StruSoft

# StruSoft StruXML Revit Add-In

## Manual: Using Revit – FEM-Design link

StruSoft AB
Fridhemsvägen 22
SE-217 74, Malmö, Sweden
www.strusoft.com

Version: June 28th, 2018

# Copyright

Copyright© 2018 by StruSoft. All rights reserved.

Content of this publication may not be reproduced or transmitted in any means without the written permission of Structural Design Software in Europe AB.

## Trademarks

FEM-Design ® is a registered trademark of StruSoft. Revit ® is a registered trademark of Autodesk.

# Disclaimer

The StruSoft StruXML Revit Add-In is a tool that enables a link between Revit and FEM-Design. However, the user must understand the assumptions and restrictions that are described in this document.

Considerable time and effort have gone into development and testing of the StruSoft StruXML Revit Add-In. We have done our best to ensure the reliability of the software and the accuracy of this document. However, the user must accept that no warranty is given by the developers or distributors concerning accuracy of this software or information found in this document.

Anyone that has doubts concerning the accuracy of the StruSoft StruXML Revit Add-In, or has suggestions regarding development of the StruSoft StruXML Add-In, is welcome to contact us at: iwona.budny@strusoft.com.

For support, please use: support.femdesign@strusoft.com. When sending support question, please remember to always attach an original Revit / FEM-Design model, struxml file, and in case of Import to Revit – a Revit rvt file, as well us explain which version of FEM-Design and StruSoft StruXML Revit Add-In have been used.

## Current plugin versions

| | |
|---|---|
| For FEM-Design 17 | - StruSoft StruXML Revit 2017 Add-In 1.1.011<br>StruSoft StruXML Revit 2018Add-In 1.1.011<br>- StruSoft StruXML Revit 2019 Add-In 1.1.011 |
| For FEM-Design 16 | - StruSoft StruXML Revit 2017 Add-In 1.1.009<br>- StruSoft StruXML Revit 2018 Add-In 1.1.009 |

## Compatibility

- Revit/Revit Structure: version 2017, 2018, 2019

## Download

- FEM-Design Download Center
- StruSoft Installer

# Table of contents


I. INTRODUCTION....3

II. TOOLS....4

1. Analytical Model .... 4
2. Material .... 6
3. Connection status .... 7
4. Wall Edge Connections .... 8
5. Floor Identifier .... 11
6. StruXML Guid .... 13
7. Guid Search .... 16

III. WORKFLOW: REVIT TO FEM-DESIGN .... 17

8. Data exchange scope .... 17

8.1. Structural Columns and Framing .... 17
8.2. Structural floors .... 18
8.3. Structural walls .... 18
8.4. Structural Foundations .... 19
8.5. Material .... 22
8.6. Eccentricity in shell elements .... 22
8.7. Eccentricity in beams .... 25
8.8. Releases of linear elements .... 27
8.9. Boundary conditions .... 27
8.10. Grids .... 28
8.11. Levels .... 29
8.12. Reference planes .... 30
8.13. Loads, load cases and load combinations .... 30

9. Export from Revit to FEM-Design .... 34

9.1. Export tab .... 35
9.2. Materials mapping .... 37
9.3. Sections mapping .... 40
9.4. Export unmapped sections .... 42
9.5. Advanced settings for Dynamo Script .... 45
9.6. Warnings and errors .... 46
9.7. Export report .... 51
9.8. Open StruXML .... 54
9.9. Errors at opening struxml in FEM-Design .... 54

10. Limitation and recognized issues .... 56

IV. WORKFLOW: FEM-DESIGN TO REVIT .... 59

11. Data exchange scope .... 59

11.1. Datum elements .... 59
11.2. Bar elements .... 60
11.3. Shell elements .... 61
11.4. Profiled panels .... 62
11.5. Foundations .... 62
11.6. Supports .... 63
11.7. Loads and load cases .... 64

12. Import FEM-Design model to Revit .... 65

12.1. Requirements: .... 65
12.2. Export FEM-Design model to struxml file .... 65
12.3. Import struxml file to Revit .... 66
12.4. Mapping .... 69
12.5. Warnings and errors .... 75

# Release Notes

## StruSoft StruXML Revit Add-In 1.1.011

### Added:

- Export load combinations to struxml. --->> Check Chapter 308.13.3.
- Tool to add and export wall edge connections. --->> Check Chapter 4.
- Tool to export floor as profiled panel or fictitious shell. --->> Check Chapter 5.
- Import struxml names to Revit. --->> Check Chapter 12.3.
- Automatic creation of materials and concrete elements (sections, walls, floors) at import. --->> Check Chapter 12.4.1
- 'What's new?' section in Help menu.

### Changed:

- Load case assigned to 'Dead Loads' load category is exported to struxml as +Struc. Dead load type. --->> Check Chapter 8.13.1.

## StruSoft StruXML Revit Add-In 1.1.010

### ADDED:

- Compatibility with FEM-Design 17.00.001 struxml schema.
- Advanced settings for Dynamo script: default edge connections are added to each wall edge. --->> Check Chapter 9.5
- Import / export of reference planes to / from Revit.

### CHANGED:

- Improved geometry check at opening struxml in FEM-Design.

## StruSoft StruXML Revit Add-In 1.1.009

### ADDED:

- Compatibility with Revit 2018.
- StruXML Guid tool that adds 'StruXML Guid' parameter to structural elements. --->> Check Chapter 6.
- Help menu is added in the StruSoft panel.

### CHANGED

StruSoft AB | Copyright© 2018 StruSoft. All rights reserved.

- View template is no longer copied from the source view when creating Connection Status view --->> Check Chapter 3.
- Analytical model and Material search can be now filtered to only show structural objects. --->> Check Chapter 1 and 2.

## FIXED:

- Wall foundation material is no longer reported as missing when using Material tool.
- Third party updater error is no longer shown when editing door and window families.
- A new warning is added, if a floor element is missing an analytical geometry.
- 'New Version' dialog is shown, if a new version of the StruSoft StruXML Revit Add-In is available.
- Some unexpected errors at Import are fixed.

# I. Introduction

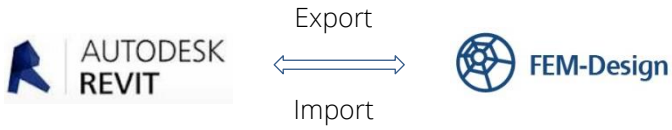
The transfer of data between Revit and FEM-Design is possible thanks to the StruSoft StruXML Revit Add-In. Data are saved into a struxml format and the file is exchanged between Revit and FEM-Design. It is not required that both Revit and FEM-Design are installed at the same computer.

Direction: Revit to FEM-Design

- Export a Revit analytical model to create a new model in FEM-Design.

Direction: FEM-Design to Revit

- Import a FEM-Design model to create a new model in Revit.




## Manual scope

This document describes the concept behind the link between Revit and FEM-Design and explains how to exchange data between those two programs using the StruSoft StruXML Revit Add-In.

The manual is divided into two main parts, each dealing with one particular direction of data exchange. Each part contains description of elements and features that can be transferred, as well as explanation of the transfer procedure.

## Installation

Download the latest version of StruSoft StruXML Revit Add-In, close Revit and run the installation file. After completion of the installation process, StruSoft tab will appear in Revit as shown in Figure I-1.

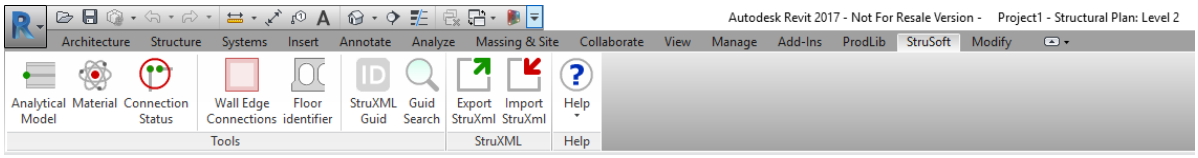



Figure I-1

# II. Tools

Three are five tools available in the StruSoft tab, in the Tools panel. The first three tools are helpful in preparation of the Revit model before the export, and are sort of "shortcut" for actions that can be done manually in Revit. The two latter tools allow adding struxml Guid, and searching for elements using the struxml Guid.

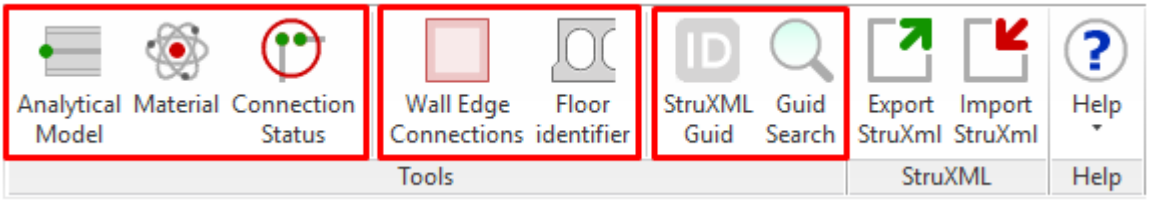



Figure II-1

It is important to understand that the tools do not bring any new functionality to Revit, but only use the existing Revit functions.

## 1. Analytical Model

Click on *Analytical Model* tool in order to check if all structural objects in your model have analytical model enabled. *Analytical model check* dialog will appear with a list of all objects without analytical model, as shown in figure below (if all objects in your model have analytical model enabled, the list will be empty).

If you check "Show only Structural elements", the list will be filtered to show only objects that are structural (this is only important for floors and walls that can be marked as non - structural elements).

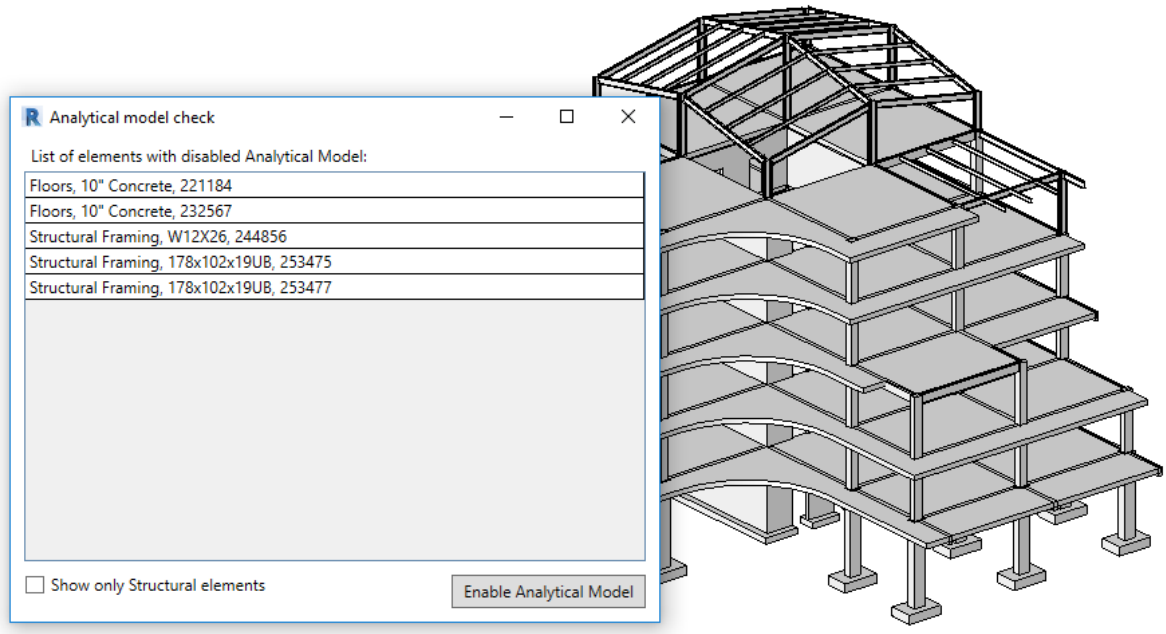



Figure II-2

This tool gives you the following options:

- Double click on one element to highlight it in the model.

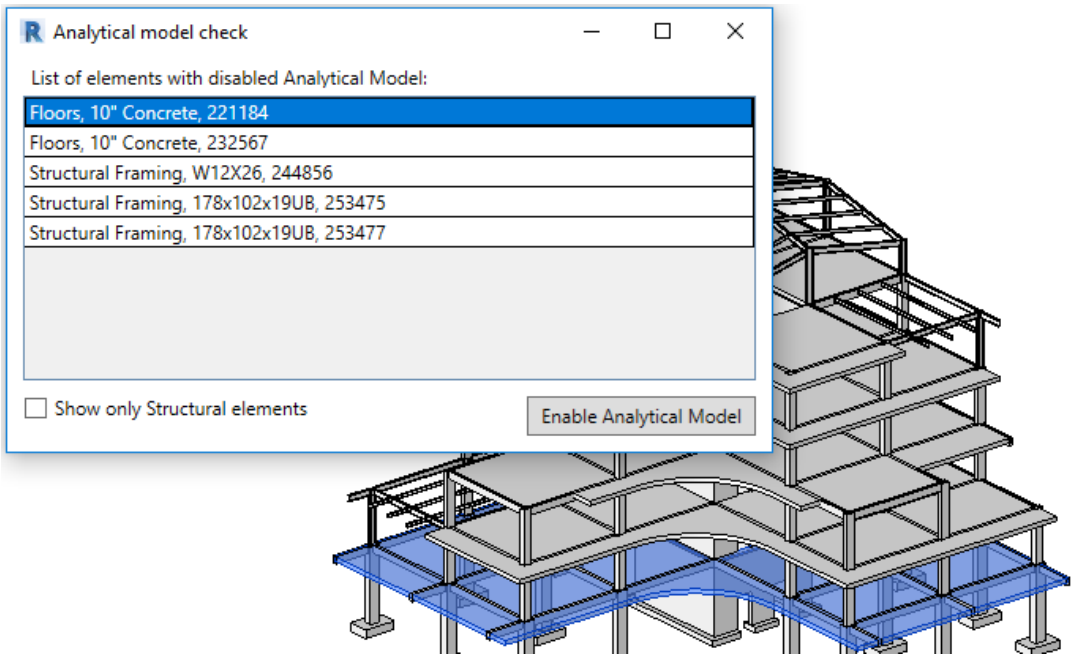



Figure II-3

- Select one or more elements (with Ctrl button) and right click, chose *Isolate selected* to isolate them in a view.

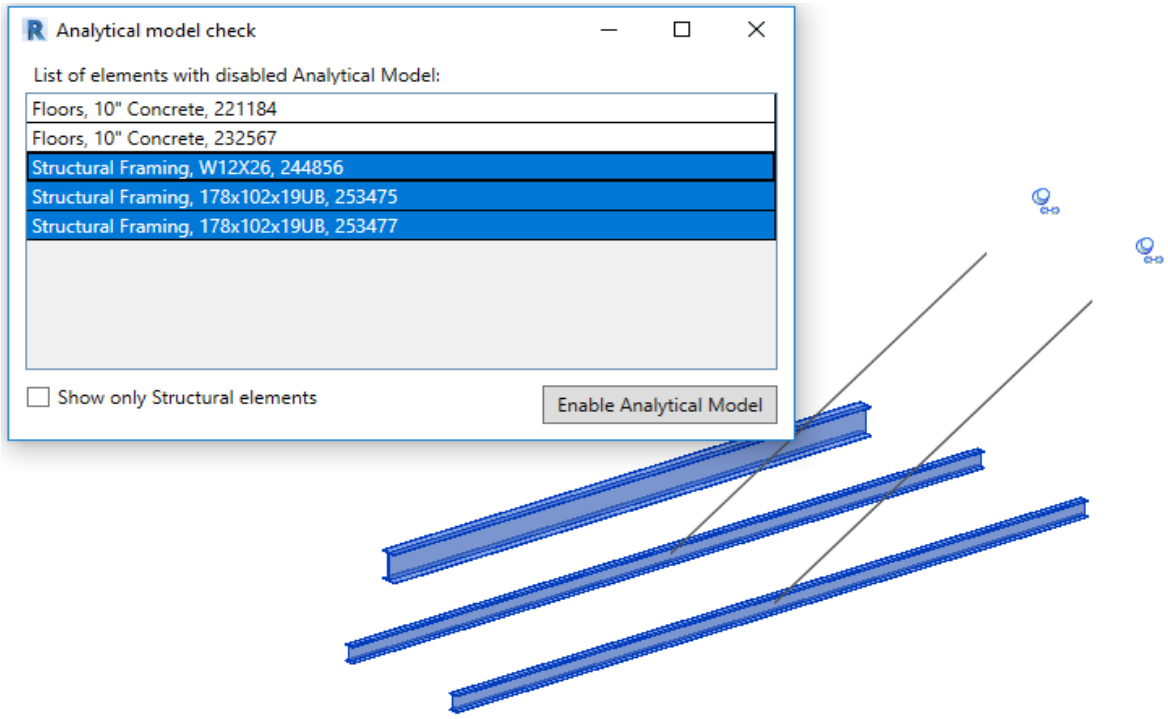



Figure II-4

Close the *Analytical model check* dialog in order to modify the objects in the *Temporary Hide/Isolate* view. To close the view click on *Reset Temporary Hide/Isolate.*

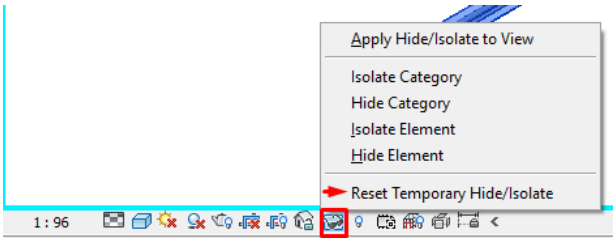



Figure II-5

- Select one, more elements (with Ctrl button), or all elements (Ctrl + A) and press *Enable Analytical Model* in order to enable the analytical model in the selected objects. If the analytical model is enabled in all elements, the dialog becomes empty.

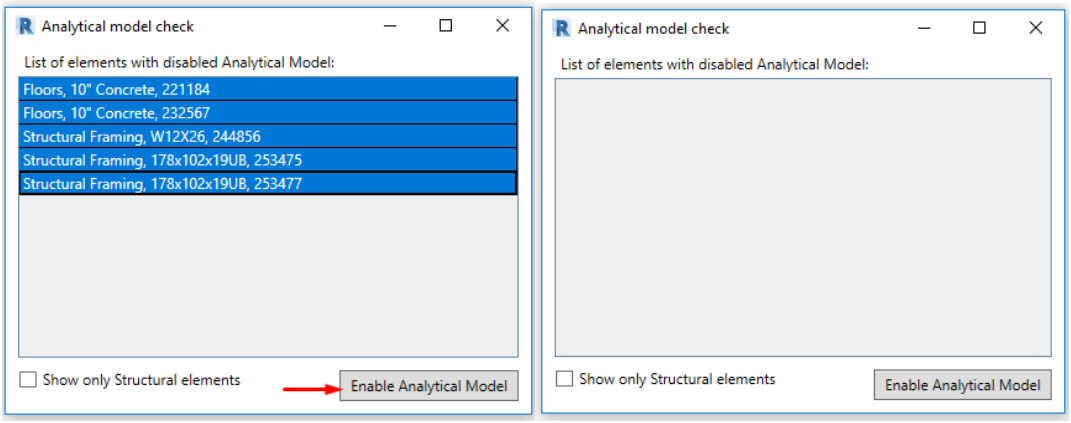



Figure II-6

In case of larger number of elements without the analytical model, it is recommended to enable it partially to a smaller number of objects at a time, rather than enabling it all at once. It requires smaller regeneration of the Revit model and will speed the process up.

## 2. Material

Click on *Material* tool in order to check if all structural objects in your model have valid structural material. *Structural material check* dialog will appear with a list of all objects without valid structural material, as shown in figure below (if all structural objects in your model have valid structural material, the list will be empty).

If you check "Show only Structural elements", the list will be filtered to show only objects that are structural (this is only important for floors and walls that can be marked as non - structural elements).

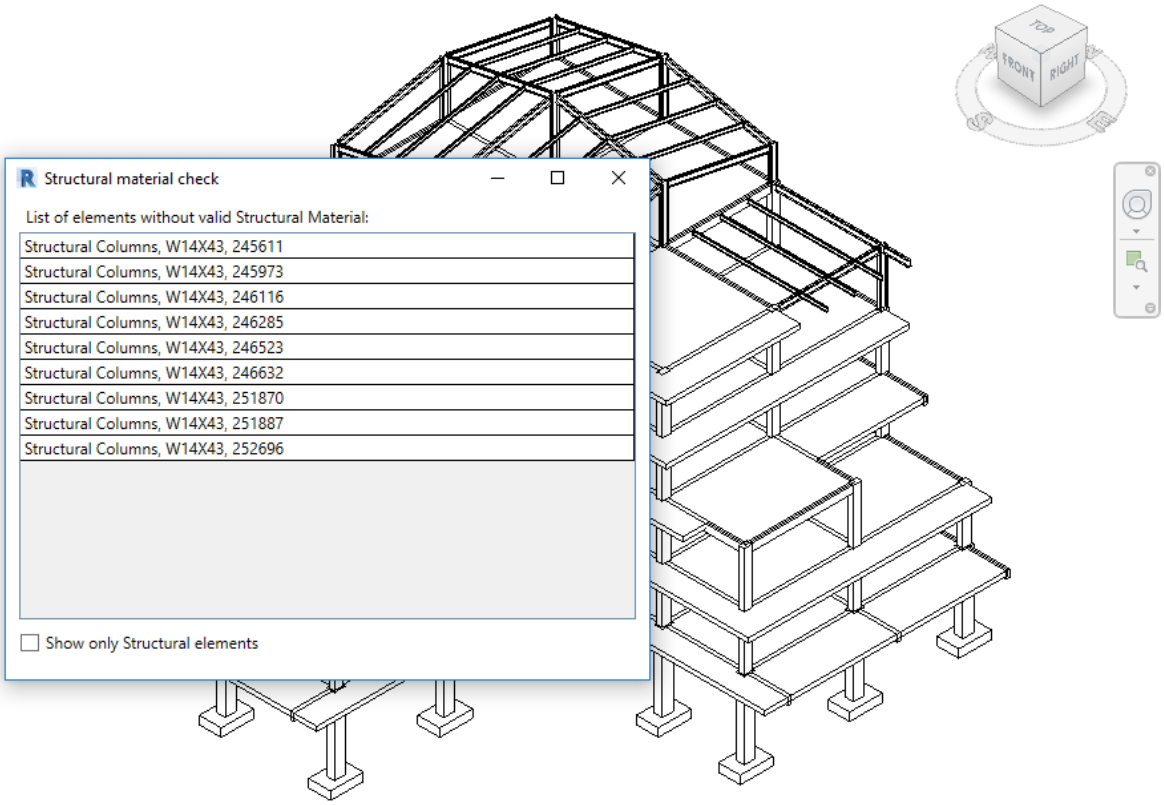



Figure II-7

This tool gives you the following options (partially similar to *Analytical Model* tool):

- Double click on one element to highlight it in the model.
- Select one, more elements (with Ctrl button), or all elements (Ctrl + A) and right click, chose *Isolate selected* to isolate them in a view. Close the *Structural material check* dialog in order to modify the objects (e.g. add the structural material) in the *Temporary Hide/Isolate* view. To close the view click on *Reset Temporary Hide/Isolate.*

## 3. Connection status

Click on *Connection Status* tool in order to create a view called *Connection Status* that will display the analytical model including analytical nodes that are filtered by their connection status (green node – connected, red node – unconnected).

The view is created as a copy of the current view, so if the displayed view is a 3D model view, a new 3D view of Connection Status will be created. Respectively, if a current view is Structural Plan, then a new structural plan showing the connection status will be created. If a current view has some view template applied, it will not be copied to Connection Status view.

The view shows all available analytical objects (excluding the analytical links, boundary conditions and loads).

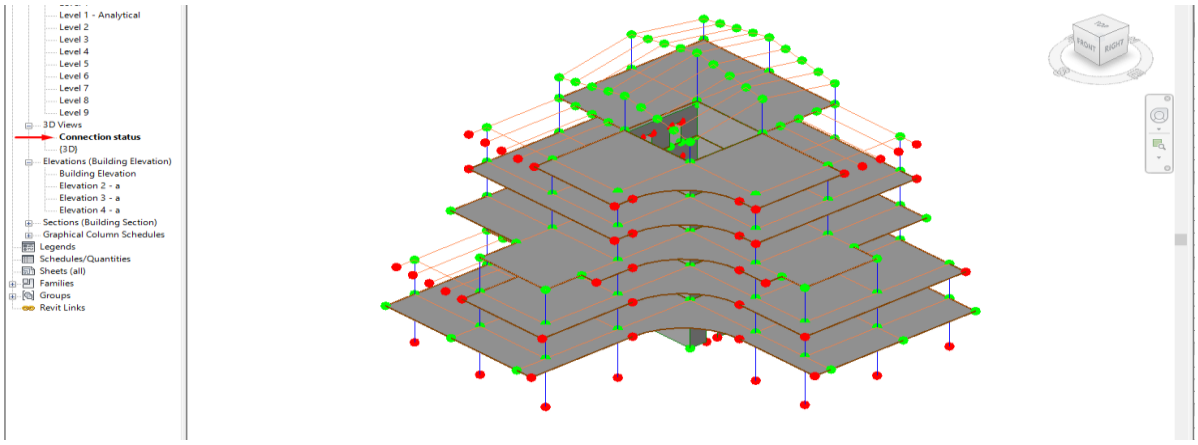



Figure II-8

Two filters are applied to that view in order to distinguish between the connected and unconnected nodes.

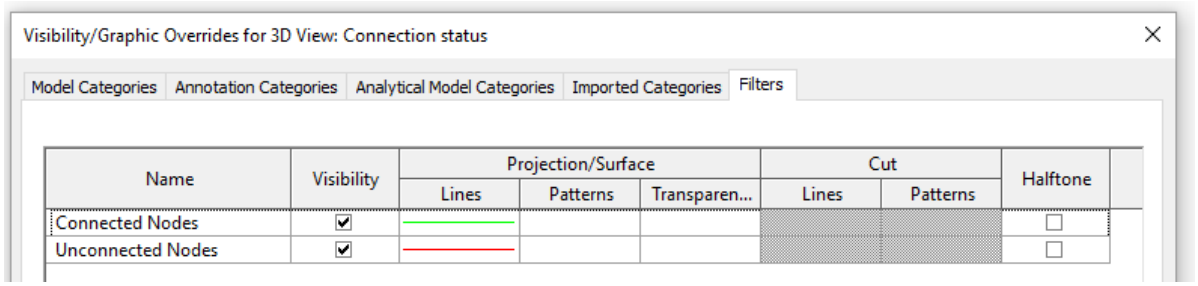



Figure II-9

## 4. 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.

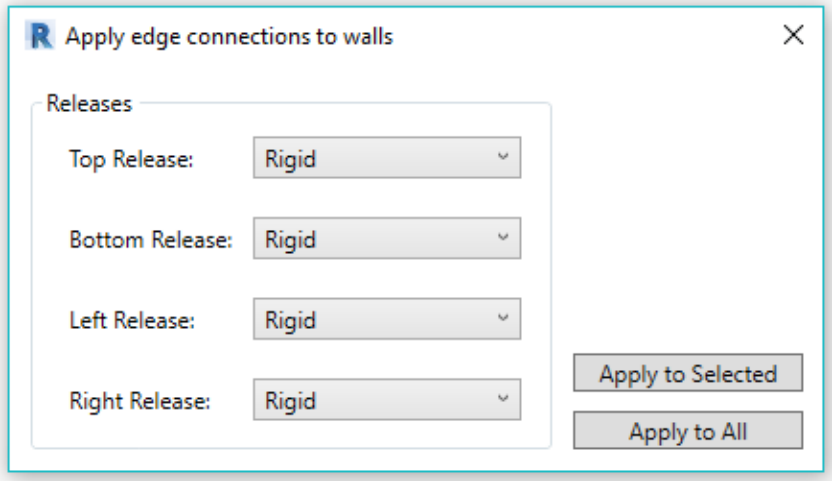



Figure II-10

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.

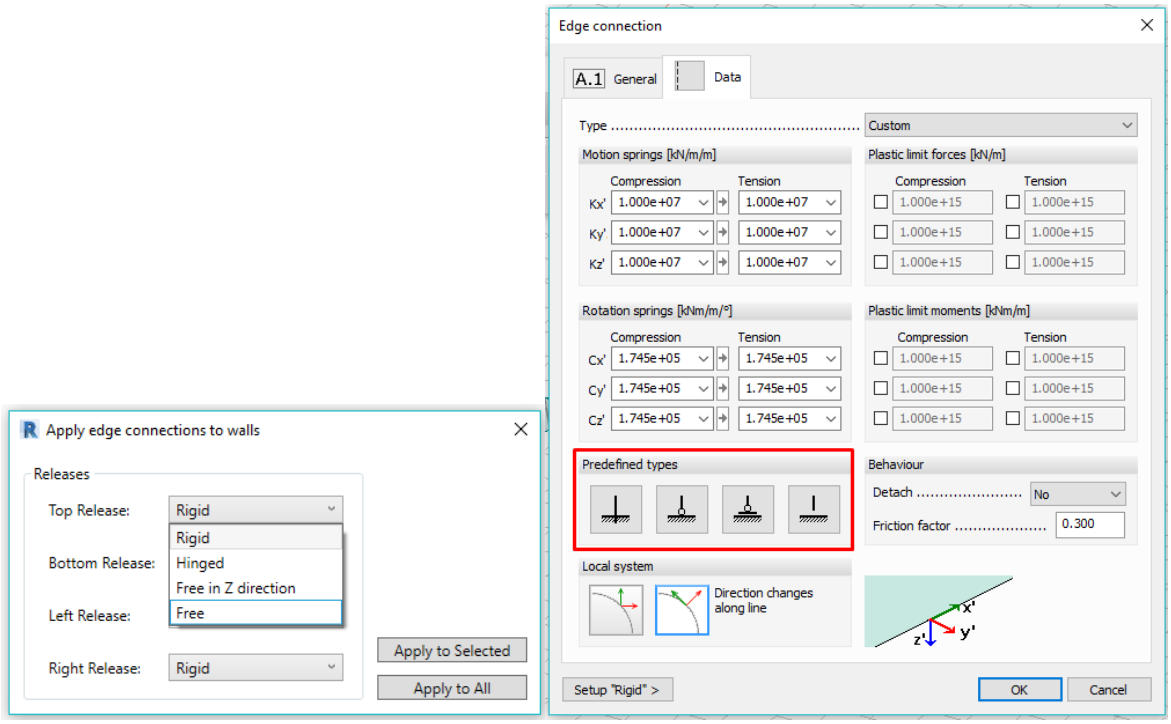



Figure II-11

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:

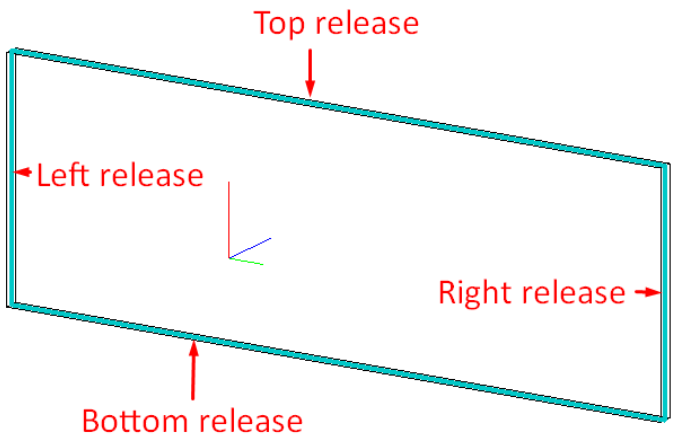



Figure II-12

<u>The Right and Left releases are solely dependent on the wall's coordinate system</u>. 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.

