APPENDIX B. IMPORT / EXPORT OF DATA ON THE CALCULATION MODEL

B.1 SUPPORTED IMPORT AND EXPORT FORMATS

B.1.1 Overview

Data exchange with other software systems is one of the most important functionalities of SP LIRA 10. Two data exchange strategies have been implemented: integration of a software module into an external software package, for instance, into Revit, and use of built-in functions for importing and exporting data to intermediate files.

SP LIRA 10.12 provides the ability to import or export source data and calculation results in a number of common formats:

- ***.msh** — FE generator files of Gmsh mesh.
- ***.stl;*stlA;*stlB** — files for data transferring on the 3D geometry of a structure from the STL program (3D stereolithography).
- ***.obj** — files for transferring geometry data from the Wavefront program.
- ***.mesh** — files for transferring data about the 3D geometry of a structure from the INRIA program.
- ***.off** — files for data transfer from RoffView, Java Off Viewer, MeshLab, GLC_Player model modeling programs.
- ***.poly** — files for transferring data from polygon modeling programs.
- ***.dxf** — an open file format for exchanging graphical information between CAD applications. In particular, it is used by the AutoCAD computer-aided design system to create design schemes for the LIRA 10 software package.
- ***.igs, *.iges** — a neutral file format designed for the exchange of 2D and 3D data of graphic editors between heterogeneous CAD systems. For import in SP LIRA 10, import/export of geometric models consisting of primitives is implemented: node (Node), rod (Beam), plate (Linear Triangle, Linear Quadrilateral).
- ***.3ds** — files used by Autodesk 3ds Max 3D for modeling, animation, and rendering. Model geometry is imported into SP LIRA 10.
- ***.neu** — text format FEMAP Neutral File Format, designed to store data about the model (this format is most conveniently used to transfer the FE model between SP LIRA 10 and Scad Office PC).
- *.**sdnf** — (Steel Detailing Neutral File) is used to import/export 3D models when working with CAD systems such as AutoCAD, Bocad-3D, Tekla Structures, etc. Describes structural elements such as bars and plates, as well as the loads on them, information about materials, etc.
- ***.byu** — used to render a 3D polygon mesh in CAD systems and other applications. Represents 3D objects as a set of polygon primitives.
- ***ifc** — a neutral file format that allows the exchange of information between different CAD systems and other construction management systems.
- ***.docx, *.xlsx** — a series of file formats for storing electronic documents of office application packages.
- *.**bmp** — raster image storage format.

- *.**dwg** — a binary file format used to store two-dimensional (2D) and three-dimensional (3D) models when working with such CAD systems as AutoCAD, Advance Steel, CorelCAD, BricsCAD, etc.
- *.**gif** — a graphic image format capable of storing compressed data without loss of quality in a format of no more than 256 colors.
- ***.png** — a raster format for storing graphic information using lossless compression using the Deflate algorithm.
- ***.tiff** — raster graphics storage format.
- ***.jpeg** — a graphic format used for storing photographs and similar images.
- ***.html** — tagged markup language for documents.
- ***.pptx** — a standard PowerPoint format that can contain slides with images, text, animation, audio, video, special effects, graphs, charts and other data.
- ***.avi** — multimedia container for audio and video data.

B.1.2 Add an imported fragment

To add an imported fragment to the calculation model, you can use the **Diagram ⇨ Import Fragment** menu command (the command of the same name on the **Add** ribbon or the button  on the toolbar). As a result, the panel of the active **Import** mode is displayed (Fig. B.1), where on the **Available Formats** tab the format can be selected from which the fragment import is to be carried out. The **Scale factor** input field specifies the scale factor to the coordinates of the imported model.

When clicking on the **Import** button, a standard file opening window opens, where the path to the file in which the imported fragment is saved, should be specified. If SP LIRA 10 does not detect any errors during reading and importing the file, the positioning the imported fragment in the interactive window of design model's visualization can be carried out.

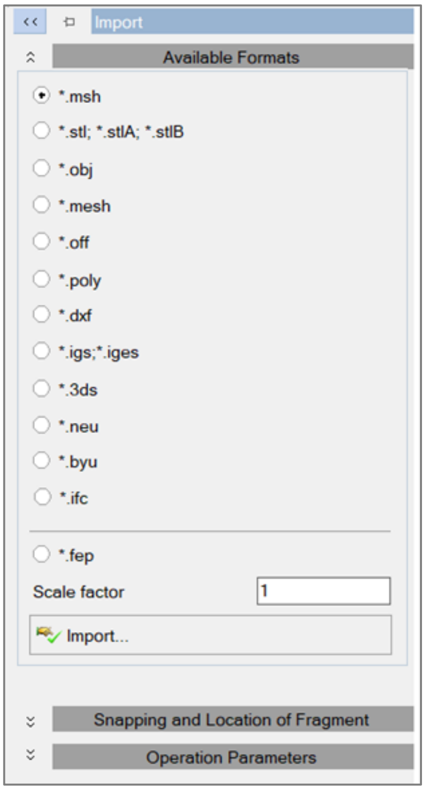


Fig. B.1. Dialog box **Import**

B.1.3 Common DXF format files import

Files of the common format DXF (import of flat elements of spatial orientation) are imported into the LIRA 10 program as a set of flat faces (shells) and bar elements. For correct data transfer in SP LIRA 10, DXF objects are drawn with the following primitives:

- LINE — rod FE 10;
- POLYLINE — rod FE 10;
- 3DFACE — shell 3-node FE 42, 4-node FE 44.

A 3DFACE face is defined by four or three points that lie in the same plane. For a three-point face, the third and fourth points must match.

Imported objects can be assigned section and material parameters. To do this, keywords (identifier) of the corresponding section and material are indicated in parentheses in the layer description. Table B.1 lists the identifiers of the implemented types of parametric sections. Table B.2 presents steel rolling sections. Table B.3 contains identifiers of materials: isotropic, as well as materials in accordance with the base of materials SP LIRA 10. For sections and materials not listed in the tables, data transfer is not provided.

Also in SP LIRA 10, the color of the layer is transmitted, which is assigned to the material and the group of elements. DXF objects drawn with the same color will belong to the same element group. Types of lines and the width of the polylines are ignored.

When importing from common format DXF files into SP LIRA 10 environment, the dxf unit of model geometry equals one meter.

Arcuate segments are not imported into SP LIRA 10. They need to be replaced with broken lines (the shorter the length of the polyline segments, the higher the import accuracy is).

Important:

1. *Section and material identifiers are written with a space.*
2. *«.» (dot) — symbol, available in the Layer Name is used as a numeric separator.*

Table B.1. Parametric section

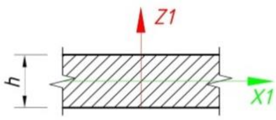
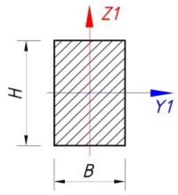
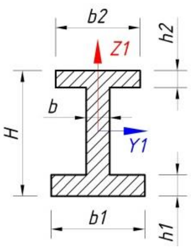
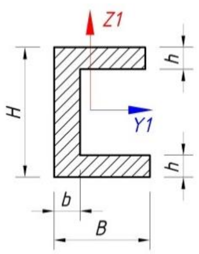
Section type	**Sketch**	**Section shape identifier**	**Note**
Plate		PLATE h-☐	☐ – numerical value, given in meters
Bar		BEAM H-☐ B-☐	☐ – numerical value, given in meters
I-beam		I_ BEAM b-☐ b1-☐ b2-☐ H-☐ h1-☐ h2-☐	☐ – numerical value, given in meters
Channel Bar		CHANNEL B-☐ b-☐ H-☐ h-☐	☐ – numerical value, given in meters

Table B.1 continuation

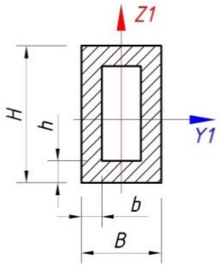
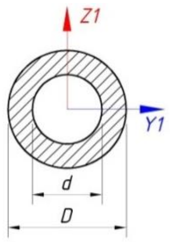
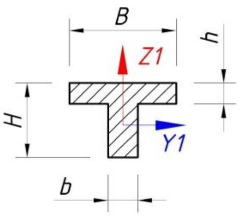
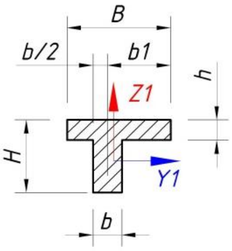
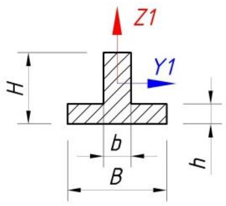
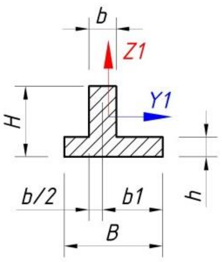
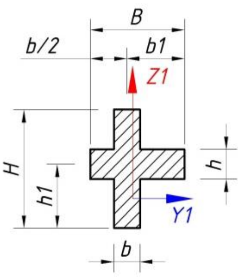
Section type	Sketch	Section shape identifier	Note
RHS		BOX B-☐ b-☐ H-☐ h-☐	☐ – numerical value, given in meters
CHS		RING D-☐ d-☐	☐ – numerical value, given in meters
Symmetrical T-beam with top shelf		T_BEAM_SYM_T B-☐ b-☐ h-☐ H-☐	☐ – numerical value, given in meters
Asymmetrical T-beam with top shelf		T_BEAM_NONSYM_T B-☐ b-☐ b1-☐ H-☐ h-☐	☐ – numerical value, given in meters
Symmetrical T-beam with bottom shelf		T_BEAM_SYM_L B-☐ b-☐ h-☐ H-☐	☐ – numerical value, given in meters
Asymmetrical T-beam with bottom shelf		T_BEAM_NONSYM_L B-☐ b-☐ b1-☐ H-☐ h-☐	☐ – numerical value, given in meters
Cross		CROSS B-☐ b-☐ b1-☐ H-☐ h-☐ h1-☐	☐ – numerical value, given in meters

Table B.1 continuation

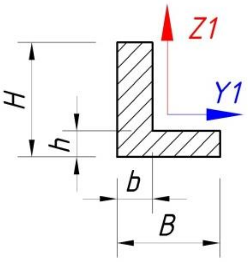
Section type	Sketch	Section shape identifier	Note
Angle		ANGLE B-☐ b-☐ H-☐ h-☐	☐ – numerical value, given in meters

