



StruSoft StruXML Revit Add-In

What's new in version 1.1.011

What's new in version 1.1.011

Added:

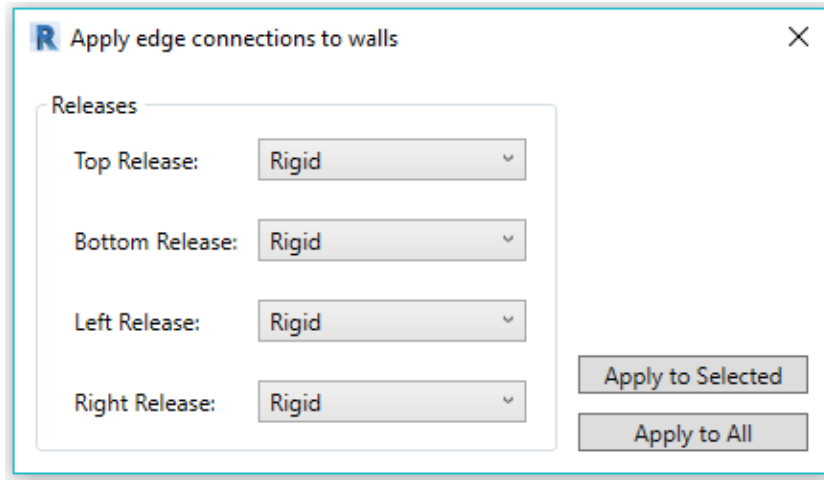
1. Tool to add and export wall edge connections.
2. Tool to export floor as profiled panel or fictitious shell.
3. Export load combinations to struxml.
4. Import struxml names to Revit.
5. Automatic creation of materials and concrete elements (sections, walls, floors) at import.

Changed:

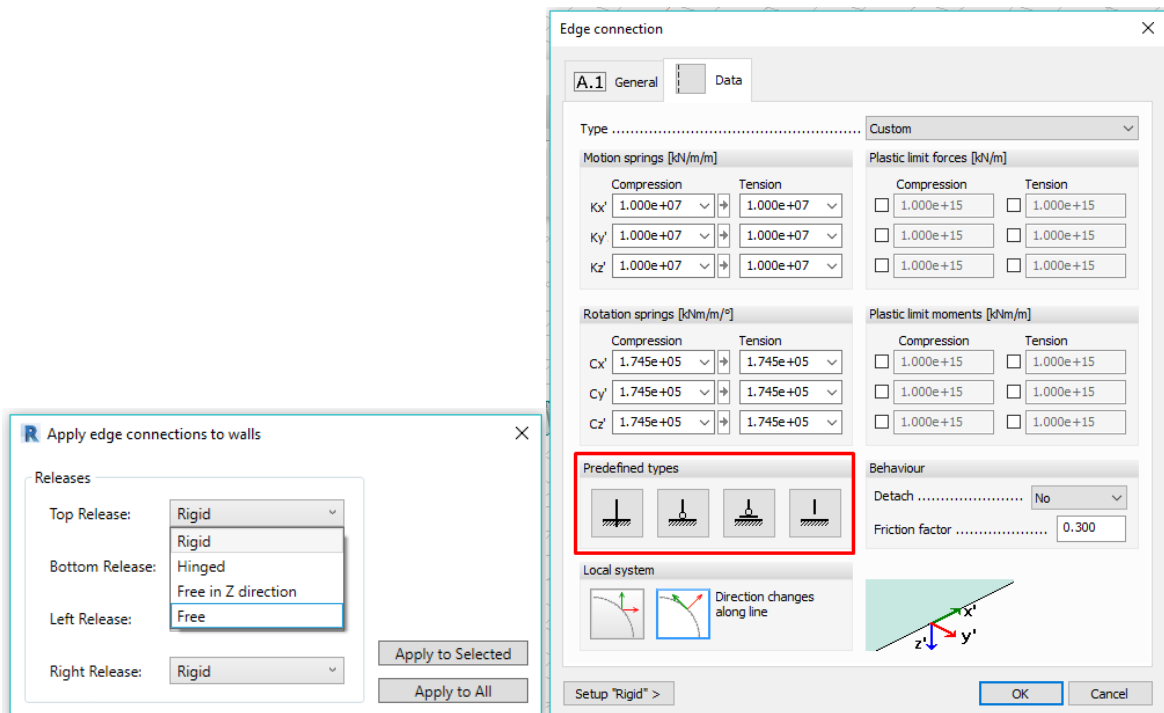
6. Load case assigned to 'Dead Loads' load category is exported to struxml as +Struc. Dead load type.

