



Manual

Export, Import, Roundtrip

Export, Import, Roundtrip

Table of Contents

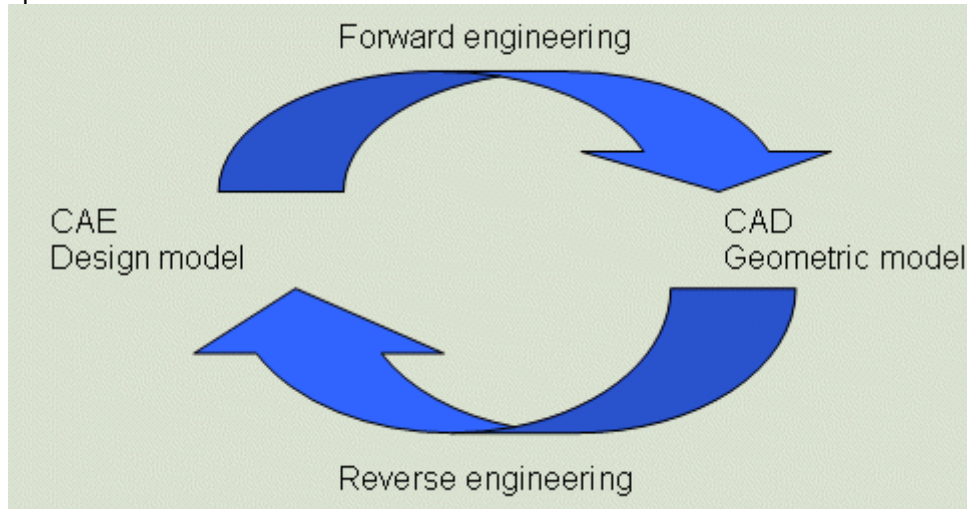
Scia Engineer	3
Scia Engineer import	3
Scia Engineer export	3
Scia Engineer update.....	3
Scia Engineer update dialogue.....	4
Allplan	7
Allplan - Importing SCIA ESA data.....	7
Allplan - Update from ESA file	8
Allplan - Exporting SCIA ESA data.....	9
Allplan - Save to ESA file.....	9
Allplan - Edit in Modeller	9
Allplan - Calculate ESA.....	10
XML.....	11
XML import.....	11
XML export.....	11
XML file editor	11
XML update	12
XML update example	12
Graphic format	21
Graphic format export	23
Export and import of DXF, DWG and VRML	24
Export from the graphical window	24
Export from the picture gallery	25
Export from the Paperspace gallery.....	25
Import into the graphical window	25
Import into the Paperspace gallery	30
Revit.....	35
Revit import	35
Revit export.....	35
Revit update.....	36
Revit link	39
IFC interface.....	57
SDNF Interface	71
DSTV.....	77
DSTV import.....	77
DSTV export.....	77
Pro Steel.....	79
Pro Steel import	79
Pro Steel export.....	80
Pro Steel update.....	82
StepSteel - analysis model.....	85
StepSteel - import of analysis model	85
StepSteel - export of analysis model	85
StepSteel - structural model	86
StepSteel - import of structural model.....	87
StepSteel - export of structural model.....	87
CEA Pland-4D	89
Tekla	91
Google Earth.....	95
Interface with ETABS.....	97
ECtools.....	109

Introduction to roundtrip

Roundtrip – what does it mean?

In order to explain the term that has slowly but surely domesticated in the vocabulary of engineers, we refer to the White Paper **Roundtrip Engineering in Construction - Integrated CAE-CAD solutions for specific market segments** by J.P. Rammant, CEO of SCIA International in March 2006.

Start of quotation:

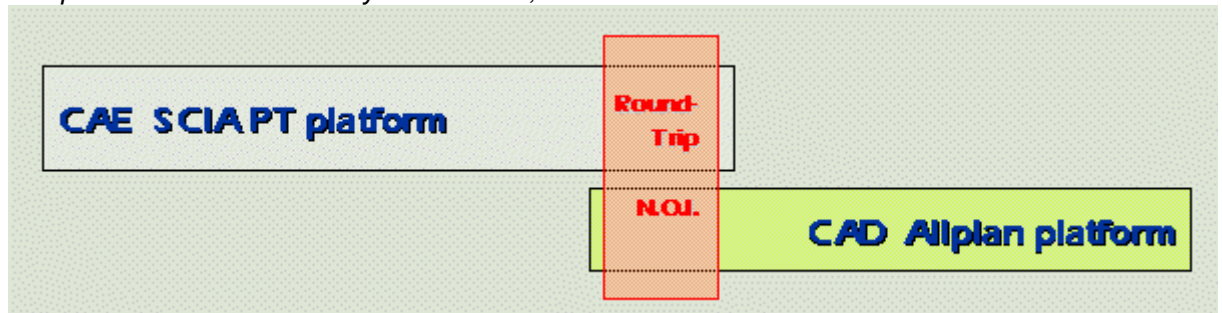


Practically every design of a construction is an iterative process of refining, reviewing, changing architectural, engineering or building (fabrication) details. Therefore, a roundtrip solution, where a designer or detailer can start or edit the model at any stage, is closest to reality and will lead to the shortest time between design and fabrication or building.

Reverse engineering reconstructs the design model from the CAD geometrical model, again an extra benefit.

Roundtrip engineering is being achieved by SCIA through a joint development with Nemetschek of a common interface between two platforms, one for CAE and for CAD.

Complex CAE calculations may be executed, hidden to the normal CAD user.



The Scia Engineer (Professional Technology) platform addresses CAE modelling, with a consistent structural and analysis model for steel and concrete building parts & structures. The structural objects are directly linked with corresponding parts within the Allplan platform, using the Nemetschek Object Interface. Steel sections and concrete constructive elements are one-to-one compliant.

In contrary to a link based on IFC between models of a same domain, here a transfer between a CAE and CAD model is being realized (in both directions).

Roundtrip is much more than import / export. Some extra benefits which are being implemented:

- the upgrade engine keeps the CAE and CAD model compatible; outside data (e.g. design data) are stored as Blob (Binary Large Objects) properties*
- labels remain positioned in associated views after changes*
- refinements and changes are possible in selected parts of a construction; they will be transferred to the other application.*

...

- fast modelling techniques in one platform are reflected in the other platform

...

SCIA is addressing the functionality of the Allplan platform concerning modelling of steel reinforcement.

Also with other CAD modelling systems such as Revit (Autodesk) SCIA is implementing the highest functional link; Revit is providing an API (Programming Interface) that allows partly roundtrip engineering. With Revit Structures SCIA will have a link to the Scia Engineer analysis model. It means that from Revit Structures the input of geometry and loading will be exported directly to SCIA .Scia Engineer. Changes to the structural model can be imported in the structural model of Revit.

...

...

...

Roundtrip engineering is drastically improving the design process; it is the most advanced implementation of Building Information Modelling technology. Integrated open general CAE-CAD platforms provide customized solutions without the necessity of dedicated programming. Ease of use and far going automation are key elements of the success.

End of quotation.

Roundtrip (including the one-way import / export) in SCIA. Scia Engineer

Format or Program	Import	Export	Update (Roundtrip)
SCIA .Scia Engineer	yes	yes	yes
XML	yes	yes	yes
IFC 2x3	yes	yes	yes
Revit	yes	yes	yes
Graphic format (excluding DXF, DWG, VRML)	---	yes	---
DXF, DWG, VRML	yes	yes	---
DSTV	yes	yes	---
ProSteel	yes	yes	yes
StepSteel (analysis model)	yes	yes	---
StepSteel (structural model)	yes	yes	---
CEA Pland-4D	---	yes	---

In the program itself (Scia Engineer), the functions are distributed over several different menus and tree menus:

- **File > Import,**
- **File > Export,**
- **File > Update,**
- service **Structure > Drawing tools > Import DWG, DXF, VRML97 file,**
- service **Tools > XML,**
- service **Structure > Import ESA project.**

Scia Engineer

Scia Engineer enables the user to perform the [import](#), [export](#) and [update](#) operations not only through "third-party format" files, but also directly through the .ESA file (i.e. the "native" format of Scia Engineer).

This capability opens the user new horizons. He/she can share data with the colleagues. He/she can add into the currently processed project parts from older projects (and thus save considerable amount of time). The work on one large project can be split among several users and the final model can be "glued" from several parts.

Scia Engineer import

The procedure to import data from .ESA file

1. Open service **Structure**.
2. Start function **Import ESA project**.
3. Browse for the required input file.
4. Confirm the file.
5. Input the "point of insertion" for the imported data.

Note: Contrary to imports through other formats, this time the specified file is added to the currently edited project.

Scia Engineer export

The procedure to export data into .ESA file

1. Start menu function **File > Export > New project ESA**.
2. Specify the required output folder.
3. Type the name of the file (i.e. the base name without extension).
4. The **Import/Export dialog** is opened on the screen.
5. Specify which types of data are to be exported (only geometric entities or the whole model)."
6. Complete the export operation.

Scia Engineer update

This function enables the user to exchange and share project data with colleagues who use Scia Engineer as well.

The principle is simple. User A makes the first version of the project and sends it to the user B. User B continues with the project and then returns it back to user A. Naturally, user A may have made some changes during this period too.

Now comes the time for Scia Engineer and its Update function. This function compares the two projects and finds added, deleted and modified entities. Everything is clearly summarised in a neat [dialogue](#). Finally, it is the user's turn again to decide which variants are those to be kept for future work.

