



LIRA-SAPR

Unified intuitive graphical user interface VISOR-SAPR

Unified – in this environment the user could go through all stages of solution: from generating design model to evaluating analysis results; it is also possible to switch to any selected mode and receive information from any stage, to present the windows of several modes simultaneously (for example, input and output data).

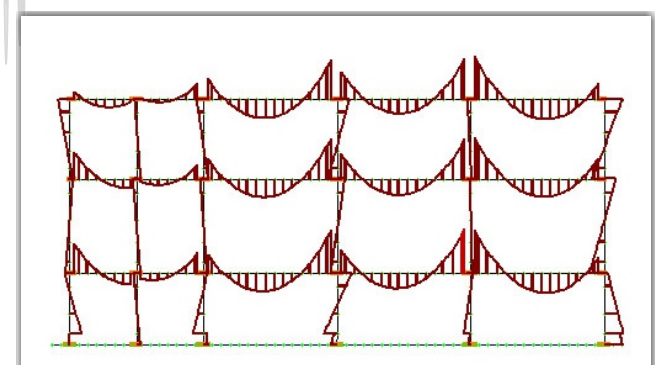
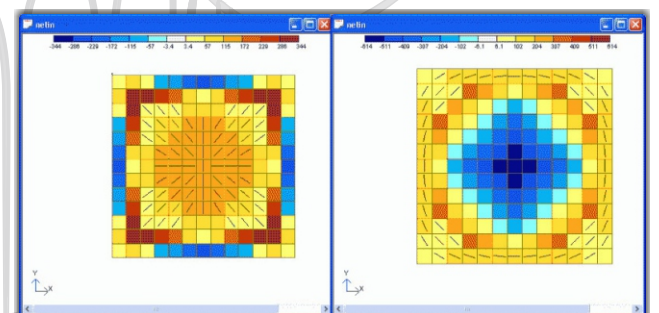
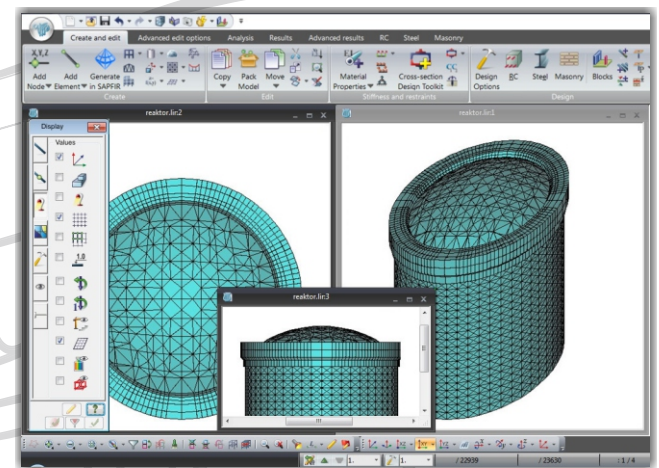
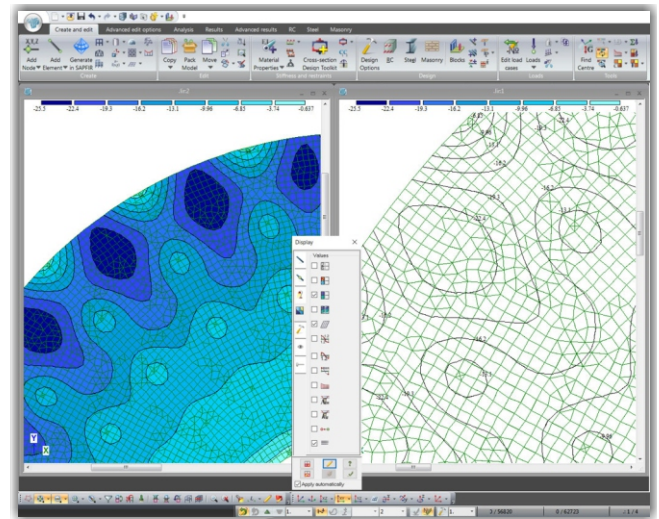
Intuitive - program interface is organized according to elements and technique for structural analysis and design. The user familiar with this technique could easily work with the program intuitively.

Graphical – for the most part, the project is presented graphically (visualization of objects in general and their parts, analysis results as deformed shapes, diagrams, contour plots, animation of dynamic processes, etc.).

The program contains a wide set of tools that enable the user to generate and evaluate computer models of arbitrary structures.

Main features:

- design model may be visualized at all stages of its generation and analysis;
- different ways for model generation and presentation (filters, flags of drawing, fragmentation, generation of standard and user-defined sections), multi-language support, different systems of measurement units, etc.;
- design model is generated with account of physical (material) nonlinearity, geometrical nonlinearity, super-elements, DCL (design combinations of loads), DCF (design combinations of forces), dynamic loads, special FE;
- multi-variant analysis (after static analysis it is possible to modify building codes, materials, element sections);
- analysis results may be evaluated in different ways (contour plots and contour lines for stress and displacement, diagrams of forces, animation of vibrations, deformed shapes, elements and their attributes are presented either in colour or with numerical labels);
- tracing routine for all stages of analysis in solver;
- error diagnostic;
- extensive context-sensitive help;
- comprehensive Help system and Tutorial.





