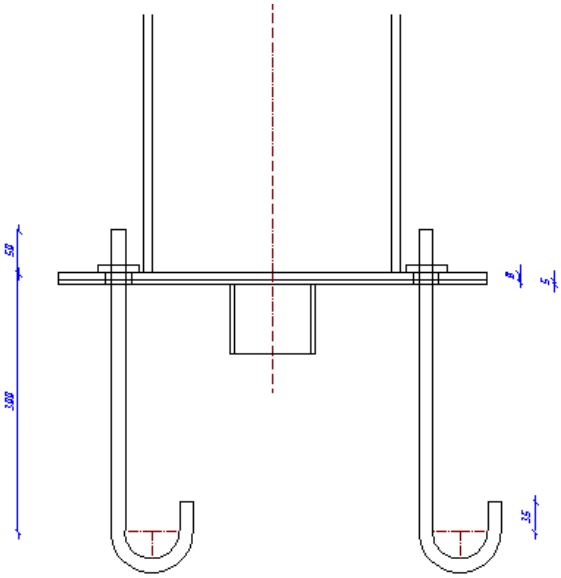


Fig. 7.4 Example of fixed column base

Concrete column base

Shortcut: 

Type of beam profile: I-beam, channel, L-bar, T-profile.
All profiles expect of T-profile can be rotated on 90 degree. T-profile – 180 degree.

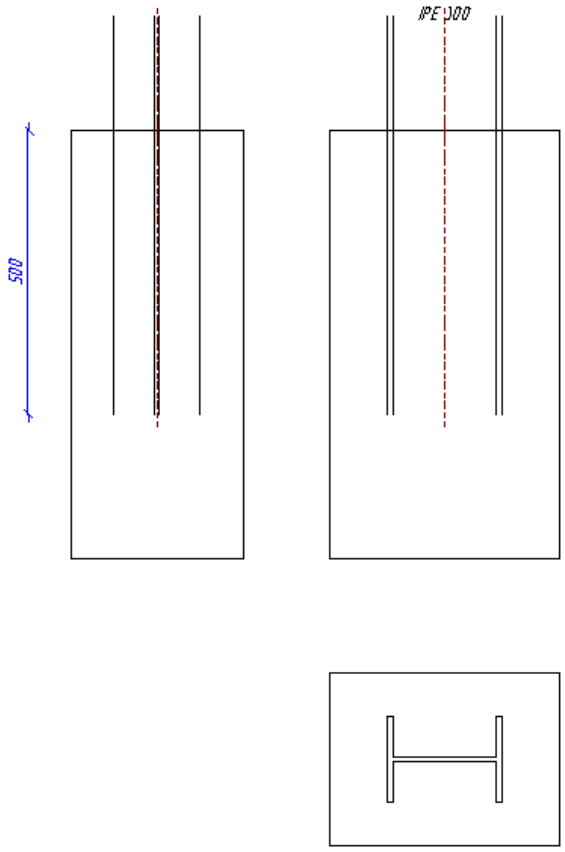


Fig. 7.5 Example of concrete column base

Beam to beam connection

Shortcut: 

Connection of column and beam: welded, bolted.
Number of bolts is unlimited, but distance between them is constant.

Position of plate (bisector): vertical  ,

perpendicular main part 

Edges of rigidity: 0 – edge is absent 

1 – triangular edge  , 

2 – rectangular edge  , 

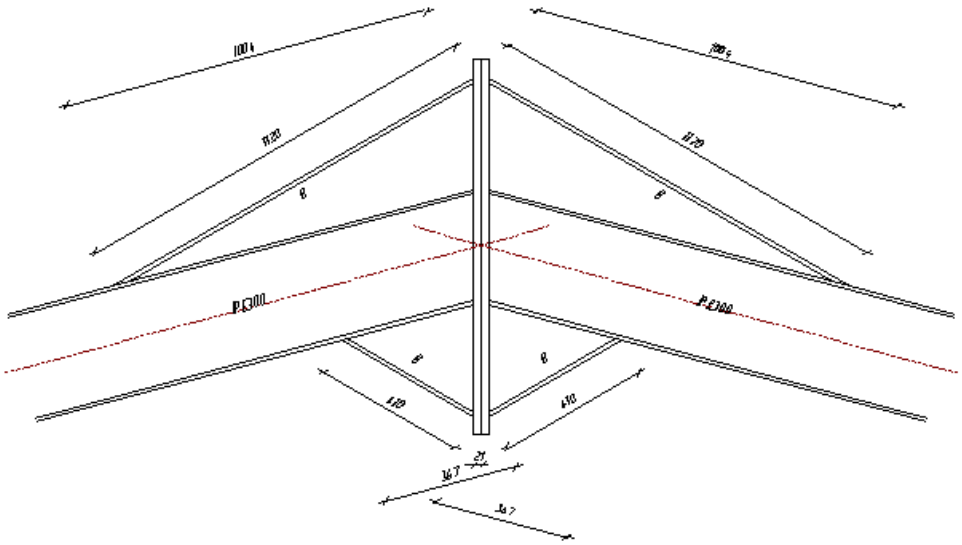


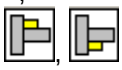
Fig. 7.6 Example of beam to beam connection

Corner of frame connection

Shortcut: 

Edges of rigidity: 0 – edge is absent 

1- triangular edge 

2 – rectangular edge 

Connection type: welded, bolted.
Number of bolts is unlimited, but distance between them is constant.

Position of beams: T-shaped  , angular 

Strengthening elements: 

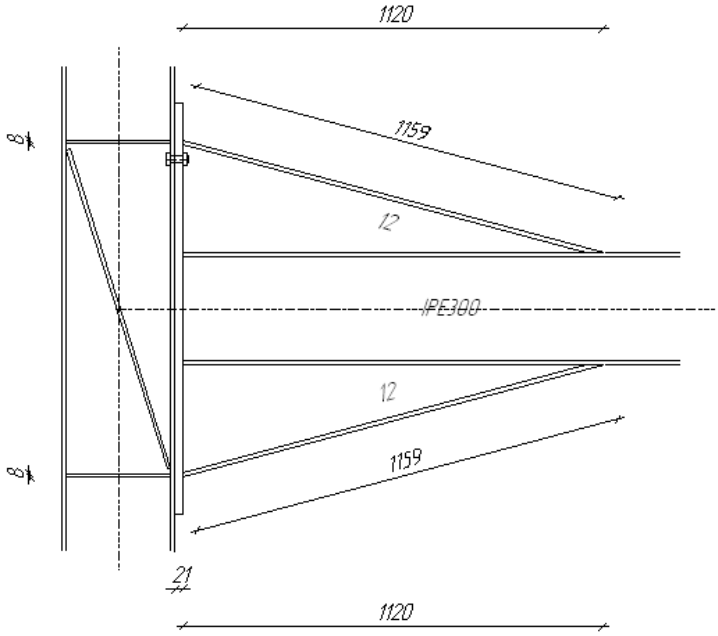


Fig. 7.7 Example of Corner of frame connection

Column to beam connection

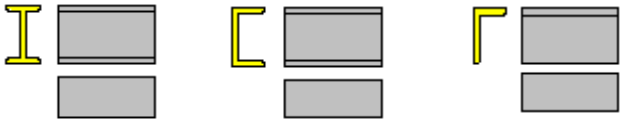
Shortcut: 

See the previous type of connection.

Beam to column connection (angled)

Shortcut: 

Beam profiles: 



Position of beam 1 - constant.
Position of beam 2 - 8 positions for L-bar and channel and 4 for I-beam.
Connection: welded, bolted.
Number of bolts is unlimited, but with sizes - 5 in lines and columns.

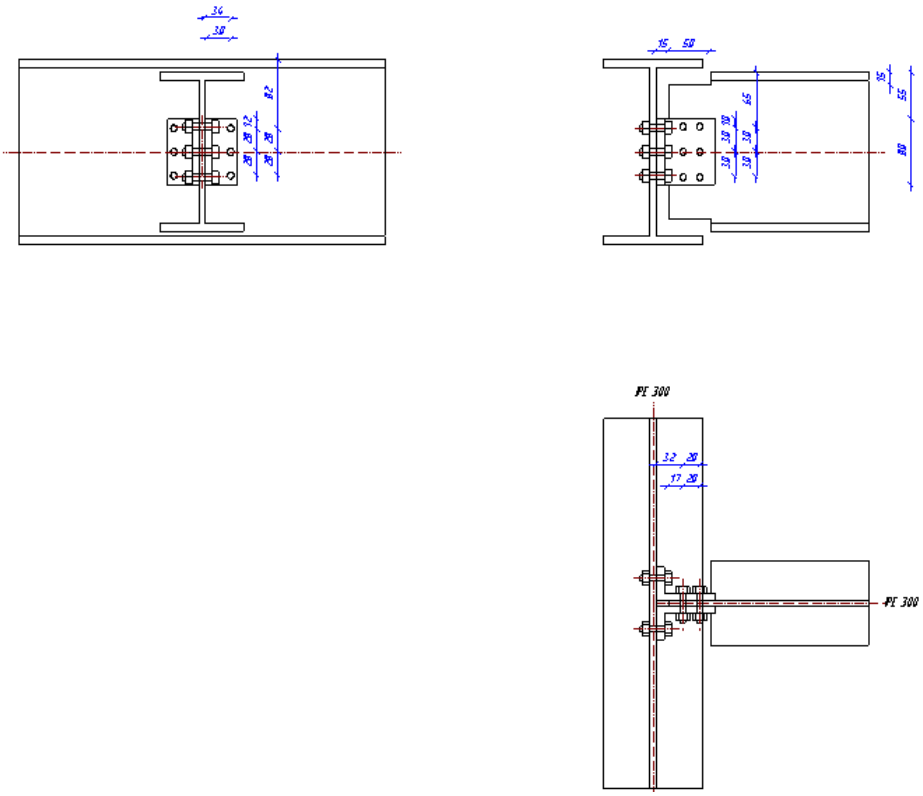


Fig. 7.8 Example of Beam to column connection (angled)

Beam to beam connection (angled)

Shortcut: 

See the previous type of connection.

Beam to column connection (flange)

Shortcut: 

See the previous type of connection.

Beam to beam connection (flange)

Shortcut: 

See the previous type of connection.

Plate-rod connection

Shortcut: 

Truncation of plate: truncation of one corner  , truncation of corners up to beam 

Type of plate corners: not truncated  truncated  strongly truncated 

Profile position on plate: fastening by wall  , fastening by flange 
Beam to plate connection: welded, bolted.
Plate connection: welded, bolted.
Number of bolts at external connection of plate is unlimited. Distance between bolts is constant.
Maximum number of bolts at connection of beams and plate = 5. Distance between bolts is set.

All beams can join with plate from one or two sides.

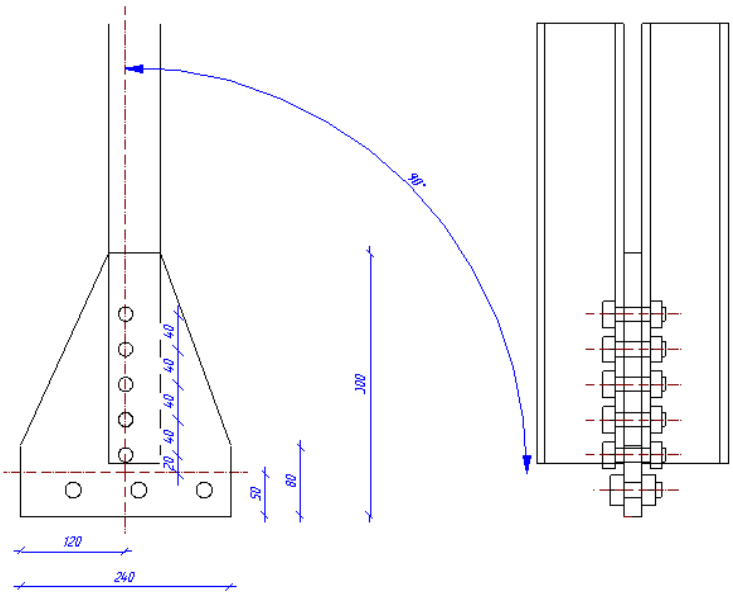


Fig. 7.9 Example of plate-rod connection

Plate-internal node connection

Shortcut: 

Plate parameters
Plate type: tetrahedral.

Type of plate corners: not truncated  truncated  strongly truncated 

Beams parameters
Beam profiles: L-bar, channel.

Profile position on plate: fastening by wall 

fastening by flange 

Number of beams: 0 - 4. 

Type of beams1,3 и 2,4: all beams separately  ,

beams 1 and 3 are jointed 

beams 2 и 4 are jointed 

At joint of beams 1 and 3 profile type, position and connection type are set on the first beam.
At joint of beams 2 and 4 profile type, position and connection type are set on the second beam.
All beams can join with plate from one or two sides.

Parameters of connection
Beams to plate connection type: welded, bolted.
Maximal number of bolts - 5. Distance between bolts is varied.

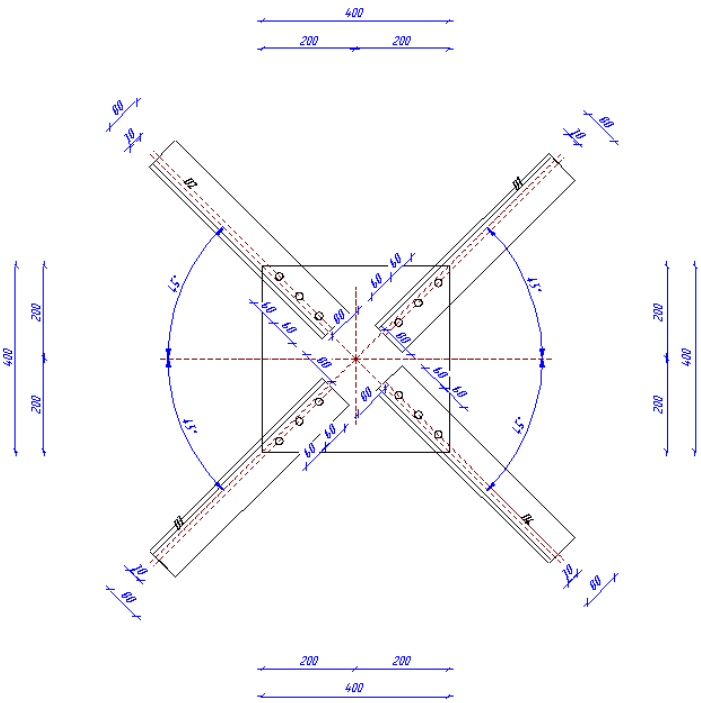


Fig. 7.10 Example of plate-internal node connection

Plate-zone node connection

Shortcut: 

Plate parameters

Plate type: tetrahedral  , hexahedral  .

Type of plate corner: not truncated  truncated  strongly truncated 

Beams parameters
Beam profiles: L-bar, channel.

Profile position on plate: fastening by wall 

fastening by flange 

Type of beams1, 2: all beams separately  , beams are jointed (horizontal)  .

Number of beams: 0 - 5. 

At joint of beams 1 and 2 profile type, position and connection type are set on the first beam.

Parameters of connection
Beams to plate connection type: welded, bolted.
Maximum number of bolts - 5. Distance between bolts is varied.
At joint of beams 1 and 2 bolts can be aligned concerning the rotation center.
All beams can join with plate from one or two sides.

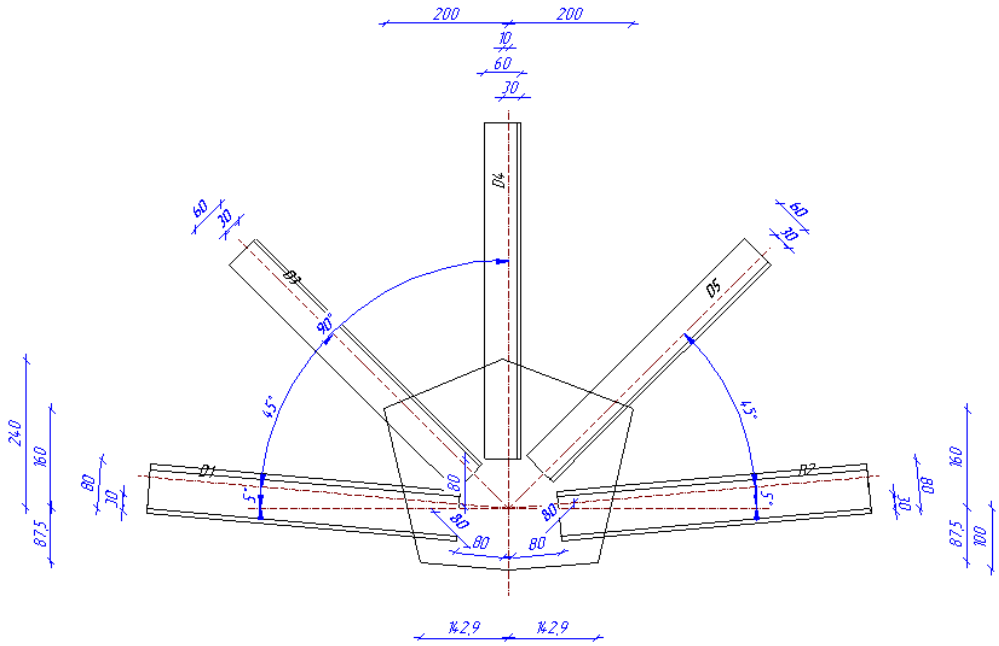


Fig. 7.11 Example of plate-zone node connection

Chapter 8. Specialized Editors

Part 1. Function Editor

Function editor APM_FNED is intended to enter and edit functions. The editor allows to use both graphics and exact function specification. The editor includes parser for entering analytical expressions describing the functions.

Function editor interface

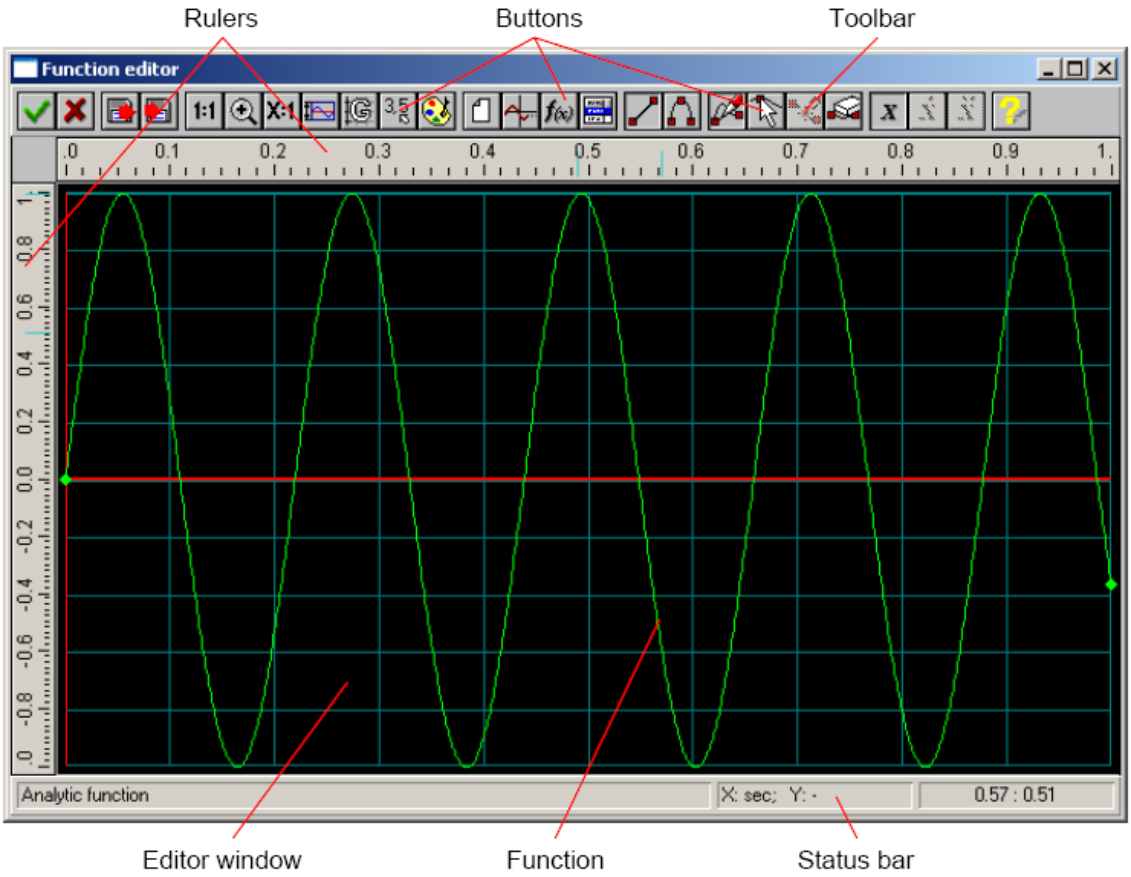


Fig. 8.1 Function editor external interface

To manage the editor, use the toolbar buttons.

Fig. 8.2 Function editor toolbar

The status bar is the bar in the lower part of the program window where various prompt information, current measurement units and cursor coordinates are displayed.

Fig. 8.3 Status bar

The user adjustments are present in function editor for convenience of work. The commands **Scale**, **Limits**, **Grid**, **Cursor step**, **Palette** are used for adjustments changing. Thus, it is necessary to note, that adjustments "by default" is convenient for creation of the most widespread functions.

Toolbar command reference

OK

Use this button to end function entering/editing, save changes and return to calling program.

Cancel

Use this button to cancel function editor without saving changes.

Load Data File

To numerical data loading, it is convenient to use this command, which invokes the dialog window with support of following formats shown below. Formats (*.prn) and (*.csv) can be created in the tabulared editor, for example MS Excel.

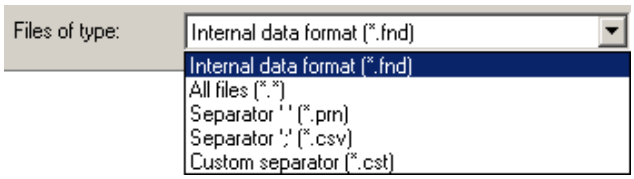


Fig. 8.4 File types that can be opened

Save Data File

To save the graph of function, it is necessary to use this command. The system allows to save data in following formats.

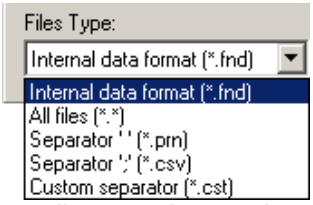


Fig. 8.5 File types that can be saved

Full Scale

Use this button to set scale 1 : 1.

Zoom In Window

Use this button to enlarge selected rectangle to the whole window. Press mouse left button and drag mouse to specify enlargement rectangle. Use right mouse button to cancel operation.

Scale

Use this button to enter specific scale. In response to this button, the *Scale* dialog box is displayed.

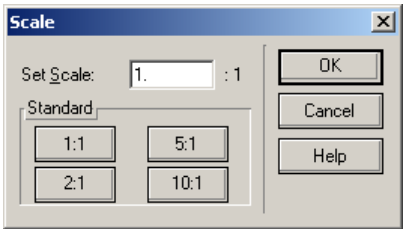


Fig. 8.6 Scale dialog box

Use *Set Scale* edit box to enter vertical scale. The buttons of *Standard* group allows you to set one of four often used scales. Note that can change the vertical scale for displacement graph only.

Limits

Use this button to enter graph limits. In invokes *Function Limits* dialog box.

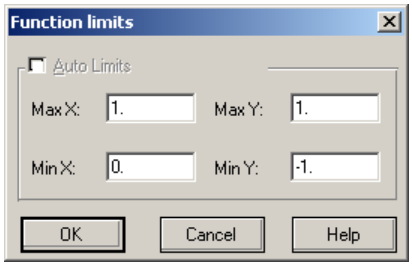


Fig. 8.7 Function limits dialog box

Grid

This button invokes *Grid Parameters* dialog box. This command allows you to change auxiliary grid settings.

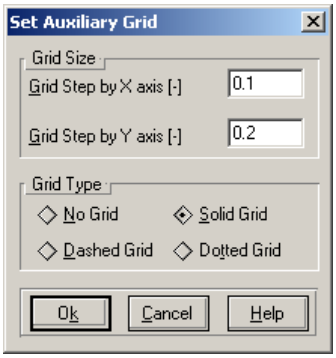


Fig. 8.8 Grid parameters dialog box

Cursor Step

This button calls *Cursor Step* dialog box. This command allows you to set cursor linear step (sensitive zone).

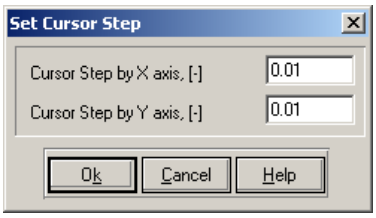


Fig. 8.9 Cursor step dialog box

Palette

This button invokes *Palette* dialog box. This command allows you to change colors of function editor elements.

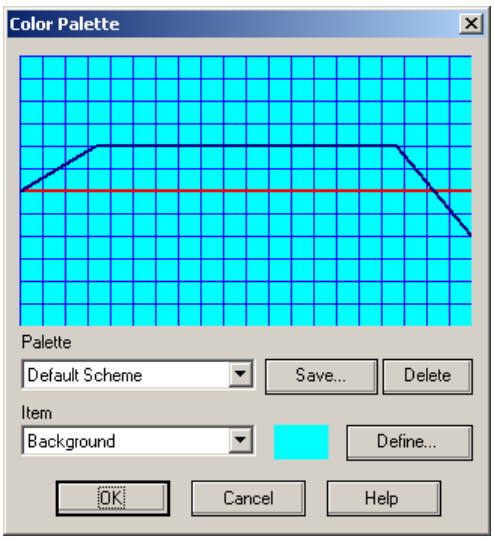


Fig. 8.10 Color palette dialog box

New Function

This button deletes existing graph and starts new one.

Lengthen Function

Using this button, you can lengthen function up to the right limit. The lengthening is performed by addition of horizontal line to existing graph.

Analytical Function

This button invokes *Analytic Function* dialog box.

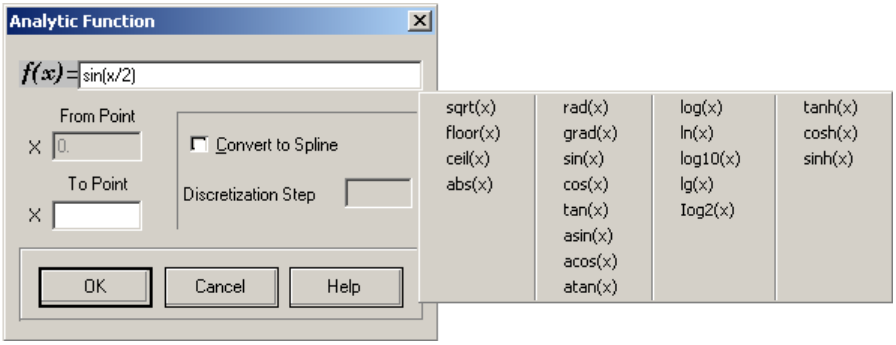


Fig. 8.11 Analytic Function dialog box

In this dialog box, you can enter parameters of analytical function. You can add analytical as graphics objects to the displacement graph.

Use *Formula* edit box to enter expression describing analytical function.

Use *To Point* edit box to enter abscissa of starting point. Ordinate is determined using function expression.

Use *From point* edit box to enter abscissa of ending point. Ordinate is determined using function expression.