Table B.2. Rolled Cross Sections

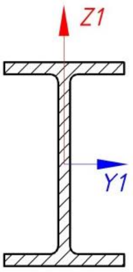
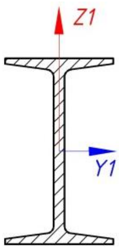
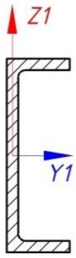
Section type	Sketch	Section shape identifier	Norms	Profile name	Note
I-beam with parallel edges of shelves		STT_I_SU N-☐	GOST 26020-83 (type Б)	10Б1	where: ☐ – profile name
			GOST 26020-83 (type Б of additional series)	10ДБ1	
			GOST 26020-83 (type К)	20К1	
			GOST 26020-83 (type Ш)	20Ш1	
			ASTM A6M (Narrow-shelf)	31У3А	
			ASTM A6M (Standart-shelf)	31Б1А	
			ASTM A6M (Middle-shelf)	20Д1А	
			ASTM A6M (Wide-shelf)	30Ш2С	
			ASTM A6M (Column)	12КС	
			STO ASChM 20-93 (Beam)	10Б1	
			STO ASChM 20-93 (Wide-shelf)	20Ш1	
			STO ASChM 20-93 (Column)	20К1	
			OST-16 (standart metrical)	12	
			OST-10016-39 (I-beams)	14	
			OST-16-1926 (Gray-Peyner)	14	
			I-beams (German standard assortment)	11	
I-beam with unparallel edges of shelves		STT_I_SU_NPRL N-☐	GOST 8239-72*	10	where: ☐ – profile name
			GOST 8239-89	10	
			GOST 5157-53 (Spetial)	18М	
			GOST 8239-56 (with changes from 1959)	10	
			GOST 8239-56 (instead of OST-10016-39)	10	
Channel with parallel edges of shelves		STT_C_SU N-☐	GOST 8240-72* (with parallel edges of shelves)	5П	where: ☐ – profile name
			GOST 8240-97 (with parallel edges of shelves)	5П	
			GOST 8240-97 (Economic)	5Э	
			GOST 8240-97 (Light series)	5Л	

Table B.2 continuation

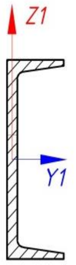
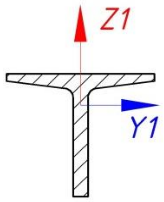
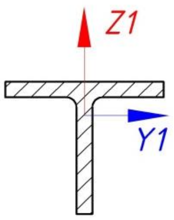
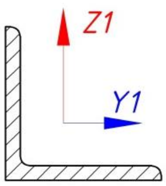
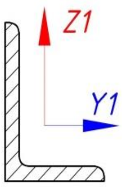
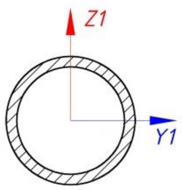
Section type	**Sketch**	**Section shape identifier**	**Norms**	**Profile name**	**Note**
Channel with unparallel edges of shelves		STT_C_SU_NPRL N-☐	GOST 8240-72 (with slope of inner edges of shelves)	5	where: ☐ – profile name
Taurus with unparallel edges of the shelf		STT_T_SU_NPRL N-☐	GOST 8239-72*	1/2 от двут. 10	where: ☐ – profile name. The presence of the symbol «/» (slash) in the name should be replaced with «_» (underscore). The profile name is written without spaces Example: 1_2отдвут.10
Taurus with parallel edges of the shelf		STT_T_SU N-☐	TU 14-2-24-72 (type Б)	10БТ*	where: ☐ – profile name. The presence of the symbol «*» (asterisk) in the name should be replaced with «^» (Degree)
			TU 14-2-24-72 (type К)	10КТ*	
			TU 14-2-24-72 (type Ш)	10ШТ*	
			TU 14-2-24-72 (type КУ)	10'КУТ1	
Engle even shelf		STT_L_EQUAL_SU N-☐	GOST 8509-86	20x20x3	where: ☐ – profile name
Engle uneven shelf		STT_L_SU N-☐	GOST 8510-72	25x16x3	where: ☐ – profile name
Pipe		STT_PIPE_SU N-☐	GOST 10704-76	83x3	where: ☐ – profile name
			GOST 8732-78	28x3	
			TU 1381-020-00186654-2011	530x7	

Table B.2 continuation

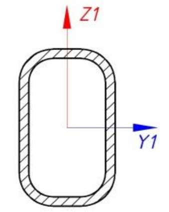
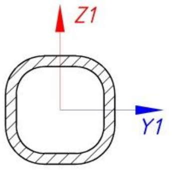
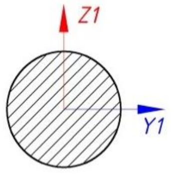
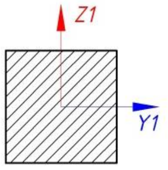
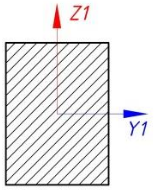
Section type	Sketch	Section shape identifier	Norms	Profile name	Note
Rectangular profile		STT_PIPE_RECT_SU N-☐	GOST 30245-2003	50x25x2	where: ☐ – profile name
			TU 67-2287-80	100x60x3	
Square profile		STT_PIPE_SQR_SU N-☐	GOST 30245-2003	40x40x2	where: ☐ – profile name
			GOST 8639-82	10x10x1	
			TU 36-2287-80	80x80x3	
Circle		STT_CIRCLE_SU N-☐	GOST 2590-88, TU 14-136-347-2001	8	where: ☐ – profile name
Square		STT_FOURSQUARE_SU N-☐	GOST 2591-88, TU14-1-4492-88	28	where: ☐ – profile name
Rectangle		STT_RECT_SU N-☐	GOST 103-76	90x9	where: ☐ – profile name

Table B.3. Materials

Item	Norms	Material identifier	Note
Isotropic	—	R-☐ E-☐ V-☐ A-☐	R – specific weight, t/m^3 E – modulus of elasticity, t/m^2 V – Poisson's ratio A – Coefficient of thermal expansion, 1/C ☐ – numerical value
Heavyweight concrete	SNiP 2.03.01-84	SNP-☐	where: ☐ – class of concrete. The presence of the symbol «/» (slash) in the name should be replaced with «_» (underscore)
Heavyweight concrete	DBN B2.6-98:2009	DBN-☐	
Heavyweight concrete	SP-52-101-2003(63-13330-2012)	SP-☐	
Heavyweight concrete	Eurocode	EC-☐	

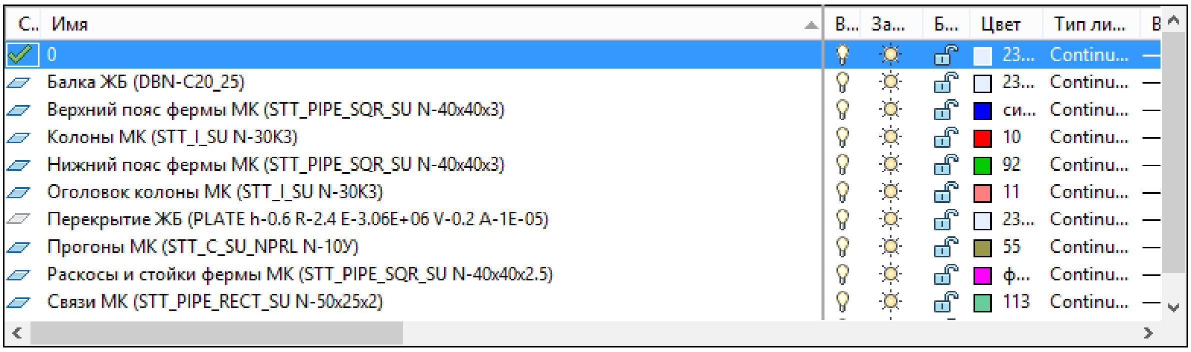


Fig. B.2. Layer design example for importing generic DXF files

B.1.4 Export of a finite element model to a common DXF file

B SP LIRA 10 implements the export of a finite element model to a DXF file of a general format. The following types of finite elements are exported to the DXF file:

Table B.4

Object LIRA 10	**Object DXF**
Piles (Single-noded FE)	Point
Bar finite elements	LINE
Plate finite elements	3DFACE
Solid finite elements	Each face of a solid element consists of 3DFACE
Loads	see table B.6

It is possible to transfer not only a finite element model to a DXF file, but also data on the design model (text representation attributes applied to elements), as well as calculation results (deformed model, mosaics of results of strength or design analysis).

Only that data is exported, which render at the time the export function is being executed.

B.2 Peculiarities of importing floor plans from DXF files

B.2.1 General information

SP LIRA 10 imports walls, slabs, columns and beams from DXF text files, which must be represented by the corresponding objects and belong to a layer with a predefined name.

Table B.5 describes how an object should be specified in a DXF file and what SP LIRA 10 model object it is converted to when imported.

In SP LIRA 10, when importing floor plans, along with the geometry of the scheme, *sections*, *materials* and *loads*, specified on the element can also be generated. Table B.6 shows the identifiers of all loads for which data transmission is provided.

Table B.5. Layers for floor import

Item	Representation in DXF	Layer name	DXF Object	LIRA 10 Object
Foundation slab / Floor slab	Midplane Contour	SLABS	P-Line	Architectural contour of the plate
Holes (foundation slab / floor slab)	Contour	SLABS_HOLES	P-Line	Subtracted from the contour of the plate
Wall	Projection of the median plane in plan	WALLS	Line	Architectural contour of the plate
Doors	Plan projection	DOORS	Line	Subtracted from the contour of the plate
Window	Plan projection	WINDOWS	Line	Subtracted from the contour of the plate
Pile	Center point of the plane projection	PILES	Point	Single node FE 56
Column	Center point of the plane projection	COLUMNS	Point	Architectural rod
Beam	Central axis	BEAMS	Line	Architectural rod

Table B.6. Loads

Item	Loading identifier	Load identifier	DXF Object	Note
Static loading, evenly distributed over the area. Considered only for the SLABS layer	LC-{arbitrary name}	A_Load-☐	P-Line	where: ☐ – numerical value
Static loading, evenly distributed over the area. Considered only for the BEAMS layer		L_Load-☐	Line	
Arbitrary load at a point. ARB_POINT_LOAD (LC-{name} FX(☐))		FX(☐), FY(☐), FZ(☐), MX(☐), MY(☐), MZ(☐)	Point	
Arbitrary load on the line. ARB_LINE_LOAD (LC-{name} QY(☐))		QX(☐), QY(☐), QZ(☐)	P-Line	
Arbitrary load on the surface. ARB_AREA_LOAD (LC-{name} PZ(☐))		PX(☐), PY(☐), PZ(☐)	P-Line	

Important!

1. Building axes, partitions and loads not listed in Table B.5 are not imported.

2. The line of walls should not be interrupted in door and window openings.

3. The contour of the slabs and holes defined by the polyline must be closed.

4. Polyline roundings are not allowed.

5. The imposition of the contours of slabs and holes is not allowed.

6. The names of the layers indicated in Table B.5 can have both section identifiers (see Tables B.1 and B.2) and materials identifiers (see Table B.3).