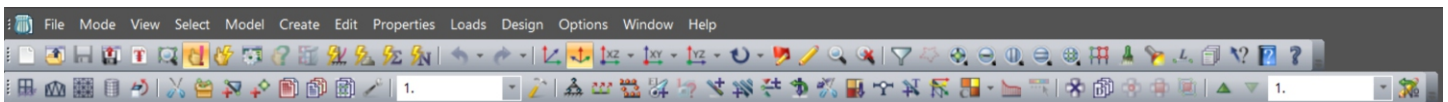
LIRA-SAPR®

Unified intuitive graphical user interface VISOR-SAPR

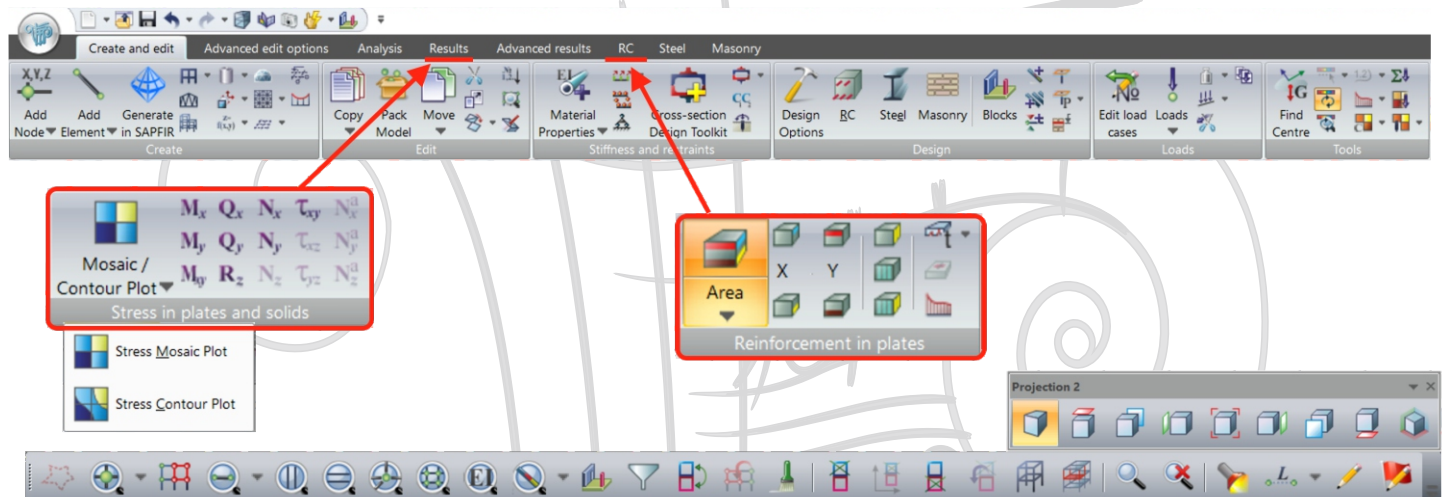
LIRA-SAPR program presents new alternative type of user interface "Ribbon" that helps the user to generate, edit and evaluate design models in a more intuitive way. The new ribbon interface is more elegant, concise; it also retains standard commands. Disadvantages of classic interface are considered in the work on ribbon user interface. Several hidden useful options are now at hand, so the program becomes more user-friendly. Nevertheless, we consider the interests of those users who have been with us for a long time, so the classic interface is still available in the program.

Classic User Interface

Continuity with previous versions LIRA 9.0, 9.2, 9.4, 9.6

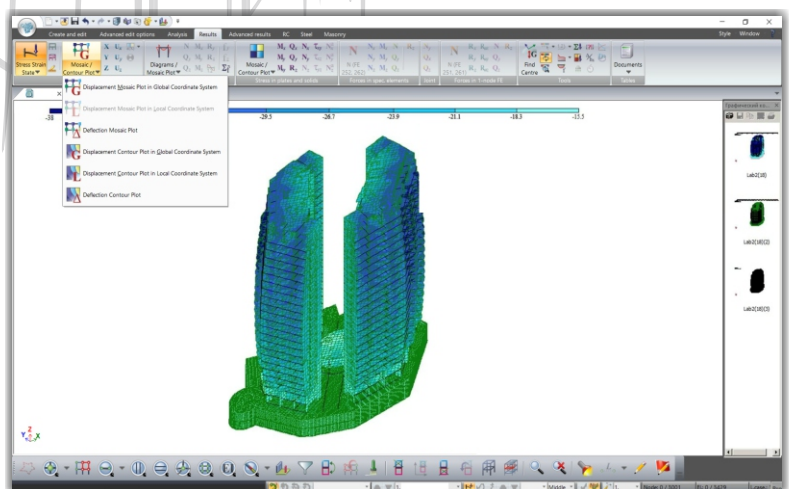


Alternative Ribbon User Interface



Program interface enables the user to adjust the following parameters of working environment:

- toolbars (content, location on the screen, visibility option);
- ribbon styles 'Ribbon RC', 'Ribbon Steel', 'Ribbon Plus' and 'User-defined ribbon' (the user obtain quick access to many tools in order to evaluate analysis results of design modules in more detail);
- shortcut keys for commands; Quick Access Toolbar may be customized by the user;
- colour palette (background of working area, colour for compound elements and objects of design model, colour for contour plots of analysis results);
- mouse pointer (rotation of design model, zoom and pan options);
- font size and type;
- path to base directories where input and output data is saved;
- units of measurement;
- language for user interface and documentation;
- 3D-graphics (software or hardware acceleration, visual effects, etc.)





LIRA-SAPR

Analysis and design of the 'SOIL-STRUCTURE' interaction

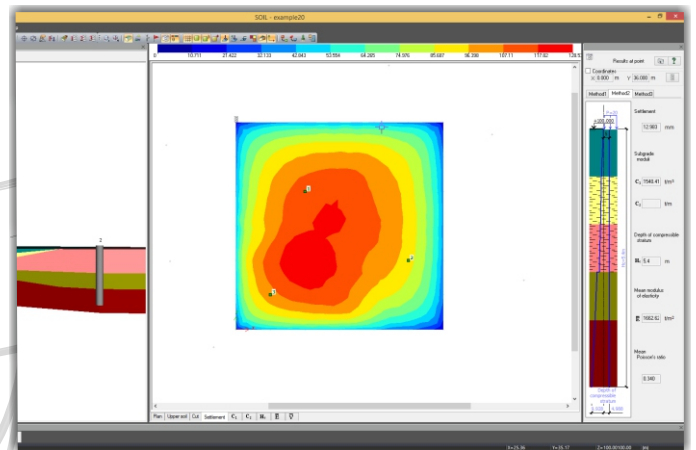
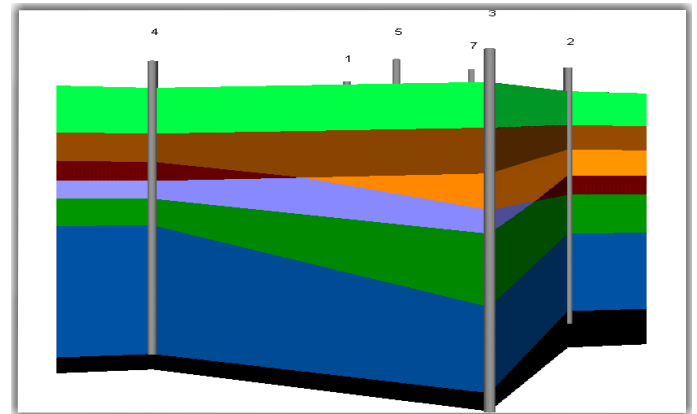
SOIL module

3D soil model is generated according to data from geological survey (location and soil properties in the specified boreholes). It is also possible to compute moduli of subgrade reaction that vary across the area of foundation slab. Neighbouring structures as well as loads on foundation slab are considered. Depth of compressible stratum and the settlement are computed.

The user could preview location of layers in arbitrary vertical and horizontal cuts of soil and contour plots for subgrade moduli.