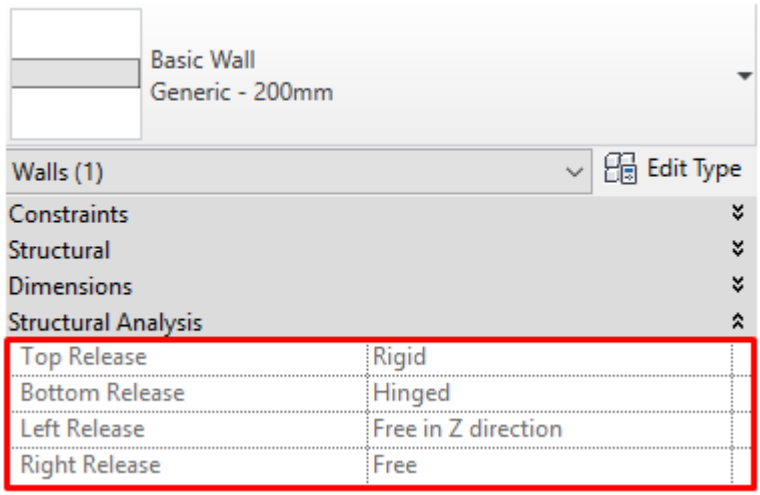



Figure II-13

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 |

Figure II-14

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.

## 5. 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.

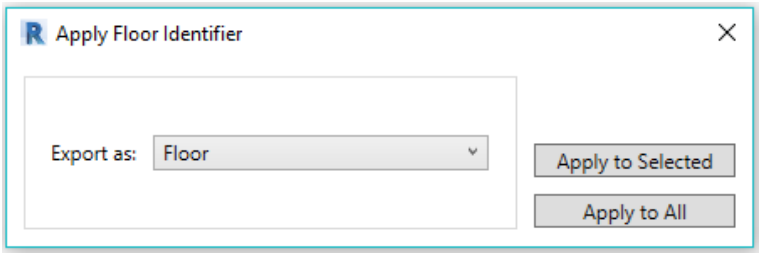



Figure II-15

One can chose 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.

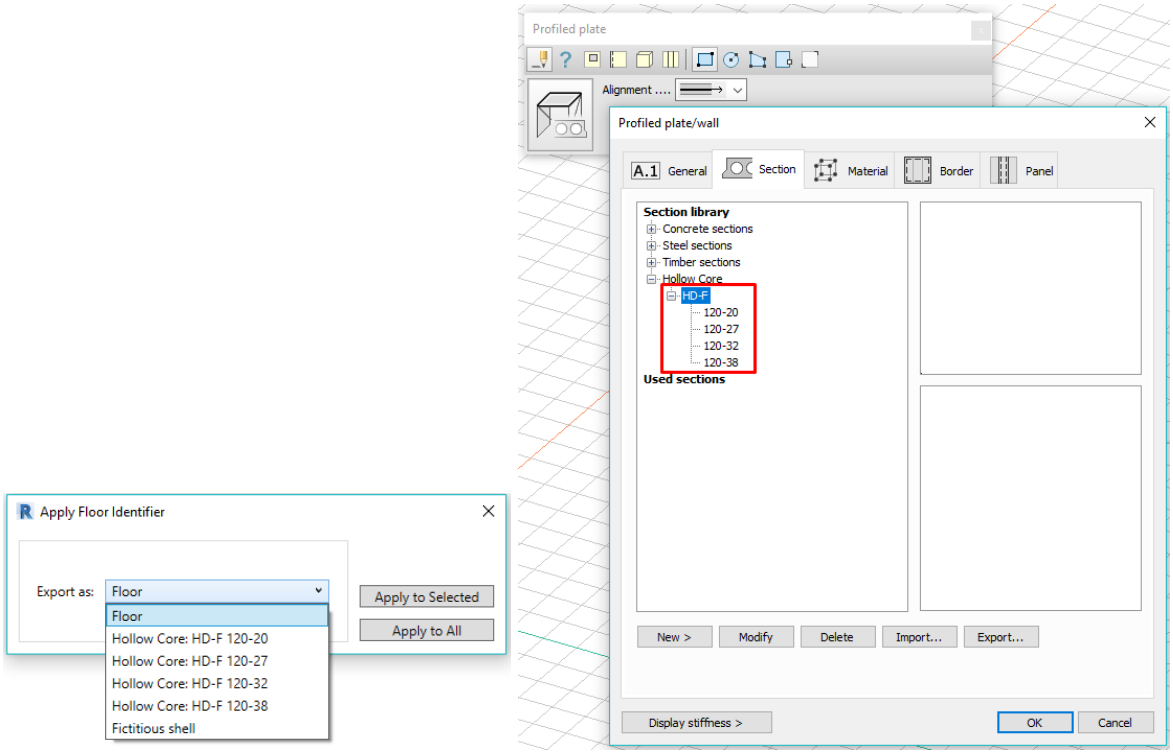



Figure II-16

In case once 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.

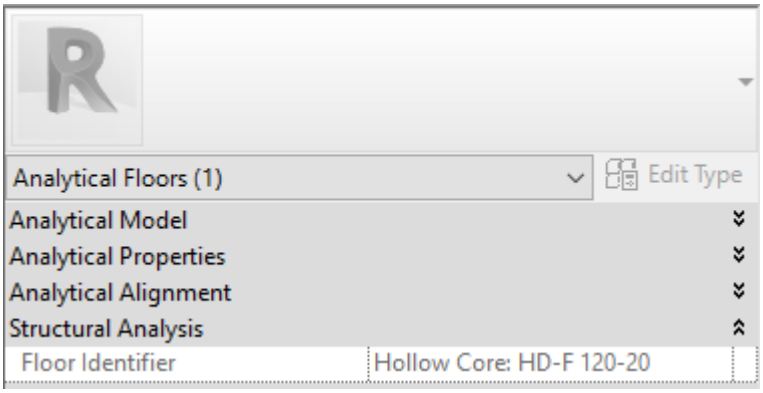



Figure II-17

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

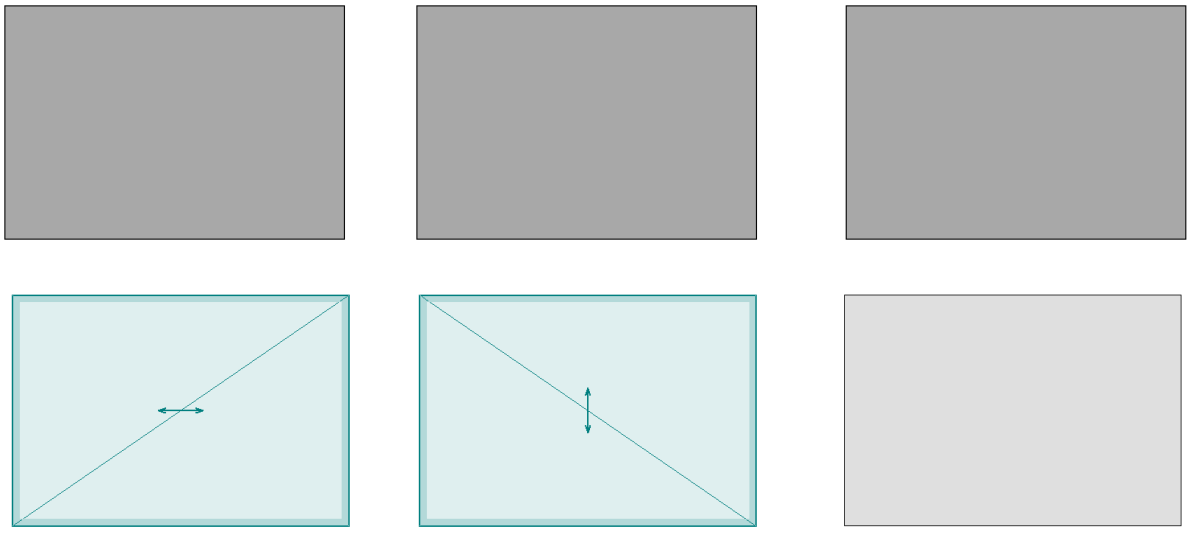


Figure II-18

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

StruSoft AB | Copyright© 2018 StruSoft. All rights reserved.

## 6. StruXML Guid

From now on, it is possible to automatically add a parameter called *StruXML Guid* to elements in the model. Upon clicking on *StruXML Guid* (*StruSoft* tab / *Tools* panel), a *StruXML Guid* parameter is created and applied to following categories:

- Structural Column
- Structural Framing
- Structural Floors
- Structural Walls
- Structural Foundations
- Analytical Column
- Analytical Beam
- Analytical Brace
- Analytical Floor
- Analytical Wall
- Analytical Isolated Foundation
- Analytical Wall Foundation
- Analytical Foundation Slab

Adding the *StruXML Guid* parameter automatically, requires presence of shared parameter file, into which, the new parameter can be added. If there is no shared parameter file loaded in the project, following message will be shown:

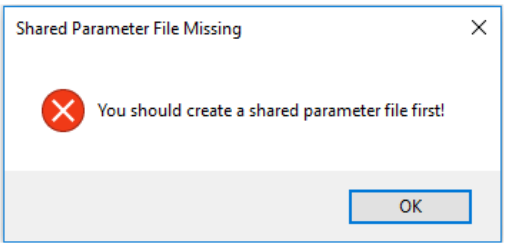



Figure II-19

## Workflow

Browse to an existing shared parameter file or create a new (can be empty) shared parameter file (Manage tab -> Shared parameters - > Create -> OK).

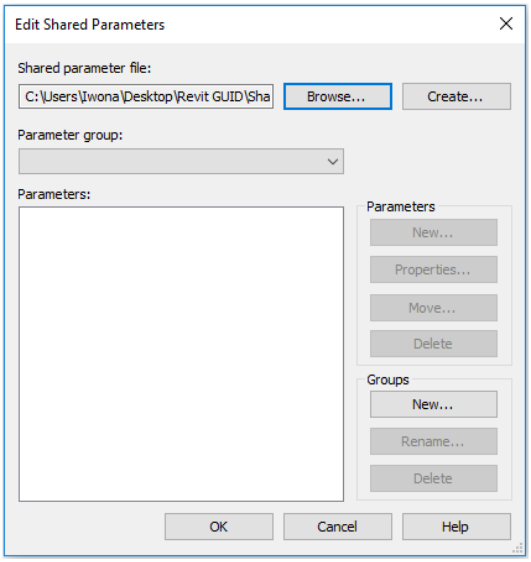



Figure II-20

Click on *StruXML Guid* (*StruSoft* tab / *Tools* panel). *StruXML Guid* parameter is now created and following message appears:

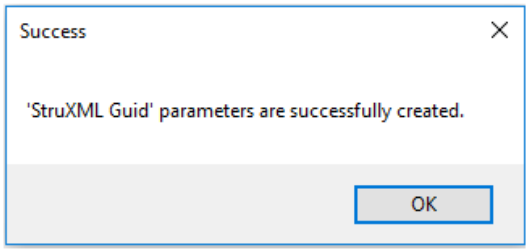



Figure II-21

*StruXML Guid* parameter is now added under *Identity data* for all element categories listed above.

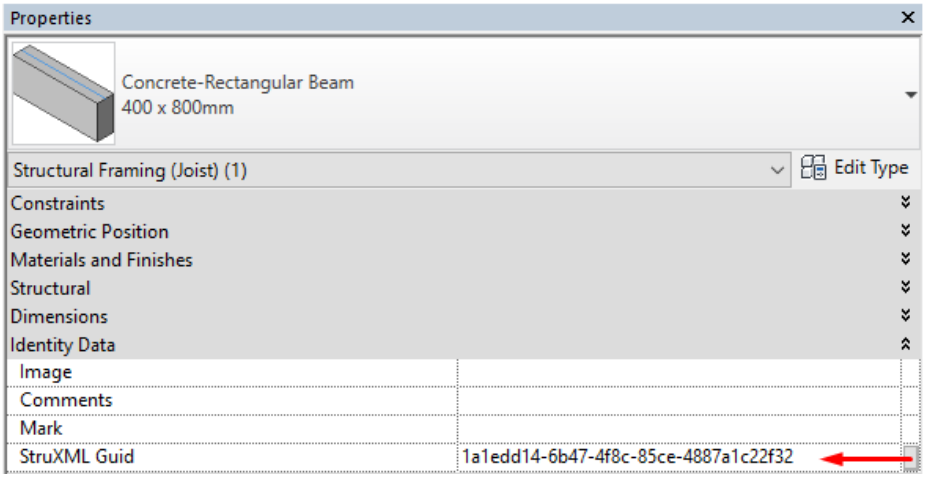



Figure II-22

## Understanding StruXML Guid

Guid stands for 'Globally Unique Identifier'. It is a 128-bit integer number used to identify objects. *StruXML Guid* is a global unique identifier of each object in struxml.

Each object exported from Revit has to have a Guid. Each object imported to Revit from FEM-Design also has a Guid.

When a model is exported from Revit (to struxml), the Guids are created randomly for each element upon exporting the model. Now, with the *StruXML Guid* tool, it is also possible to first add the StruXML Guid parameters and then export the model (in such case, the Guids are created before the model is exported).

When importing a FEM-Design (struxml) model to Revit, all the objects already have Guids (they are assigned to them while the model is saved to struxml). It is, of course, possible to add the StruXML Guid parameter to imported elements in Revit.

It is important to understand that adding StruXML Guid parameters to your Revit elements is not required for successful import or export - it is only an additional feature, for those who are interested in more advanced data exchange (e.g. using Dynamo).

Almost all of the structural element types in struxml actually have two different guids. For example a beam object in struxml has:

- bar guid
- bar_part guid

Using the StruXML Guid tool, one can now apply the guids into Revit elements. The rule is following (shown in Figure II-23):

- StruXML Guid of a physical element corresponds to the struxml main guid (using beam example that would be *bar guid*)
- StruXML Guid of an analytical element corresponds to the struxml part guid (using beam example that would be *bar_part guid*)

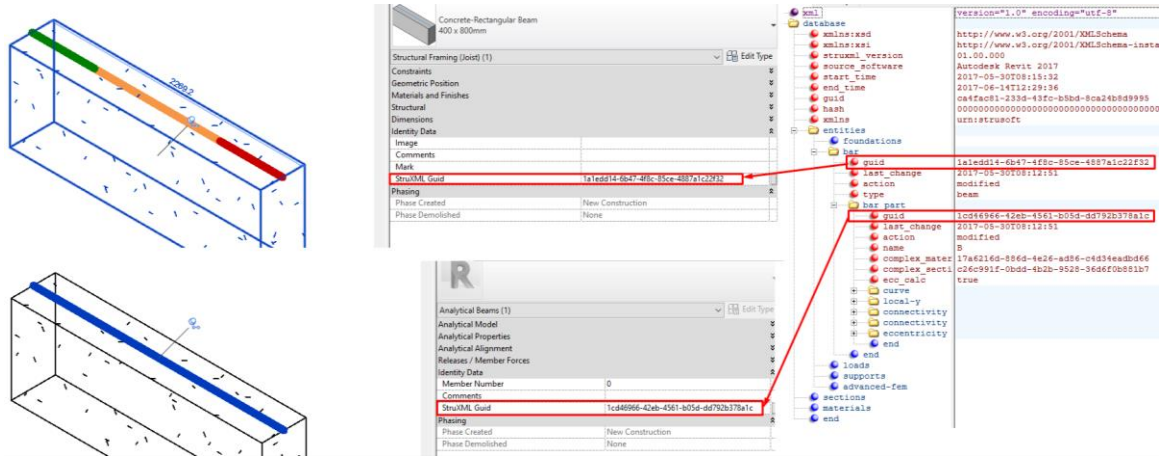


Figure II-23

Both guids can be used to locate the same element in FEM-Design (Tool -> Find | Guid).

Exceptions

There are few exceptions where it is impossible to directly connect a Revit object with an object in struxml (FEM-Design) using the Guid.

a. More than one floor element created within one floor boundary

If more than one floor element is created within one floor boundary (as shown in Figure III-2), it will be divided into number of separate elements when exported to struxml, each with its own Guid.

b. Curved wall

Curved walls drawn in Revit (Figure III-3) will be divided into a number of straight walls while being exported to struXML file, as shown in Figure III-3. This is the way curved walls are represented in FEM-Design. So a curved wall in Revit will have one Guid, but in struxml the walls will be divided into several elements, each with its own Guid.

## 7. Guid Search

One can use Guid Search in order to locate an object in Revit model by their StruXML Guid. It is possible to locate physical and analytical objects. Object can be located in any of the existing views.

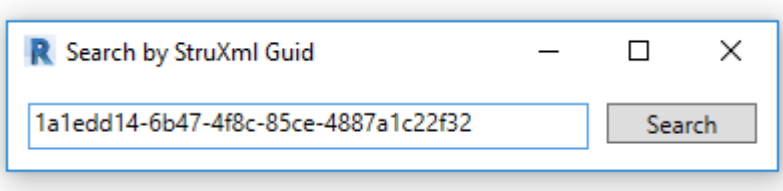



Figure II-24

# III. Workflow: Revit to FEM-Design

The transfer of structural elements from Revit to FEM-Design is done by exporting an analytical model of an instance of an element along with its properties, i.e. material, section/profile/thickness, release conditions, eccentricity.

If a structural element in Revit does not have analytical model enabled, it will not be exported to FEM-Design.

## 8. Data exchange scope

Following elements and properties can be exported from Revit to FEM-Design using the StruSoft StruXML Revit Add-In:

Elements:

- Structural Columns
- Structural Framing elements (Beams, Beam systems, Braces, Trusses)
- Structural Floors* (as floors, profiled panels or fictitious shells)
- Structural Walls
- Structural Foundation
- Grids, levels and reference planes
- Loads, load cases and load combinations

Properties:

- Material of an element
- Section / Profile / Thickness of an element
- Releases of linear elements* (and wall edge connections)
- Boundary conditions
- Eccentricity in case of floor slabs, walls and beams

### 8.1. Structural Columns and Framing

Following properties of *Structural Columns* and *Structural Framing* elements are exported from Revit to FEM-Design along with the geometry of the *Analytical Column* or *Analytical Beam* element:

- Section / profile
- Material*
- Release conditions
- Eccentricity (only of beams)**

* Only the Structural material is exported. Read more in Chapter 8.5.

** Read more about the eccentricity export rules in Chapter 8.7.

Slanted columns in Revit can be exported to struxml and read in FEM-Design, but a message at the export will appear (both in the Export dialog and in the Export Report) in order to warn about slanted elements in the model, as shown in Figure III-1.

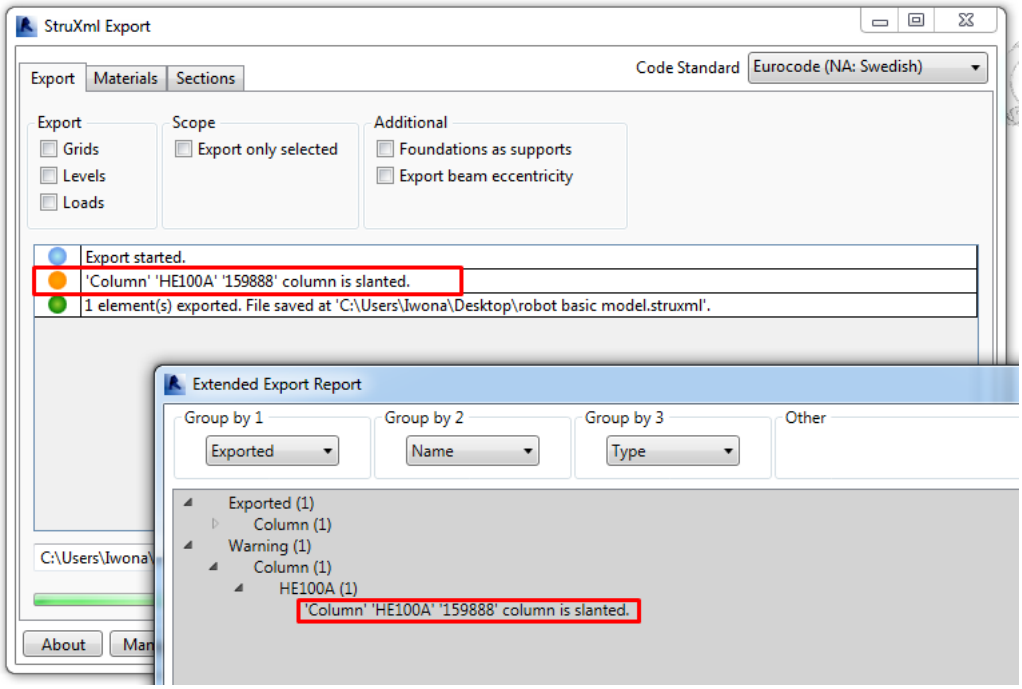



Figure III-1

## 8.2. Structural floors

Following properties of *Structural Floors* are exported from Revit to FEM-Design along with the geometry of the *Analytical Floor* element:

- Thickness of the structural layer
- Material*
- Eccentricity**

If one wants to export a Revit model containing floor elements, he needs to be aware that each structural floor element in Revit has to be a separate instance i.e. only one individual floor should be drawn using the *Create Floor Boundary* command.

If more than one floor element is created within one floor boundary (as shown in Figure III-2), it will be divided into number of separate elements when exported to struxml.

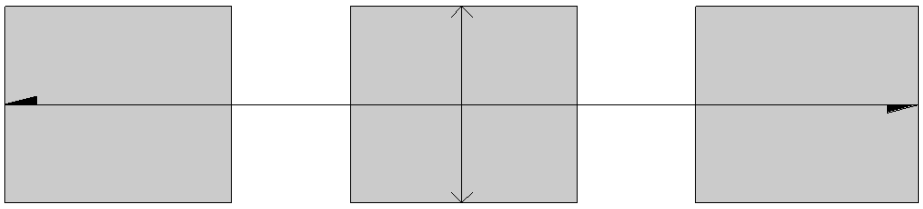


Figure III-2

* Only the Structural material is exported. Read more in Chapter 8.5.

## 8.3. Structural walls

Following properties of *Structural Walls* are exported from Revit to FEM-Design along with the geometry of the *Analytical Wall* element:

- Thickness of the structural layer
- Material*
- Eccentricity**

* Only the Structural material is exported. Read more in Chapter 8.5.
** Read more about the eccentricity export rules in Chapter 8.6.

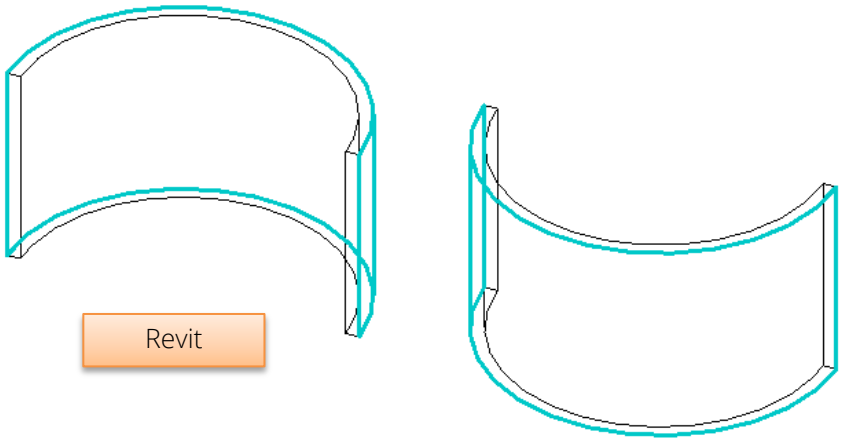



Figure III-3

Curved walls drawn in Revit (Figure III-3) will be divided into a number of straight walls while being exported to struXML file, as shown in Figure III-3. This is the way curved walls are represented in FEM-Design.

Figure III-4 shows the convention of the local coordinate system that curved walls will be assigned with in FEM-Design.

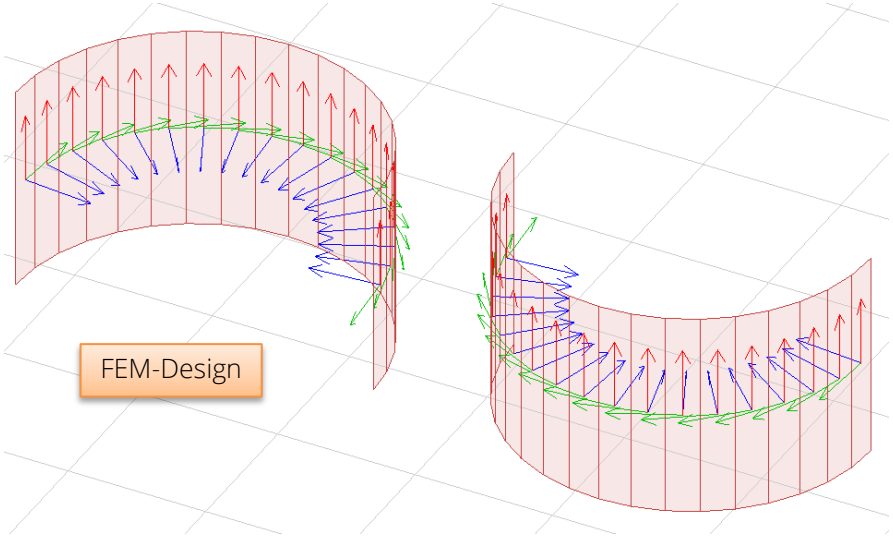



Figure III-4

## 8.4. Structural Foundations

User can decide if to export *Structural Foundations* as supports or as foundation elements. By default, foundations are always exported as foundation elements. The option to export foundations as supports is present in the *StruXML Export* dialog as shown in Figure III-5.

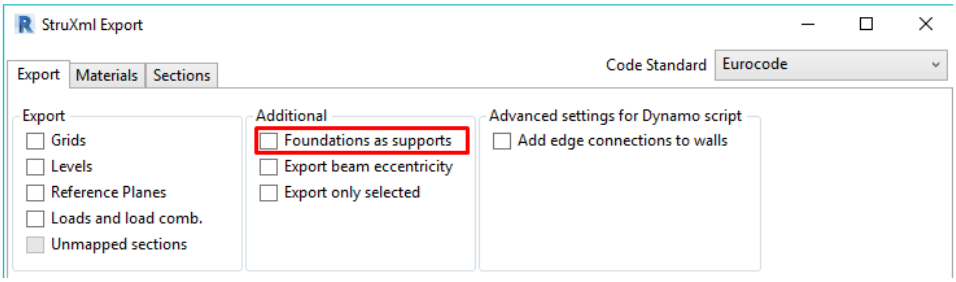



Figure III-5

### 8.4.1. Export as foundation elements

If *Foundation as Supports* option is not selected, *Structural Foundations* will be exported from Revit to FEM-Design as foundation elements according to the following manner:

## Structural Foundation: Isolated

It is exported to FEM-Design as *Isolated foundation* with the following default settings:

- Analytical system: Point support group
- Bedding modulus: 10000 kN/m2/m

An analytical model of isolated foundation is exported to FEM-Design along with the material properties and the exact position of the physical model in respect to the analytical model (In FEM-Design recognized as *Connection point*) as shown in Figure III-6.

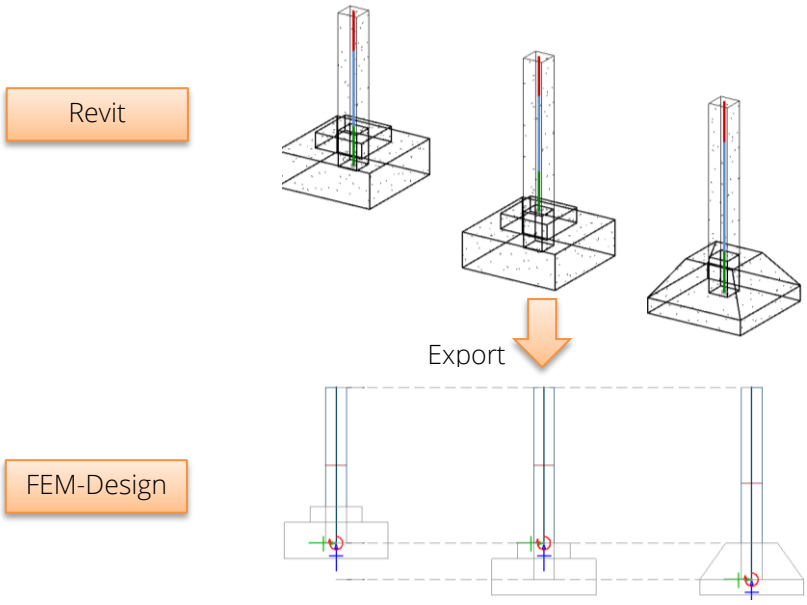



Figure III-6

It is only possible to export the geometry of foot pads with non-curved edges. All the other kinds of isolated foundations, i.e. foot pads with round edges, piles, pile caps, etc. will be exported as a *Point Support group* with the default state *Hinged*. The point support will be added in each analytical foundation (point).

### Structural Foundation: Wall

It is exported to FEM-Design as *Wall foundation* with the following default settings:

- Analytical system: Line support group
- Bedding modulus: 10000 kN/m2/m

An analytical model of wall foundation is exported to FEM-Design along with the material properties and the exact position of the physical model in respect to the analytical model (in FEM-Design recognized as *Insertion point*) as shown in figure below.

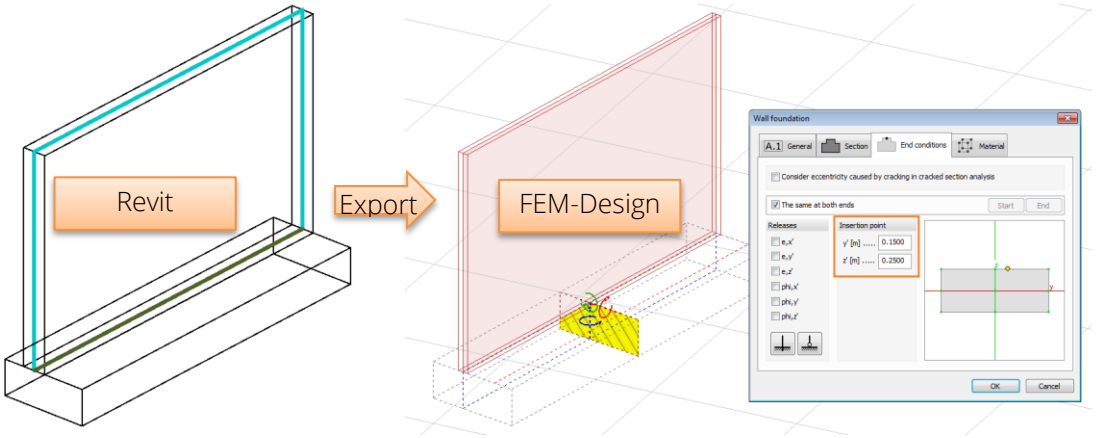



Figure III-7

### Structural Foundation: Slab

It is exported to FEM-Design as *Foundation slab* with the following default settings:

- Bedding modulus: 10000 kN/m2/m

An analytical model of slab foundation is exported to FEM-Design along with the material properties and the analytical alignment definition. Read about the analytical alignment rules in Paragraph 8.6.