The procedure to import and compare the project processed by a third-party user

1. Open your version of the project.
2. Call function **File > Update > ESA file**.
3. Browse for the file with the version that should be compared.
4. The program reads the project file and opens the [Update dialogue](#).
5. Decide which changes are to be accepted and which ones are to be declined.
6. Confirm with button **[Accept]**.

Scia Engineer update dialogue

The dialogue is a three-pane dialogue with a toolbar. The panes are:

- preview window,
- property window,
- merge window.

Toolbar

The control icons are mostly taken over from a standard Scia Engineer graphical window. Therefore, their meaning will not be discussed in detail here.

View parameters icons

These icons enable the user to adjust "what" and "how" should be displayed.

View-adjustment icons

These icons make it possible to set the view direction.

Zoom icons

These icons enable the user to zoom-in or -out the model.

Clipping box icons

These icons control the clipping box.

Special icons

Generate report		This icon generates a report about the update.
Display entities	original	If ON, the original entities are displayed.
Display entities	merged	If ON, the merged entities are displayed.

Preview window

The Preview window is a standard graphical window used throughout Scia Engineer.

The window supports standard features of Scia Engineer graphical windows:

- pop-up menu with a set of zoom, print, save etc. functions,
- [Ctrl] + [Shift] + right-click** and drag to zoom in and out the drawing,
- [Shift] + right-click** and drag to move the drawing,
- [Ctrl] + right-click** and drag to rotate the drawing.

The window displays the two merged projects in one view.

Moreover, the entity currently selected in the merge window (see below) is highlighted in this window.

Property window

This window is linked to the Merge window. If a particular entity is selected in the Merge window, then the corresponding properties are shown in the Property window. This is especially useful when the user has to track changes made in the two variants of the project.

Merge window

This window lists all the entities that somehow do not coincide in the two projects that are being merged.

Added, deleted and changed entities are shown here. The user may select which changes are to be accepted and which ones are to be ignored.

Individual groups (new, deleted, merged entities) will now be described in more details.

New entities

Accepting the group

By default, the check box at this item is selected (ticked). It means that new entities (as a group) will be accepted in the final merged project. If you deselect this check box, no new entities will be considered by the update function at all.

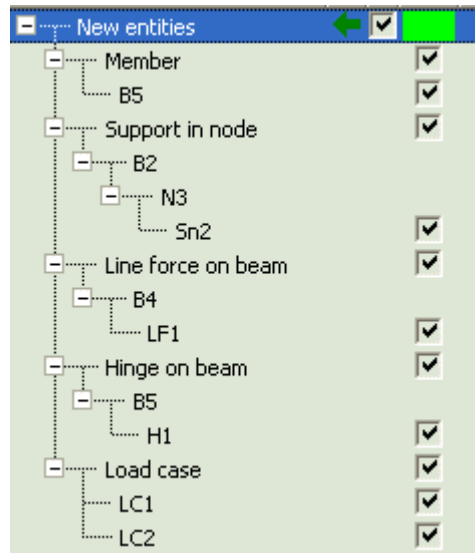
Specifying the colour of the group

You may adjust the colour for the new entities. Simply click on the coloured box on the right-hand side of the "New entities" line and then select the required colour.

Accepting individual items from the group

You may select which particular entity (or sub-group) from the "New entities" group should be accepted and which one should be declined. Just mark (tick) or unmark (untick) the appropriate item. Please note that once you put cursor (click) on a specific item, its properties are shown in the middle pane (property window) and the entity itself is highlighted in the preview window. Some items may be "grouped" in sub-groups such as load cases in the picture below. This enables you to decline (if required) the whole subgroup by unmarking the corresponding subgroup check box.

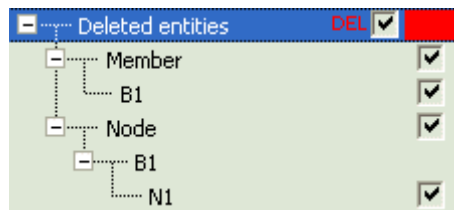
Example:



Deleted entities

This is analogous to "New entities" described above.

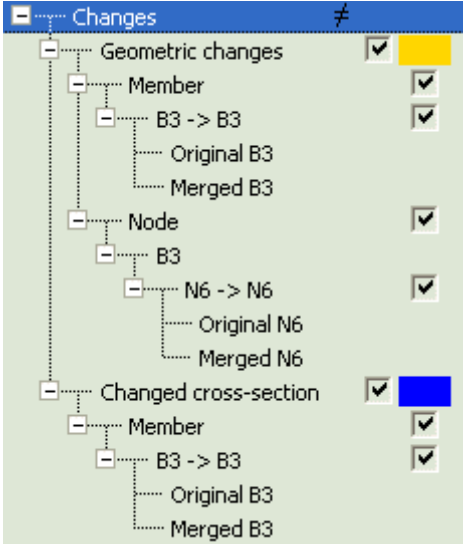
Example:



Changes (Merged entities)

This part of the merge window lists all the entities that have different properties in the two compared projects. The list always contains the "original" and the "merged" entity. The properties of both are shown in the property window (one by one – not together). If you select (tick) a particular listed item, the changes made in the merged (second) project will be accepted. If you do not select (unmark) a particular listed item, the original (the first project) will be preserved.

Example:



Note: The Update function can process geometric entities, load cases, loads, supports, hinges, etc.

Allplan

The exchange of data between Allplan and Scia Engineer can be realised in several ways. First, standard export and import routines can be applied. Second, direct interlink of the two programs can be used.

The first approach uses standard Scia Engineer project files. Allplan is capable of both saving the project into this format and reading the project from this format. The second approach leads to very fast connection – Scia Engineer is opened directly from within the Allplan environment and the changes made in one program are automatically taken into account in the second program. For example, the following scenario is possible (the aim of this particular example is to give you the idea of what is possible, not to present the most effective style of work, even though it may even be the case for certain situations and types of structures):

1. The project is started in Allplan. Some Scia Engineer steel templates are inserted into the model.
2. Scia Engineer is opened and one or two 1D members in the model are modified (moved, changed cross-section, etc.).
3. The model is sent back to Allplan and a new Scia Engineer steel template is added.
4. Scia Engineer is opened again and a part of the model is removed.
5. The model is sent back to Allplan and the model is completed.

Note: The direct communication between the two programs has inevitably some limitations. The user must decide which type of entities will be used: either Allplan architectural elements or Scia Engineer steel templates. If this prerequisite is satisfied, the full roundtrip between the two programs is fully functioning. On the other hand, if both the above-mentioned types are used in one project, the direct and complete exchange of the data between the two programs is not possible.

The following list summarises all Allplan data-exchange functions:

- [Importing SCIA ESA data](#),
- [Update from ESA file](#),
- [Exporting SCIA ESA data](#),
- [Save to ESA file](#),
- [Edit in Modeller](#),
- [Calculate ESA](#).

Allplan - Importing SCIA ESA data

Allplan does not create a new project but adds the imported data to the existing currently opened model.

The import function requires that Scia Engineer is installed on the computer where Allplan is run – a demo-version is sufficient.

The procedure to import .esa file to Allplan

1. Start function **Create > Interfaces > Importing SCIA ESA data**.
2. Browse for the file to-be-imported.
3. A dialogue with import parameters is opened on the screen.
4. Fill in the required parameters (see below).
5. Confirm the settings to proceed with the import.
6. If the option **Make special adjustment** was selected for the **View parameters**, the **View parameters dialogue** opens on the screen. In that case, make the necessary settings and end the import.
7. The import is done.

Import parameters dialogue

Colour

Use colours from ESA project	The imported entities are assigned colours as they were adjusted in Scia Engineer.
Colour from Allplan	The colours are taken from the settings made in Allplan.

View parameters

Use adjustment from ESA project	The view parameters are taken exactly as they were set in Scia Engineer.
Make special adjustment	The user may adjust the view parameters manually during the import procedure.
Precise surface member	If ON, the outline of cross-sections is drawn with utmost precision.

3D geometry

One volume	If ON, the whole imported structure is made a single object in Allplan.
-------------------	---

Allplan element

BIE Macro	If ON, the imported data are stored as Allplan BIE macros.
Structural element	If ON, the the imported data are stored as Allplan structural elements (Konstruktive Fertigteil). The imported data cannot be edited by Scia Engineer editing functions integrated in Allplan. The editing can only be done through standard Allplan functions.

Allplan - Update from ESA file

This function reads a Scia Engineer data file and adds its contents to the currently opened project in Allplan.

The function requires that iScia Engineer is installed on the computer where Allplan is run – a demo-version is sufficient.

The imported data update the existing members in the currently open project (this, however, applies to steel templates only).

The procedure to update data in Allplan from .esa file

1. Start function **Create > Add-On modules > Steel/Concrete Construction > Update from ESA file**.
2. Browse for the file to-be-imported.
3. A dialogue with import parameters is opened on the screen.
4. Fill in the required parameters (see [Importing SCIA ESA data](#)).
5. Confirm the settings to proceed with the import.
6. If the option **Make special adjustment** was selected for the **View parameters**, the **View parameters dialogue** opens on the screen. In that case, make the necessary settings and end the import.
7. The update is done.

Allplan - Exporting SCIA ESA data

This function exports the whole project created or edited in Allplan into Scia Engineer data file. The export function requires that Scia Engineer is installed on the computer where Allplan is run – a demo-version is sufficient.