1. Wall Edge Connections

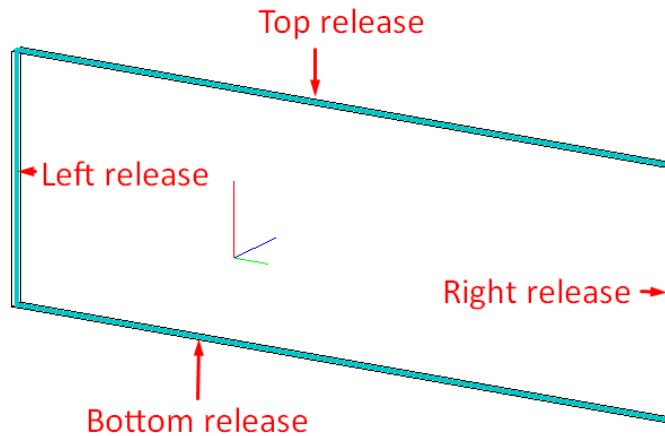
From now on, it is possible to apply edge connections to wall elements, and export walls along with their edge parameters to struxml. Upon clicking on *Wall Edge Connections* (*StruSoft* tab / *Tools* panel), an *Apply edge connections to walls* dialog appears.



One can select one of the four predefined releases to be applied to wall edges. The four predefined releases correspond exactly to the four predefined edge connections in FEM-Design that one can find in the Edge connection dialog.



Set of edge releases can be applied to selected wall(s) or to all walls in the project. The Top / Bottom / Left / Right releases are applied according to following schema:



The Right and Left releases are solely dependent on the wall's coordinate system. The Right release is applied to that edge of the wall that is in the positive directions of the y (green) in-plane axis. The Left release is applied to that edge of the wall that is in the negative directions of the y (green) in-plane axis. So the coordinate system of the wall decides on the Left / Right release position!

The releases are applied to walls as Shared parameters and are only read-only. By default they are applied to both *Wall* and *Analytical wall* and are located under *Structural Analysis* tab. One can modify those default settings in the Project Parameters.

Basic Wall	
Generic - 200mm	
Walls (1)	Edit Type
Constraints	⌵
Structural	⌵
Dimensions	⌵
Structural Analysis	⌶
Top Release	Rigid
Bottom Release	Hinged
Left Release	Free in Z direction
Right Release	Free

In order to modify/edit the release conditions, one has to reapply the edge connections on selected walls.

- Select the wall(s) you want to modify and start the Wall Edge Connections tool.
- The tool will display the releases applied to the selected wall(s).
- Now you can modify the releases.

The shared parameters can be added to the schedules.

<Wall Schedule>				
A	B	C	D	E
Family and Type	Top Release	Bottom Release	Left Release	Right Release
Basic Wall: CW 102-85-215p				
Basic Wall: CW 102-85-215p	Rigid	Hinged	Rigid	Rigid
Basic Wall: CW 102-85-215p	Rigid	Hinged	Rigid	Rigid
Basic Wall: Generic - 200mm				
Basic Wall: Generic - 200mm	Rigid	Hinged	Rigid	Rigid
Basic Wall: Generic - 200mm	Rigid	Hinged	Rigid	Rigid
Basic Wall: Generic - 200mm	Rigid	Hinged	Rigid	Rigid
Basic Wall: Generic - 200mm	Rigid	Hinged	Rigid	Rigid

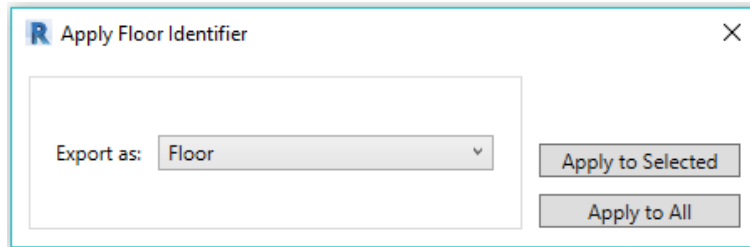
Once applied, edge connections will be exported automatically with the walls to struxml.