7. Layer names of doorways that do not have identifier parameters are imported by default, doorway height is H = 2.1 m.

8. The names of the layers of window openings that do not have identifier parameters are imported by default, the setback from the bottom of the ceiling h = 0.8 m, window height H = 1.5 m.

9. The layer name ***PILES****, both for the general *.dxf format and for floor plans, can have the following parameters (Rx-☐ Rux-☐ Ry-☐ Ruy-☐ Rz-☐ Ruz-☐):*

- *Rx is the axial stiffness of link under tension-compression along global/local axis Y of the node;*
- *Ry is the axial stiffness of link under tension-compression along global/local axis Y of the node;*
- *Rz is axial stiffness of link under tension-compression along the global/local Z-axis*
- *Rux is the rotational stiffness of link under rotation around the global/local X-axis;*
- *Ruy is the rotational stiffness of link under rotation around the global/local Y-axis;*
- *Ruz is the rotational stiffness of link under rotation around the global/local Z-axis;*
- *☐ — numerical value of stiffness.*

Numerical values of identifiers specified in tables B.1, B.3 for floor-by-story import can be in arbitrary units. These units will be assigned in the ***Floor Plans Import*** *dialog box.*

10. In the Layer Name, identifiers should be typed in the following manner: Layer Name (section identifier; material identifier; loading states editor identifier; load identifier).

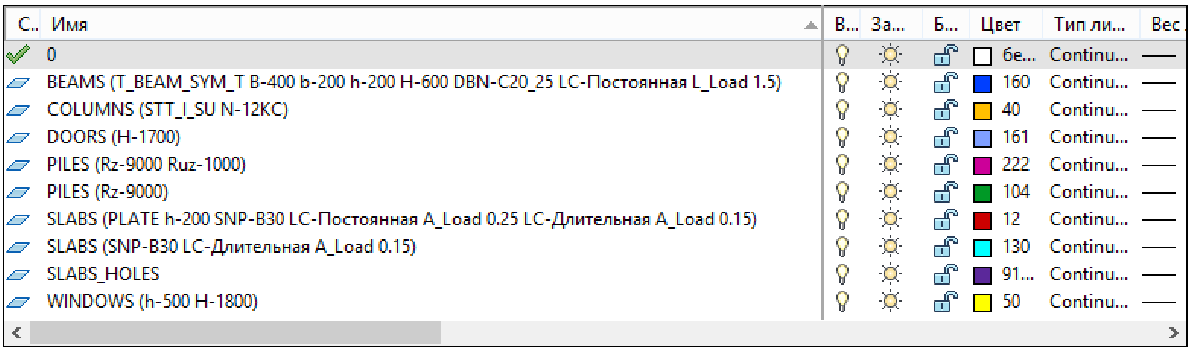


Fig. B.3. Example of layer design for floor-by-floor import of DXF files

B.2.2 Preparing a DXF file in the AutoCAD environment

- Create the required layers using the **AutoCAD ⇨ Create Layer** commands.
- Fill the layers with objects LINE, POINT, POLYLINE, in accordance with the above tables B.1–B.3, B.5, B.6.
- Save the resulting file in DXF format. DXF file names may have (_h☐) suffix, where ☐ is the floor height (ex.: 0_h3000.dxf). When specifying the numerical value of the floor height in the file name, the units of measurement of the model geometry must be taken into account.
- For each floor plan of a multi-story building, repeat the steps above in a separate file.

DXF files prior to 2000 and R12/LT2 versions replace spaces and parentheses in layer names with underscores, destroying layer properties.

When importing only one floor, the DXF file must be named **0.dxf**.

When importing multiple floor plans, each floor plan must be saved in a separate file. File names are given only by numbers. File names can be specified in two sequences:

- **First**: — **0.dxf** — for the first floor, **1.dxf** — for the second floor, **2.dxf** — for the third floor, and so on.
- **Second**: — **-1.dxf** — for the first floor, 0.dxf — for the second floor, 1.dxf — for the third floor, and so on.

For any sequence, the presence of the **0.dxf** file is mandatory; without this file, import is not possible. The generated floor plan files must be placed in the same folder.

The coordinates of the created floor plans must be agreed. The world coordinate system is used.

During the import process, in the **Floor plans import from DXF** dialog box (Fig. B.4), the user can set the units of measurement for the model geometry, section, pile stiffness and load. The specified units of measure will be used when calculating the parameter values in the layer description.

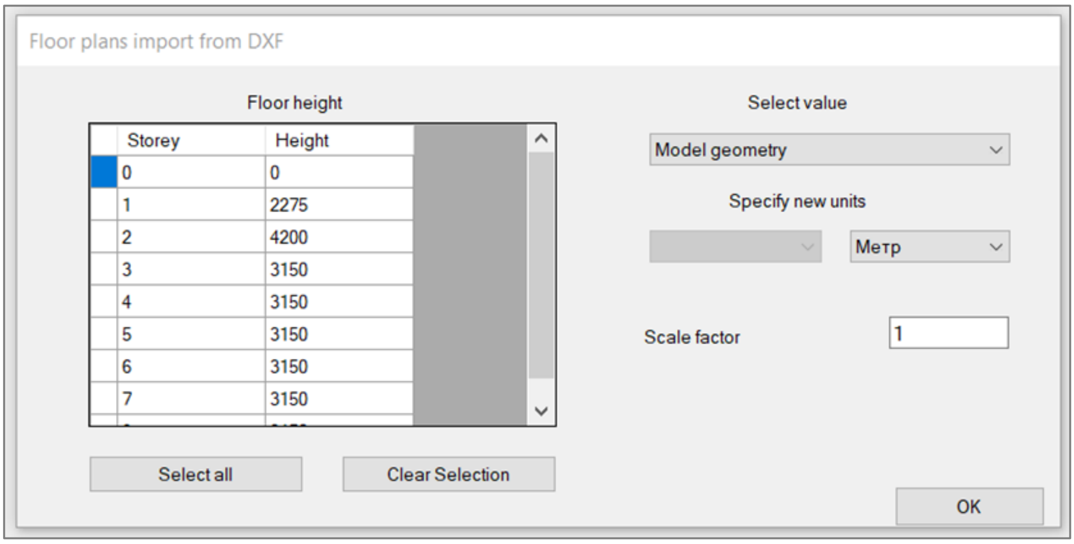


Fig. B.4. Dialog box for importing floor plans from DXF

When importing floor plans, one dxf unit equals one user-specified unit in the ***Floor plans import from DXF*** ⇨ ***Model geometry*** *dialog box.*

B.3 PECULIARITIES OF IMPORTING FILES WITH EXTENSION (*.IFC)

Import

When importing files in the IFC format, support for the IFC2x3 version is implemented.

SP LIRA 10 imports bar and plate elements.

For bar elements, geometry and section data are transmitted. Import of standard parametric sections is implemented.

For plate elements, geometry (including holes) and thickness data are transmitted.

Here is the list of the main implemented classes of the IFC format:

- Rods:
  - IfcColumn, IfcBeam, IfcMember;
- Plates:
  - IfcWall, IfcWallStandardCase, IfcPlate, IfcOpeningElement, IfcBuildingElementProx, IfcSlab.

For IFC files created in Tekla Structures or AVEVA Bocad Steel, automatic comparison of rolled steel sections is implemented. When processing an imported section of rolled steel, described in an IFC file, SP LIRA 10 checks for the presence of such a profile in its own assortment databases.

If there are no matches, the user is prompted to independently match the name of the imported profile with the selected section in the window **Material comparison form** (Fig. B.5).

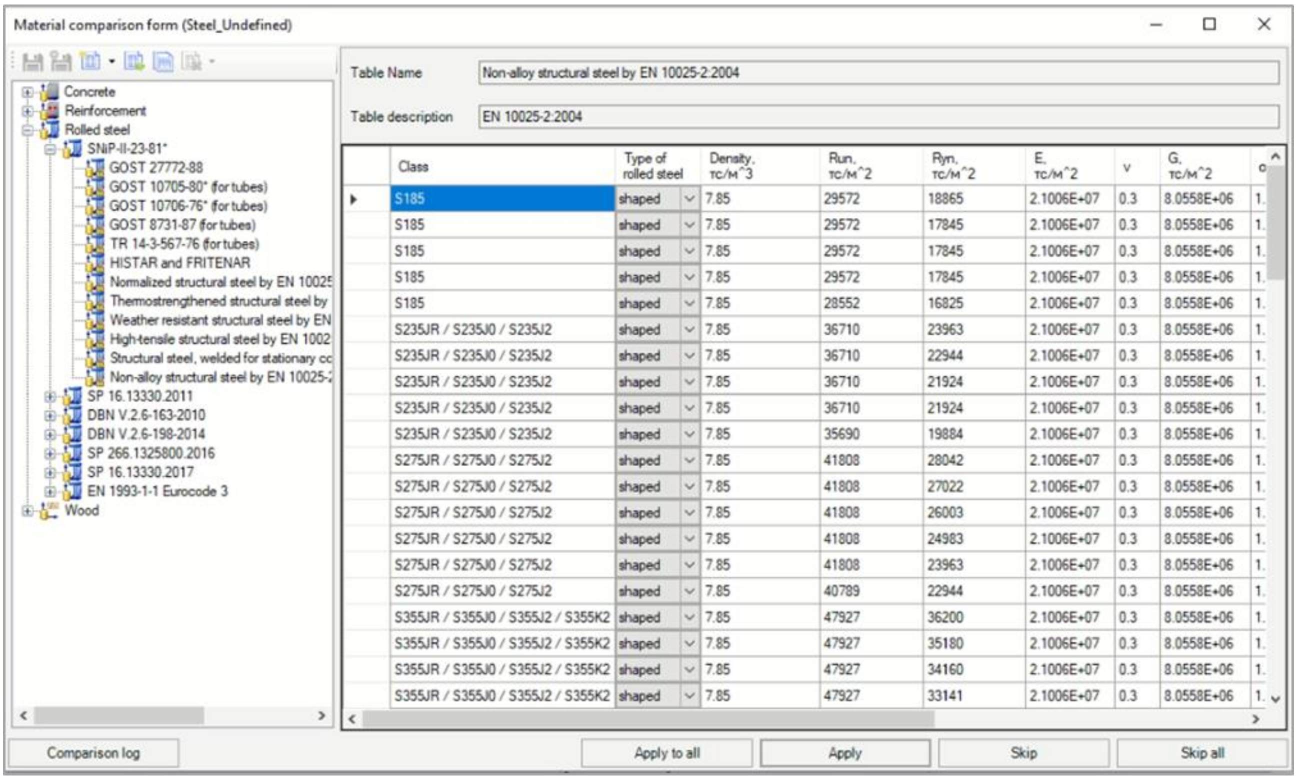


Fig. B.5. Dialog box **Material comparison form**

To compare sections, the user needs to select the required section from the rolled steel table and click the **Apply** button. This section will be automatically assigned to all imported bar elements with the same section description in IFC format.