The procedure to export data into .esa file from Allplan

1. Start function **Create > Interfaces > Exporting SCIA ESA data**.
2. Browse for the required folder and specify the file name.
3. Export is done.

Allplan - Save to ESA file

This function saves the project created or edited in Allplan into Scia Engineer data file.

The function requires that Scia Engineer is installed on the computer where Allplan is run – a demo-version is sufficient.

The function enables the user to save either the whole project or just a selected part of it into the Scia Engineer data file.

The procedure to save data into .esa file from Allplan

1. Start function **Create > Add-On modules > Steel/Concrete Construction > Save to ESA file**.
2. Select the entities to be saved or save the whole project.
3. Browse for the required folder and specify the file name.
4. Save-operation is complete.

Allplan - Edit in Modeller

If you are creating a model of a structure composed of Scia Engineer steel templates, you may freely swap between Allplan and Scia Engineer and work in both programs almost at the same time. The scenario (given also in the introductory chapter) can be like this:

1. The project is started in Allplan. Some Scia Engineer steel templates are inserted into the model.
2. Scia Engineer is opened and one or two 1D members in the model are modified (moved, changed cross-section, etc.).
3. The model is sent back to Allplan and a new Scia Engineer steel template added.
4. And so on and so forth.

Any change of the model made in either program is automatically reflected in the other program.

The procedure to perform this combined editing

1. In Allplan start with the model (i.e. insert some Scia Engineer steel templates).
2. Start function **Modify > Add-on Modules > Steel/Concrete Construction > Edit in Modeller**.
3. Select the entities to be sent to Scia Engineer Modeller.
4. Scia Engineer is opened and the Roundtrip dialogue appears in the screen..
5. Do whatever is needed.
6. Call function (in Scia Engineer) **File > Close**.
7. A small Save-Cancel dialogue opens on the screen.
8. Select the option that saves the changes.
9. You are back in Allplan.

Note: Please note that Allplan function **Create > Add-on Modules > Steel/Concrete Construction > Start Modeller** starts the Scia Engineer Modeller and opens an empty project in it. It does not transfer the data from your current Allplan project.

Note: It is not recommended to combine in one project Allplan architectural elements and Scia Engineer steel templates. If both the types are used in one project, the direct and complete exchange of the data

between the two programs is not possible. On the other hand, separate import and export functions are still fully applicable.

Allplan - Calculate ESA

If you are creating a model of a structure composed of architectural elements, you may freely swap between Allplan and Scia Engineer and work in both programs almost at the same time. The general scenario is the same like for [Editing in Modeller](#). However, you may decide which change of the model is taken into account.

The procedure to perform this combined editing

1. In Allplan start with the model.
2. Start function **Modify > Add-on Modules > Steel/Concrete Construction > Calculate ESA**.
3. Select the entities to be sent to Scia Engineer.
4. Scia Engineer is opened.
5. Do whatever you need.
6. Call function (in Scia Engineer) **File > Close**.
7. A small Save-Cancel dialogue opens on the screen.
8. Select the option that saves the changes.
9. You are back in Allplan.

The important feature comes into play when you are swapping to Scia Engineer for the second (third, fourth, etc.) time. Scia Engineer can recognise that you have been already editing the same project in it and (in order to help you tracing the changes made in Allplan) opens the dialogue that compares two variants: (i) the data that are currently sent from Allplan to Scia Engineer and (ii) the data that were saved the last time you were working in Scia Engineer. A special designed dialogue guides you through the changes and allows you to decide which changes are to be approved and which ones are to be discarded. This is the Scia Engineer [update dialogue that is described in a separate chapter](#).

Note: It is not recommended to combine in one project Allplan architectural elements and Scia Engineer steel templates. If both the types are used in one project, the direct and complete exchange of the data between the two programs is not possible. On the other hand, separate import and export functions are still fully applicable.

XML

Scia Engineer allows for the exchange of data with third-party applications through the popular and powerful XML format. Moreover, XML format can be used to develop tailor-made applications that use Scia Engineer as a "hidden" engine working on the background and performing calculations of company-specific problems.

Note: The aim of this text is to present the principles of XML interface. The full explanation of all features and all capabilities of XML import/export/update would occupy dozens and maybe even hundreds of pages and exceeds the extent and scope of this text. Users who are interested in the application of XML interface in their practice are kindly asked to contact the help-desk of SCIA Company. SCIA specialists will be happy to give concrete piece of advice that could help solve the particular problem of your company.

XML import

The procedure to import data from XML file

1. Start menu function **File > Import > XML file**.
2. Browse for the required .xml file.
3. Confirm the selection of the file.
4. If required by the import procedure, specify the national standard to be used.
5. Finish the import.

Note: The import function creates a new empty project and the specified file is imported into it.

XML export

The procedure to export data into XML file

1. Use the tree menu to start function **Tools > XML**.
2. [XML file editor](#) is opened on the screen. Before it is opened, the program may ask you to select the default XML document template. You may select one or just close the dialogue without choosing anything. If you want to export the whole structure into a XML file, it is convenient to select the **default.TDX** template.
3. In the XML file editor define the structure of the XML document.
4. Use icon **Export** on the toolbar to export the created document into a XML file.
5. Close the XML file editor.

Note: Please remember that the order of Scia Engineer entities stored in the exported XML file must guarantee that all the references are made to entities stored earlier in the file. It means, for example, that materials must come before cross-sections, cross-sections and nodes must come before 1D members, load cases must come before load case combinations, etc.

XML file editor

The XML file editor is very similar to the Document of Scia Engineer. The principle is that the user defines tables describing individual entities of the Scia Engineer projects and there order. This table-form can be easily previewed (it is in fact identical to the standard Scia Engineer document). When ready, the final XML file contents can be transformed into the real XML format through the **Export** function.

The dialogue contains the following parts:

XML	document	Selects the required XML document if more
------------	-----------------	---

selection box	than one document has been created.
button to open the XML document manager	Opens the XML document manager where you can create new documents or delete the existing ones. It is a standard Scia Engineer database manager.
XML file contents window	This window shows the contents and structure of the currently edited document. This is analogous to the document tree window.
property window	Displays parameters related to currently selected item in the XML file contents window. This is analogous to the document property window.
icon toolbar	Contains buttons (icons) that perform various actions. The toolbar is identical with the Document window toolbar.
control buttons	There are two control buttons: [New] – adds a new item to the XML file contents window. [Close] – closes the XML file editor.
action button	There is just one action button – it refreshes the preview window of the XML file editor.
preview window	Displays the currently edited XML file in table-form.

XML update

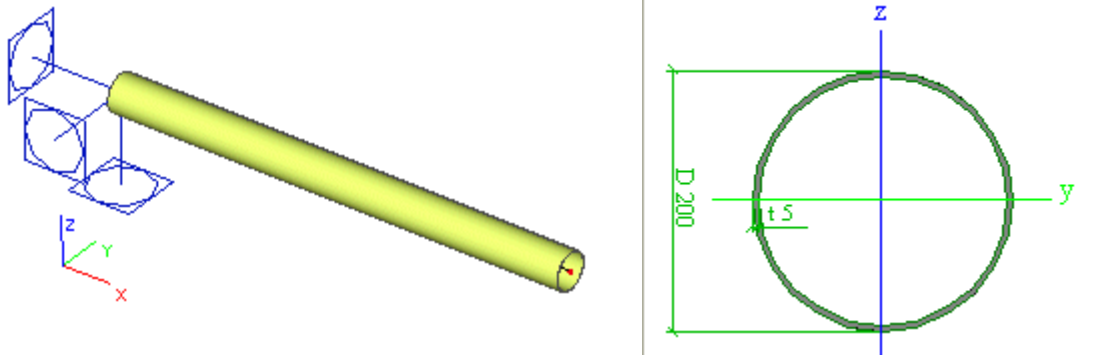
The procedure to update the project through XML file

1. Open in Scia Engineer the project to-be-updated.
2. Call function **File > Update > XML file**.
3. Browse for the file to-be-merged.
4. Confirm the selection of the file.
5. The project in Scia Engineer is updated by the data read from the file.

Note: For more information on the update function read chapter [XML update example](#).

XML update example

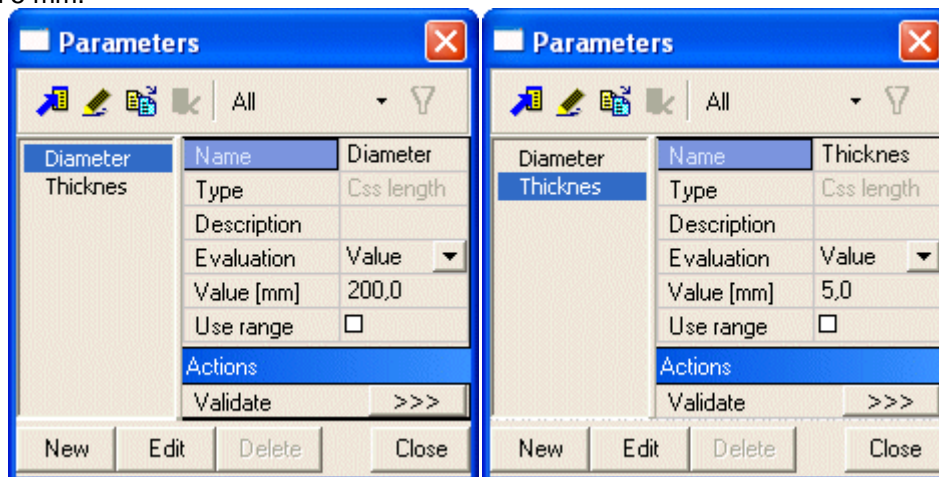
Let us have a project named **Cantilever.esa** with a simple cantilever beam of a circular cross-section.