## 8.4.2. Export as supports

If *Foundation as Supports* option is selected, *Structural Foundations* are exported from Revit to FEM-Design as *Supports* according to the following manner:

- *Structural Foundation: Isolated* in FEM-Design will be converted into *Point Support Group* with the default type: *Hinged*
- *Structural Foundation: Wall* in FEM-Design will be converted into *Line Support Group* with the default type: *Hinged*
- Structural Foundation: Slab in FEM-Design will be converted into a Surface support group.

## 8.5. Material

Each analytical element that is meant to be exported from Revit to FEM-Design needs to have a structural material assigned to its physical model.

Element, which structural material is set to be *<By Category>* will not be exported to FEM-Design. An error about lack of structural material assigned to an element will appear after attempt to export such element. <-- Read about recognized issue in Chapter 10.

In case of multilayer walls or multilayer floor slabs, analytical model is exported along with the material (and thickness) property of a layer that is marked as a *Structural Material*. In the example shown in Figure III-8, the wall element would be exported from Revit to FEM-Design as a 215 cm thick element with the material: *Masonry – Concrete Block.*

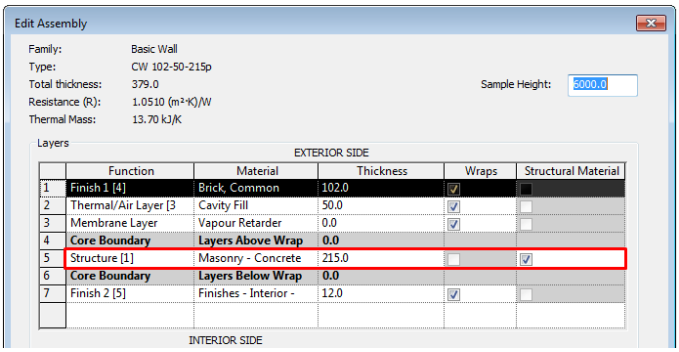



Figure III-8

## 8.6. Eccentricity in shell elements

In Revit user can decide about the position of *Analytical Wall* and *Analytical Floor / Analytical Foundation Slab* in relation to the physical model. Those settings are exported to FEM-Design as *Eccentricity* and the Alignment is set to one of the three values: Top/Left, Center or Bottom/Right. Example of an alignment setting in case of floor element is shown in Figure III-9.

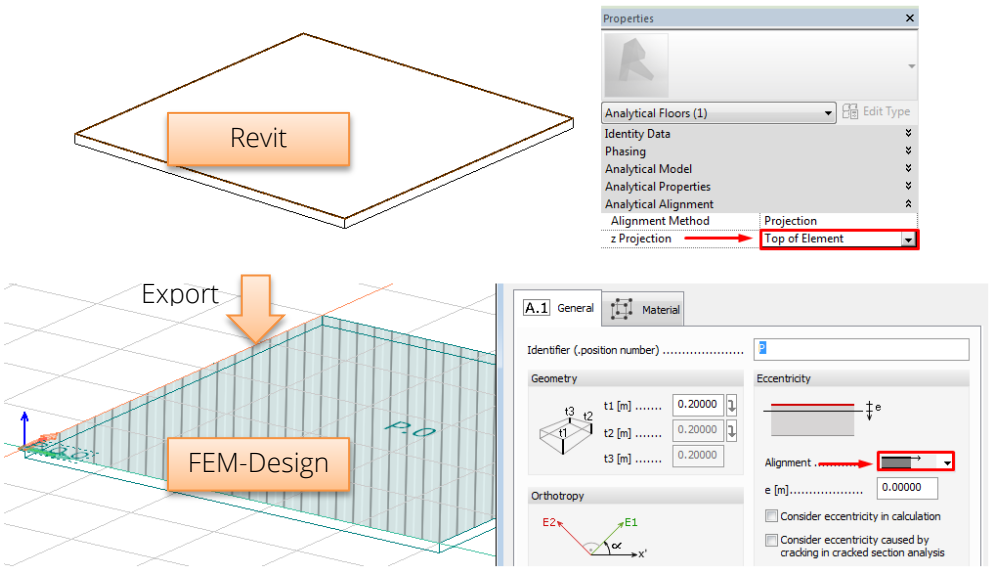



Figure III-9

### 8.6.1. Analytical Floor / Analytical Foundation Slab

Following Analytical Alignment definition of an Analytical Floor / Analytical Foundation Slab is exported to FEM-Design:

Alignment Method: Projection

z Projection: Top of Slab
Center of Slab
Bottom of Slab

In any other case, i.e. *Projection* is set to a *Level* or a *Reference Plane,* or the *Alignment Method* is set to be *Auto detect*, the default position of *Analytical Floor* / *Analytical Foundation Slab* in relation to physical element after export to FEM-Design will be: *Center*.

Example of different settings in Revit and the final result after opening the exported model in FEM-Design is shown in Table III-1 and in Figure III-10.

| | 1 | 2 | 3 | 4 | 5 |
|---|---|---|---|---|---|
| Revit Alignment: | Projection | Projection | Projection | Projection | Auto detect |
| Method: | Top of Slab | Center of Slab | Bottom of Slab | Level 1 | - |
| FEM-Design alignment | Top/Left | Center | Bottom/Right | Center | Center |

Table III-1

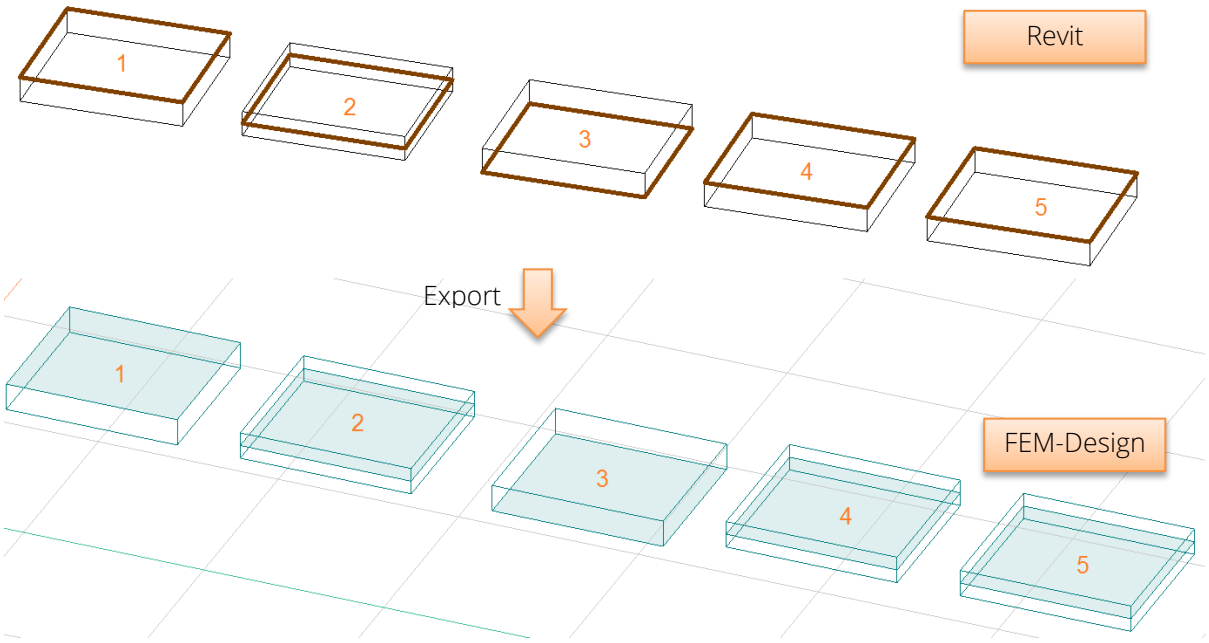



Figure III-10

### 8.6.2. Analytical Wall

Following *Analytical Alignment* settings of an *Analytical Wall* are exported to FEM-Design:

Alignment Method: Projection

Projection: Center of Element
Interior Face
Center of Core
Exterior Face

In any other case, i.e. *Projection* is set to a *Grid* or *a Reference Plane* or the *Alignment Method* is set to be *Auto detect*, the default position of analytical model in relation to physical element after export to FEM-Design will be: *Center*.

Example of different settings in Revit and the final result after opening the exported model in FEM-Design is shown in Table III-2 and in Figure III-11.

| | 1 | 2 | 3 | 4 | 5 |
|---|---|---|---|---|---|
| Revit Alignment: | Projection | Projection | Projection | Projection | Auto detect |
| Method: | Center of Element | Interior Face | Center of Core | Exterior Face | - |
| FEM-Design alignment | Center | Right | Center | Left | Center |

Table III-2

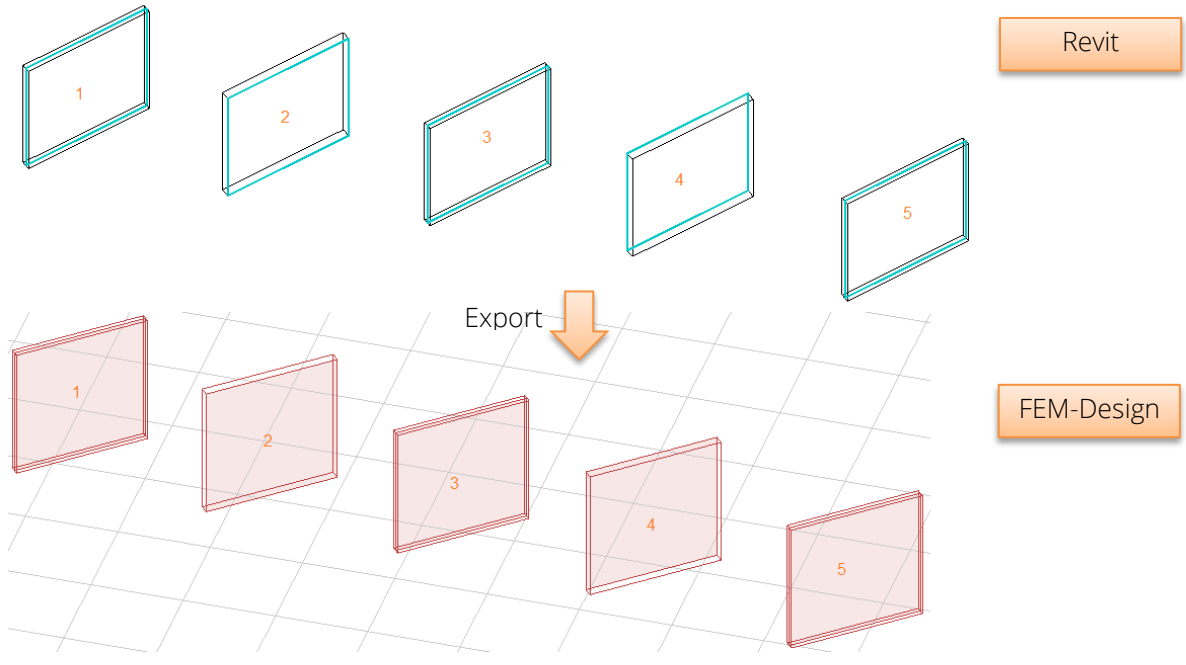



Figure III-11

## 8.7. Eccentricity in beams

Almost any configuration of analytical beam model in regards to its physical representation can be transferred to struxml and read in FEM-Design as *Eccentricity*.

In order to export physical eccentricity in beams, one has to check the *Export beam eccentricity* box in the *Export StruXML* dialog, as shown in Figure III-12. <-- Read about recognized issues in Chapter 10.

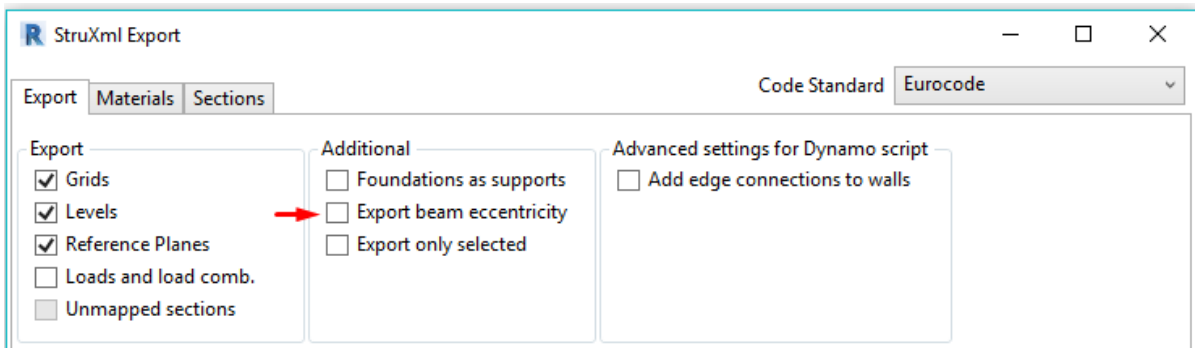



Figure III-12

Example of beam eccentricity export is shown in Figure III-13

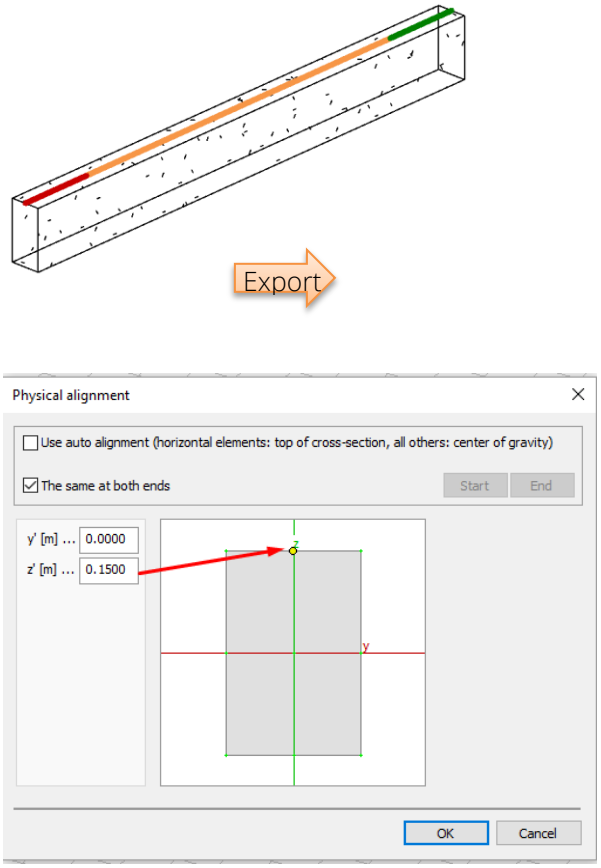



Figure III-13

The eccentricity configuration that cannot be exported to struxml is when the start and end part of the analytical beam are not the same.  Example of such configuration is shown in Figure III-14.

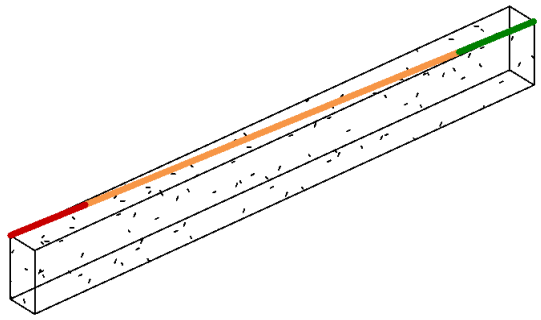


Figure III-14

In this case, a warning will be thrown at the export (Figure III-15).

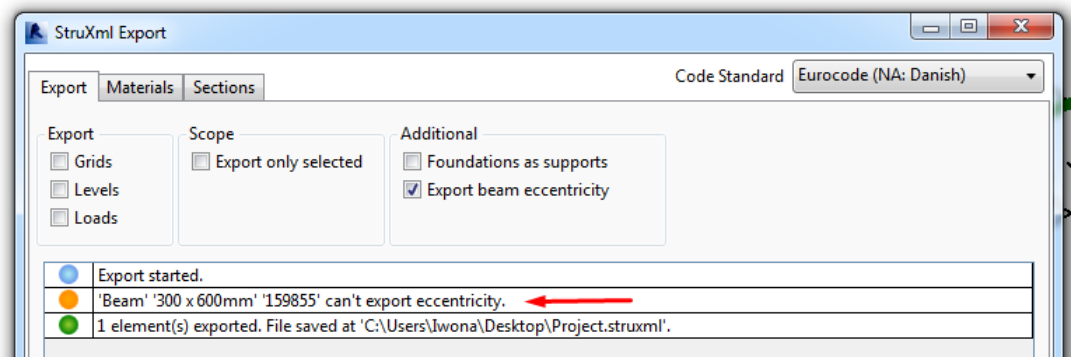



Figure III-15

The analytical position of the beam will be exported to struxml (as usually), but eccentricity will not. It means that the cross section will be simply assigned along the length of the analytical beam, with y'=0 and x'=0 eccentricity values (Figure III-16).

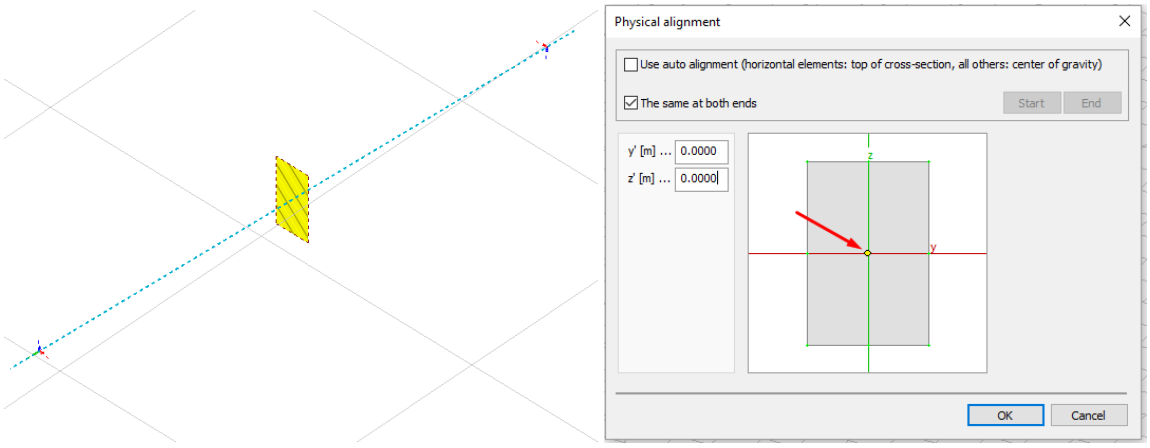



Figure III-16

## 8.8. Releases of linear elements

All linear analytical elements are exported from Revit to FEM-Design along with the settings of *Top Release* and *Bottom Release*. The exact definition of the three predefined releases: *Pinned*, *Fixed*, *Bending Moment* as well as *User Defined* release are exported to FEM-Design. Figure III-17 shows release conditions originally set in Revit and the result of the export to FEM-Design.

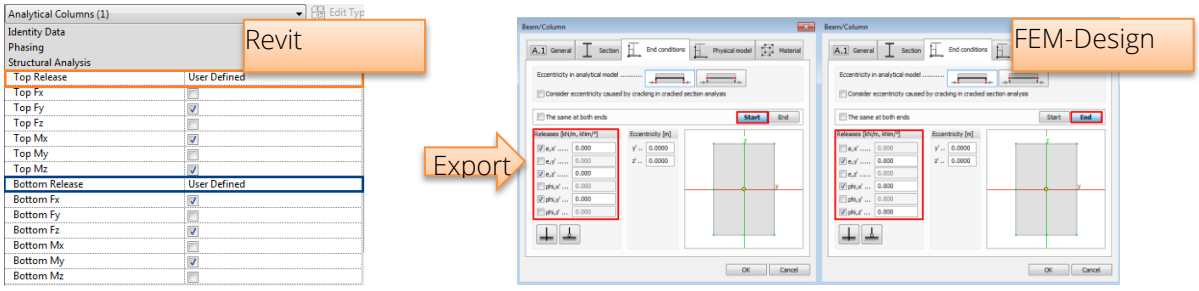



Figure III-17

## 8.9. Boundary conditions

In Revit, *Boundary Conditions* command allows to choose between three kinds of boundary conditions: *Point*, *Line*, and *Area*. User has to assign it with one of predefined or user defined *State* in X, Y and Z direction: *Fixed*, *Pinned*, *Roller*, and *User*. The exact definition of boundary conditions can be exported from Revit to FEM-Design as follows:

Boundary Condition Type:

- *Point*: exported to FEM-Design as *Point Support Group*
- *Line*: exported to FEM-Design as *Line Support Group*
- *Area*: exported to FEM-Design as *Surface Support Group*

State:

- *Fixed*: exported to FEM-Design as type *Hinged*
- *Pinned*: exported to FEM-Design as type *Rigid*
- *Roller*: exported to FEM-Design according to Revit definition
- *User*: exported to FEM-Design according to Revit definition

Example: *Point* boundary condition type with the *Pinned* state is exported to FEM-Design as *Point Support Group*, type: *Hinged* as shown in Figure III-18.

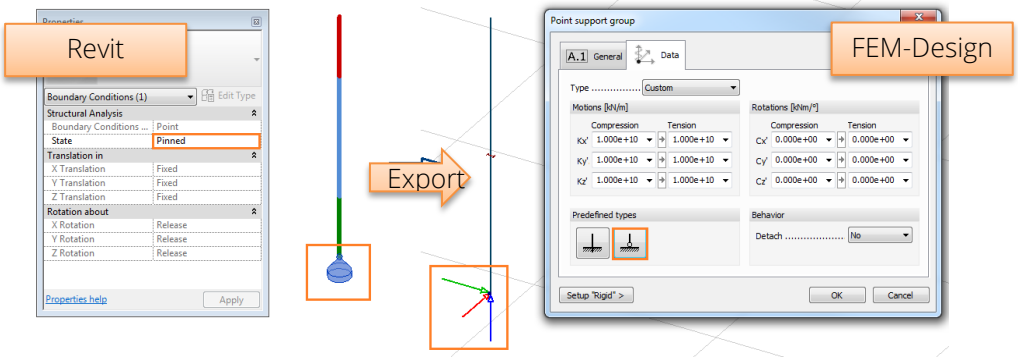



Figure III-18

The "Rigid" value exported from Revit to FEM-Design is equal to: 1e+15 kN/m for motion, and 3.046e+11kNm/° for rotation.

Line boundary conditions under curved walls are special case. Each curved wall exported from Revit, in FEM-Design will be divided into a number of straight walls. But line boundary condition will remain curved, as shown in the left part of the figure below. It is therefore, user's responsibility to verify the support condition under curved walls in order to obtain a desired solution as shown in the right part in figure below.

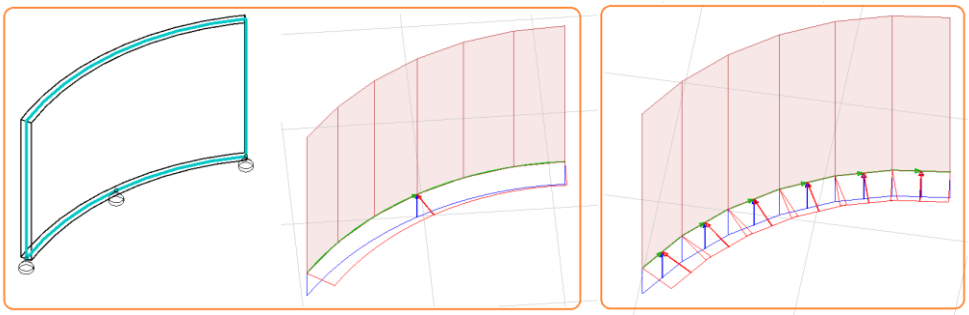


Figure III-19

## 8.10. Grids

*Grids* defined in Revit model may be exported to FEM-Design where they will be recognized as *Axes. Grids* are exported with following assumptions:

- If the option to *Export Grids* (Figure III-23) is selected, all the grids will be exported to FEM-Design.
- The exact length of the each grid is exported.
- All the grids are always exported into 00.000 m level in FEM-Design.
- Names of the grids are not exported. Each axis in FEM-Design will receive a new number based on the order of grid creation in Revit, as shown in Figure III-20. If necessary, renaming can be performed in FEM-Design.

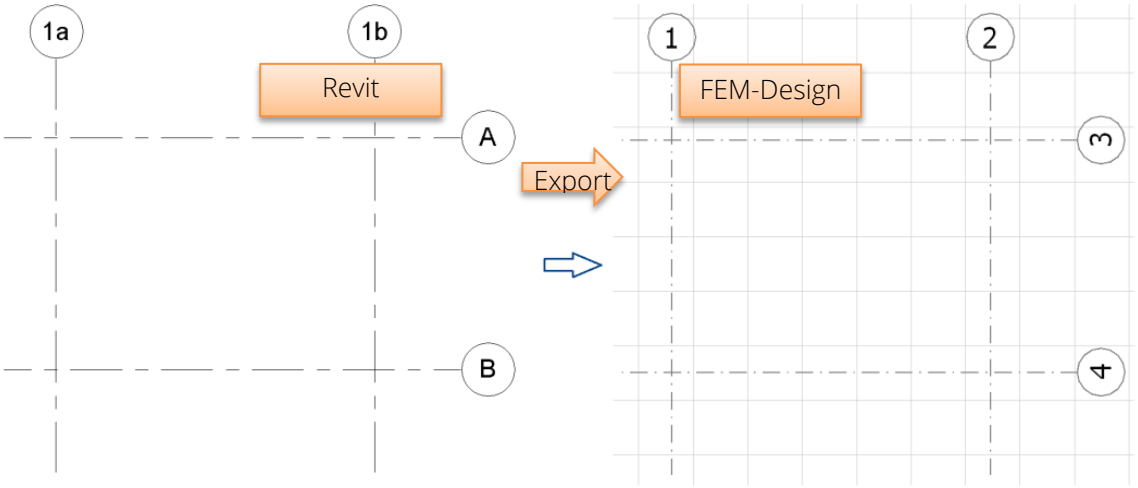



Figure III-20

In FEM-Design only straight line axes are allowed. Therefore,

- *Multi-Segment Grid* after export from Revit to FEM-Design will be replaced with a number of straight line axes equivalent to number of segment.
- curved grid after export from Revit to FEM-Design will be replaced with a straight line axis between the start and end point of the original curved grid, as shown in Figure III-21.

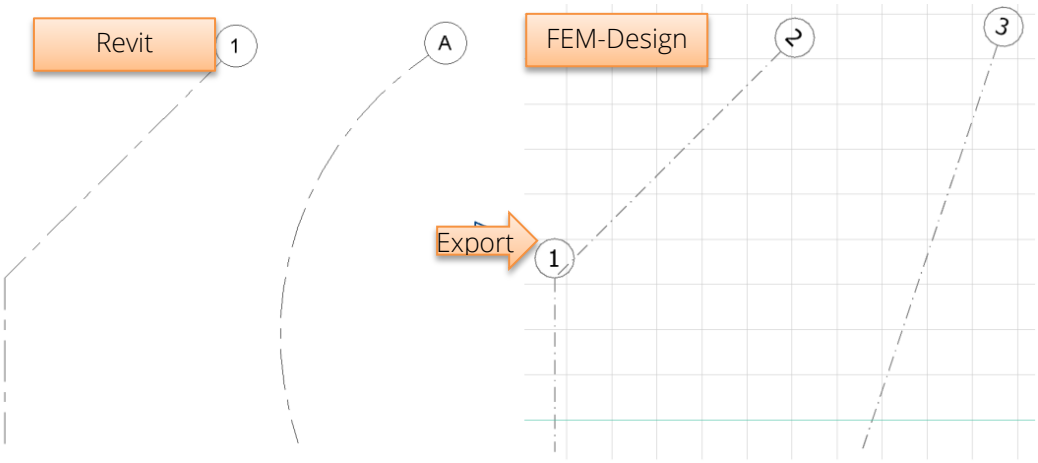



Figure III-21

## 8.11. Levels

*Levels* defined in Revit model can be exported to FEM-Design, where they will be recognized as *Storeys*. *Levels* are exported with following assumptions:

- If the option to *Export Levels* is selected (Figure III-23), all the levels will be exported to FEM-Design.
- Name of the levels are exported to FEM-Design.
- Size of the *Storey* in FEM-Design is calculated based on the placement of most outer elements in the Revit model and included as a *Horizontal size of building* in the *Storey* dialog, as shown in Figure III-22.
- *Levels* cannot be exported if no elements are created in the Revit model because size of a storey cannot be generated.

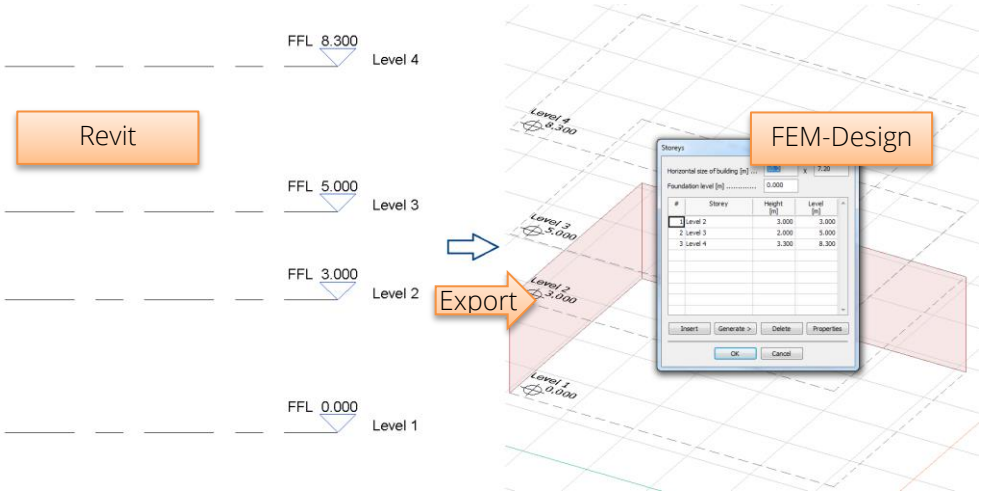



Figure III-22

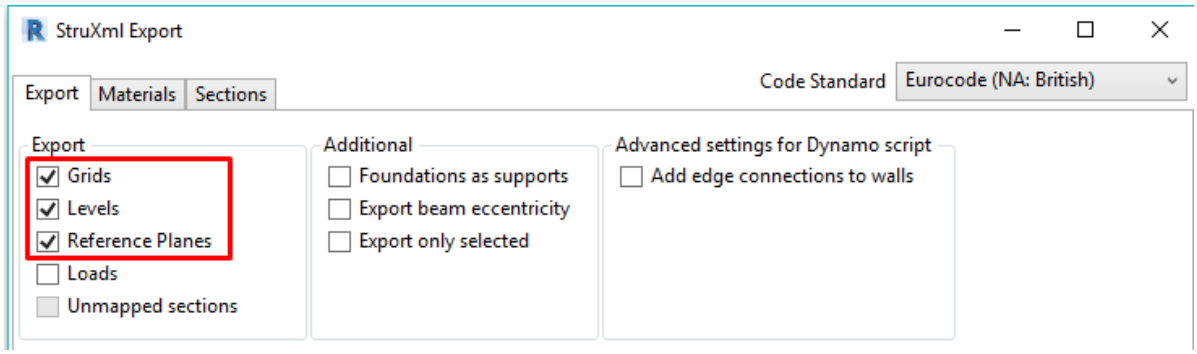



Figure III-23

## 8.12. Reference planes

*Reference planes* defined in Revit model may be exported to FEM-Design where they will be recognized as *Reference planes.*

- If the option to *Export Reference planes* is selected (Figure III-23), all the reference planes will be exported to FEM-Design.
- Name of the reference planes are exported to FEM-Design.