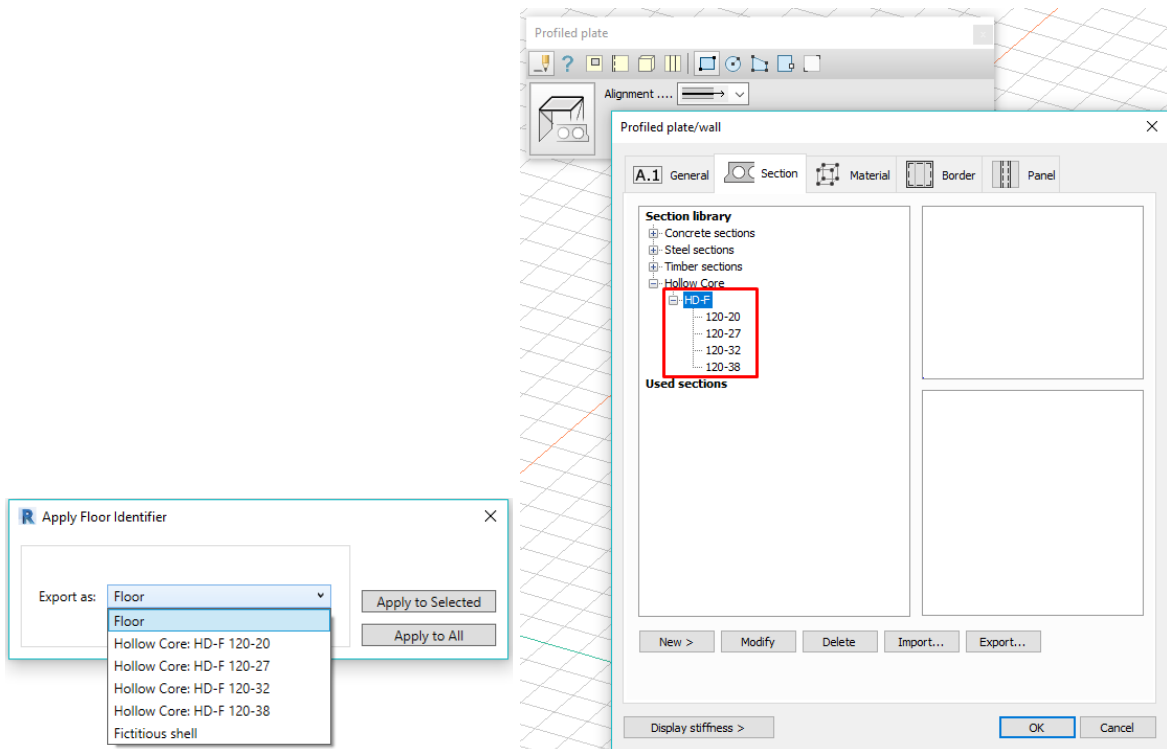
Adding the *Wall Edge Connections* parameters, requires presence of shared parameter file, into which, the new parameter can be added.

2. Floor Identifier

From now on, there is an option to export floor as profiled panel or fictitious shell. Upon clicking on *Floor Identifier* (*StruSoft* tab / *Tools* panel), an *Apply Floor Identifier* dialog appears.



One can choose to export a floor as one of the four predefined profiled panel types or as a fictitious shell. The four predefined Hollow core types correspond to the four predefined hollow core sections available in the default Profiled plate library in FEM-Design.



