

Revit to FEM-Design

Using Revit StruXML 1.0.004 Add-In

Manual: Getting started using the Revit to FEM-Design connection.

Iwona Budny

iwona.budny@strusoft.com

18/12/2013



TABLE OF CONTENTS

1. INTRODUCTION.....	3
2. INSTALLATION	3
3. EXPORTED ELEMENTS AND FEATURES.....	4
3.1. STRUCTURAL COLUMNS AND FRAMING ELEMENTS.....	4
3.2. STRUCTURAL FLOORS.....	4
3.3. STRUCTURAL WALLS.....	5
3.4. STRUCTURAL FOUNDATIONS	6
3.5. MATERIAL	8
3.6. ECCENTRICITY - ANALYTICAL ALIGNMENT	9
3.7. RELEASES OF LINEAR ELEMENTS	11
3.8. BOUNDARY CONDITIONS	11
3.9. GRIDS	12
3.10. LEVELS	13
3.11. LOADS AND LOAD CASES.....	14
4. EXPORT FROM REVIT TO FEM-DESIGN	18
4.1. EXPORT.....	19
4.2. MATERIALS MAPPING	21
4.3. SECTIONS MAPPING	24
4.4. WARNINGS AND ERRORS	26
4.5. EXPORT REPORT	31
5. OPEN STRUXML IN FEM-DESIGN.....	34
6. UPDATE FEM-DESIGN MODEL.....	34
6.1. PROCEDURE.....	35
6.2. LIMITATIONS	36
6.3. EXAMPLE.....	36

1. INTRODUCTION

In May 2012 a new Revit to FEM-Design Add-in that allows exporting a Revit analytical model into a struXML file has been developed. Main new features in comparison to the previous Add-In that was based on the .r2f file are:

- Export of grid lines and levels.
- Export of curved walls.
- Export of eccentricity settings in case of floor slabs and walls.
- Export of structural foundations.
- Export of loads and load cases.
- New mapping dialog with the option to load FEM-Design Code sensitive libraries.
- Better system of warning and error messages.
- Export report.

2. INSTALLATION

StruSoft StruXml Revit 2013 Add-In 1.0.003 Win32 (or Win64) and *StruSoft StruXml Revit 2014 Add-In 1.0.003 Win32 (or Win64)* can be found in the *StruSoft Installer* (select *FEM-Design* and go to *Info* tab) or at the download page:

<http://download.strusoft.com/FEM-Design/inst130x/>

Run the installation file. After completion of the installation process, *StruSoft FEM-Design* panel will appear in the *Analyze* tab in Revit along with the *Export StruXml* command as shown in Figure 2-1.

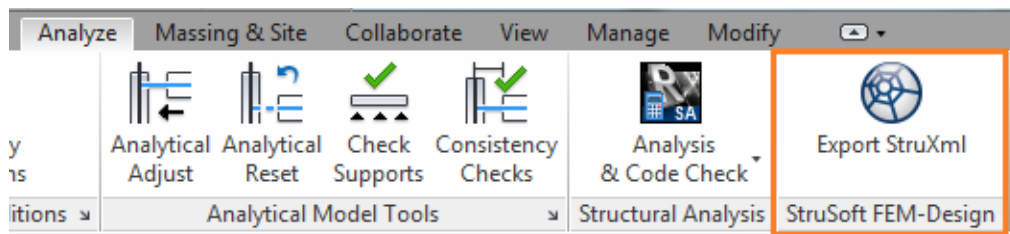


Figure 2-1

If a new version of the Add-In has been released, upon pressing the *Export StruXml* command, a *New Version* dialog will appear, as shown in Figure 2-2.

Press [Go to download page!](#) if you want to be redirected to the StruSoft download page.

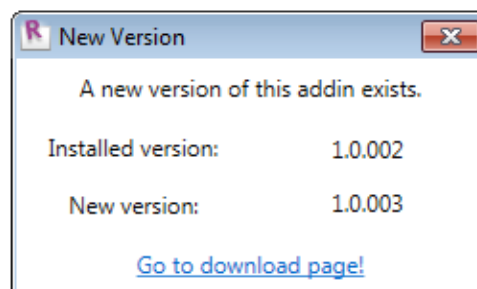


Figure 2-2

3. EXPORTED ELEMENTS AND FEATURES

Following elements and properties can be exported from Revit to FEM-Design using the struXML file based Add-In:

Elements:

- Structural Columns
- Structural Framing elements (Beams, Beam systems, Braces, Trusses)
- Structural Floors
- Structural Walls
- Structural Foundation
- Grids and levels
- Loads and load cases

Properties:

- Material of an element
- Section / Profile / Thickness of an element
- Releases of linear elements
- Boundary conditions
- Eccentricity in case of floor slabs and walls

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, it will not be exported to FEM-Design.

3.1. STRUCTURAL COLUMNS AND FRAMING ELEMENTS

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
- Material¹
- Release conditions

3.2. STRUCTURAL FLOORS

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 as shown in Figure 3-1. If more than one floor element is created within one floor boundary (as shown in Figure 3-2), those elements will be exported as one floor instance that is equivalent into one plate element in FEM-Design.

¹ Only the Structural material is exported. Read more in Chapter 3.5.

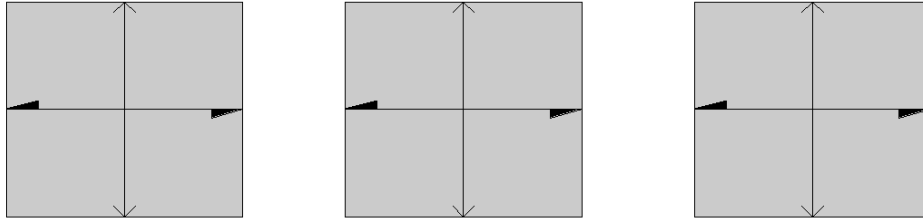


Figure 3-1

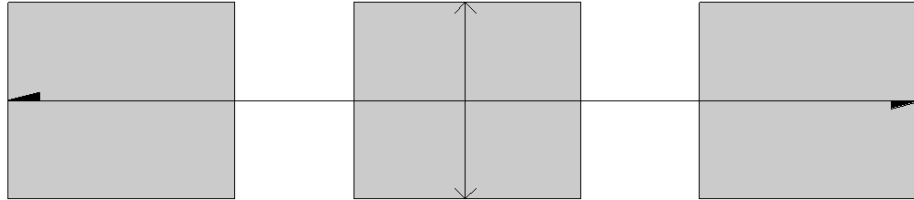


Figure 3-2

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
- Material¹
- Eccentricity²

3.3. 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
- Material¹
- Eccentricity²

Curved walls can now be exported from Revit to FEM-Design. Each curved wall drawn in Revit (Figure 3-3) will be divided into a number of straight walls while being exported to struXML file, as shown in Figure 3-4. This is the way curved walls are handled in FEM-Design.

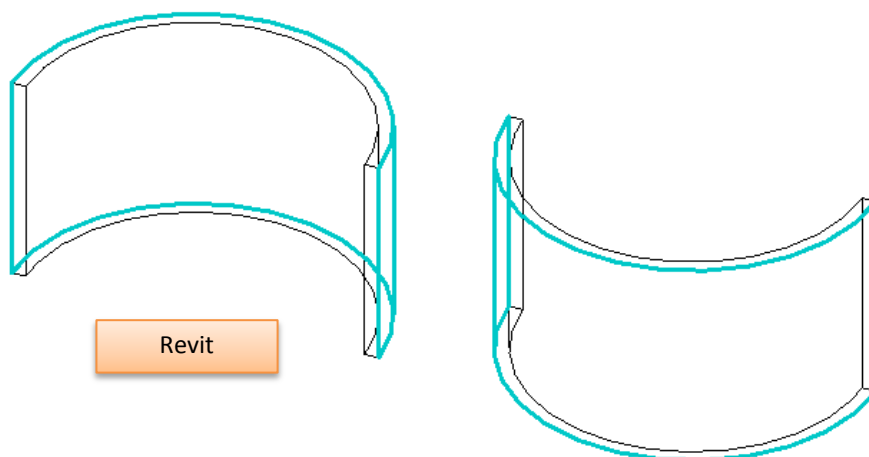


Figure 3-3

² Read more about the rules of eccentricity export in Chapter 3.6.

Figure 3-4 shows the convention of the local coordinate system that curved walls will be assigned with in FEM-Design.

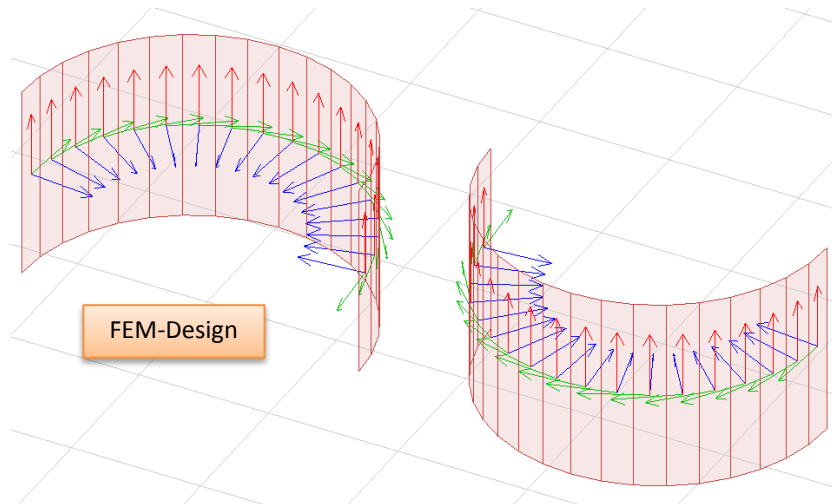


Figure 3-4

3.4. 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 3-5.

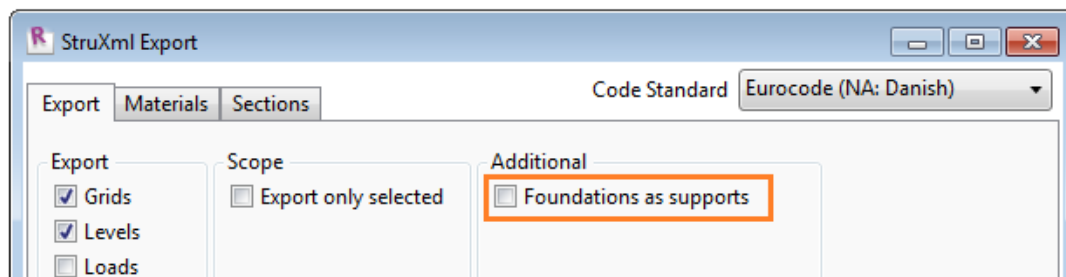


Figure 3-5



Structural Foundations can be exported as foundation elements only to FEM-Design in version higher than 12.00.003

3.4.1. 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 3-6.

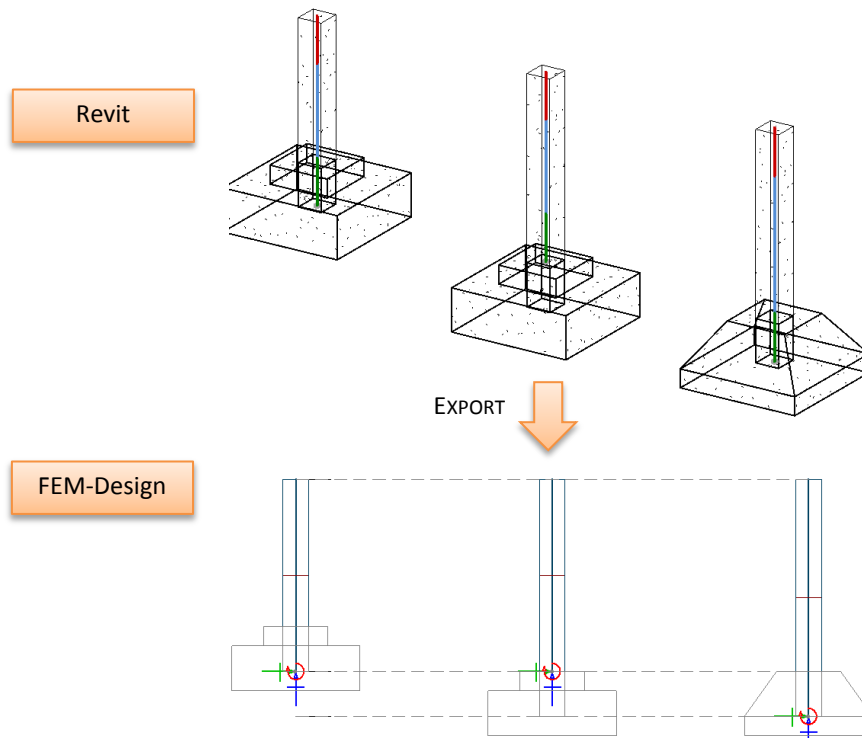


Figure 3-6

However, 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*.

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.

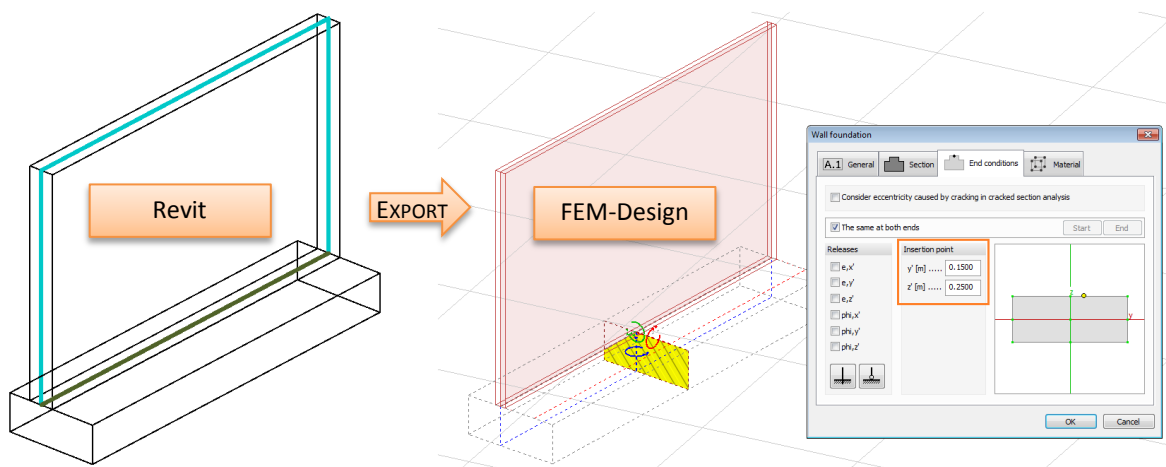


Figure 3-7

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 3.6.