If the *Convert to Spline* checkbox is checked the analytical function will be converted to spline with given discretization step. In this case, the restrictions acting in the case of analytical function will be removed, but the accuracy decreases.

Using *Discretization Step* edit box you can enter the step used to convert analytical function to spline. Since the restriction are imposed on the number of spline points, when using too small step you get the corresponding warning message.

Table

This button displays dialog box *Specify Function by Table*.

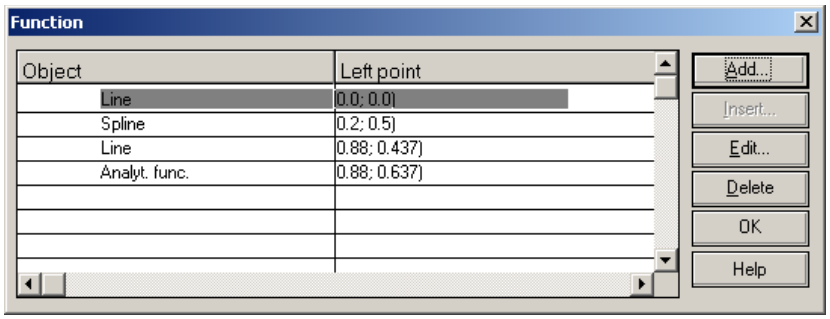


Fig. 8.12 Function dialog box

In this dialog box the objects are listed including their boundary coordinates.

Use **Add** button to select type of object to be added - line, spline, analytical function.

Use **Insert** button to select type of object to be inserted - line, spline, analytical function. The object is inserted before current one (highlighted by selection bar). Analytical function cannot be inserted. The object cannot be inserted if analytical function is specified.

Use **Edit** button to change the current object. Some restrictions are imposed to object editing if analytical function is specified.

Use **Delete** button to delete the current object. Some restrictions are imposed to object deletion if analytical function is specified.

If you select the object of 'line' type (when you add or insert the object), the *Line* dialog box is displayed.

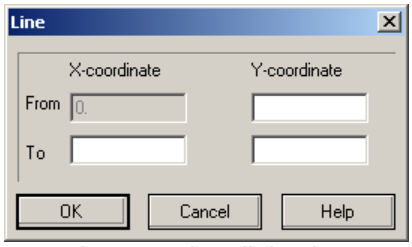


Fig. 8.13 Line dialog box

You can enter line parameters using this dialog box.

In some modes (adding, insertion, editing) some of the edit boxes become unavailable.

If you select the object of 'spline' type (when you add or insert the object), the *Spline* dialog box is displayed.

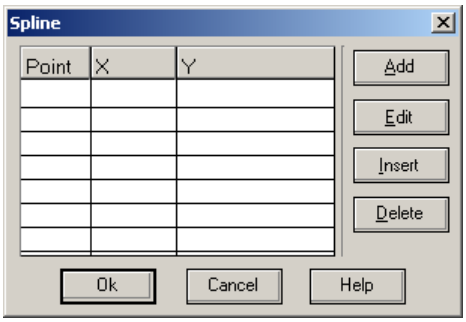


Fig. 8.14 Spline dialog box

The points used to construct spline are listed in this dialog box.

Use **Add** button to add new point to the spline.

Use **Edit** button to change current point of the spline.

Use **Insert** button to insert new point. The point is inserted before current one (highlighted by selection bar).

Use **Delete** button to delete current point of the spline.

Use **OK** button to confirm spline editing. The spline is checked for the presence of at least four points, as well as for agreement between spline end points and boundaries.

If you select the object of type 'analytical function' the *Analytical Function* dialog box is displayed (see above).

Line

This button switches the program to line drawing.

Spline

This button switches the program to spline drawing.

Add Object

This button switches the program to object adding mode.

Line drawing

To draw the line place cursor to one of it end points, press mouse left button and holding it move cursor to other end point, then release the button. Pressing of right mouse button cancels the drawing operation. If you continue the drawing, the first point will be the end point of the previous object. If the line you are drawing is the first object, the X coordinate is 0, the Y coordinate is determined by cursor position.

Spline drawing

When you draw the spline its first point is set automatically: if you continue the drawing, the first point will be the end point of the previous object, if the spline you are drawing is the first object, the X coordinate is 0, the Y coordinate is determined by cursor position. To enter the second point press mouse left button and holding it move cursor to required point, then release button. As usually mouse right button pressing cancels the operation.

To enter following points of the spline use the same actions as for line drawing. To delete existing spline point move cursor to it and press mouse right button.

Use Ctrl + mouse right button combination to cancel spline drawing.

To draw spline one should enter at least four points. The set of points used to draw spline must not contain the points with coinciding coordinates, otherwise spline drawing is canceled.

To finish spline drawing, press space bar.

Edit Function

To edit the node, located between the objects, move cursor to it and press mouse left button. Holding left button you can move the node. If you press mouse right button at the moment, you cancel the editing. Releasing left button you set new position of the node. Now you can edit the objects, located to the left and to the right of the node. The editing performed just in the same way as the drawing does. Pressing of mouse right button (as well as combination of Ctrl + mouse right button for the spline) returns the object to initial state.

To edit the spline, move cursor to it and press mouse left button. The program will operate in editing mode. The combination Ctrl + mouse right button returns the object to initial state. To finish editing, press space bar.

To edit analytical function, move mouse cursor to it and press mouse left button. The dialog box will be displayed; using it, you can change parameters of analytical function.

Insert Object

To insert new object, move the cursor to the node that corresponds to objects you want to insert new object between. If you press mouse left button, the right object is displaced to distance that is equal to distance between last point of object to right boundary of graph. Now you can enter new object just in the same way as you do it in the drawing mode. When you finish object insertion, the drawing is displaced to the left with leveling of Y coordinate. So editing of displaced object may be required. You cannot insert analytical function. In presence of analytic function, there exist some restrictions on object deletion.

Delete Object

To delete object, move cursor either object or to node, the object located right of. Then press mouse left button to select object to be deleted. In this moment, pressing right button you will cancel deletion. Having release left button you delete the object. After deletion the object located to the right of deleted one moves to the left with Y coordinate leveling off. So editing of displaced object may be required. If analytic function is entered there exist some restrictions on object deletion.

Function

This button switches the program to mode of function entering.

First Derivative

This button switches the program to mode of first derivative entering.

Second Derivative

This button switches the program to mode of second derivative entering.

Help

This button displays help contents.

Part 2. A table editor

Any (for example, heat and other) kinds of loads can be specified with help of a function of several variables defined in form of a table. When you set the load necessary to select *"Table"*, and then click "Edit".

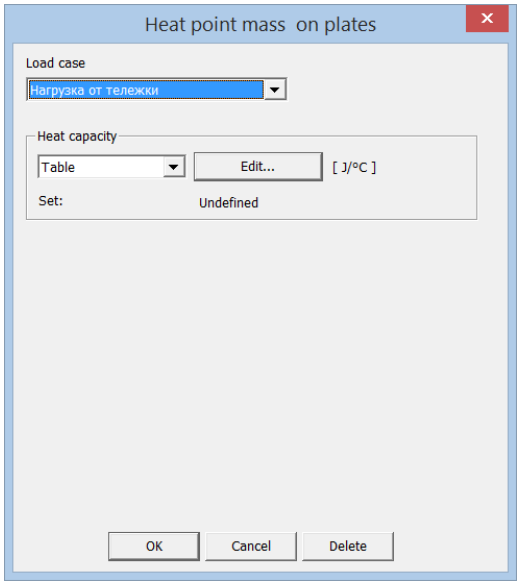


Fig. 8.1 Setting of load with help of a table.

In the dialog window that appeared, one may create a new table or open the existing.

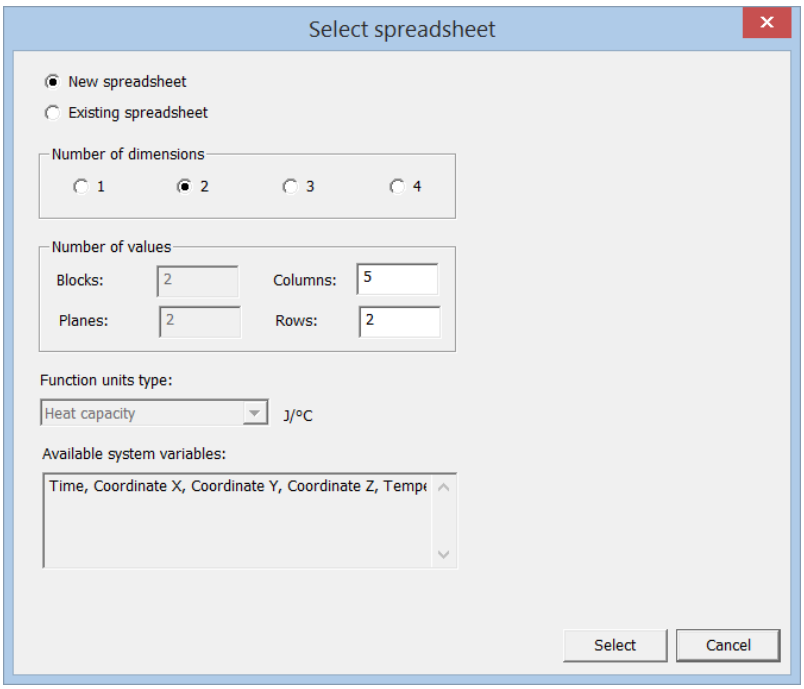


Fig. 8.2 Creation of a new table.

During creation of a table the number of dimensions from 1 to 4 (i.e. the number of independent variables) must be specified and the number of values on each dimension must be specified. A number of values must be larger than two for each dimension. As a result a table with the mentioned parameters will be created. If one dimension is specified, a column with the mentioned number of rows vector-column will be created.

At the choice of two dimensions a rectangular matrix will be created. So the example of a matrix creation with the 2nd rows and 5 columns is given in the Fig. above.

At the number of dimensions a three-dimensional matrix containing in each plane a rectangular matrix of specified sizes is created for 3 measurement.

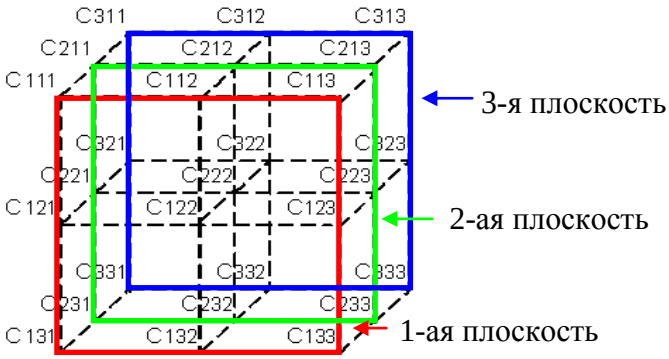


Fig. 8.3 Three-dimensional matrix.

Numbering of each 3D matrix element starts with indication of the plane number, then the row number, and the column number. Four dimensions allow to create the 4th measuring matrix consisting of the blocks each of which contains a three-dimensional matrix.

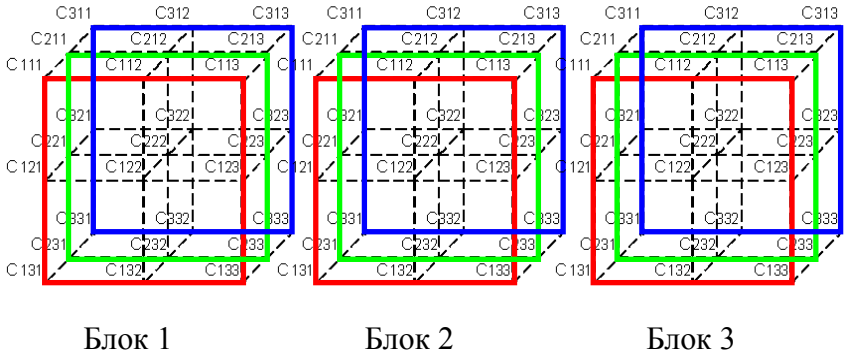


Fig. 8.4 4D-matrix consisting of three blocks

Numberings of four-dimensional matrix elements similar: First the block number, then the plane, the row, and the column are specified. During creation of the table the number of elements in each dimension must be not less than 2.

After creating the table needs to be filled.

Filling in of the table and table editing

In the drop-down list for each measurement (rows, columns, etc.) an independent variable must be selected from a list of available ones. After the choice of an independent variable, units of measurement for each quantity will be displayed in the cells. Further enter values of independent variables (by the rows and columns) to corresponding table cells (they are highlighted with color). In a case of planes and blocks, double click on an element or push the button "Change" and enter its value in a window that appeared.

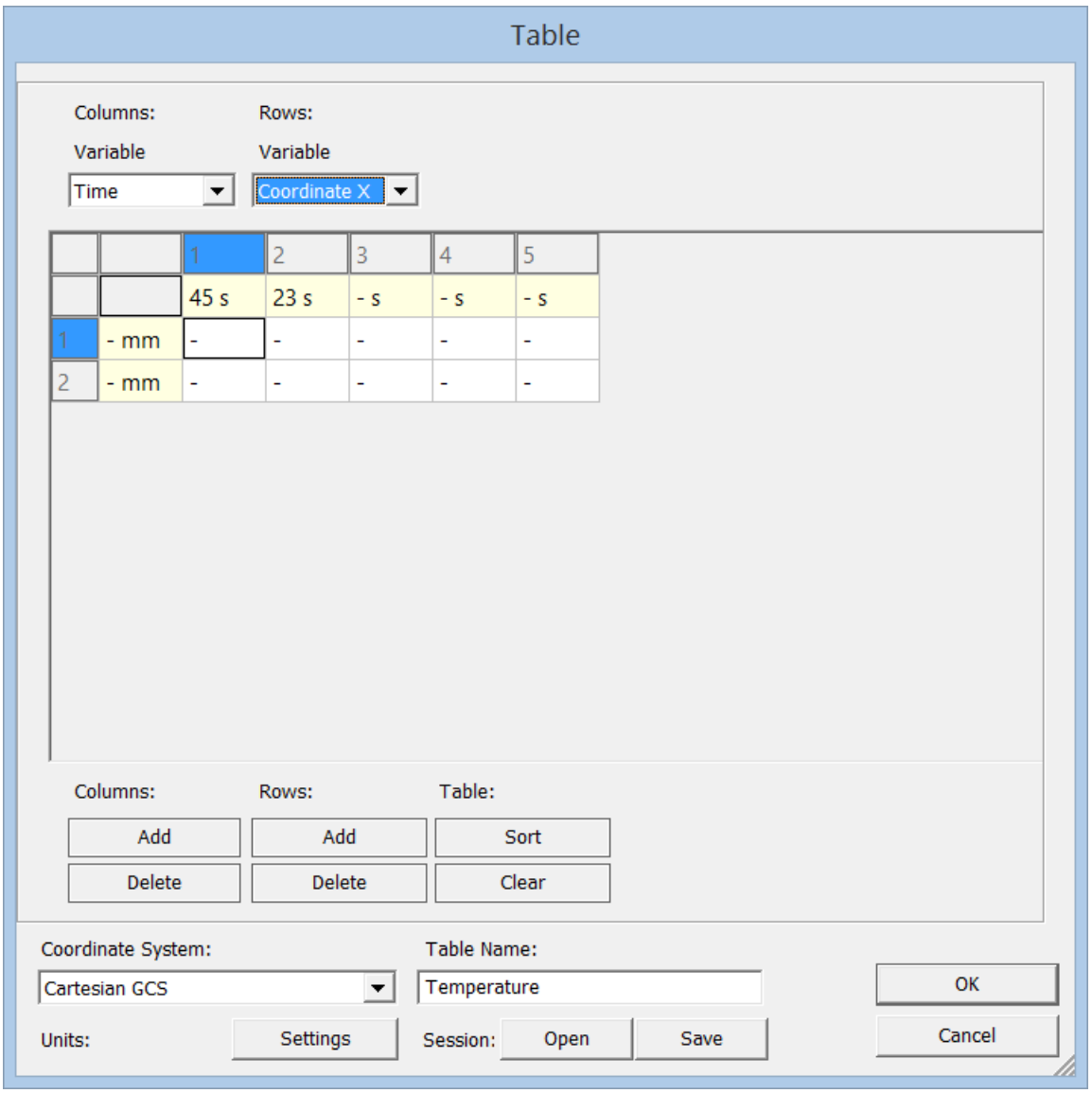


Fig. 8.5 Definition of a multivariate function with help of the table.

After entry of independent variables values should fill values of the function (the white cells in the table). The editable cell is being selected by a frame, as well as the headers of the row and column where the cell is selected.

In a case of need one can change units of variables measurement using the button "Customize". The new dialog where one can select necessary units of measurement for each quantity will be open. Further select option to convert entered data into new units of measurement or leave data without change and change only units.

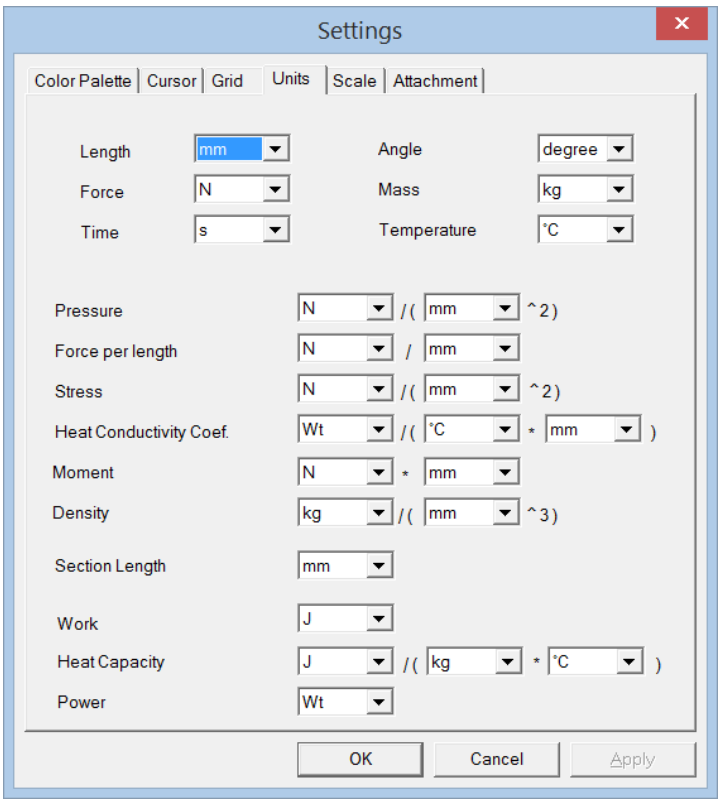


Fig. 8.6 Setting of units of measurement.

A possibility to choose the coordinate system where the variable values are specified is provided for. When changing the coordinate system, if the variable has in its other units, warning is issued, and then the units automatically are replaced.

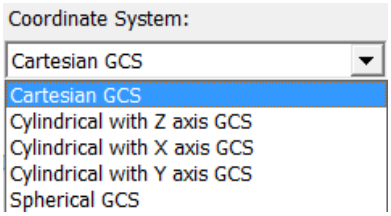


Fig. 8.7 Coordinate systems

The button "Save" allows to save a created table to a file with *.bctbl extension. By default the file name coinciding with the table name will be suggested.

The button "Open a session" and the button "Open" are loading a previously saved table from a file, but make it by the miscellanea.

The button "Open a session" opens an arbitrary table from a file on a disk, i.e. de facto it creates a new table with the same parameters and data as in a saved file.

The button "Open" will load a table from a file only if the number of measurements in a saved table coincides with the number of measurements in the one that is created, in the opposite case a message is given to the screen.

In APM Studio, if the function in the table is specified in a user coordinate system and such a system is absent in a current project, the necessary coordinate system will be created during opening of the table.

In APM Structure3D the user coordinate system is not created during opening of the table, but a global system of coordinates of the same type will be used.

Also the user may not create a new table, but he can use as a template the previously created tables which are available in the tab "Existing table".

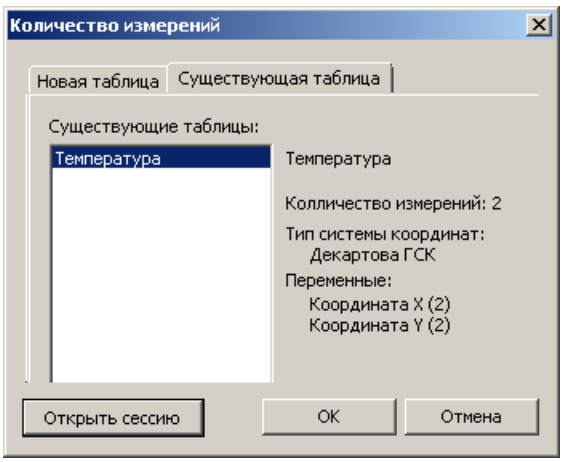


Fig. 8.8 Use of an existing table as a template

Addition and deletion of cells

If in the stage of a table creation the number of rows, columns, etc., is incorrectly specified, one may add or delete the necessary amount of cells in each dimension. For it one must use the available buttons "Add" and "Remove".

For addition of a row or column an arbitrary cell must be selected and the row or the column will be added after a current column or row. If there is no selected column or row, addition occurs to the first position. To remove selection, one must press ESC or click gray cells for removing in the left top corner of the table.

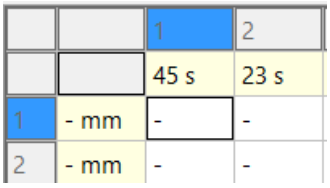


Fig. 8.9 The area of rows and columns deselect

Addition of planes and blocks is occurring in a similar way: The element is added after a dedicated one. For addition of an element to the first place one must remove selection having pressed ESC.

In general, the procedure of elements deletion for each measurement is similar to the addition procedure. A necessary element in the dimension must be selected and the corresponding button "Delete" must be pushed. If it is present in measurement of 2 elements, removal is not possible, a corresponding message about which will appear.

After data entry to the table it can be sorted (the button "Sort"). The table is sorted based on the increase of data in headers of rows, columns, etc.

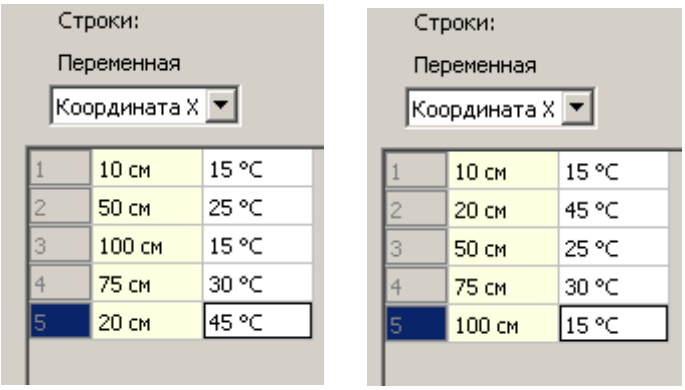


Fig. 8.10 The example of the table sorting

The button "Clean" removes values of a function from the table, leaving values of independent variables.

Part 3. Expression editor

Dialog of an expression definition

The editor of analytical expressions is intended for definition of source data in parameter form. Independent variables (parameters), prevalent mathematical functions and operations, a conditional statement for determination of a function defined by different expressions at different values of an independent variable can be used in the expression.

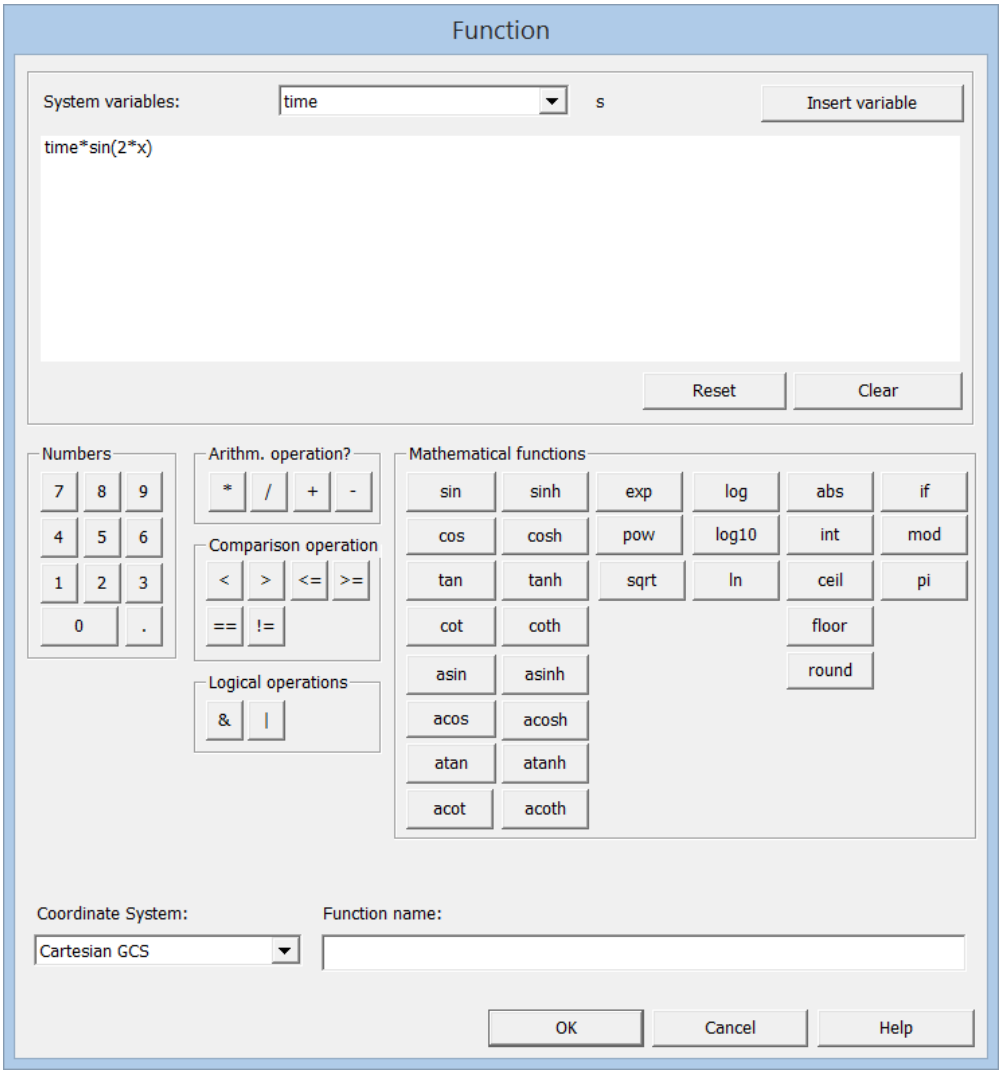


Fig. 8.11 A dialog window of an editor of analytical expressions

1. Coordinates system

If space coordinates of «x», «y», «z» are used in the expression, a possibility to choose the coordinate system where the value of an expression will be counted is provided to the user.

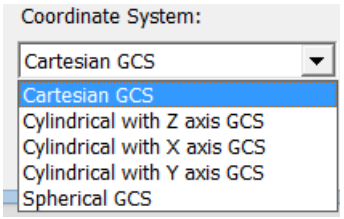


Fig. 8.12 Coordinate system selection

2. Saved objects

Frequently for different loads one and the same expression is used. If these expressions have identical units of measurement and a set of independent variables, the expression can be used again. It is sufficient to select it from a drop-down list. If the user for different loads selected one expression and changed it, the changes will be reflected in other loads using this object "Expression".