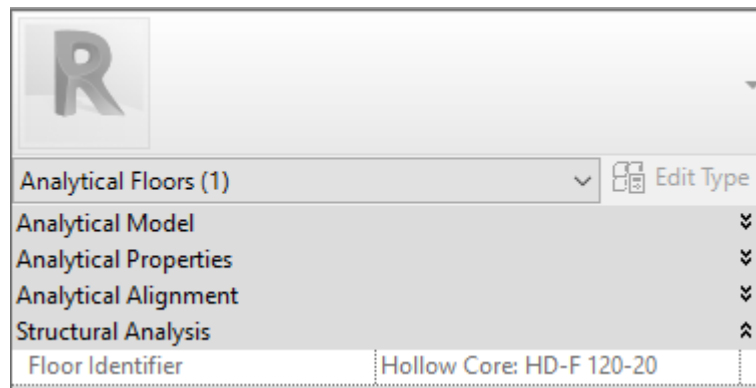
In case one chose the export floor as one of the hollow core profiles, following properties are exported:

- Geometry (contour) of the floor
- The selected hollow core profile
- Span direction
- Material

All the other parameters are set to default in FEM-Design.

In case once chose the export floor as fictitious shell, only geometry (contour) of the floor is exported. All the other parameters are set to default in FEM-Design.

The Floor Identifier is applied to floors as Shared parameters and is read-only. By default it is applied to both *Floor* and *Analytical floor* and is located under *Structural Analysis* tab. One can modify those default settings in the Project Parameters.

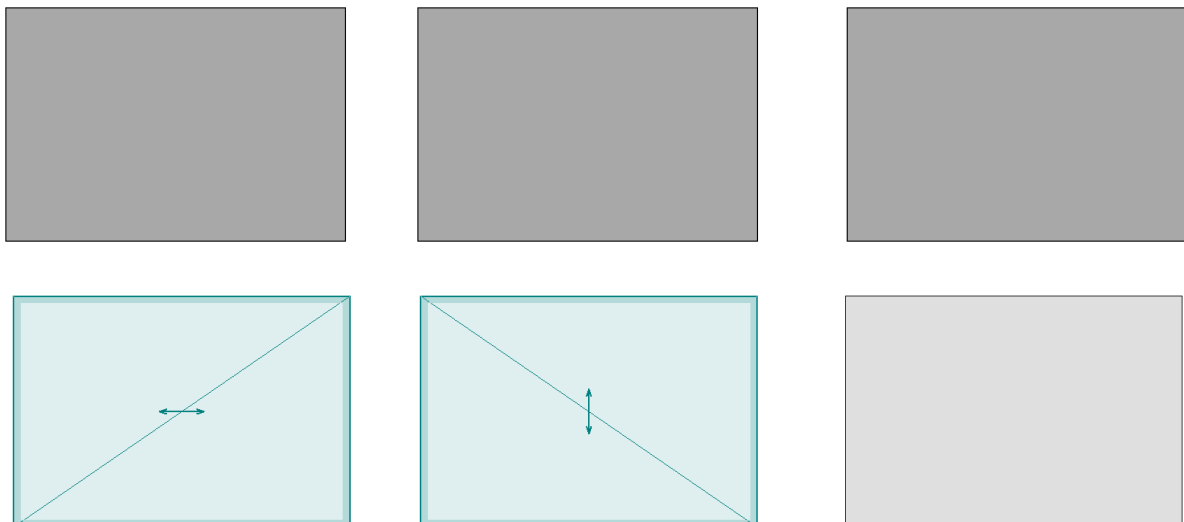


In order to modify/edit the floor identifier, one has to reapply it on selected floor(s).

- Select the floor(s) you want to modify and start the Floor Identifier tool.
- The tool will display the current setting applied to the selected floor(s).
- Now you can modify the settings.

Example of three floors exported from Revit to FEM-Design with following settings:

Hollow Core HD-F 120-20 (- span) | Hollow Core HD-F 120-32 (| span) | Fictitious shell



Applying Floor Identifier, requires presence of shared parameter file, into which, the new parameter can be added.