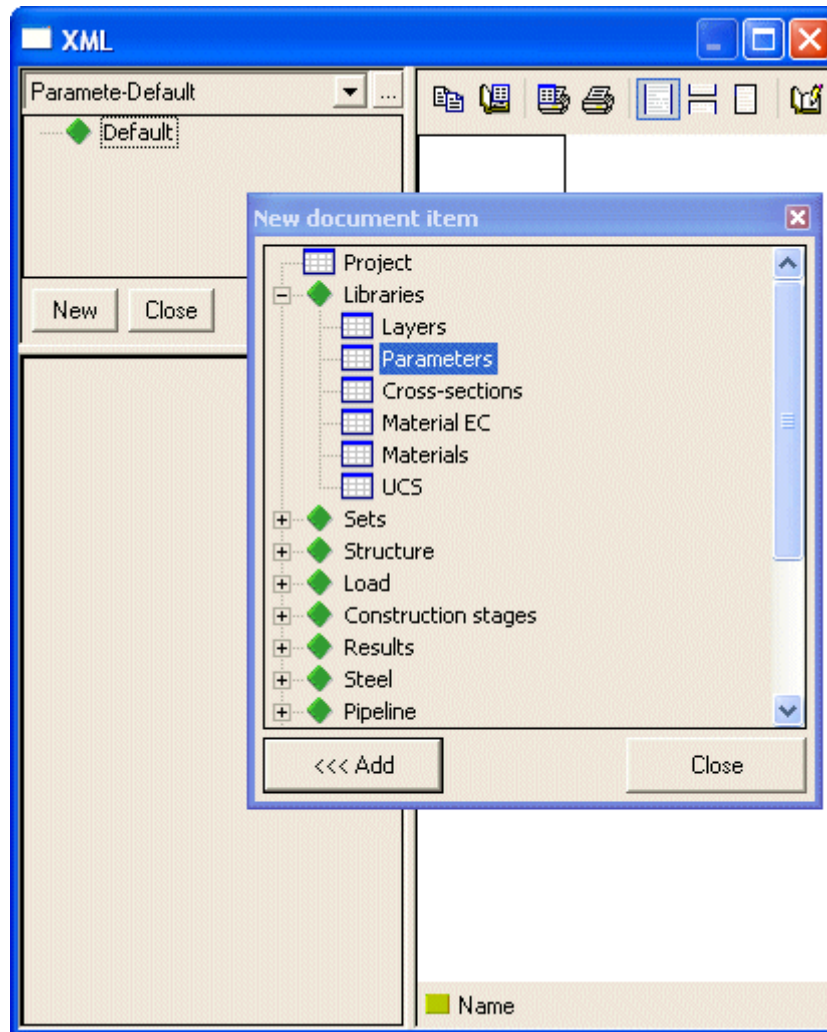
The dimensions of the cross-section are defined through two parameters: Diameter and Thickness.

Name	CS1
Selector switch	<input type="checkbox"/>
Type	Tube
Detailed	200, 5
Parameters	
Material	S 235
D [mm]	Diameter
t [mm]	Thicknes
General	
Draw color	Normal colour
Colour	
Properties editable	<input type="checkbox"/>

Both the parameters are of type Cross-section length (Css length) and their initial values are 200 mm and 5 mm.



Now we have to open the XML file editor (tree menu function **Tools > XML**). There we have to create a new "XML document" containing the table of defined parameters. To do so, we add the table **Libraries > Parameters**.



The inserted table looks like this:

name	Unique ID	Type	Evaluation	Use range	Value
Diameter	{FDD9298F-1A51-406B-9887-8C9302981A47}	Css length	Value	false	Real 0,2
Thicknes	{9E8E662D-4B12-43EE-8A56-A33948D079A1}	Css length	Value	false	Real 0,005

The created XML document must be exported into XML file. This can be achieved through function **Export** (available on the toolbar of the preview window in the XML file editor). The generated XML file (named **CantileverParameters.xml**) looks like:

XML file	Comment
<pre><?xml version="1.0" encoding="UTF-8" standalone="yes" ?> <project xmlns="http://www.scia.cz"> <def uri="CantileverParameters.xml.def" /> <container id="{5C62EA11-86D4-11D4-B3AB-00104BC3B531}" t="EP_Parameters.EP_Param.1"> <table id="B4DDEC1E-5869-4B8C-8B4B-</pre>	

<pre> 1A8C07B76699" t="EP_Parameters.EP_Param.1"> =<h> <h0 t="Name" /> <h1 t="UniqueID" /> <h2 t="Type" /> <h3 t="Evaluation" /> <h4 t="Use range" /> <h5 t="Value" /> </h> =<obj id="1" nm="Diameter"> <p0 v="Diameter" /> <p1 v="{FDD9298F-1A51-406B-9887- 8C9302981A47}" /> <p2 v="11" t="Css length" /> <p3 v="0" t="Value" /> <p4 v="0" /> =<p5 t=""> =<h> <h0 t="Real" /> </h> =<row id="0"> <p0 v="0.2" /> </row> </p5> </obj> =<obj id="2" nm="Thicknes"> <p0 v="Thicknes" /> <p1 v="{9E8E662D-4B12-43EE-8A56- A33948D079A1}" /> <p2 v="11" t="Css length" /> <p3 v="0" t="Value" /> <p4 v="0" /> =<p5 t=""> =<h> <h0 t="Real" /> </h> =<row id="0"> <p0 v="0.005" /> </row> </p5> </obj> </table> </container> </project> </pre>	<p>Parameter Diameter</p> <p>value = 0.2 m</p> <p>Parameter Thickness</p> <p>value = 0.005 m</p>
--	--

We will not explain the meaning of every single line of the XML document. The example above demonstrates the structure of the XML file and shows the way the two parameters from our example are stored.

Note: When the export function is invoked, the XML file editor generates in fact two files: XML file and the corresponding definition file (DEF file). For example, if the name of the XML file is MyFile.XML, the name of the definition file is MyFile.xml.DEF. Briefly said, the definition file contains the definition of all the characteristics that are referred to in the XML file.

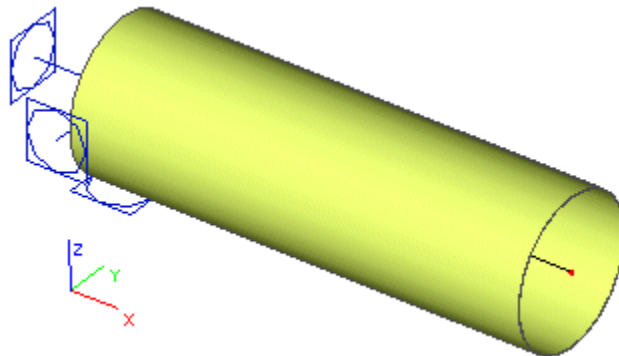
Manual XML update

And now, we may demonstrate the principle of the XML update function. Let us assume that we have a tool that can read and modify the XML file. Using this tool we can change the value of the parameters in the XML file. Let us assume that the value of the Diameter was changed to 0.8 m.

Note: Later we will show how exactly this can be done. For the time being, let us take it for granted that the tool changed the XML file and modified parameter Diameter.

Let us go back to Scia Engineer where our original project is still open. Call function **File > Update > XML file** and browse for our modified file.

The program updates the data of the project (in our example - the values of the parameters) and we can see the result on the screen. The tube has increased its diameter four times.



Automatic XML update

Now, let us return to the idea of an external tool that is capable of modifying the XML file. Probably every engineering office uses a kind of spreadsheet processor such as MS EXCEL. Taking into account the fact that the Microsoft's product is probably the most widespread tool available in the market, we will use it in our example.

List of operations carried out by the EXCEL application

The EXCEL "application" will do the following operations:

- A. it will take our input of the diameter and thickness of the cross-section,
- B. it will generate the appropriate XML file,
- C. it will open the Scia Engineer project with the cantilever in question,
- D. it will update the cross-section using the values from the generated XML file,
- E. it will perform the calculation,
- F. it will generate output document containing the calculated deformation of the cantilever,
- G. it will import this document into the EXCEL sheet,
- H. it will show the vertical displacement of the free end of the cantilever directly in the EXCEL sheet.

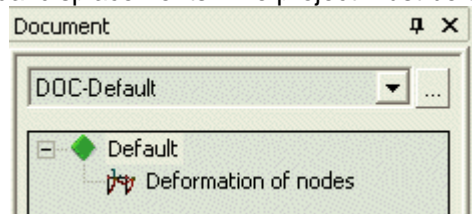
Note: The EXCEL application could naturally display also other results. In the interests of brevity, clarity, and comprehension, we will deal in our example just this single result value.

We will prepare a VBA script (run from a XLS sheet) that will perform all the described operations. Scia Engineer as a calculation engine will be running "somewhere" on the background. The user won't even spot it on the screen and will be able to think that the EXCEL sheet itself does everything.

To start with, we have to pre-prepare something in Scia Engineer.