Matching data is saved in the comparison log, which is located in the SP LIRA 10 database directory. When the IFC file is subsequently imported, the import system will automatically check profiles unknown to it with sections stored in the comparison log.

Pressing the **Preview** button, the steel profile will be shown similar to it, as a parametric one.

To view the comparison log, click the **Comparison log** button, after which the **Cross-section comparison log** dialog box will appear (Fig. B.6).

To remove a matching line from the log, select it and click the **Delete** button.

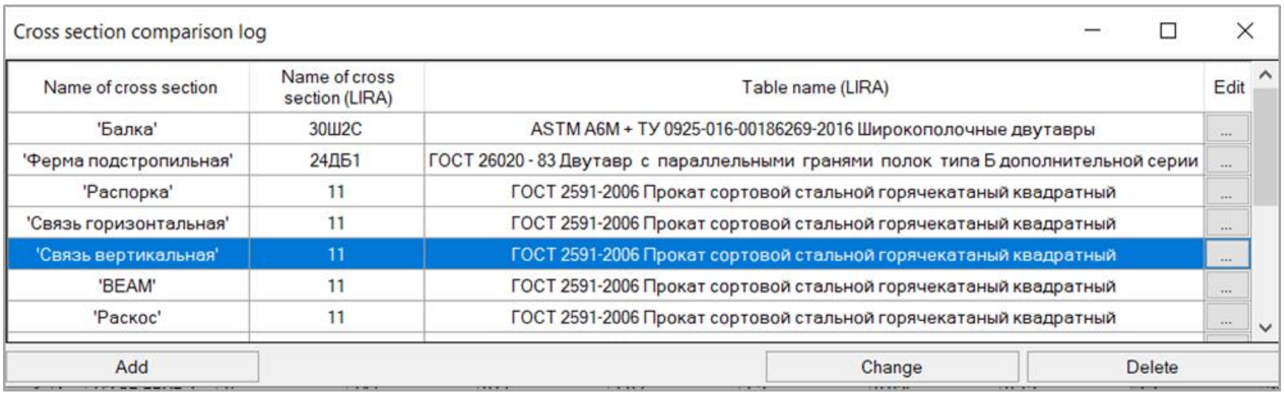


Fig. B.6 Dialog box **Cross section comparison log**

The change mechanism always uses only the log, which is located in the SP LIRA 10 database catalog and which has the name ***JournalOfComparisonSectionIfc****. To use the other comparison log, in the database directory, you are to replace it and rename it to* ***JournalOfComparisonSectionIfc****.*

When you click the **Skip** SP button, LIRA 10 tries to convert the current profile into a parametric or user-defined bar cross-section. The **Skip all** button does the same for all sections, i.e., in the current session matching will no longer run.

All bars with an unconverted IFC profile will be imported without a cross-section, i.e. bar architectural elements will not be assigned with cross-sections.

Following the same algorithm, using the comparison log, the import of materials from IFC is implemented.

Export

In SP LIRA 10, the model export to IFC format version 2x3 is implemented only for the architectural model.

Architectural elements, depending on their position in model space, are exported in IFC format as:

- IfcBeam — for an architectural bar that is located horizontal or occupies an arbitrary position in space.
- IfcColumn — for an architectural bar of vertical position.
- IfcWall — for an architectural plate of vertical position.

- IfcSlab — for an architectural plate horizontal position.
- IfcPlate — for an architectural plate of arbitrary position.

For plate elements, geometry and thickness information is transmitted.

For bar elements, information about the coordinates of the beginning and the end of the bar is transmitted, and the assigned parametric sections are also exported.

Rolled steel cross-sections are exported to the IFC format as parametric sections similar in shape to rolled steel. For bars with cross-sections that are not compatible with the IFC format (for example, a numerical description of the cross-section), a fictitious cross-section is created in the form of a square with a side of 0.2 m.

Information about the position of floors is described in groups of elements.

The following is not exported to the IFC format:

- finite elements;
- loads;
- physical properties of materials.

B.4 Advance Steel integration — LIRA 10

SP LIRA 10 provides for the exchange of section base data with Advance Steel. The connection between the programs is implemented by a plug-in that is built into Advance Steel.

Plugin implemented for Advance Steel 2020 and Advance Steel 2021.

The **LIRA 10** tab was added to the main Advance Steel button bar for importing and exporting the model and the button that starts databases synchronization of rolled steel sections (Fig. B.7).

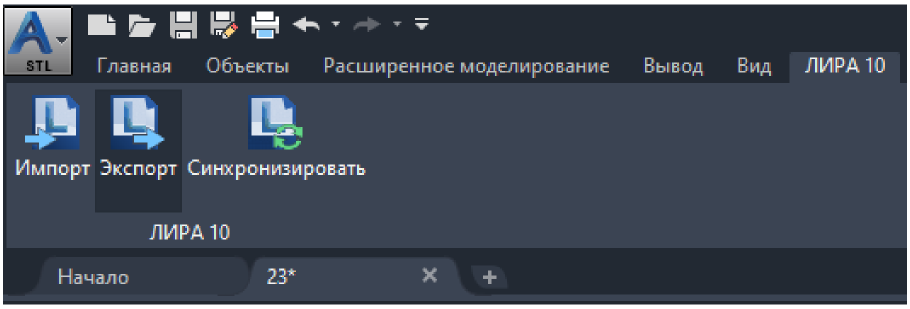


Fig. B.7. External tools

Import and export were carried out for the main structural types of elements, stored in DWG files, as well as their sections and materials.

The steel section database synchronization form provides import and export, replacement and merging of tables into the AstorProfiles database.

Implemented for AstorProfiles 2020 and AstorProfiles 2021.

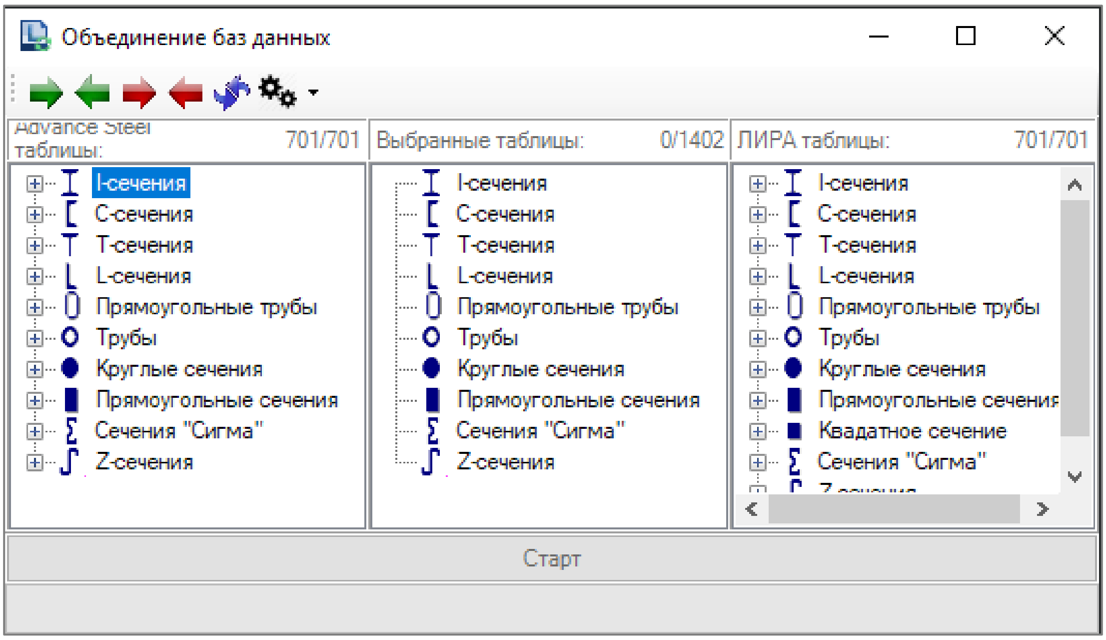


Fig. B.8. Dialog box **Databases Merge**

Before exporting a model from Advance Steel to SP LIRA 10, the databases must be synchronized. To do this, click the **Synchronize** button on the **LIRA 10** panel, after which the **Database Merge** dialog box opens (Fig. B.8). Double-click the left mouse button to select the desired sections, they will be displayed in the middle column **Selected Tables**, then click the **Start** button. Now the sections are available in LIRA 10. After that, you can export the model by clicking on the **Export** button on the **LIRA 10** panel (Fig. B.7).

Import from LIRA 10 to Advance Steel is carried out in a similar way. Select the cross-sectional data by double-clicking the mouse from the **LIRA table** and pressing the **Start** button. After that, you can import the model by clicking on the **Import** button on the **LIRA 10** panel.

It is also possible to manipulate all the missing sections at once using the buttons:

 — Select dissimilar tables and replace in LIRA 10;

 — Select dissimilar tables and replace in Advance Steel;

 — Select duplicate tables and replace in LIRA 10;

 — Select duplicate tables and replace in Advance Steel;

 — Select duplicate tables and combine;

 — User mode settings, where in the drop-down list **On repetitions** you need to select **Synchronize data** or **Replace**.

B.5 Export from Renga

It is possible to export a BIM model from Renga to SP LIRA 10 using a plugin built into Renga.

On the main panel of the program there is a button for exporting the model to a FEP file (Fig. B.9).

Fig. B.9. External export tools

When exporting, a form is displayed with information about the progress of the export process (Fig. B.10). Export is carried out for the main structural types of elements.

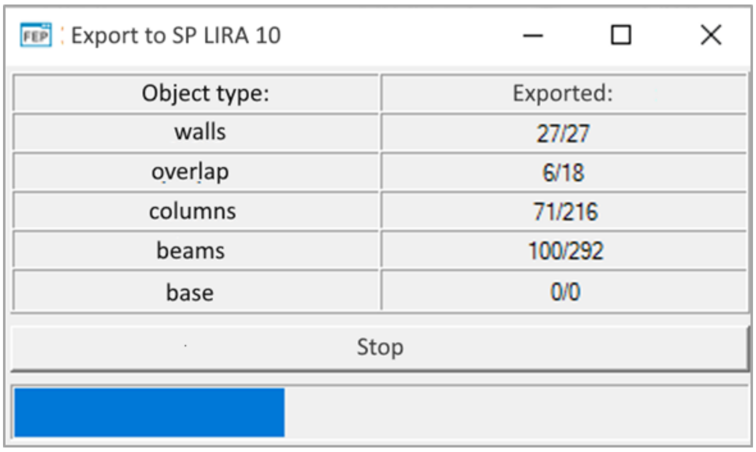


Fig. B.10. Progress Dialog box

B.6 Revit — LIRA 10 integration

SP LIRA 10 provides for the exchange of model data and analysis results with Revit. The link between the programs is implemented by a plug-in, built into Revit.