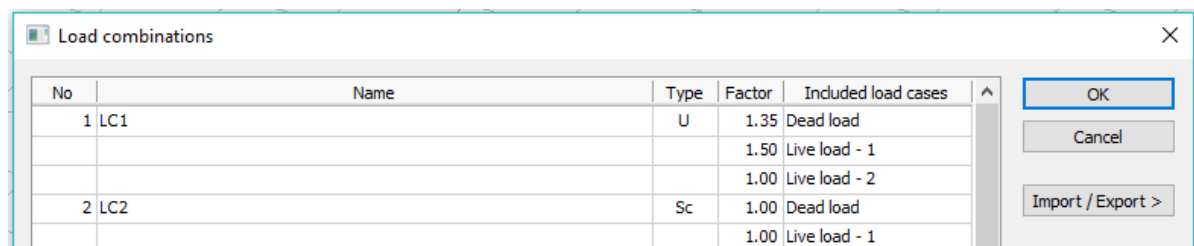
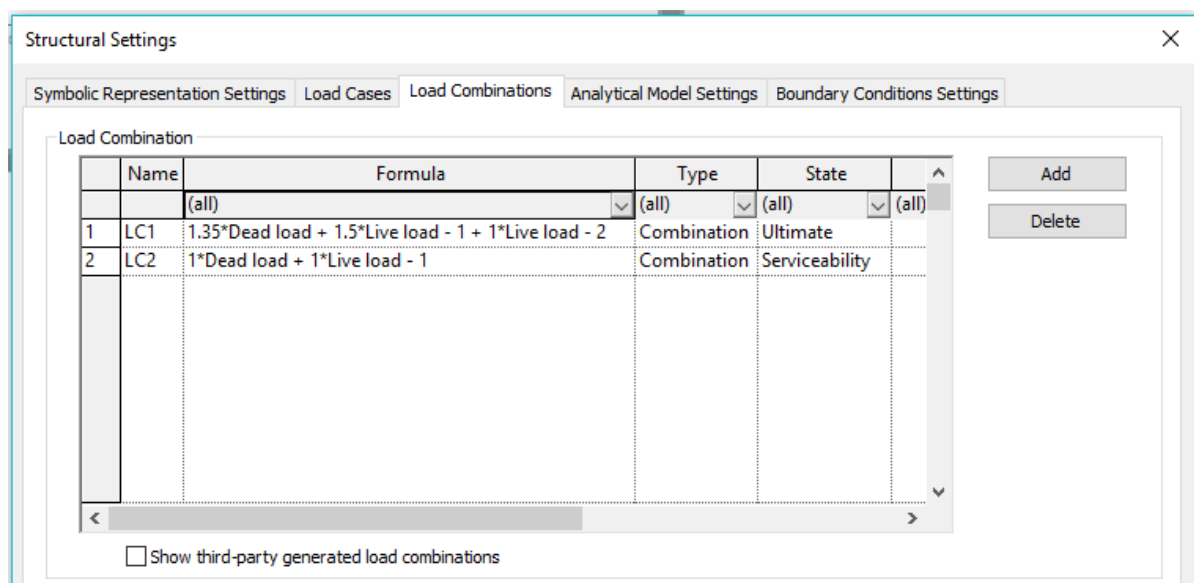
3. Load combinations

From now on, it is possible to export Load combinations from Revit to struxml. In order to export load combinations, one should select the *Loads and load comb.* to be exported in the *StruXml Export* dialog.

Following load combinations parameters are exported:

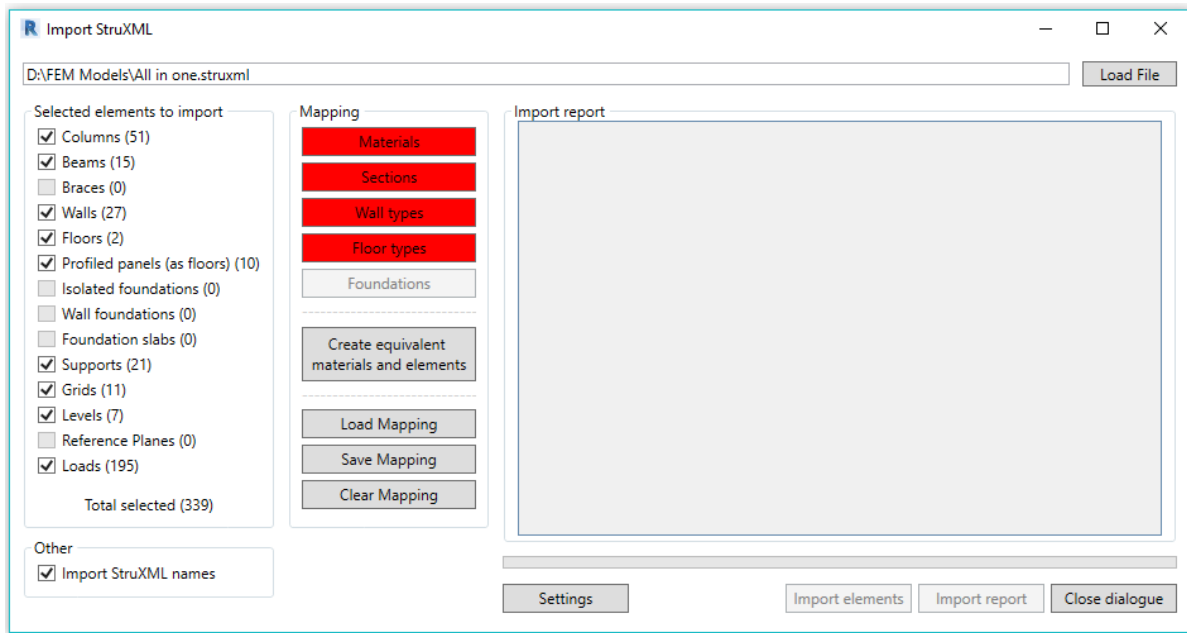
- Name
- Formula
- State
 - o Ultimate corresponds to Ultimate Limit State in FEM-Design,
 - o Serviceability corresponds to Serviceability - characteristic limit state in FEM-Design.