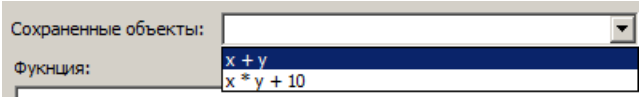


Fig. 8.13 The choice of an expression from a drop-down list

3. Field of an expression definition
4. Units of measurement of an expression result.

For the majority of objects units of measurement results of an expression are fixed and can't be selected. But in a number of other cases perhaps the choice of units of measurement type must be left to the user. Also current units of measurement are shown on the right.

5. List of the independent variables (parameters) available for use that can be used in an expression. Also units of a changed selected variable are shown on the right.
6. Button of a selected independent variable to an expression input field adding.
7. Buttons of prevalent mathematical functions to an expression input field adding.
8. A current change reset button.
9. An expression field clearance button.

Syntax of expressions

Mathematical operators

The mathematical operators in accordance with their priority are presented in the table.

Table 8.2 the Mathematical operators

A notation	The name	Priority
-	An unary minus	1
Of a %	A remainder of division	2
^	Raising to a power	3
*	Arithmetic multiplication	4
/	Arithmetic division	4
+	Arithmetic addition	5
-	Arithmetic subtraction	5
&	Logical multiplication («и»)	6
|	Logical addition («or»)	6
>	More (comparison)	7
<	(Comparison) is smaller	7
>=	(Comparison is) larger or equal	7
<=	(Comparison) is smaller or equal	7
!=	(Comparison) is not equal	7
==	Also (comparison)	7

The unary minus can be used for logical expressions. For it the expression necessary to bracket. An example: «(x > 10 | - (y > 0 & y < 10)».

A conditional operator

The conditional «if» operator which has three parameters can be used in the expression: A condition, an expression in case of satisfaction of a condition, and an expression in case of non-satisfaction of a condition.

The conditional operator has the following syntax: if ("condition", expression, «when a condition is satisfied», «expression, when is the condition not satisfied»).

Priority of a conditional operator is to be read off as grouping with brackets.

Brackets

The parts of an expression can group by brackets «(», «)».

✍ *A remark. Into input field of the expression will be impossibly to use characters «{», «}», «[», «], as they are using for templates in operation with units of measurement.*

Functions

In the expression one can use prevalent mathematical functions introduced in the table.

Table 8.3 Mathematical functions

A notation	Description	A notation	Description
sin(x)	A sine	pow(a, b)	A «a» number to a degree of «b»
cos(x)	A cosine	sqrt(x)	A square root of a number
tg(x)	A tangent	floor(x)	Rounding in smaller direction
ctg(x)	A cotangent	ceil(x)	Rounding in large direction
asin(x)	An arcsine	lg(x)	A vulgar logarithm
acos(x)	An arccosine	ln(x)	A natural logarithm
atan(x)	An arctangent	log(b, a)	A logarithm of a b number to an a base
exp(x)	An exponential of a x number	abs(x)	A modulus of a number

Examples of expressions

1) ctg (x + 10) + sin (if (y < 0, -1, if (y > 10, 10, y * 2)

A cotangent of a sum of an independent variable of a x and 10, as well as a sum of this expression and a sine of piece-wise function are calculated in this expression. The piece-wise function receives value -1 on a segment (-∞, 0), value 10 receives on the segment (10, +∞), the value of y * 2 receives on the segment (0, 10).

2) if (x > 0 & x < 10, 5, exp(x))

This example is distinguished from the previous one by the fact that a compound condition of a conditional operator is used here. The piece-wise function accepts value 5 on a segment (0, 10) and a value of exp(x) at other values of an independent variable of a x.

3) if ((x > -10 & x < 0) | (x > 0 & x < 10) , x, if (x == 0, 100, sin (x)))

In this example the piece-wise function receives the value of a x on segments (-10, 0 and (0, 10), in point 0 receives value 100 and sin(x) – at other x values.

Units of measurement

Customizable measurement units are used in the APM Structure3D system. At change of units of measurement the expression is automatically converted so that it would turn identical with consideration for new measurement units.

For conversion of units of measurement «the conversion coefficient» and the «delta» are used (it is used for temperature).

The expression field at a species template is displayed: «coefficient*» {of "expression"} is «the delta», for a variable a species template is displayed: [«Coefficient*» («variable» + «delta»)].

In a case, when at change of measurement units a coefficient is equal «to 1.0» or the delta – equal «to 0.0», they do not appear in an expression and the template may not be used.

✍ *A remark.*

It is unlikely to remove parts of templates. So the expression outside brackets «{», «}» can't be changed in some way. For change of an expression in this case the button "Clear" must be used.

It is unlikely to change the expression inside brackets in any way «]». For deletion of a variable together with a template all the template together with brackets in this case must be selected and it must erase with "Delete" or "Backspace" keys.

An error check

If the user defined an incorrect expression, he will be notified by a message with description of an error and the cursor in an input field will be placed to the error position.

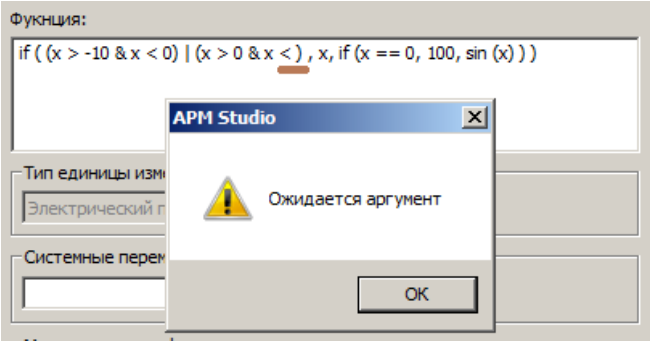


Fig. 8.14 An incorrect argument input error

Chapter 9. Finite Element Analysis

Theoretical basis

The main idea of the method is to break object into areas (finite elements) and to describe object behavior in each area by a set of functions presenting stress and displacements in the given area. Realization of finite element method for strained state of a body is based on displacement calculation method. So the displacements in arbitrary point within finite element are described by a set of some functions - usually polynomials - of point coordinates. Substitution in these functions of nodal coordinates of finite element allows to write down the displacements u(x) of any point of an element through unknown nodal displacements.

$$\vec{u}(x) = \sum_{i=1}^{n} N_i(x)\vec{u}_i \qquad \text{or} \qquad \vec{u}(x) = N(x)\vec{U}$$

where $N_i(x)$ - element shape function, $\vec{u}_i$ - displacement vector of i-th node of element, $N(x)$ - matrix of element shape function, $\vec{U}$ - vector of element nodal displacements.

Let's consider element strain state

Equation $\vec{\sigma} = D\vec{\varepsilon}$ describes relations of stresses σ(x) with strains ε(x) for linear-elastic material, where D – elasticity matrix of Hooke's law.

Strain can be expressed through nodal displacements of an element

$$\vec{\varepsilon} = B\vec{u}$$

Complete potential energy of element is stated as

$$\Pi^{(e)} = 1/2\int_v \vec{\varepsilon}^T D\vec{\varepsilon} dV - \int_v \vec{u}^T \vec{p} dV - \int_s \vec{u}^T \vec{q} dS \quad ,$$

where $\vec{p}$ и $\vec{q}$ – vectors of volume and surface forces accordingly.

Substituting strain vector through nodal displacements

$$\Pi^{(e)} = (1/2\vec{U}^T \int_v (BN)^T DBN dV)\vec{U} - (\int_v \vec{p}^T N dV + \int_s \vec{q}^T N dS)\vec{U}$$

Equation for potential energy can be written down as

$$\Pi^{(e)} = 1/2\vec{U}^T K\vec{U} - f^T\vec{U}$$

where: $K^{(e)} = \int_v (BN)^T DBN dV$ – element stiffness matrix; $f^T = \int_v \vec{p}^T N dV + \int_s \vec{q}^T N dS$ – vector of reduced nodal forces.

Complete potential energy of system is the total energy from all of its elements

$$\Pi = \sum_e \Pi^{(e)} \ ,$$

Minimization of potential energy results in FEA (Finite Element Analysis) equation system

$$KU = F$$

where *K* global stiffness matrix и *F* nodal force vector obtained by summation correspondent stiffness matrix $K^{(e)}$ elements and force vectors *f* of single finite elements.

Finite elements

The following finite elements are supported:

- Beam (tension/compression, bending, torsion)
- Truss (tension/compression)
- Cable
- Pipe;
- Curved pipe;
- 4-noded shell
- 3 - noded shell
- 8 - noded solid (hexahedron)
- 6 - noded solid (prism)
- 5 - noded solid (pyramid)
- 4 - noded solid (tetrahedron)
- 10 - noded solid (tetrahedron)
- 13 - noded solid (pyramid)
- 15 - noded solid (prism)
- 20 - noded solid (hexahedron)

Coordinate systems

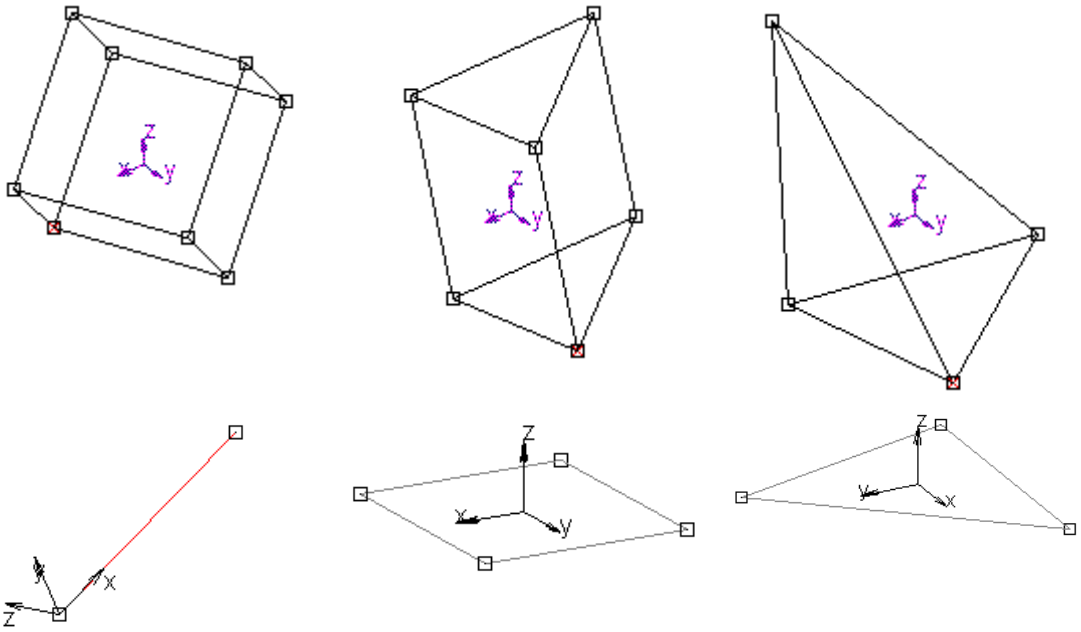
Each structure element has its own local coordinate system. For convenience, all coordinate systems are right-handed. Orientation of local coordinate system for different elements is shown below.

Node coordinate system may have any orientation by its rotation. By default, it coincides with the global coordinate system. The following node attributes are specified in node coordinate system: supports, elastic supports, nodal displacements in direction of fixed degree of freedom.

Rod coordinate system is always oriented so that X axis is directed along its axis. Rod cross-section orientation is strictly attached to its coordinate system. Besides, loads on rod are also specified in rod coordinate system.

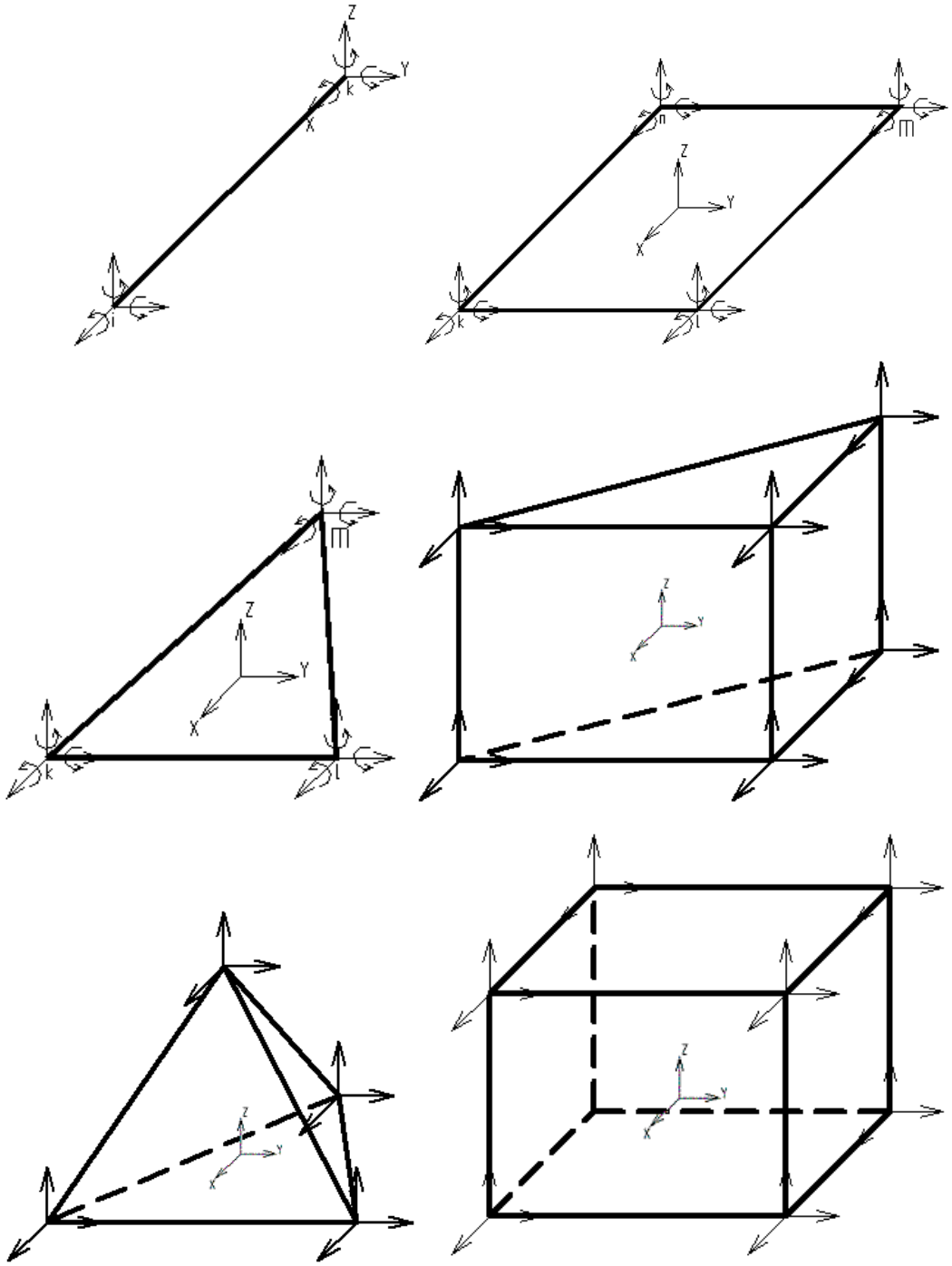
Plate coordinate system is always oriented so that Z axis is normal to the plate surface. X axis is parallel to one of the plate sides and Y axis completes the system to right-handed orientation. Plate coordinate system can be inverted (make opposite normal direction). Plate distributed load is specified in the plate coordinate system.

Solid element coordinate system coincides with global one.



Degrees of freedom. A node has six degrees of freedom. A rod has twelve degrees, a plate has 18 or 24 depending on the number of its nodes. Solid elements have 12, 18 or 24 degrees of freedom, 3

translational displacements in each node. Supports remove degrees of freedom from the system. Degrees of freedom and local coordinate systems for different elements are shown below:



In order to create stiffness, mass and geometric matrices, we need to know shape functions for every element, allowing us to obtain displacements in every point of the element expressed in terms of node displacements. Procedures of obtaining form functions from base polynomials are discussed in books on finite element method. Base polynomials for every element are shown below.

Triangular plate element

For lateral displacements of triangular plate element, we use part of 3rd order polynomial in terms of L-coordinates with 9 unknowns

$$\left[L1 \quad L2 \quad L3 \quad \left(L1\cdot L2^2+\frac{L1\cdot L2\cdot L3}{2}\right) \quad \left(L3\cdot L2^2+\frac{L1\cdot L2\cdot L3}{2}\right) \quad \left(L1\cdot L3^2+\frac{L1\cdot L2\cdot L3}{2}\right) \quad \left(L2\cdot L3^2+\frac{L1\cdot L2\cdot L3}{2}\right) \quad \left(L2\cdot L1^2+\frac{L1\cdot L2\cdot L3}{2}\right) \quad \left(L3\cdot L1^2+\frac{L1\cdot L2\cdot L3}{2}\right)\right]$$

For displacements in plane – 1st order polynomial

$$(1 \quad x \quad y)$$

For lateral displacements of quadrangular plate element – part of 4th order polynomial with 12 unknowns

$$\begin{bmatrix} 1 & x & y & x^2 & x{\cdot}y & y^2 & x^3 & x^2{\cdot}y & x{\cdot}y^2 & y^3 & x^3{\cdot}y & x{\cdot}y^3 \end{bmatrix}$$

For displacements in plane – part of 2nd order polynomial

$$\begin{pmatrix} 1 & x & y & x{\cdot}y \end{pmatrix}$$

For lateral displacements of rod element in one plane – part of 3rd order polynomial

$$\begin{bmatrix} 1 & x & x^2 & x^3 \end{bmatrix}$$

For axial and torsion displacements – 1st order polynomial

$$\begin{pmatrix} 1 & x \end{pmatrix}$$

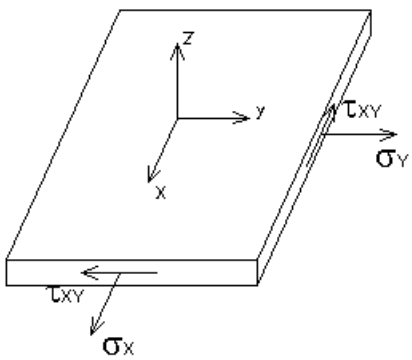
Working with finite elements

It is advisable to follow these rules while modeling using finite elements. The angles between adjacent edges of 4-noded plate element should be closer to 90°. The angles of the 3-noded plate should be closer to 60°. The angles should be never equal or greater than 180° which would otherwise make the element degenerate and the result – indeterminate.

Examples of degenerate finite elements

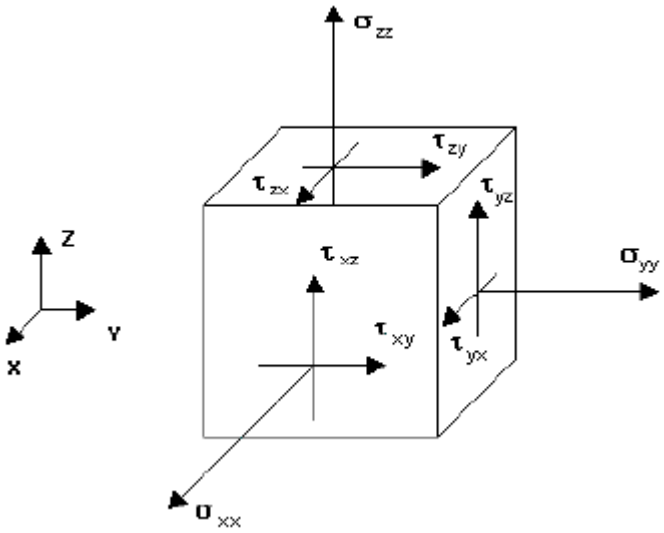
For calculation of formulae for shell elements, thin-plate theory was used, assuming that plate thickness is 5 and several orders lower than linear dimensions of the plate.

Stresses in plate elements



We consider that the line being normal to non-deformed equilateral surface and straight, remains straight after deformation and normal to the deformed surface of the element. All stresses lying in plane of the element are considerably greater than those perpendicular. Stresses have non-uniform distribution along the element's height.

Solid elements



Glossary on Finite Element Method

Bate K., Wilson E. Numerical Methods of Analysis and Finite-Element Method.

Gallager R. Finite-Element Method. The Basics.

Darkov A., Shaposhnikov N. Structural Mechanics.

Zenkevitch O. Finite-Element Method for Technics.

Smirnov A. Analysis of Framed Structures, Plates and Shells With the Help of Computers.

Sinitzin A.. Finite-Element Method for Dynamics of Structures.

Agapov V. Finite-Element Method for Statics, Dynamics and Buckling of Spatial Thin-Walled Supported Structures.

Ivanov B. Solving Structural Units Dynamics and Buckling Problems By Finite-Elements Method.

Hetchumov R. Application of Finite-Elements Method to Structure Calculation.

Chapter 9. ELECTROMAGNETIC FIELDS ANALYSIS (*EMA*)

1. Overview of electromagnetic fields analysis

The tools of electromagnetic fields analysis that are implemented in the module APM Structure3D, are available starting with version 13 and can be used to study various effects of electromagnetism, such as self-induction, magnetic flux density, the distribution of the magnetic field lines, loss of electric power and other related phenomena. These tools effective in the analysis of such devices as solenoids (inductors), magnetic starters, electric motors, sources of constant magnetic fields, transformers, electromagnets, etc. APM Structure3D has the capacity to solve problems of microwave equipment (calculation of waveguides, resonators and antennas).

Three types of electromagnetic analysis are available:

- Three-dimensional stationary electromagnetic fields;
- Low frequency three-dimensional variable electromagnetic fields;
- High frequency three-dimensional electromagnetic fields.

The finite-elemental formulation used in an APM Structure3D modulus of a considered type of analysis is based on Maxwell equations for electromagnetic fields. By putting of scalar or vector potential in these equations and establishment of defining relations, the user can get the equations which are convenient for finite-elemental analysis.

Analysis in the low frequency and high frequency area

Electromagnetic field analysis can be divided depending on speed of change of (induction and briskness) vectors characterizing state of field into low frequency (0 ~ 1000 Hz) and high frequency (about 1 MHz ~ 10 GHz) area.

Problems, inter alia, represent practical interest for the calculations in the low frequency area and stationary – (0 Hz), those related to such electrical devices: Electric motors, electromagnetic drives, transformers, etc.

During solution of tasks in the high frequency area usually wave processes of the proliferation of electromagnetic waves in the space are explored, or the electronic characteristics and SHF – devices of such ones are investigated: antennae, resonators, waveguides, micro-strip transmission lines, etc.

Types of analysis

In the low-frequency electromagnetic fields are the following types of problems that arise, which can be solved with the help of EMA:

1) Electrostatics;
2) Electricity kinetics;
3) Magneto statics;
4) Electromagnetic transients;

Electrostatic calculation (Electrostatics)

The EMA tools used for electric field analysis affect two areas of electric phenomena: Direct current flow (conductors, electrostatics (dielectrics). They are assigned to the typical parameters representing interest: current density, electric field strength, the stress distribution, the thermal effect of current, power and strength of the electric field, electrostatic capacitance, current and voltage drop.

Three-dimensional tasks arising in development of different devices such as accumulation buses, transmission lines, high voltage insulators screening covers, capacitors, etc., can be solved.

Laplace's equation is used as theoretical basis for analysis of stationary electric field in the program. Largely unknown (nodal degrees of freedom), is determined by a finite-element solutions are electric potentials (voltage). The rest of the parameters are calculated over their values.

Electrostatic field analysis is used for a calculation of electric field characteristics and a potential distribution conditioned by a system of electric charges or voltage drop. Two types of loads are allowed: A

difference of potentials and charge density. It is assumed linear analysis is carried out, i.e. the parameters, characterizing electric field, linearly depend on applied voltage.

Solution consists in receipt of quantities of electric potentials in nodes, gives a possibility to find electric field intensity and current density.

Calculation of direct currents field (Electricity kinetics)

The EMA program can be used for finding current density and a distribution of electric potentials (a voltage) arising in electric circuits in a course of direct current or at the expense of voltage drop. Two types of loads are considered as input parameters: Current and voltage. Analysis is assumed linear, i.e. magnitude of electric current in separate sections of a circuit is proportional to input current.

The task of direct electric current flow is solved with use of a potential function and reduces to calculation of electric potentials (current or voltage density) in model nodes.

Static electromagnetic fields (Magneto statics)

Analysis of static electromagnetic field is possible for three-dimensional tasks in a linear statement. The three-dimensional task of magneto statics is a result of minimization of a magnetic energy functional associated with a three-dimensional potential vector. There is a possibility to model conductors and permanent magnets in form of sources.

The conductors are being modeled by finite elements or with help of solid-state primitives in form of a straight or circular rod and coil rings. The user has a possibility to model iron cores and non-magnetic materials (air).

EMA program provides the user linear magnetic substances, including the values of the magnetic permeability of isotropic and orthotropic materials. .When post processing of the results it is possible to get a picture of the vector potential, magnetic flux density and magnetic field strength.

Variable stationary electromagnetic low frequency field

Electromagnetic analysis can be done for tasks in three-dimensional statement. In the analysis of non established transient potential vectors are computed, as the induction of the magnetic field flux density and intensity of the electromagnetic field.

For a solution of these equations an implicit scheme of integration by the time of Krank-Nikolson is used. The scheme of Krank- Nikolson integration is the discrete procedure with help of which the vector of field potentials is being computed in separate points of a time interval.

Analysis is carried out for electromagnetic high frequency field on the basis of a complete system of Maxwell equations i.e. with consideration for the circulation of electromagnetic waves. Such a kind of analysis is required in those cases when wavelength is comparable to determining device gauges.

It is important to understand for high frequency electromagnetic field the «edge» finite elements whose discrete connections are carried out not through the nodes, but through the edges with which the degrees of freedom are associated – projections of electric field strength vector to an edge are applying.

For high frequency electromagnetic field modal analysis is available.

Modal analysis

Modal analysis is used for determination of the natural frequencies and forms of fluctuations for hollow resonators. Analysis must precede any dynamic calculation of a resonator because knowledge of fundamental modes of oscillations and oscillation frequencies gives a possibility to accordingly characterize transients in a system.

To solve the task of eigen values, Lanczos method is used. Modal analysis can be used for determination of a system resonant properties, inter alia, with consideration for dielectric and surface loss. At the same time the losses are assumed small, not even having effect on the natural frequencies of a system.

Analysis in the low frequency area

An electrostatic calculation

During solution of an electrostatic task it is believed all the objects are stationary (not being changed in time) and there is no current in conductors (in other words, the conductors are found in electrostatic equilibrium state). All conductors are considered ideal and equi-potential, therefore, inside conductors electric field is absent – the conductors can't be calculated in this form of a calculation, a calculation, of the electrostatic task to be posted only for dielectrics. The unknown calculation value is electric scalar potential through which calculated strength and induction (offset) of the electric field are being computed.

Calculating model development

For solution of electrostatics tasks the user needs to define geometry of a calculated area, using three-dimensional finite elements of the first order – four-node (tetrahedrons), six-node (triangular prisms), or eight-node (hexahedrons). It is required for a calculation passing that the model did not contain any other finite elements and was associated, in other words, It was a single entity and not held separately spaced nodes.

All the materials that are used in a model must have a property – relative (relative to an electric constant ε_0=8, 854187817·10^{-12} [F]/[m]) dielectric permeability. Table below means of determination of this property available for a user are presented.

Table 10.1 Relative dielectric permeability

Anisotropy	Units of Measurements	Ways of definition	Available independent variables
An isotropic one, An orthotropic one	[]	Constant value, Graph. Table, Function	*X* coordinate Y coordinate Z coordinate

At setting of an orthotropic property one is to consider the direction of axes of orientation is coinciding with the local coordinate of the three-dimensional element for which this material is assigned.