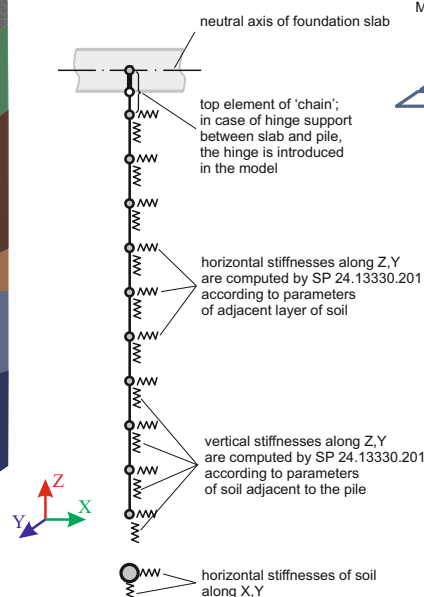
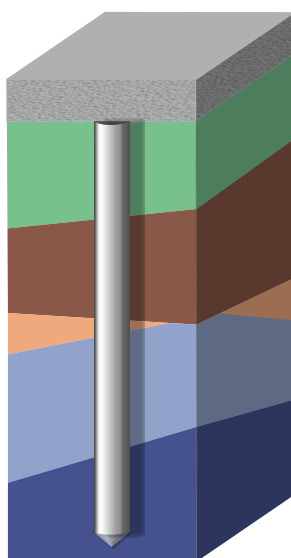
Values of subgrade moduli for every FE are automatically transmitted to the whole computer model for further analysis of structure together with soil.

Based on 3D soil model, it is possible to generate FE model and compute stiffness of every FE depending on its location in different layers of soil. This model may be used to analyse the system 'overground structure – foundation – soil'.

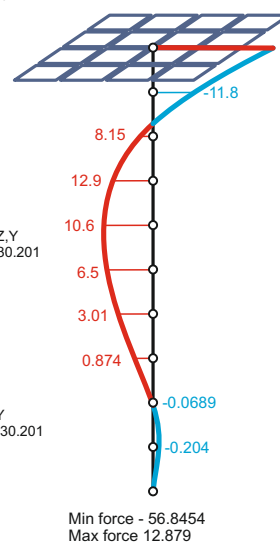


Computation of pile stiffness

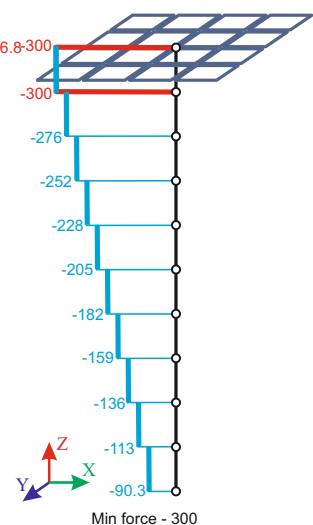
Pile may be simulated with a set of bars with springs of finite rigidity introduced at intermediate nodes. These springs simulate combined behaviour of pile and soil. Stiffness of spring is computed automatically according to soil properties at soil layers adjacent to the pile. Mutual interaction of neighbouring piles is considered. After analysis procedure, the output data contains diagrams of forces M , M_y , M_z , Q_y , Q_z along the pile length. It is also possible to analyse reinforcement for the pile.



Load case 1
Diagram M_y
Measurement units - t*m



Load case 1
Diagram N
Measurement units - t





LIRA-SAPR®

Multi-functional solvers

LIRA-SAPR includes a set of solvers for different types of problems. All solvers work on 64-bit and 32-bit operating systems and take full advantage of multicore computers. The procedure of solving algebraic equations that realize modern concepts for solving of system of linear equations forms the basis of all solvers. It ensures that the problems with great number of unknowns will be solved in the shortest possible time.

Method of super-elements makes it possible to remove any limitations on the size of problem.

Linear solver is mentioned to carry out linear analysis for wide set of problems on static (force and deformed) and dynamic loads.

FE library of linear solver contains extensive set of finite elements that enable the user to simulate arbitrary structural models.

Nonlinear step-type - iteration

solver is mentioned to carry out nonlinear analysis for wide set of problems.

Modules of physical nonlinearity on the basis of different nonlinear relations (including account of downward path, unloading path that is different from loading one). These modules enable the user to simulate the loading process for both mono- and bi-material RC structures and to trace crack propagation, creep strain and yield until the complete damage of the structure.

Modules of geometrical nonlinearity enable the user to analyse both geometrically stable structures (flexible plate, shells, trusses, etc.) and geometrically unstable structures that take load due to considerable change in its initial shape (separate ropes, guy trusses, guy shells, tent structures, membranes).

Modules for account of structural and genetic nonlinearity (one-way restraints, time varying structural models, etc.).

Dynamics-plus solver enables you to perform time history analysis of structures according to defined different types of time varying loads (accelerograms, forced vibrations, impact, etc.). In this case, physical and geometrical nonlinearity, material damping of the structure and soil, structural dampers are considered.

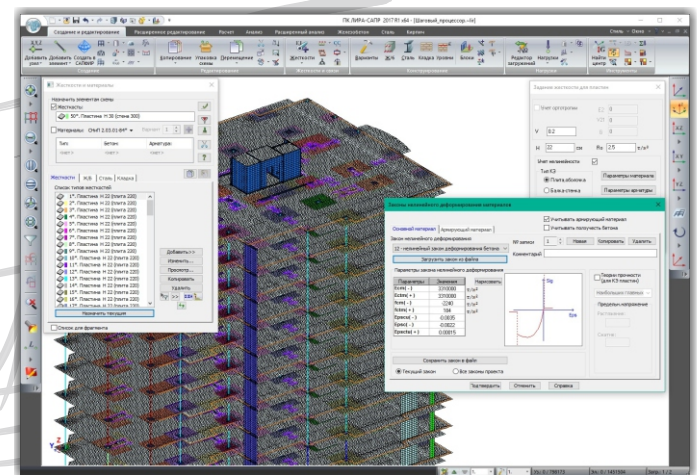
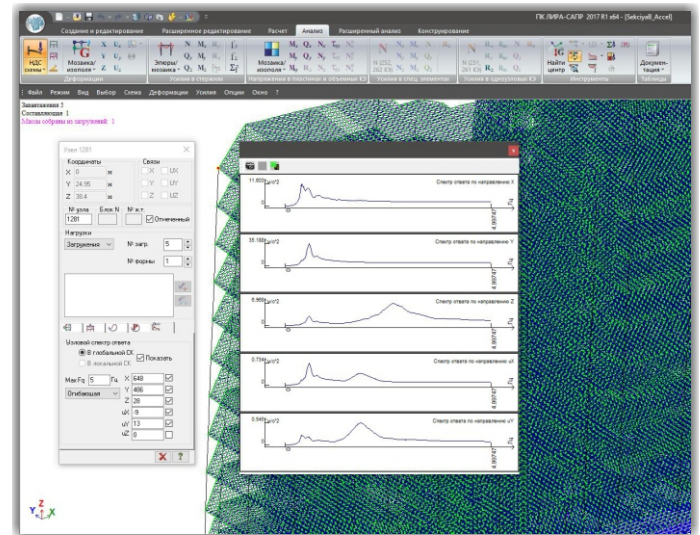
Special solver **ASSEMBLAGE-plus** enables you to simulate erection (assemblage) of the multi-storey buildings with account of frequent changes in design model. For monolithic reinforced concrete structures it is allowed to disassemble formwork, temporary supports, change stiffness and strength of concrete due to temporary freezing of concrete mix and other factors