## 8.13. Loads, load cases and load combinations

In order to export loads and load cases one should select the *Loads and load comb.* to be exported in the *StruXml Export* dialog, as shown in Figure III-23.

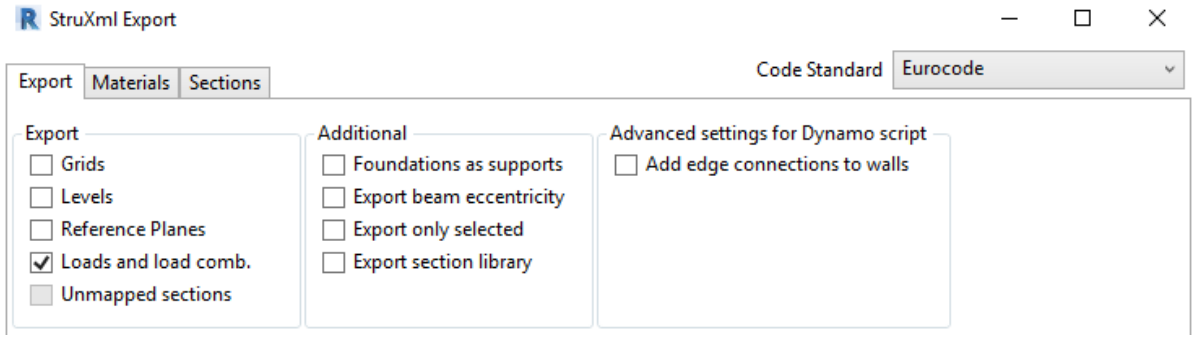



Figure III-24

### 8.13.1. Load cases

If the *Export Loads* option is selected, all *Load Cases* existing in Revit will be exported to FEM-Design (including those loads cases that do not contain any loads assigned).

The only property of the load case that is exported from Revit to FEM-Design is a *Name*. Definition of *Nature* and *Category* is not relevant for the export to FEM-Design, except in one case. 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.*

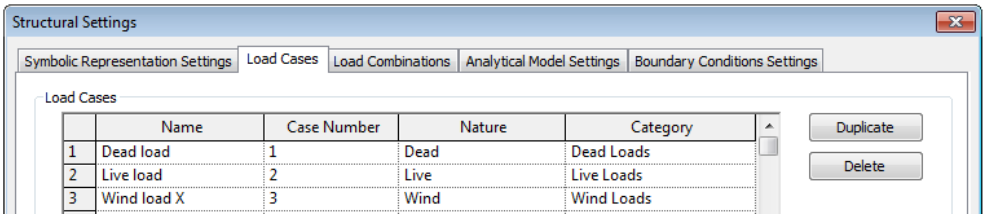



Figure III-25

Example of loads cases defined in Revit (Figure III-25) and exported to FEM-Design is shown in Figure III-26.

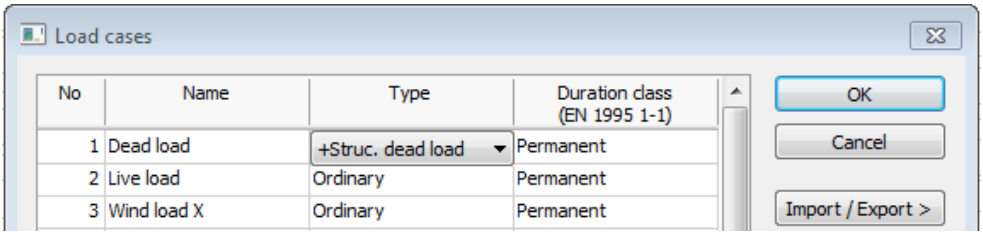



Figure III-26

### 8.13.2. Loads

Following *Load* types are exported from Revit to FEM-Design:

- Point Load
- Line Load
- Area load
- Hosted Point Load
- Hosted Line Load
- Hosted Area load

The example on how the loads are exported from Revit to FEM-Design is shown in following Figures.

Figure III-27 shows a vertical force of 1kN is applied to a column in Revit. The value of the force is *-1kN* so the force acts downward. The force is applied into a load case: *Dead load*.

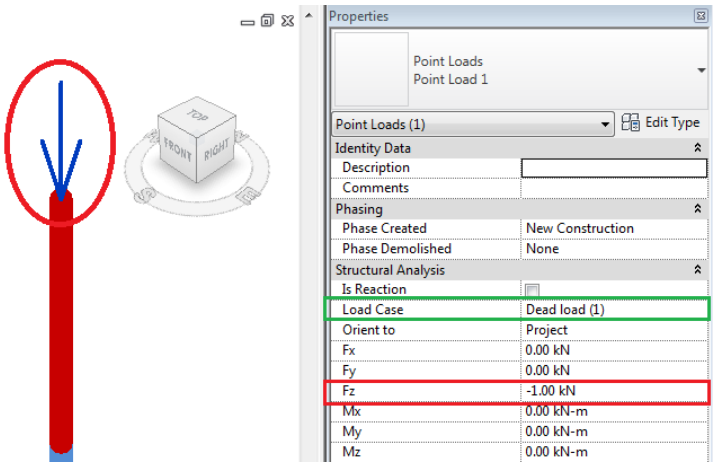



Figure III-27

Figure III-28 shows how the model looks like after exporting it to FEM-Design.

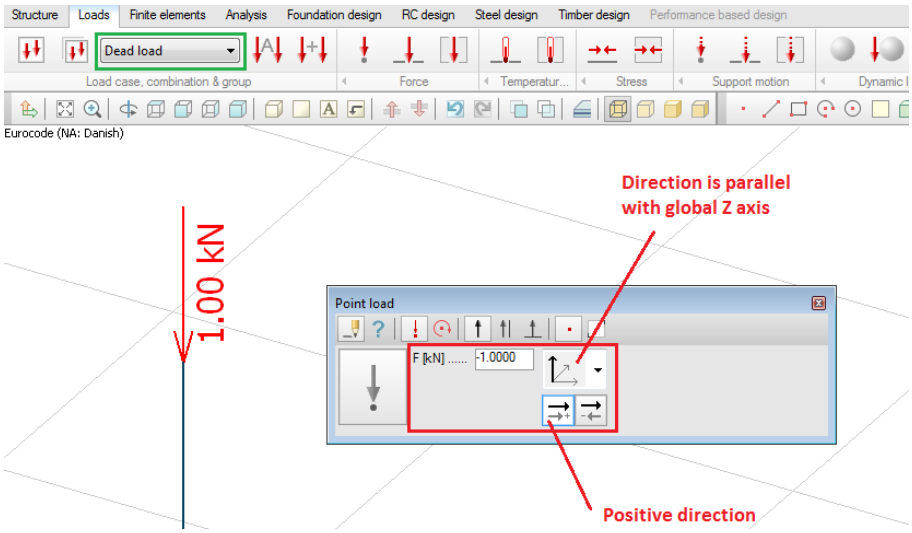



Figure III-28

The load value is *–1 kN*, as defined in Revit. The correct direction of the load is preserved by setting a positive direction (with the global Z axis) that is consistent with the positive direction of Revit coordinate system. The load is assigned to a corresponding load case *Dead load*.

If more force components is assigned into one point load or one surface load, the visible force in Revit will be displayed as resultant force, as shown in Figure III-29.

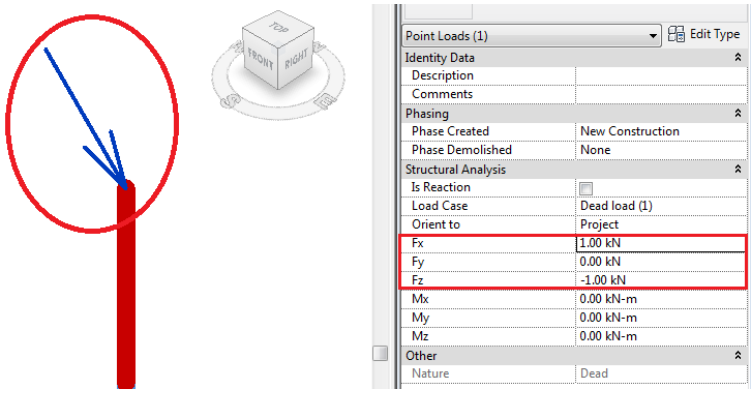



Figure III-29

After exporting it to FEM-Design, the point (or surface load) will be divided into separate components. So, the column from figure above will be loaded with a force of 1kN acting in the X – direction, as shown in Figure III-30.

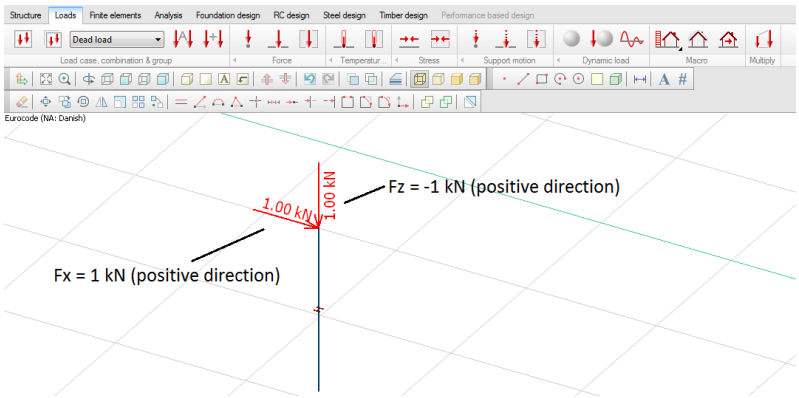



Figure III-30

### 8.13.3. 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
    - Ultimate corresponds to Ultimate Limit State in FEM-Design,
    - Serviceability corresponds to Serviceability - characteristic limit state in FEM-Design.

Type and Usage are ignored.

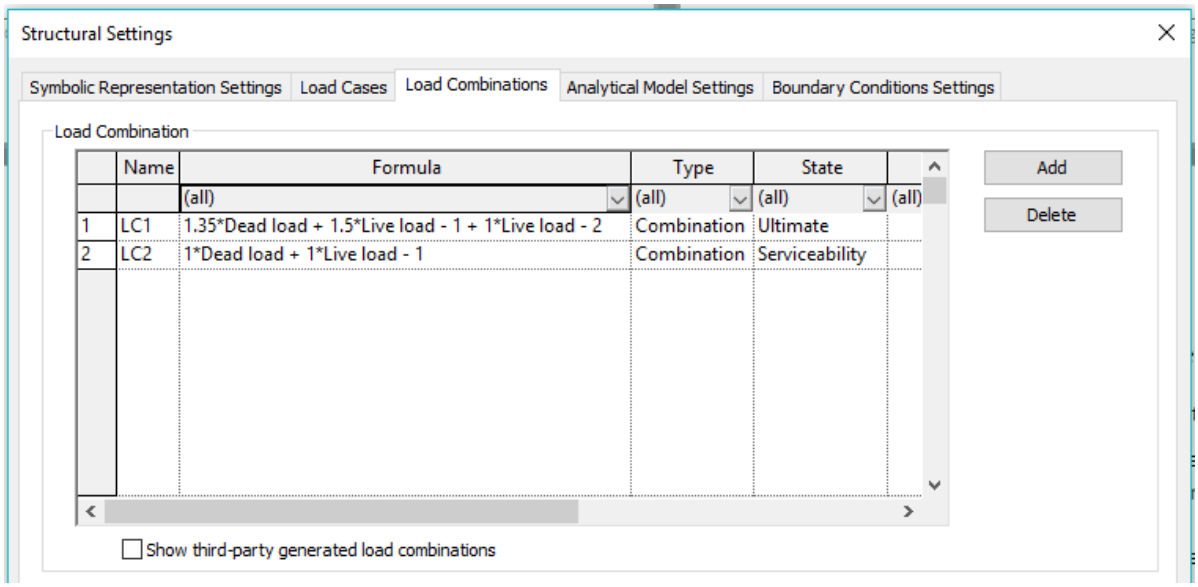



Figure III-31

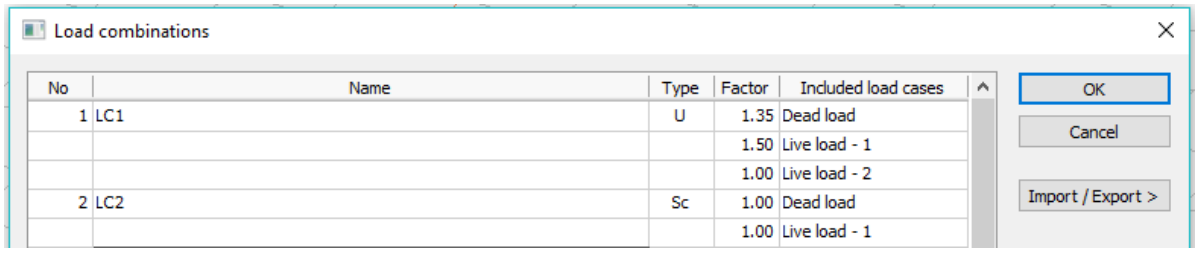



Figure III-32

## 9. Export from Revit to FEM-Design

Before exporting a model from Revit to FEM-Design, make sure that:

- all the elements you wish to export have analytical model enabled,
- the analytical model of the structure is consistent and is arranged the way you want it to be exported to FEM-Design,
- all the elements you wish to export have a structural material assigned.

If the analytical model is prepared, proceed with the *Export StruXML* command that is placed in the *StruXML panel* in the *StruSoft* tab in Revit.

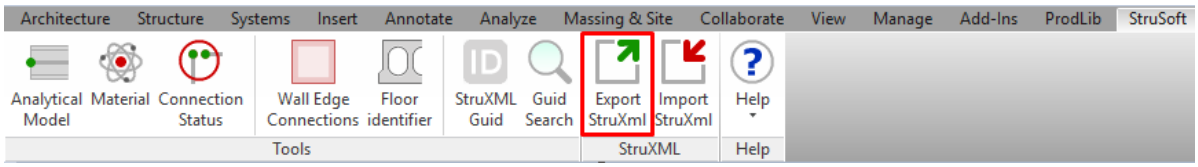



Figure III-33

Upon choosing the *Export StruXML* command, a *Code* dialog shown in Figure III-34 will appear. Before exporting model to FEM-Design, it is necessary to select one of the design codes. Chosen code and national annex will influence the material and sections library that will be used for mapping. After choosing the desired code standard, check *Set as default* box, if you want to save this choice.

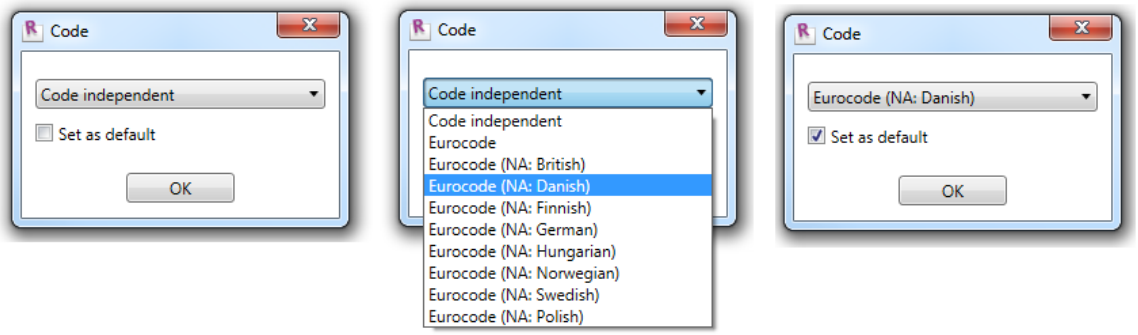



Figure III-34

Upon pressing OK in the *Code* dialog, a *StruXML Export* dialog will appear, as shown in Figure III-35.

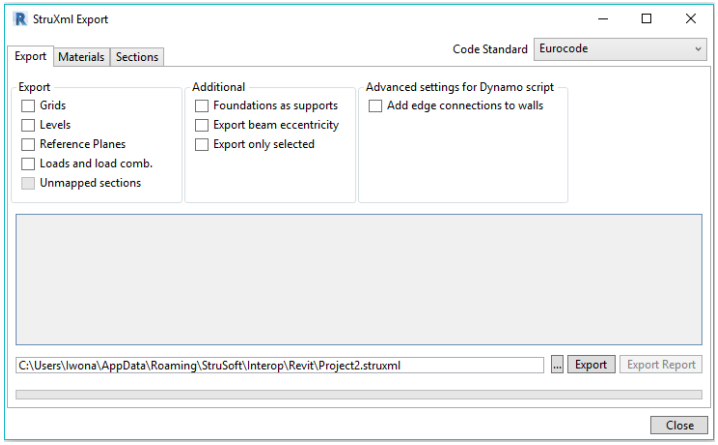



Figure III-35

## 9.1. Export tab

In the *Export* tab of a *StruXML Export* dialog user can:

- change the previously chosen *Code Standard,*
- decide if to export Grids, Levels, Reference planes and Loads,
- decide if to export *Loads and load combinations*,
- decide if to export *Unmapped sections,*
- decide if foundation elements should be exported as supports (check *Foundation as Supports*),
- decide if beams eccentricity should be exported (check *Export beams eccentricity*)
- decide if to export only selected elements (check *Export only selected*)
- add default edge connections to walls (only for post processing)
- see the list of warnings (orange bullets) and errors (red bullets), as well as export results listed in the status window,
- export the model to struXML file,
- see the *Export Report* (that becomes active when the export is done).

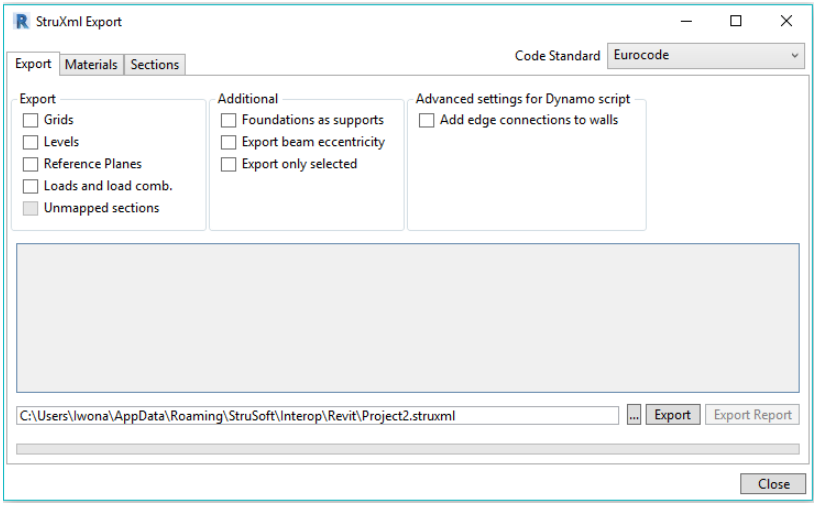



Figure III-36 (newest version of the dialog)

After launching StruXML Export dialog there are no warnings about unmapped elements, as shown in Figure III-36.

It means that all the materials and profiles used in your current project have already been mapped and saved before, and no mapping is needed. However, it is recommended to go to *Materials* and *Sections* tabs in order to check and approve the previously saved mapping.

In this situation, the procedure is as follow:

1. In the *Export* tab, press an icon with three dots to define location and name of the struxml file.
2. Press the *Export* button to export the model into struxml file.

When the export is finished, an export status will be displayed in the status window as shown in Figure III-37 and the Export Report will become active.

Go to Paragraph 0 in order to read about meaning of possible warnings and errors. Go to Paragraph 0 in order to read more about the *Export report*.

3. Press *Close* button to close the dialog.

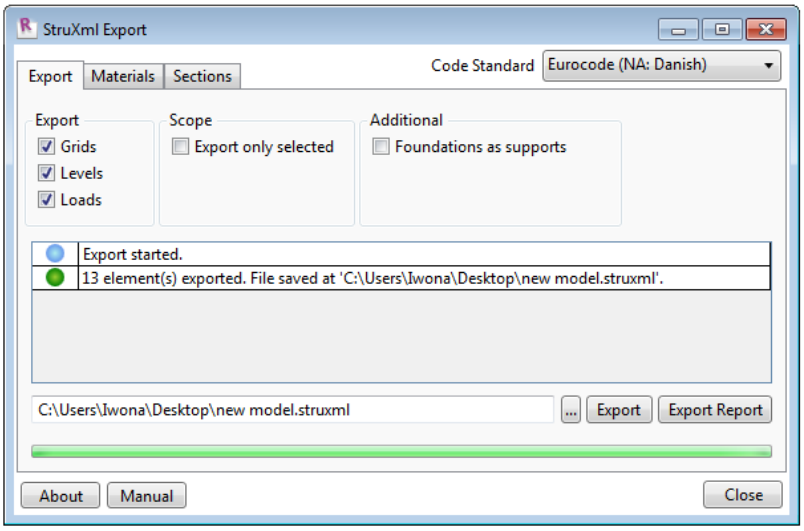



Figure III-37 (older version)

After launching StruXML Export dialog there is a list of warnings about unmapped elements, as shown in Figure III-38.

It means that all or some of the materials and/or sections included in the Revit model have to be mapped to the corresponding materials/sections in FEM-Design.

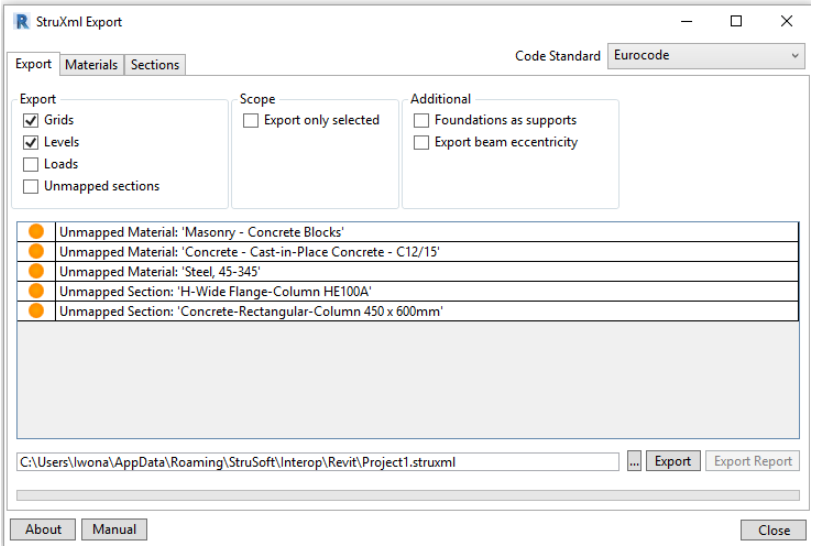



Figure III-38

In this situation, the procedure is as follow:

1. Go to *Materials* tab and map the listed materials (read more about materials mapping in Paragraph 9.2).

2. Go to *Sections* tab and map the listed sections and profiles (read more about sections mapping in Paragraph 9.3) or chose to export unmapped sections.
3. Go back to *Export* tab, and if there is no warning left in the status window, follow the procedure from workflow 1 in order to export the model.

## 9.2. Materials mapping

In order to map the materials included in the Revit model one should move to *Materials* tab, shown Figure III-39. The dialog is divided into two parts: *Revit* materials in the left part and *FEM-Design* materials in the right part.

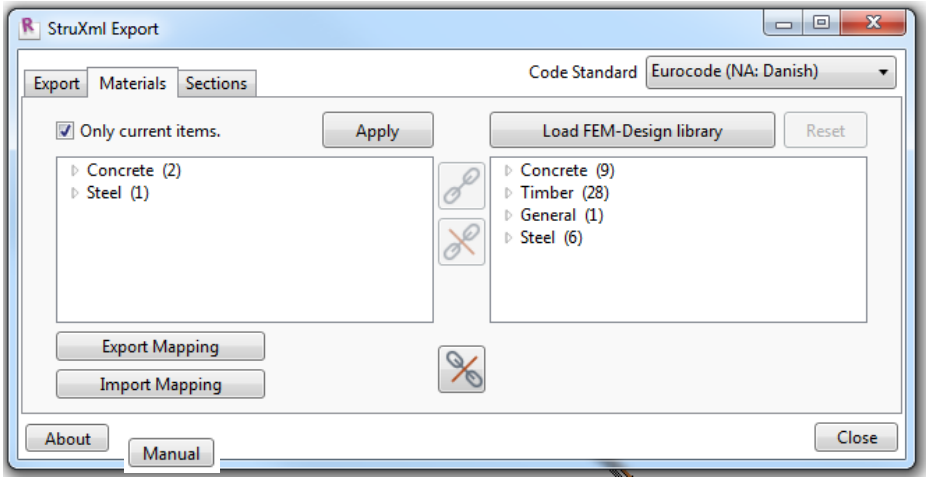



Figure III-39

Only current items

You can decide if you want to see (and map) all the materials from the Revit model library or only those that are currently being used in the model. *Only current items* option is always checked by default. Uncheck it in order to see all the materials from in the Revit library.

In the left window materials, based on their Revit category: *Concrete*, *Metal*, *Wood,* are sorted into following groups*: Concrete*, *Steel*, *and Timber.* Material of any other category than *Concrete*, *Metal*, *and Wood* will be sorted under group *Unassigned*.

Mapping procedure

1. Click on the small triangular next to the material category to open a drop down list of materials to be mapped within this category and select one of the materials. Any material that has to be mapped will be displayed in red. Any of material that have already been mapped before will be displayed in black.
2. Find a corresponding material in the FEM-Design library in the right part of the dialog. Click on the small triangular next to the material category to open a drop down list of available materials from the FEM-Design library and select desired material.
3. Press the *Map selected item* button. Material that has just been mapped turned black.
4. In order to save the material mapping (and update the warning list in the status window in the *Export* tab) press *Apply*.

Unmapping procedure

- If you wish to unmap any of the previously mapped materials, select the material in the Revit materials window and press the *Unmap Selected Item* button.
- If you wish to unmap all of the previously mapped materials, do not select any material – just press the *Unmap All Mapped Items* button.

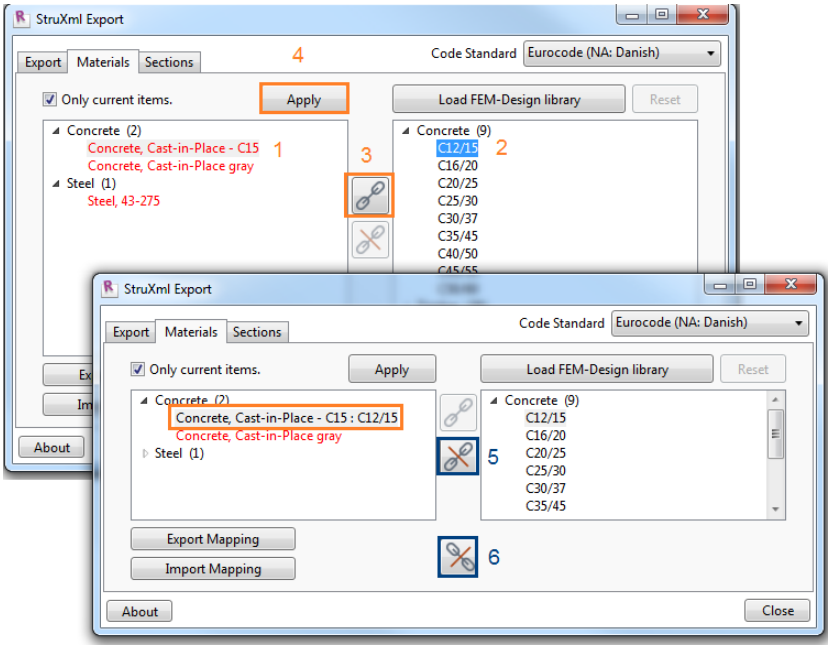



Figure III-40

Load FEM-Design library

If you added additional materials into FEM-Design materials library and you wish to perform mapping into those materials, you need to update the default FEM-Design library in the *StruXML Export* dialog to the desired one. The code of the new materials library imported from FEM-Design has to be consistent with the chosen *Code Standard* in the *StruXML Export* dialog.

The procedure is as follows (shown in Figure III-41 and Figure III-42):

1. In FEM-Design go to the *Default settings* of e.g. *Colum.*
2. Go to *Material* tab.
3. Press the *Export…* button.
4. Chose *Material library (*.struxml)* in Save as type, find a location and name the library.
5. Chose *Save* button.
6. Go back to the *Materials* tab in the *StruXML Export* dialog and press *Load FEM-Design library.* Find and select the desired struXML file.
7. Press *Open* button.

Remember to always load the current FEM-Design library. This will prevent from mapping into a material that may not exist in the current library used in FEM-Design.

Press *Reset* button in order to reset the materials library to the default one.

Export / Import Mapping

Once performed materials mapping can be saved to struXML file using *Export Mapping* button. Use *Import Mapping* button in order to load mapping from struXML file.

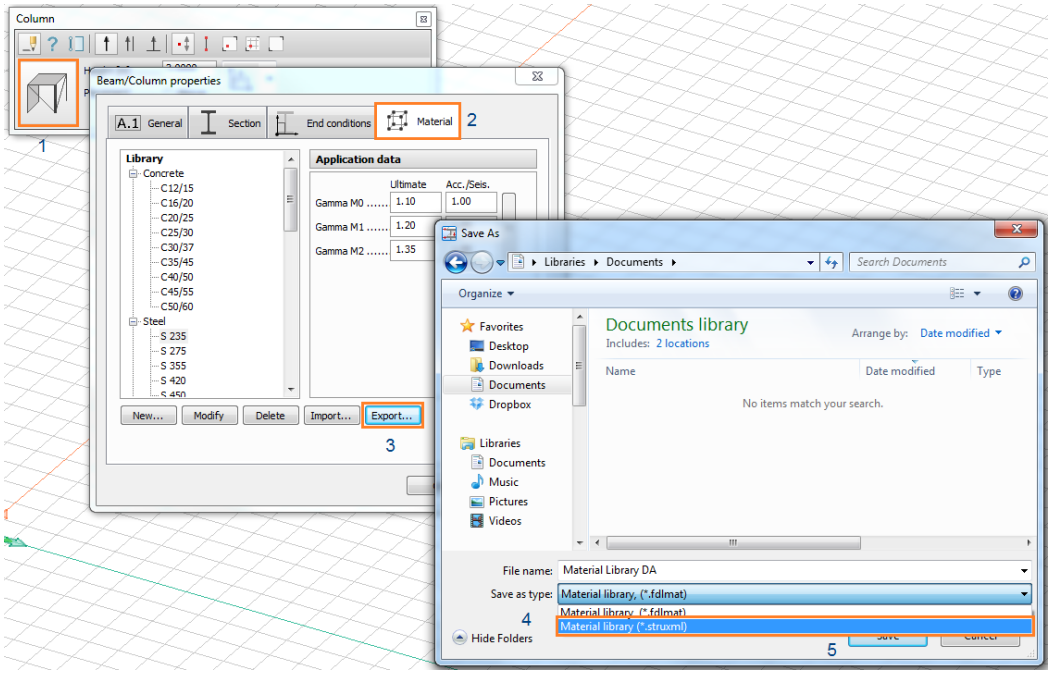



Figure III-41

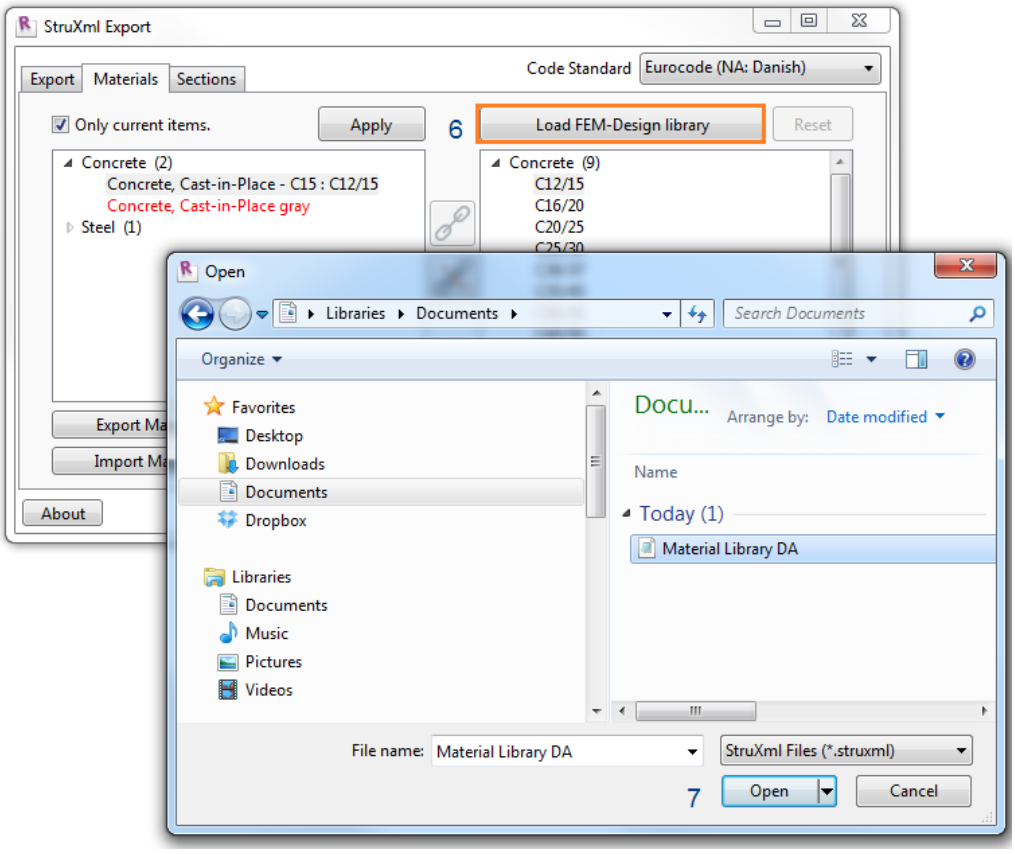



Figure III-42

## 9.3. Sections mapping

In order to map the sections one should move to *Sections* tab shown in Figure III-43. The dialog is divided into two parts: *Revit* sections in the left part and *FEM-Design* sections in the right part.