Definition of loads and boundary conditions

For modeling of electrostatics tasks the following types of loads and boundary conditions are available to a user:

1) Electric charge;

2) Electric charge density;

3) Electric potential.

Information about these objects is presented in table 10.2

Table 10.2 Electrostatic loads

Name	Type	Units of Measurements	Application objects	Ways of definition	Available independent variables
Electric charge	Scalar one	[Cl]	Nodes	Constant value, Graph. Table, Function	*X* coordinate Y coordinate Z coordinate
Electric	Scalar one	[Cl]/[*length**] ^{3}	Three-	Constant value,	*X* coordinate

charge density			dimensional elements	Graph. Table, Function	*Y* coordinate Z coordinate
Electric potential	Scalar one	[B]	Nodes, Three-dimensional elements	Constant value, Graph. Table, Function	*X* coordinate *Y* coordinate Z coordinate

Length* is determined by a value selected in the drop-down list. The ***length* of the dialog window ***'Settings'*** in tab ***"Units"***.

Electric charge

Electric charge is a load for the tasks of electrostatics. At setting to a nodes group of a constant value a given value will be assigned to each node. At setting to a node group of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned to each node.

If one **Loading** includes several loads of this type, which contain the same node, it will be taken into account in the calculation for the node, the sum of all loads values of this type of **Loading,** which includes the node.

Electric charge density

Electric charge is a load for the tasks of electrostatics. At setting to a group of three-dimensional elements of a constant value a certain* value will be assigned to each node of all selected three-dimensional elements. At setting to a group of three-dimensional elements of a variable value (graph, table, function) a certain* value defined by coordinates of a node will be assigned to each node of all selected three-dimensional elements.

If one **Loading** includes several loads of this type, which contain the same three-dimensional element, it will be taken into account in the calculation for a given three-dimensional element, the sum of the values of all loads of this Loading type, which includes the three-dimensional element.

**A certain value is defined as an integral weight of a node for specified density.*

Electric potential

Electric potential is the boundary condition for tasks of electrostatics. At setting to a nodes group of a constant value, a given value will be assigned to each node. At setting to a node group of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned to each node.

At setting to a group of three-dimensional elements of a constant value a given value will be assigned to each node of all selected three-dimensional elements. At setting to a group of three-dimensional elements of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned into each node of all selected three-dimensional elements.

If one **"Loading"** includes several loads of this type, which contains one and the same node (three-dimensional element), then the calculation will be considered for this node (three-dimensional element) value of the last created this type.

All the described types of loads can be added to a document by the choice of a corresponding shortcut menu item for node **Electric loads** on the **Objects** panel (fig. 4.64).

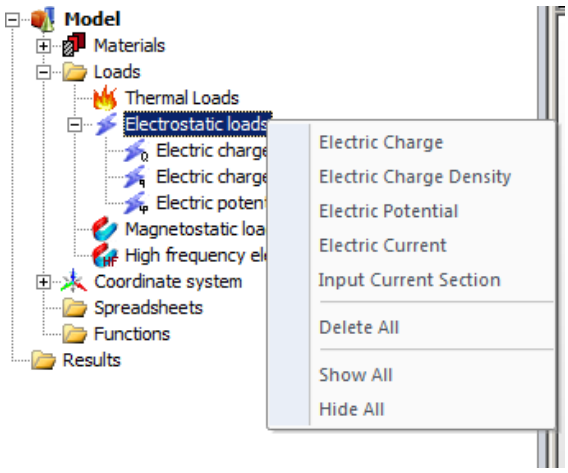


Fig. 10.1 Electrostatic loads

Performing calculation

To carry out **the Electrostatic calculation** is necessary to select in the horizontal menu **Calculate**,... and then a **dialog window Calculation** will appear.

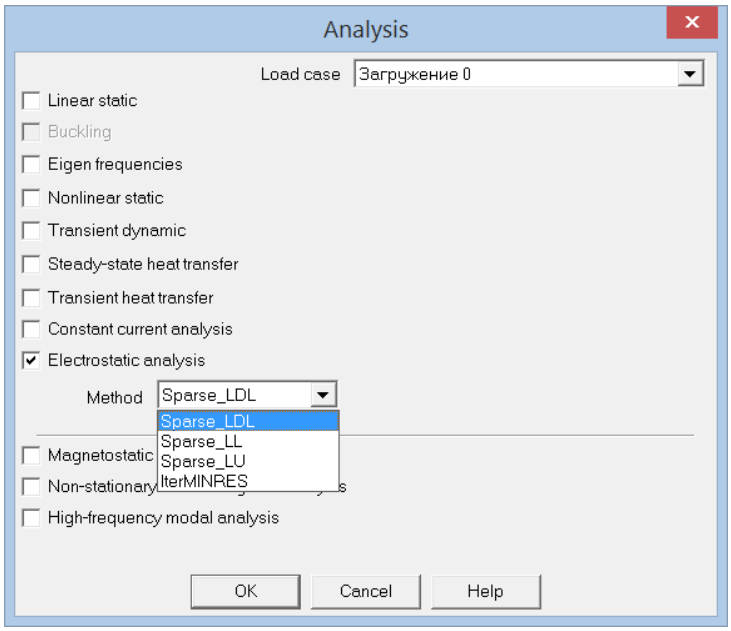


Fig. 10.2 Start of an electrostatic calculation

In the dialog box **Calculation** the user must mark the **Electrostatic calculation** section, in the drop-down list for loading must select **Loading,** for which user wants making a calculation, to select in the drop-down list Method by which the system of linear algebraic equations for a posed discrete task will calculate. After a press of an **OK** button the **Electrostatic calculation** will be made.

If the user in the **Loading** for the calculation of which is held, did not give any type of Electrical load capacity, which includes at least one node or three-dimensional element, than he will receive the ambiguous decision (with precision up to a constant term). In addition, the matrix of the linear algebraic equations system for the posed discrete task soonest of all, will be ill-conditioned, which can lead to impossibility of the task solution.

Selecting a method for solving a system of linear algebraic equations for a posed discrete task must base on the following positions:

1) Sparse_LL method is intended for systems of linear algebraic equations with a symmetric positively definite matrix, based on expansion of Kholetskii.

2) Sparse_LDL method is intended for systems of linear algebraic equations with a symmetric positively indefinite matrix, based on LDL decomposition.

3) Sparse_LU method is intended for systems of linear algebraic equations with an asymmetric matrix, based on LU decomposition.

4) IterMINRES method is the iteration method intended for tasks of linear algebraic equations systems with a symmetric positively indefinite matrix, in particular, with the degenerate one.

In most cases all the electrostatics tasks are reduced to systems of linear algebraic equations by a positively definite matrix. Therefore Sparse_LL is usually more preferable, but in a number of cases (the sufficiently small coefficients are on the main diagonal of a matrix conditioned by a crummy finite-elemental mesh or material properties), the matrix may be positively uncertain. At the same time the Sparse_LL method may not solve the task (a message about which the user will receive), then one can choose the Sparse_LDL or Sparse_LU methods which can solve that task. The key feature of Sparse_LL, Sparse_LDL, and Sparse_LU methods is that for their work (implementation of decomposition) the specified amount of random access memory which is increasing with dimensions of a task that is solved is necessary. In addition, «greediness» towards random access method memory is increasing in the following sequence (Sparse_LL–>Sparse_LDL–>Sparse_LU). Therefore, such cases are possible, when for the problem is not enough RAM (a message about which the user will receive), this case, it is advisable to choose a method IterMINRES, which is much less demanding on the size of the RAM, due to the iterative nature it is slower.

View of results

After an electrostatic calculation in the bar **Objects** in the **Results** node new menu item **Electrostatic calculation** will appear.

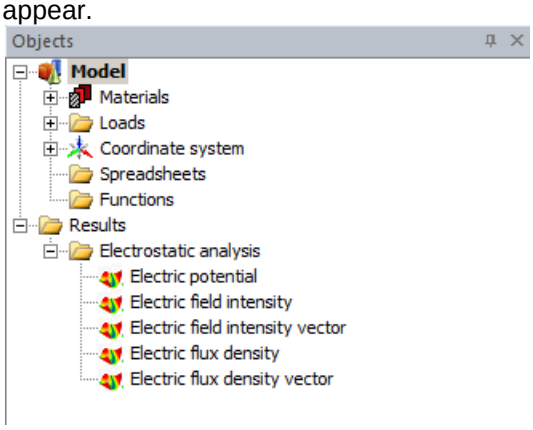


Fig. 10.3 Electrostatic calculation results

The user is available for viewing the results of five types of cards:

1) Electric potential of [V] (contoured);

2) Electric field strength [V] / [length *] (contoured);

3) Vectoral strength of electric field [V] / [length *] (vectorial);

4) Electric induction [Kl]/^{2} [length*] (contoured);

5) Vector electric induction [Kl]/^{2} [length*] (vectorial).

Length* is determined by a value selected in the drop-down list. The ***length* of the dialog window ***'Settings'*** in tab ***"Units"***.

For details work with Cards of results see in the section "Results" in the user manual "Work with a Project Tree".

Calculation of direct currents field

During resolution of a calculation task of direct currents field it is considered that all the objects are stationary (not changed in time), the currents in conductors are compensated (in other words, the sum of all external currents is equal to zero). The dielectrics can't be calculated in this form of such a calculation. A calculation of direct currents field hear is for conductors only. The unknown calculation value is electric

scalar potential through which then strength of electric field and electric current density are being computed.

Calculating model development

For solution of a calculation of field of direct currents task the user needs to define geometry of a calculated area, using three-dimensional finite elements of the first order – four-node (tetrahedrons), six-node (triangular prisms), or eight-node (hexahedrons). It is required for a calculation passing that the model did not contain any other finite elements and was associated, in other words, It was a single entity and not held separately spaced nodes.

All the materials that are used in the model must have a property – specific electric conductivity. In Table 4.2.3 means of this property available for a user are presented.

Table 10.3 is Specific electric conductivity

Anisotropy	Units of Measurements	Ways of definition	Available independent variables
An isotropic one, An orthotropic one	1/([Ohm] · [*length**])	Constant value, Graph. Table, Function	*X* coordinate Y coordinate Z coordinate

Length* is determined by a value selected in the drop-down list. The ***length* of the dialog window ***'Settings'*** in tab ***"Units"***.

At setting of an orthotropic property one is to consider the direction of axes of orientation is coinciding with the local coordinate of the three-dimensional element for which this material is assigned.

Definition of loads and boundary conditions

For tasks modeling the calculation of direct current field the following types of loads and boundary conditions are available to a user:

1) Electric current;

2) Electric potential.

Information about these objects is presented in table 10. 4

Table 10.4 Loads of direct current field

Name	Type	Units of Measurements	Application objects	Ways of definition	Available independent variables
Electric current	Scalar one	[Cl]	Nodes	Constant value	-
Electric potential	Scalar one	[B]	Nodes, Three-dimensional elements	Constant value, Graph. Table, Function	*X* coordinate *Y* coordinate Z coordinate

Electric current

Load for the field of direct currents tasks is **electric current**. At setting to a nodes group of a constant value, a given value will be assigned to each node. But this load is essentially points' and it is recommended to set it to one node..

If one **Loading** includes several loads of this type, which contain the same node, it will be taken into account in the calculation for the node, the sum of all loads values of this type of **Loading,** which includes the node.

Electric potential

Electric potential is the boundary condition for the tasks of direct currents field. At setting to a nodes group of a constant value, a given value will be assigned to each node. At setting to a node group of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned to each node.

At setting to a group of three-dimensional elements of a constant value a given value will be assigned to each node of all selected three-dimensional elements. At setting to a group of three-dimensional elements of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned into each node of all selected three-dimensional elements.

If one **"Loading"** includes several loads of this type, which contains one and the same node (three-dimensional element), then the calculation will be considered for this node (three-dimensional element) value of the last created this type.

All the described types of loads can be added to a document by the selection of a corresponding shortcut menu item for the node Electric load on the panel **Objects** (Fig. 4.66).

Performing calculation

To carry out the Electrostatic calculation is necessary to select in the horizontal menu **Calculate**, and then a dialog window **Calculation** will appear.

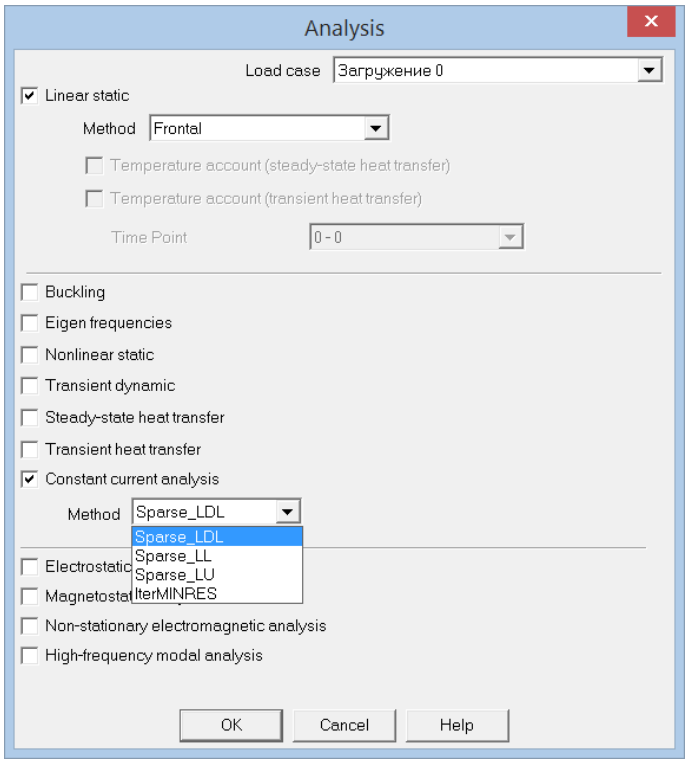


Fig. 10.4 Start of a direct currents calculation

In the dialog window Calculation the user must mark the item **Calculation of direct currents**, in the drop-down list for **loading** must select **Loading**, for which a user wants making a calculation, to select in the drop-down list Method the method by which the system of linear algebraic equations for a set discrete task will count. After a press of an **OK** button the **Calculation of direct currents** will be made.

If a user in the **loading**, for which calculation is performing, did not give any type of **electrical load capacity**, where at least one node or three-dimensional element is included, the result will be ambiguous (with accuracy to a constant term). In addition, the matrix of the linear algebraic equations system for the posed discrete task soonest of all, will be ill-conditioned, which can lead to impossibility of the task solution.

About selection of a solution method for the linear algebraic equations system for a posed discrete task see point 2.1.3.

View of results

After an electrostatic calculation, in the bar **Objects** in the **Results** node new menu item **Electrostatic calculation** will appear.

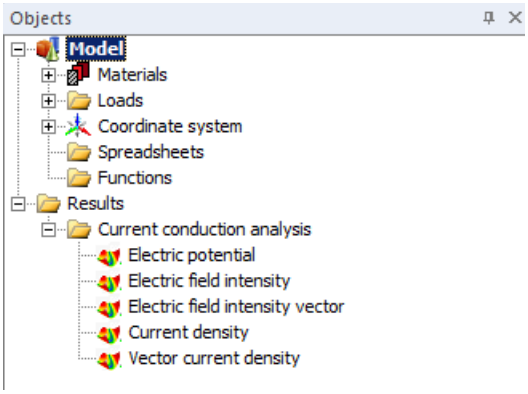


Fig. 10.5 - Results of direct currents fields calculation

The user is available for viewing the results of five types of cards:

1) Electric potential of [V] (contoured);

2) Electric field strength [V] / [length *] (contoured);

3) Vectorial strength of electric field [V] / [length *] (vectorial);

4) [A]/^{2} [length*] current density (contoured);

5) Vectorial [A]/^{2} [length*] current density (vectorial).

Length* is determined by a value selected in the drop-down list. The ***length* of the dialog window ***'Settings'*** in tab ***"Units"***.

For details work with Cards of results see in the section "Results" in the user manual "Work with a Project Tree".

Magnetostatics calculation

During resolution of a magnetostatics task it is considered that all the objects are stationary (not being changed in time). The unknown calculation value is magnetic vector potential through which then strength and induction of magnetic field are being computed.

Calculating model development

For solution of magnetostatics tasks a user needs to define geometry of a calculated area, using three-dimensional finite elements of the first order – four-node (tetrahedrons), six-node (triangular prisms), or eight-node (hexahedrons). It is required for a calculation passing that the model did not contain any other finite elements and was associated, in other words, It was a single entity and not held separately spaced nodes.

All the materials that are used in the model must have a property – relative (relatively to a magnetic constant of $\mu_0=4\pi\cdot10^{-7}$ [Hn]/[m]) magnetic permeability. In Table 10. 5 means of this property determination available for a user are presented.

Table 10.5 Relative magnetic permeability

Anisotropy	Units of Measurements	Ways of definition	Available independent variables
An isotropic one, An orthotropic one	[]	Constant value, Graph. Table, Function	*X* coordinate Y coordinate Z coordinate

At setting of an orthotropic property one is to consider the direction of axes of orientation is coinciding with the local coordinate of the three-dimensional element for which this material is assigned.

Definition of loads and boundary conditions

For modeling of electrostatics tasks the following types of loads and boundary conditions are available to a user:

1) Electric current density;

2) A residual magnetization vector;

3) A magnetic vectorial potential.

Information about these objects is presented in table 10.6

Table 10.6 Magnetostatics loads

Name	Type	Units of Measurements	Application objects	Ways of definition	Available independent variables
Electrical current density	Vectorial one	[A] /[*Length**] is ^{2}	Three-dimensional elements	Constant value, Graph. Table, Function	*X* coordinate Y coordinate Z coordinate
A residual magnetization vector	Vectorial one	[A]/[*length**]	Three-dimensional elements	Constant value, Graph. Table, Function	*X* coordinate *Y* coordinate Z coordinate
Vectorial magnetic potential	Vectorial one	[Вб] /[*Length**]	Nodes, Three-dimensional elements	Constant value, Graph. Table, Function	*X* coordinate *Y* coordinate Z coordinate

Length* is determined by a value selected in the drop-down list. The ***length* of the dialog window ***'Settings'*** in tab ***"Units"***.

Electrical current density

Electric current density is a load for the tasks of magnetostatics, it is applying for a part of the calculated model which is a conductor with current. At setting to a group of three-dimensional elements of a constant value a certain* value will be assigned into each node of all selected three-dimensional elements. At setting to a group of three-dimensional elements of a variable value (graph, table, function) a

certain* value defined by coordinates of a node will be assigned to each node of all selected three-dimensional elements.

If several loads of this type entered into one **loading**, which held one the same three-dimensional element, in the calculation for this one three-dimensional element the sum of a value from all loads of this type of this **Loading** in which the three-dimensional element is included, will be considered.

**A certain value is defined as an integral weight of a node for specified density.*

A residual magnetization vector

The residual magnetization Vector is the load for the tasks of magnetostatics, it is applying for a part of the calculated model which is a permanent magnet. At setting to a group of three-dimensional elements of a constant value a certain* value will be assigned into each node of all selected three-dimensional elements. At setting to a group of three-dimensional elements of a variable value (graph, table, function) a certain* value defined by coordinates of a node will be assigned to each node of all selected three-dimensional elements.

If several loads of this type entered into one **loading**, which held one the same three-dimensional element, in the calculation for this one three-dimensional element the sum of a value from all loads of this type of this **Loading** in which the three-dimensional element is included, will be considered.

**A certain value is defined as an integral weight of a node for specified density.*

Vectorial magnetic potential

Vector magnetic potential is the boundary condition for the tasks of magnetostatics, the user, for this vector load, may specify not all components of the vector. The not specified components will be computed on the input of magnetostatics calculation. At setting to a nodes group of a constant value, a given value will be assigned to each node. At setting to a node group of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned to each node.

At setting to a group of three-dimensional elements of a constant value a given value will be assigned to each node of all selected three-dimensional elements. At setting to a group of three-dimensional elements of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned into each node of all selected three-dimensional elements.

If one **"Loading"** includes several loads of this type, which contains one and the same node (three-dimensional element), then the calculation will be considered for this node (three-dimensional element) value of the last created this type.

For correct setting of a task the tangential **Vectorial Magnetic Potential** on the external surfaces of a pattern components must be specified by constant value 0.

All the described types of loads may be added to a document by the choice of a corresponding shortcut menu item for the node **Magnetic loads** on the panel **Objects**.

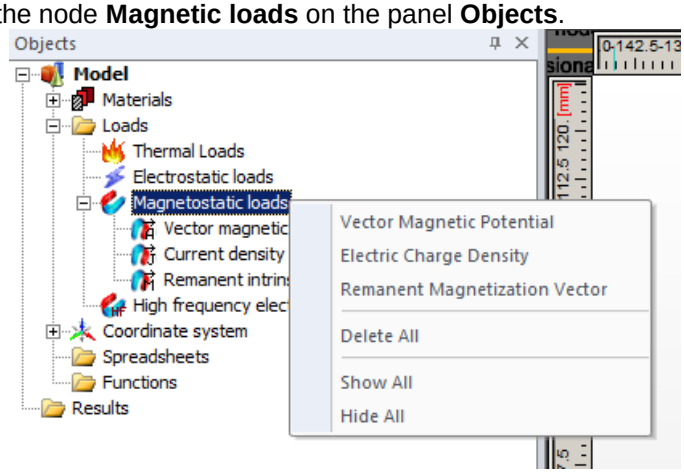


Fig. 10.6 Magnetostatics loads

Performing calculation

After fulfillment of the **Magnetostatic calculation** is necessarily to select the item **Calculation...**. In the horizontal menu **Calculations** the dialog window **Calculation** will appear.

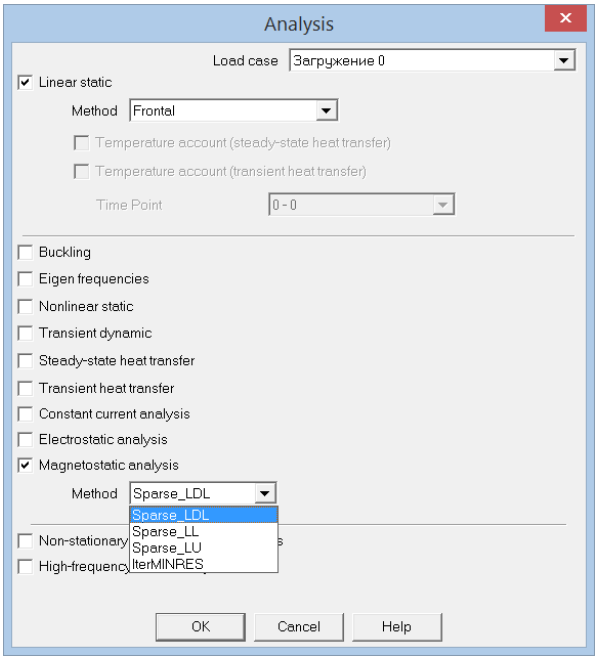


Fig. 10.7 Launch of a magnetostatics calculation

In the dialog window **Calculation** the user must mark the **"Magnetostatics Calculation"** section, in the drop-down list for loading must select **"Loading"**, for which a user wants to fulfill a calculation, to select in the drop-down list **"Method"** the method by which the system of linear algebraic equations for a posed discrete task will calculate. After a press of **"OK"** button the **"Magnetostatics Calculation"** will be made.

In a case if the user in **Loading** for which calculation is performed, did not specify any load of the type Vector magnetic potential, where at least one node or the three-dimensional element is turned on, he will receive an ambiguous result (with accuracy to a constant term). In addition, the matrix of the linear algebraic equations system for the posed discrete task soonest of all, will be ill-conditioned, which can lead to impossibility of the task solution.

About selection of a solution method for the linear algebraic equations system for a posed discrete task see point 2.1.3.

View of results

After an electrostatic calculation, in the bar "Objects" in the "Results" node new menu item "Electrostatic Calculation" will appear.

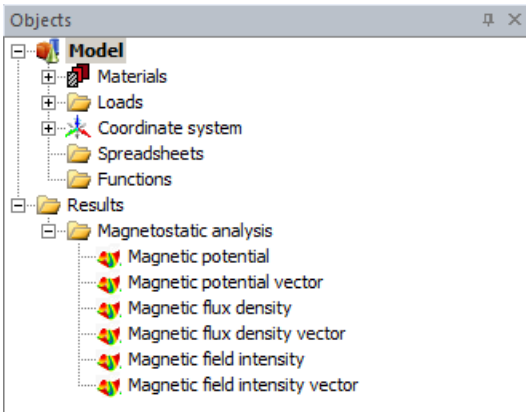


Fig. 10.8 Electrostatic calculation results

The user is available for viewing the results of the six types of cards:

1) Magnetic potential of [Vb]/[length*] (contoured);

2) Vector magnetic potential of [Vb]/[length*] (vectorial);

3) Magnetic induction of [Tl] (contoured);

4) Vector magnetic induction of [Tl] (vectorial);

5) Magnetic field force [A]/[length*] (contoured);

6) Vectoral magnetic field force [A]/[length*] (contoured).

Length* is determined by a value selected in the drop-down list. The ***length* of the dialog window ***'Settings'*** in tab ***"Units"***.

For details work with Cards of results see in the section "Results" in the user manual "Work with a Project Tree".

Nonstationary electromagnetic calculation

During solution of stationary problems of electromagnetic field it is considered that all the objects can change in time. In the model magnets, conductors with current density specified (familiar), conductors with unknown current density, permanent magnets can be present. The unknown of a calculation quantity is magnetic vector potential and the time integral of electric potential through which then buoyancy and induction of magnetic field, as well as voltage of electric field and electric current (for medium with the non-zero electric conductivity) density are being computed.

Calculating model development

For a solution of stationary electromagnetic field tasks the user needs to define geometry of a calculated area, using three-dimensional finite elements of the first order – four-node (tetrahedrons), six-node (triangular prisms), or eight-node (hexahedrons). It is required for a calculation passing that the model did not contain any other finite elements and was associated, in other words, It was a single entity and not held separately spaced nodes.

All the materials that are used in a model must have two properties:

1) Relative (relative to a magnetic constant $\mu_0=4\pi\cdot10^{-7}$ [Hn]/[m]) magnetic permeability (table 10.5);

2) Specific electric conductance (table 10.3).

Definition of loads and boundary conditions

For modeling of stationary electromagnetic field tasks the following types of loads and boundary conditions are available to a user:

1) Time integral of electric potential;

2) Input section of current;

3) Electric current density;

4) Residual magnetization vector;

5) Vectorial magnetic potential.

Information about these objects is presented in table 10.7

Table 10.7 Electrostatics loads

Name	Type	Units of Measurements	Application objects	Ways of definition	Available independent variables
Time integral of electric	Scalar one	[B]·[c]	Nodes, Three-	Constant value, Graph.	*X* coordinate *Y* coordinate

potential**			dimensional elements	Table, Function	Z coordinate
Input section of current	Scalar one	[A]	Nodes,	Constant value, Graph. Table, Function	Time
Electrical current density	Vectorial one	[A] /[*Length**] is ^{2}	Three-dimensional elements	Constant value, Graph. Table, Function	*X* coordinate Y coordinate Z coordinate
A residual magnetization vector	Vectorial one	[A]/[*length**]	Three-dimensional elements	Constant value, Graph. Table, Function	*X* coordinate *Y* coordinate Z coordinate
Vectorial magnetic potential	Vectorial one	[Вб] /[*Length**]	Nodes, Three-dimensional elements	Constant value, Graph. Table, Function	*X* coordinate *Y* coordinate Z coordinate

Length* is determined by a value selected in the drop-down list. The ***length* of the dialog window ***'Settings'*** in tab ***"Units"***.