It is possible to simulate changes in the stress strain state of the structure in time caused by **Creep** (including thermal creep) of concrete.

PUSHOVER solver enables you to carry out nonlinear analysis on earthquake loads according to DBN B.1.1-12:2014

Thermal conductivity is computed: thermal fields are distributed, then stress strain state of the structure is computed.

NonLinear Engineering solver enables you to consider reduced stiffness of RC elements in analysis on all loads and then compute DCF, DCL.





LIRA-SAPR

Document system

New document system **Report Book** includes interactive screen copies of design model and conceptually new tables with input data, output data from FEA and analysis of RC, steel & masonry structures. When you save the screen copy, the state of design model is saved together with the image presented on the screen. It is possible to return the model to this saved view or fragment (with its state) at any time. New tables of *.csv format provide comprehensive evaluation of output data in tabular form. The contents of the screen copies as well as tables may be automatically updated following the changes in design model. Report Book enables you to organize your elements into hierarchical tree-structured database, add arbitrary text and graphic objects. It is possible to arrange elements of Report Book as single file of DOCX format and then print them.

Documentation is generated in less time due to templates. Documentation templates are available for any fragment of the model and structural block. The program contains a set of standard templates. The user could either modify these templates or add the new templates for documentation.

The autotext and repeater options considerably simplify the work with the report. The user could define that names for images of the model should be assigned automatically (e.g. by levels and axes of the fragment and factors of stress strain state presented on the image).

Cross-section Design Toolkit

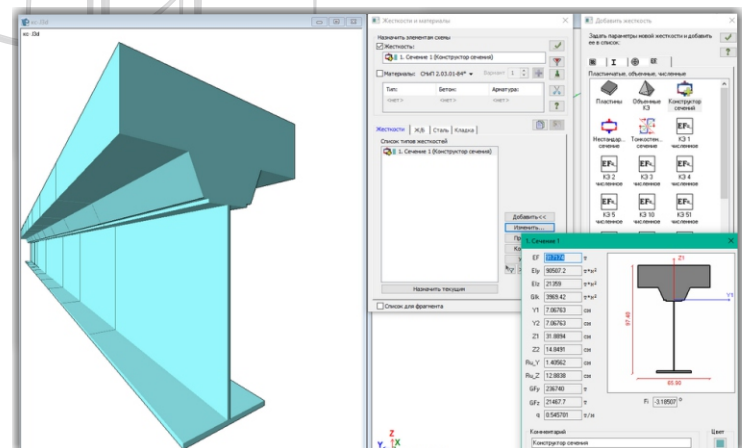
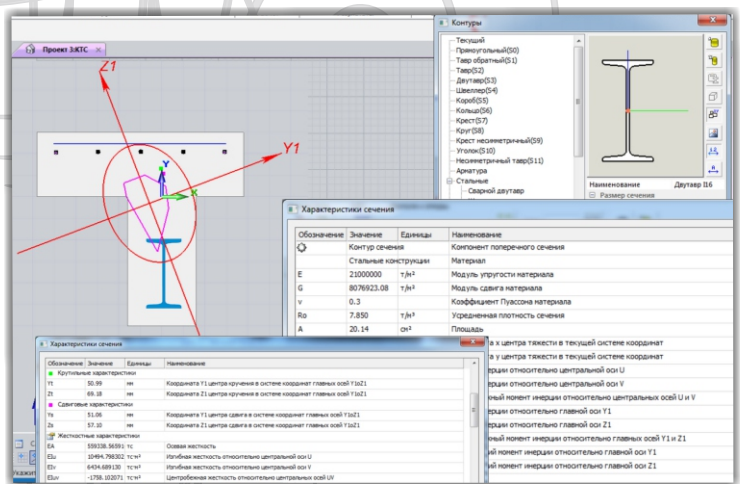
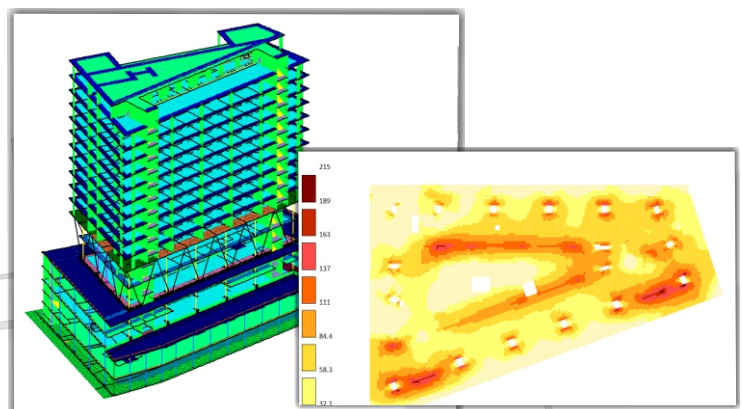
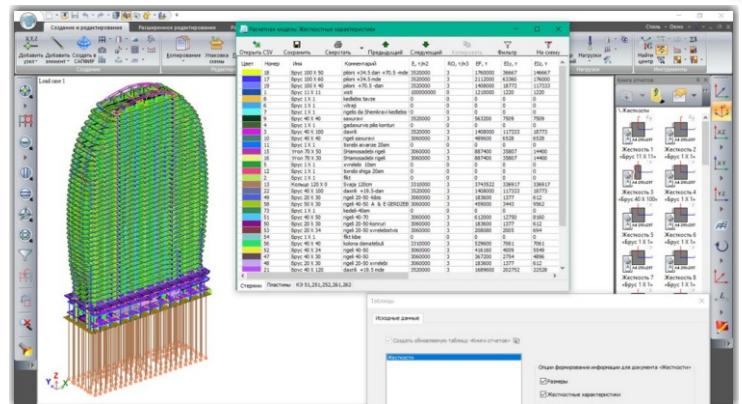
Stiffness parameters are computed: flexural, torsional, shear, warping for mono and multi-material arbitrary sections. Sections may be solid, thin-walled and combined. It is possible to include strip elements, rolled shapes and reinforcement.