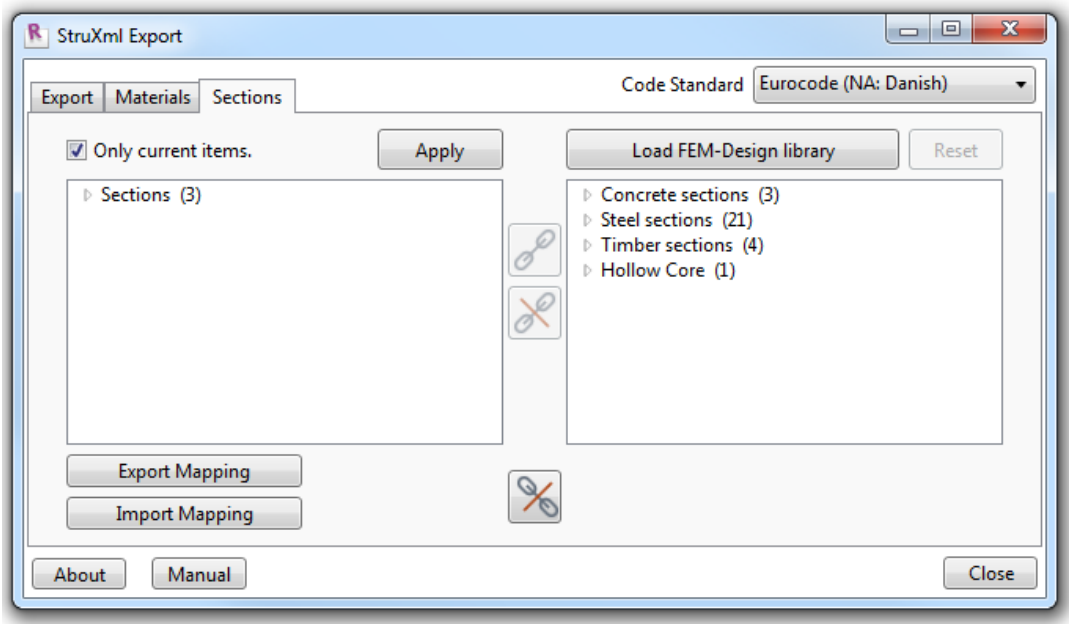



Figure III-43

Only current items

At a start you can decide if you want to see (and map) all the sections from the Revit model library or only those that are currently being used in the model. *Only current items* option is always checked by default. Uncheck it in order to see all the sections and profiles from in the Revit library.

In the left window Revit sections are listed without being grouped to any category.

Mapping procedure

a) Click on the small triangular next to the *Sections* to open a drop down list of sections to be mapped and select one item. Any section that has to be mapped will be displayed in red. Any of section that have already been mapped before will be displayed in black.
b) Find a corresponding section in the FEM-Design library in the right part of the dialog. Click on the small triangular next to the section category to open a drop down list of available sections from the FEM-Design library and select a desired one.
c) Press *Map selected item* button. A section that has just been mapped will turn black.
d) In order to save the sections mapping (and update the warning list in the status window in the *Export* tab) press *Apply*.

Unmapping procedure

- If you wish to unmap any of the previously mapped sections, select the section in the Revit sections window and press the *Unmap Selected Item* button.
- If you wish to unmap all of the previously mapped sections, do not select any section – just press the *Unmap All Mapped Items* button.

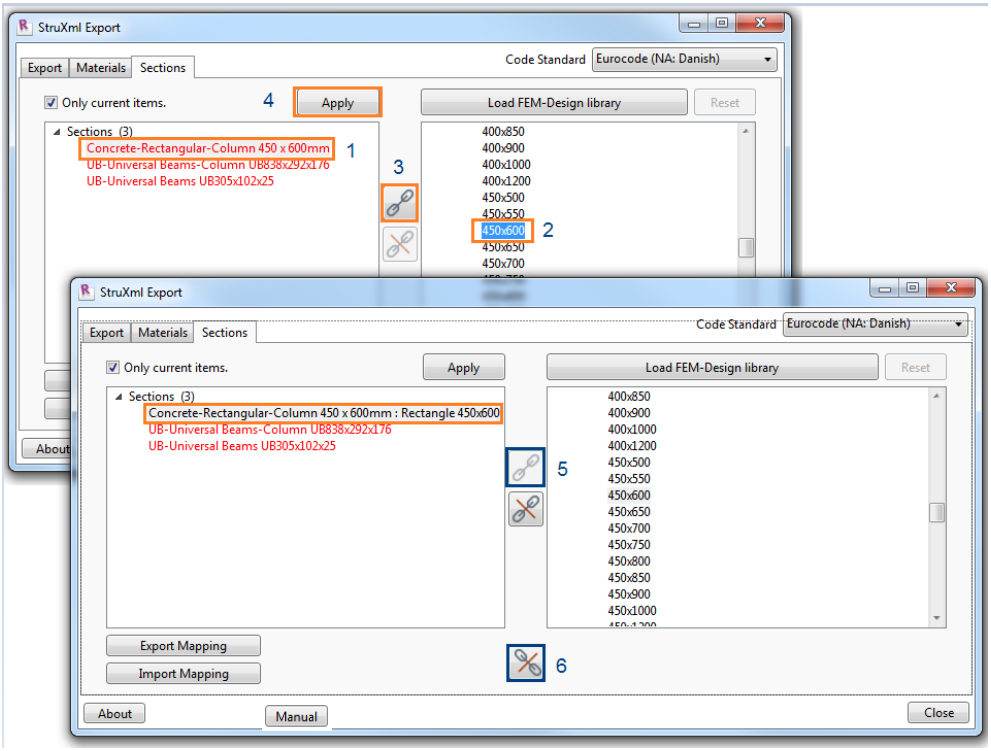



Figure III-44

Load FEM-Design library

If you added additional sections into FEM-Design sections library and you wish to perform mapping into those sections, you need to update the default FEM-Design library to the desired one. The code of the new sections library imported from FEM-Design has to be consistent with the chosen *Code Standard* in the *StruXML Export* dialog.

The procedure is as follows:

1. In FEM-Design go to the *Default settings* of e.g. *Colum.*
2. Go to *Sections* tab.
3. Press the *Export...* button.
4. Chose *Sections library (*.struxml)* in Save as type, find a location and name the library.
5. Chose *Save* button.
6. Go back to the *Sections* tab in the *StruXML Export* dialog and press *Load FEM-Design library.* Find and select the desired struXML file.
7. Press *Open* button.

Remember to always load the current FEM-Design library. This will prevent from mapping to a section that may not exist in the current library used in FEM-Design.

Press *Reset* button in order to reset the materials library to the default one.

Export / Import Mapping

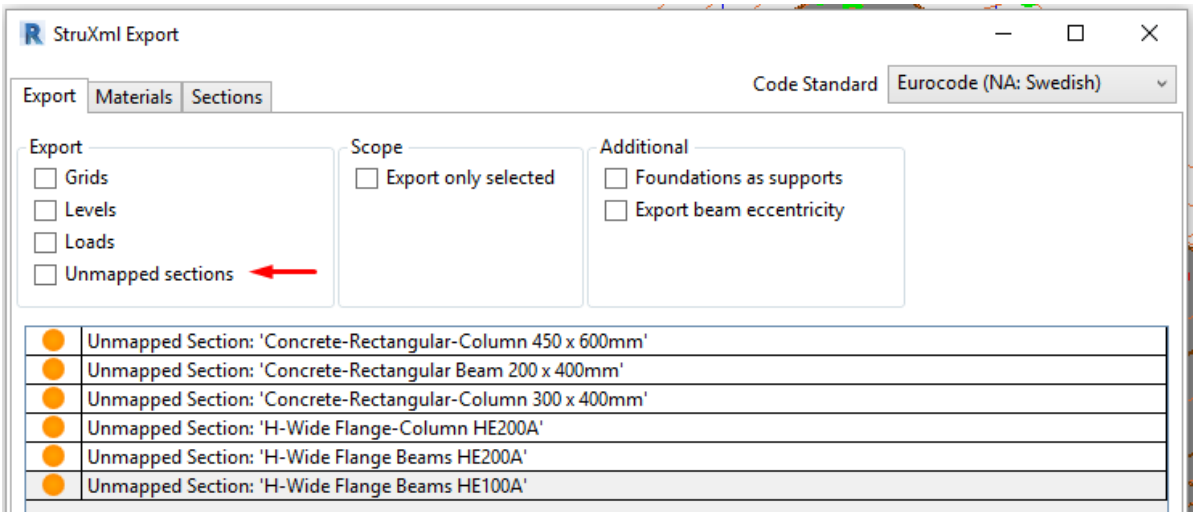
Once performed sections mapping can be saved to struXML file using *Export Mapping* button. Use *Import Mapping* button in order to load mapping from struXML file.

## 9.4. Export unmapped sections

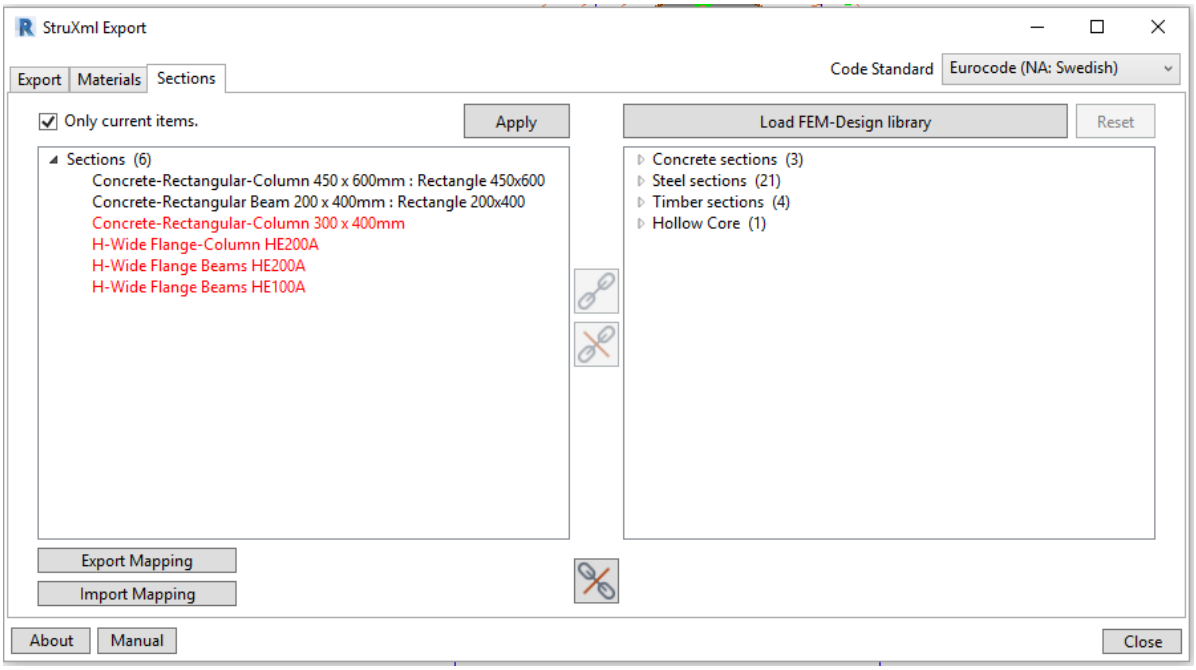
A new option has been introduced in the StruXML Export that allows for exporting unmapped section.

In previous versions of the StruXml StruSoft Add-In, both all the materials and sections had to be mapped to corresponding objects from FEM-Design library, otherwise elements with unmapped material or section were not exported.

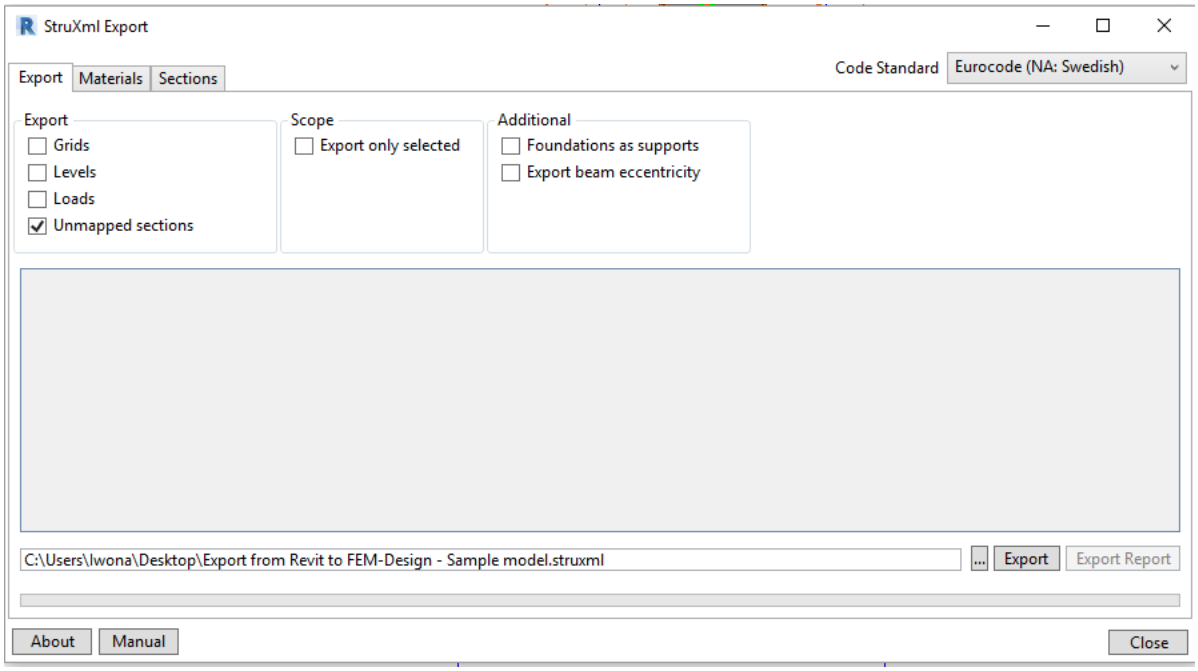
From now on, it is possible not to map the sections (the materials still have to be mapped) and let the StruXML StruSoft Add-In create them automatically in FEM-Design.



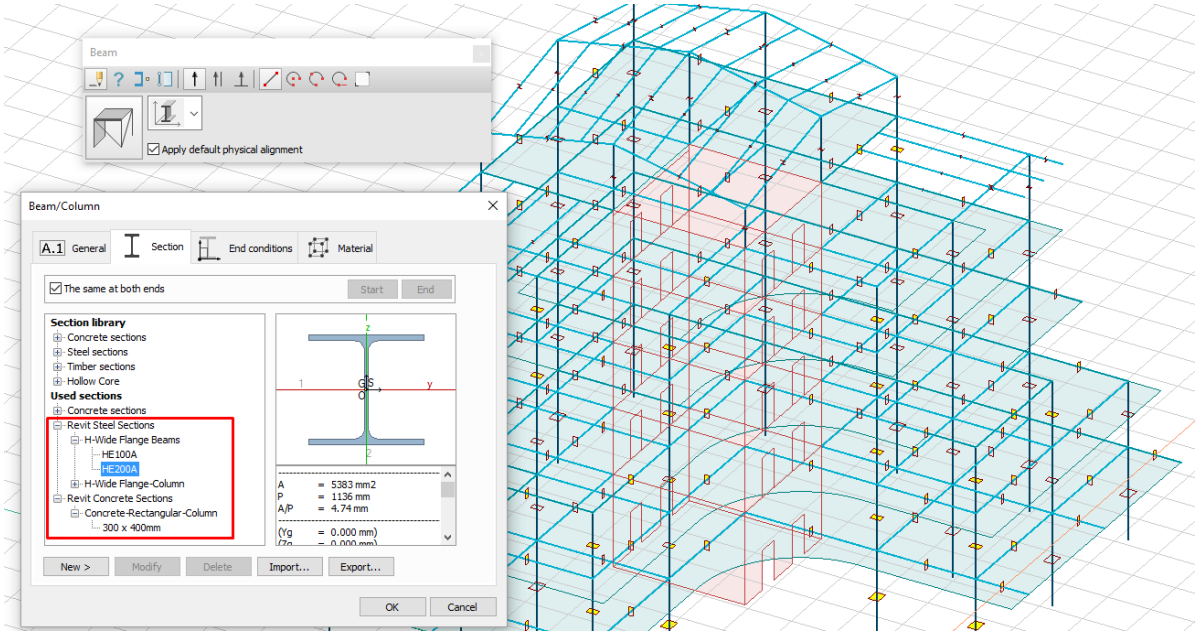

It is possible not to map any section, or map only some sections, and let the Add-In create the rest, as shown in the example below.




In the Export tab, simply chose to Export unmapped sections (when this option is chosen, the list of warnings about unmapped sections will disappear).

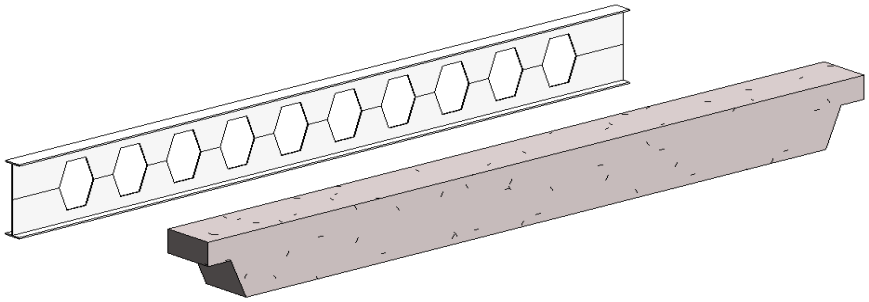


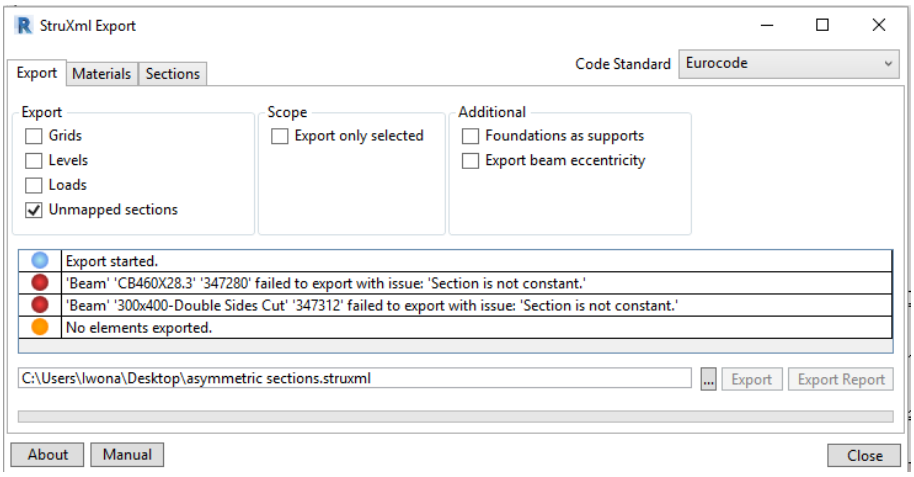

In FEM-Design, the unmapped Revit sections are listed under Used sections, and are named as the sections in Revit.




Keep in mind that if you choose to export unmapped sections, new sections will be created in FEM-Design, even though similar sections exist in FEM-Design library. If you want your Revit object to have the original FEM-Design sections, use the regular mapping procedure.

In case of an attempt of export unmapped sections that do not have a constant section (examples shown below) the Add-In will send a warning that such section cannot be exported (since it cannot originally exist in FEM-Design).

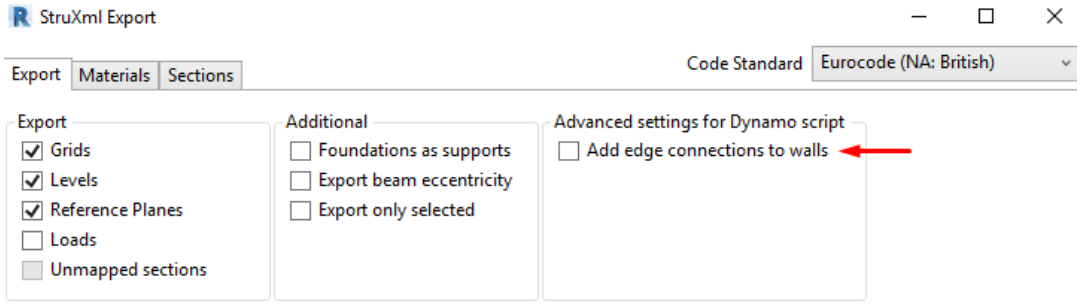


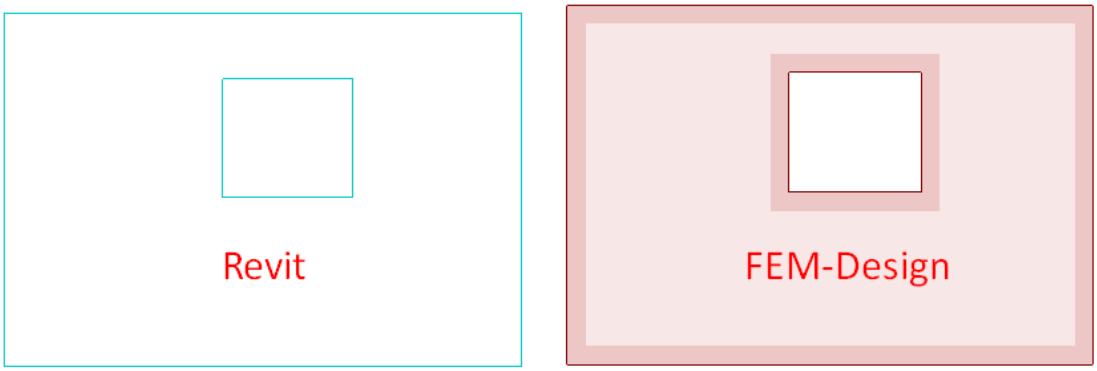

StruXml Export
Export
Materials
Sections
Code Standard
Eurocode
Export
Grids
Levels
Loads
Unmapped sections
Scope
Export only selected
Additional
Foundations as supports
Export beam eccentricity
Export started.
'Beam' 'CB460X28.3' '347280' failed to export with issue: 'Section is not constant.'
'Beam' '300x400-Double Sides Cut' '347312' failed to export with issue: 'Section is not constant.'
No elements exported.
C:\Users\Iwona\Desktop\asymmetric sections.struxml
Export
Export Report
About
Manual
Close

## 9.5. Advanced settings for Dynamo Script

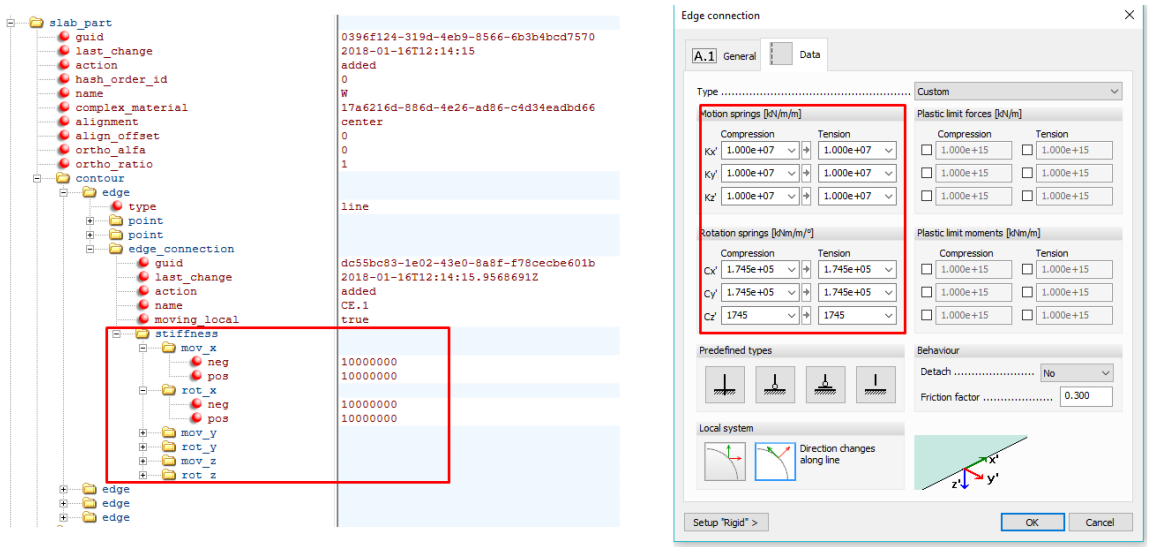
### 9.5.1. Add default edge connections to walls

If you check the option to *Add edge connections to walls,* each wall edge including openings will be exported with en edge connection parameter in struxml.




Here is how the edge connections are defined in struxml and how this edge connection looks like in FEM-Design.




This feature is implemented for those, who are post processing the struxml file, before it is opened in FEM-Design, for example with the Dynamo. When the edge connections are defined for each edge, they can be easily modified or deleted externally.

## 9.6. Warnings and errors

Here is a list of all possible warning and error messages that can appear while using the Revit to FEM-Design StruXML Add-In:

| | |
|---|---|
| Warning: | Unmapped Material: “Name of the material from Revit” |
| Explanation: | This warning may appear in the status window after launching StruXML Export dialog. It only warns that there are some materials in the Revit model that have not been mapped yet. The warning will disappear after the material is mapped and the material mapping is confirmed with the Apply button. |

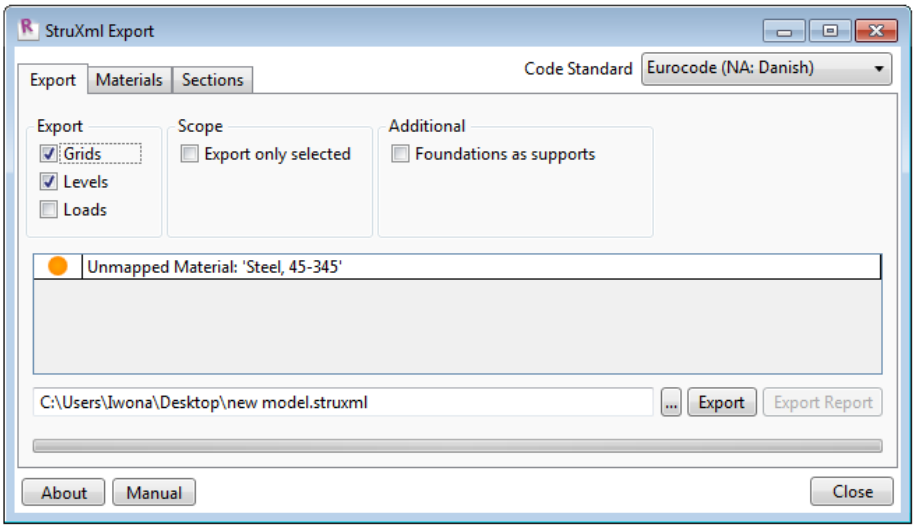



Figure III-45

| | |
|---|---|
| Warning: | Unmapped Section: “Name of the section from Revit” |
| Explanation: | This warning may appear in the status window after launching StruXML Export dialog. It only warns that there are some sections in the Revit model that have not been mapped yet. The warning will disappear after the sections are mapped and the sections mapping is confirmed with the Apply button. |

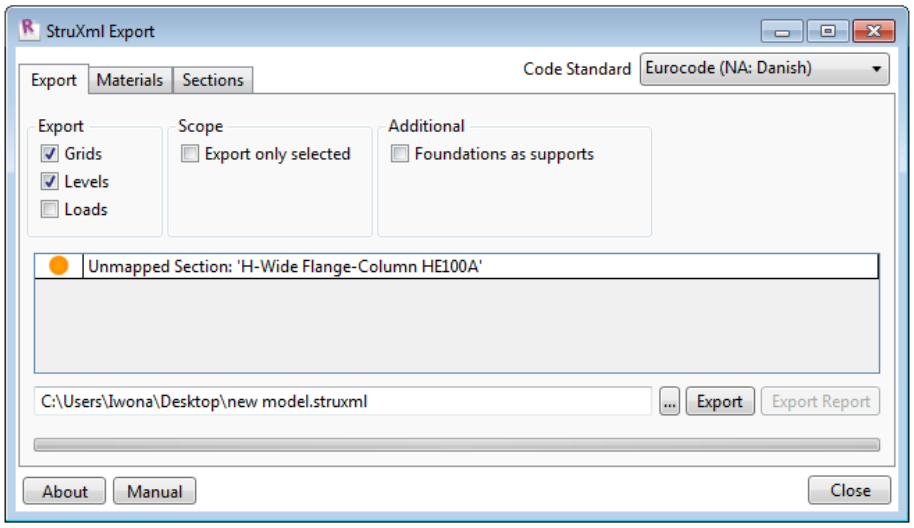


Figure III-46

| Warning: | Element: “Type” ”Number“ can’t export eccentricity. |
|---|---|

Explanation: This warning may appear in the status window after exporting a beam section for which eccentricity settings could not be exported. The eccentricity configuration that cannot be exported to struxml is when the start and end part of the analytical beam are not the same. Read more in Chapter 8.7.