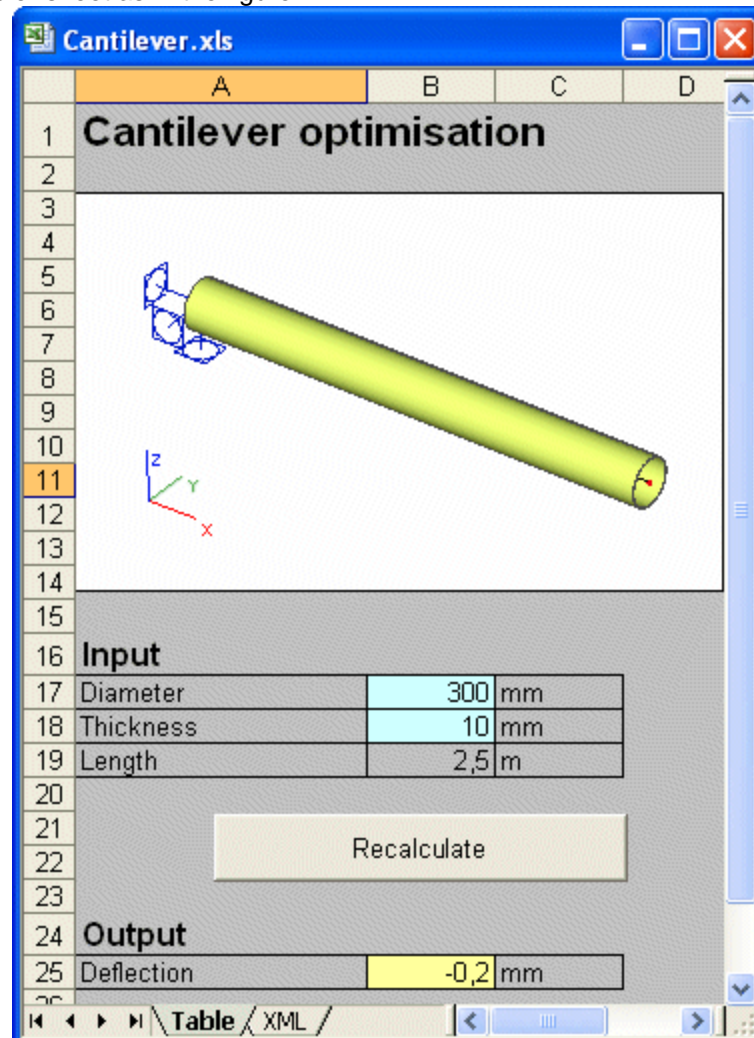
First, we have to generate the .XML "template" file and the corresponding .DEF file. This was already done earlier in our example and we can simply take the produced .XML file. Unless we decide to use different folder (different then Scia Engineer project folder) for our EXCEL application, we do not have to bother about the .DEF file. It will be in the right place by default.

Second, we have to create a document in our Scia Engineer project. The document must contain the table with calculated nodal displacements. The project must be then saved.



Let us now start MS EXCEL, create a new document and define two sheets. Let us name them "Table" and "XML". The first one (Table) will be our "User interface", i.e. it will be the sheet that will be used by the user. The other one (XML) will be used as an auxiliary sheet to hold the contents of the XML file we have to generate (Please note that there are other possibilities (in terms of programming techniques) to store the contents of the XML-file-to-be-generated that may prove more efficient in terms of speed for large projects with large XML files. Our example is quite a simple one and we may afford to store the data in the sheet).

Design the "Table" sheet as in the figure:



You can see that the sheet takes two values as input (the blue cells B17 and B18) and displays one value as output (the yellow cell B25). The sheet contains a simple illustrative picture (scanned from Scia Engineer) and a button that starts all the operations listed above.

The second sheet holds the contents of the XML file that is needed for the XML update functionality of Scia Engineer.

	A	B
1		<?xml version="1.0" encoding="UTF-8" standalone="yes"?>
2		<project xmlns="http://www.scia.cz">
3		<def uri="CantileverParameters.xml.def">
4		<container id="{5C62EA11-86D4-11D4-B3AB-00104BC3B531}" t="EP_Parameters.EP_Param.1">
5		<table id="B4DDEC1E-5869-4B8C-8B4B-1A8C07B76699" t="EP_Parameters.EP_Param.1">
6		<h>
7		<h0 t="Name">
8		<h1 t="UniqueID">
9		<h2 t="Type">
10		<h3 t="Evaluation">
11		<h4 t="Use range">
12		<h5 t="Value"></h>
13		<obj id="1" nm="Diameter">
14		<p0 v="Diameter">
15		<p1 v="{FDD9298F-1A51-406B-9887-8C9302981A47}">
16		<p2 v="11" t="Css length">
17		<p3 v="0" t="Value">
18		<p4 v="0">
19		<p5 t="">
20		<h>
21		<h0 t="Real ">
22		</h>
23		<row id="0">
24	d	<p0 v="0,25">
25		</row>
26		</p5></obj>
27		<obj id="2" nm="Thicknes">
28		<p0 v="Thicknes">
29		<p1 v="{9E8E662D-4B12-43EE-8A56-A33948D079A1}">
30		<p2 v="11" t="Css length">
31		<p3 v="0" t="Value">
32		<p4 v="0">
33		<p5 t="">
34		<h>
35		<h0 t="Real ">
36		</h>
37		<row id="0">
38	t	<p0 v="0,005">
39		</row>
40		</p5></obj>
41		</table>
42		</container>
43		</project>

The XML content is in the second column (B). The first column (A) is almost(!) empty. It just contains letters "d" and "t" in rows where the values of the diameter and thickness are stored. This arrangement was selected with regard to the algorithm programmed in our sample VBA script. The simplest way to fill in this "XML" sheet is to open the exported **CantileverParameters.xml** file in a simple text editor, remove all blank lines and copy+paste it to the second column of the "XML" sheet. The blank lines have been removed to allow for a very simple VBA script.

Now we have to define the procedure that is triggered by clicking the button [Recalculate] from the "Table" sheet. The procedure may look like:

```

Private Sub Recalculate_Click()
' Open XML file for writing
Dim fs, f
Set fs = CreateObject("Scripting.FileSystemObject")
Set f = fs.CreateTextFile("E:\SCIA\EsaData\CantileverParameters.xml",
True)
' we assume that the data are stored in folder E:\SCIA\EsaData
' Generate XML file using the two input values
Dim SomethingToWrite As Boolean
SomethingToWrite = True
Dim mystring As String
Dim i As Integer
i = 1
Do
mystring = Worksheets("XML").Cells(i, 2).Value
If mystring <> "" Then
If Worksheets("XML").Cells(i, 1).Value = "d" Then
' this line stores the value of the diameter
mystring = " <p0 v="" & Str(Worksheets("Table").Cells(17, 2).Value /
1000) & """/>"
End If
If Worksheets("XML").Cells(i, 1).Value = "t" Then
' this line stores the value of the thickness
mystring = " <p0 v="" & Str(Worksheets("Table").Cells(18, 2).Value /
1000) & """/>"
End If
f.WriteLine (mystring)
i = i + 1
Else
SomethingToWrite = False
End If
Loop While SomethingToWrite = True
f.Close
' Invoke the update of the project +
' Run the calculation+
' Export the document into XLS file
Shell ("D:\ESA\Esa_XML.exe LIN E:\SCIA\EsaData\Cantilever.esa
E:\EsaData\SCIA\CantileverParameters.xml /tHTML
/oE:\SCIA\EsaData\CantileverParameters.xls")
'Assumptions:
' Scia Engineer installed in folder D:\ESA
' project file stored in fodler E:\SCIA\EsaData
' XML file generated to folder E:\SCIA\EsaData
' Note:
' File Esa_XML.exe is an integral part of Scia Engineer installation
' Parameters of "SHELL" command
' D:\ESA\Esa_XML.exe = location of Esa_XML.exe
' LIN= linear calculation
' E:\SCIA\EsaData\Cantilever.esa = original project file
' E:\EsaData\SCIA\CantileverParameters.xml = XML file for the update
function
' /tHTML /oE:\SCIA\EsaData\CantileverParameters.xls = export the
document stored inside the project file as HTML file with location and
name E:\SCIA\EsaData\CantileverParameters.xls

Application.Wait (Now + TimeValue("0:00:15"))

```



```
' wait in order to guarantee that Scia Engineer completes the calculation
before the control of the operation is returned to the VBA script
' Read result into XLS sheet +
' Insert the result value into the appropriate cell of the "Table"
sheet
Dim myresult As Variant
Workbooks.Open Filename:="E:\SCIA\EsaData\CantileverParameters.xls"
myresult = Sheets("CantileverParameters").Cells(8, 5).Value
' Cell (8,5) of the "document file" contains the required vertical
displacement
' to find out this, you have to open the generated XLS file manually
(only once) and note down the cell coordinates
Workbooks("CantileverParameters.xls").Close
Worksheets("Table").Cells(25, 2).Value = myresult

End Sub
```

Now we can start playing with the EXCEL application. Just type the two input parameters, click button [Recalculate] and (after a few seconds) you can see the vertical displacement shown in the "Deflection" cell.

Note: Please, consider this chapter to be only a brief introduction to the XML update and contact SCIA support team for more information and/or guidance.

Graphical format

Graphic format

Export

Scia Engineer enables the user to export the graphical representation of the project into a graphical file. This type of export usually work with the data that are displayed in the graphical window of Scia Engineer at the moment the export procedure is started. Formats VRML, DXF and DWG are exceptions as they allow the user to export the whole project and even specify special export parameters.