3.4.2. 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*.

3.5. 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.

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 3-8, the wall element would be exported from Revit to FEM-Design as a 190 cm thick element with the material: *Masonry – Concrete Block*.

Family:	Basic Wall				
Type:	Exterior - Render on Brick on Block				
Total thickness:	429.0		Sample Height: 6000.0		
Resistance (R):	0.0000 (m ² ·K)/W				
Thermal Mass:	0.00 kJ/K				
Layers					
EXTERIOR SIDE					
	Function	Material	Thickness	Wraps	Structural Material
1	Finish 1 [4]	Render - Tan, Textured	25.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>
2	Finish 1 [4]	Render - Beige, Smooth	25.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>
3	Finish 1 [4]	Masonry - Brick	102.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>
4	Thermal/Air La	Misc. Air Layers - Air S	50.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>
5	Thermal/Air La	Insulation / Thermal B	50.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>
6	Membrane Lay	Vapour / Moisture Barr	0.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>
7	Core Boundary	Layers Above Wrap	0.0	<input type="checkbox"/>	<input type="checkbox"/>
8	Structure [1]	Masonry - Concrete Bl	190.0	<input type="checkbox"/>	<input checked="" type="checkbox"/>
9	Core Boundary	Layers Below Wrap	0.0	<input type="checkbox"/>	<input type="checkbox"/>
10	Finish 2 [5]	Plasterboard	12.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Figure 3-8



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. Furthermore, *Structural material* has to be an Instance parameter in case of Beams, Columns, Isolated foundations and a Type parameter in case of Floors, Walls, Wall foundations and Foundation Slabs.

3.6. ECCENTRICITY - ANALYTICAL ALIGNMENT

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*.

3.6.1. ANALYTICAL FLOOR / ANALYTICAL FOUNDATION SLAB

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

Alignment Method: Projection

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 3-1 and in Figure 3-9.

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

Table 3-1

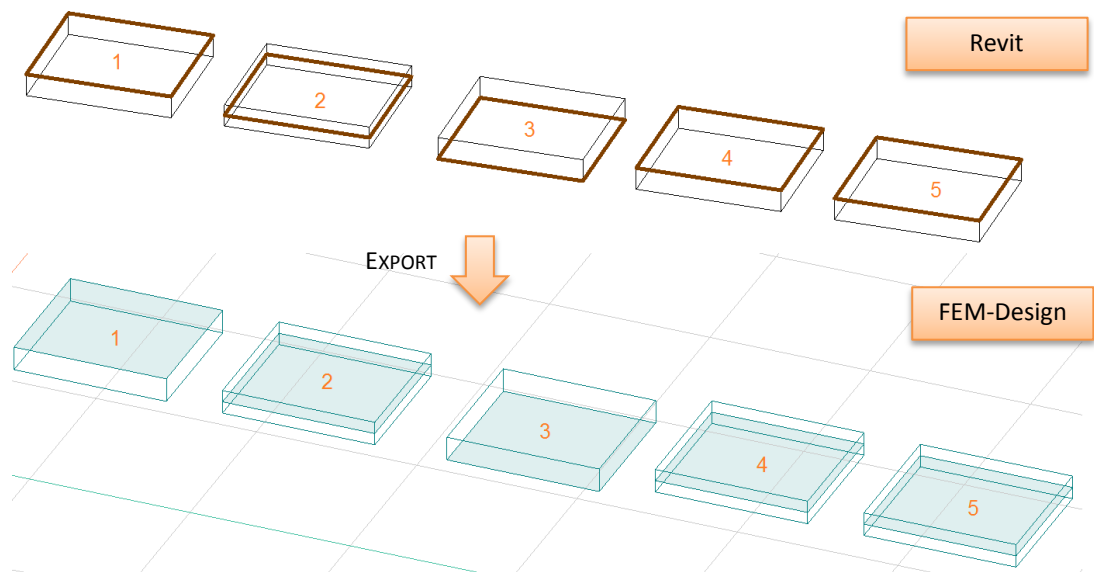


Figure 3-9

3.6.2. 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 3-2 and in Figure 3-10.

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

Table 3-2

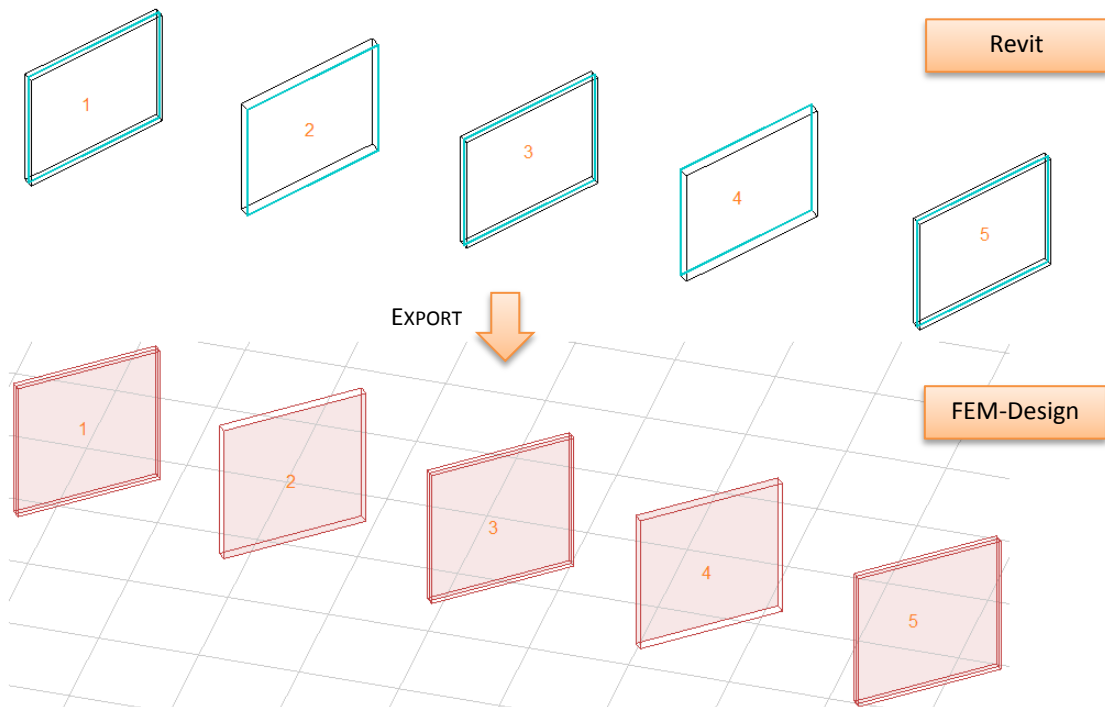


Figure 3-10

3.7. 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 3-11 shows release conditions originally set in Revit and the result of the export to FEM-Design.

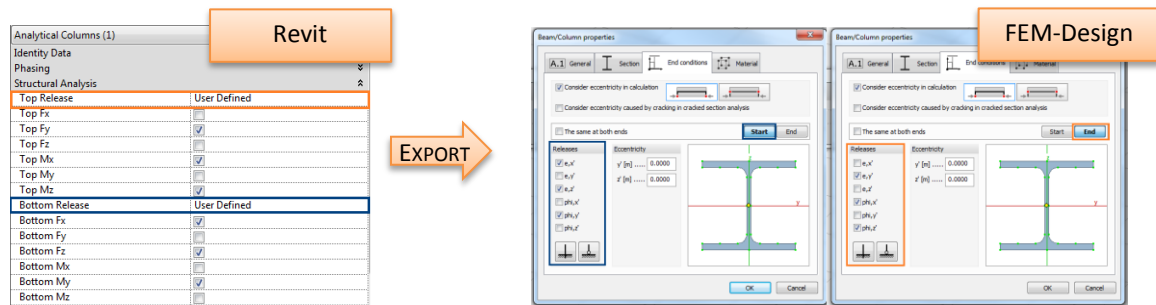


Figure 3-11

3.8. 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 3-12.

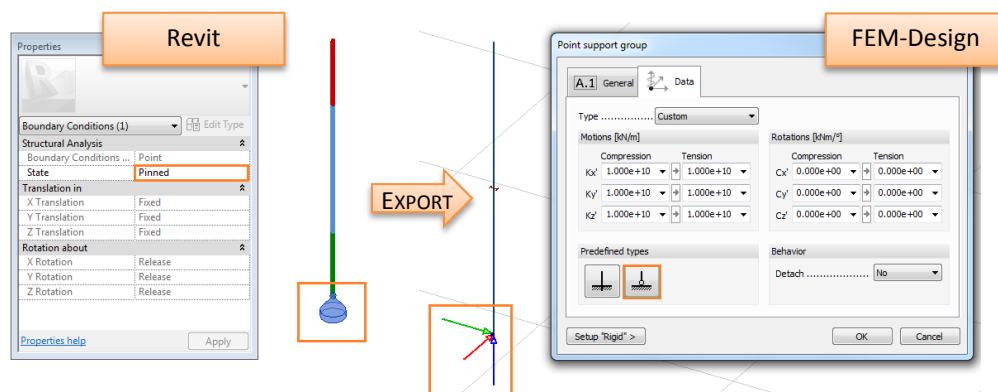


Figure 3-12

The “Rigid” value exported from Revit to FEM-Design is equal to: $1\text{e}+15$ kN/m for motion, and $3.046\text{e}+11$ kNm/° 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.

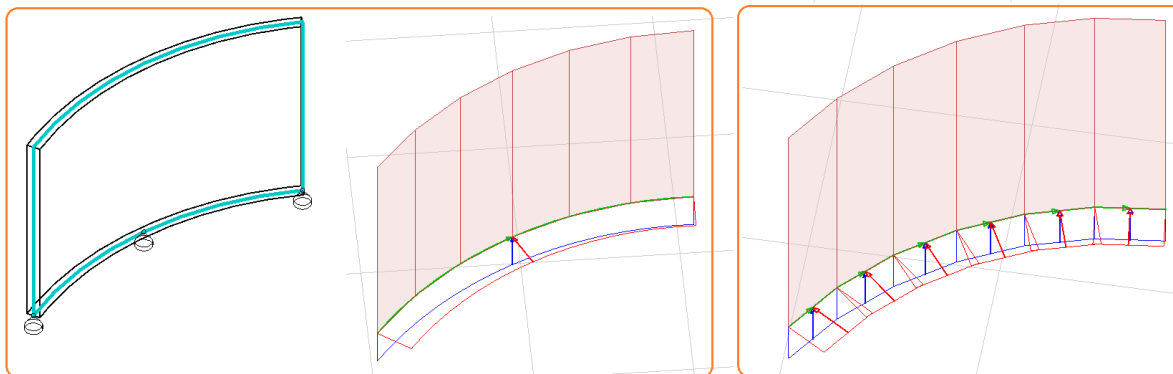


Figure 3-11

3.9. 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 3-16) 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 3-13. If necessary, renaming can be performed in FEM-Design.

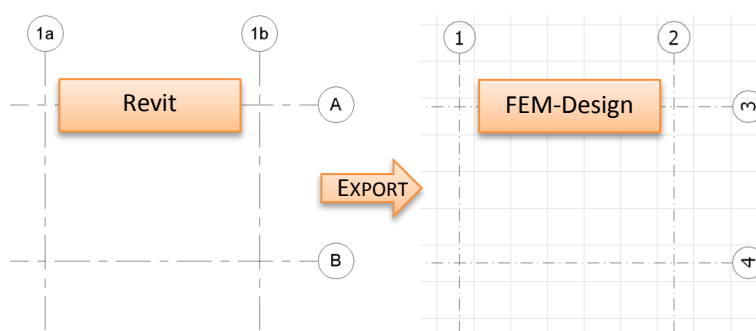


Figure 3-13

- 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 3-14.

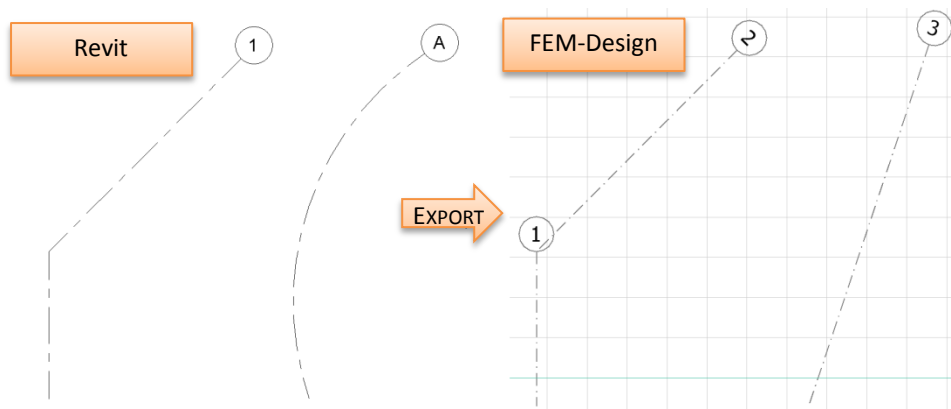


Figure 3-14

3.10. 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 3-16), 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 3-15.
- *Levels* cannot be exported if no elements are created in the Revit model because size of a storey cannot be generated.