**The time integral of electric potential is also defined as electric potential (it is an object of the same type), there is dimension of [V] in the channel interface

Time integral of electric potential

The time Integral of electric potential is the boundary electromagnetic field condition for stationary tasks. At setting to a nodes group of a constant value, a given value will be assigned to each node. At setting to a node group of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned to each node.

At setting to a group of three-dimensional elements of a constant value a given value will be assigned to each node of all selected three-dimensional elements. At setting to a group of three-dimensional elements of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned into each node of all selected three-dimensional elements.

If one **"Loading"** includes several loads of this type, which contains one and the same node (three-dimensional element), then the calculation will be considered for this node (three-dimensional element) value of the last created this type.

For the not conducting areas of a pattern this boundary condition must be applied with constant value 0.

Input section of current

The Input section of current is load for the stationary electromagnetic field tasks, it is applying for determination of a conductor section with unknown current density, on the opposite section the **time Integral of electrical potential** with constant value 0 must be specified to all nodes.

Electrical current density

Electrical Current Density is load for the stationary electromagnetic field tasks, it is applying for a part of the calculated model which is the sink explorer. At setting to a group of three-dimensional elements of a constant value a certain* value will be assigned into each node of all selected three-dimensional elements. At setting to a group of three-dimensional elements of a variable value (graph,

table, function) a certain* value defined by coordinates of a node will be assigned to each node of all selected three-dimensional elements.

If several loads of this type entered into one **loading**, which held one the same three-dimensional element, in the calculation for this one three-dimensional element the sum of a value from all loads of this type of this **Loading** in which the three-dimensional element is included, will be considered.

**A certain value is defined as an integral weight of a node for specified density.*

A residual magnetization vector

The remanence vector is load for the stationary electromagnetic field tasks, it is applying for a part of the calculated model which is the permanent magnet. At setting to a group of three-dimensional elements of a constant value a certain* value will be assigned into each node of all selected three-dimensional elements. At setting to a group of three-dimensional elements of a variable value (graph, table, function) a certain* value defined by coordinates of a node will be assigned to each node of all selected three-dimensional elements.

If several loads of this type entered into one **loading**, which held one the same three-dimensional element, in the calculation for this one three-dimensional element the sum of a value from all loads of this type of this **Loading** in which the three-dimensional element is included, will be considered.

**A certain value is defined as an integral weight of a node for specified density.*

Vectorial magnetic potential

Vectorial Magnetic Potential is the boundary condition for the stationary electromagnetic field tasks, a user – for this vector load, may specify not all components of the vector. The not specified components will be computed on the input of magnetostatics calculation. At setting to a nodes group of a constant value, a given value will be assigned to each node. At setting to a node group of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned to each node.

At setting to a group of three-dimensional elements of a constant value a given value will be assigned to each node of all selected three-dimensional elements. At setting to a group of three-dimensional elements of a variable value (graph, table, function) a value defined by coordinates of a node will be assigned into each node of all selected three-dimensional elements.

If one **"Loading"** includes several loads of this type, which contains one and the same node (three-dimensional element), then the calculation will be considered for this node (three-dimensional element) value of the last created this type.

For correct setting of a task the tangential **Vectoral Magnetic Potential** on the external surfaces of a pattern components must be specified by constant value 0.

All the described types of loads can be added to a document by the selection of a corresponding shortcut menu item for the site the **"Electrical Load"** or the site **"Magnetic Load"** in the bar **"Objects"**.

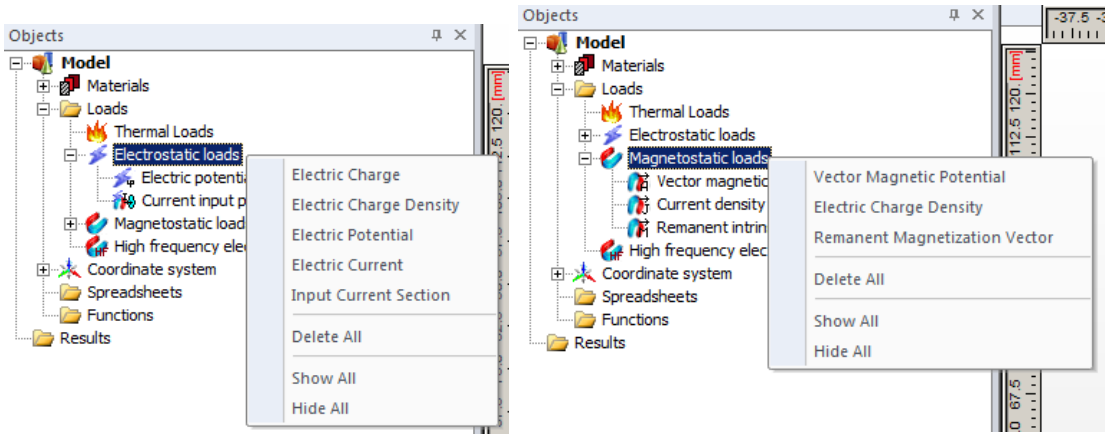


Fig. 10.9 is the Loads for stationary electromagnetic field tasks

Performing calculation

After what for the spending of the **Static Electromagnetic Calculation** is necessarily to select the point **"Calculation..."** in the horizontal menu **"Calculations"** the dialog window **"Calculation"** will appear. (Fig. 4.73).

In the dialog window "Calculation" the user must mark the **Stationary electromagnetic detachment** section, in the drop-down list for loading must select **"Loading"** for which he wants to do a calculation to the field **Interval**: 0 enter the positive value which defines end time of the modeling. To the field of the **"Moments of Time"** the user must enter the positive integer value which determines time of how many even intervals of modeling will be divided. The field **"Parameter of an Integration Circuit of** Thetha" [0; 1] the user must enter a positive value in the range of [0; 1]. After a press of an **"OK"** button the **Static Electromagnetic Calculation** will be made.

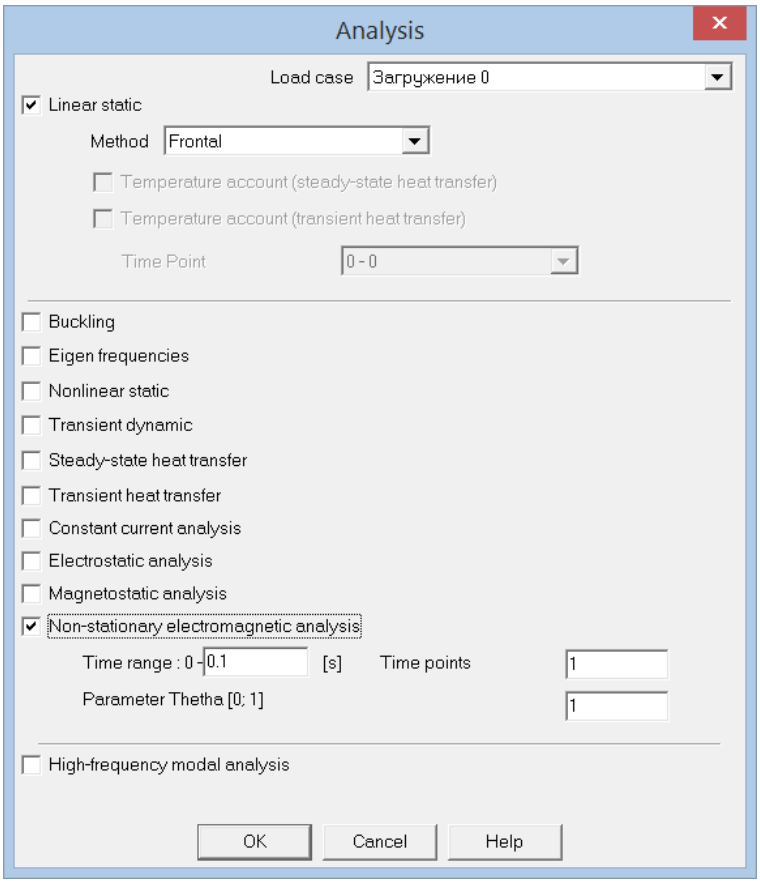


Fig. 10.10 Launch of the static electromagnetic calculation

The parameter of an integration circuit is, theoretically, [0; 1], defines a one-layered method of integration of equations by the time. The certainly stable schemes for the range, however, of [0.5; 1] and them at a sufficiently large pitch are possible to oscillation of a solution. Particular methods of integration are reduced below for some values of the parameter:

0 is the Euler method with a direct increment by the time (a difference ahead) (classical scheme of numerical integration of differential first order equations);

1 - Euler method with a reverse step by the time (with a difference backward);

0,5 - the method of Krank Nicholson (with the central difference);

2/3 - Galerkin method.

If there is a pattern of permanent magnets, and areas with a predetermined current density, in the first few steps of integration is possible to obtain oscillations in the solution. It's due to the fact that the problem is solved with trivial initial conditions, whereby the components of the electromagnetic field changes abruptly.

View of results

The new item after spending of an electrostatic design to appear in the bar **Objects** in the Results node **Stationary an electromagnetic calculation**. After the calculation of the electrostatic, on the panel Objects, in the node Results the new item Non-stationary electromagnetic calculation will appear.

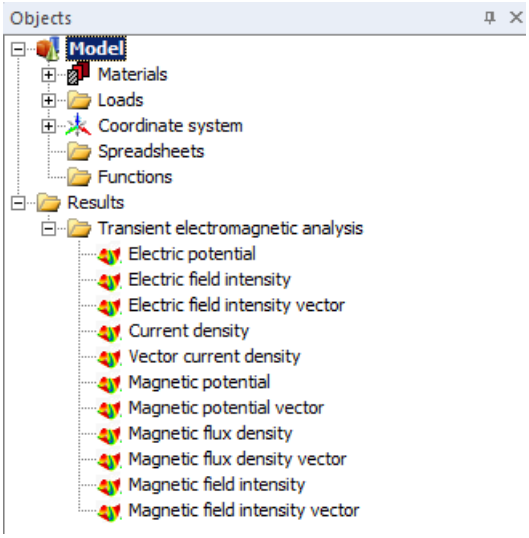


Fig. 10.11 Results of a non-stationary electromagnetic calculation

For the user to view, eleven types of results card are accessibly:

1) Electric potential of [V] (contoured);

2) Electric field strength [V] / [length *] (contoured);

3) Vectorial strength of electric field [V] / [length *] (vectorial);

4) [A]/^{2} [length*] current density (contoured);

5) Vectorial [A]/^{2} [length*] current density (vectorial).

6) Magnetic potential of [Vb]/[length*] (contoured);

7) Vectorial magnetic potential of [Vb]/[length*] (vectorial);

8) Magnetic induction of [Tl] (contoured);

9) Vectorial magnetic induction of [Tl] (vectorial);

10) Magnetic field strength [A]/[length*] (contoured);

11) Vectorial magnetic field strength [A]/[length*] (vectorial).

Length* is determined by a value selected in the drop-down list. The ***length* of the dialog window ***'Settings'*** in tab ***"Units"***.

The user has a possibility to animate each type of results card.

For details work with Cards of results see in the section "Results" in the user manual "Work with a Project Tree".

Analysis in the high frequency area

High frequency modal analysis

High frequency modal analysis is intended for a calculation of the natural frequencies (cut-off frequencies) and forms of electromagnetic structures (waveguides, resonators) working at high frequencies (~ 1 MHz ~ 10 GHz).

During decision of this type tasks it is considered that there is no attenuation of wave processes. The unknown calculation value is projection of electric field strength vector to «edges» of the grid through which strength of electric and magnetic field is being computed.

Calculating model development

To meet the tasks of calculating the natural frequencies of electromagnetic structures the user must determine the geometry of the computational area, using three-dimensional finite elements of the first order are four-node (tetrahedrons), six- node (triangular prisms) or eight-node (hexahedrons). It is required for a calculation passing that the model did not contain any other finite elements and was associated, in other words, It was a single entity and not held separately spaced nodes. During resolution of the task the three-dimensional elements transform into "edged"

All the materials that are used in a model must have two properties –

1) Relative (relative to an electric constant ε_0=8, 854187817·10^{-12} [F]/[m]) dielectric permeability (table 4.2.1);

2) Relative (relative to a magnetic constant μ0=4π·10-7 [Hn]/[m]) magnetic permeability (table 4.2.5).

Definition of loads and boundary conditions

The calculation own the frequencies in total one boundary condition **Perfect electric conductor** is available for modeling of stationary problems to a user of electromagnetic structures.

Perfect electric conductor

This boundary condition can be set to model edges and defines an equality to a zero of the projection of the electric field strength vector on these edge. The perfect electric conductor must be set to all external borders of a pattern with an exception of planes of symmetry. In addition, in the model such "walls" may exist, which are reflecting the electromagnetic wave, for such facilities as necessary to set this boundary condition.

Above described boundary condition may be added to the document by the way of the selection the item of the corresponding shortcut menu for the node **"High Frequency Loads"** in the panel **"Objects"**

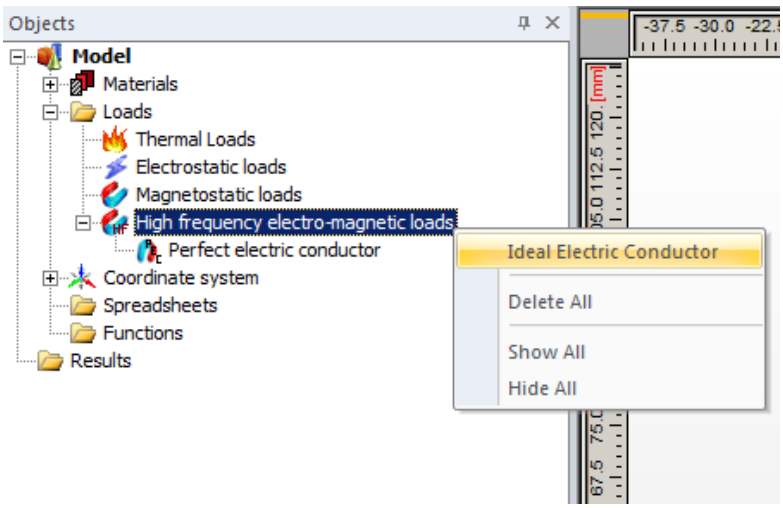


Fig. 10.12 Loads for high frequency modal analysis

Performing calculation

For fulfillment of **High frequency modal analysis** in the horizontal menu **"Calculations"** a point **"Calculation..."**ought to be selected after what the dialog window **"Calculation"** will appear.

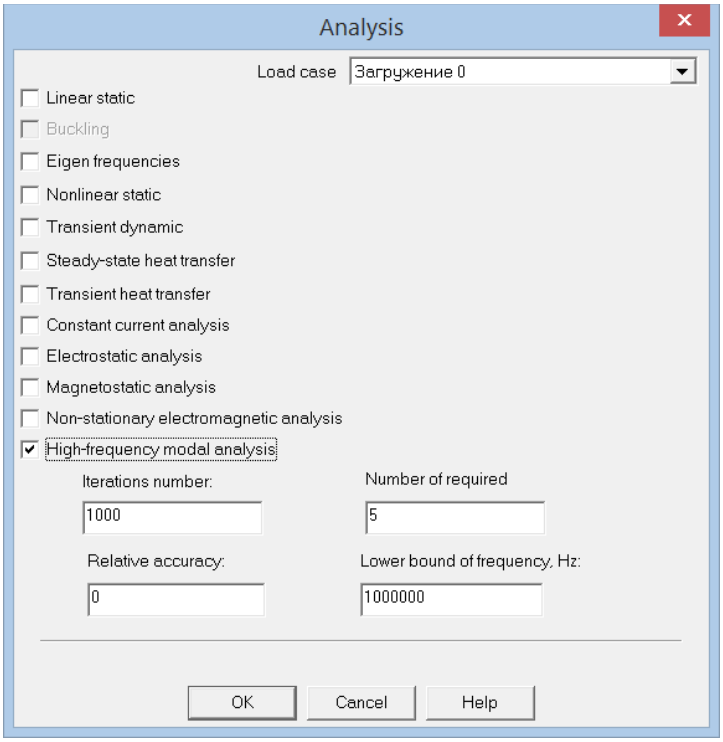


Fig. 10.13 Launching of high frequency modal analysis

In the dialog window **"Calculation"** the user must mark the point **"High Frequency Modal Analysis"**, to select in the drop-down list of loading **"Loading"** for which a user wants to make the calculation, to input into the field **"Searched Frequencies Number"** the positive integer value which defines the lowest natural frequency number and forms corresponding to frequencies that will be found. In the field **"Lower Limit"** of frequency, Hz, the user must enter the positive value which determines the frequency in Hertzes in less than its value non-search of the natural frequencies. Into the field **"Number of Iterations"** the user must enter the positive integer value in the range which specifies the maximally possible number of iterations in the Lantsosh method. In the field **"Relative error"** a user must enter a positive lower value, less than 1, in the range, what a precision of convergence of the own vectors determinate in the Lantsosh method. After a press of an **"OK"** button **High frequency modal analysis** will be performed.

The fields **"Number of Iterations"** and **"Relative Error"** are recommended to leave with values by default with 1000 and 0. If not all the natural frequencies (a message about which the user will receive) were found during solution, one must repeat a calculation having increased the **"Iteration Number" or** "Relative Error". It is not recommended to enter the value of a relative error more than 10^{-6}.

View of results

After the high-frequency modal analysis on the panel **"Objects"** in the **"Results"** node a new menu item **3D calculation of the natural frequencies of the electromagnetic fields** will appear.

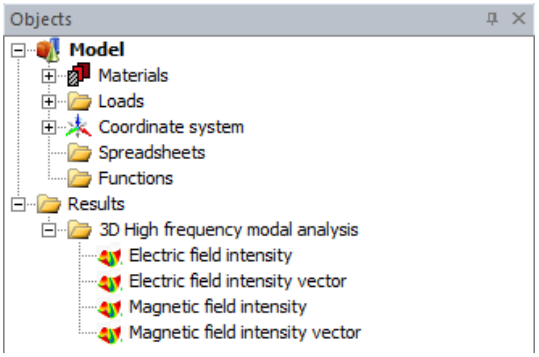


Fig. 10.14 Results of high frequency modal analysis

Four types of results cards are available to the user for view:

1) The electric field strength [V]/[length*] (contoured);

2) Vectorial strength of electric field [V]/[length*] (vectorial);

3) The magnetic field force [A]/[length*] (contoured);

4) Vectorial magnetic field force [A]/[length*] (vectorial).

Length* is determined by a value selected in the drop-down list. The ***length* of the dialog window ***'Settings'*** in tab ***"Units"***.

For details work with Cards of results see in the section "Results" in the user manual "Work with a Project Tree".

Brief theoretical information

Maxwell equations, describing state electromagnetic field may be written under a differential form in the following form:

$$\begin{cases} \nabla \times \left(\vec{H}\right) = \vec{J} + \dfrac{\partial \vec{D}}{\partial t}; \\ \nabla \times \left(\vec{E}\right) = -\dfrac{\partial \vec{B}}{\partial t}; \\ \nabla \cdot \left(\vec{B}\right) = 0; \\ \nabla \cdot \left(\vec{D}\right) = \rho, \end{cases} \tag{10.1}$$

where t - time [s]; $\vec{D}$ - Electric induction vector (offset) [Tl] / [m2]; $\vec{E}$ - The electric field vector [B] / [m]; - $\vec{B}$ The vector of magnetic induction [Tl]; $\vec{H}$ - Magnetic field force vector [A] / [m]; $\vec{J}$ - Vector electric current density [A] / [m2]; ρ - the volume density of electric current; - $\nabla \cdot (\ )$ Operation of divergence; $\nabla \times (\ )$ - The operation of the rotor.

Vectors describe the state of the electromagnetic field are related as follows:

$$\vec{D} = \varepsilon_a \vec{E}, \tag{10.2}$$

Where – $\varepsilon_a = \varepsilon\varepsilon_0$ absolute dielectric permeability of the environment [F]/[м], ε are relative dielectric (relative to vacuum) permeability of the environment []; ε_0 – the dielectric constant (absolute dielectric permeability of vacuum) is 8, 854187817·10^{-12} [F]/[m].

The expression (10.2) is true only in the isotropic electric environment, if environment is orthotropic, it must be written in appearance:

$$\vec{D} = [\varepsilon]\vec{E}, \tag{10.3}$$

where [ε] is an absolute dielectric medium permeability matrix:

$$[\varepsilon] = \varepsilon_0 \begin{pmatrix} \varepsilon_{xx} & 0 & 0 \\ 0 & \varepsilon_{yy} & 0 \\ 0 & 0 & \varepsilon_{zz} \end{pmatrix}, \tag{10.4}$$

Here ε_{xx}, ε_{yy}, and ε_{zz} are relative dielectric permeability of environment in x, y, and corresponding z directions.

$$\vec{B} = \mu\vec{H}, \tag{10.5}$$

Where – $\mu_a = \mu\mu_0$ absolute magnetic permeability of the environment [Hn]/[m], μ – relative (relatively to vacuum) magnetic permeability of the environment [], μ_0 are a magnetic constant (absolute magnetic permeability of vacuum) of the $4\pi\cdot10^{-7} \approx 1,25663706\cdot10^{-6}$ Hn]/[m].

The expression (10.5) is true only in isotropic magnetic medium, if medium is orthotropic, it must be written in appearance:

$$\vec{B} = [\mu]\vec{H}, \tag{10.6}$$

where [μ] is the magnetic environment permeability matrix:

$$[\mu] = \mu_0 \begin{pmatrix} \mu_{xx} & 0 & 0 \\ 0 & \mu_{yy} & 0 \\ 0 & 0 & \mu_{zz} \end{pmatrix}, \tag{10.7}$$

Here μ_{xx}, μ_{yy}, and μ_{zz} are relative (relatively to vacuum) magnetic permeability of the environment in directions of x, y, and z respectively [].

In case of presence of permanent magnet the correlation (10.6) assumes appearance:

$$\vec{B} = [\mu]\vec{H} + \mu_0\vec{M}_0, \tag{10.8}$$

Where – the $\vec{M}_0$ vector of the own residual magnetization of permanent magnet [A/m].

Very often it is necessary to express the force of magnetic field:

$$\vec{H} = [\nu]\vec{B} - \mu_0[\nu]\vec{M}_0, \tag{10.9}$$

where [ν] is a matrix of magnetic resistance of the environment:

$$[\nu]=\frac{1}{\mu_0}\begin{pmatrix}\frac{1}{\mu_{xx}} & 0 & 0\\ 0 & \frac{1}{\mu_{yy}} & 0\\ 0 & 0 & \frac{1}{\mu_{zz}}\end{pmatrix}. \qquad (10.10)$$

An equation of inviolability of electromagnetic field follows from the first equation of the system (10.1) after application of a divergence operation:

$$\nabla\cdot\left(\vec{J}+\frac{\partial\vec{D}}{\partial t}\right)=0. \qquad (10.11)$$

One is also to note electric current density is bound with electric field strength (the Ohm's law) by magnetic induction (the Amp's law):

$$\vec{J}=[\sigma]\left(\vec{E}+\vec{v}\times\vec{B}\right), \qquad (10.12)$$

where [σ] is the specific electric orthotropic environment 1 conductance/([Ohm] ·[m]) matrix:

$$[\sigma]=\begin{pmatrix}\sigma_{xx} & 0 & 0\\ 0 & \sigma_{yy} & 0\\ 0 & 0 & \sigma_{zz}\end{pmatrix}, \qquad (10.13)$$

Here σ_{xx}, σ_{yy}, and σ_{zz} are specific electric conductance of medium in directions of a x, y, and z respectively to 1/(the [Ohm] ·[m]); $\vec{v}$ velocity of a conductor in magnetic field [m]/[s].

Electrostatics

Electrostatic field, if magnetic field is absent, even electric currents naturally are absent, can arise in medium when there are electric charges. In this case Maxwell (10.1)'s equations will assume an appearance:

$$\begin{cases}\vec{H}=0;\\ \nabla\times\left(\vec{E}\right)=0;\\ \vec{B}=0;\\ \nabla\cdot\left(\vec{D}\right)=\rho.\end{cases} \qquad (10.14)$$

It follows from the second equation (10.14) that electric field is potential. Therefore, for it some scalar potential determined by a following expression exists:

$$\vec{E}=-\nabla(\varphi), \qquad (10.15)$$

Where φ is electric potential [V]; a gradient $\nabla(\ )$ operation.

Substituting to fourth equation (10.14) for expressions (10.15) and (10.3), we get Poisson's following equation for electrostatic field:

$$-\nabla\cdot\left([\varepsilon]\nabla(\varphi)\right)=\rho. \qquad (10.16)$$

After solution of equation (10.16) of an electric field vector they are restored over expressions (10.15) and (10.3).

Field of direct currents

If only the conductors where direct current is flowing are examined, in other words, if change in time of magnetic, as well as electric field is absent, Maxwell (10.1)'s equations will assume an appearance:

$$\begin{cases} \nabla \times \left(\vec{H}\right) = \vec{J}; \\ \nabla \times \left(\vec{E}\right) = 0; \\ \nabla \cdot \left(\vec{B}\right) = 0; \\ \nabla \cdot \left(\vec{D}\right) = \rho; \end{cases} \tag{10.17}$$

It (10.17) follows from the second equation that electric field is potential. Therefore, for it some scalar potential (10.15 ones, although also are determining from an equation, it is exists). Equation (10.11) of continuity in this case apparently assumes an appearance:

$$\nabla \cdot \left(\vec{J}\right) = 0. \tag{10.18}$$

Given there are no mobile conductors, substituting to equation (10.18) for expressions (10.15) and (10.12), we get Laplace's following equation for a calculation of direct currents field:

$$\nabla \cdot \left(\left[\sigma\right] \nabla \left(\varphi\right)\right) = 0. \tag{10.19}$$

After solution of an equation (10.19) a vector of strength of electric field is restored by an expression (10.15), and a density of electric current by the expression vector is being restored are got from (10.12) taking into consideration of absence of mobile conductors:

$$\vec{J} = \left[\sigma\right] \vec{E}. \tag{10.20}$$

Magnetostatics

For a calculation of magnetic field in the absence of change in time of magnetic, as well as electric field a system (10.17) will be used and its equations on the third are going that magnetic field is solenoidal. Therefore, for it some vector potential determined by a following expression exists:

$$\vec{B} = \nabla \times \left(\vec{A}\right), \tag{10.21}$$

Where $\vec{A}$ – vectorial magnetic potential is [Vb]/[m]

For non-ambiguity definition of vectorial magnetic potential we will introduce Coulon's calibration:

$$\nabla \cdot \left(\vec{A}\right) = 0. \tag{10.22}$$

Substituting to first equation (10.17) for expressions (10.21), (10.22) and (10.9), we get the following vector equation for a calculation of magnetic field:

$$\nabla \times \left(\left[\nu\right] \nabla \times \left(\vec{A}\right)\right) - \nabla \left(\nu_e \nabla \cdot \left(\vec{A}\right)\right) = \vec{J} + \nabla \times \left(\mu_0 \left[\nu\right] \vec{M}_0\right), \tag{10.23}$$

ν_e is medium magnetic resistance of medium ($\nu_e = \dfrac{1}{3\mu_0}\left(\dfrac{1}{\mu_{xx}} + \dfrac{1}{\mu_{yy}} + \dfrac{1}{\mu_{zz}}\right)$)

Afterwards the solutions of equation (10.23) of a magnetic field vector are restored by expressions (10.21) and (10.9).