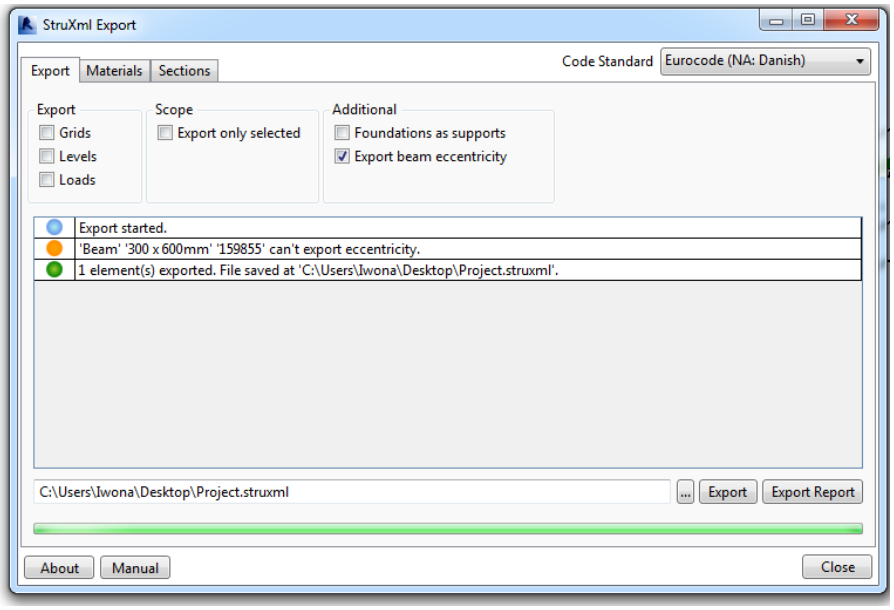



Figure III-47

| Warning: | Element: “Type” ”Number“ column is slanted. |
|---|---|

Explanation: This warning may appear in the status window after exporting a column that is slanted. This is to warn the user in case a slanted column was just a modeling mistake.

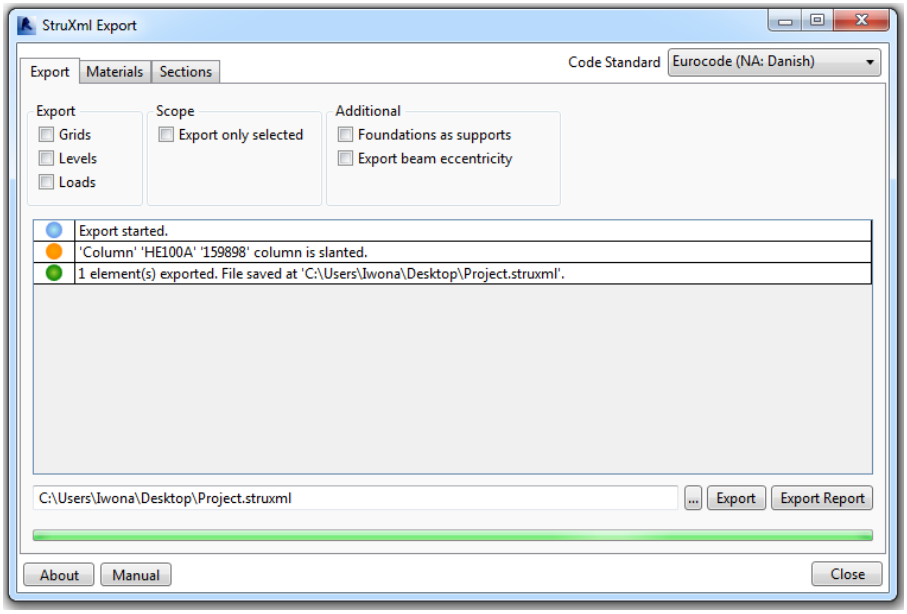



Figure III-48

| | |
|---|---|
| Error: | The code standard of the file and application differ. |

Explanation: This error will appear if you try to *Load FEM-Design library* that has a different *Code Standard* than the chosen *Code Standard* in the *StruXML Export*. Both the code standards have to be consistent in order to perform a valid mapping.

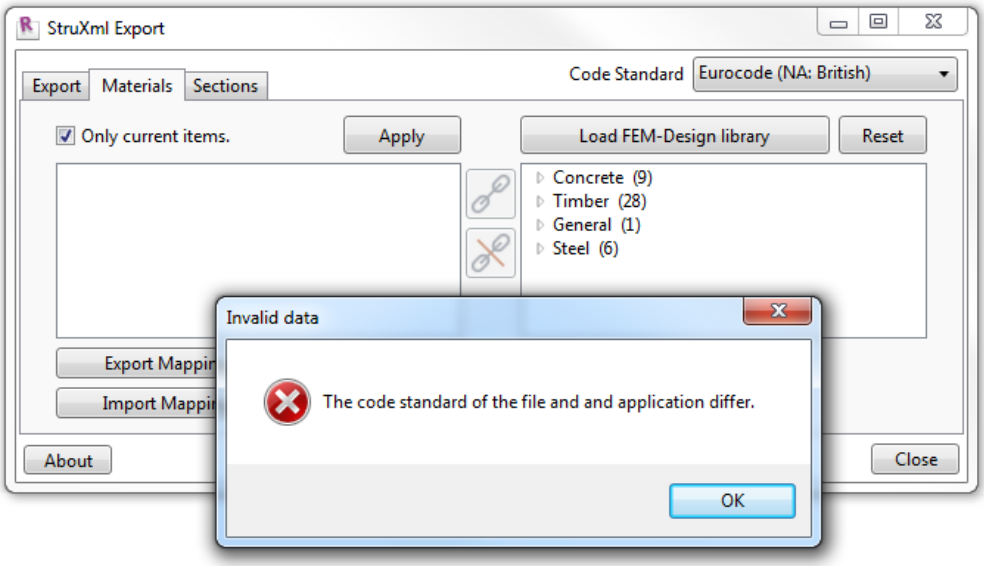



Figure III-49

| | |
|---|---|
| Error: | There are no structural elements in the document, therefore levels cannot be exported. |

Explanation: This error will appear if you try to *Export Levels* but there are no elements in the Revit model. This is not possible as the size of the *Storey* in FEM-Design cannot be then calculated if there are no elements in the Revit model.

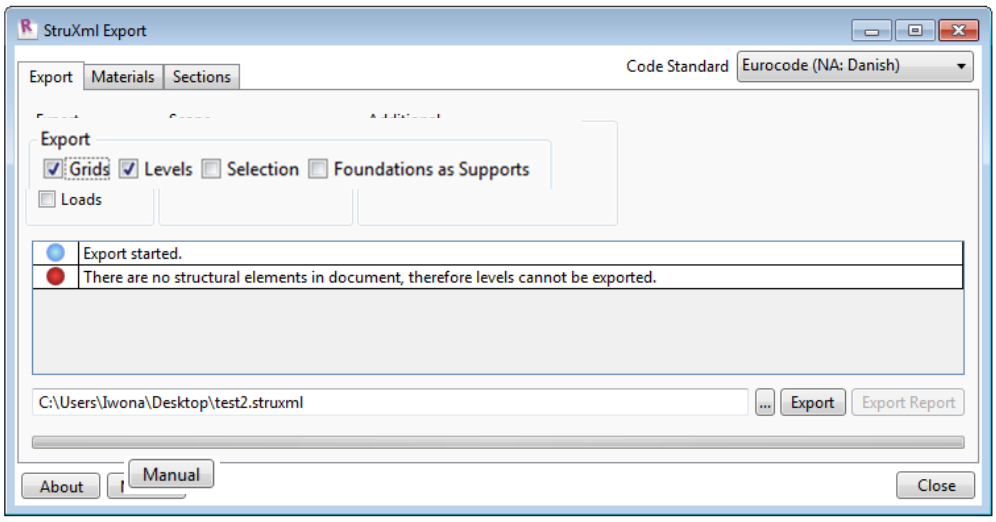



Figure III-50

| | |
|---|---|
| Error: | "Element type" "Type" "Number" failed to export with issue: 'Lack of analytical model' |

Explanation: This error will appear after an attempt to export a Revit model that includes an element that do not have analytical model enabled. Elements with no analytical model cannot be exported to FEM-Design. This will warn you about it and give you a chance to correct your model and repeat the export.

In case there are some elements that should not be exported to FEM-Design and therefore, they do not have analytical model enabled (e.g. partition walls), this error can then be ignored.

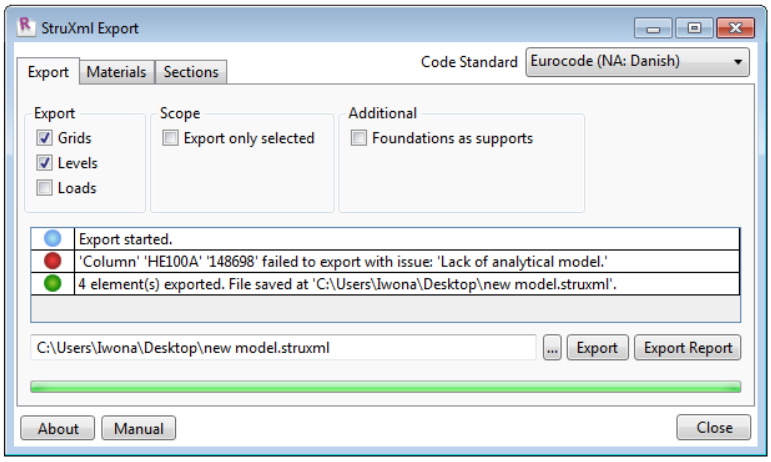



Figure III-51

| | |
|---|---|
| Error: | "Element type" "Type" "Number" failed to export with issue: 'No structural material assigned to it' |

Explanation: This error will appear after an attempt to export a Revit model that includes an element that do not have a structural material assigned to it. If a structural material is set to be *<By Category>* it cannot be exported to FEM-Design. This will warn you about it and give you a chance to correct your model and repeat the export.

This kind of error may appear only for elements that have analytical model enabled. In case there are elements in your Revit model that are not meant to be exported and therefore, do not have analytical model and structural material assigned, you will first receive information about lack of analytical model in those elements.

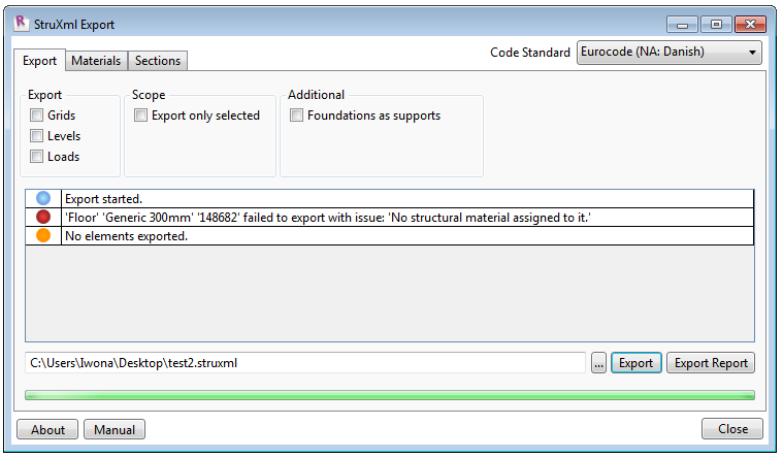



Figure III-52

| | |
|---|---|
| Error: | Element: "Type" "Number" failed to export with issue: 'Material has not been mapped' |

Explanation: This error will appear if the warning about unmapped material has been ignored and the export was performed anyway. Element without mapped material cannot be exported to FEM-Design.

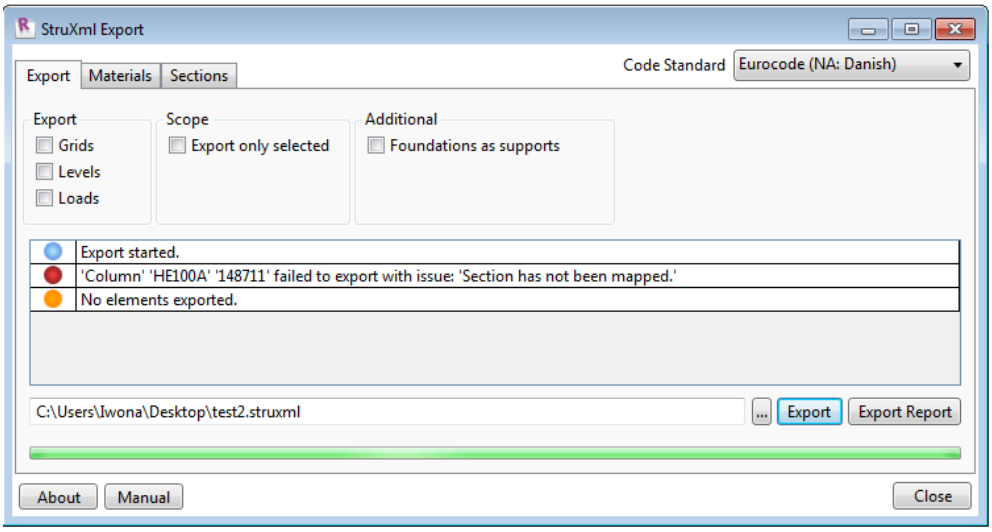



Figure III-53

| | |
|---|---|
| Error: | Element: "Type" "Number" failed to export with issue: 'Section has not been mapped' |

Explanation: This error will appear if the warning about unmapped section has been ignored and the export was performed anyway. Element without mapped material cannot be exported to FEM-Design.

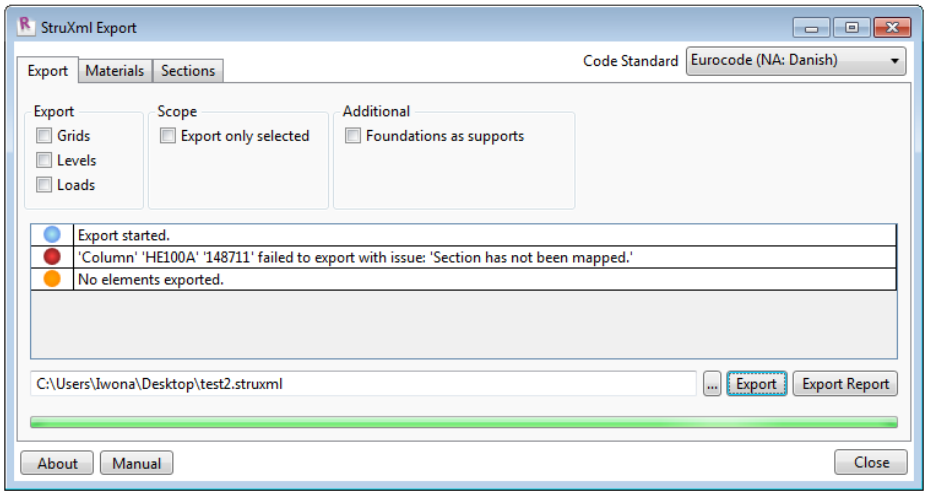



Figure III-54

Double click on the warning message in order to zoom into the element with an error.

## 9.7. Export report

After a Revit model is exported to struxml, an *Export Report* button will become active In the *StruXml Export* dialog.

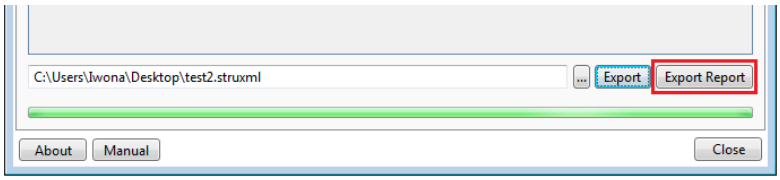



Figure III-55

Upon pressing the *Export Report* button, the Extended Report dialog will appear. Report is based on listing of elements in three groups. User can decide according to which criterion the elements should be listed in each of the groups.

One can chose between:

- *Exported*: elements are grouped based on the fact if they were successfully exported or not.
- *Name:* elements are grouped by the elements group name, e.g. Beam, Column, Grid line, etc.
- *Type:* elements are grouped by the type of the family instance.
- *ID:* elements are grouped by the ID number.
- *Message:* elements are grouped by the type of the message / warning message.

Upon clicking in the small triangular symbol next to each group of elements listed in the report, this group can be minimized. Three different examples are presented below:

### Exported → Name → Type.

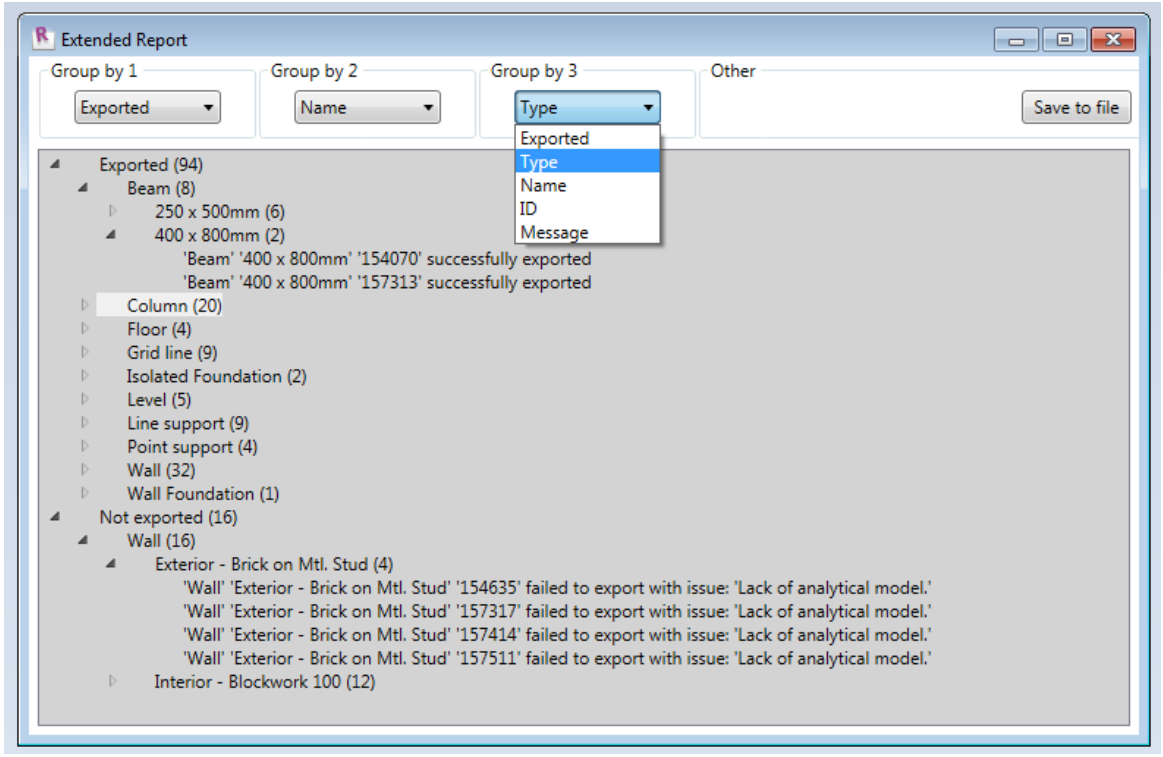


Figure III-56

Figure III-56 shows an example of how the report can be arranged. In the first group, elements are grouped by *Exported* into exported (94 elements) and not exported elements (16). Then elements are grouped by *Name*. There are 8 beams exports, 20 columns, 4 floors, 9 grid lines, etc. There are also 16 walls that were not exported. Finally, the elements are grouped by *Type*.

We can see that out of 8 beams that were exported, 6 of them were of a type 250 x 500mm and 2 of them 400 x 800mm, etc. We can also see that there were two types of walls that were not exported and the reason for that was the lack of analytical model.

## Name → Exported → Type.

Figure III-57 shows another suggested way of arranging the report (of the same model as in the previous case).

Elements in a first place are grouped by name. Afterward, they are split into elements of this group that were exported and those that were not. In the last step, they are divided into types. For instance, we can see there were 48 walls, out of which 32 were exported and 16 were not exported due to lack of analytical model. We can also see how many walls of which type were and were not exported.

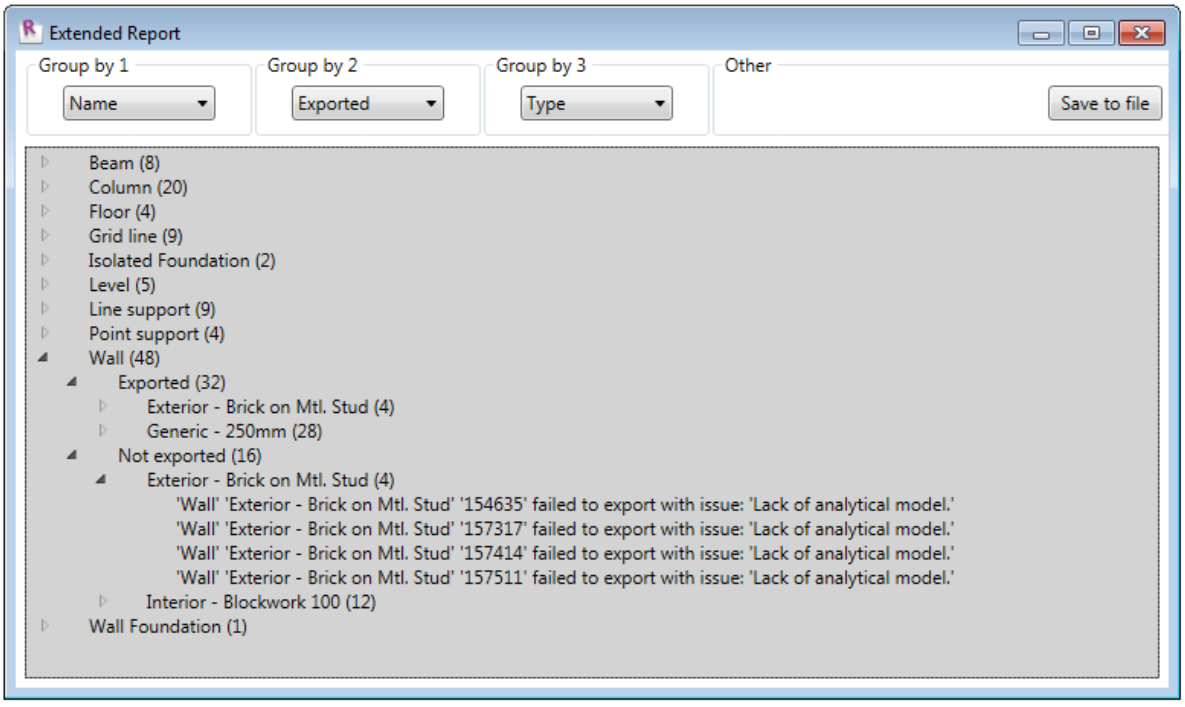



Figure III-57

## Name → Type → Exported

The last example shows how to arrange elements if you want to see how many elements of a given type were exported and how many were not.

In the Figure III-58, we can see that there were 8 walls of a given type, 4 of them were exported and 4 of them failed due to lack of analytical model.

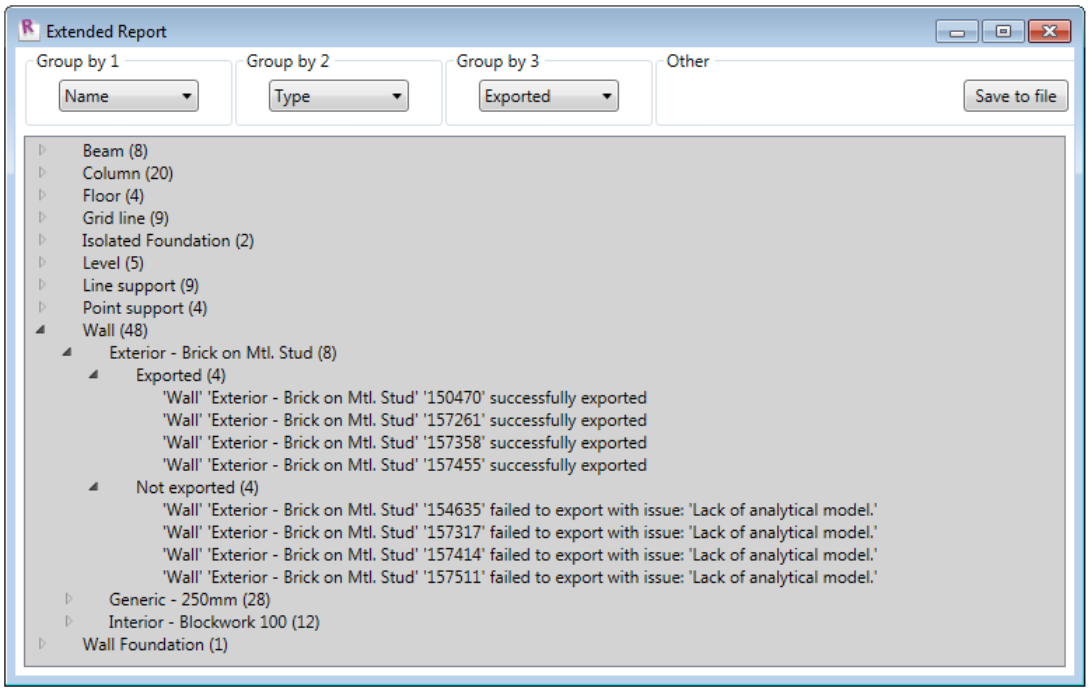


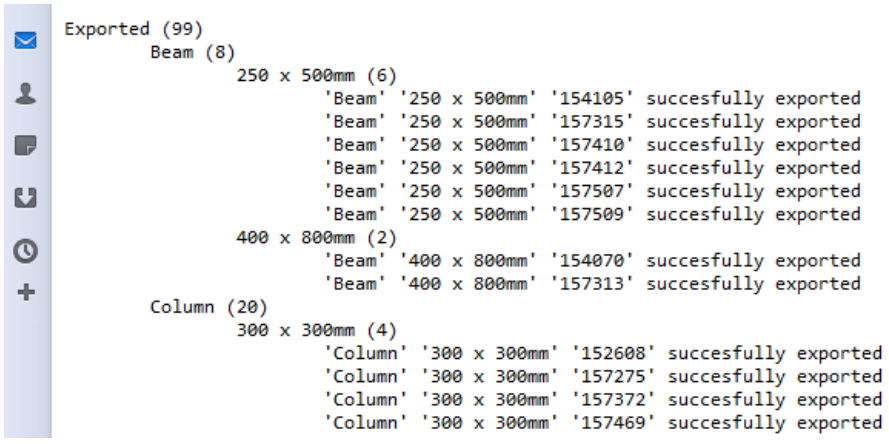

Figure III-58

## Save the report

Upon pressing the *Save to file* button, one can save the generated report into txt file. The .txt file can be nicely open in WordPad / Word or any Internet browser. We do not recommend opening it in Notepad, as the structure of the report is not preserved.

In order to open the .txt file in other instance than Notepad, right click on the file and chose Open with. Then select the desired program (WordPad, Word, Internet browser, etc.). Be aware that when you export the report into a file, all the groups will be maximized displaying all elements listed below each group. However, you can easily modify the report in the text editor and remove the unnecessary lines before printing the report.

Figure III-59 shows how the exported report can look like, when opened in the Internet browser.



```
Exported (99)
        Beam (8)
                250 x 500mm (6)
                        'Beam' '250 x 500mm' '154105' succesfully exported
                        'Beam' '250 x 500mm' '157315' succesfully exported
                        'Beam' '250 x 500mm' '157410' succesfully exported
                        'Beam' '250 x 500mm' '157412' succesfully exported
                        'Beam' '250 x 500mm' '157507' succesfully exported
                        'Beam' '250 x 500mm' '157509' succesfully exported
                400 x 800mm (2)
                        'Beam' '400 x 800mm' '154070' succesfully exported
                        'Beam' '400 x 800mm' '157313' succesfully exported
        Column (20)
                300 x 300mm (4)
                        'Column' '300 x 300mm' '152608' succesfully exported
                        'Column' '300 x 300mm' '157275' succesfully exported
                        'Column' '300 x 300mm' '157372' succesfully exported
                        'Column' '300 x 300mm' '157469' succesfully exported
```

Figure III-59

## 9.8. Open StruXML

StruXML file containing exported model from Revit has to be open directly in FEM-Design. StruXML files can only be open in the *3D Structure and 3D Frame* module of *FEM-Design.*

Procedure is as follows:

1. Open FEM-Design 3D Structure.
2. From the main menu choose: *File* → *Open.*
3. Choose StruSoft common structure XML files (*.struxml) in the Files of type and browse for the desired file.
4. Press *Open* button.

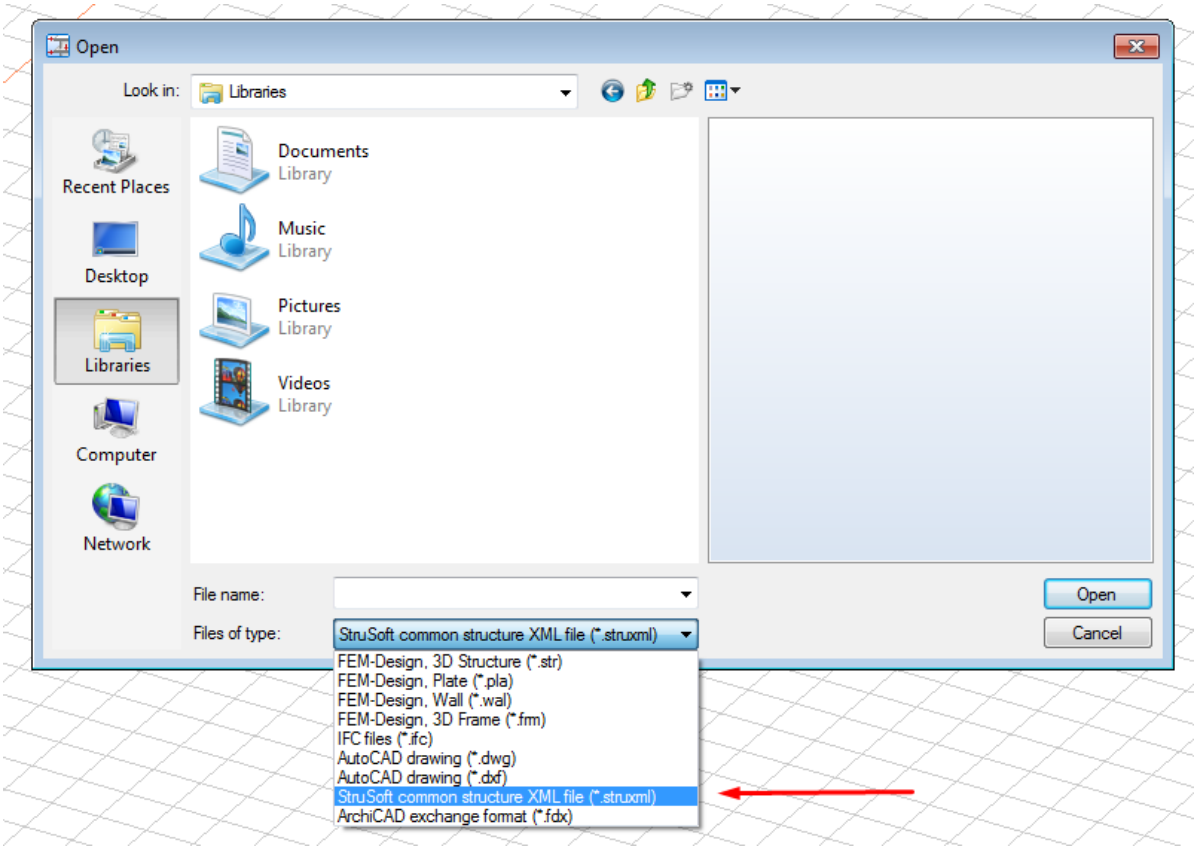


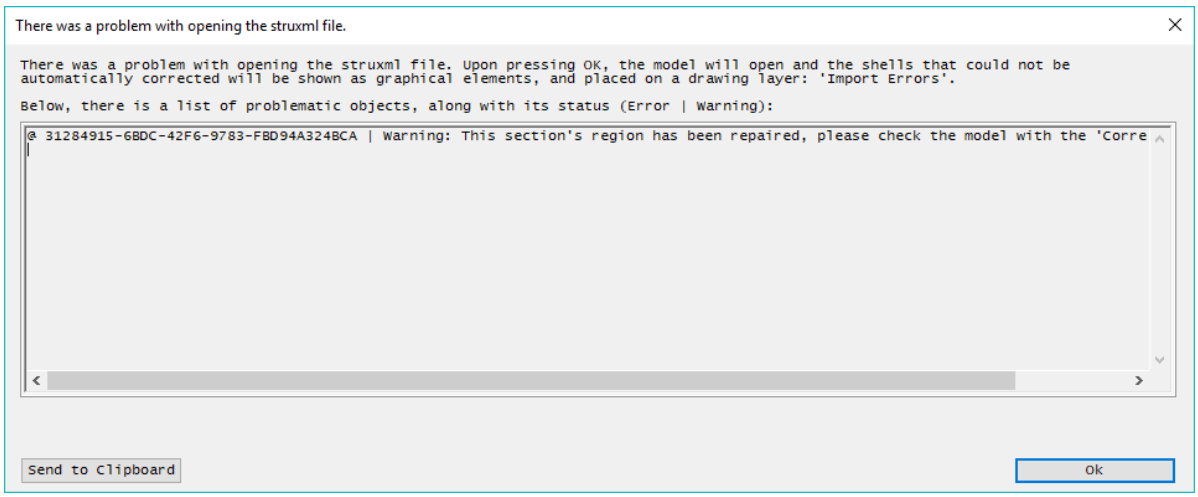

Figure III-60

## 9.9. Errors at opening struxml in FEM-Design

It can happen that the exported struxml file can contain some geometry definitions that are not compatible with FEM-Design. To be very general, two things can happen in such situation:

- Certain bad element cannot be imported, and will be excluded from the opened model in FEM-Design
- Certain bad element can be corrected automatically, and be imported along with the rest of the model to FEM-Design.