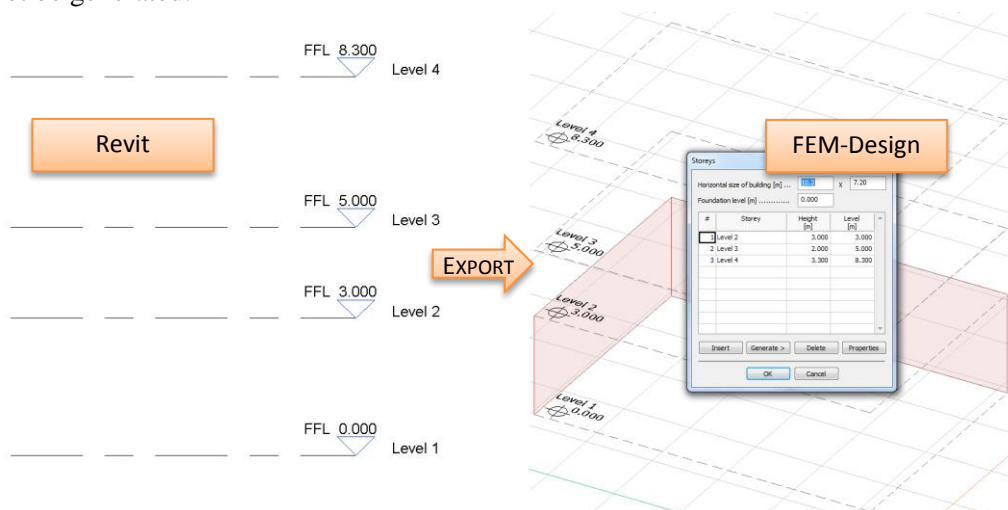


Figure 3-15

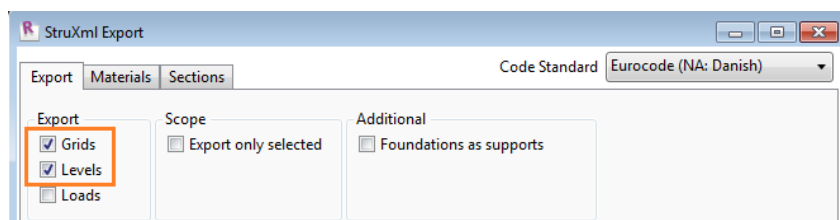


Figure 3-16

3.11. LOADS AND LOAD CASES

Loads and *Load Cases* can be exported from Revit to FEM-Design.

In order to export loads and load cases one should select the *Loads* to be exported in the *StruXml Export* dialog, as shown in figure below:

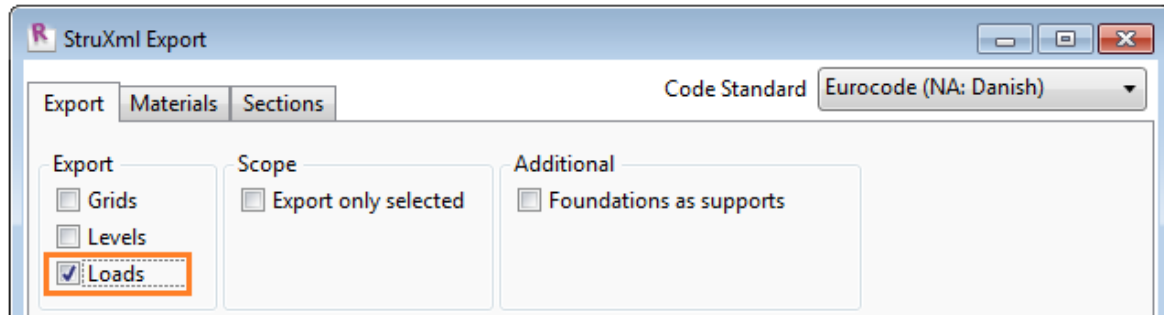


Figure 3-17

3.1.1 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.

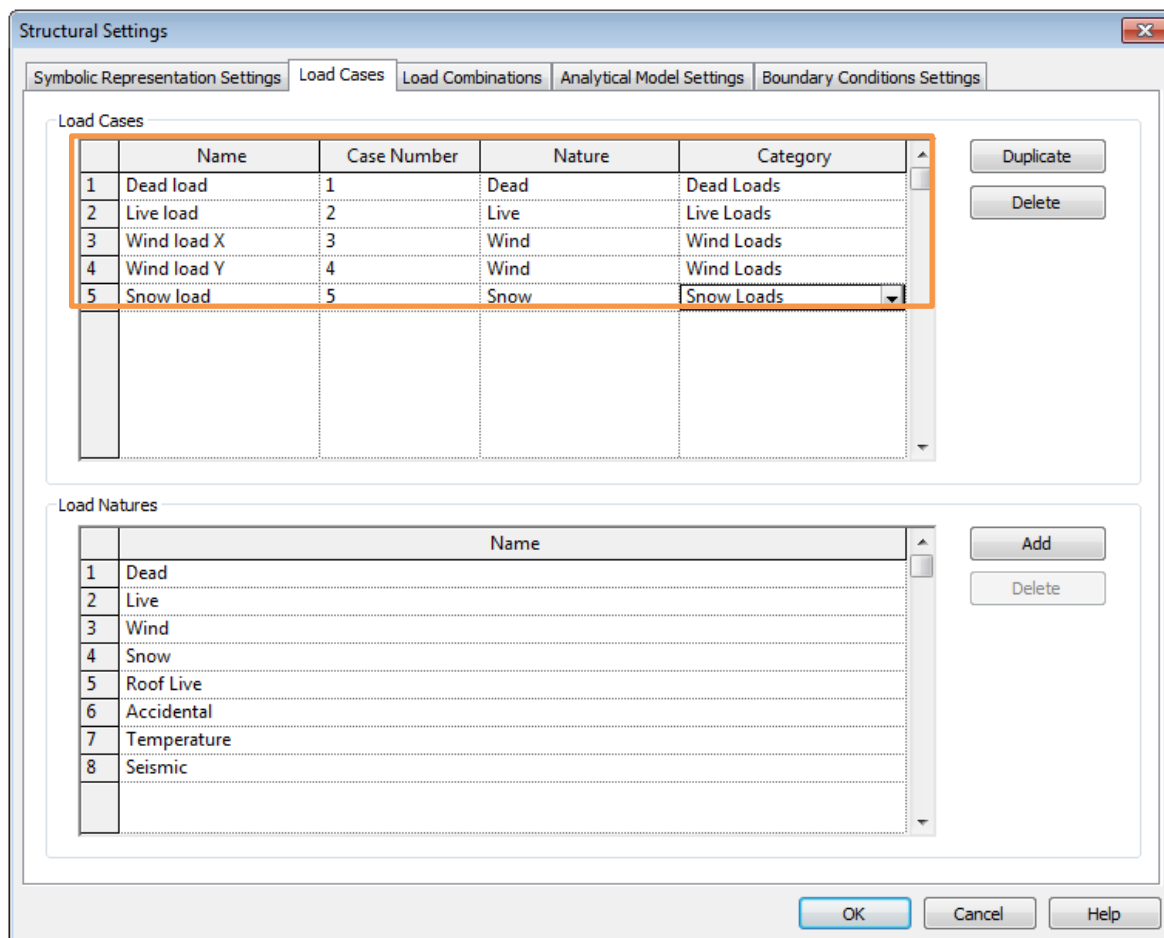


Figure 3-18

Each of the *Load Cases* exported from Revit, in FEM-Design will be assigned with the *Type: Ordinary* that is suitable to any kind of load nature.

Example of loads cases defined in Revit (Figure 3-18) and exported to FEM-Design is shown in Figure 3-19.

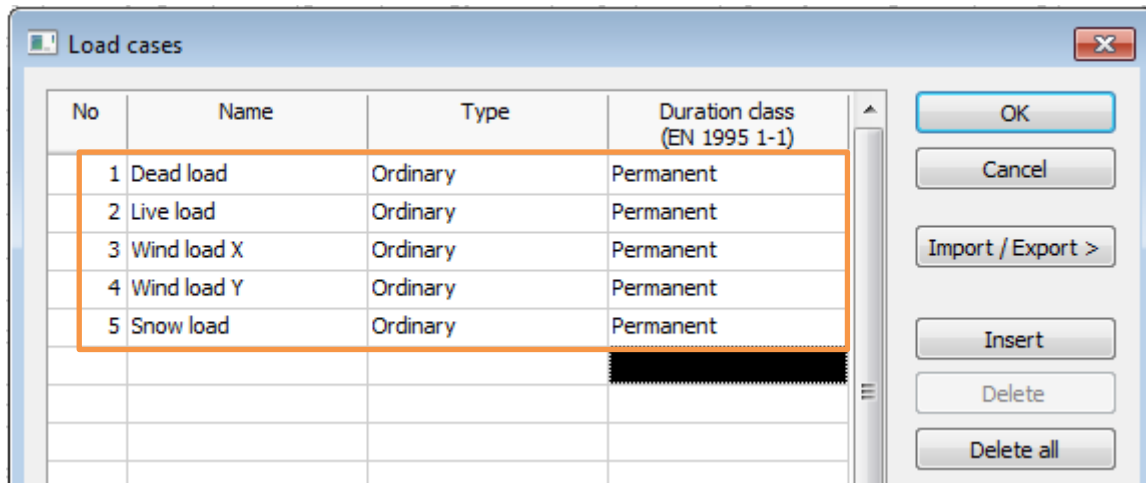


Figure 3-19

It is user's responsibility to decide which of the cases will additionally contain self-weight of structural elements. It can be done by changing the load case *Type* from *Ordinary* to *+Struc. Dead load*.

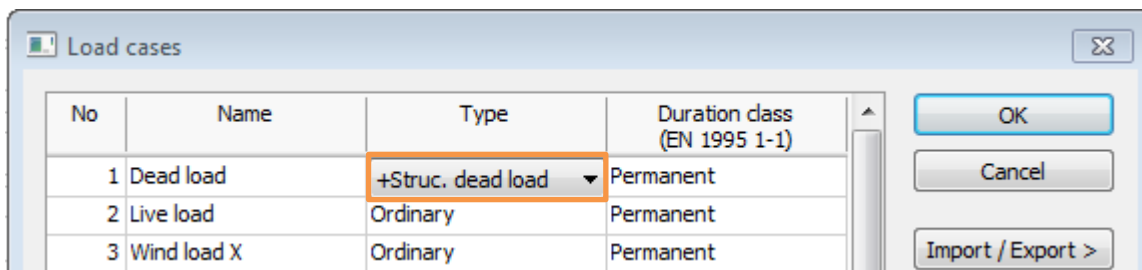


Figure 3-20



A Revit model that contains only load cases and no structural elements cannot be exported to FEM-Design.

3.1.2 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 3-22 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*.

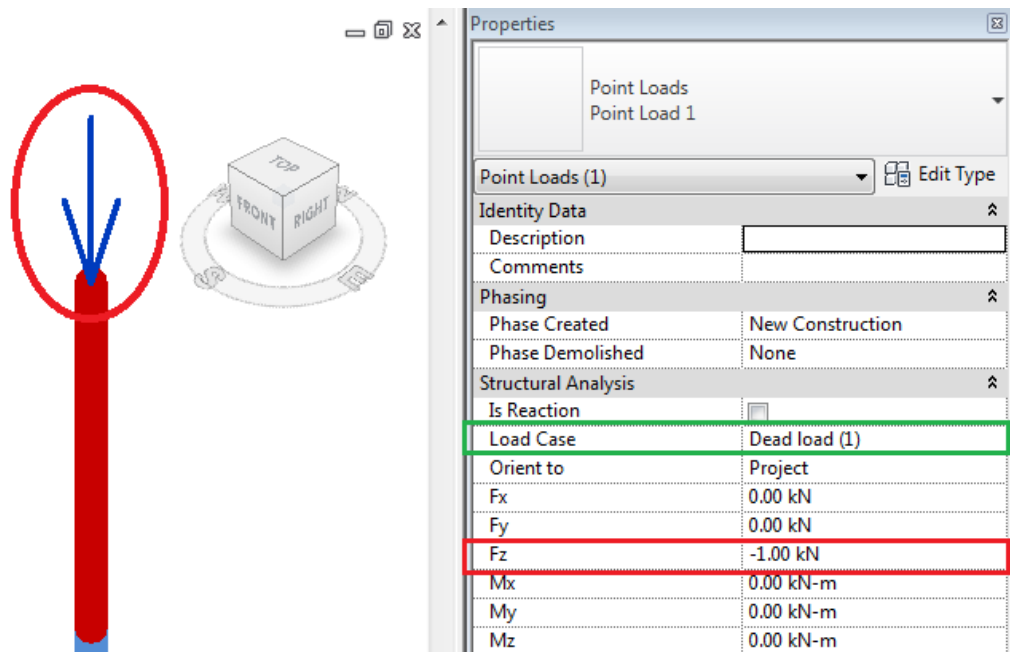


Figure 3-21

Figure 3-22 shows how the model looks like after exporting it to FEM-Design.

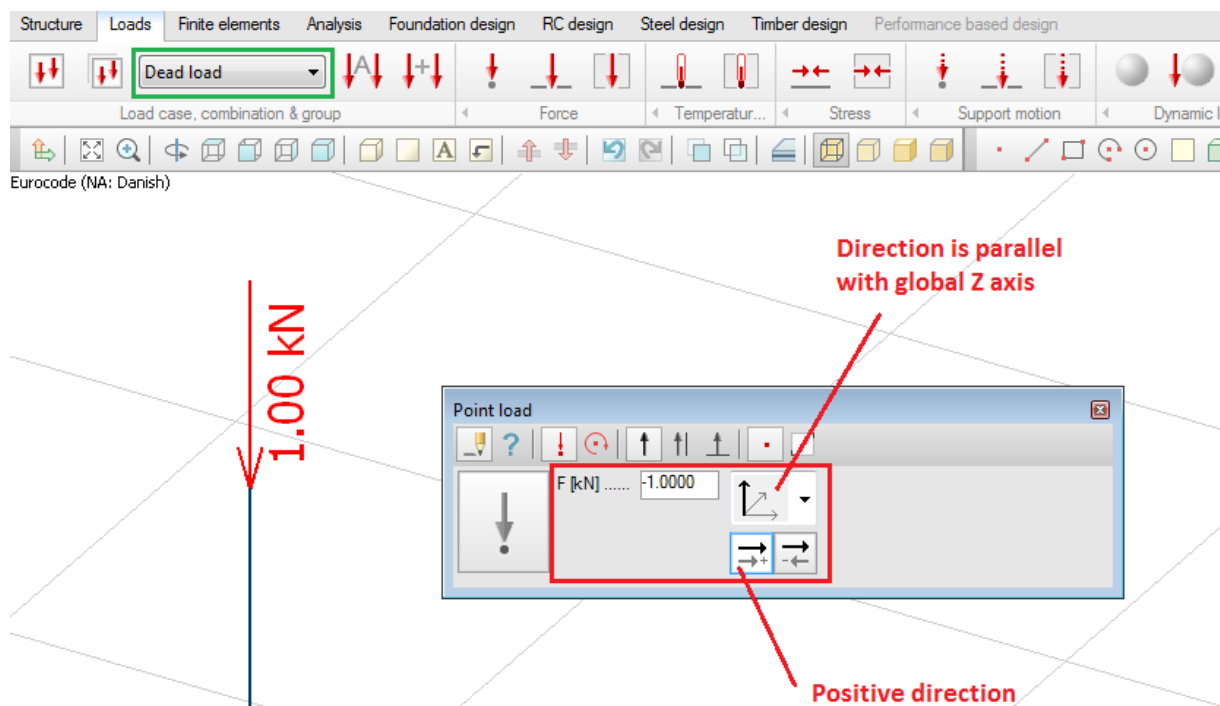


Figure 3-22

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 3-23.