Type and Usage are ignored.

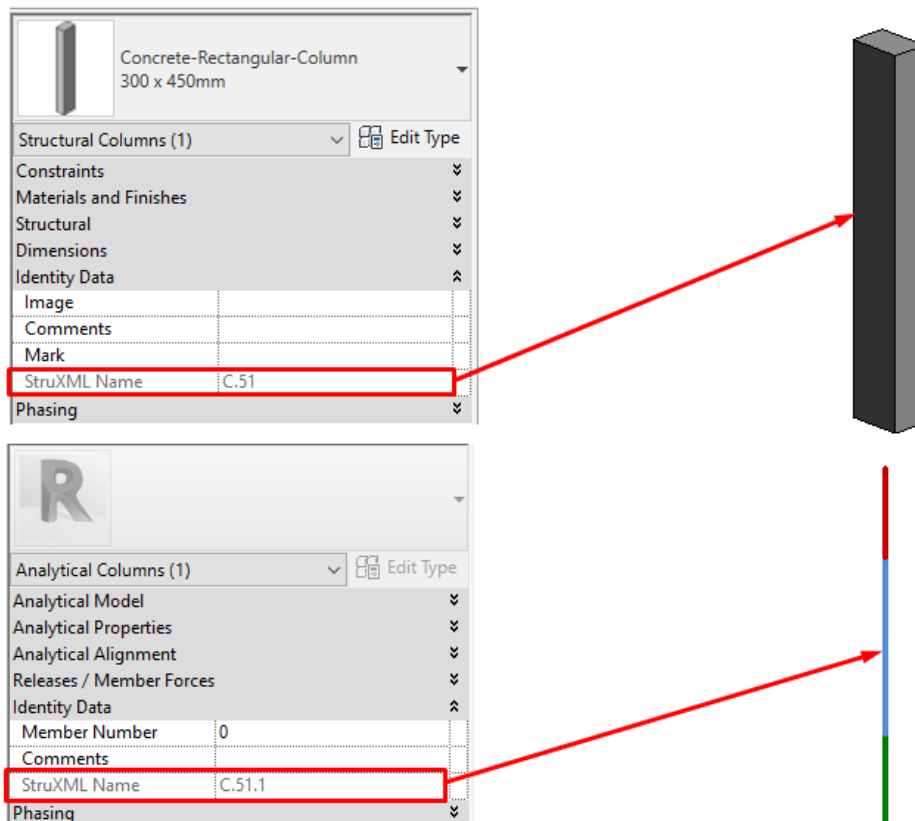


4. Import StruXML names

Press Import StruXML names, if you want to add *StruXML Name* parameter into each imported structural element.



The *StruXML Name* is applied to analytical and physical elements as Shared parameters and is read-only. By default it is located under *Identity Data* tab. One can modify it in the Project Parameters. Physical elements ID can only be imported starting from FD 17.01.001.