If forces on section are defined, stresses (normal, shear, equivalent) across the section area are computed by different criteria of rupture.

Stresses are visualized as contour plots with different colour palettes for inclusions with different stiffness parameters and as diagrams by user-defined lines.

Vectors (direction) of principal and equivalent stress are visualized.

User-friendly interface based on SAPFIR tools.





LIRA-SAPR

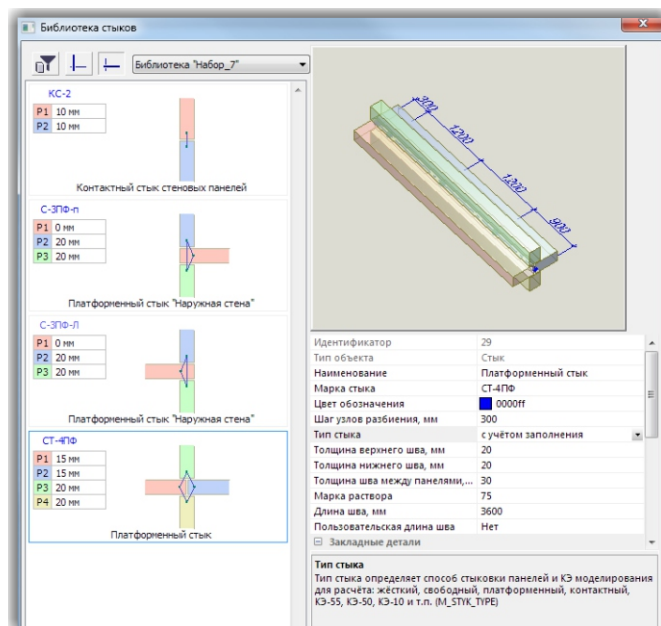
Additional special solvers and graphic modules

Large Panel Buildings

Analysis model of the large panel building is generated. Analysis is carried out, and then you could obtain parameters of the stress strain state for elements of the large panel building. Constantly enlarged and editable library of joint types is provided. This library ensures adaptability of the system to new types of large panel buildings.

The library contains different variants of the following types of joints: platform joint, contact joint, vertical joints of wall panels with or without embedded items, etc. According to selected type, the user could compose certain samples of joints and place them to the building model.

Linear and nonlinear analyses are available for large panel buildings. The FE library contains new elements of panel joints. Nonlinear analysis enables you to carry out analysis by step-type method (simulation of loading process) and iteration method based on NonLinear Engineering Design concept (NL Engineering). With this concept it is possible to carry out analysis in standard method (analysis on several load cases, composing DCF and DCL, analysis of reinforcement, structural elements of joints and embedded items) with indirect consideration of nonlinear behaviour of structure. After analysis you will obtain all parameters of the stress-strain state of elements of the large panel building including diagrams of contact stresses in joints of the building.

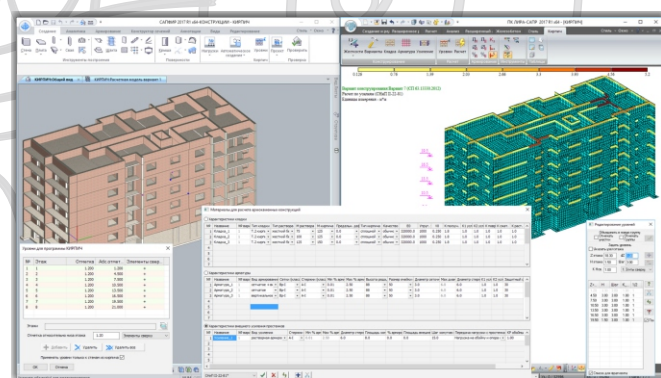
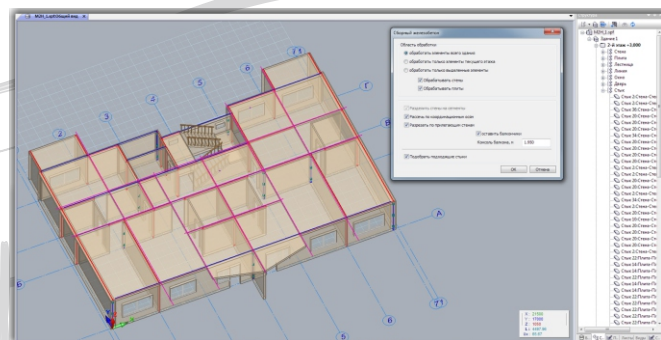


Masonry structures

The model is generated in preprocessors SAPFIR or VISOR-SAPR. The user assigns horizontal levels in structural model (at the elevation of min section of pier, at elevation where floor slabs are supported, etc.), the strength of masonry is checked at these levels. When forces are computed by the solver, combined 3D behaviour of load-bearing brick and reinforced concrete elements of the building is considered. Required number of mesh reinforcement as well as rebars of vertical reinforcement is determined. It is possible to analyse several variants for different division of wall segments in different design options defined by the user.

For every level it is possible to obtain the sketch of working drawing with number of masonry courses in which it is necessary to place reinforcement meshes.

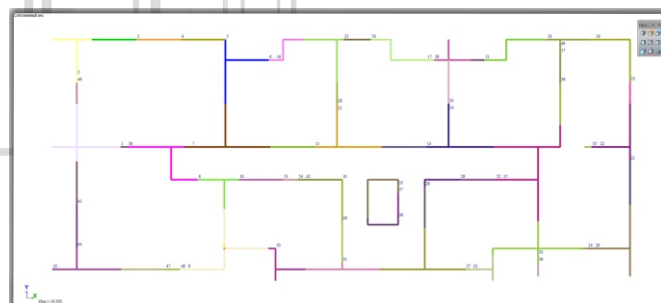
In the same project it is possible to define different types of masonry structures, different types of stone, slag stone, shell limestone, tuff, etc.



METEOR (Method of unified complete result)

Enhanced options for MODEL VARIATION system

The system enables the user to merge problems with the same topology – nodal coordinates, FE model, geometry of sections. Problems may differ in load cases, stiffness and boundary conditions.





LIRA-SAPR®

Design modules for Steel Structures

STC-SAPR

Analysis of elements in steel structures

Selection and check of elements of steel structures and their joints according to ultimate and serviceability limit states.