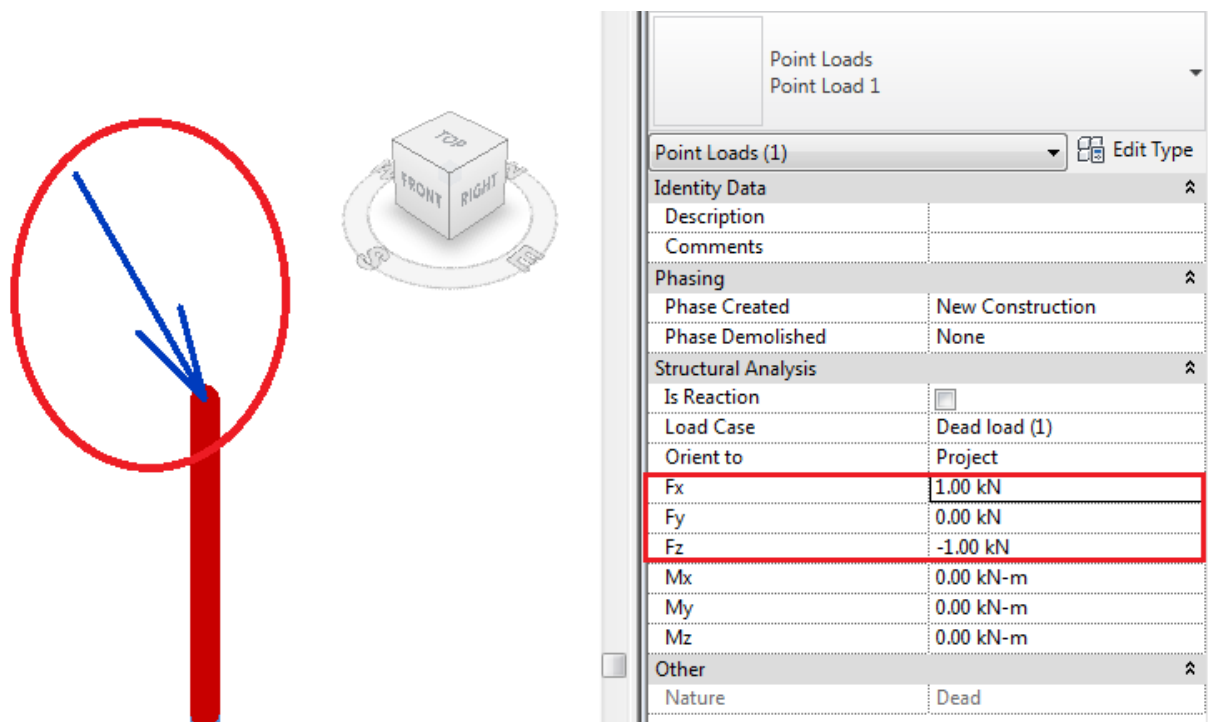


Figure 3-23

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 3-24.

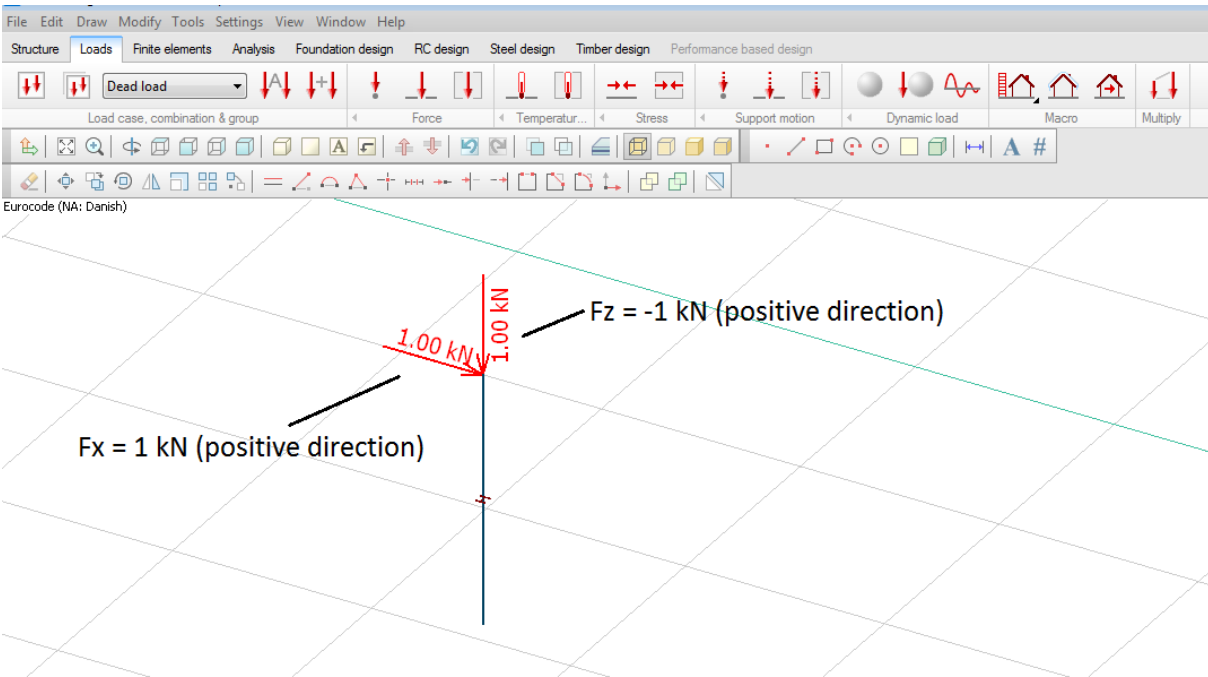


Figure 3-24

4. EXPORT FROM REVIT TO FEM-DESIGN

Before trying to export the 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,
- 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 *StruSoft FEM-Design* panel in the *Analyze* tab in Revit.

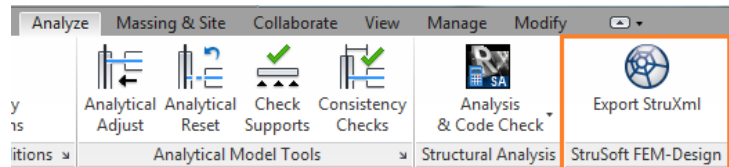


Figure 4-1

Upon choosing the *Export StruXML* command, a *Code* dialog shown in Figure 4-2 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.

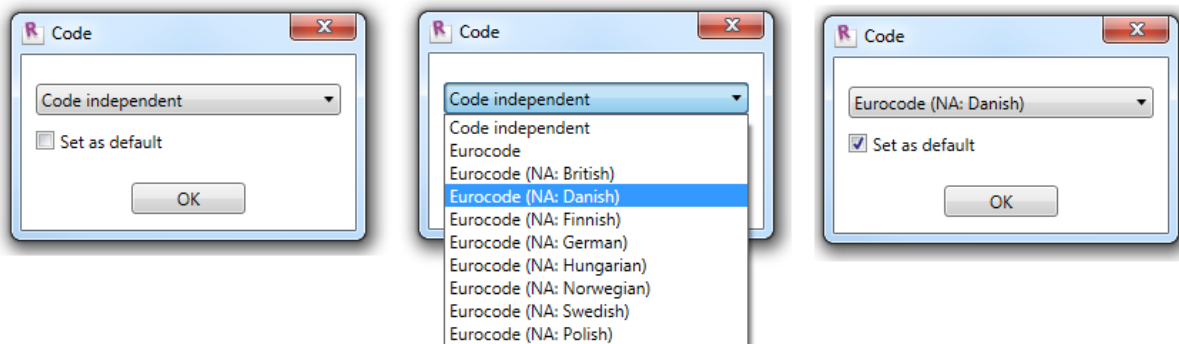


Figure 4-2

Upon pressing *OK* in the *Code* dialog, a *StruXML Export* dialog will appear, as shown in Figure 4-3.

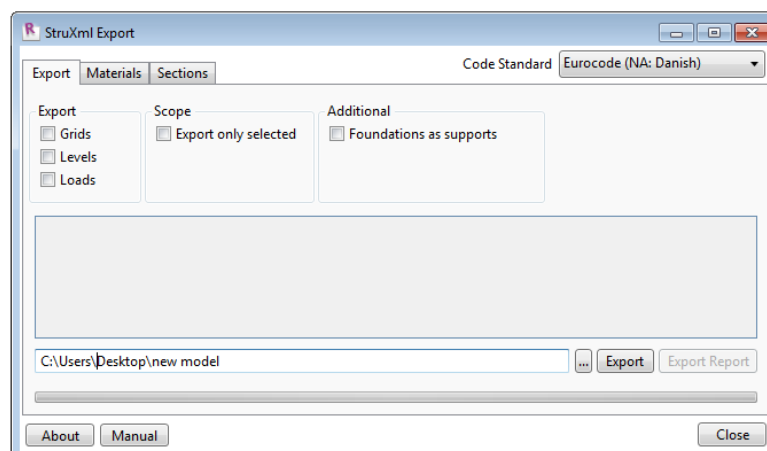


Figure 4-3

4.1. EXPORT

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

- change the previously chosen *Code Standard*,
- decide if to export *Grids*, *Levels* and *Loads*,
- decide if to export only selected elements (check *Export only selected*)
- decide if foundation elements should be exported as supports (check *Foundation as Supports*),
- 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* (becomes active when the export is done).

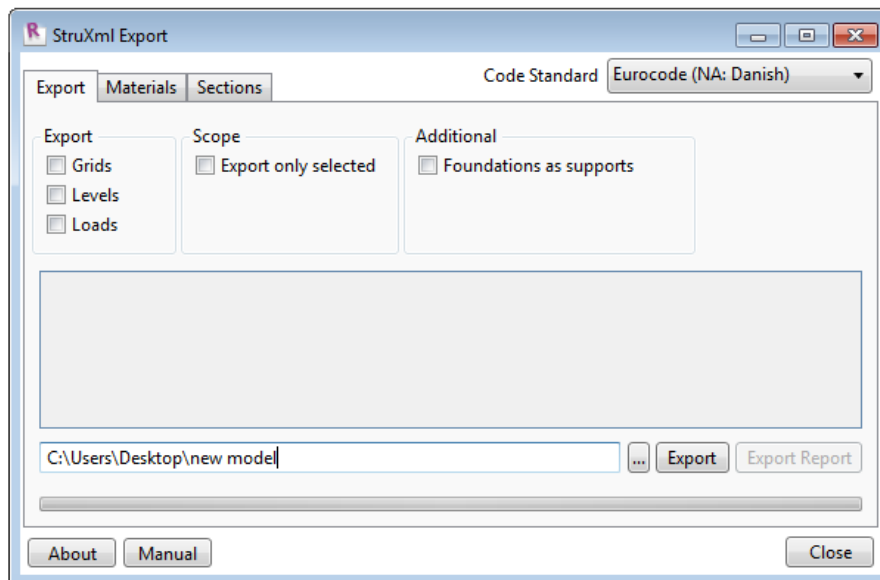


Figure 4-4

Workflow 1: After launching StruXML Export dialog there are no warnings about unmapped elements, as shown in Figure 4-4.

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 4-5 and the Export Report will become active.



Go to Paragraph 4.4 in order to read about meaning of possible warnings and errors. Go to Paragraph 4.5 in order to read more about the *Export report*.

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

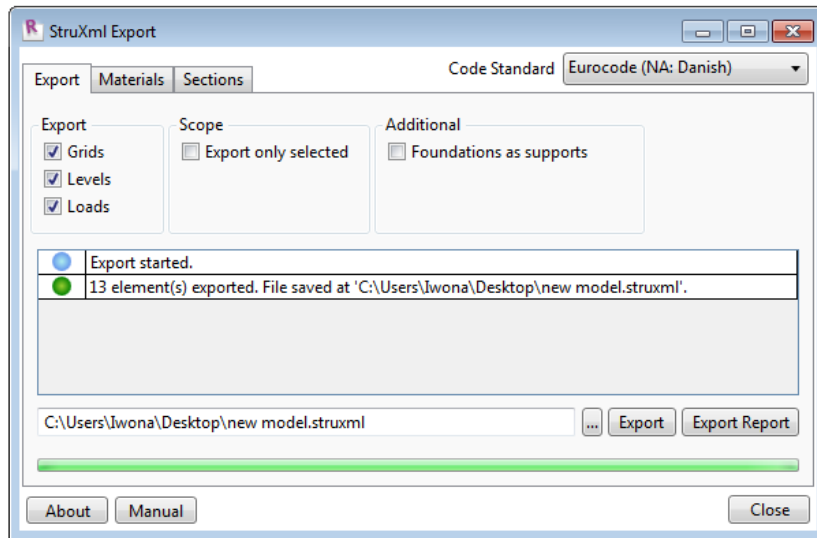


Figure 4-5

Workflow 2: After launching StruXML Export dialog there is a list of warnings about unmapped elements, as shown in Figure 4-6.

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.

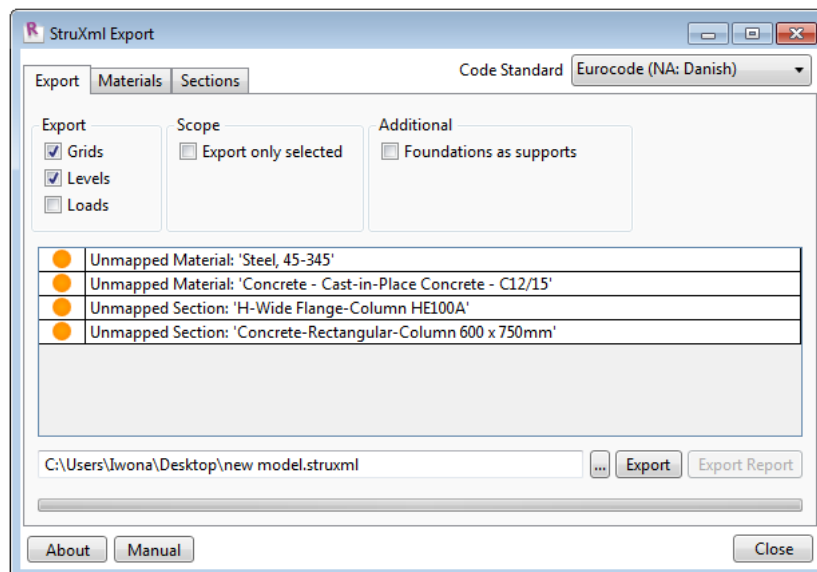


Figure 4-6

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 0).
2. Go to *Sections* tab and map the listed sections and profiles (read more about sections mapping in Paragraph 4.3).
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.