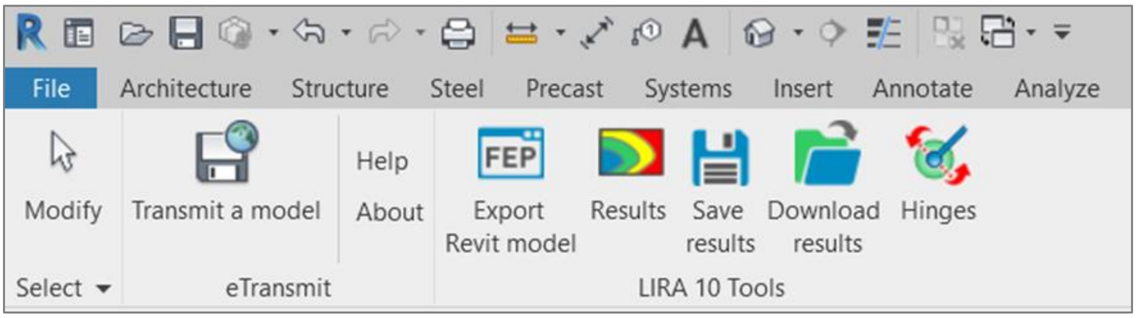


Fig. B.11. External tools

An additional plug-in is installed during the installation of SP LIRA 10 in the presence of a previously installed Revit program.

After starting Revit, this add-on will be displayed on the **Add-Ins** tab (Fig. B.11). The plugin consists of the following commands:

- **Export Revit model** (export to LIRA 10) — exports the Revit analytical model to SP LIRA 10.
- **Results** (Show LIRA 10 Calculation Results) — visualizes the calculation results and updates them.
- **Save result** — saves the results.
- **Download result** — it is possible to load rebar selection results from a file in ***.lrr** format.
- **Hinges** — displays of hinges on the Revit analytical model (Fig. B.12).

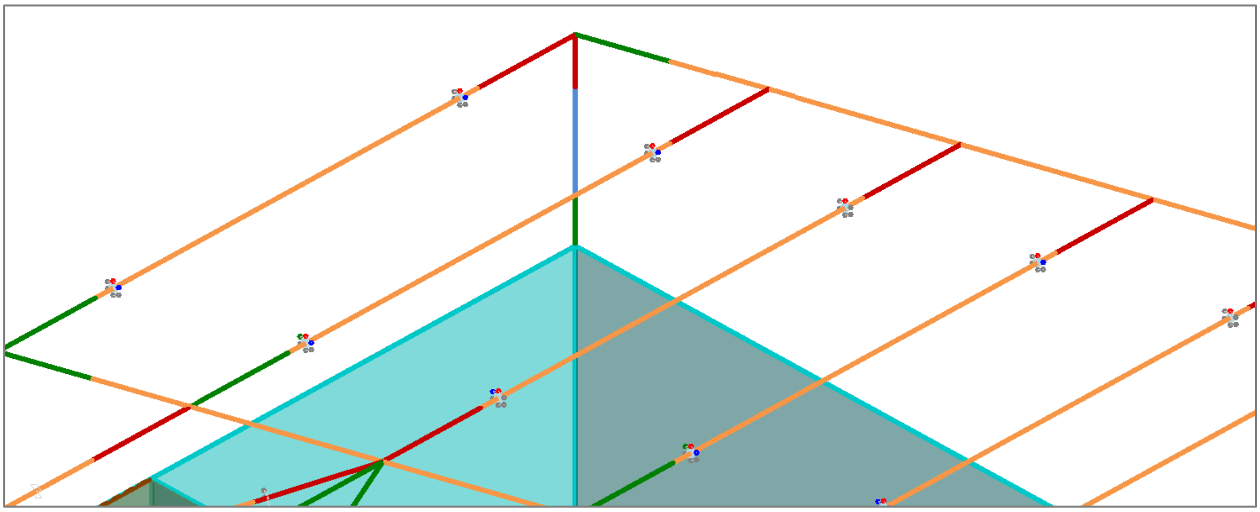


Fig. B.12. Hinge visualization

Model export

To transfer a Revit analytical model to SP LIRA 10, use the **Export Revit model** button. After pressing the button, the **Export wizard** dialog box is activated (Fig. B.14).

This window allows you to manage imported parameters, view logs of comparison of sections and materials, and set additional import parameters. It is also possible to select the interface language (English, Russian, Ukrainian).

Checkboxes **Cross section**, **Material**, **Restrains**, **Loading States** indicate whether the marked parameters will be imported into the *.fep file.

Using the **Snap** checkbox, the **Select the model for snapping** button, you can connect an SP LIRA 10 model with the current Revit model. By calling the menu command again or clearing the checkbox, the models can be unlinked. The snapping will be useful in cases where the model in Revit has been slightly changed, and its parameters need to be transferred to a previously created SP LIRA 10 file, which was also manipulated.

In the case of a snapping-aware export, the following element parameters are updated:

- geometry of the floor contour, slabs, walls, holes;
- geometry and placement in space of beams and bars.

Updating the geometry of the contours of plate elements means highlighting changes with red lines on the current contour of the element (Fig. B.13).

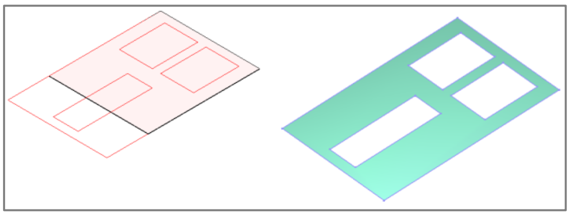


Fig. B.13. Highlight the changes with red lines

The **Triangulation** checkbox allows you to add the parameters specified on the triangulation panel (Fig. B.15). This panel can be called up by clicking on the **Settings** button located to the right of the checkbox.

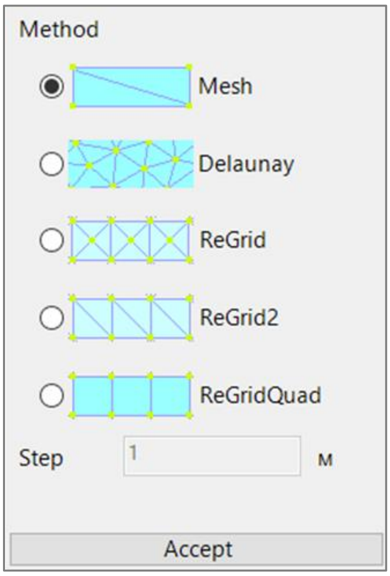


Fig. B.15. Triangulation Options Panel

The **Dead load** checkbox allows you to assign a dead weight to the materials. The load selection panel for self-weight assignment can be called up by clicking on the **Settings** button located to the right of the checkbox. This panel contains a list of materials, load cases and self-weight coefficients (Fig. B.16).

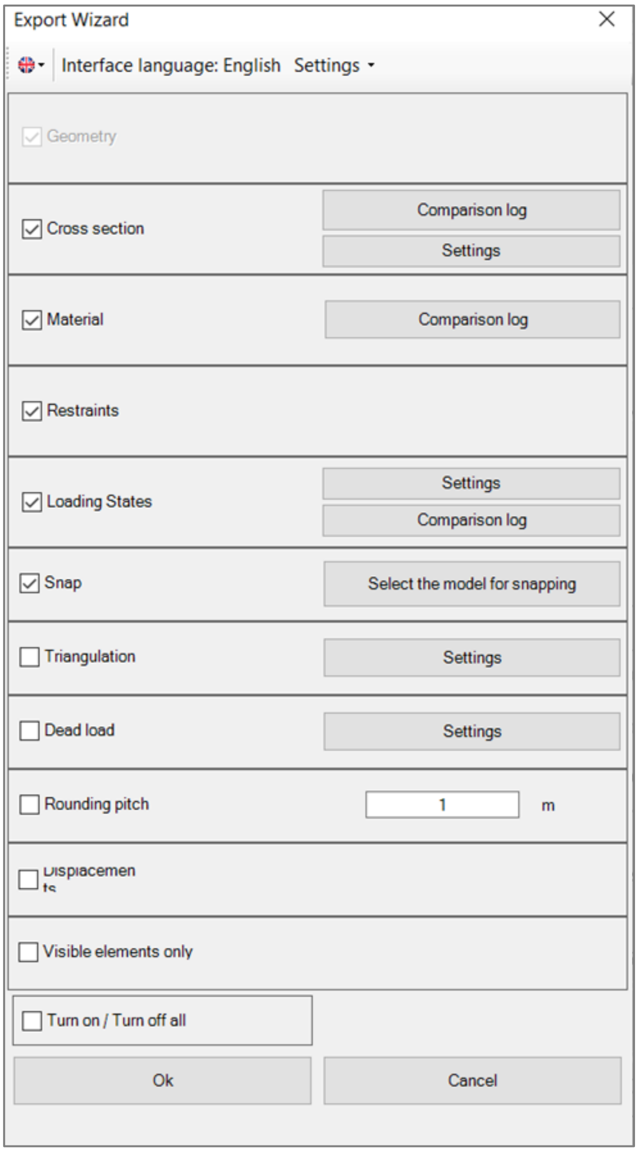
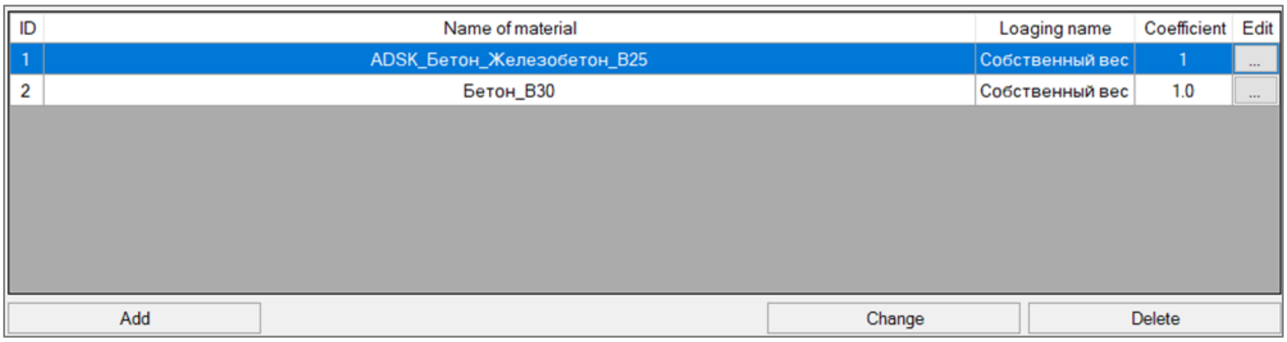


Fig. B.14. Dialog box **Export Wizard**



ID	Name of material	Loaging name	Coefficient	Edit
1	ADSK_Бетон_Железобетон_B25	Собственный вес	1	...
2	Бетон_B30	Собственный вес	1.0	...