Selection procedure - for every element of steel structure included into the building model the program selects steel cross-section (of min area) capable to take loads defined for design model. To reduce the number of selected cross-sections, the elements may be either united into structural elements or unified.

Check procedure - the program checks that steel structures defined for design model will take the load and satisfy requirements of all building codes.

Analysis of steel structures is carried out based on the normative data about design parameters of steel, dimensions and geometric properties of the steel sheets and profiles. The user could either add or edit this data in the SRS-SAPR module - Steel Tables (Steel Rolled Shapes).

The steel sections subjected to corrosion may be also checked.

The program supports the following building codes: DBN B.2.6-198:2014, SNIP II.23-81*, SP 16.13330.2011, Eurocode 3.1.1 ENV 1993-1-1:1992, LRFD (AISC) 2nd edition. Steel elements with the following cross-sections may be analysed: I-section, built-up I-section, T-section, angle section.

Analysis of elements in steel structures is available for the following cross-section types: I-section, built-up I-section, T-section, angle section, double angles, channel sections, built-up channel section, C-shapes section, double channels, closed section, laced and battened section, solid (square, round and rectangular) section and rope (more than 30 types of sections).

Analysis and check are carried out for the following types of joints in steel structures: beam-to-column connection (pinned joint), beam-to-column connection (rigid joint), beam splice, beam-to-beam joint, column splice (slip resistant connection), column base (pinned joint), column base (rigid joint), brace joint, beam flange joint, column flange joint, beam-to-column flange joint, truss joints, truss joints from pipes (more than 70 prototypes of joints).

SRS-SAPR

Steel Tables (Steel Rolled Shapes)

In this module you could create new and edit existing steel tables of rolled and welded sections. Extensive database of steel tables and shapes of CIS, Europe and the USA is provided.

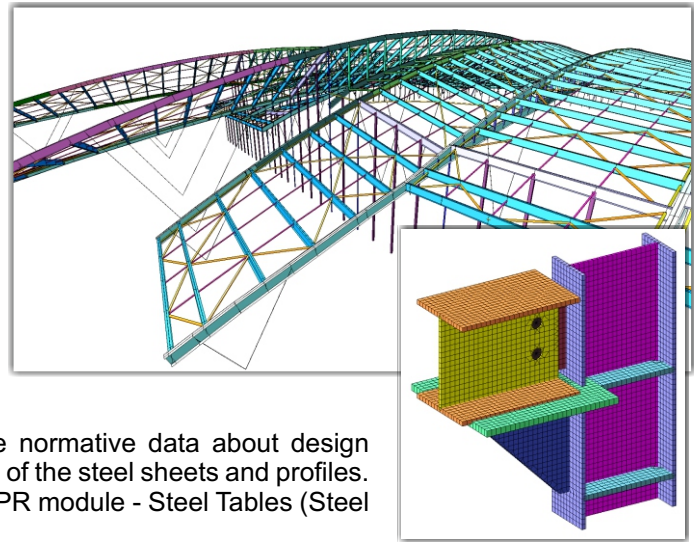
KM-SAPR (available only in Russian)

















Design of steel structures

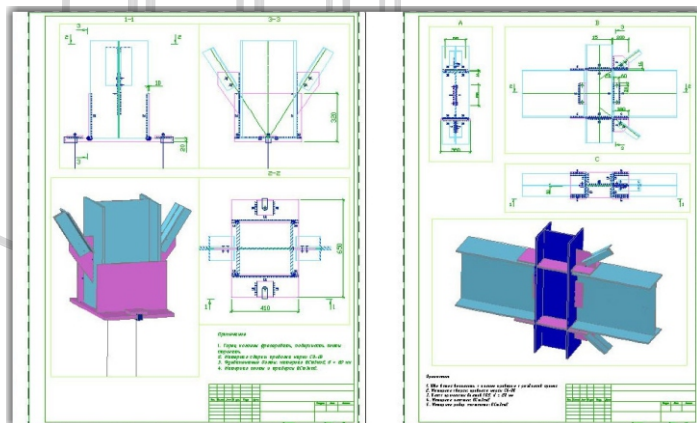
KM-SAPR module is mentioned to automatically generate the whole set of working drawings KM (for reference, it is called 'drawings of steel structures' in Russian).

According to the data imported from VISOR-SAPR module (design model, analysed and unified element sections, analysed and unified joints), design model is generated as well as lists of elements, drawings of joints, necessary notes, specifications. The user could make layout of drawings, arrange model presentations, separate items, tables and notes on the sheets of drawings.

In KM-SAPR module the user could import models from other graphical systems of steel structures (Tekla, Bocad, RealSteel, AdvanceSteel).



16.13330.2011	
Сварные стыки	
Тип сечения	   
Тип резки	   
Формальный	+
Исходный	+
Внецентрично-скалыв	+
Тип элемента	   
Формальный	+
Исходный	+
Внецентрично-скалыв	+
Тип элемента	   
Формальный	+
Исходный	+
Внецентрично-скалыв	+





LIRA-SAPR®

Design modules for Reinforced Concrete Structures

ARM-SAPR

Analysis & design of reinforced concrete structures

In this mode it is possible to determine areas of reinforcement for columns, beams, slabs and shells according to ultimate and serviceability limit states. Drawings of beams and columns are presented after analysis; they may be saved as DXF files. The following building codes are supported: DBN B.2.6-98:2009, SP 52-101-2003, SNIP 2.03.01-84, Eurocode, TSN 102-00, DSTU 3760-98.

LARM-SAPR

Analysis & design of reinforced concrete structures (Local mode)

Analysis & design of the separate RC bar or plate element according to appropriate building codes. Areas of reinforcement are determined and then the specified reinforcement is checked for selected element. The following building codes are supported: DBN B.2.6-98:2009, SP 52-101-2003, SNIP 2.03.01-84, Eurocode, TSN 102-00, DSTU 3760-98.

SAPFIR-RC

The following modules are included: SLAB, DIAPHRAGM, COLUMN, BEAM, DOWELS.