This is how the dialog will look like if you come across some errors / warning at opening the struxml in FEM-Design.




Now, when one imports a struxml file that contains some bad shell geometry (for example shell with points of plane), FEM-Design 17 is trying to repair it to a FEM-Design compatible format. When the repaired model is imported to FEM-Design, it is necessary to run the Correct Model tool, to adjust the model geometry.

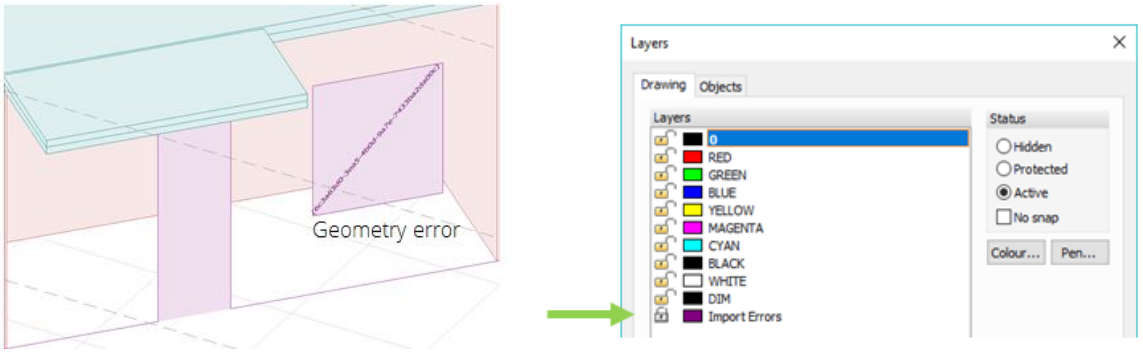
Warning message in such will look like this:

@ 0F4E02AE-11C7-4C48-82DA-FE5D2B7B453B | Warning: This wall's region has been repaired, please check the model with the 'Correct model' tool!

However, if a shell region cannot be automatically fixed and imported, it is now visualized and placed on special layer called Import errors. It is just a graphic image that shows what the bad object looks like and where it is located.

Error message in such will look like this:

@ 51DE2BAA-FBDB-433D-B05A-D1196F962E8C | Error: This wall's improper region cannot be automatically corrected!



There can be a number of other warning or error messages that can appear at opening incompatible struxml file. There is solution to most of the errors, and if you cannot manage it on your own, contact the Support.

Some messages are pure informational, like for example this one:

@ 772BCA3B-712A-49A3-A0DC-8FDE5BE7142D | Warning: The 'Revit Steel Sections, HPC' has been automatically corrected!

This only means that some section part of geometry was corrected when section was automatically created at the export from Revit.

## 10. Limitation and recognized issues

Structural material given to adjacent object

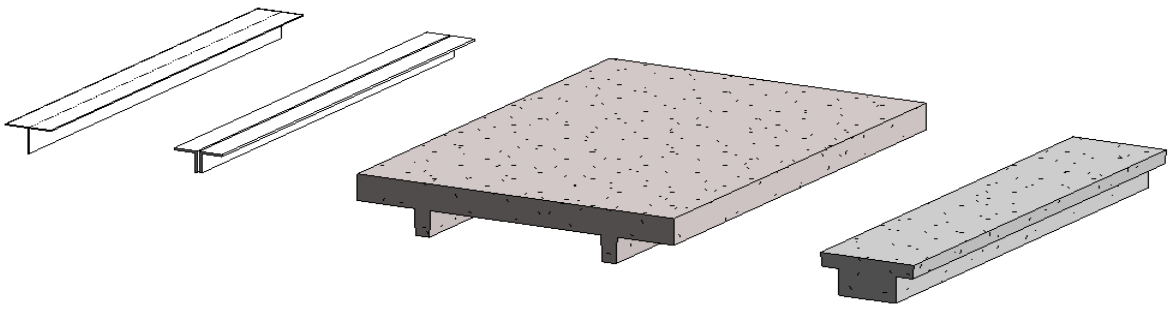
If a single linear object (beam, columns) does not have a valid structural material (is set to <By Category>), it cannot be exported due to lack of material. Also the Structural material search tool will recognize it as object without a valid material.

But, in some situations when this linear object is connected (in a certain way) to another object e.g. floor that has a valid structural material, Revit will by default apply this material to our linear object. This is however, not clear from the user interface – our object's material will still be set to <By Category>. In such case, the linear object will not be recognized as one without a material and will be exported with the material given by the adjacent object.

It is not known to us what the exact situations are and when the adjacent object's material will be given by Revit to a connected object without a material.

Eccentricity issues in export of (some) mono-symmetric profiles

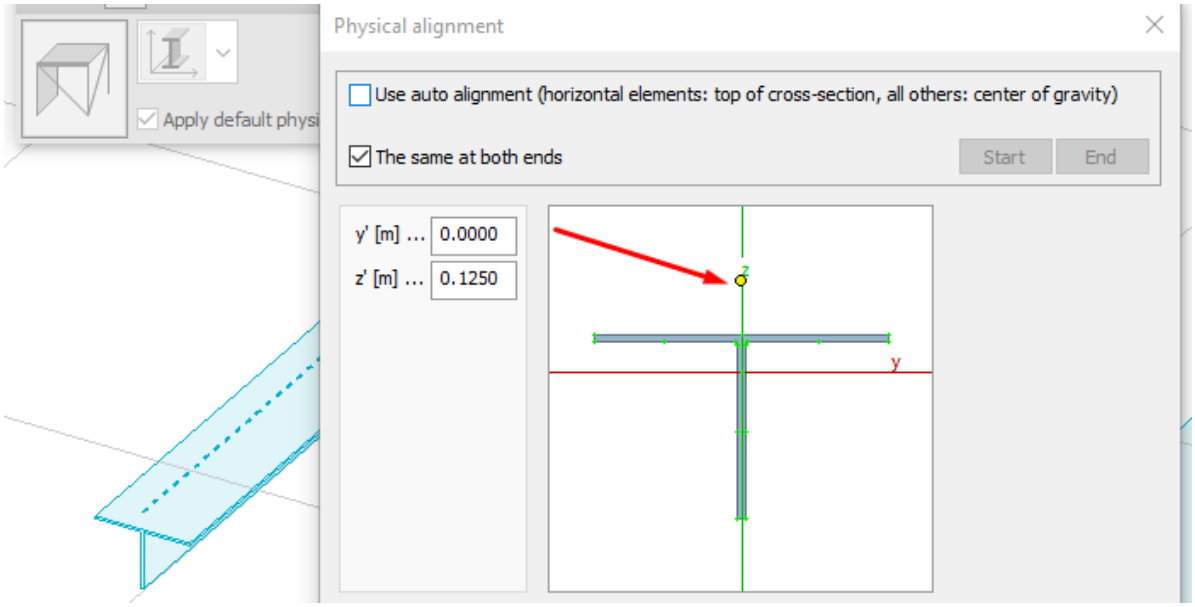
Some of mono-symmetric sections families in Revit (examples shown below) often have a reference line defined in the middle of section height, which is not a center of gravity of that section.



When such section is exported to FEM-Design with the option to export beams eccentricity, the center of the gravity is applied correctly, but the eccentricity settings (how a physical section is

located in relation to the beam's analytical model) is not read correctly due to the difference between Revit and FEM-Design's approach.

Example of such is shown in figure below. The physical eccentricity should be set to top, but instead is placed outside it the section.

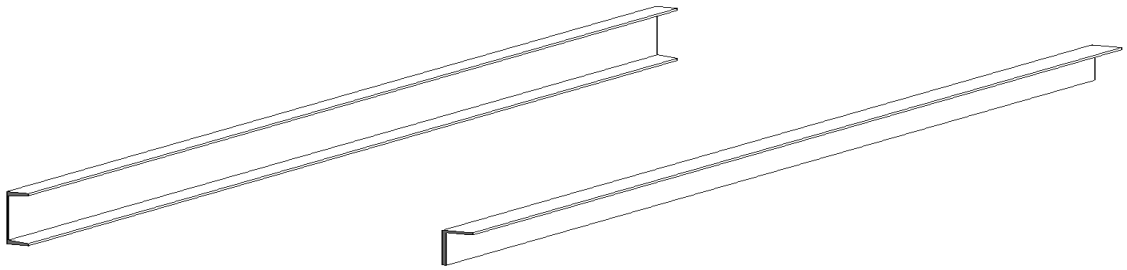



The solution is to:

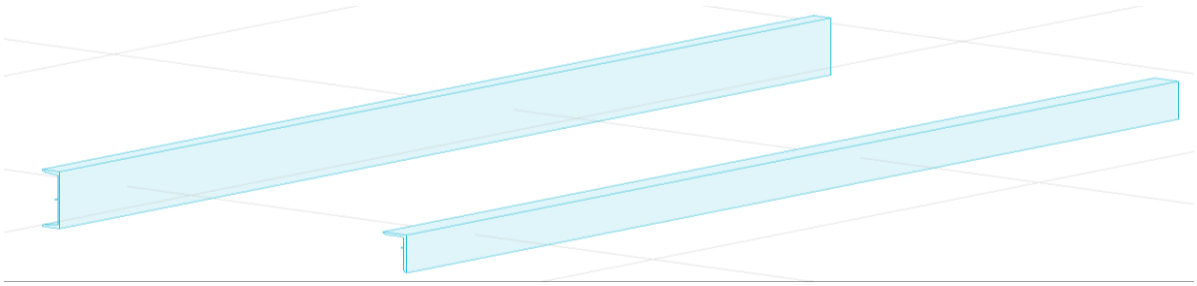
- adjust the physical eccentricity manually in FEM-Design,
- or do not export beam's eccentricity - in such case the physical eccentricity in FEM-Design will be applied to center of gravity of a section). This is the default option in export settings.

Mirrored section issue in export of asymmetric and (some) mono-symmetric profiles

Example of asymmetric and mono-symmetric profiles in Revit:

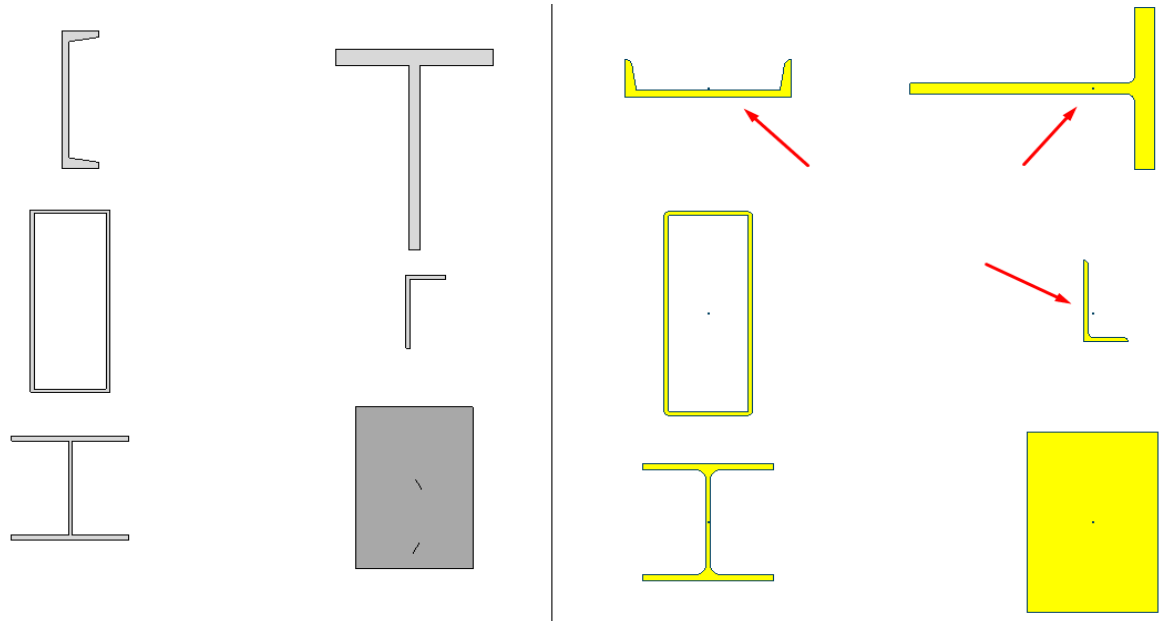


This is how these sections will looks like after importing them to FEM-Design: the sections are mirrored.



Rotation problem in export of asymmetric and (some) mono-symmetric column profiles

Example of different column profiles in Revit and the result of exporting unmapped sections to FEM-Design. As visible, the asymmetric and mono-symmetric profiles are not exported with the correct rotation.



At the moment we are not able to solve any of the issue, but we will do our best to provide some solution in one of the coming Add-In releases.

# IV. Workflow: FEM-Design to Revit

## 11. Data exchange scope

Following elements can be imported from FEM-Design to Revit using the StruSoft StruXML Revit Add-In:

Elements:

- Datum elements (axes, storeys and reference planes)
- Bar elements (beams, columns, trusses)
- Shell elements (walls and plates)
- Profiled plate panels (imported as plates)
- Foundation elements (isolated foundations, wall foundations, foundation slabs)
- Supports
- Loads and load cases

Detailed description on how particular elements are imported, and which properties can be transferred along with the element, are presented in the following chapters.

### 11.1. Datum elements

Datum elements in FEM-Design, i.e. *storeys* and *axes* can be saved into struxml file and imported to Revit project.

#### 11.1.1. Storeys

Definition of *Storeys* is imported to Revit project as *levels* however, the *default* Storey name from FEM-Design is not imported; the imported levels will be called Level 1, Level 2, etc. as shown in the example in Figure IV-1.

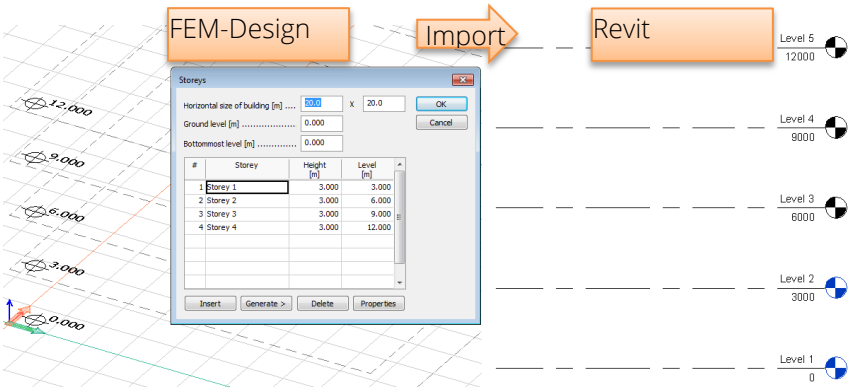



Figure IV-1

#### 11.1.2. Axes

Axes defined in FEM-Design are imported to Revit Project as grids. The exact length and geometry is transferred however, axes names are not imported. In Revit grids will be numbered accordingly with the order of their creation in FEM-Design, as shown in the example in Figure IV-2.

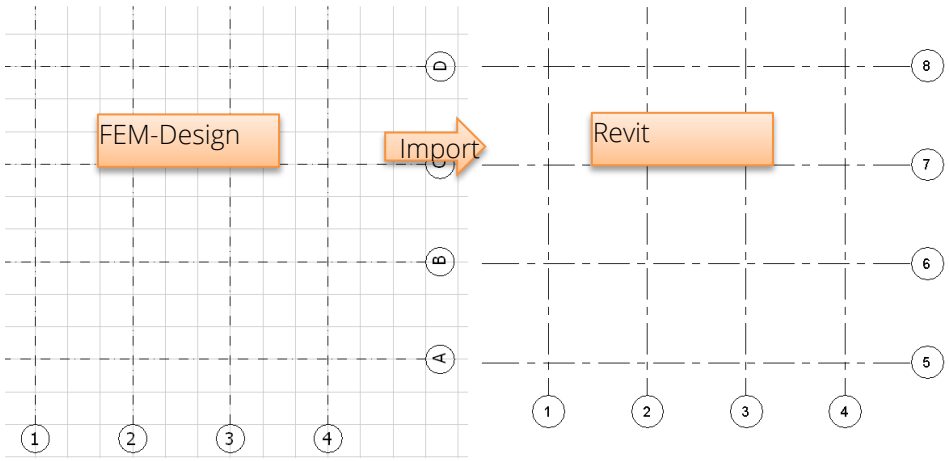



Figure IV-2

### 11.1.3. Reference planes

Reference planes defined in FEM-Design are imported to Revit Project as reference planes.

## 11.2. Bar elements

Every section and material of bar elements defined in FEM-Design has to be mapped into corresponding materials and family types in Revit project. Read more about the mapping process in Chapter 12.4.

### 11.2.1. Columns

A column element defined in FEM-Design is imported to Revit project as a *Structural Column* element. Along with the geometry of a column, following properties are transferred:

- Rotation
- *Releases*: assigned as *Start* and *End release* parameter of analytical column.
- *Structural Material*: mapping required.

Eccentricity of a column defined in FEM-Design cannot be transferred into Revit project. The default position of an analytical column, in relation to physical column, will always be *Center*.

### 11.2.2. Beams

A beam element defined in FEM-Design is imported to Revit project as a *Structural framing: Beam* element. Along with the geometry of a beam, following properties are transferred:

- *Rotation*: assigned as *Cross-section rotation* instance parameter of a physical beam.
- *Releases*: assigned as *Start* and *End release* parameter of analytical beam.
- *Structural Material*: mapping required.
- *Eccentricity*: assigned as *y Offset Value* and *z Offset value* instance parameter of a physical beam.

StruSoft AB | Copyright© 2018 StruSoft. All rights reserved. 

### 11.2.3. Truss elements

A truss element defined in FEM-Design is imported to Revit project as a *Structural framing: Brace* element. Along with the geometry of a truss element, following properties are transferred:

- *Rotation*: assigned as *Cross-section rotation* instance parameter of a physical beam.
- *Structural Material*: mapping required.

There is no possibility of defining *release conditions* of a truss element in FEM-Design therefore, imported braces will be assigned with the default Revit release conditions, i.e. user defined (start) / pinned (end). There are no eccentricity settings for a truss element in FEM-Design therefore, braces will be imported with the default position of the analytical model, which is center of element.

## 11.3. Shell elements

Each of the shell elements (plates and walls) has to be mapped separately into corresponding type of a floor of wall element in Revit. Read more about the mapping process in Chapter 12.4.

### 11.3.1. Plates

A plate element defined in FEM-Design is imported to Revit project as a *Floor: Structural.* Along with the geometry of a plate, following properties are transferred:

- *Structural Material*: as parameter of a floor type chosen at mapping.
- *Eccentricity*: one of the three alignment settings from FEM-Design is imported to FEM-Design as *z Projection* parameter of an analytical floor element, according to the following rule:
    - Top /Left alignment from FD --> z Projection: Top of Slab
    - Center alignment from FD --> z Projection: Center of Slab
    - Bottom/Right alignment from FD --> z Projection: Bottom of Slab

Be aware that a manual definition of an eccentricity value for a plate in FEM-Design (e[m]) cannot be imported along with the plate element.

### 11.3.2. Walls

A wall element defined in FEM-Design is imported to Revit project as W*all: Structural.*. Along with the geometry of a wall, following properties are transferred:

- *Structural Material*: as parameter of a floor type chosen at mapping.
- *Eccentricity*: alignment settings from FEM-Design are imported to FEM-Design as *z Projection* parameter of an analytical wall element, according to the following rule:
    - Left alignment from FD --> z Projection: Interior Face
    - Center alignment from FD --> z Projection: Center of element
    - Right alignment from FD --> z Projection: Exterior Face

Be aware that a curve wall in FEM-Design is modeled as a number of regular walls and this is also the way it will be imported to Revit.

## 11.4. Profiled panels

Profiled plate panels created in FEM-Design model can be saved to struxml as plate elements and imported to Revit as *Floor: Structural.* The same rules apply as in case of importing plate elements. Each group of profiled panels will be exported as one floor boundary.

## 11.5. Foundations

Each of the foundation elements has to be mapped separately into corresponding type of an isolated foundation, wall foundation or foundation slab. Read more about mapping process in Chapter 12.4.

### 11.5.1. Isolated foundation

An isolated foundation element defined in FEM-Design is imported to Revit project as a *Structural foundation: Isolated* element. Along with the geometry of an isolated foundation element, following properties are transferred:

- *Structural Material*: mapping required.

Isolated foundation does not need a host element (e.g. column) to be imported to Revit.

### 11.5.2. Wall foundation

A wall foundation element defined in FEM-Design is imported to Revit project as a *Structural foundation: Wall* element. Along with the geometry of an isolated foundation element, following properties are transferred:

- *Structural Material*: mapping required.

Wall foundation requires a host element in Revit - Structural wall therefore, wall foundations do not attached to a wall element in FEM-Design will not be exported (warning will be thrown). There are no properties of Wall foundation in Revit that would allow for setting the eccentricity therefore, eccentricity of wall foundation defined in FEM-Design will not be imported.

There is one recognized issue at the import of wall foundation. If wall foundations are added to a wall that contains a wall opening, after import of those elements to Revit, a wall foundation will also be added to the top edge of the opening, as shown in Figure IV-3. This issue will hopefully be solved in one of the next release of the StruSoft StruXML Revit Add-In.

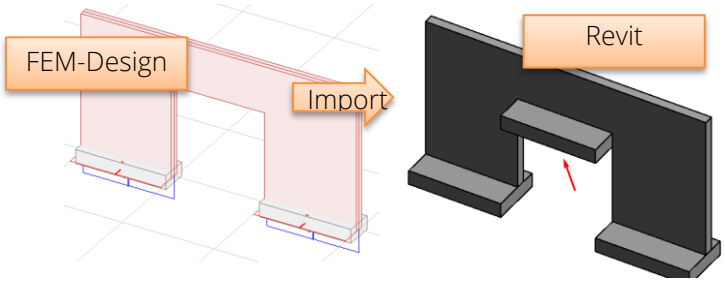



Figure IV-3

### 11.5.3. Foundation slab

A foundation slab element defined in FEM-Design is imported to Revit project as a *Structural foundation: Slab* element. Along with the geometry of a foundation slab element, following properties are transferred:

- *Structural Material*: as parameter of a floor type chosen at mapping.
- *Eccentricity*: alignment settings from FEM-Design are imported to FEM-Design as *z Projection* parameter of an analytical foundation slab element, according to the following rule:
    - Top /Left alignment from FD --> z Projection: Top of Element
    - Center alignment from FD --> z Projection: Center of Element

Be aware that a manual definition of an eccentricity value for a foundation slab in FEM-Design (e[m]) cannot be imported along with the foundation element.

## 11.6. Supports

Both the supports and supports group defined in FEM-Design can be imported to Revit project. The exact support definition is imported to Revit as *User* boundary condition. There are however, substantial differences between the elements type that supports can be added to between FEM-Design and Revit. Please find the details in below paragraphs.

### 11.6.1. Point support and point support group

Point support and point support group defined in FEM-Design are imported to Revit project as *Point Boundary Condition* of a *State*: *User*.

Point boundary condition in Revit requires a host element and can only be applied to an end of a beam, column or brace element. Therefore, if a point support (group) in FEM-Design is applied to a wall or plate element, or not to an edge of linear element, or is independent, a warning informing about the lack of reference element will be thrown at the import.

There is one recognized issue about the import of point support group. If a point support group is defined in a local coordinate system of an element that is different than the global coordinate system of the FEM-Design model, and the stiffness of this point support group is not the same in all directions, an error will be thrown and this support will not be imported.

### 11.6.2. Line support and line support group

Line support and line support group defined in FEM-Design are imported to Revit project as *Line Boundary Condition* of a *State*: *User*.

Line boundary condition in Revit requires a host element and can be applied to a beam, column, wall and floor element. Therefore, if a line support (group) in FEM-Design is applied to a brace, or is independent, a warning informing about the invalid reference will be thrown at the import.

### 11.6.3. Surface support group

Surface support group defined in FEM-Design are imported to Revit project as *Are Boundary Condition* of a *State*: *User.*

Area boundary condition in Revit requires a host element and can be applied to a wall or a floor element. Therefore, if a surface support group in FEM-Design is independent, a warning informing about the lack of reference element will be thrown at the import.

## 11.7. Loads and load cases

Following loads types defined in FEM-design can be imported to Revit:

- Point load
- Line load
- Surface load

Load cases defined in FEM-Design can be imported to Revit project under condition that there is at least one load applied to the structure.

The load cases imported from FEM-Design will be added to the existing load cases in Revit (if any). Type and the duration class parameters are not imported to Revit; all the imported load cases will be assigned with the Dead Nature and Dead loads Category. Figure IV-4 and Figure IV-5 shows an example of the imported load cases.

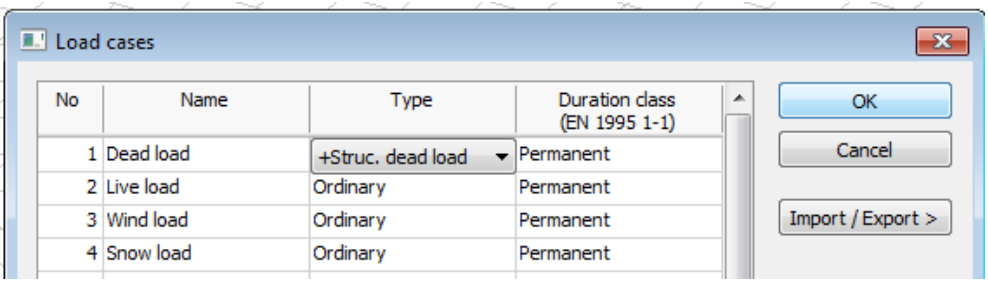



Figure IV-4

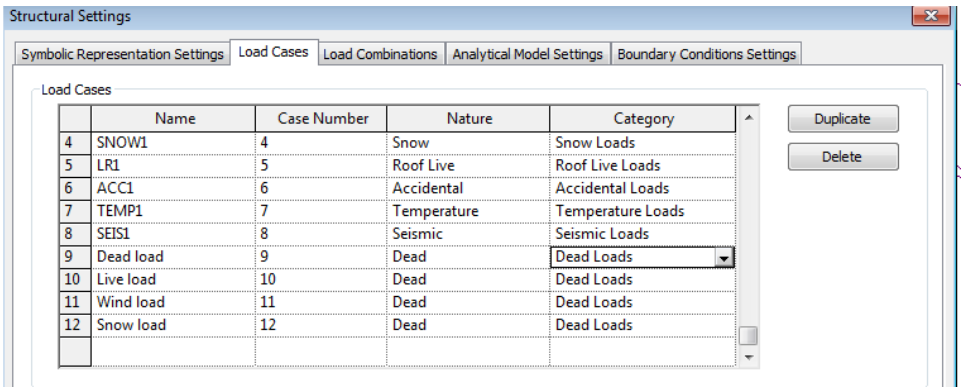



Figure IV-5

If a name of the load cases imported from FEM-Design is the same as name of the existing load case in FEM-Design, import will not be possible and an error message will be thrown.

# 12. Import FEM-Design model to Revit

## 12.1. Requirements:

There has to be at least one level defined in the Revit project into which a FEM-Design model will be imported, or at least one storey has to be included in the imported FEM-Design model. It is required because elements in Revit have to be placed on a level or with a certain reference to a level.

All the frame sections and materials, wall and floor types and foundation element types that are equivalent to the elements and materials existing in the FEM-Design model, has to be included in the Revit project, or proper families and types should be created there prior to mapping. Read more about it in the Chapter 12.4. In one of the next releases of StruXML StruSoft Revit Add-In this will be improved and user will have a chance to create materials and element types from the level of mapping dialog.

## 12.2. Export FEM-Design model to struxml file

The first step is to export the FEM-Design model into a file. This is done by saving it as a *.struxml file type. In FEM-Design go to *File* → *Save as* → type a name in the *File name* field → chose *.*struxml* in the *Save as type* field → *Save*.