Add Change Delete

Fig. B.16. Self-weight setting panel

The **Rounding pitch** checkbox is used for curved elements such as rounded walls and slabs, where the step of splitting the curve with straight lines is set.

The **Displacement** checkbox is used to set the displacement of the model during import.

The **Visible element only** checkbox allows you to import only elements that are not hidden in the current Revit view.

The **Turn on/Turn off all** checkbox activates and deactivates all checks.

Only those elements that have an analytical presentation model are exported in SP LIRA 10. For elements that do not have an analytical model, data transfer is not provided.

When exporting, the following initial data about the model is transferred:

- For bar elements:
  - information about geometry;
  - section;
  - material (physical characteristics of isotropic/orthotropic material);
  - boundary conditions — fixings;
  - hinges.
- For plate elements:
  - geometry information (including hole data);
  - section (thickness);
  - material (physical characteristics of isotropic/orthotropic materials);
  - boundary conditions — fixings;
  - door and window openings with an analytical model.

Comparison of materials and rolled steel is implemented using comparison logs. Comparison logs work in conjunction with Revit in the same way as with IFC. A detailed description of the import of rolled steel is given in B.3.

*The Revit steel comparison log is also located in the SP LIRA 10 database catalog and is named **JournalOfComparisonSectionRevit**. To use a different comparison log, the current log in the database catalog should be replaced with another one and renamed to **JournalOfComparisonSectionRevit**.*

All the load cases (with the name preserved) and their combinations are imported. Each load case implements the import of the following types of loads specified in the global coordinate system:

- concentrated nodal loads;

- uniformly distributed and trapezoidal loads on the rods;
- uniform and trapezoidal loads distributed on the plates along the line;
- distributed and trapezoidal loads on the plates.

Export of calculated results to Revit Structure

Use the **Result** button to update the model and load calculation results from SP LIRA 10 into Revit Structure. After pressing the **Update** button, the **Select file** dialog box appears on the screen. After selecting the required *.fep file, click on the **Open** button.

If the system does not detect any errors during reading and exporting the results, the **Results** dialog box appears on the screen (Fig. B.17). The dialog box provides tools for managing the display of results.

After selecting the results of the selection of reinforcement required for visualization, click the **Display** button. After that, the elements should be selected, for which the mosaic is to be shown, and click the **Finish** button.

If several checkboxes of longitudinal reinforcement inclusions are selected, then their total values will be displayed.

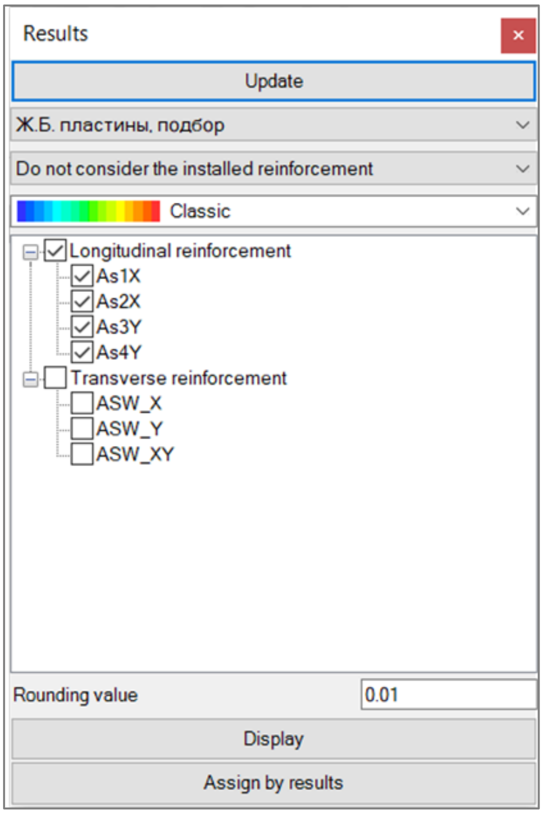


Fig. B.17. Visualization of results

When exporting the results, it is possible to display the verification of steel structures. The displayed results are configured through the “tree” of checks. The results are displayed as diagrams for bar analytical elements (Fig. B.18).

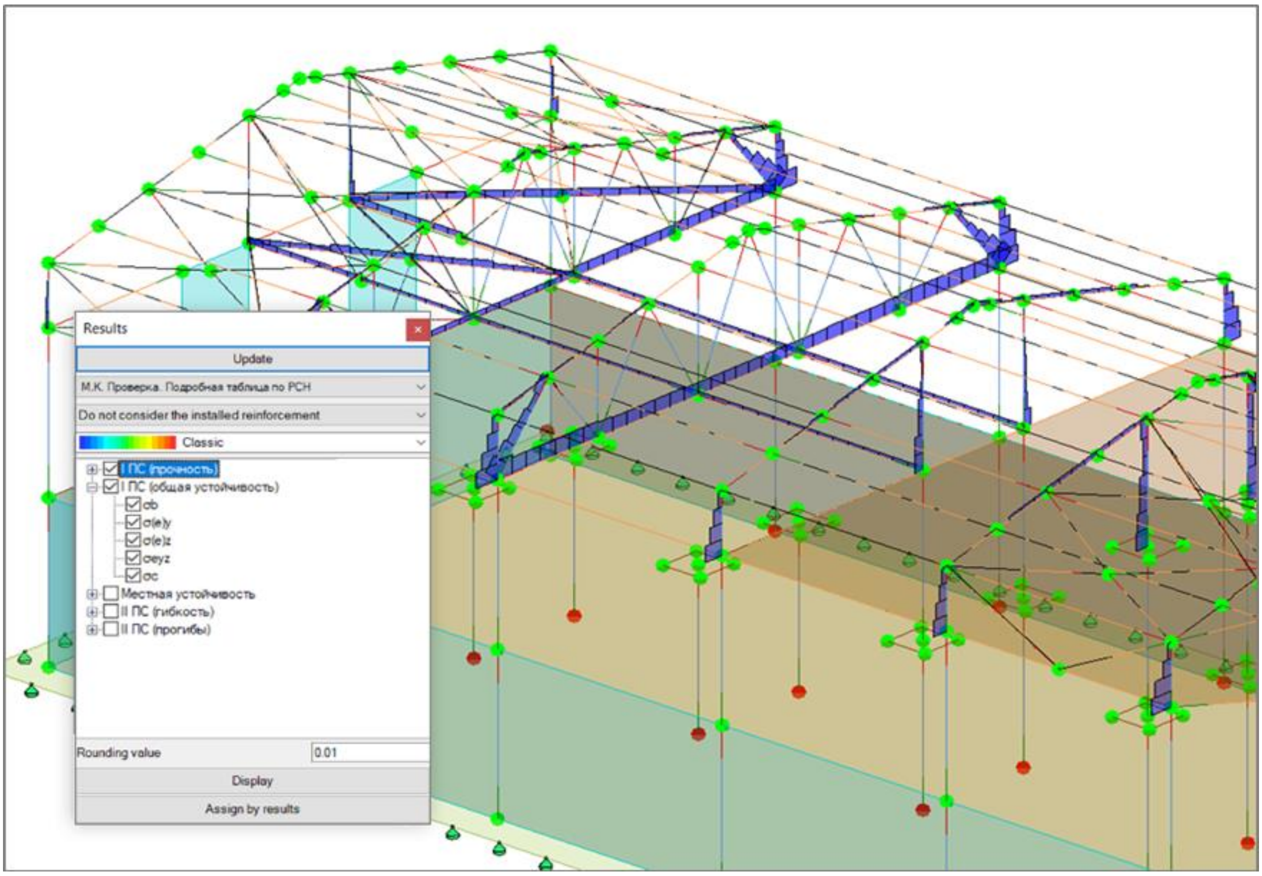


Fig. B.18. The results of metal structures verification

It is possible to consider the installed reinforcement set on the elements in Revit. To do this, the **Consider installed reinforcement** checkbox is to be activated. After that, the results display the difference between the selected and installed reinforcement (Fig. B.19).

*If the **Consider installed reinforcement** checkbox is activated, and the reinforcement values on the scale are negative, then the initially installed reinforcement (using Revit tools) has a safety factor greater than that calculated in SP LIRA 10.*

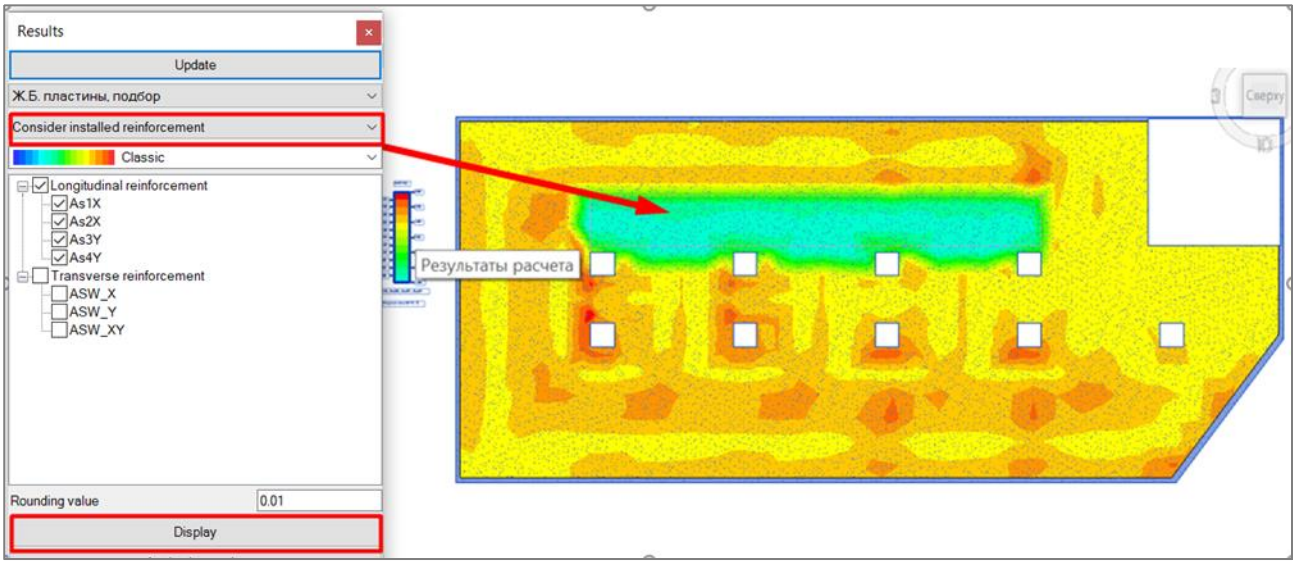


Fig. B.19. The results of the selection of reinforcement, considering the already existing one

The **Assign by result** button allows you to create reinforcement meshes in plate elements based on the results of reinforcement selection (Fig. B.20). The position of the created reinforcement meshes can be adjusted using the following parameters:

- **Full reinforcement** — defines the minimum value of the selected reinforcement, for which the installation of an additional reinforcement element is required.
- **Max distance between nodes** (finite element model) — determines the distance at which the results of finite elements will be combined into groups to build reinforcement elements.
- **Min reinforcement area** — allows you to limit the creation of reinforcing elements for small areas.
- **Distance between bars** — allows you to set the distance between the reinforcing elements.