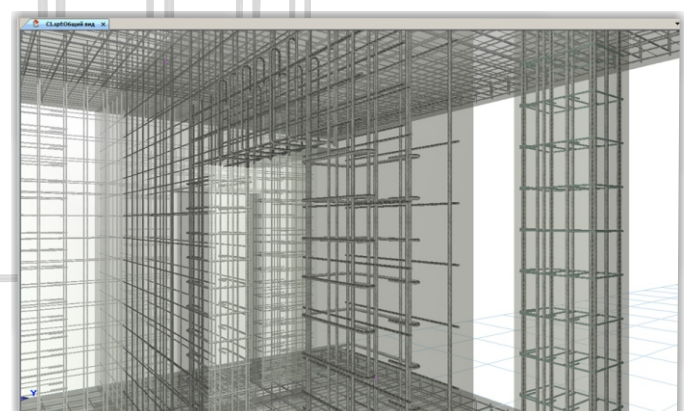
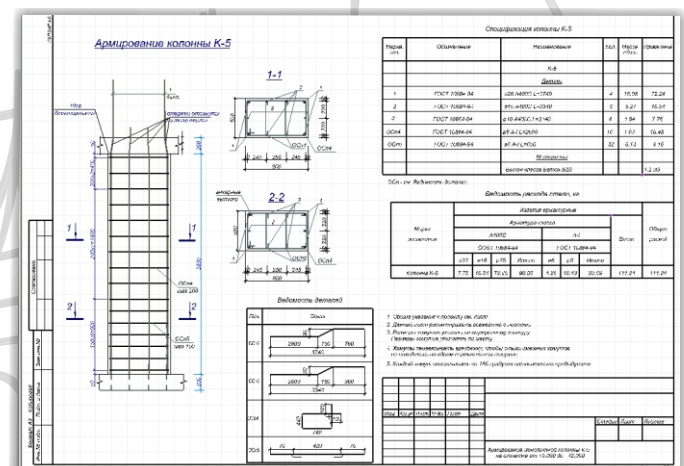
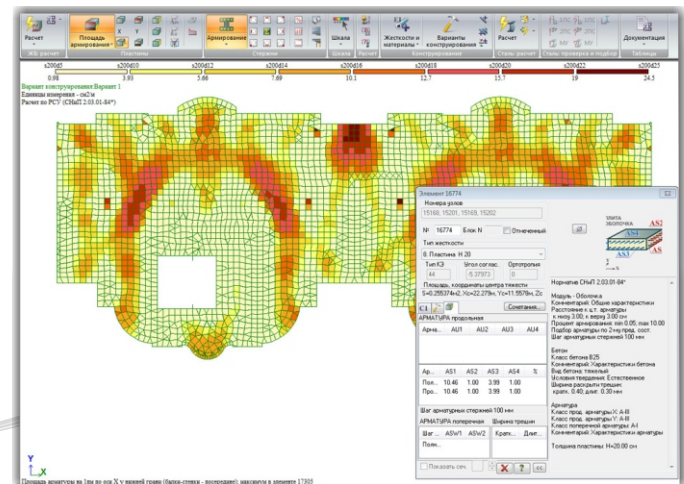
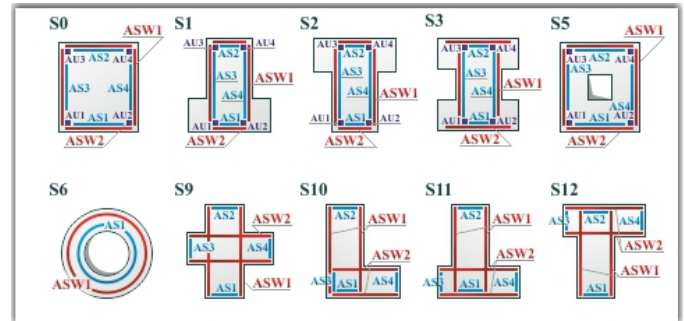
In **SLAB** module the user could generate contour plots of design reinforcement; obtain user-friendly tools to define, locate and unify pattern zones for additional reinforcement with account of defined main reinforcement for every mark of a slab. It is also possible to frame the openings with reinforcement, etc. Drawings may be generated along each of the two directions of rebar patterns along top and bottom edges of a slab.

In **DIAPHRAGM** module the user could generate contour plots of design reinforcement for defined marks of diaphragms (walls). For design of diaphragm, rebars are arranged automatically along two directions for both edges of diaphragm. Rebar pattern generated automatically may be modified by the user later. It is also possible to frame the openings with reinforcement, etc.

According to mosaic plots of design reinforcement on the model of whole structure, in **COLUMN** module you could obtain tools to generate required types of reinforcement for defined marks of columns. Column reinforcement is arranged automatically according to assigned type of reinforcement. Rebar pattern generated automatically may be modified by the user later.

Based on diagrams of design reinforcement, in **BEAM** module the user could assign marks with account of unification for beams. Beam is reinforced in automatic mode according to adjustable parameters of reinforcement. Breakage of longitudinal rebars, their anchorage and zones of transverse reinforcement with various spacing are available. Automatic reinforcement pattern for rebars may be modified by the user. Changes in reinforcement are visualized on the diagram of materials.

In the **DOWELS** module, based on marked plans of columns, diaphragms and reinforcement in these elements, the user could automatically obtain drawings of dowels from the foundation slab.





LIRA-SAPR

Preprocessor SAPFIR-Structures

This module enables you to generate design model of building or structure based on 3D information model. The module provides powerful tools for 3D simulation; they help the user to generate information model from scratch and edit models generated in other BIM-oriented programs like Allplan, ArchiCAD, Revit, Tekla, etc.

Physical (real) and analytical components of information model are supported. It is allowed to design in both presentations.