Non stationary electromagnetic field

For description of stationary electromagnetic field we will connect a vector of electric field strength with electric potential and magnetic vector potential as follows:

$$\vec{E} = -\frac{\partial \vec{A}}{\partial t} - \nabla(\varphi). \tag{10.24}$$

Substituting expressions (10.24) (10.21) (10.22), (10.12) (10.9) to first equation (10.1) and (10.11) of the system, we will get the following equations, describing stationary electromagnetic field:

for areas with the current density distribution unknown:

$$\begin{cases} \nabla \times \left([\nu] \nabla \times \left(\vec{A} \right) \right) - \nabla \left(\nu_e \nabla \cdot \left(\vec{A} \right) \right) + [\sigma] \frac{\partial \vec{A}}{\partial t} + [\sigma] \nabla(\varphi) - \vec{v} \times \left([\sigma] \nabla \times \left(\vec{A} \right) \right) = 0; \\ \nabla \cdot \left([\sigma] \frac{\partial \vec{A}}{\partial t} - [\sigma] \nabla(\varphi) + \vec{v} \times \left([\sigma] \nabla \times \left(\vec{A} \right) \right) \right) = 0, \end{cases} \tag{10.25}$$

for the rest of the areas:

$$\nabla \times \left([\nu] \nabla \times \left(\vec{A} \right) \right) - \nabla \left(\nu_e \nabla \cdot \left(\vec{A} \right) \right) = \vec{J} + \nabla \times \left(\frac{1}{\nu_0} [\nu] \vec{M}_0 \right). \tag{10.26}$$

After solution of equations (10.25) and (10.26) the magnetic field vectors are recovered by expressions (10.21) and (10.9), the vector of electric field strength restore by expression (10.24) and the electric current density vector, an expression (10.12).

One is to note for convenience of integration by the time it is convenient to introduce an integral of electric potential into consideration:

$$\varphi^* = \int \varphi dt, \tag{10.27}$$

then the system of equations (10.25) will assume the view:

$$\begin{cases} \nabla \times \left([\nu] \nabla \times \left(\vec{A} \right) \right) - \nabla \left(\nu_e \nabla \cdot \left(\vec{A} \right) \right) + [\sigma] \frac{\partial \vec{A}}{\partial t} + [\sigma] \nabla \left(\frac{\partial \varphi^*}{\partial t} \right) - \vec{v} \times \left([\sigma] \nabla \times \left(\vec{A} \right) \right) = 0; \\ \nabla \cdot \left([\sigma] \frac{\partial \vec{A}}{\partial t} - [\sigma] \nabla \left(\frac{\partial \varphi^*}{\partial t} \right) + \vec{v} \times \left([\sigma] \nabla \times \left(\vec{A} \right) \right) \right) = 0. \end{cases}$$

Electromagnetic high frequency field

For high frequency electro-dynamic processes Maxwell's equations can be reduced to a Helmholtz equation (relatively complex vector of electric field strength $\bar{E}$ [V]/[m]) of an aspect:

$$\nabla \times \left([\mu]_r^{-1} \left(\nabla \cdot \left(\bar{E} \right) \right) \right) - k_0^2 [\varepsilon]_r \cdot \bar{E} = j\omega\mu_0 \bar{J}_S, \tag{10.28}$$

Where $[\mu]_r$ - matrix of relative complex magnetic permeability of the medium []; "-1" - the operation matrix inversion; j - imaginary unit; ω - operating angular frequency [Rad / s]; $[\varepsilon]_r$ - matrix of relative complex permittivity of the medium []; k_0 - wave number of the vacuum [Rad] / [m]; - $\overline{J}_S$ Complex vector density excitation current [A] / [m^2].

Education examples

Calculation of a capacitor (electrostatics)

1. Create the new document where one will have to create a cube of three-dimensional elements divided into 10 parts in every of the directions, Fig. 4.78. This cube will be dielectric between plates of a capacitor. In the nodes charge on the opposite faces of the cube will be specified and these facets will represent plates of a capacitor themselves.

2. For the Steel material add the property "Relative dielectric permeability" with constant value 1.

3. Specify this material to all the model. For that all the model must be selected when the layer "Model" is activated and in the shortcut material menu the item "Specify highlighted" must be selected.

4. Add two loads – "Electric potential". For it in the shortcut menu of a node "Electrical loads" choose a corresponding menu item.

5. Activate only the layer "Potential 0" and for the first load specify all nodes of this layer. Leave the value of load by default (0 V).

6. For the second load activate only the layer "Potential 5" and set electric potential of the 5th Century to all nodes of the layer.

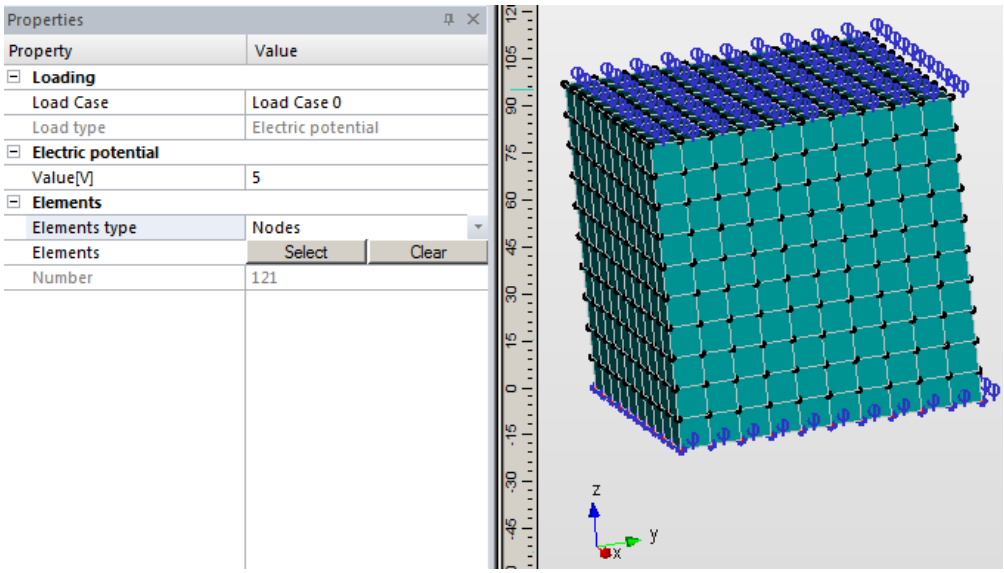


Fig. 10.15 Setting of electric potential

7. Further execute a calculation having selected an item of the horizontal menu "Calculations/Calculation"... and having specified an electrostatic calculation with parameters by default. The result will appear in the tree.

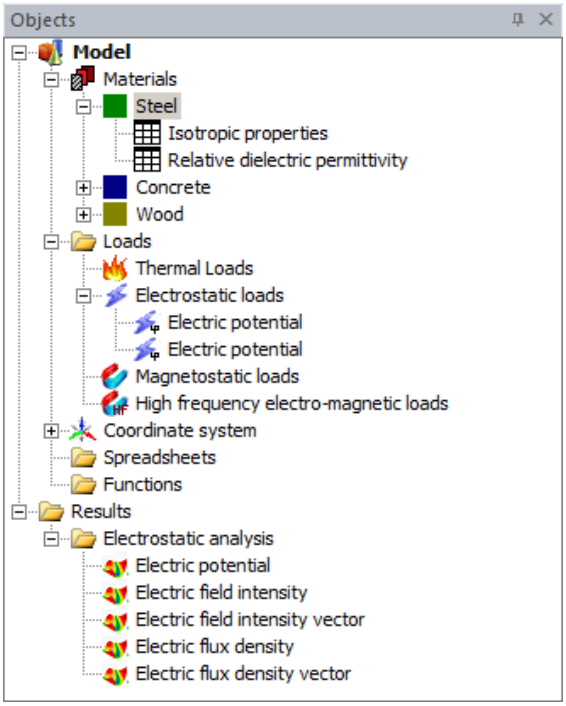


Fig. 10.16 Tree with results of an electrostatic calculation.

The received results are presented in Fig.s 10.17 – 10.21.

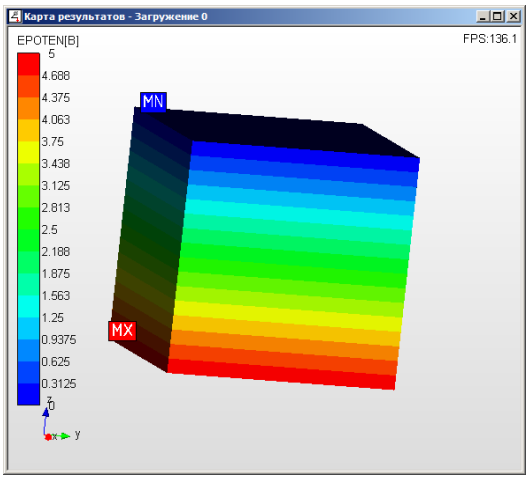


Fig. 10.17 Results map – an electric potential distribution

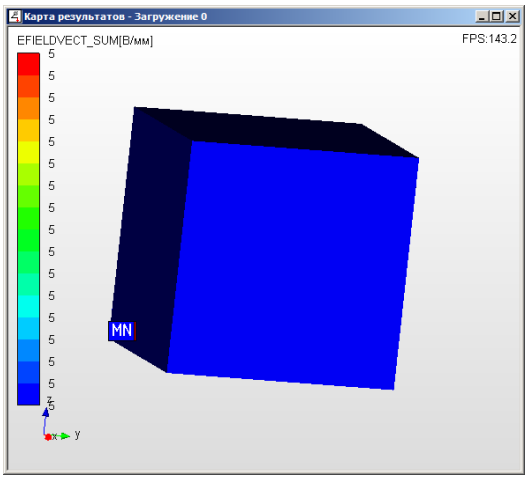


Fig. 10.18 Results map – intensive of electric field are a total value

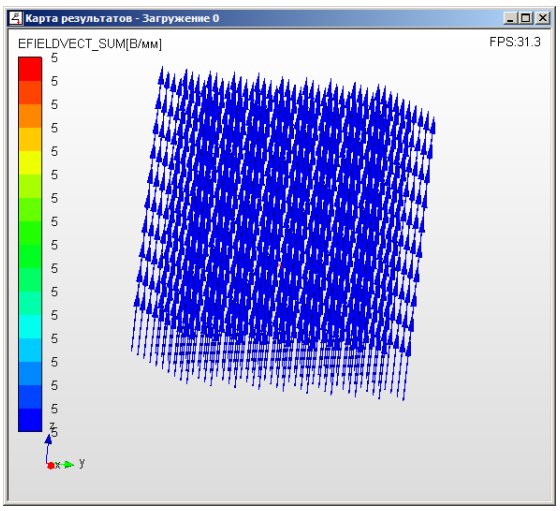


Fig. 10.19 – Results map – total vectorial electric field strength

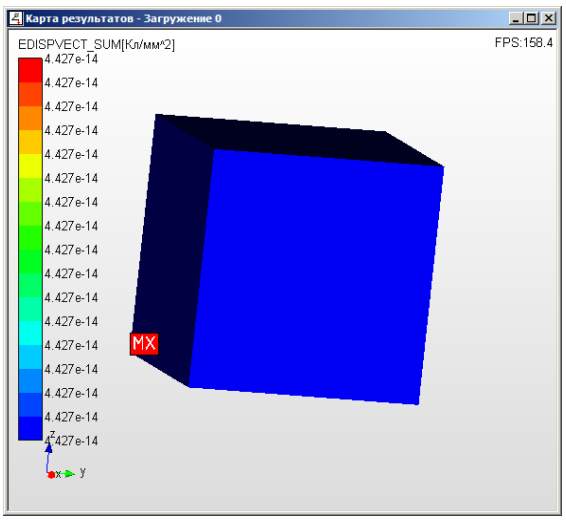


Fig. 10.20 Results map – the total value of electric induction

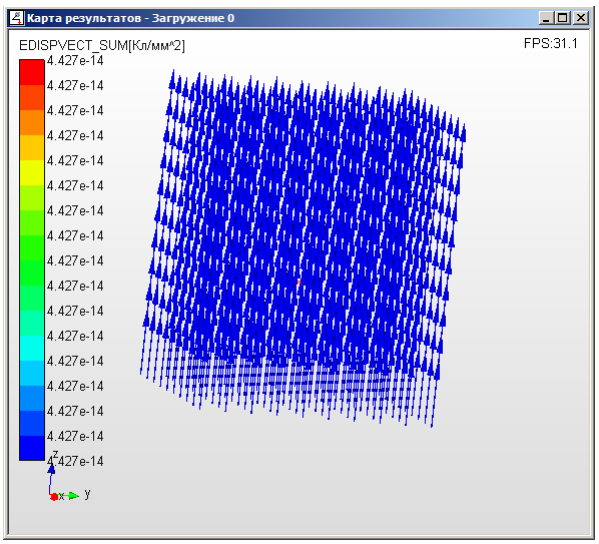


Fig. 10.21 Results map – total value of vectorial electric induction

The calculation of the magnetic circuit (magnetostatics).

Create a new file. A model of a magnetic circuit of rectangular shape consisting of several layers, is created in the file, Fig. 4.83. It is desirable to divide a three-dimensional model with the same parameters as in the Fig.. We specify the layers with the names "Steel", "Magnet", and "Air", see fig. 4.83.

Definition of materials in the model

1. Deactivate all layers except the layer "Steel".

2. Select a part of the model lying in this layer.

3. In the tree in the shortcut menu of the steel material one must select the item "Specify highlighted"

4. Add to steel material the property "Relative magnetic permeability" with constant value 2500.

5. Deactivate the layer "Steel" and turn on the layer "Magnet".

6. Add new general material with the name "Magnet" and property "Relative magnetic permeability" with constant value 1.

7. Select a visible part of the model and specify it the produced magnet material.

8. Deactivate the layer "Magnet" and turn on the "Air" layer.

9. Create the third "Air" material with parameters of the previous one.

10. Define the remainder of the material model "Air'.

The result of operations performed is presented in figure below.

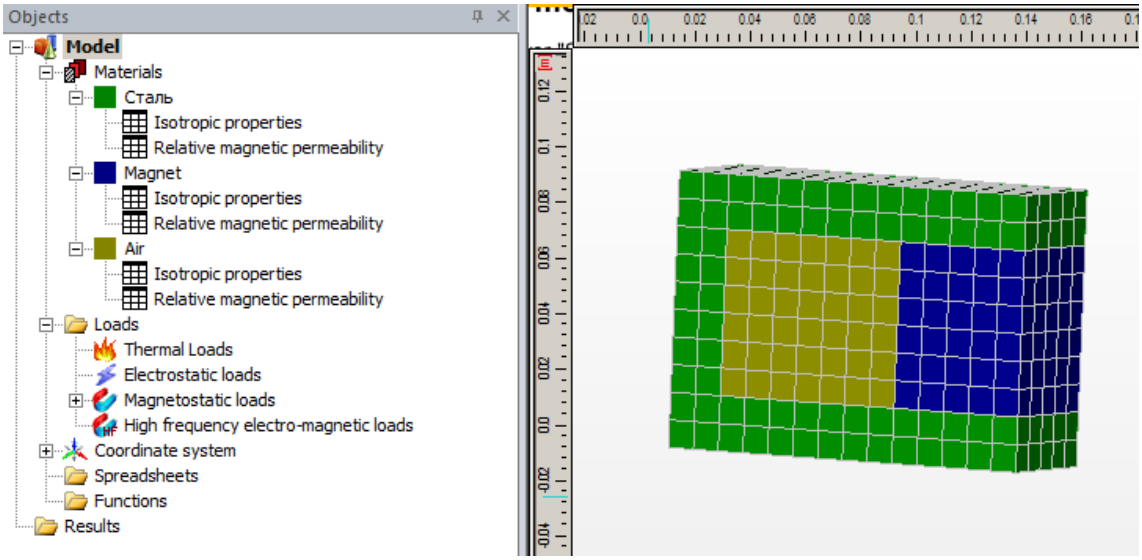


Fig. 10.22 Model with specified materials.

Definition of loads in the model

All the elements on which, is necessarily to set loads, are carried out to separate layers.

1. Switch off all layers except the layer "Magnet".

2. Add the load "Residual magnetization vector".

3. Select a type of elements – "Three-dimensional elements".

4. Push the button "Install" and select all elements of the layer.

5. Push the button "Apply". Number of elements, to which load is applied, must be equal to 150.

6. Specify an equal value of load in direction of Z to 50000 A/m, leaving zero values to the directions of X and Y.

7. Switch on all the model layers.

The result is presented in figure below.

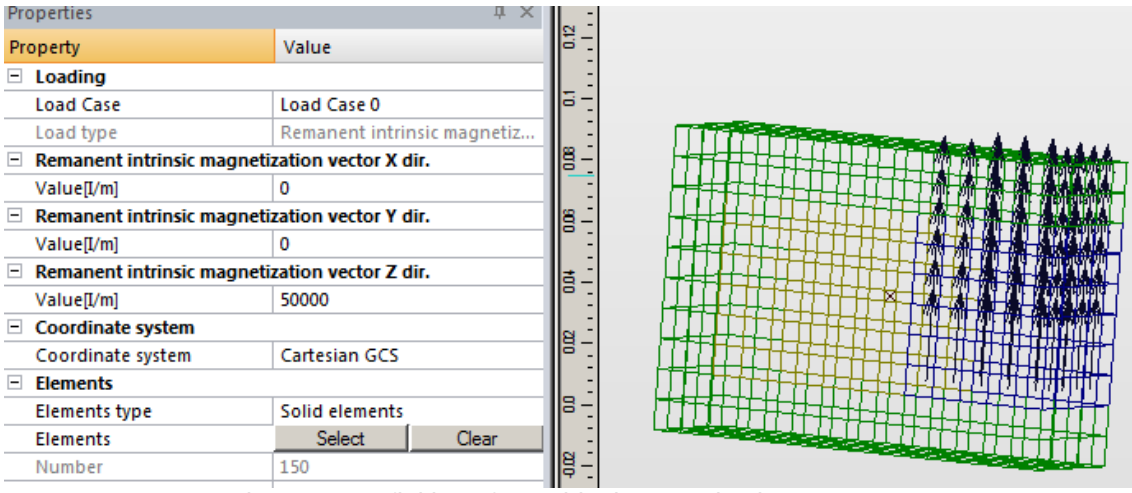


Fig. 10.23 Definition of a residual magnetisation vector.

8. Switch off all layers and activate the layer "Normal along Z axe".

9. Add the load "Vectorial magnetic potential".

10. Select a type of elements "Nodes" and apply to nodes in this layer.

11. Remove a switch in direction of Z, leave the rest of values as zero

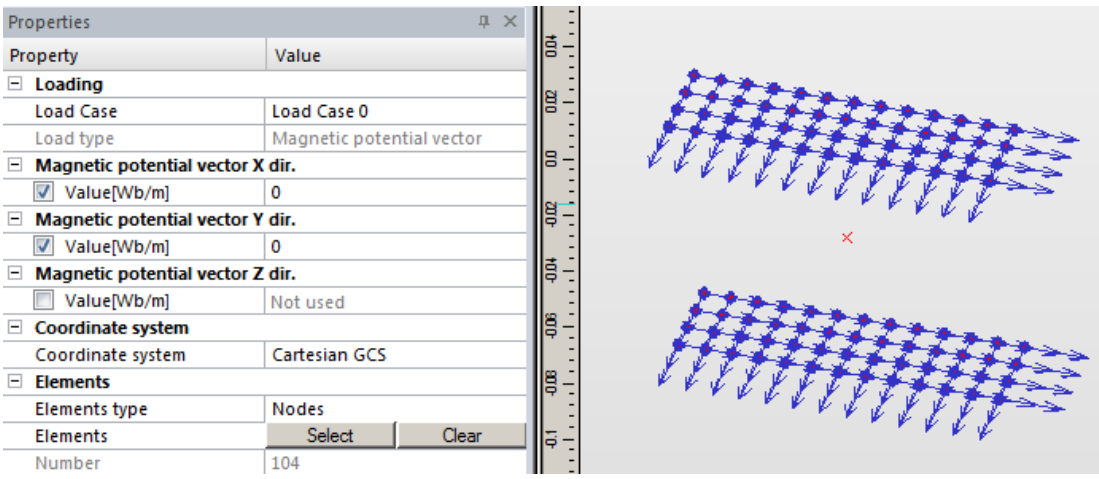


Fig. 10.24 Definition of vectorial magnetic potential

12. Similarly set vectoral magnetic potential to the nodes "Perpendicular to a Y axis" and "Perpendicular to a X axis" in layers, deactivating corresponding components of a vector.

13. Final vectorial magnetic potential is applying on elements of an "Edge" layer. Values are zero in all components of a vector.

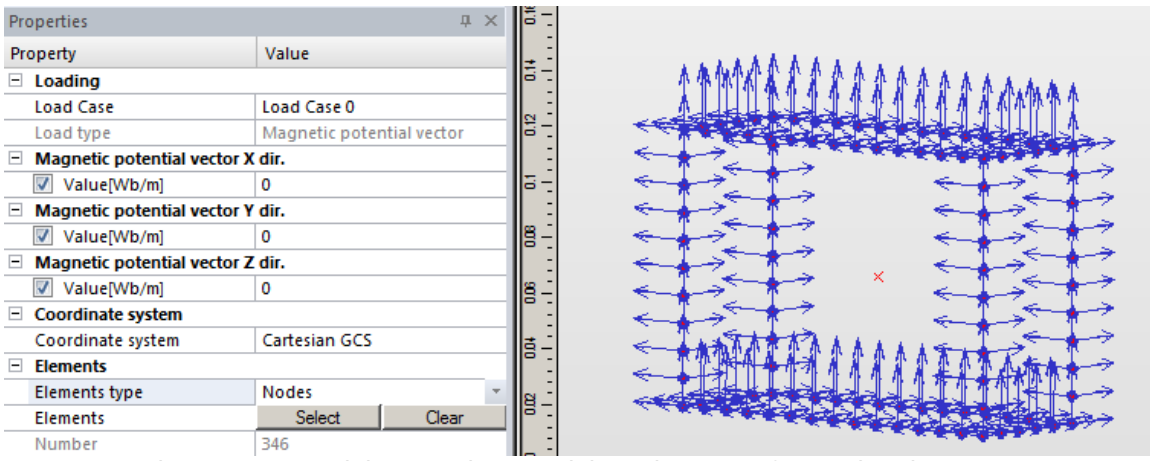


Fig. 10.25 Vectorial magnetic potential on elements of an "Edge" layer

Calculation and view of results

1. The open dialog window **"Calculation"** being called to remove in the menu **Calculations/Calculation...** the check mark of the **Linear static calculation** and set the **Magnetostatic calculation**. The calculation method of **Sparse_LDL**. Push **OK**.

2. After the completion of the calculation in the tree in the node "Results" will appear the subsite node "Magnetostatic calculation" containing the following results.

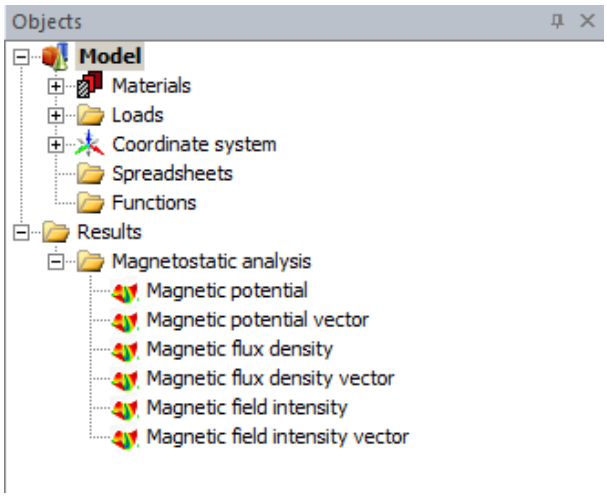
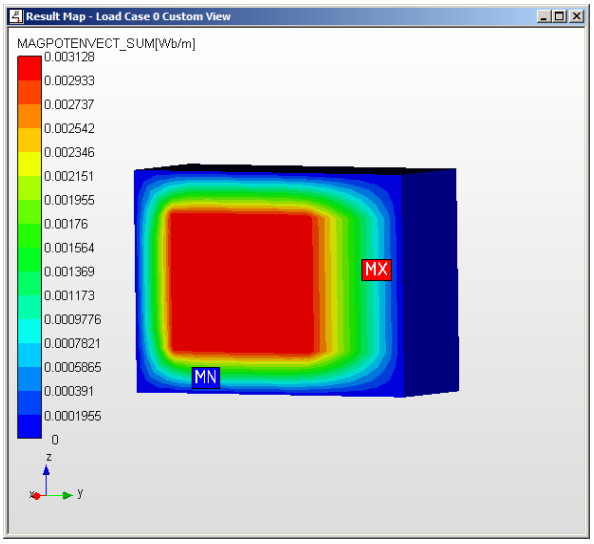