[The procedure to export data into a graphical file is given in a separate topic.](#)

Various graphical formats are available in Scia Engineer:

BMP	Standard Windows bitmap
VRML	Virtual Reality Modeling Language (V-R-M-L) A standardised language used to transmit information about 3D images over the web (WWW). Since the image is built up on the local computer VRML information transmission is more efficient than sending the actual image. For more information on this format read below the table.
EP3	Internal graphical format of Scia Engineer
EMF	Extended (Enhanced) Windows Metafile Format
WMF	Windows Meta File (W-M-F) A graphic file format used by Microsoft to transfer graphics between Windows applications. Both bitmap and vector graphics are supported by WMF. A variation on WMF is and extended version known as EMF.
3D DWG AutoCAD R14, 2000 3D DWG AutoCAD R11 – 2006	DWG (AutoCAD) format.
2D DWG AutoCAD R14, 2000	Please note that only versions R14 and 2000 are supported.
3D DXF AutoCAD R14, 2000 3D DXF AutoCAD R11 - 2006	DXF (e.g. AutoCAD) format.
2D DXF AutoCAD R14, 2000	Please note that only versions R14 and 2000 are supported.

Import

For special graphical formats DXF, DWG, VRML also the import procedure is supported in Scia Engineer. This is described in [separate topics](#).

VRML

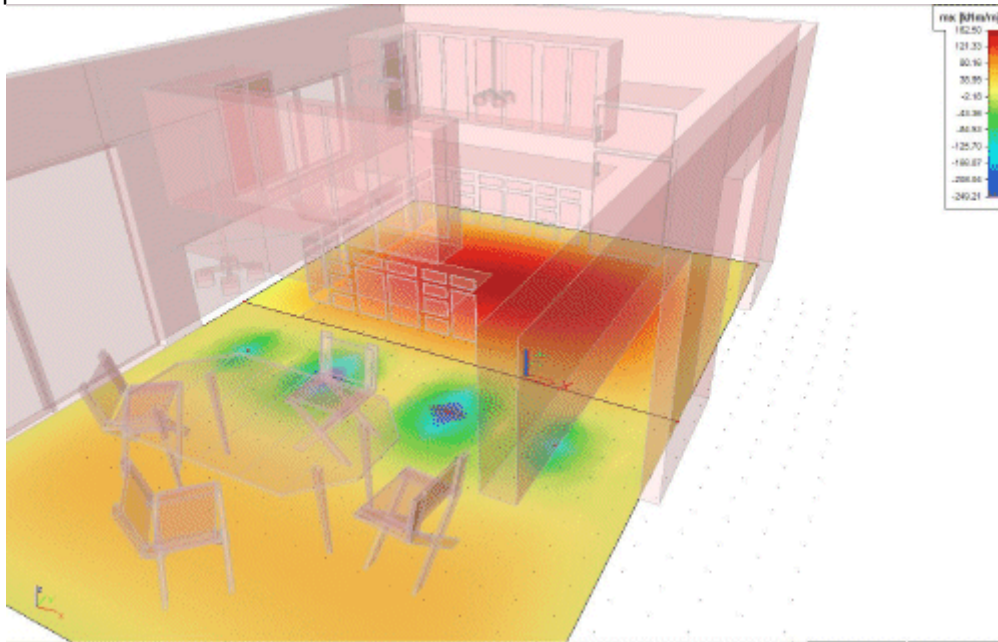
This format represents a powerful tool enabling the users to publish virtual objects via VRML and also import them into Scia Engineer. Vast number of CAD software applications support the export of VRML files (AllPlan, AutoCAD, CEA Systems, etc.). The import of VRML files is done the same way as for DXF and DWG files. For example, the imported data can be used as a 3D grid or as a background. They are not taken into account for the analysis.

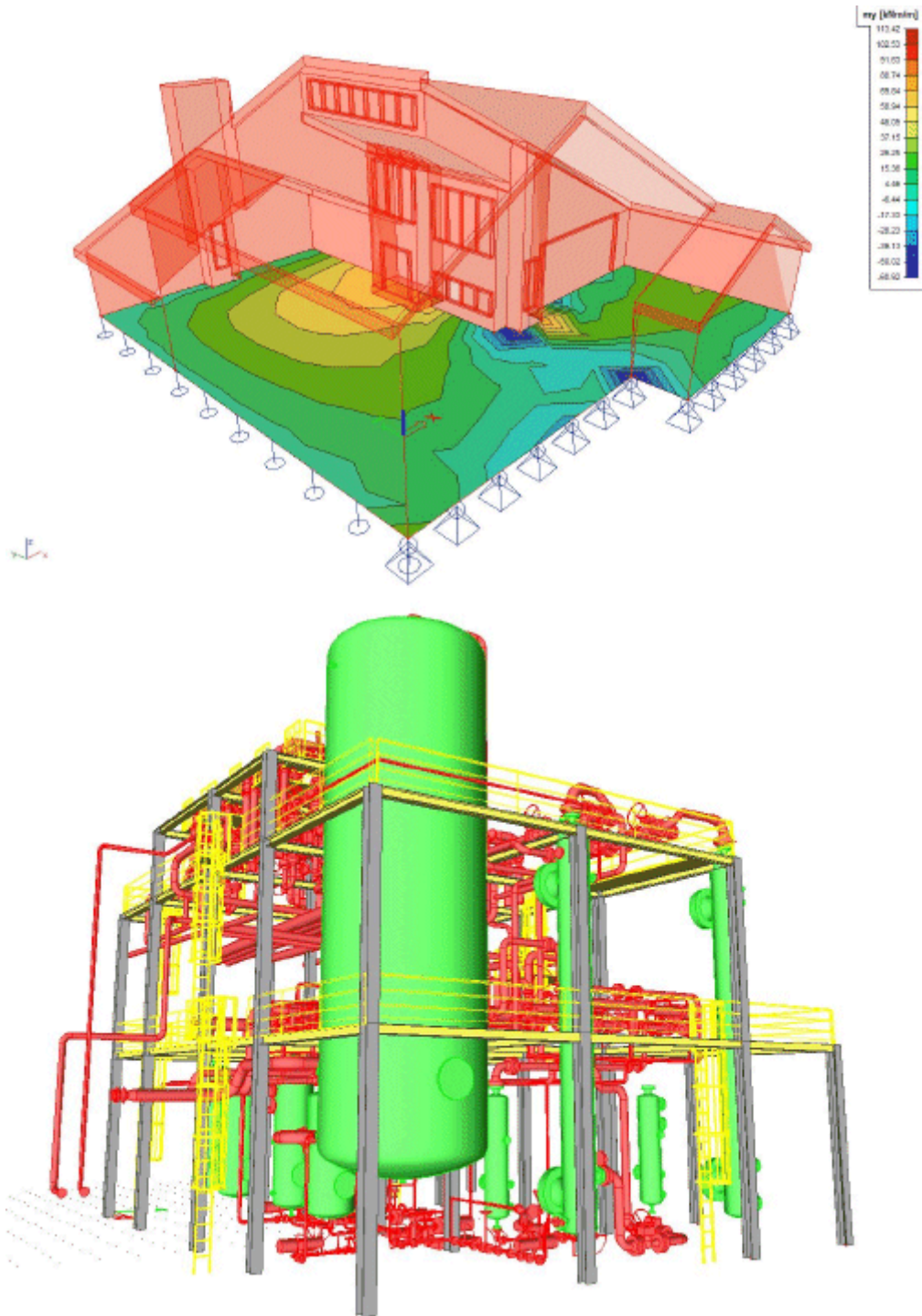
The virtual objects can be used:

- for the visualization architectural elements sent from 3rd parties,
- as a drawing tool – snapping functionality,
- for pro-active viewing of changes and consequences due to optimisation or structural adaptations (e.g. deformations),
- for visualization of the analysed structure in its environment,
- for the visualization of piping generated by Plant design applications,
- etc.

The virtual objects can afterwards be exported again to VRML format file for 3D publishing of the structure including the results of analysis.

Examples:





Graphic format export

The procedure to export data into a graphical file

1. Start menu function **File > Export > Graphic format**.
2. Select the [format of the graphical file](#).
3. Specify the required output folder.
4. Type the name of the file.
5. Complete the export.
6. **Note:** A special dialogue opens for DWG and DXF formats in which you can specify some [export parameters](#).