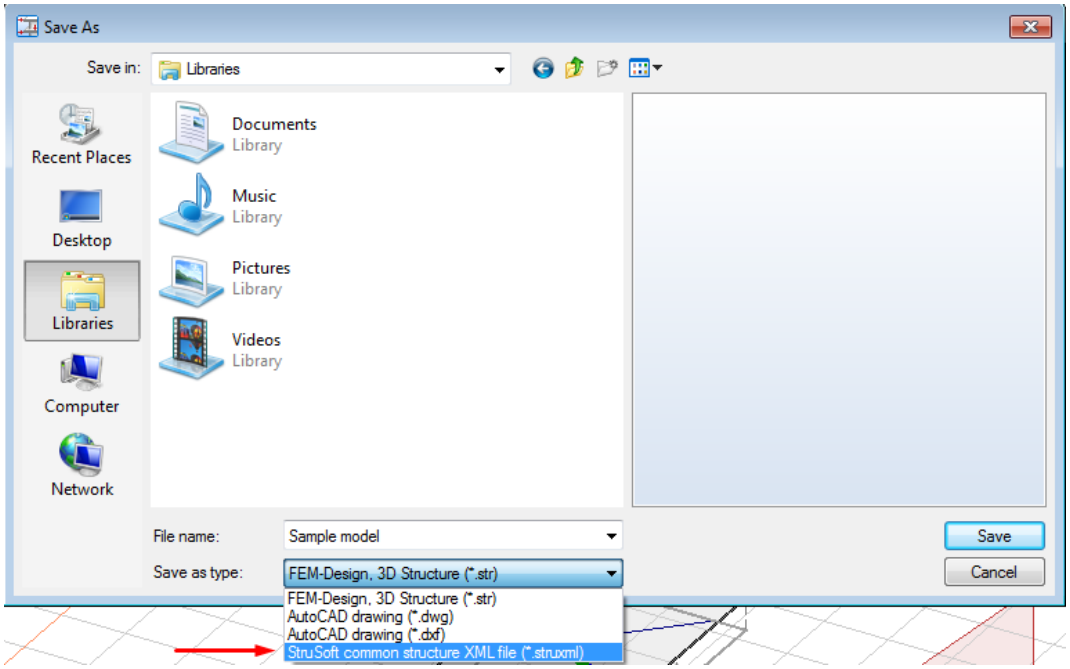



Figure IV-6

## 12.3. Import struxml file to Revit

Go to *Analyze* tab and find *Import StruXML* command in the *StruSoft FEM-Design* panel (Figure IV-7). This is the only command that one has to use in order to import a FEM-Design model to Revit.

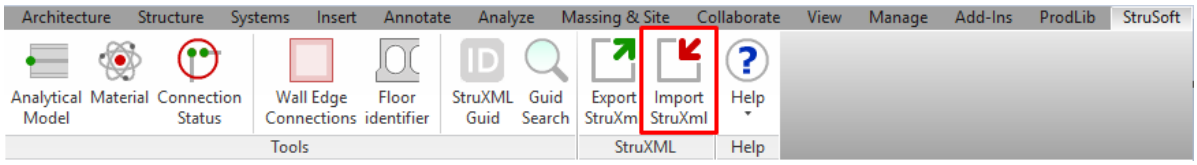



Figure IV-7

Upon pressing the *Import StruXML* command, an *Import StruXML* dialog will pop out, as shown in Figure IV-8

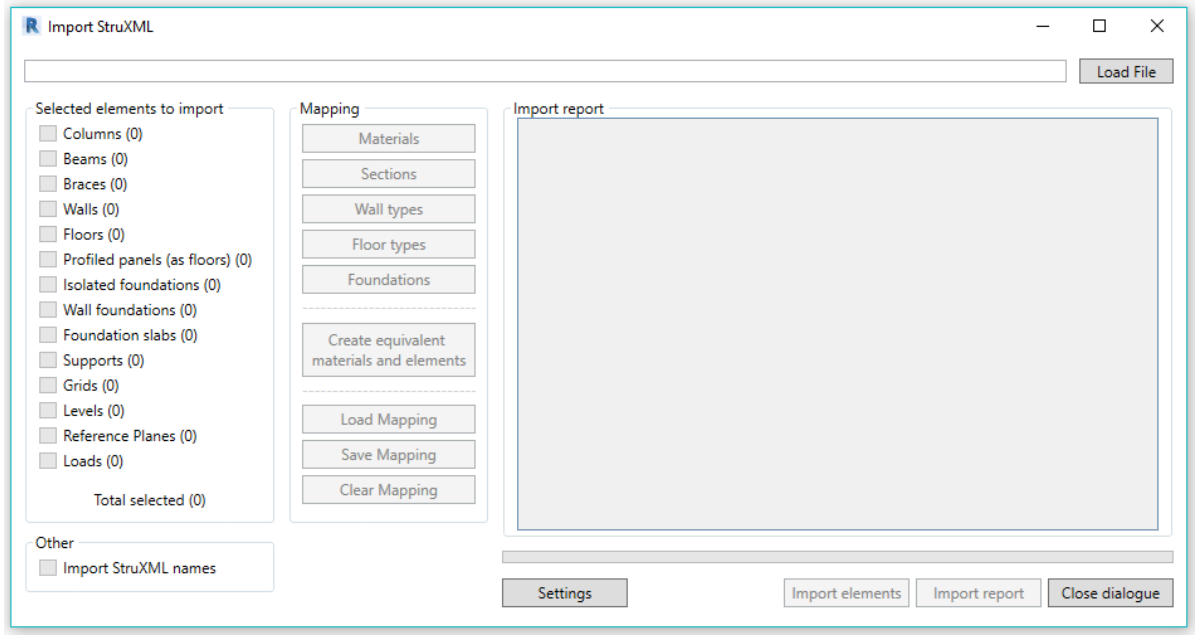
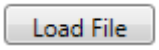



Figure IV-8

### Load file




Use this button in order to load the struxml file to be imported.

Upon loading a struxml file, *Selected elements to import* and *Mapping* panels will become active.

### Select elements to import

*Selected elements to import* panel contains a list of all possible element categories that can be imported to Revit. Only the categories that are present in the struxml file will become active (for instance if there are no grids and levels in the imported model, those categories will remain inactive).

In the *Selected elements to import* panel user has a chance to see how many elements of a given category are present in the file and how many elements in total will be imported. One has also a chance to unselect a certain category that is present in the struxml fie so it will not be imported to Revit project (check Figure IV-9).

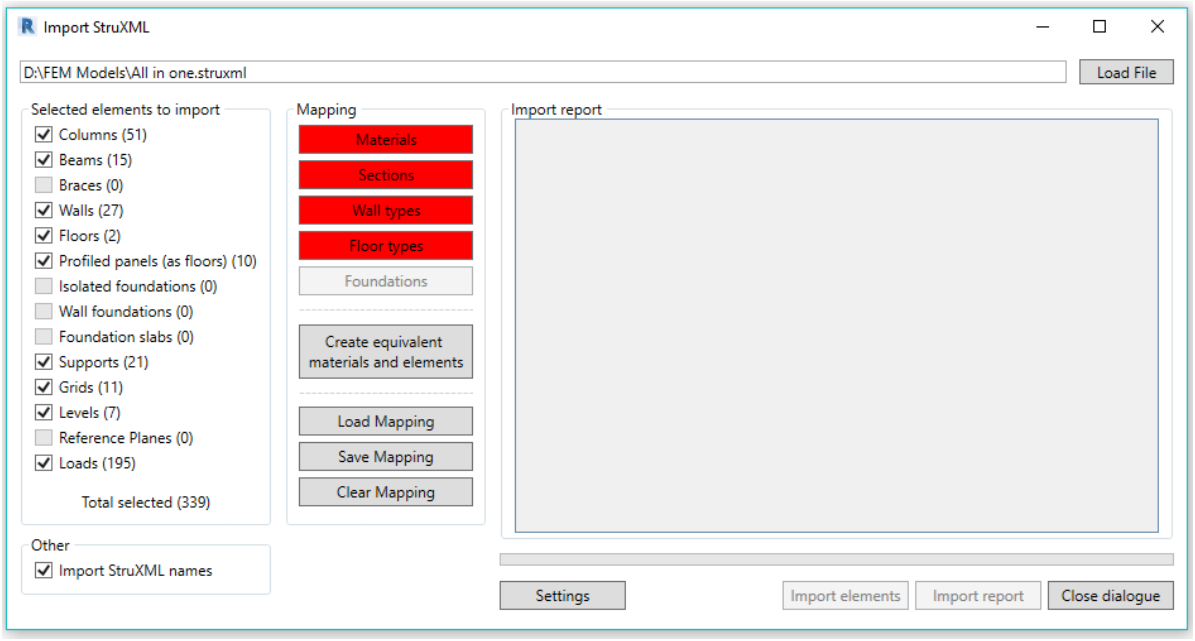



Figure IV-9

## 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. Physical elements ID can only be imported starting from FD 17.01.001.

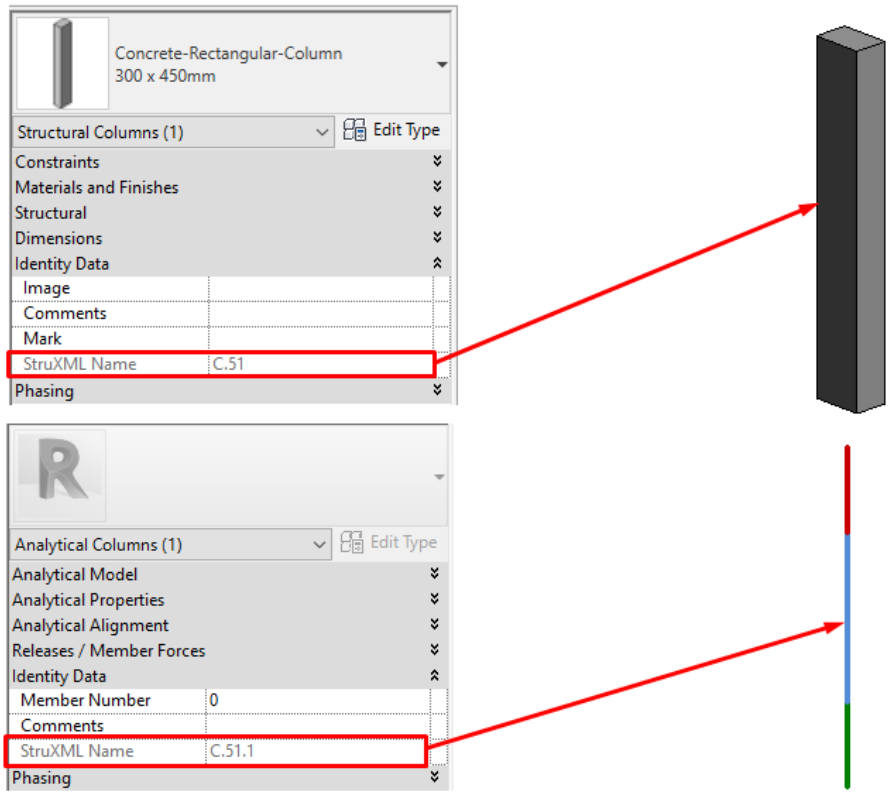



Figure IV-10

## Mapping

In the *Mapping* panel, one should map all elements that are about to be imported into Revit project.

*Mapping* panel contains five mapping buttons that correspond to different kind of element types. Again, only mapping categories for elements that are present in the struxml file, and were selected to be imported in the *Selected elements to* panel, will be active.

If all elements within one category are mapped, the button will turn green. Otherwise it will remain red, until mapping is completed. Read more about the mapping procedure in the Chapter 12.4.

Once done mapping can be saved to struxml, and loaded next time to be used with another model.

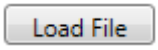
There are two methods of mapping: manual and automatic. Read more about the mapping procedure in the Chapter 12.4.

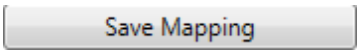
## Import

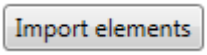
When the mapping procedure is finished, *Import elements* button will become active. It will only become active if all the elements (and materials) were mapped.

Upon pressing *Import elements* button, selected elements will be imported to the Revit project and an *Import report* dialog will become active. There, one can see a list of possible warnings, errors and total amount of elements imported to Revit (check Figure IV-11). Press Import report button, to see a complete overview of all imported elements.

## Import procedure

1. Click on 

and chose a struxml file to be imported.
2. In the *Selected elements to import* panel, decide which elements to export (among those that are present in the struxml file).
3. Optionally: check the *Import StruXML names box.*
4. Perform the mapping (read more about the mapping procedure in the Chapter 12.4.)
5. Optionally: press 

in order to save mapping into struxml.
6. Press 

in order to import the selected elements into Revit project.
7. Optionally: press 

to see an overview of all imported elements.
8. Press 

in order to close the Import StruXML window.

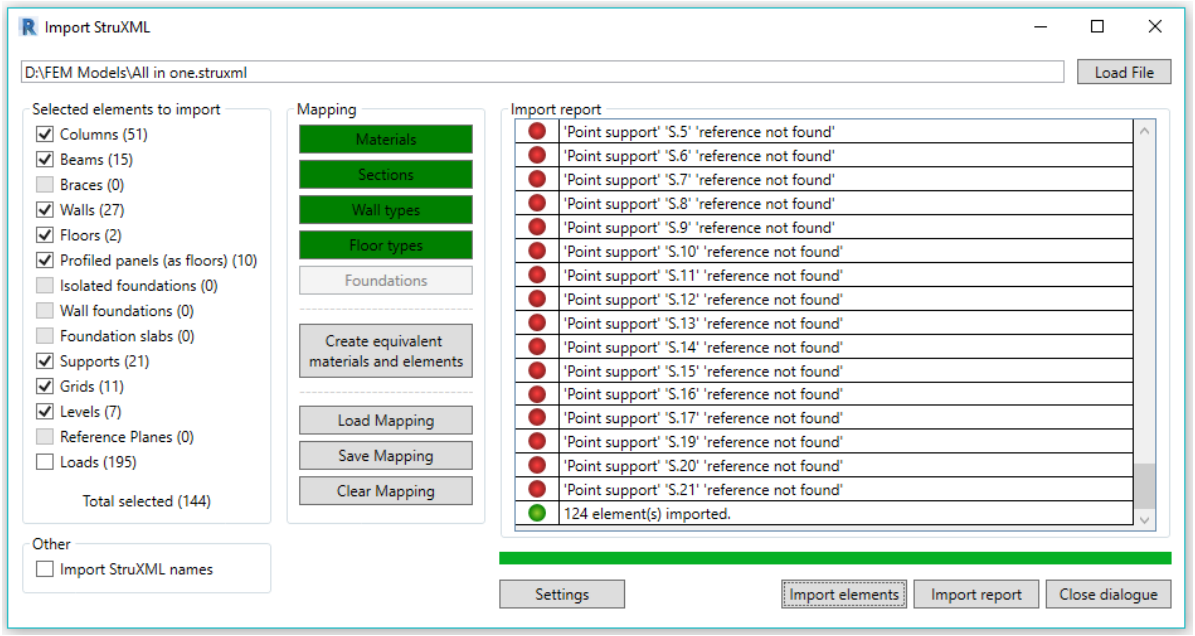



Figure IV-11

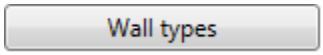
Import procedure may take up to several minutes in case of large models with many elements. Observe the green line to see the progress of the import (available only in Revit 2014).

## 12.4. Mapping

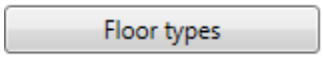
*Mapping* panel is divided into five different mapping buttons:


Here materials for standard component families: columns, beams, braces, isolated foundations should be mapped. Mapped material will be assigned to a *Structural material* parameter under condition that it is an Instance parameter.


Here sections for standard component families: columns, beams, braces) should be mapped.




Here wall types should be mapped. Remember that material of a wall is a Type parameter and is included in Wall type.




Here floor types and profiled panels should be mapped. Remember that material of a floor is a Type parameter and is included in Floor type.


Here all kind of foundations (isolated, wall and foundation slab) should be mapped. Remember that material of a wall foundation and foundation slab is a Type parameter and is included in wall foundation and foundation slab type.

Every mapping dialog consists of two columns:

- on the left hand side there is a column where all FEM-Design materials / sections / wall types / etc. used in the imported model are listed,

- on the right hand side there is a column where corresponding Revit material / section / wall type / etc. should be loaded. The Revit library appears upon clicking on the white field in a certain row.

  Be aware that the material / sections / wall and floor types / foundations libraries only contain materials / sections / wall types / etc. that existed, or have been loaded or created in the Revit project prior to importing the struxml file.

If a certain material / section / wall type / etc. is missing during the mapping process, one has to terminate the import process and create this certain material / section / wall type / etc. in the Revit project first. Since version 1.1.011 one can use Create equivalent sections button, to create concrete materials, sections and shells in Revit.

If you see the same material, section (of column, beam, truss, etc.) or type (of slab, wall, etc.) listed more than one time in the mapping dialog, it means that there are more elements that have the same materials, section or type, but they have different properties and therefore, have to be mapped separately.

### 12.4.1. Automatic mapping

This feature is introduced in version 1.1.011.

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 creates 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.

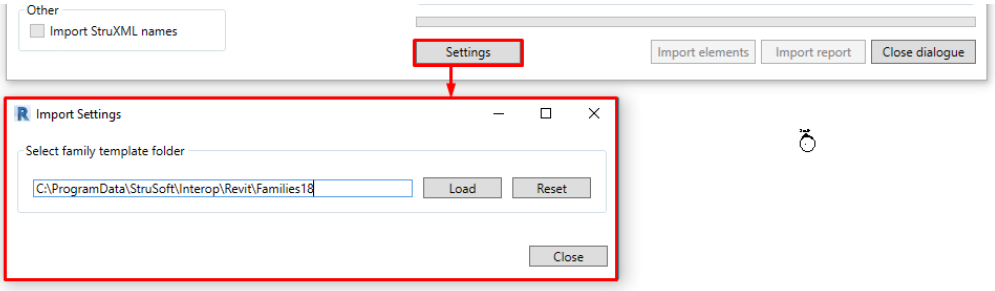


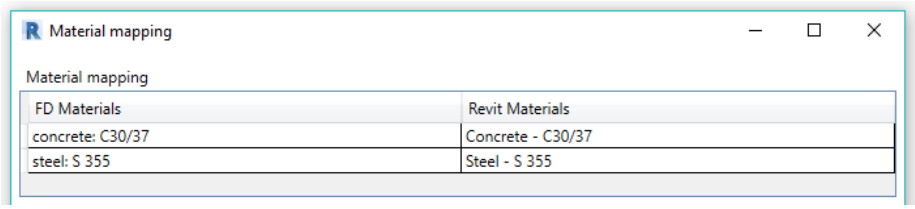

Figure IV-12

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:

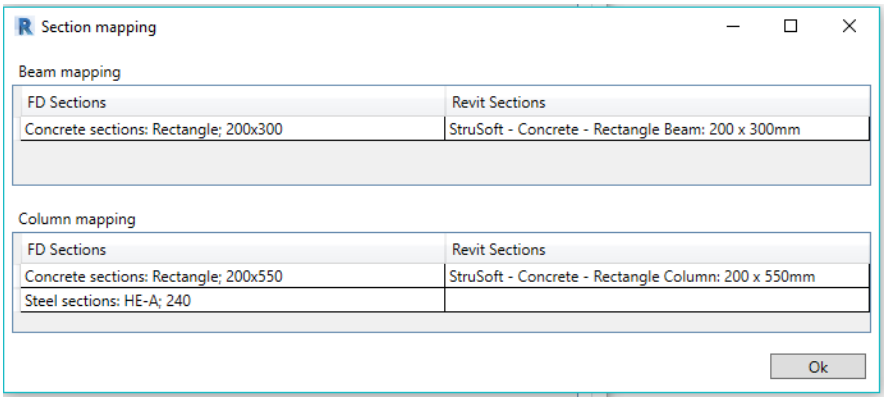
## Materials:



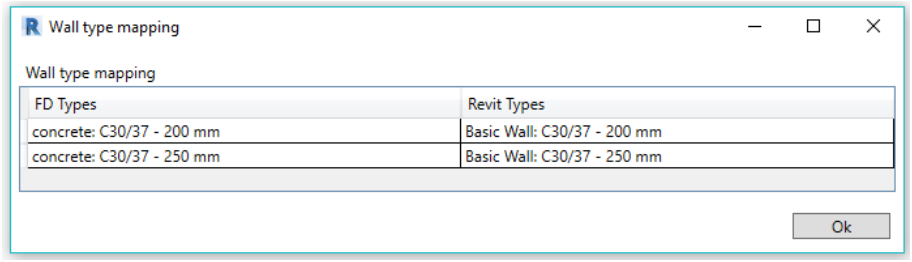

## Sections:

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 e created automatically and has to be mapped manually.

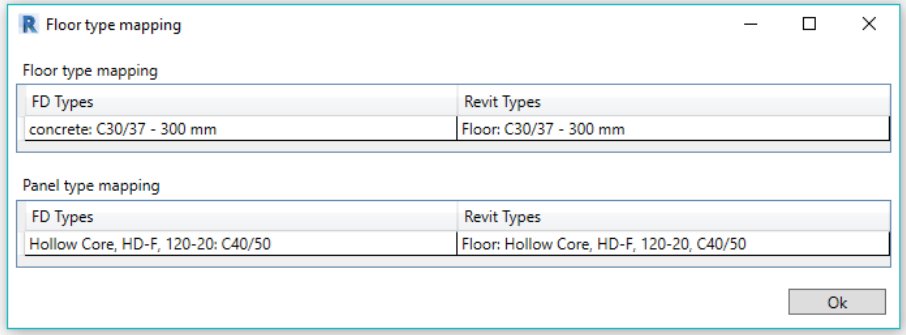
The same happens in case of any other section than those that are supported at the moment (Concrete rectangle beam, Concrete rectangle column, Concrete round column) e.g. steel section. These sections have to be mapped manually.




## Walls:



Floors:




### 12.4.2. Manual materials mapping

Upon pressing *Materials* button, *Materials mapping* dialog will pop out. Here, in the left column (*FD Materials*) one can see a list of all the materials that were assigned to columns, beams, truss members and isolated foundations in the FEM-Design model.

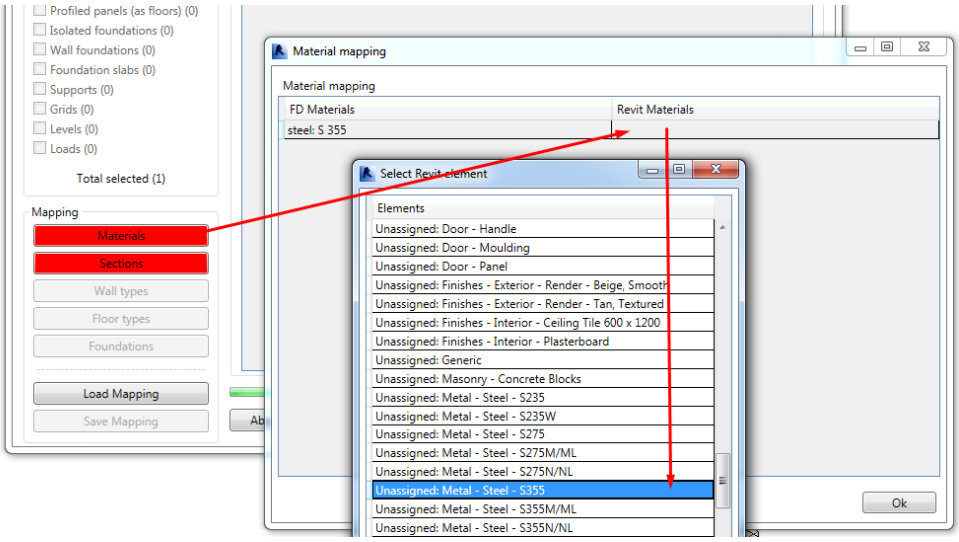



Figure IV-13

In order to map a selected material into a corresponding material from the Revit library, one has to click on the white field in the right hand side column (*Revit materials*). A *Select Revit element* dialog will pop out. It contains all materials from the Revit project material library that one can browse to find a matching material. Double click to select a material.

Example is shown in Figure IV-13.

It is also possible to perform multiple mapping, i.e. select the same material / section / type / etc. to a several elements imported from FEM-Design. Select several elements in the mapping dialog (with Ctrl button), right click and chose *Assign all selected,* as shown in Figure IV-14

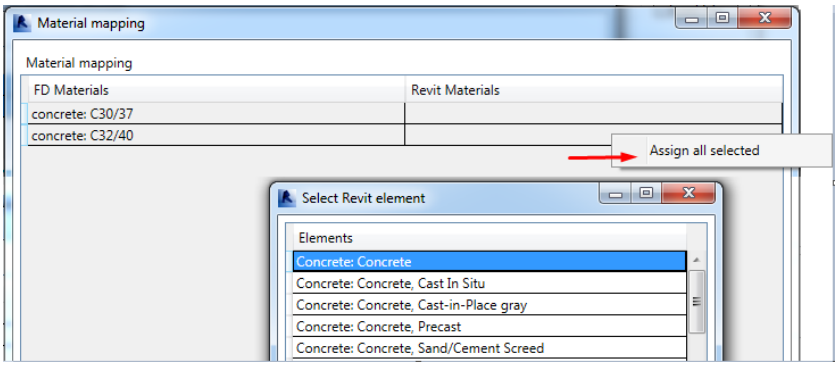



Figure IV-14

### 12.4.3. Manual Sections mapping

Upon pressing *Sections* button, *Sections mapping* dialog will pop out. Dialog is divided into two parts: *Beam mapping*, where beams and braces should be mapped and *Column mapping*, where columns should be mapped.

In the left column (*FD Materials*) one can see a list of all sections used FEM-Design model. In order to map a selected section into a corresponding section from the Revit library, one has to click on the white field in the right hand side column (*Revit Sections*). A *Select Revit element* dialog will pop out. It contains all the beam (or column) sections included in the Revit project that one can browse to find a matching section.

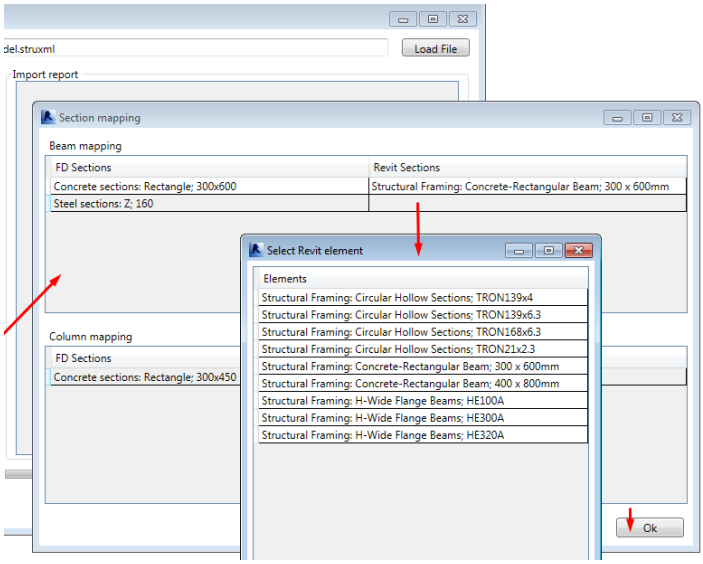



Figure IV-15

### 12.4.4. Manual Wall types mapping

Upon pressing *Wall types* button, *Wall types mapping* dialog will pop out.

In the left column (*FD Types*) one can see a list of all wall types used FEM-Design model. In order to map a selected wall into a corresponding wall type from the Revit library, one has to click on the

white field in the right hand side column (*Revit Types*). A *Select Revit element* dialog will pop out. It contains all the wall types included in the Revit project that one can browse to find a matching one.

Example is shown in Figure IV-16.

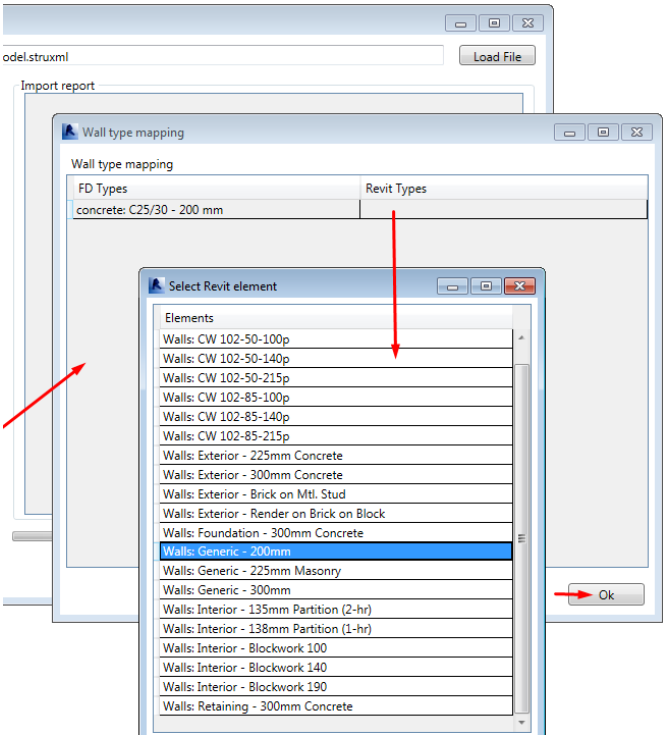



Figure IV-16

*Structural Material* property of a wall is a type parameter that is included in the wall type that one has to choose in the mapping process. Therefore, make sure that a wall type that you chose from the Revit library has a proper material assigned.
An example can be the *concrete C25/30 – 200 mm* wall that was created in FEM-and was mapped into *Wall: Generic – 200mm* (Figure IV-16). This particular wall has a certain material assigned to it, what can be seen when checking its type properties (Figure IV-17).

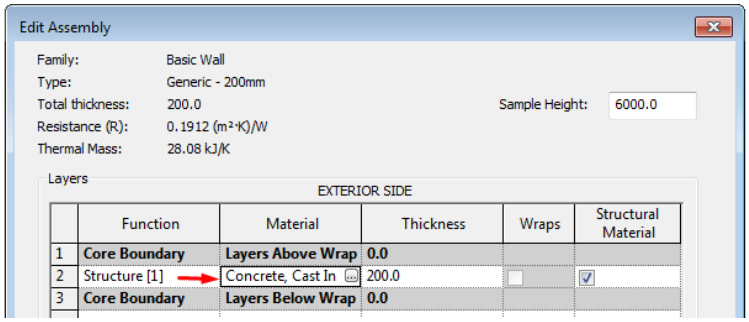



Figure IV-17

### 12.4.5. Manual Floor types mapping

Floor types mapping is equivalent to the Wall types mapping procedure. In this dialog profiled panels can also be mapped into corresponding floor types.

*Structural Material* property of a floor is a type parameter that is included in the floor type that one has to choose in the mapping process. Therefore, make sure that a floor type that you chose from the Revit library has a proper material assigned.

### 12.4.6. Foundations mapping

Upon pressing Foundations button, Foundation mapping dialog will pop out. Dialog is divided into three parts: Isolated foundations mapping, Wall foundation mapping, and Foundations slab mapping.

*Structural Material* property of a *Wall foundation* and *Foundation slab* is a type parameter that is included in the wall foundation and foundation slab type that one has to choose in the mapping process. Therefore, make sure that a floor type that you chose from the Revit library has a proper material assigned.

## 12.5. Warnings and errors

There is couple of situations in which import of certain elements will not be possible, and a warning or error message will be thrown in the *Import report* dialog:

Most of the error occurs because of the different approach to certain situation between Revit and FEM-Design. Most of those situations are related to supporting structures and their hosts. Revit simply do not allow for situations that are allowed in FEM-Design, for example to attach a point support to a wall or floor element. Below, all recognized situations are listed:

1. If a support (point support, line support, and surface support) in FEM-Design is attached to an element that is not allowed by Revit, or is not attached to any element, an error message will be thrown in a format:

"Point support / Line support / Surface support", "support name and ID from FEM-Design", "reference not found"

2. The next situation when an error will be thrown, is an import of a wall foundation without a host wall, or attached to other element than a wall – situation like this is not possible in Revit and wall foundation can only be attached to a wall element. An error message will be thrown in a format:

"Wall foundation", "wall foundation name and ID from FEM-Design", "wall not found"

3. Sometimes, an opening in a wall cannot be recreated while importing the wall into Revit and an error message will be thrown in a format:

"Wall", "wall's name and ID from FEM-Design", "Revit function failed to create an opening for a certain wall"