Fig. 10.26 Tree after a magnetostatic calculation passing

3. The results are shown in Fig. 10.27 - 10.32.



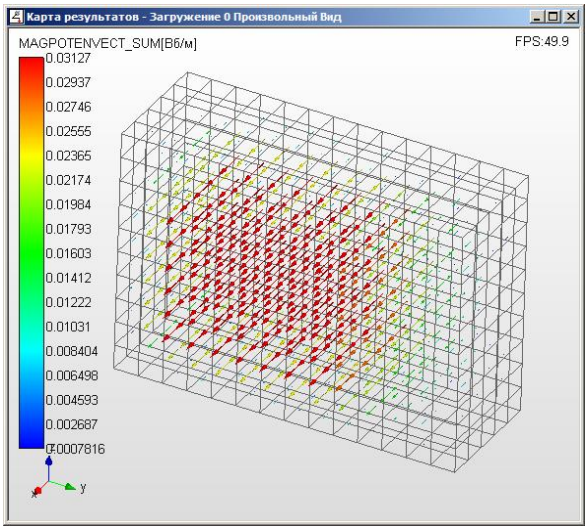


Fig. 10.27 Map of total magnetic potential results

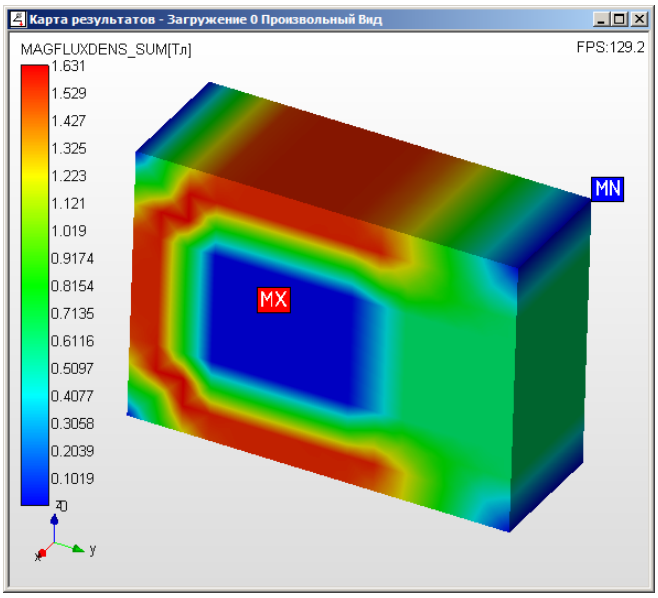


Fig. 10.28 Map of total magnetic vectoral pote

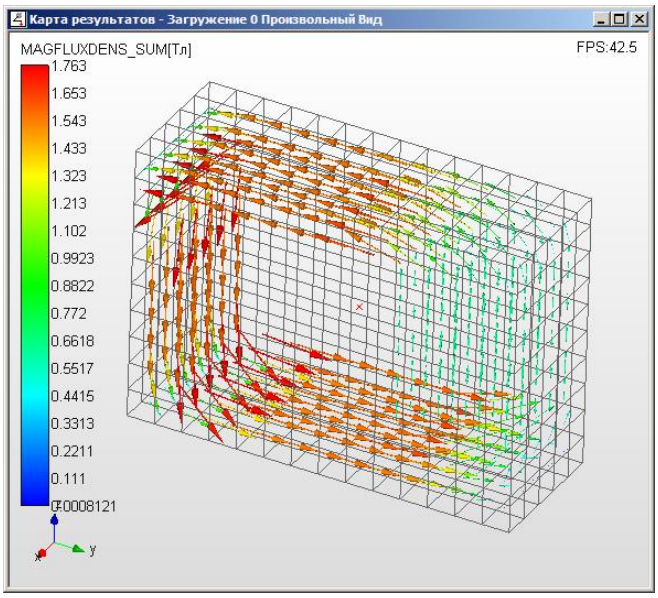


Fig. 10.29 Map of magnetic induction total result

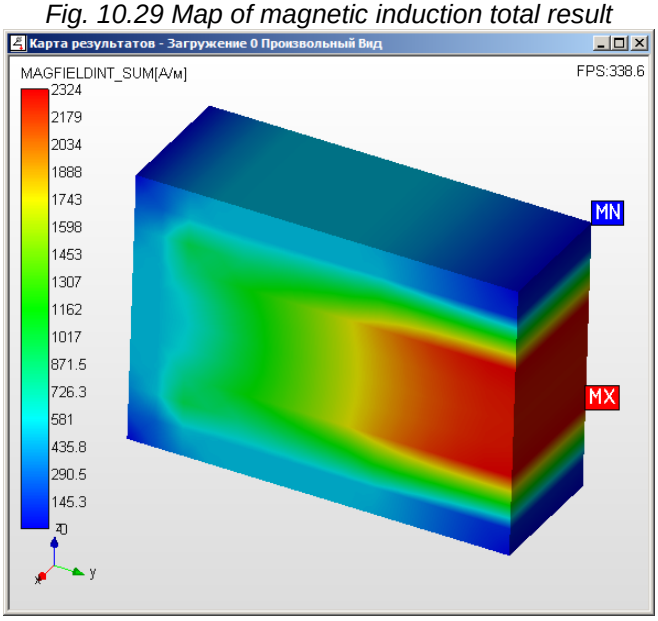


Fig. 10.30 Map of total results of magnetic field force

Fig. 10.31 Map of total magnetic vectorial indu

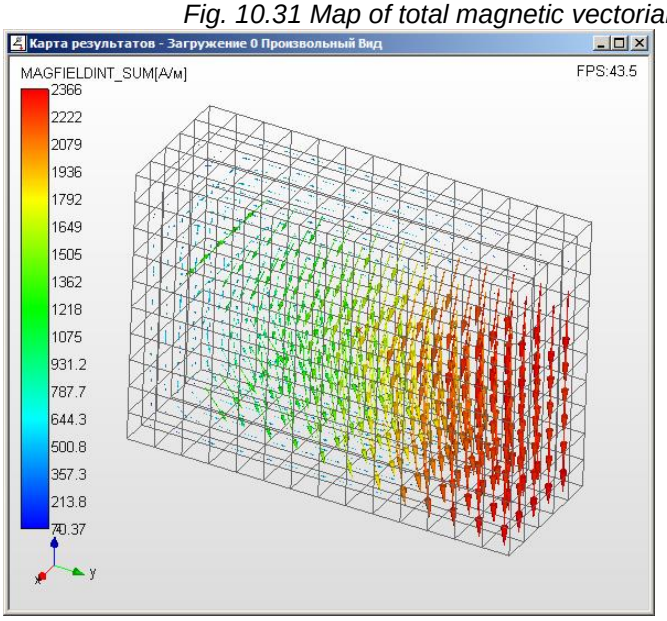


Fig. 10.32 Map of total vectorial magnetic field

Calculation of direct currents field

1. Create a cube of three-dimensional elements by extent 6 x 6 x 6 which will correspond to the area of conductor where one needs to get a distribution of direct currents field.

2. Distribute nodes of this cube to different layers in accordance with fig. below.

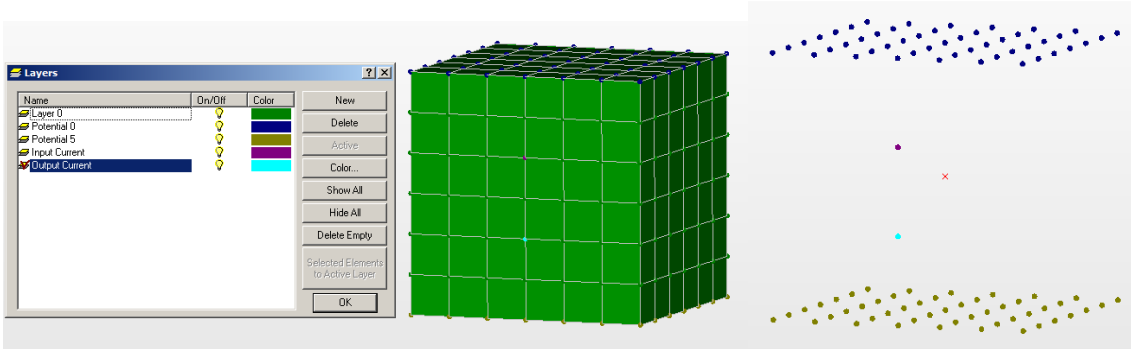


*Fig. 10.33 Different layers of a cube model with coloring in layers. On the right **Layer 0** is turned off.*

The Potential 0 layer – nodes on the upper facet of the cube, **Potential 5** – in the lower , the Layers are **Input current** and **Output current** – nodes on the front side of the cube

3. Create for the steel material the property "Specific electric conductance" with constant value 0,025 1/(the *mm Ohm).

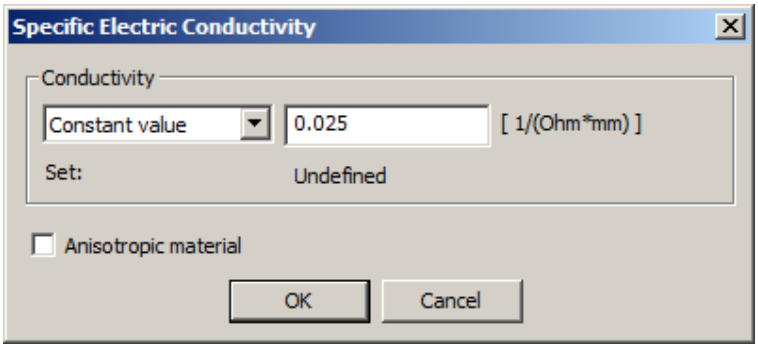


Fig. 10.34 Definition of specific electric conductance of material

4. Specify this material for the whole model, using the section "Assign to All" of the site "Steel" shortcut menu.

5. Deactivate all layers except the layer "Potential 5".

6. Create the electricity constant 5V value load "Electric potential", select all nodes in a visible layer and apply produced load to them.

7. Fulfill the similar acts with the layer "Potential 0". Leave the value of load by default. Result of effects is presented in fig below.

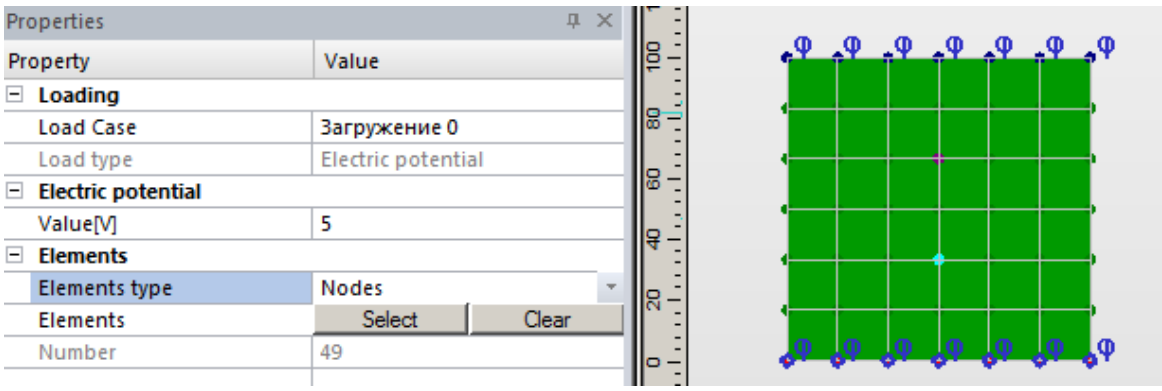


Fig. 10.35 Setting of electric potential to model nodes

1. Create two more loads "Electric current" and place to nodes in layers "Input current" and "Output current". The value of an incoming current is 1A, the value of outgoing current is -1A.

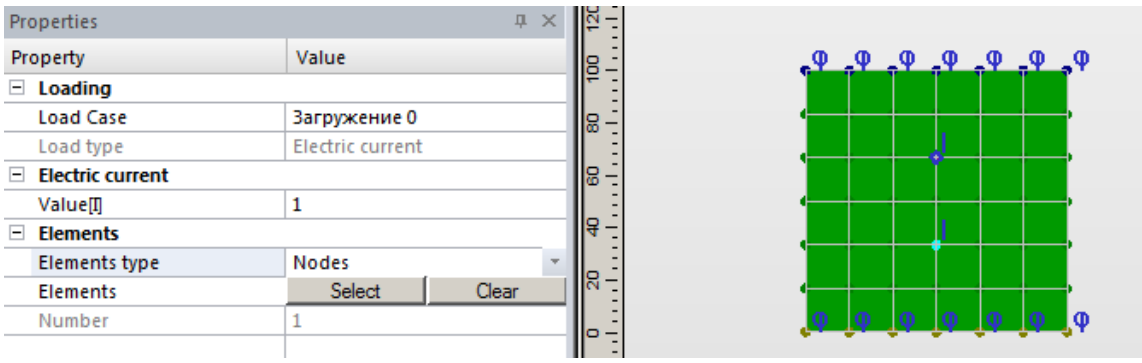


Fig. 10.36 Setting of electric current to model nodes.

2. Carry out a calculation. Select the menu item **Calculations\Calculation…** and choose "**Calculation of direct currents**" with default parameters.

3. After a calculation passing a node with corresponding results will appear in the tree.

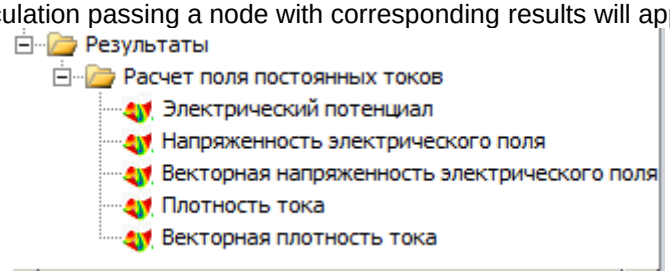


Fig. 10.37 View results in a project tree

The received results are presented in Fig. 4.99 – 4.103.

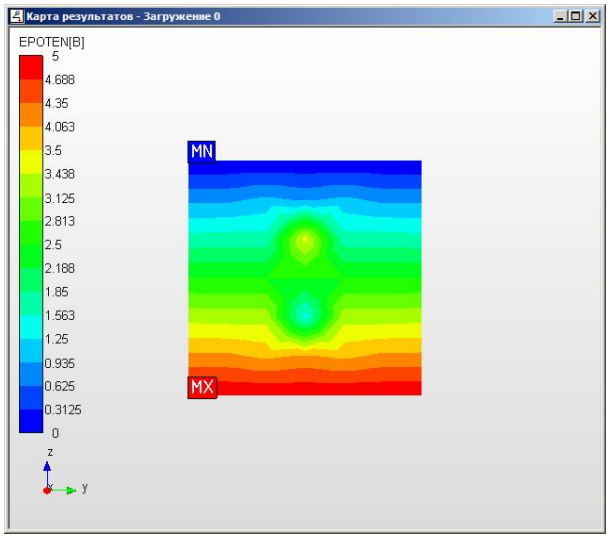


Fig. 10.38 Map of electric potential distribution

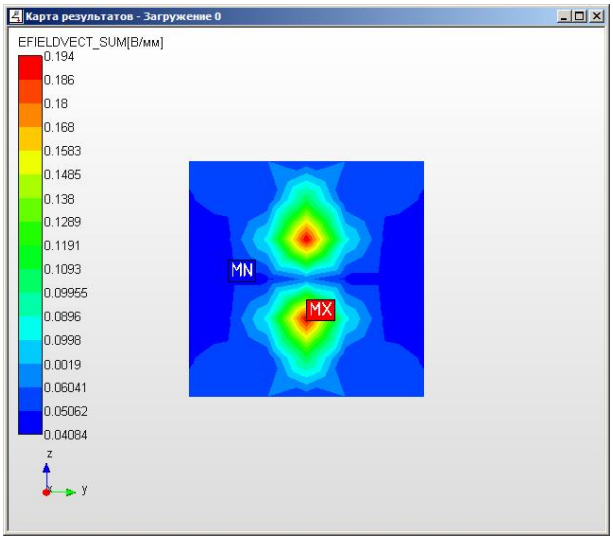


Fig. 10.39 Map of total strength of electric field distribution

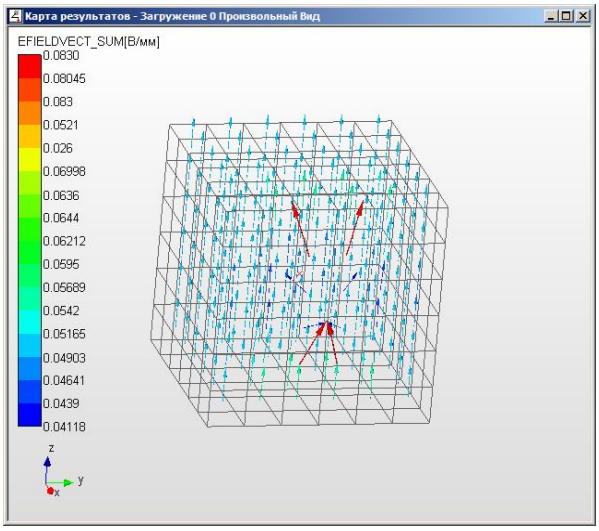


Fig. 10.40 Map of total strength of electric field in vectoral representation distribution

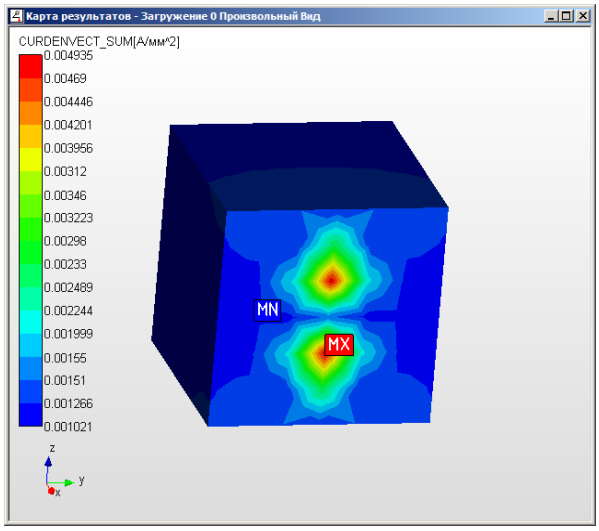


Fig. 10.41 Map of total electric field density distribution

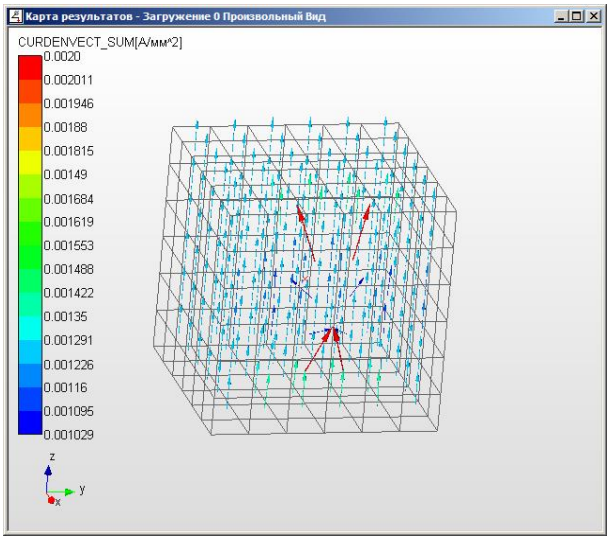


Fig. 10.42 Map of total electric field density distribution in vectorial representation

Ring conductor (Non stationary electromagnetic calculation)

1. Create a model from three-dimensional elements of a semi-ring. In the model the exterior nodes of a toroidal surface are to be placed to a separate layer.

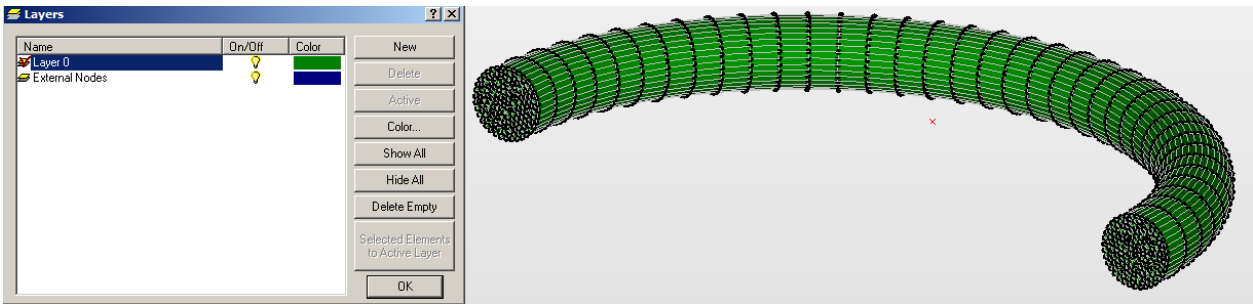


Fig. 10.43 Model with external nodes on a toroidal surface placed to the layer "External nodes"

2. For the material "Steel" set two properties: Relative magnetic permeability with constant value 1 and specific electric conductance, a constant value of 0.25 1/Ohm*m.

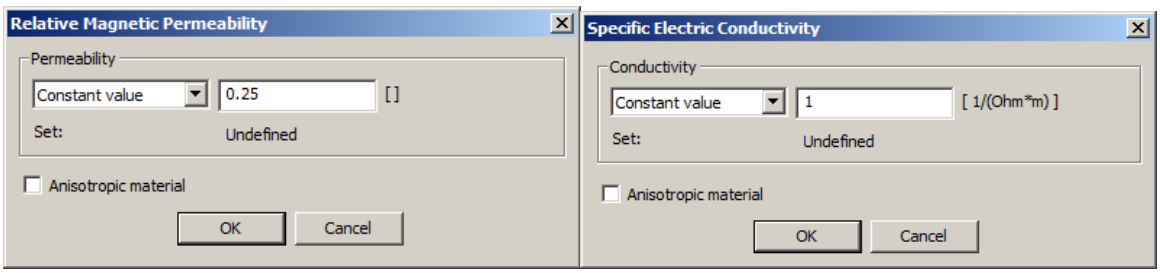


Fig. 10.44 Definition of electromagnetic material properties.

3. Specify material for all the model.

4. Create the electrical load "Input section of current" with a functional dependence of view of sin (2*pi()* time) and place it to all nodes of one the model ends. At definition of a function make sure angle is measuring in radians.

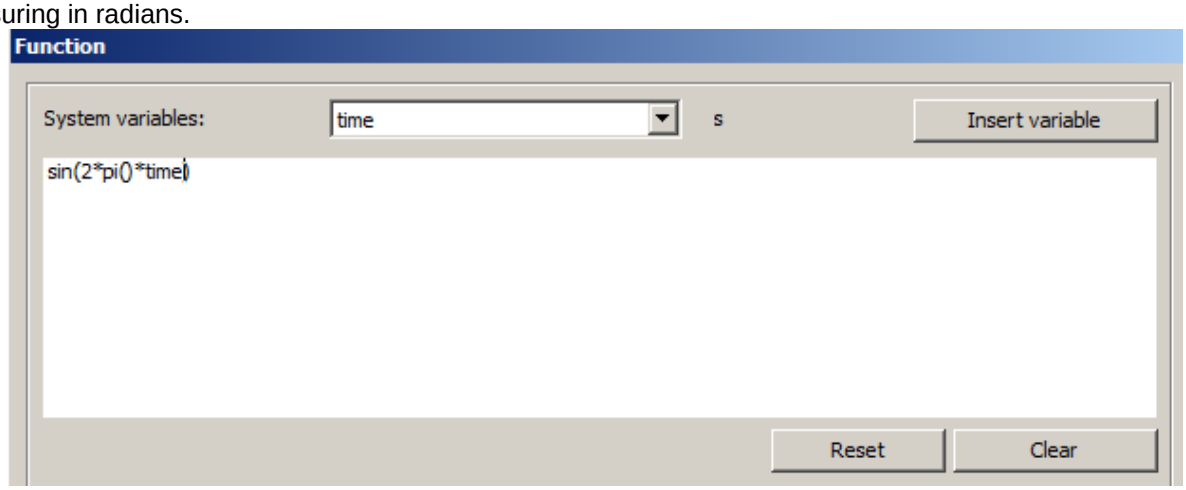


Fig. 10.45 Definition of a functional dependence for the load "Input section of current"

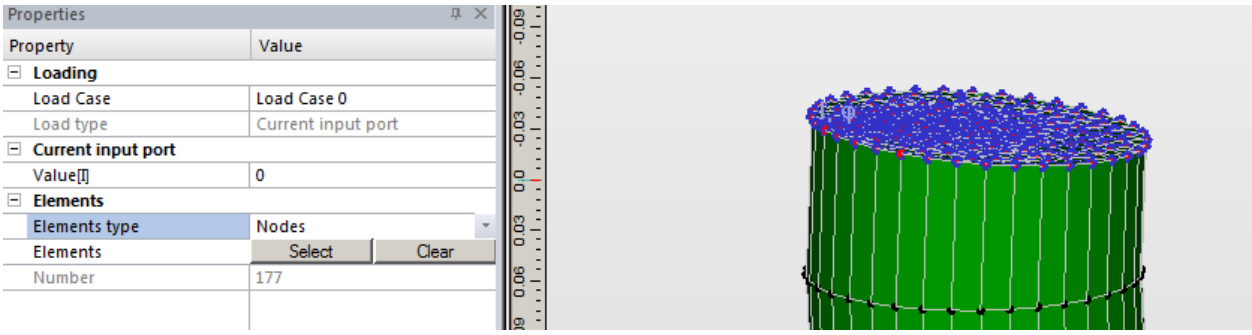


Fig. 10.46 Setting of an input current section to model end nodes

5. Set electric potential 0 V to all the nodes of the opposite end of the model.

6. Leave only the layer "External nodes" activated and specify to all nodes of this layer the load "Vectorial magnetic potential" with the zero values on all components of the vector.

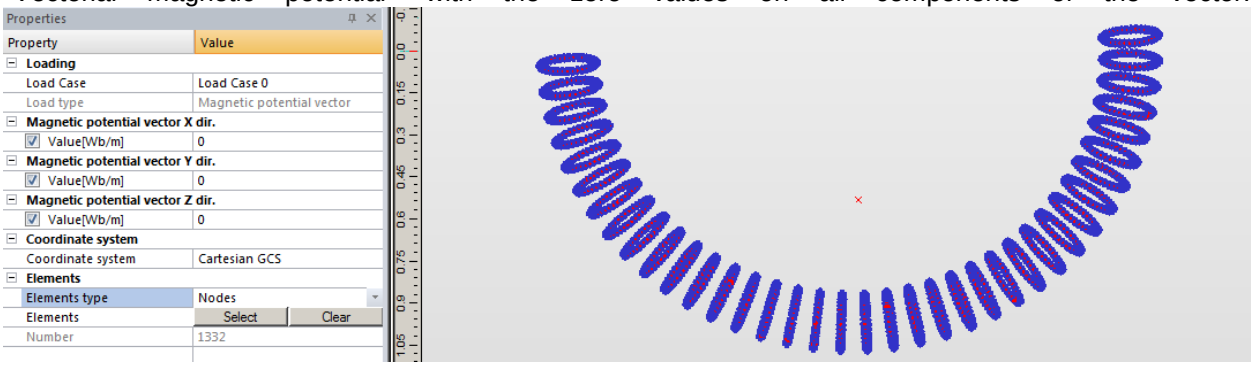


Fig. 10.47 Vectorial magnetic potential applied to external model sites.

7. Deactivate the layer "External nodes" and turn on "Layer 0".

8. Create one more load "Vectorial magnetic potential" with the zero values in directions of X and Z, but the Y direction is not used (for it one needs to deactivate a corresponding switch).

9. Specify this load on nodes of both ends of the model.

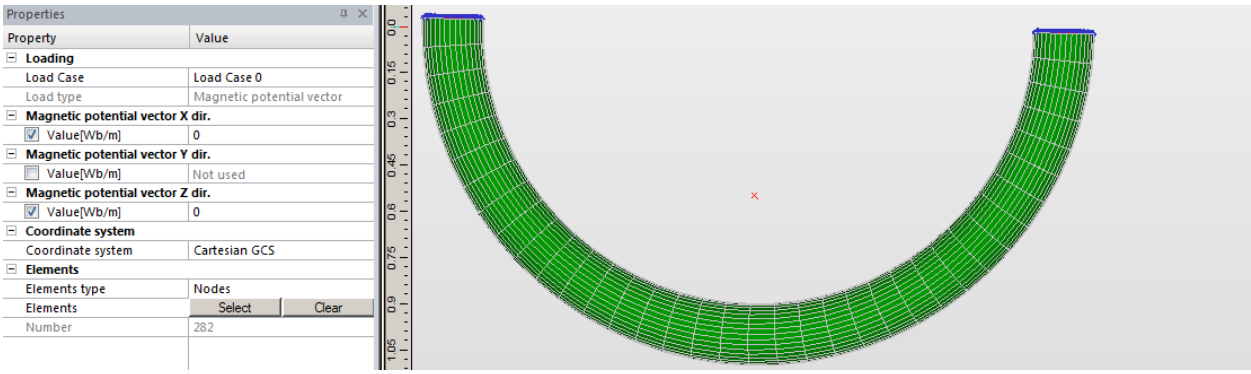


Fig. 10.48 Application of Vectorial magnetic potential on model end nodes.

10. Select "**Stationary electromagnetic calculation**" from the menus **Calculations/Calculation...**. Set the interval 0 until 2 seconds, moments of time as 40 ones. Carry out a calculation.