4.2. MATERIALS MAPPING

In order to map the materials included in the Revit model one should move to *Materials* tab, shown Figure 4-7. The dialog is divided into two parts: *Revit* materials in the left part and *FEM-Design* materials in the right part.

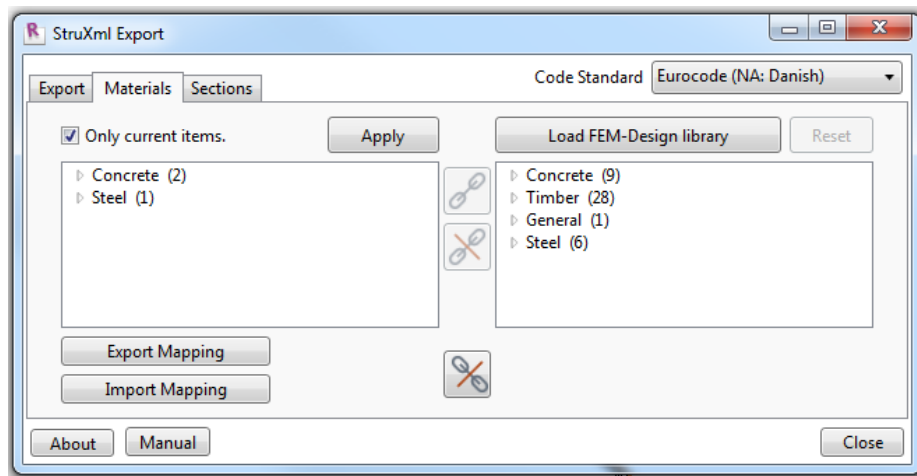


Figure 4-7

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

5. 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.
6. If you wish to unmap all of the previously mapped materials, do not select any material – just press the *Unmap All Mapped Items* button.

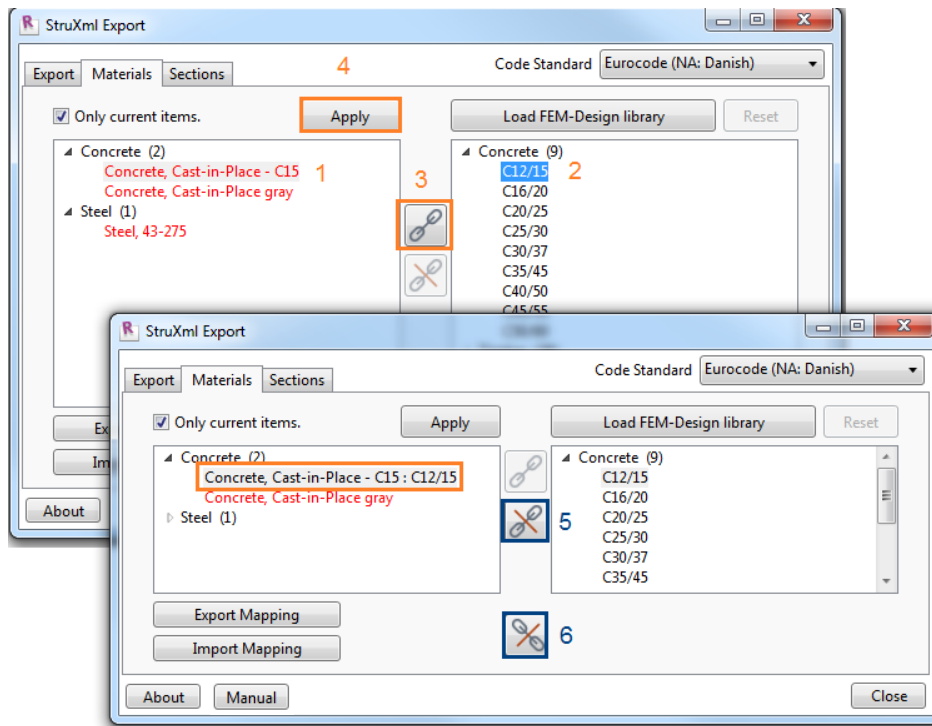


Figure 4-8

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 4-9 and Figure 4-10):

1. In FEM-Design go to the *Default settings* of e.g. *Column*.
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.

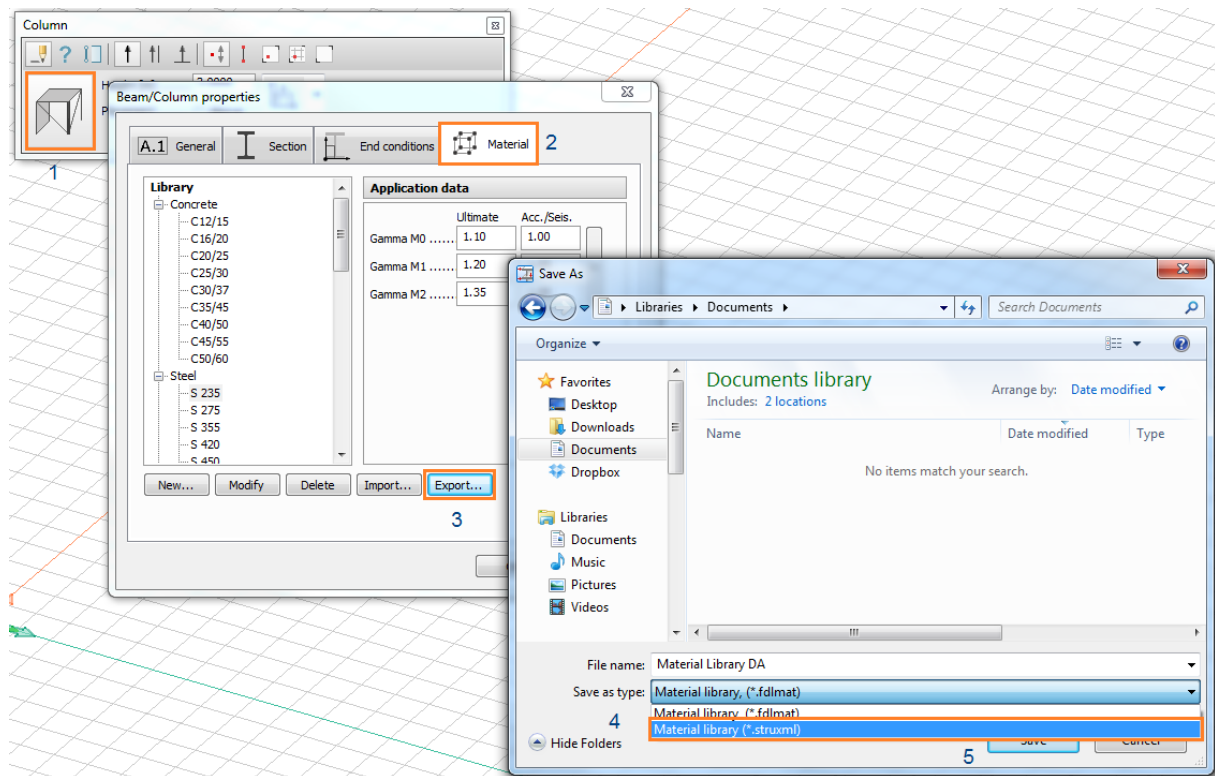


Figure 4-9

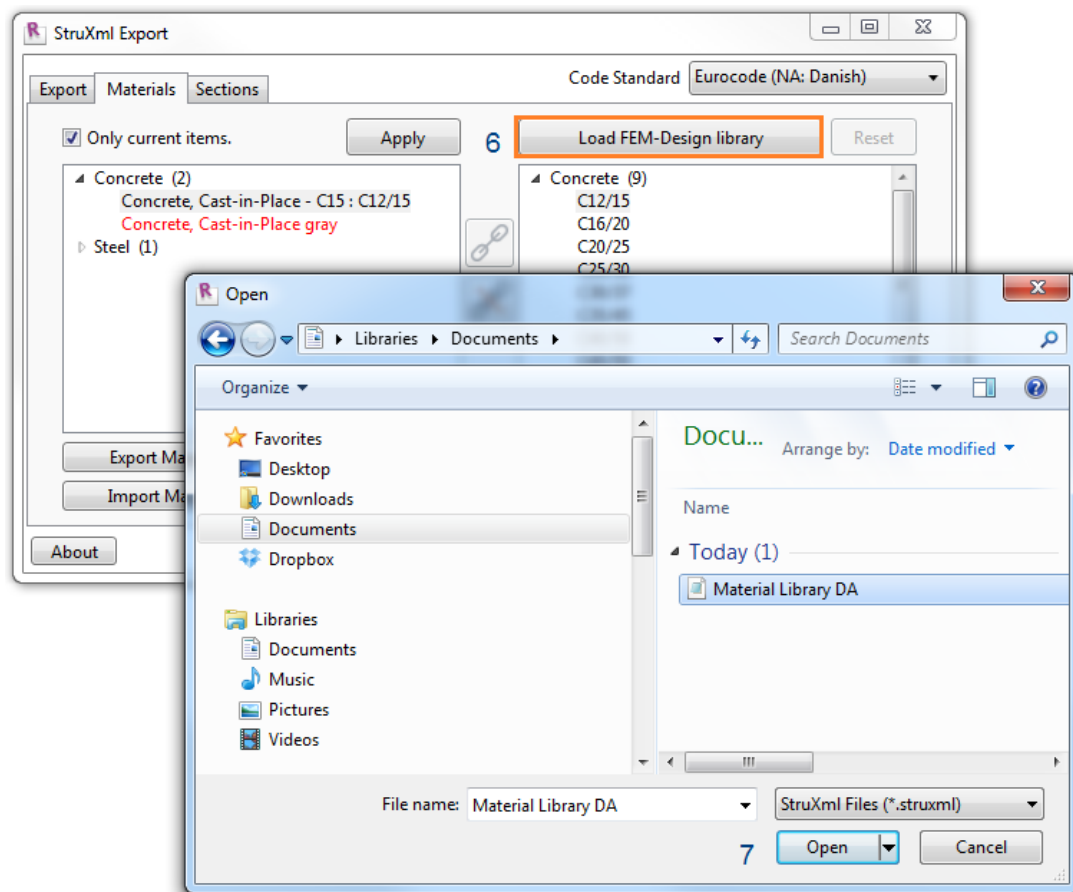


Figure 4-10

4.3. SECTIONS MAPPING

In order to map the sections one should move to *Sections* tab shown in Figure 4-11. The dialog is divided into two parts: *Revit* sections in the left part and *FEM-Design* sections in the right part.

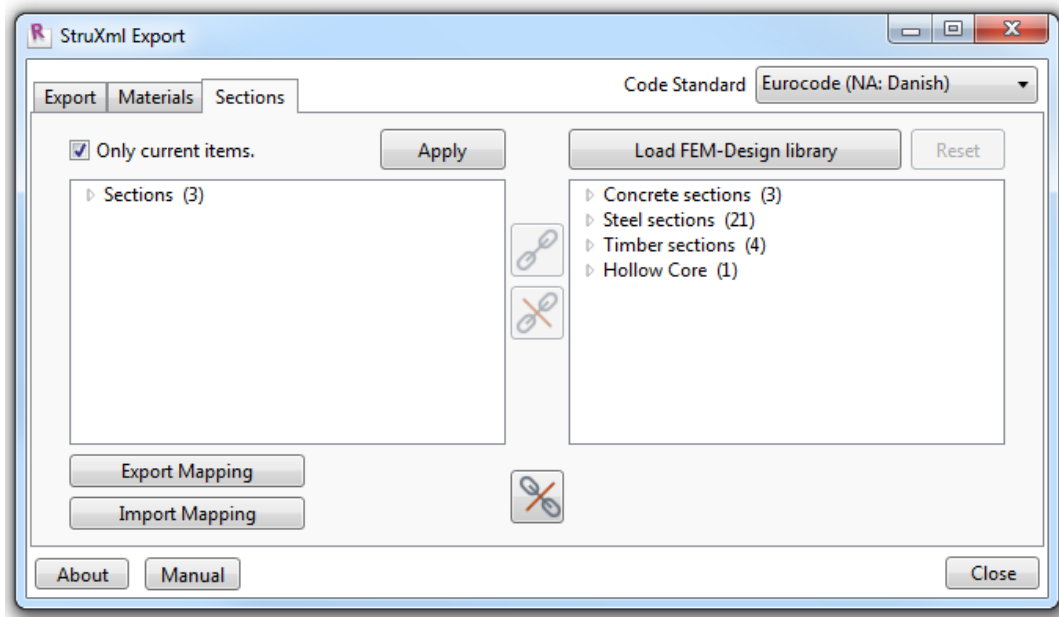


Figure 4-11

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

1. 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.
2. 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.
3. Press *Map selected item* button. A section that has just been mapped will turn black.
4. In order to save the sections mapping (and update the warning list in the status window in the *Export* tab) press *Apply*.

Unmapping procedure

5. 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.
6. If you wish to unmap all of the previously mapped sections, do not select any section – just press the *Unmap All Mapped Items* button.

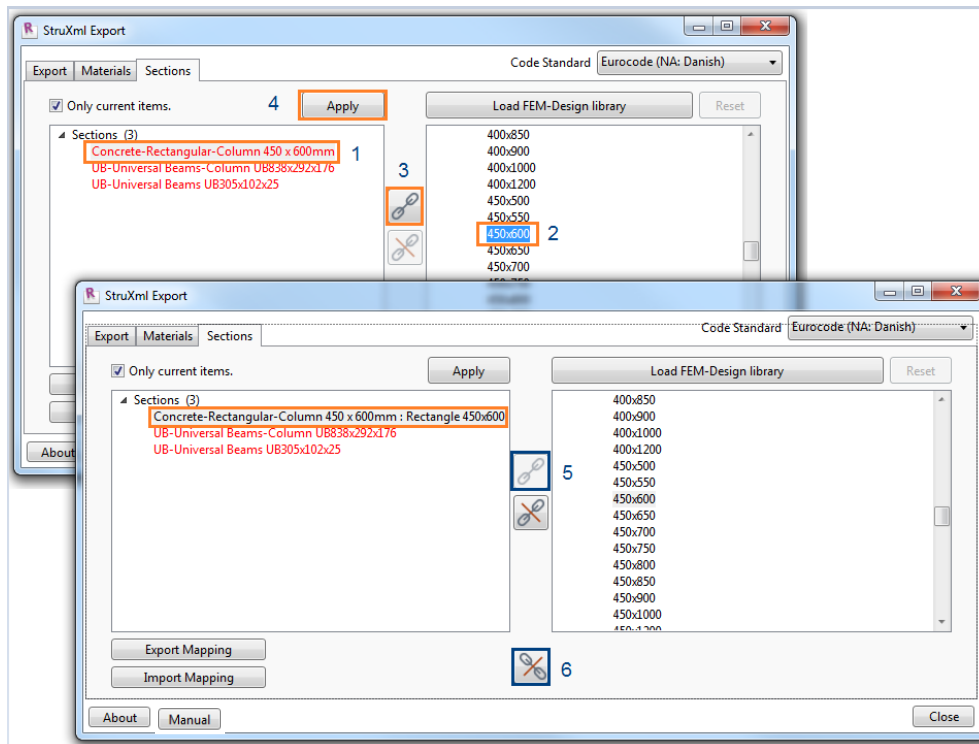


Figure 4-12

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. *Column*.
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.