From the list below, a specific diameter and class of reinforcement can be selected or several types can be included, and the program will select the reinforcement according to these parameters.

The check marked **Simple contours of reinforcement** simplifies the contours of reinforcement areas to a rectangle.

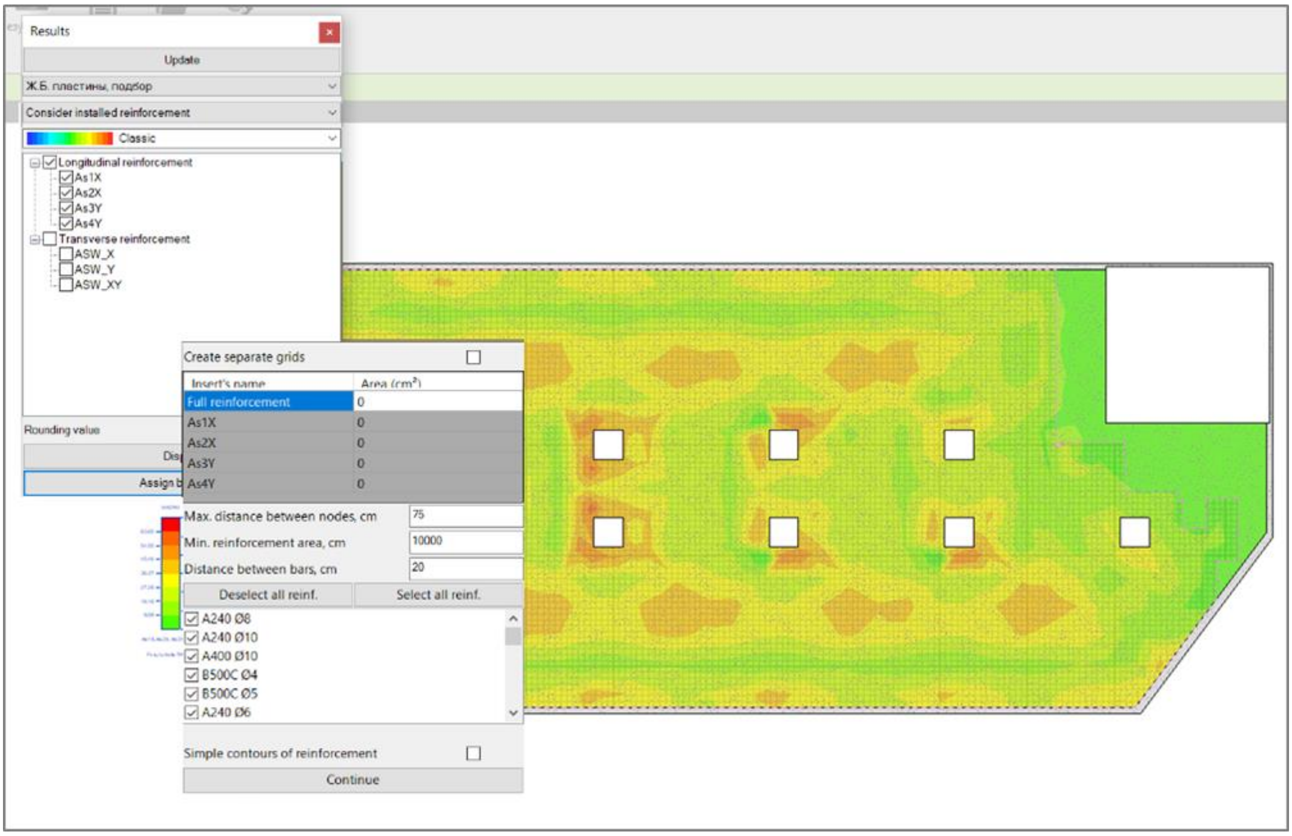


Fig. B.20. Calculation results-based reinforcement

It is also possible to export the results of the punching calculation in the form of a punching contour and required transverse reinforcement (Fig. B.21).

When exporting column elements (Fig. B.22), you can:

- export them as piles;
- set the application of loads on the surface, not snapping to the element;
- indicate whether ARBs (absolutely rigid bodies) will be created during triangulation.

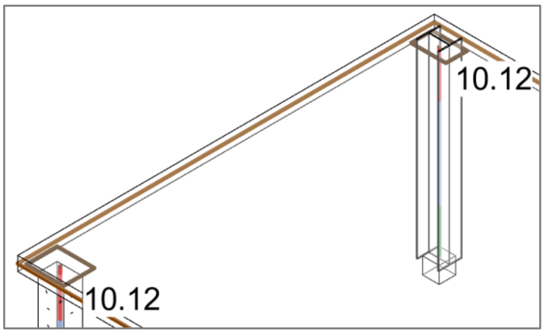


Fig. B.21. Punching contour

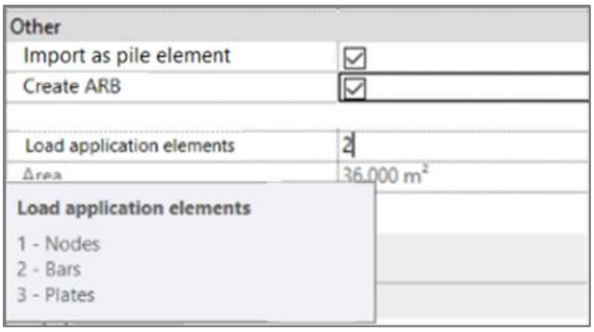


Fig. B.22. Export of column elements

B.7 TEKLA STRUCTURES — LIRA 10 INTEGRATION

SP LIRA 10 provides for the exchange of model data with Tekla Structures. The connection between the programs is implemented by a plug-in that is built into Tekla Structures.

An additional plug-in is installed during the installation of SP LIRA 10 considering Tekla Structures is already installed.

To start connection, an analytical model in Tekla Structures is to be created. To do this, open the **Analysis & Design** submenu and call the **Analysis & Design Models** dialog box (Fig. B.23) using the relevant button.

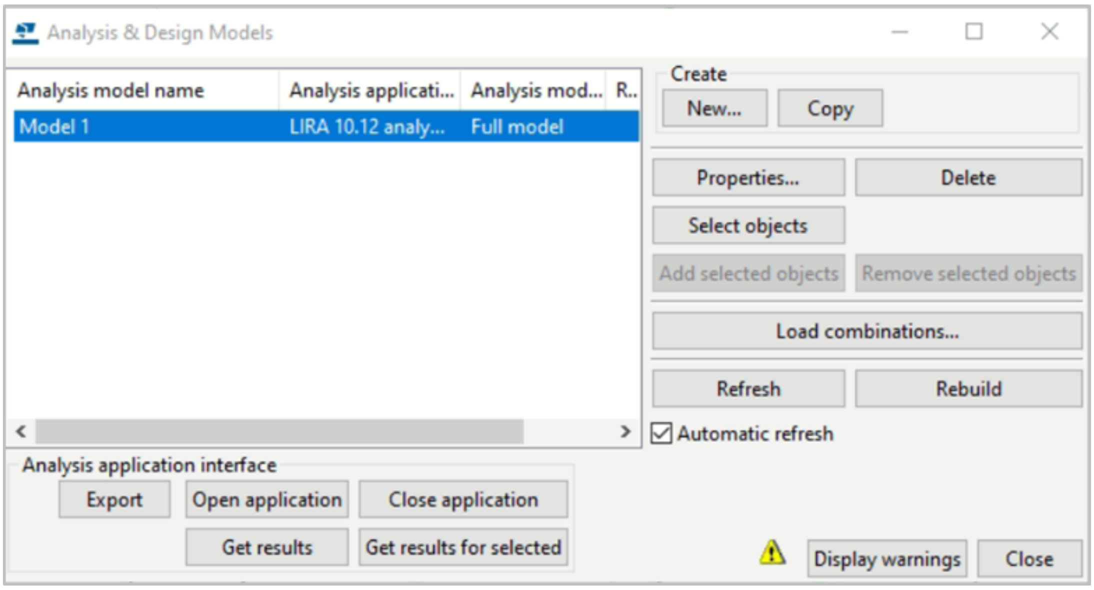


Fig. B.23. Dialog box **Analysis & Design Models**

In the window that opens, click the **Create** ⇨ **New** button to call up the **Analysis Model Properties** dialog box (Fig. B.24). Next, select LIRA 10 as the analysis application, configure the rest of the settings, and click **OK**.

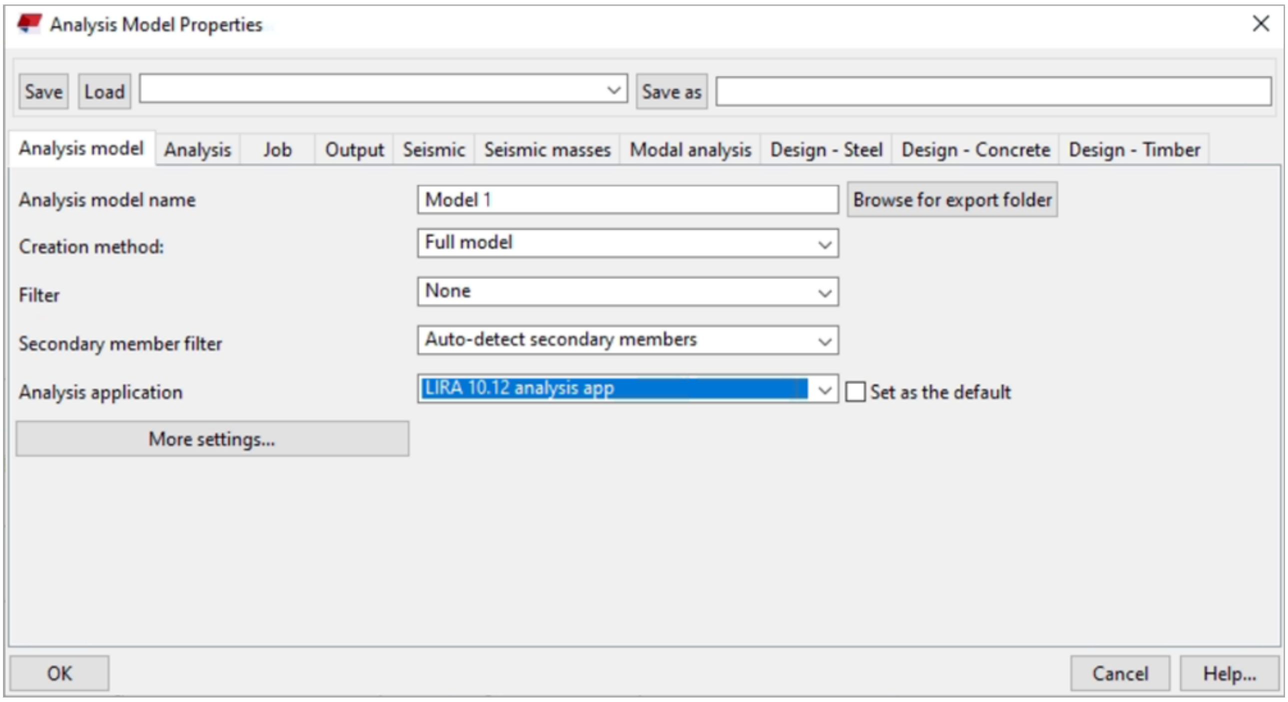


Fig. B.24. Dialog box **Analysis Model Properties**

Several different analytical models can be added to the list. Choose the analytical model that suits you. After that, the buttons on the **Analysis application interface** panel will become active.