11. After accomplishment of the calculation corresponding results will be displayed in the tree.

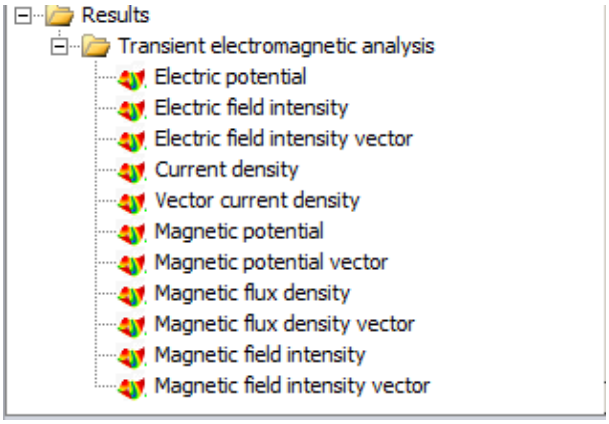
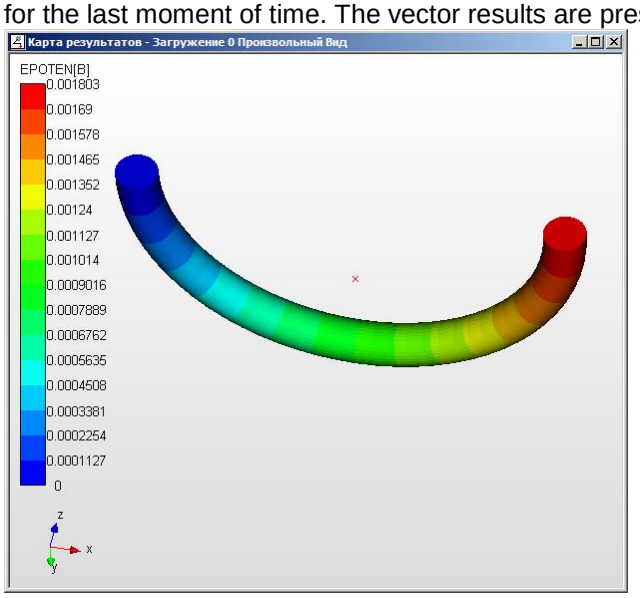


Fig. 10.49 Results of a stationary electromagnetic calculation in a project tree.

12. The detailed results are presented in Fig. 10.50 -10.60. For obviousness the results are given for the last moment of time. The vector results are presented by a part of a model for bigger obviousness.



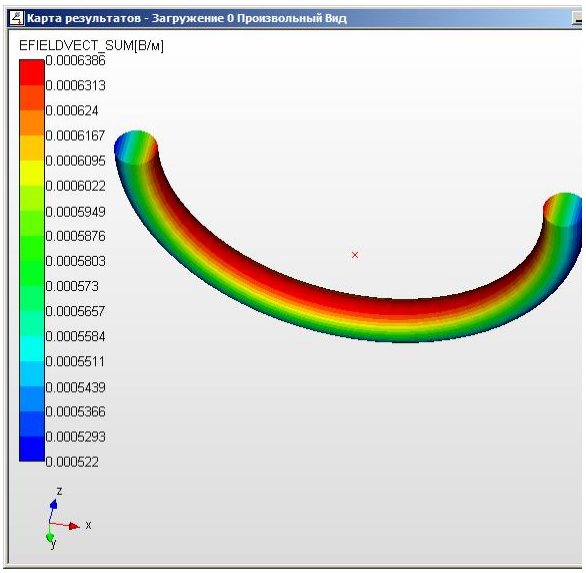


Fig. 10.50 Map of electric potential distribution

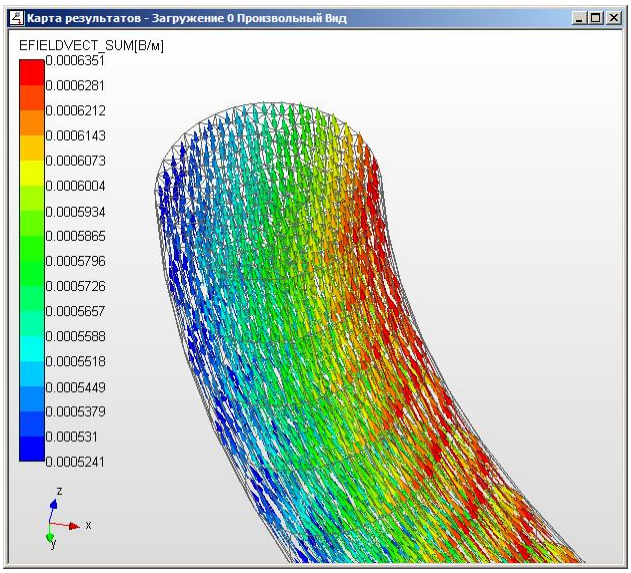


Fig. 10.51 Map of total electric strength field distribution

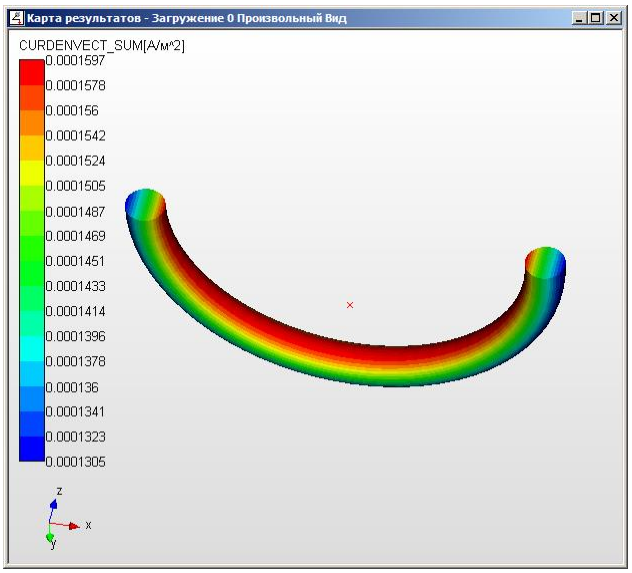


Fig. 10.52 Map of total electric field strength in vectoral representation

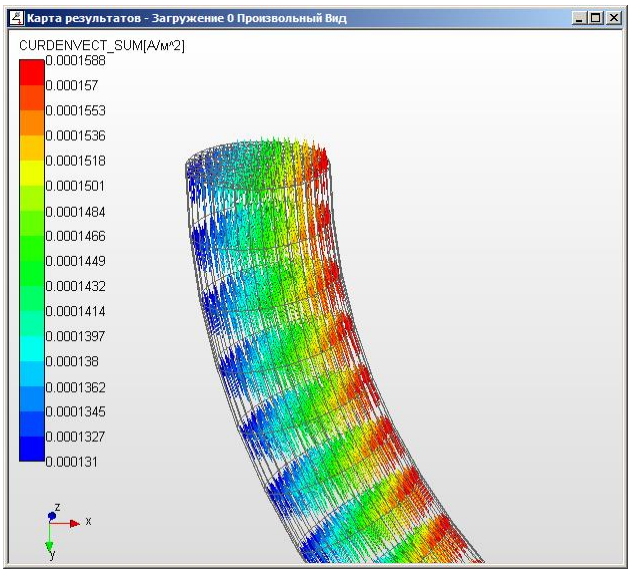


Fig. 10.53 Map of total electric current density distribution

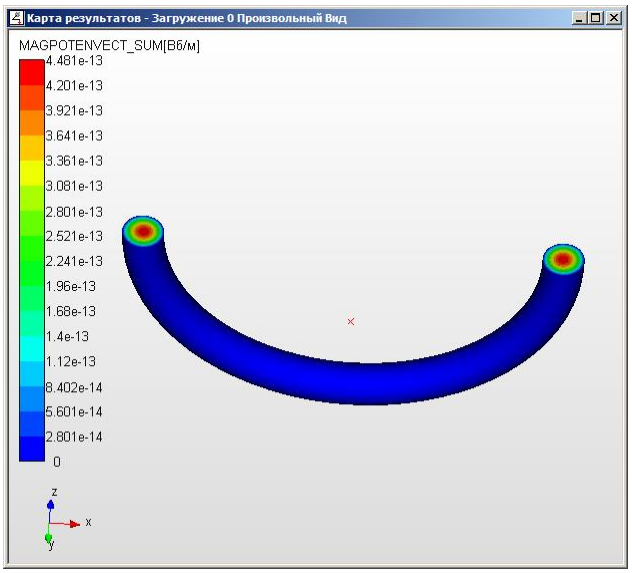


Fig. 10.54 Map of total electric current density distribution in vectorial representation

Fig. 10.55 Map of total magnetic potential distribution

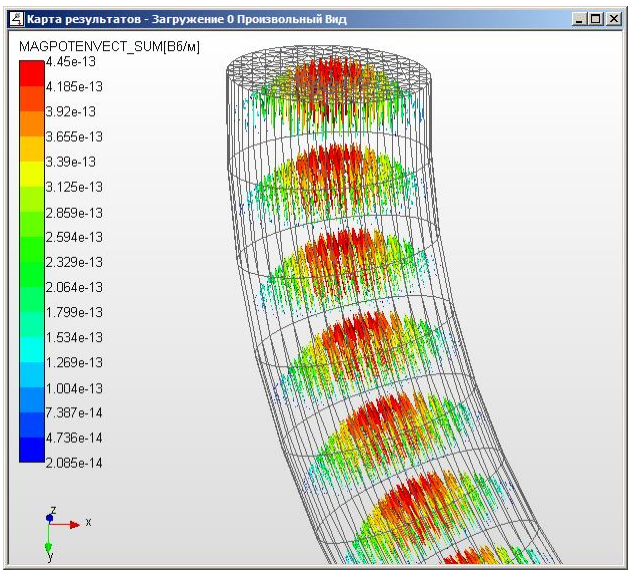


Fig. 10.56 Map of total magnetic potential distribution in vectorial representation

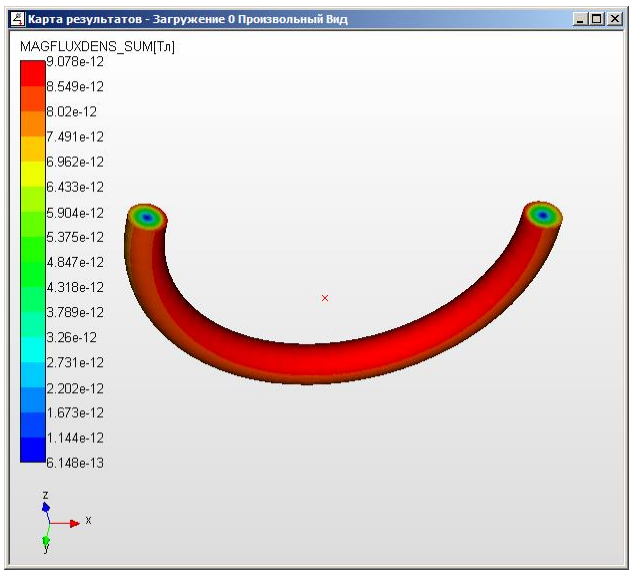


Fig. 10.57 Map of a total magnetic induction distribution

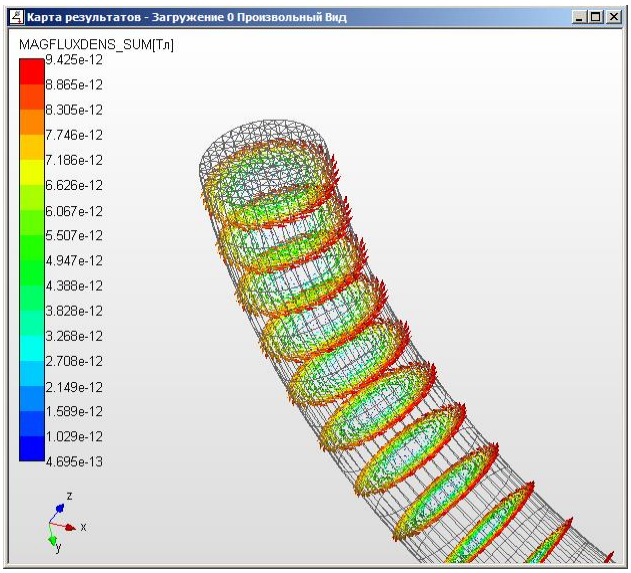


Fig. 10.58 Map of a total magnetic induction distribution in vectorial representation

Fig. 10.59 Map of total magnetic field force distribution

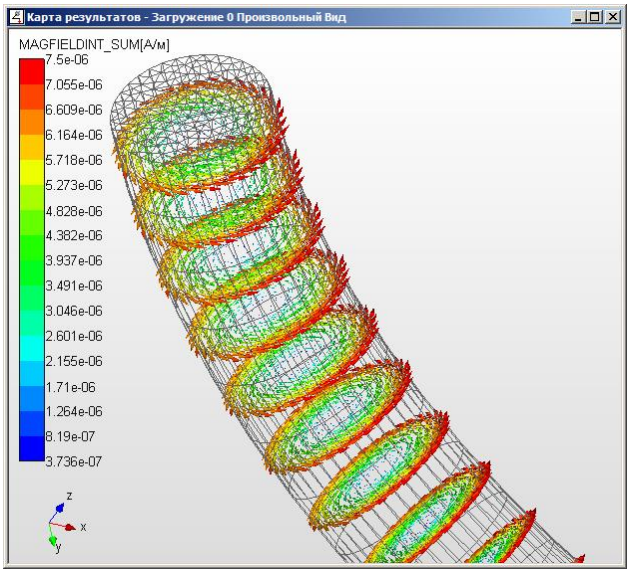


Fig. 10.60 Map of a total magnetic field force distribution in vector representation

A calculation of a waveguide resonator (High frequency modal analysis)

1. A model of the inner cavity of the waveguide segment from three-dimensional elements must be created, see fig. 4.123.

2. For the material "Steel", besides properties of isotropic material, add two more properties: Relative dielectric permeability with constant value 2,05 and relative magnetic permeability with a constant value equal to 1.

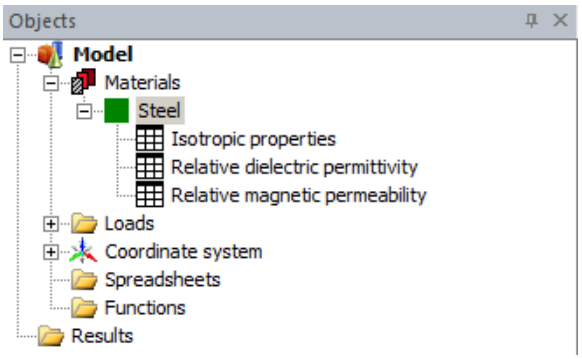


Fig. 10.61 Definition of electromagnetic material properties

3. Set this material for all the model: An item of the shortcut material menu "To assign to everyone".

4. Create the load "Perfect electric conductor". For it select a corresponding menu item in the shortcut menu of the site "High frequency loads".

5. Select a type of edge elements in a "Property" panel and select all the pattern. For this type of load all the outer edges of a model will be selected.

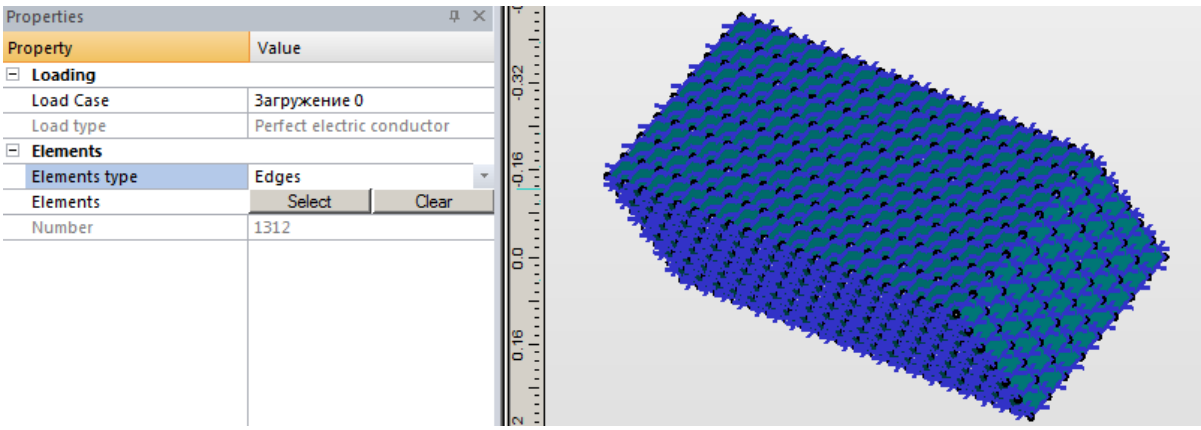


Fig. 10.62 Definition of the load "Perfect electric conductor" of the outer edges of the model

6. Further select an item of the menu "Calculations/Calculation…" and specify high frequency modal analysis with parameters by default.
7. After implementation of analysis corresponding results will appear in the tree.

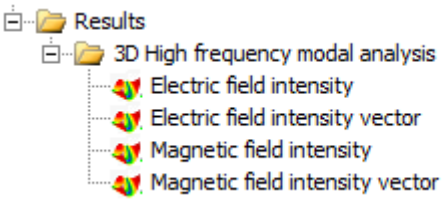


Fig. 10.63 Showing results in the project tree

In greater detail the results are presented in Fig. 10.64 – 10.71.

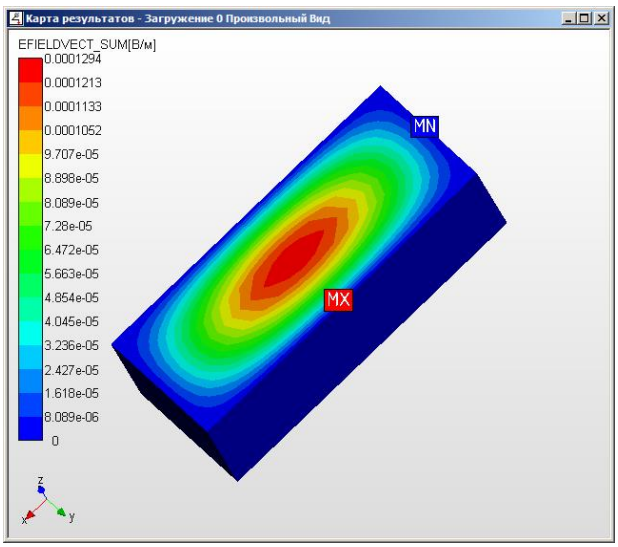


Fig. 10.64 Card of total electric field strength at frequency of 2,83497 e+08 Hz

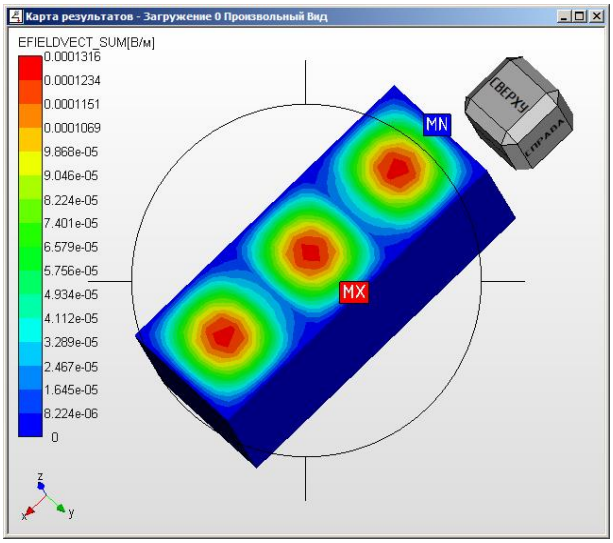


Fig. 10.65 Card of total electric field strength at frequency 4, 12153e+08 Hz

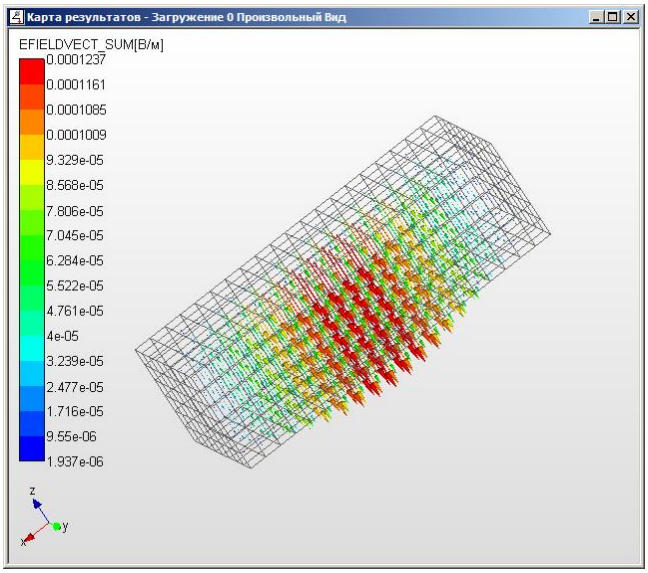


Fig. 10.66 Card of vectorial strength of total electric field at frequency 2,83497 e+08 Hz

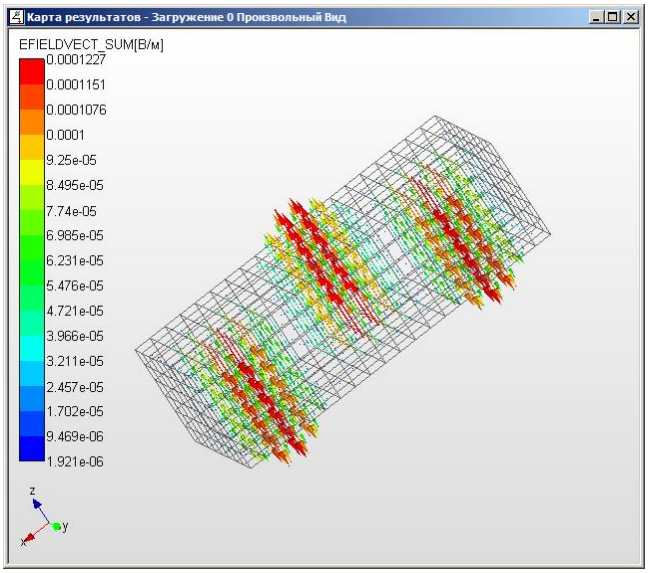


Fig. 10.67 Card of vectorial strength of total electric field at frequency 4, 12153e+08, Hz

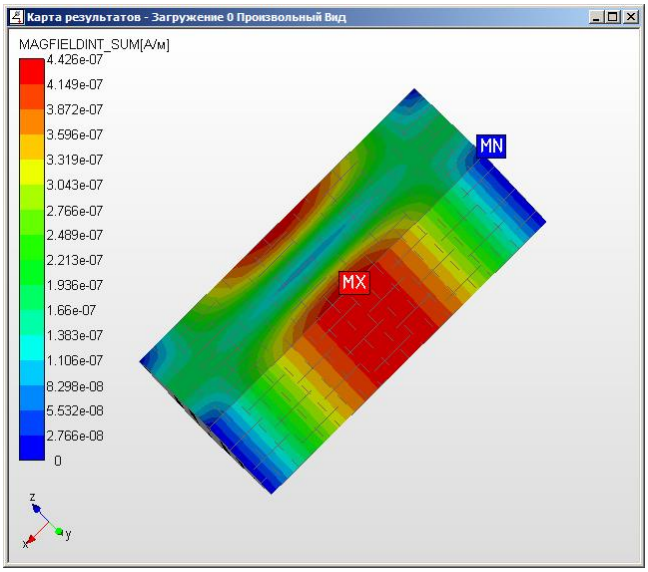


Fig. 10.68 Card of total magnetic field force at frequency 2,83497 e+08 Hz

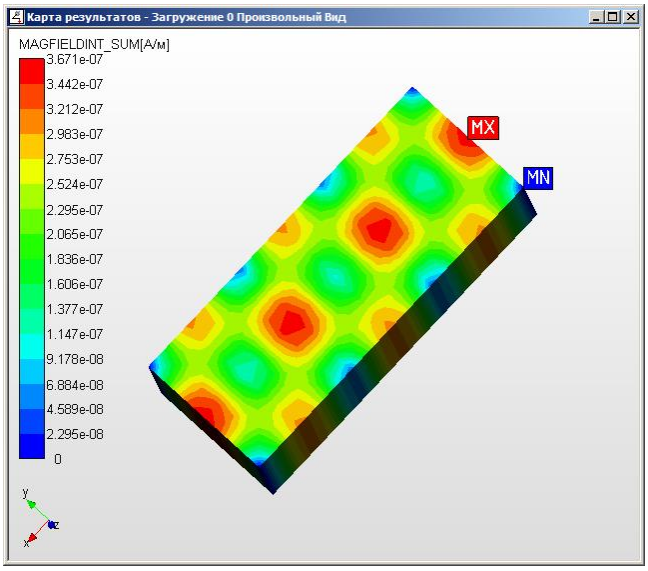


Fig. 10.69 Card of total magnetic field force at frequency 4, 12153e+08, Hz

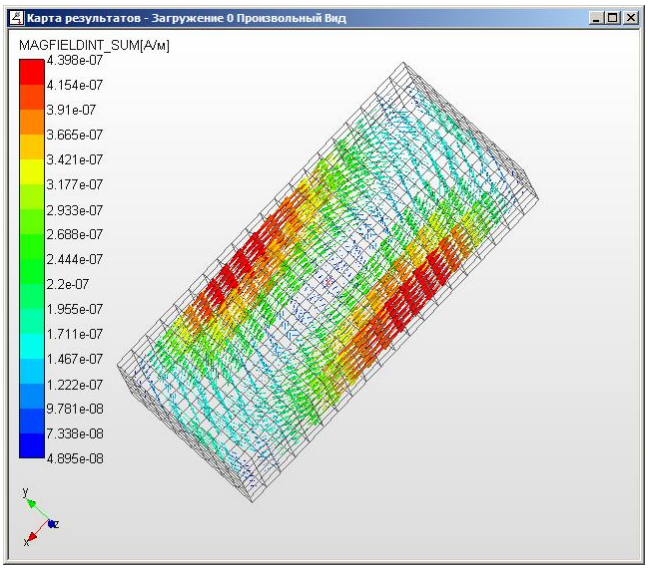


Fig. 10.70 Card of vectorial total magnetic field force at frequency 2,83497 e+08 Hz

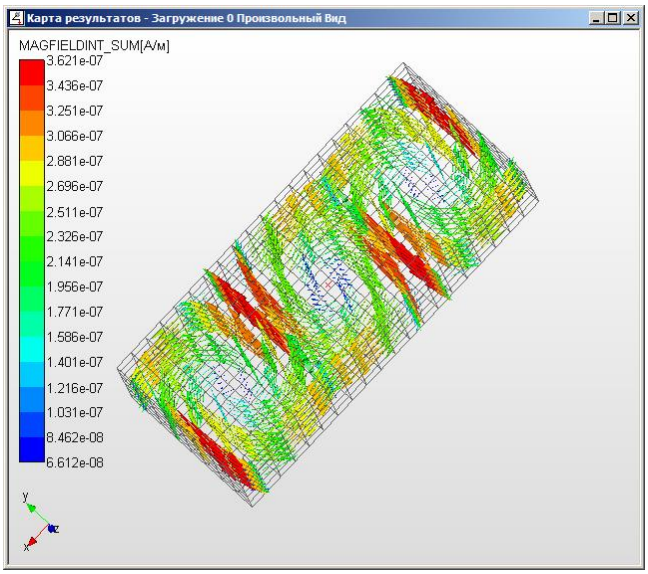


Fig. 10.71 Card of vectorial total magnetic field force at frequency 4, 12153e+08, Hz

Having pushed on the header of a table column, one can sort supports in the ascending/descending order of a reaction in a selected table column.

Non-mating by forces and moments (sum of reactions and external efforts) is derived in a global coordinate system. This table can also be routed to a printer or to a file in RTF.

The show of reactions vectors is vector map and values for the selected reactions in the table of support.

Filters – allow incl. / off. display pillars in certain areas of GCS and unilateral pillars.