4.4. 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.

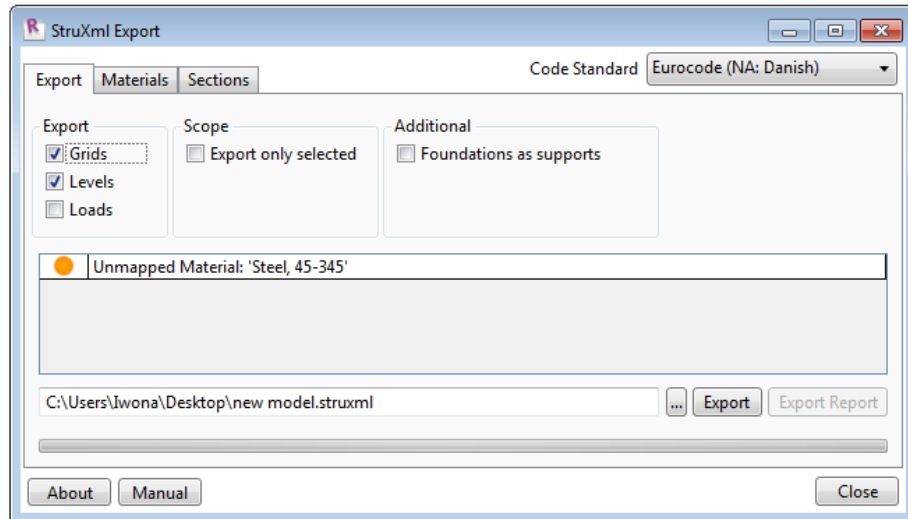


Figure 4-13

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.

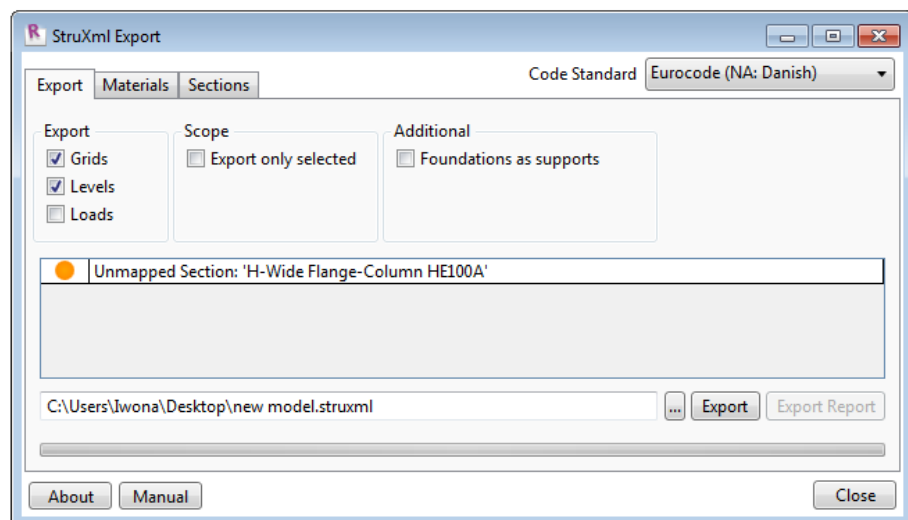


Figure 4-14

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.

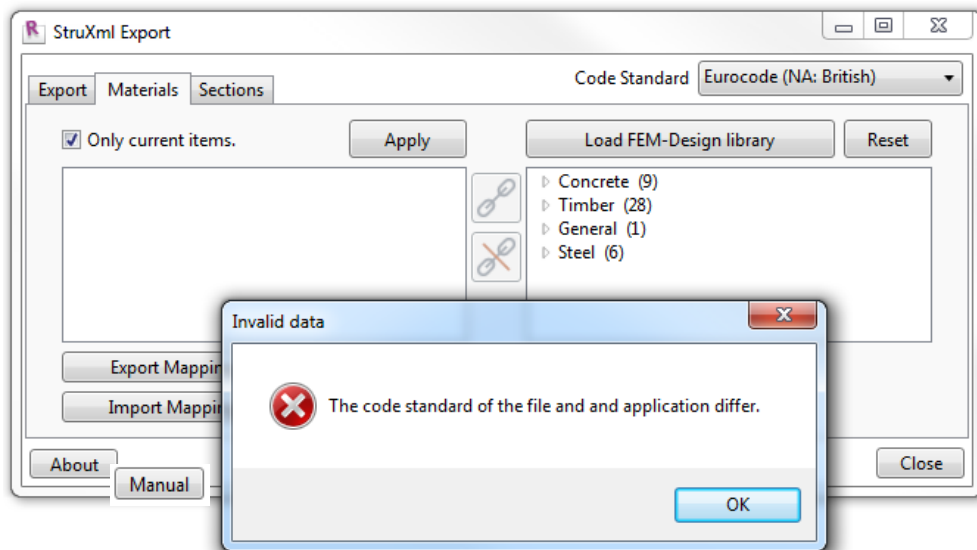


Figure 4-15

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.

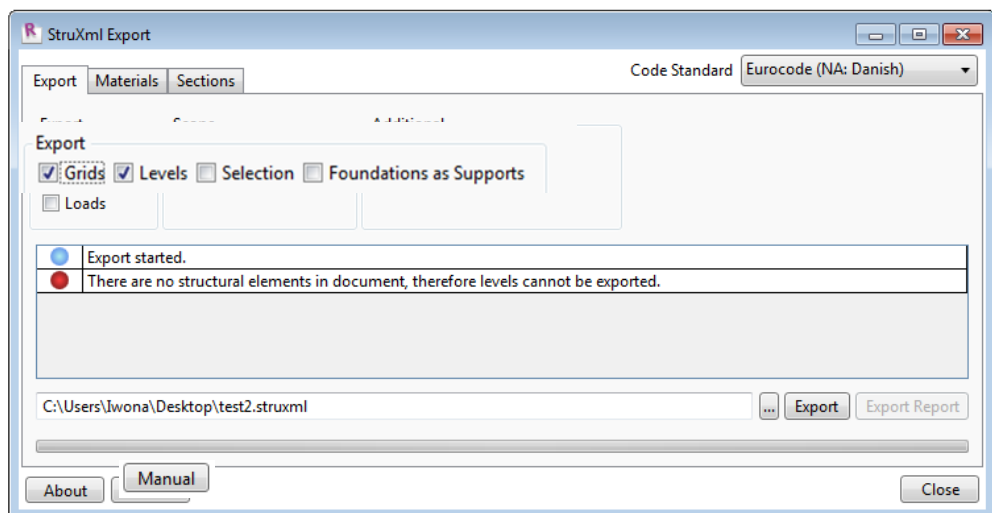


Figure 4-16

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.

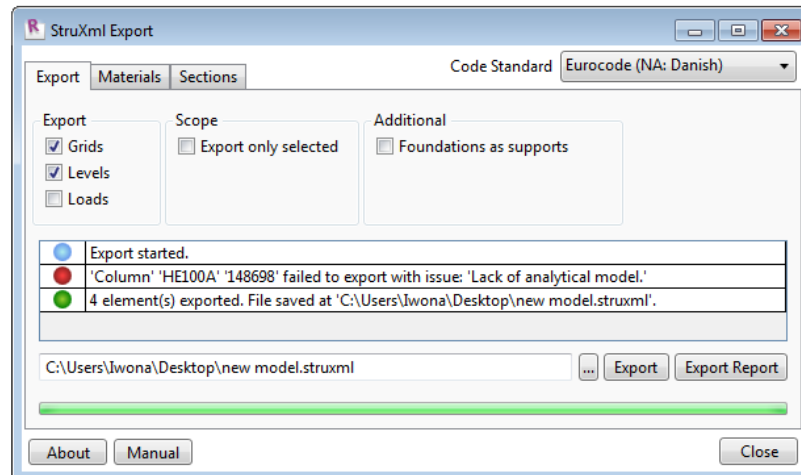


Figure 4-17

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.

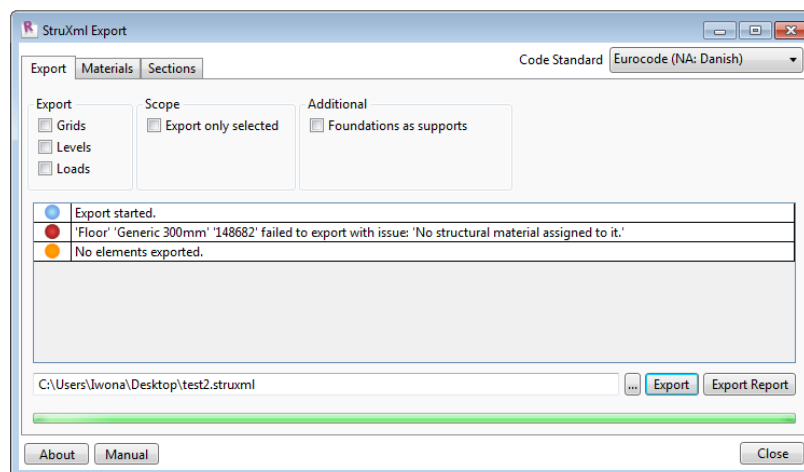


Figure 4-18

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.

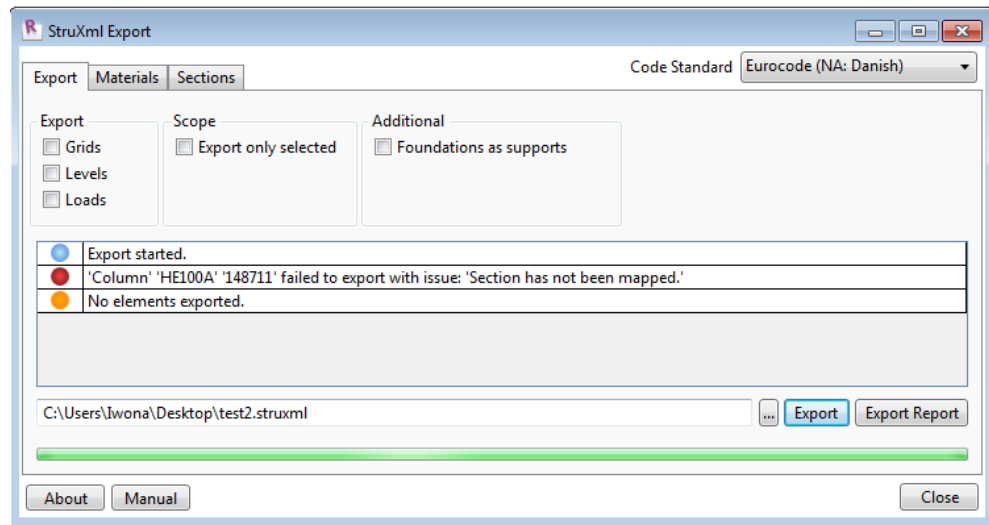


Figure 4-19

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.

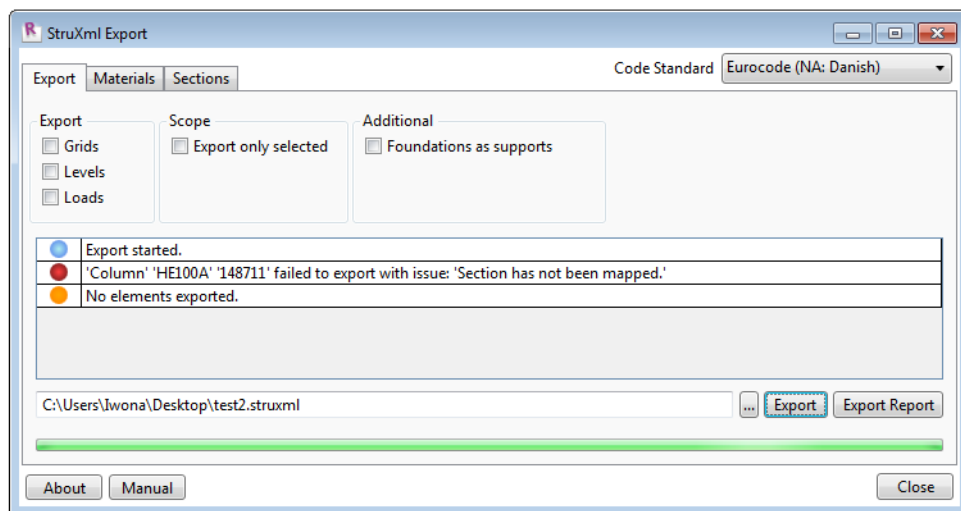


Figure 4-20



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

This part is relevant for users that exported their models using the StruXML Add-In, in version 1.00.001 and 1.00.002

If you try to open a project that has been saved in a Revit instance with StruXML Add-In installed, in a Revit instance that does not include the StruXML Add-In, a following warning may appear at some special situations as shown in Figure 4-21.

It is because Revit recognizes that some data has been saved to the model when the Add-In was present, and warns you that those data will not be recognized in the Revit instance without the Add-In installed.