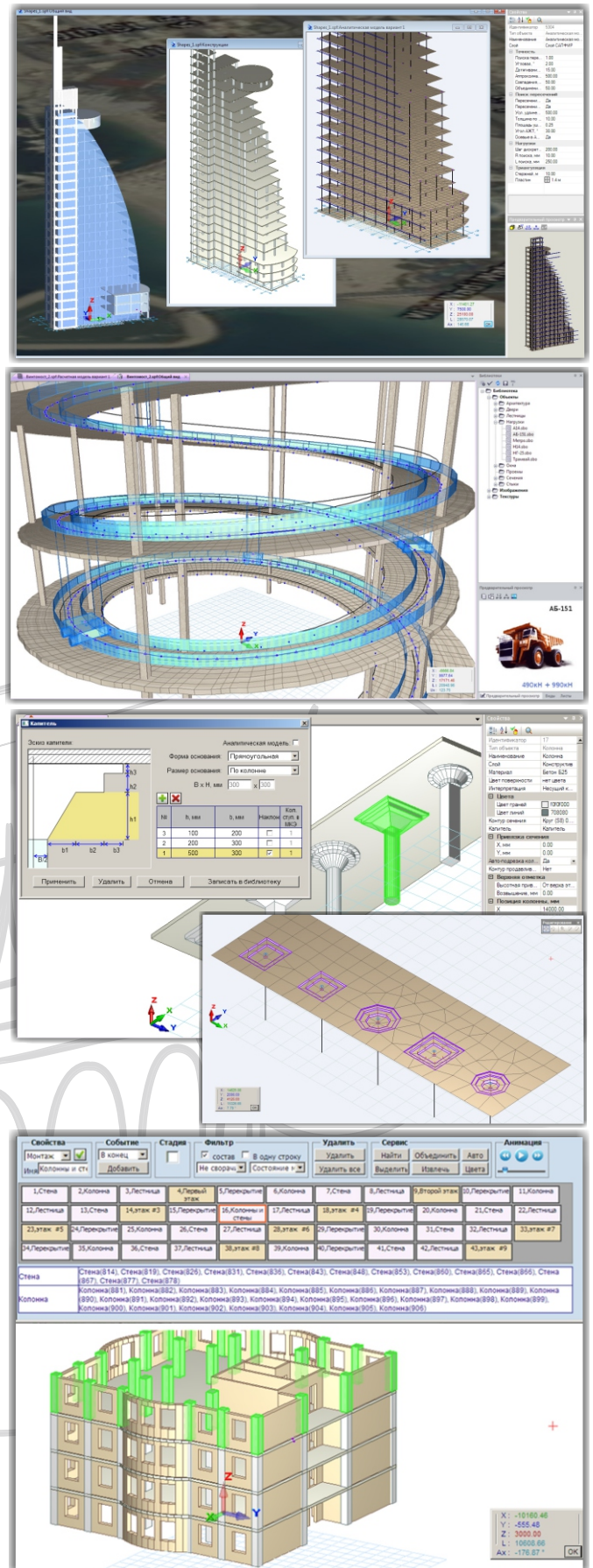
FE library of automatic meshers. Every mesher generates linear triangular and quadrilateral FE mesh according to its own rules. Combined behaviour of meshes is provided by uniting them at intersections with common nodes, perfectly rigid bodies, groups of coupled DOF and other boundary conditions.

The user could define and edit the following loads: concentrated, distributed along line, across area, with varying intensity, wind, earthquake, maintenance at premises and moving transport loads on superstructures.

It is also possible to define capitals, thickening and thinning of slabs, ramps, stairs, trusses, complex curvilinear shapes and surfaces. Punching shear contours are generated automatically with account of the slab shape and presence of the openings.

Main advantages:

1. When design is based on the information model, it eliminates the need to re-enter data, makes the procedure less time-consuming, reduces the number of errors, and provides an adequate design model.
2. Option to automatically generate 3D information model from 2D drawings AutoCAD, backgrounds and floor plans.
3. Powerful graphical tools enable the user to generate and edit the model in physical and analytical presentation in a quick and easy way.
4. The choice of triangulation algorithms and individual adjustment of parameters ensures that the optimal meshes are obtained depending on the features of the object.
5. User-friendly interface where it is possible to specify loads and load history, to generate tasks for computing DCF and DCL, including combinations for appropriate building codes.
6. Interactive graphical control over assemblage history for ASSEMBLAGE-plus solver with visual illustration in physical and analytical presentation.
7. Advanced procedures for diagnostic as well as multi-criterial check for model quality with interactive report about evident errors and potential incorrectness enable the user to obtain the adequate design models for the structures.





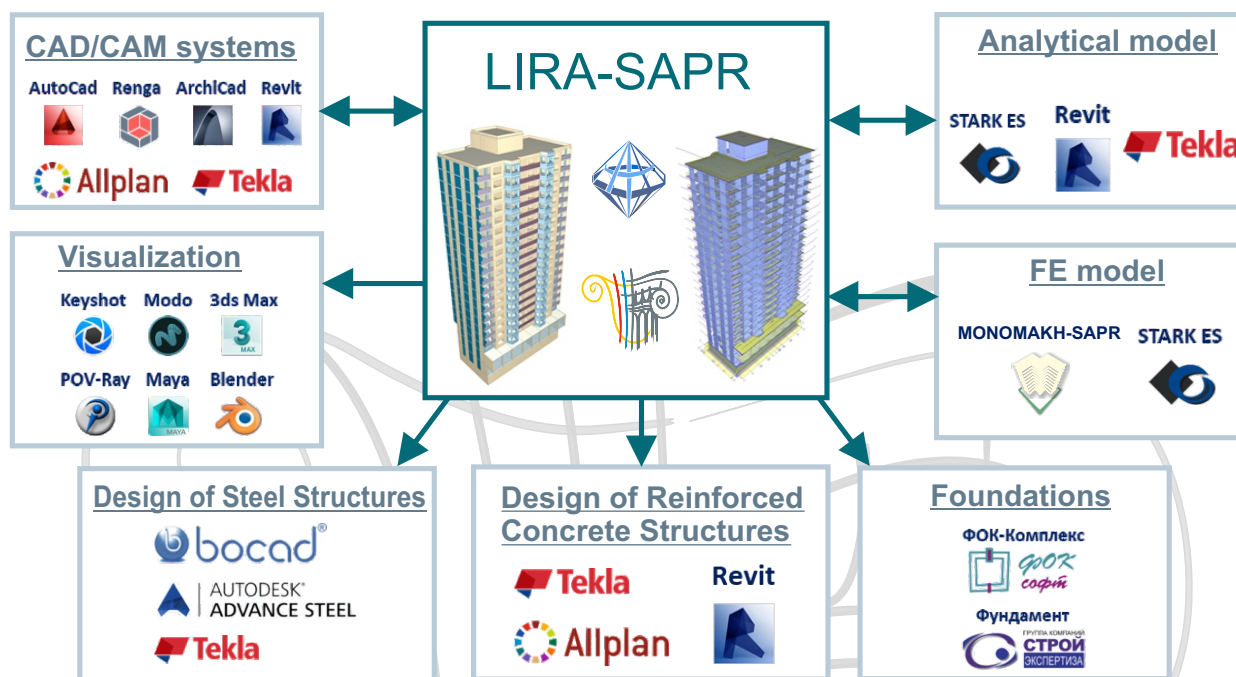
LIRA-SAPR®

Integration of LIRA-SAPR with other software

LIRA-SAPR is a component of BIM-technology. The program supports an open approach to information modelling of buildings.

The LIRA-SAPR tools enable the user to achieve complete integration (information models, coordinated processes and methods, common terminology) between software packages for various purposes, to import architectural models from various formats and export analysis results to various graphic programs.

The following data formats are used: *.msh, *.stl, *.dxf, *.obj, *.IFC, *.3ds, *.mdb, *.stp, *.sli, *.docx, *.xlsx, *.bmp, *.gif, *.txt, *.liraKM, etc.



The data exchange technology between Tekla Structures and LIRA-SAPR, REVIT and LIRA-SAPR enables the user to significantly reduce the time required for the project and take full advantage of each software in the integrated technology for design of buildings and structures.