Export and import of DXF, DWG and VRML

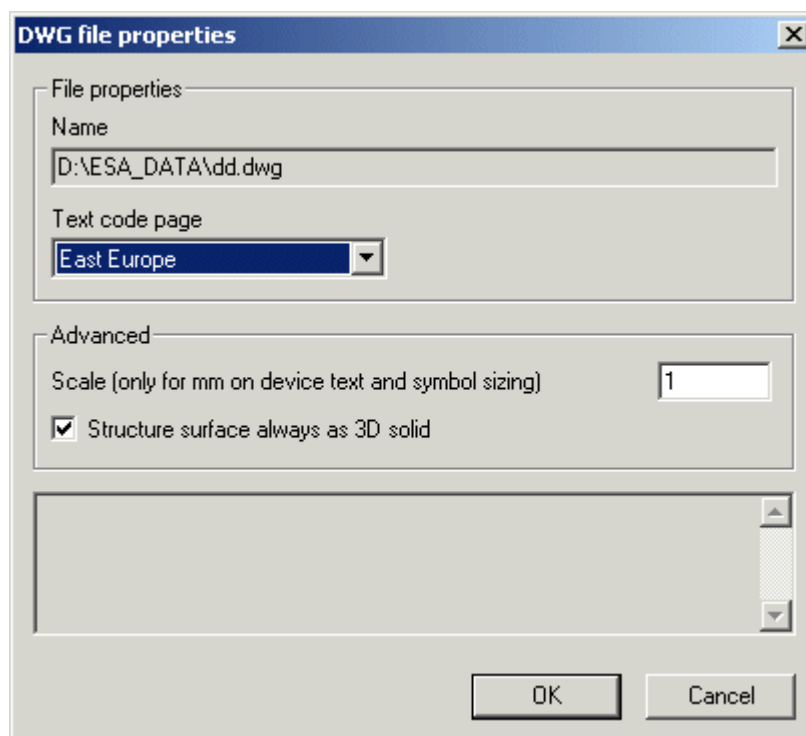
Export from the graphical window

The procedure to export DXF / DWG file from graphical window

- 1) Use one of the following functions:
 - a) **File > Export > Graphic format.**
 - b) Right-mouse-button-pop-up menu > **Save picture to file.**
- 2) It is possible to choose from the following formats:
 - a) 3D DWG AutoCAD R11 - 2007 (*.dwg)
 - b) 2D DWG AutoCAD R14, 2000 (*.dwg)
 - c) 3D DXF AutoCAD R11 - 2007 (*.dxf)
 - d) 2D DXF AutoCAD R14, 2000 (*.dxf)

(Note: this list is only informative, available options may vary from version to version).
- 3) When you enter the file name and path, confirm with button **[Save]**. Another dialogue is opened on the screen where you can define some details of the export.

•



- 4) Adjust the properties (see below) and confirm with **[OK]**.

Export properties

File properties	group name
Name	Pathname of the exported file.
Text code page	Windows ANSI or Eastern Europe coding can be adjusted.
Advanced	group name
Scale	Scale for text and graphical symbols that will be in the exported drawing.
Structure surface always	If ON, the individual members will be exported including the surface.

as 3D solid

Note: 2D DXF and 2D DWG export does not support option Structure surface always as 3D solid, which is quite understandable.

Note: See also Saving the picture into an external file.

Export of FE mesh to AutoCAD as 3D-face

It is possible to export the generated finite element mesh as 3D-face to both DWG and DXF file. To do so, the mesh must be displayed in Scia Engineer. There is no other action required. If the FE mesh is displayed in the graphical window at the moment the export to DWG/DXF is started, the mesh is automatically exported as a separate layer called "3D-FACE". By default, this layer is hidden. In order to see it in AutoCAD, the layer must be switched ON manually.

Export from the picture gallery

The pictures from the **Picture gallery** may be exported into DWG or DXF file in the same way as [from the graphical window](#).

The procedure to export DXF / DWG file from the Picture gallery

1. Open the **Picture gallery**.
2. Select the picture to be exported.
3. Click icon **Save picture to file**.
4. Input the file name and path.
5. Adjust file parameters (see [Export from graphical window](#)).
6. Confirm with **[OK]**.

Export from the Paperspace gallery

The pictures from the paperspace may be exported into DWG or DXF file in the similar way as from the [graphical window](#).

The procedure to export DXF / DWG file from the Paperspace gallery

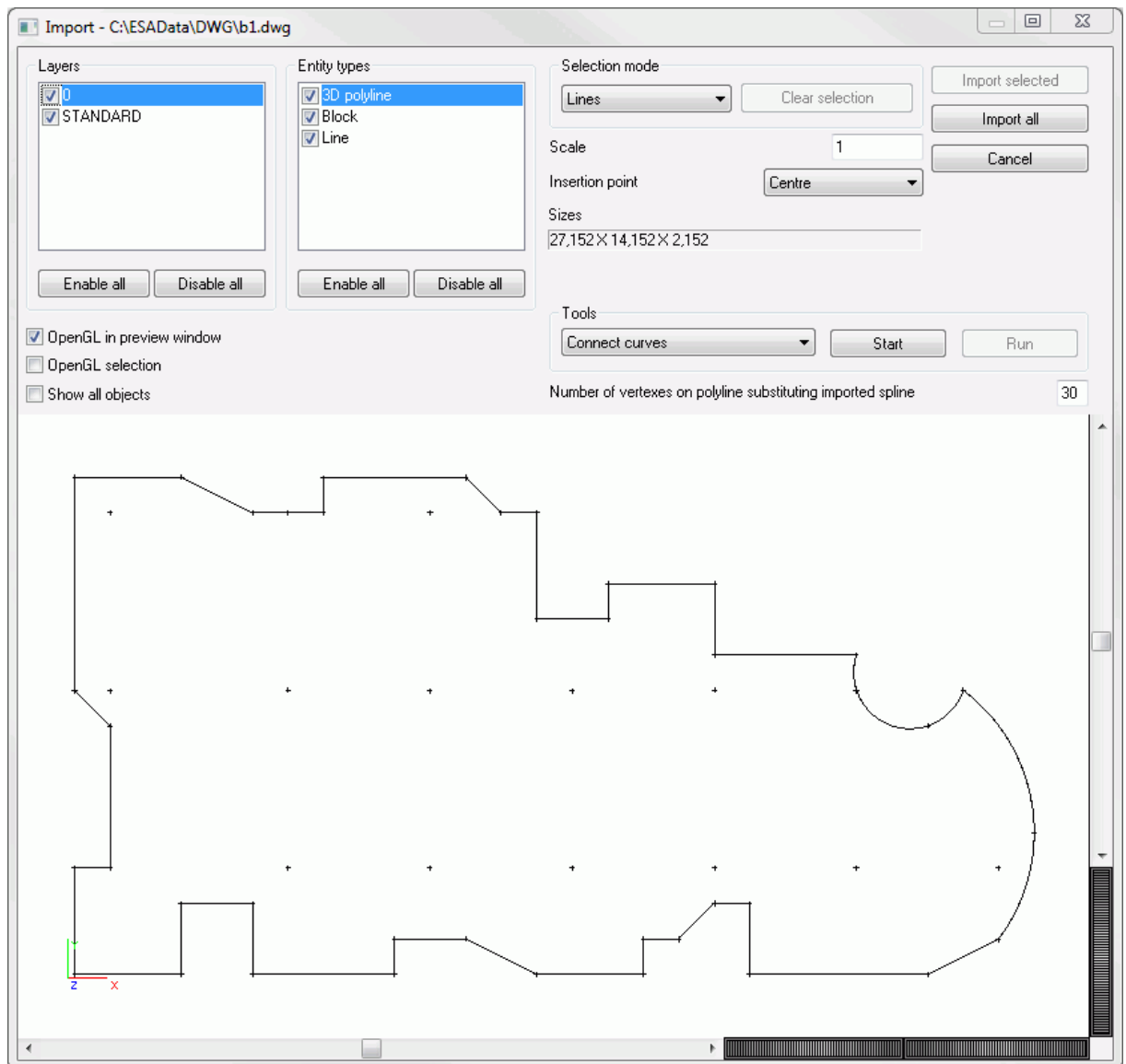
1. Open the **Paperspace gallery manager**.
2. Select the picture to be exported.
3. Click icon **Save picture to file**.
4. Input the file name and path.
5. Adjust file parameters (see [Export from graphical window](#)).
6. Confirm with **[OK]**.

Note: The export from Paperspace gallery is limited to 2D DWG and 2D DXF format.

Import into the graphical window

Procedure for the import of DWG, DXF or VRML file into the graphical window

1. Start the tree menu function **Structure > Drawing tools > Import DWG, DXF, VRML 97 file**.
2. The **Open file** dialogue is opened on the screen.
3. Browse for the file and confirm.
4. A special import dialogue is opened on the screen.



5. Adjust the required parameters in dialogue and perform the import itself – see below.

Layers

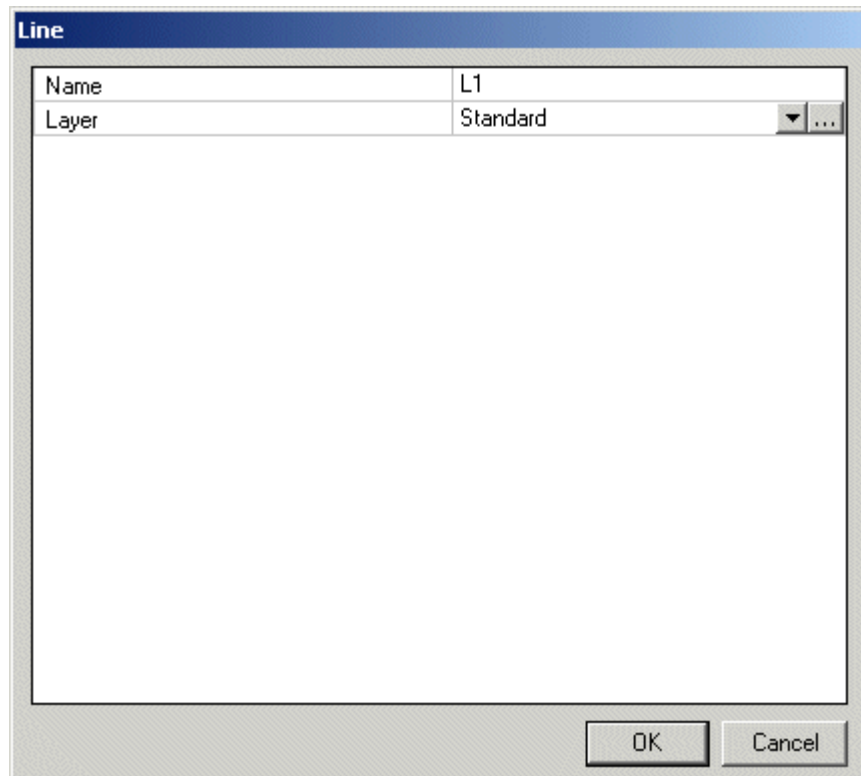
This list box contains the layers that were defined in the original DWG/DXF file. Only selected layers are shown in the preview window of the **Import** dialogue.