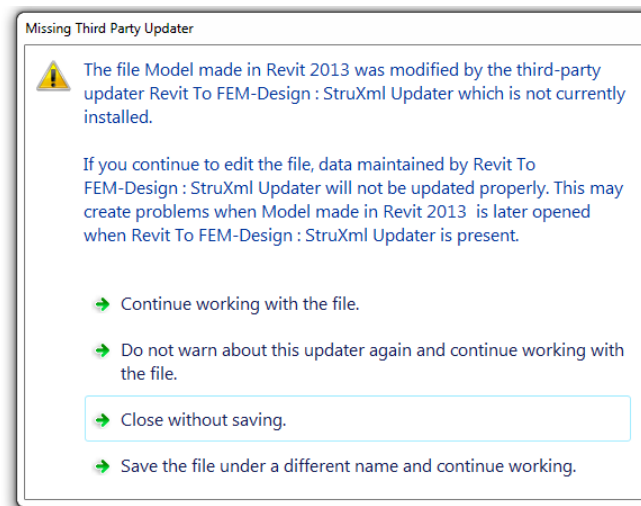


Figure 4-21

It is only a warning generated by Revit and in this case one should choose to *Do not warn about this updater again and continue wirking with the file.*

Starting from version 1.00.003, the publicity level of the data has been changed, so for all new models that were exported with the newest version of the Add-In (and not ever edited with the previous versions), Revit should not generate that kinds of warnings any more.

Another kind of warning (Figure 4-22) may be thrown by Revit, if one attempts to open two different Revit models containing different data schemas in one Revit sessions. In this case, in order to open the Revit model, restart of Revit is required.

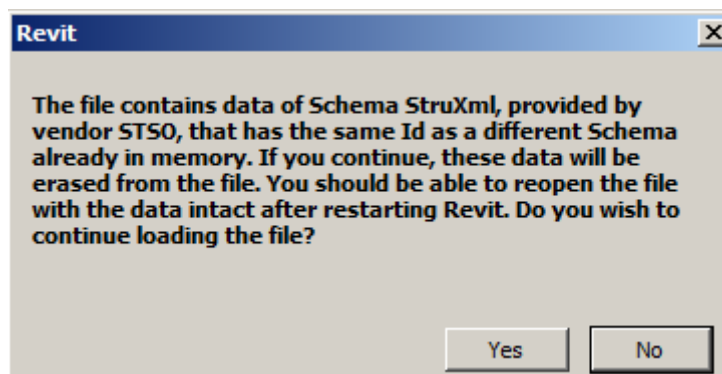


Figure 4-22

4.5. EXPORT REPORT

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

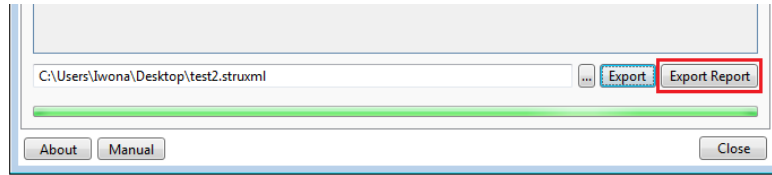


Figure 4-23

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.

Examples:

- Exported → Name → Type.

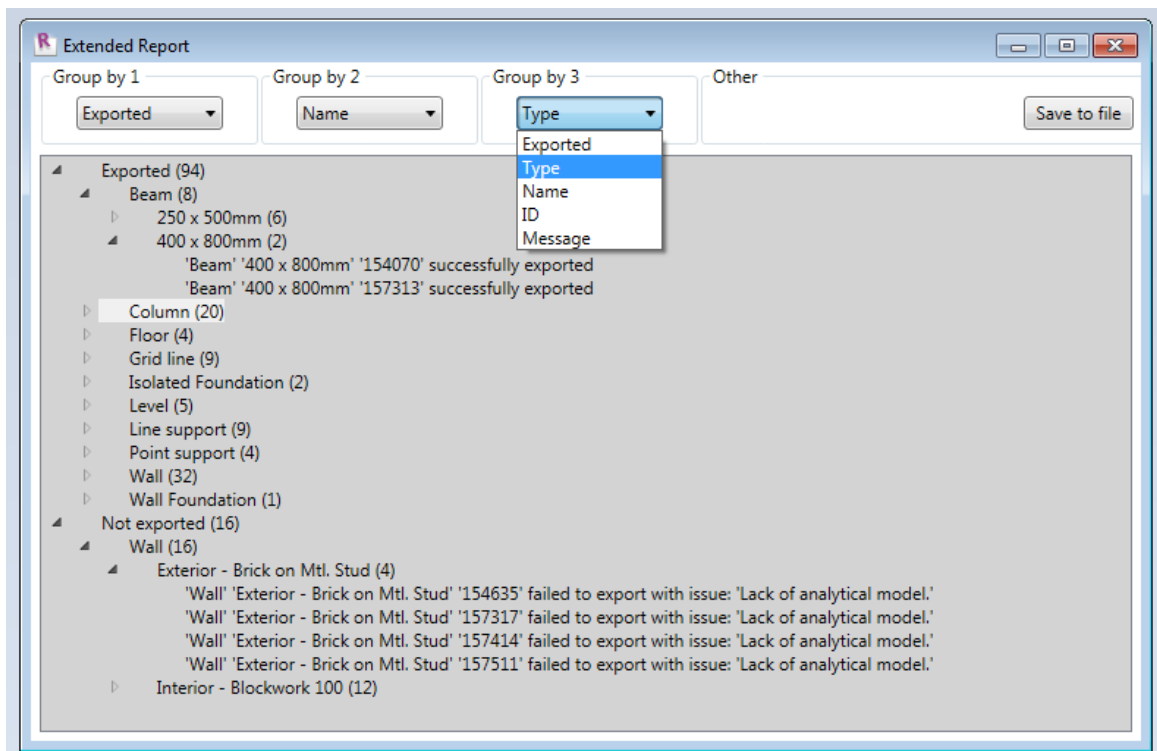


Figure 4-24

Figure 4-24 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 4-25 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.

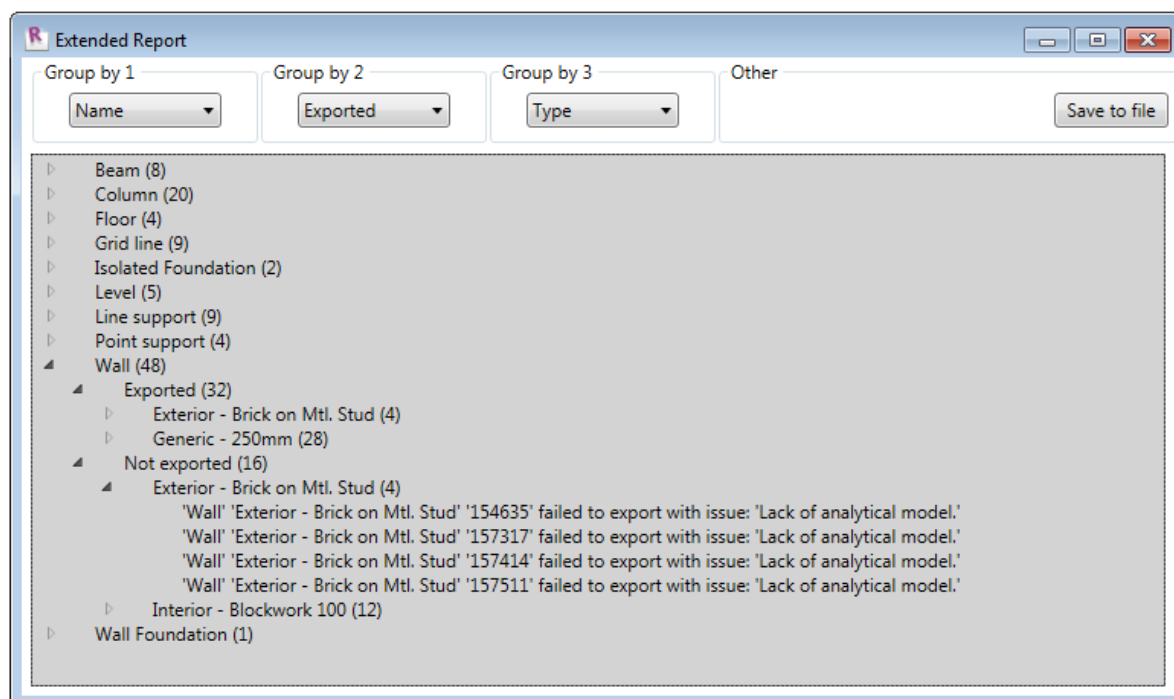


Figure 4-25

- 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 4-26, 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.

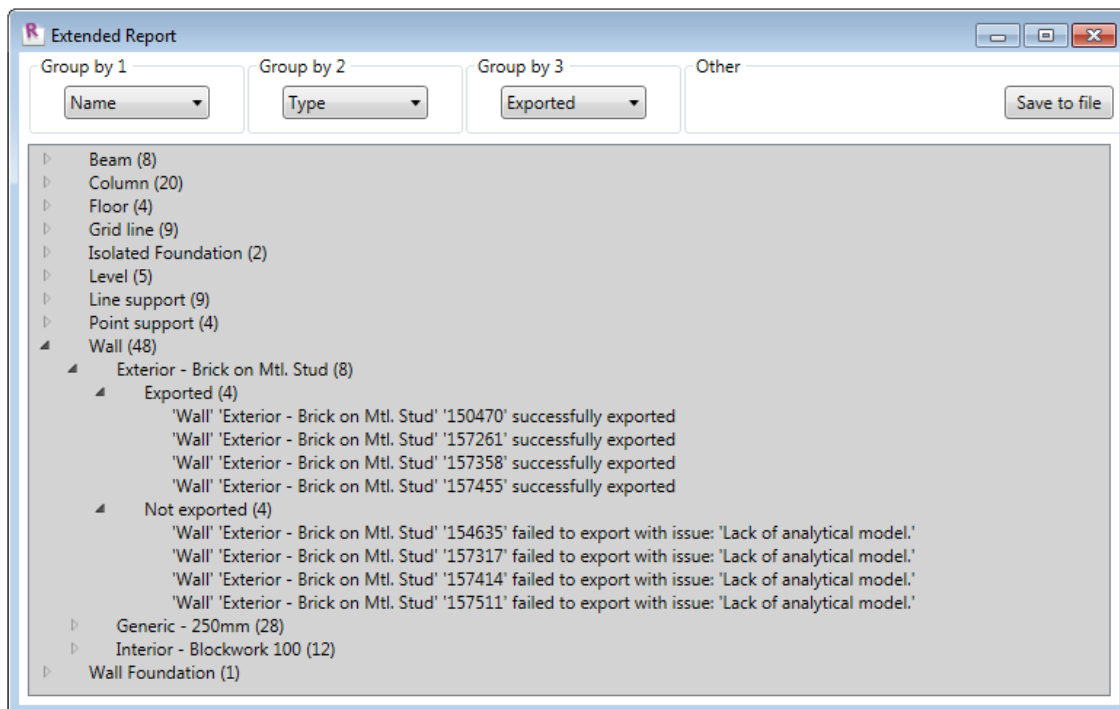


Figure 4-26

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 4-27 shows how the exported report can look like, when opened in the Internet browser.

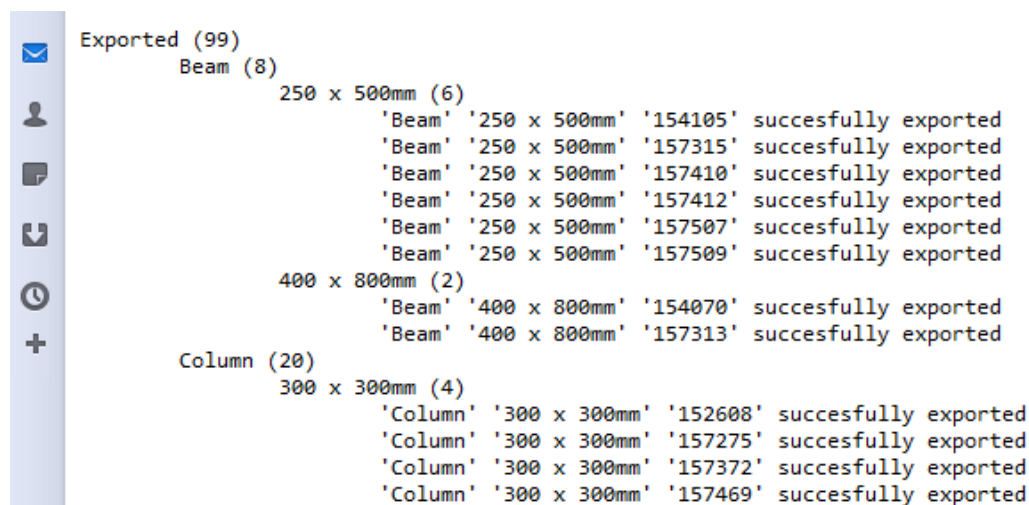


Figure 4-27

5. OPEN STRUXML IN FEM-DESIGN

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* module of *FEM-Design 11* (starting from version 11.01.001) and *FEM-Design 12.0*.

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.

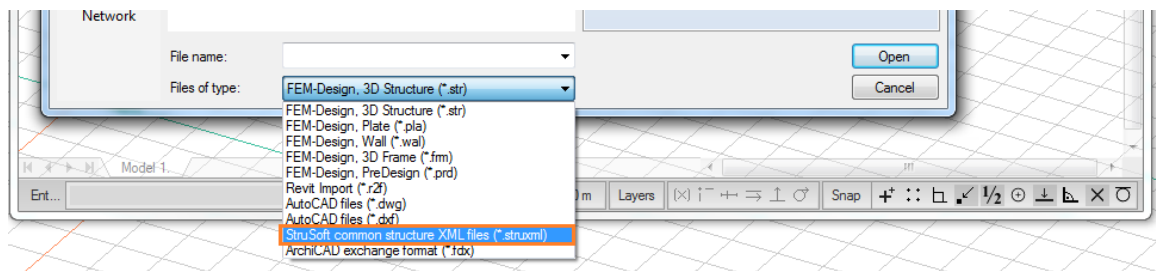


Figure 5-1

6. UPDATE FEM-DESIGN MODEL

A new command *Update File* has been introduced in FEM-Design. It is enabled in *FEM-Design 11* starting from version 11.01.003 and in *FEM-Design 12*. It is accessible from the main menu: *Edit* → *Update File*.

The *Update File* command gives a possibility to update initially exported elements to FEM-Design with the following changes that have been made in Revit afterwards:

- Material change
- Section / Thickness change
- Geometry change

Changes in material/profile/thickness/geometry properties are detected in the file containing the updated model in relation to the elements that are exported to FEM-Design at the moment.

By updating the model, one can also add to FEM-Design model additional elements that have been created in Revit.

Remember to always save the Revit model after you export the first model to struxml. By doing that, the struxml file receives a unique database identification number that will then be the same for every export file generated for the same model. It is crucial if one wants to be able to update the model in FEM-Design.