The **Export** and the **Open Application** buttons are used to export the selected Tekla Structures analytical model to SP LIRA 10. If an export is carried out using the **Open application** button, the model is opened with SP LIRA 10 immediately after export.

After starting the export, an intermediate the **Import Wizard** dialog box is opened (Fig. B.25). It allows to configure imported settings and view mapping logs.

The **Get results** and the **Get results for selected** buttons allow to get steel profile sections from SP LIRA 10 and update the load information. After clicking these buttons, the **Export Wizard** dialog box is being opened (Fig. B.26). It allows to configure export options and view the comparison log history. If the **Find the closest profile in size** check box is selected, SP LIRA 10 searches Tekla Structures' assortments for the closest profile in terms of characteristics, otherwise only profiles that fully match each other will be selected.

*To select steel cross-sections in SP LIRA 10 and update information about them in Tekla Structures, after selecting it`s necessary to use the **Convert results to original data** mode.*

Upon completion of the export of the results, all profiles proposed for replacement can be viewed in the **Optimization Result** window (Fig. B.27).

*The sections in the Tekla Structures model will be replaced only if the change is accepted in the **Optimization Result** window (Fig. B.27).*

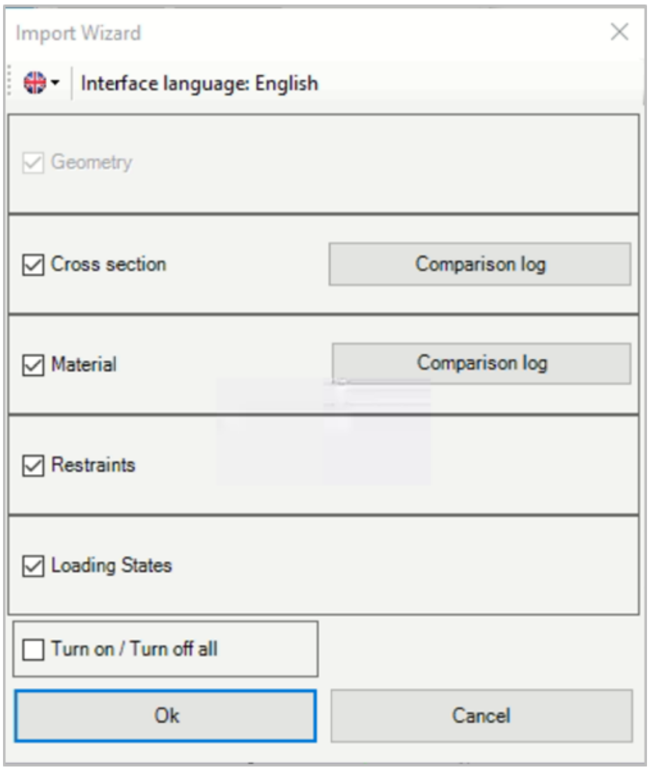


Fig. B.25. Dialog box **Import Wizard**

Fig. B.26. Dialog box **Export Wizard**

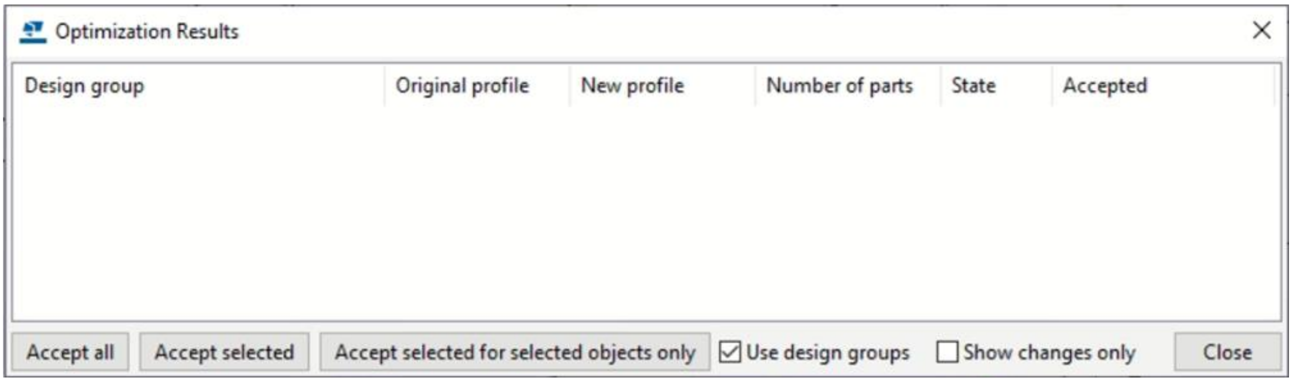


Fig. B.27. Dialog box **Optimization Result**

B.8 IMPORT/EXPORT FROM THE COMMAND LINE

Any file can be opened via the command line, which is supported in SP LIRA 10.12. To do this, in the command line the path of the SP LIRA 10 exe-application should be specified and added the path to the calculation file separated over a space.

Example

There is a 1.fep file in the root of drive D (Fig. B.28). It needs to be exported to 2.dxf.

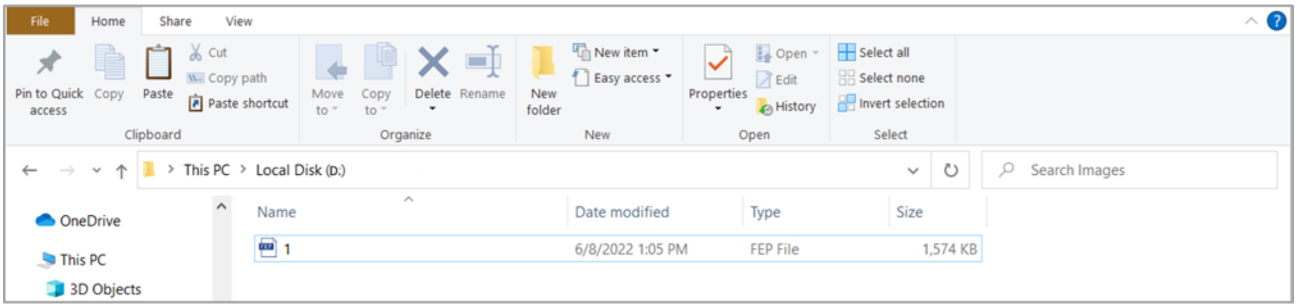


Fig. B.28. The path to the file

In the command line, go to the directory where the *.exe is located — the LIRA 10 file (Fig. B.29): cd C:\Program Files\Lira Soft\Lira10.12\bin64.

Enter:

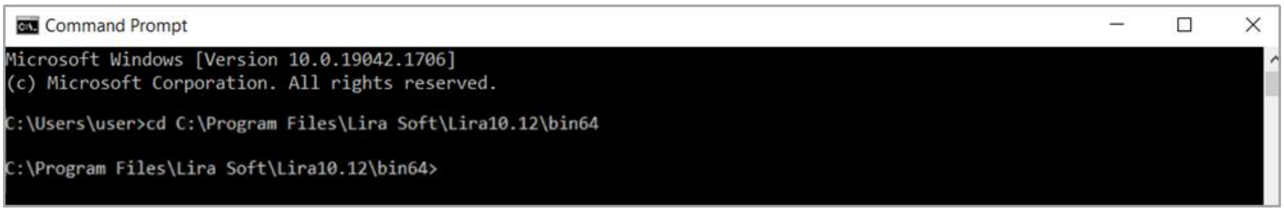


Fig. B.29. An example of entering a command line path

Open the file D:\1.fep (txt, dxf, etc. — any format imported by SP LIRA 10 is possible) and export it to D:\2.dxf.

If you write ***lira.exe "D:\1.fep"***, then the LIRA 10 file opens.

Besides opening a file, you can optionally enter the following commands on the command line over a space:

- **CALCULATION** — start the file for calculation.
- "**Export=B**" — file export to a SP LIRA 10 format. Here **B** is the full path to the file where the model will be exported. For example: *"Export=D:\2.dxf"*.
- **"RUNAPI=ApiName"** is an API application call, where **ApiName** is the name of the application, registered in the application list of SP LIRA 10 installed on the user's computer.
- **EXIT** — close an open file.

Consider some examples of command line entries:

Example 1:

lira.exe "D:\1.fep" "Export=D:\2.dxf"

After pressing **Enter**, the LIRA 10 file will be opened and exported to D:\2.dxf (Fig. B.30).

Fig. B.30. Example 1

Example 2:

lira.exe "D:\1.fep" "Export=D:\2.dxf" EXIT

After pressing **Enter**, the LIRA 10 file will be opened, be exported to D:\2.dxf, and then closed.

Example 3:

lira.exe "D:\1.fep" "Export=D:\2.dxf" CALCULATION EXIT

After pressing **Enter**, the LIRA 10 file will be opened, exported to D:\2.dxf, calculated and closed.

Example 4:

lira.exe "D:\1.fep" "Export=D:\2.dxf" CALCULATION

After pressing **Enter**, the LIRA 10 file will be opened, exported to D:\2.dxf, then calculated.

Example 5:

lira.exe "D:\1.fep" CALCULATION "RUNAPI=SampleLiraAPI" " Export= D:\2.neu" EXIT

After pressing **Enter**, the LIRA 10 file is opened, calculated, the registered API application named ***SampleLiraAPI*** will be launched, the model is exported to D:\2.neu and closed.