5. Automatic mapping

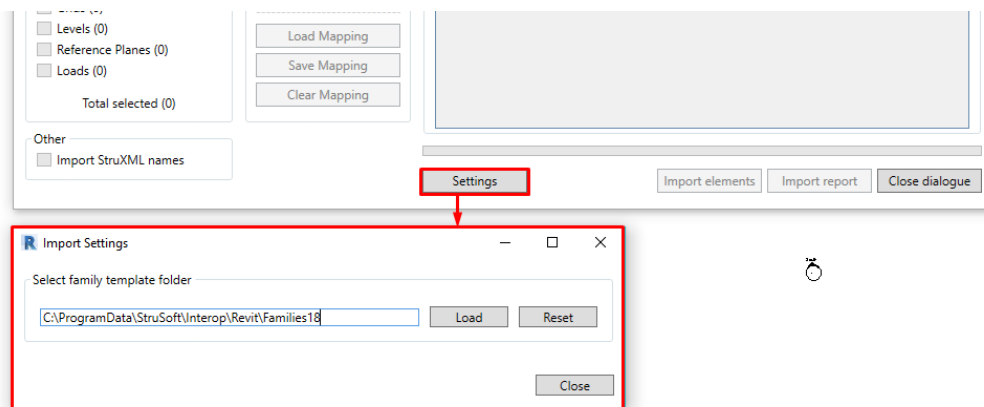
From now on it is possible to automatically generate desired materials, concrete sections types and walls / slabs instances in Revit project. The purpose of this feature is to speed up the import process in case of typical concrete sections and wall/floor elements that so far, always had to be created manually in Revit before the import process. The automatic section creation can be combined with traditional manual mapping.

In this version, the automatic section generation is only available for following sections:

- Concrete rectangle beam
- Concrete rectangle column
- Concrete round column

Sections are created as types of predefined families that are by default installed with the Add-In in following location: C:\ProgramData\StruSoft\Interop\Revit\Families18 (and Families 17 folder for Revit 2017).

It is possible to change location of the families. Simply go to Settings and select new path, as shown in below figure.



One can use their own families, but in this version the requirement is that the naming will be consistent with the default families naming. In the future version we will introduce option to load user families. Also, it is important that the family definition, especially the dimensions b , h , D will be consistent with those in the default families.

The concrete materials, section types and walls and floors are created upon pressing *Create equivalent materials and elements*.

Check the examples below, to understand the naming convention:

Materials:

FD Materials	Revit Materials
concrete: C30/37	Concrete - C30/37
steel: S 355	Steel - S 355

Sections:

Concrete sections are created based on the FEM-Design section naming, e.g. 200x300. This is the only acceptable format. In case of different name in struxml, section cannot be created automatically and has to be mapped manually.

The same happens in case of any other section than those that are supported at the moment (Concrete rectangle beam, Concrete rectangle column, Concrete round column) e.g. steel section. These sections have to be mapped manually.

Section mapping	
Beam mapping	
FD Sections	Revit Sections
Concrete sections: Rectangle; 200x300	StruSoft - Concrete - Rectangle Beam: 200 x 300mm
Column mapping	
FD Sections	Revit Sections
Concrete sections: Rectangle; 200x550	StruSoft - Concrete - Rectangle Column: 200 x 550mm
Steel sections: HE-A; 240	
Ok	

Walls:

Wall type mapping	
Wall type mapping	
FD Types	Revit Types
concrete: C30/37 - 200 mm	Basic Wall: C30/37 - 200 mm
concrete: C30/37 - 250 mm	Basic Wall: C30/37 - 250 mm
Ok	

Floors:

Floor type mapping	
Floor type mapping	
FD Types	Revit Types
concrete: C30/37 - 300 mm	Floor: C30/37 - 300 mm
Panel type mapping	
FD Types	Revit Types
Hollow Core, HD-F, 120-20: C40/50	Floor: Hollow Core, HD-F, 120-20, C40/50
Ok	

6. 'Dead Loads' load category

If the Category is set to Dead Loads, the load case will be exported as load case *Type: +Struc. Dead load*. All the other load cases exported from Revit to FEM-Design will be assigned with the *Type: Ordinary*.

Example of loads cases defined in Revit and exported to FEM-Design is shown below.