After the initial model is imported to FEM-Design, it has to remain at the original coordinates exported from Revit. Otherwise, the update of the model with changes performed in Revit will not be successful.

6.1. PROCEDURE

The procedure to update a model in FEM-Design is as follows:

1. Export the updated Revit model into new .struxml file.
2. Open a FEM-Design project containing the model that has been originally imported from Revit.
3. Go to *Edit → Update File*.
4. Select the new .struxml file containing the updated model and press *Open*.
5. A dialog with updateable entities will appear.
 - Here you can see which elements were added to the model (wall element that was no present in the FEM-Design model, therefore has no FD entity's data and is marked as "+").
 - You can also see the modified elements (all of them that have both the FD entity's data and imported entity's date), they are marked as ">>"
6. Select all the elements to be updated or only a particular one(s) and click on the *Update selected* button.

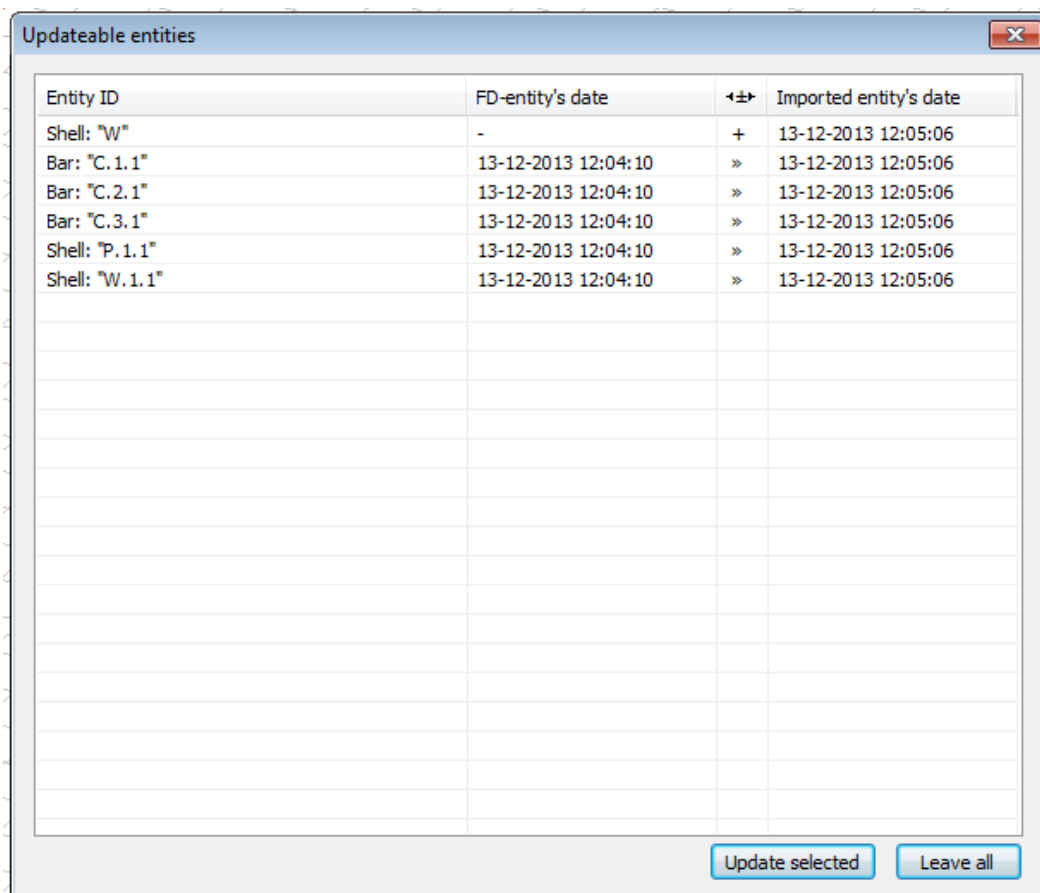


Figure 6-1

All the elements in the dialog are ordered by the time of creation, and the general order is:

- new elements - ordered by time,
- updated elements (time forward) - ordered by time,
- then updated elements (time backward) - ordered by time,
- deleted elements - ordered by time.

6.2. LIMITATIONS

The functionality of the *Update File* command at the current state of development has some limitations concerning elements that have been deleted in the updated Revit model or in FEM-Design:

- If an element has been deleted in the updated Revit model, after using the update command, this element will not be deleted from FEM-Design model.
- If an element has been deleted in FEM-Design model (but not in updated Revit model) and then updated with other changes made in Revit, it will reappear in FEM-Design.

6.2. EXAMPLE

Here is a brief example showing functionality of the *Update File* command.

1. In the first step, model shown in Figure 6-2 has been exported from Revit to struxml file and opened in FEM-Design (Figure 6-3).

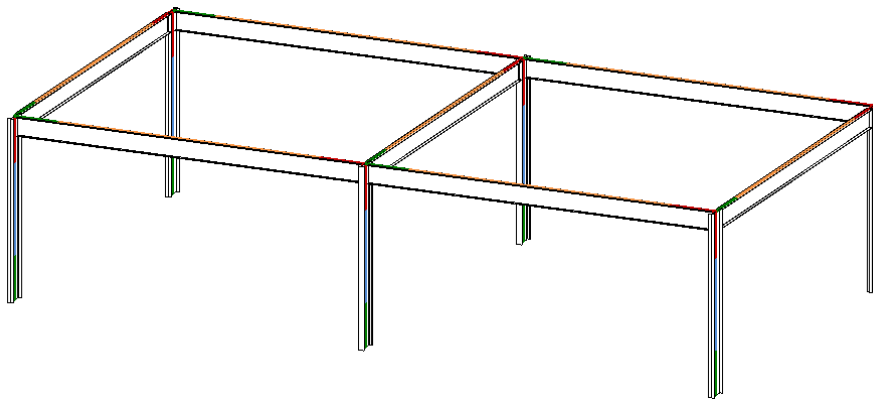


Figure 6-2

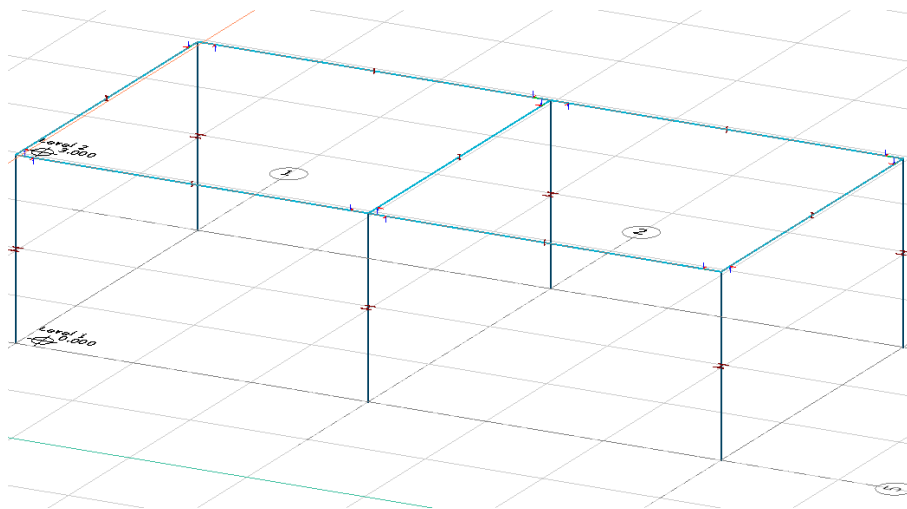


Figure 6-3

2. In FEM-Design following actions have been performed:

- Added support.
- Added loads and load combinations.
- Added bracing.

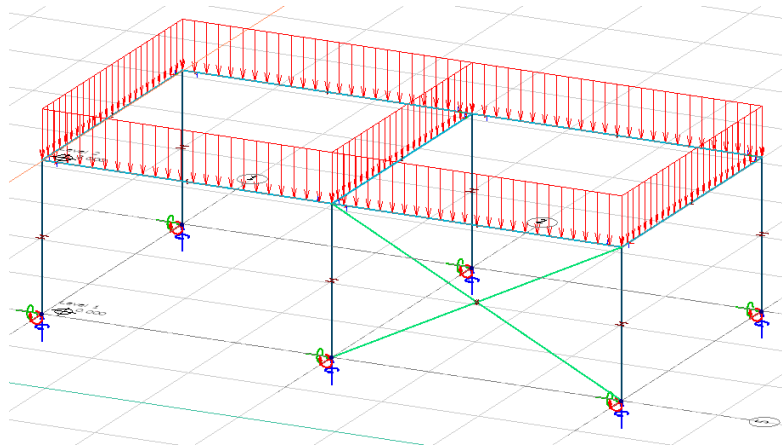


Figure 6-4

3. Meanwhile some changes have been applied to Revit model, and the updated model has been exported to new .struxml file. Following changes have been applied to the initial Revit model:
 - First span is now shorter than initially - columns and one beam have been moved, and the other beams got shorter.
 - Additional span covered with concrete floor has been added.
 - Profile of the two columns supporting the concrete floor changed and they have been rotated 90 degrees in relation to the original position.

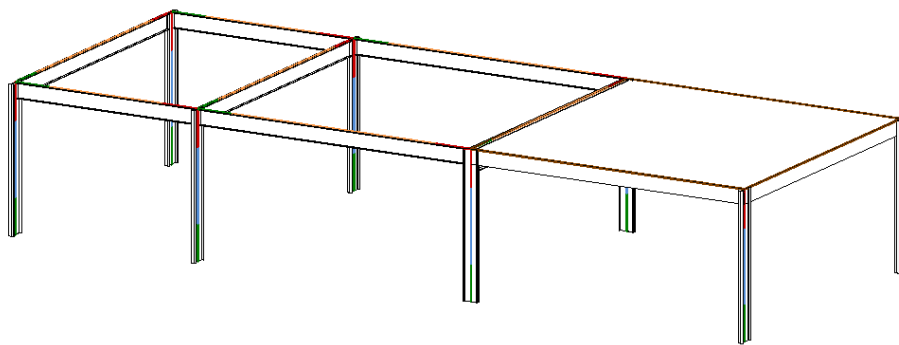


Figure 6-5

4. FEM-Design model (at the stage as shown in Figure 6-4) has been updated with the new .struxml file containing changes made in Revit since the first export.

The effect of the update is shown in Figure 6-6.

- The first span is now shorter.
- Additional span and concrete floor has been added.
- Two columns supporting the concrete floor changed their profile and direction/rotation.

However,

- the applied load will not adjust itself if the geometry changed - it has to be done manually (e.g. using *Move* or *Stretch* command). The load cases and load combinations remain in the FEM-Design model.
- since the first span got shorter and two columns have been moved, the supports that have been applied in FEM-Design need to be adjusted to the new position of columns.

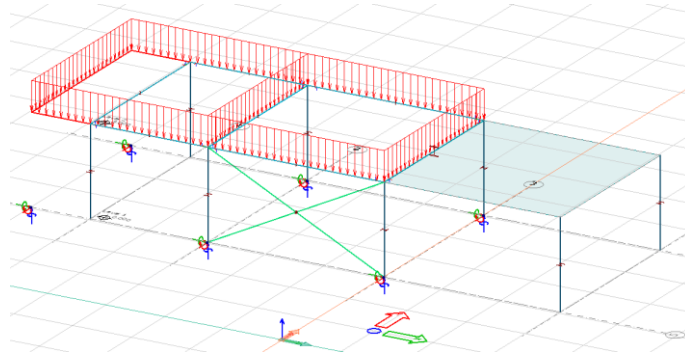


Figure 6-6

5. The model has now been adjusted in FEM-Design and some additional area loads have been applied to the concrete floor. The effect is visible in Figure 6-7.

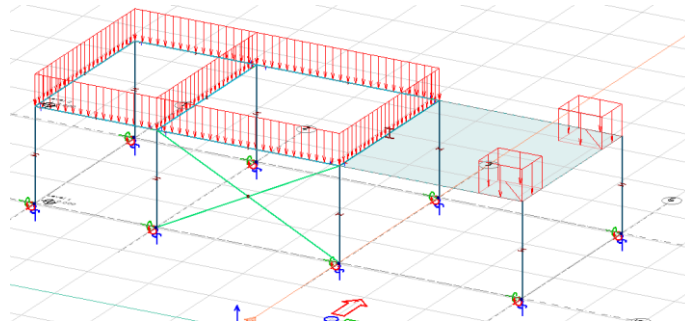


Figure 6-7

6. Meanwhile one more change has been applied in the Revit model - a hole was made in the concrete floor. The updated model was exported yet to another .struxml file.

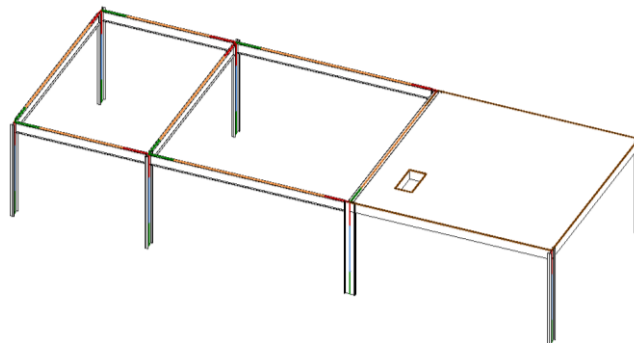


Figure 6-8

7. FEM-Design model (at the stage as shown in Figure 6-7) has been updated with the newest .struxml file containing changes made in Revit since the previous export. The effect of the update is shown in Figure 6-9 – a hole has been added to the floor, nothing else has changed.

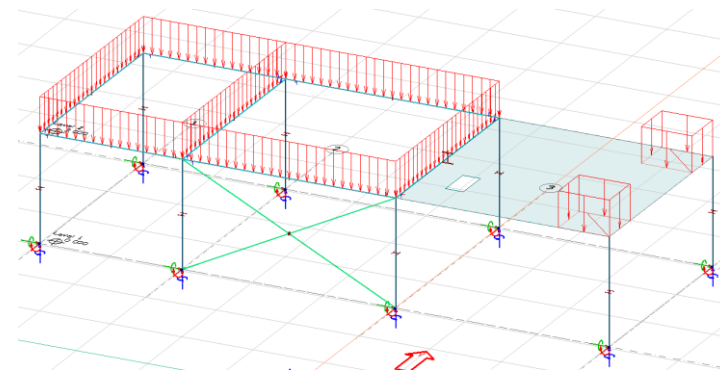


Figure 6-9