Entity types

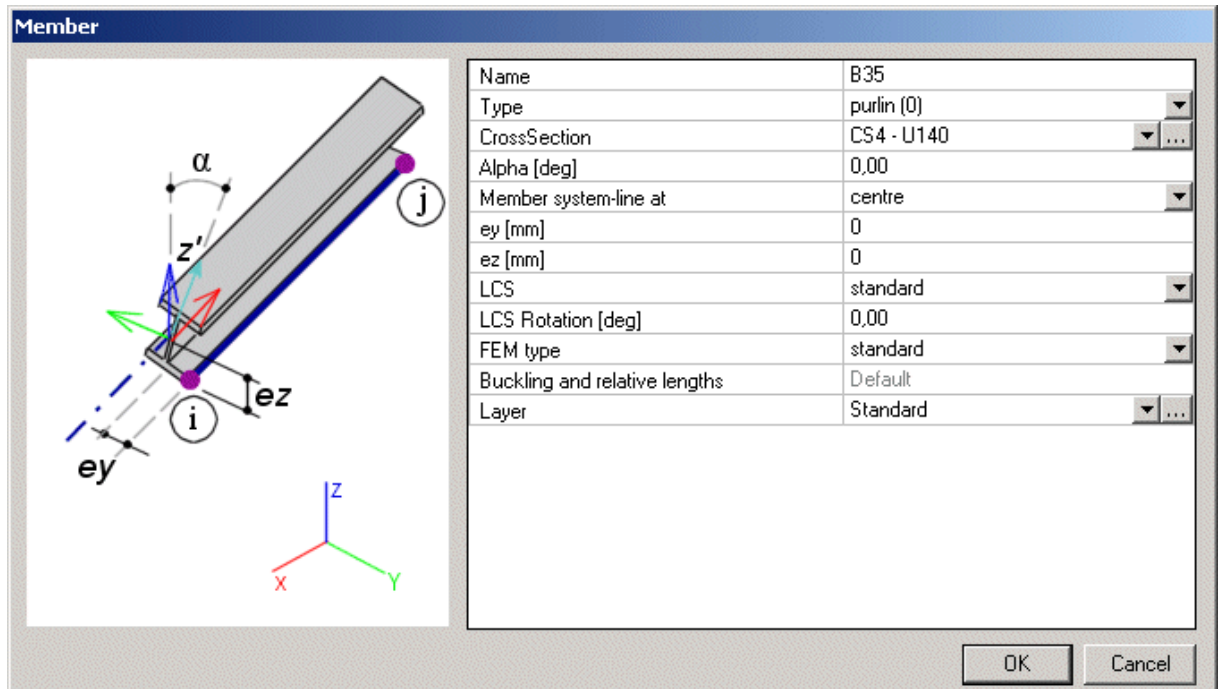
This list contains available entity types. Only selected types are shown in the preview window of the **Import** dialogue.

Selection mode

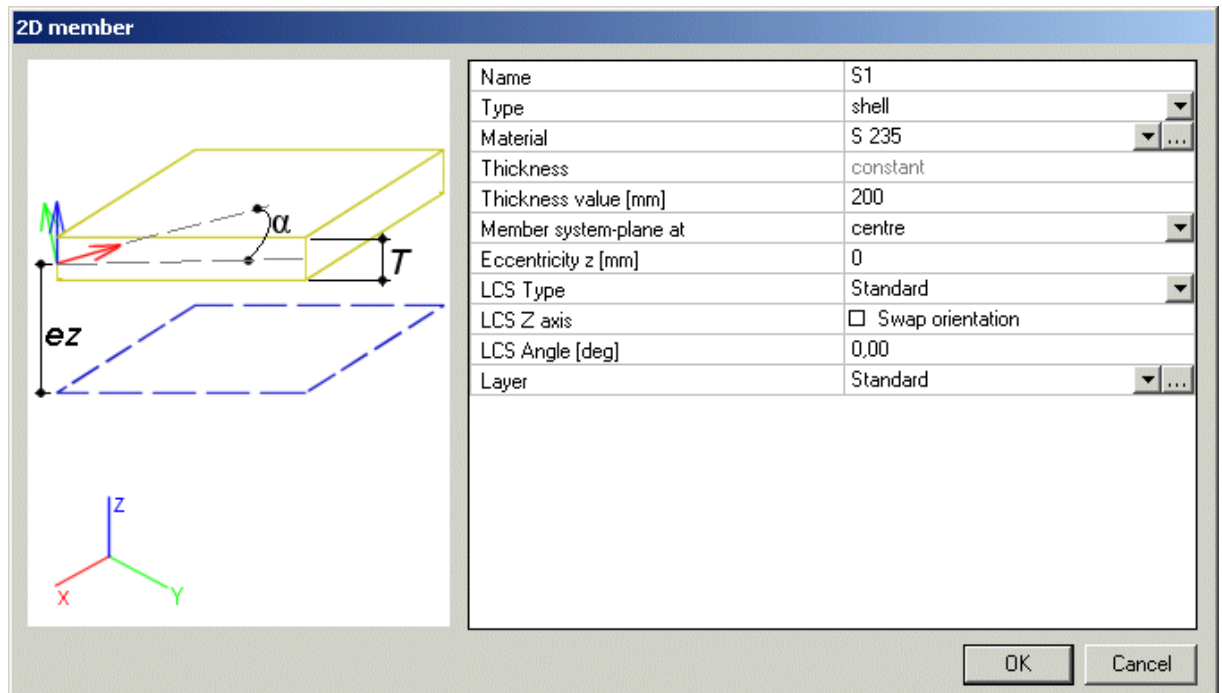
Lines	The drawing is read as lines. The lines may be assigned to particular layers. On pressing the "Import" button, the property dialogue is opened and you may specify the required parameters.
--------------	---

**Beams**

The drawing is read as 1D members. On pressing the "Import" button, the property dialogue is opened and you may specify the required parameters.

**Slabs**

The drawing is read as slabs. On pressing the "Import" button, the property dialogue is opened and you may specify the required parameters.

**Solids**

The drawing is read as solid objects that are displayed together with the analysed structure but are neglected during the calculation.

Note: When slabs are to be imported, it is required that the closed polygon was created already in the file that will be imported.

Scale

The scale for the import. It may be necessary when the drawing is not in SI units. The item provides for the transformation from "imaginary" units of the DWG/DXF file and metres (used in Scia Engineer as the basic unit).

Note: If the scale is set to 1 (one), Scia Engineer assumes the data to be stored in metres.

Insertion point

The user can define the insertion point:

Centre

The centre of the model in the imported file is selected as the insertion point (where the cursor is) and by this point the user can place the imported model into the graphical window. The "centre" is the centre of a bounding rectangle circumscribed around the imported model.

Original

The original insertion point of the imported model is selected as the insertion point (where the cursor is) and by this point the user can place the imported model into the graphical window.

Origin in 0,0,0

The origin of the coordinate system of the imported file is placed into the origin (0,0,0) of the coordinate system in Scia Engineer.

Sizes

This is an informative item, which shows the dimensions calculated from the input scale.

Tools (Connect curves, Connect solids, Solids smoothing)

These tools allow the user to slightly "modify" the DWG/DXF file before it is imported into Scia Engineer.

Connect curves

This tool connects individual lines from the imported file to a polyline. This may be useful to reduce the total number of lines in the imported model. It may also simplify the manipulation with the imported model.

The procedure:

1. Select **Connect curves**.
2. Click **[Start]**.
3. Select the lines to be connected together.
4. Press **[Run]**.
5. Repeat as many times as required.
6. Click **[End]**

Connect solids

This tool connects individual solids from the imported file. Similarly to connecting of lines (above) it can reduce the total number of solids in the imported model and simplify the manipulation with the imported model.

The procedure:

1. Select **Connect solids**.
2. Click **[Start]**.
3. Select the solids to be connected together.
4. Press **[Run]**.
5. Repeat as many times as required.
6. Click **[End]**

Solids smoothing

This tool allows the user to "improve" the imported solid by smoothing the surface, which means reducing the number of edges. This is especially useful if the imported solid was not modelled in an ideal way and contains redundant edges that are not important for the description of the proper shape of the solid.

The procedure:

1. Select **Solids smoothing**.
2. Click **[Start]**.
3. Select the part of the imported solid that is to be smoothed. Alternatively, use the window selection to select everything (click in the top left corner of the preview window and drag the mouse to the bottom right corner, then release the button).
4. Press **[Run]** and specify the smoothness parameter: 0 = the minimum possible number of edges will be used to represent the shape of the imported solid (in some situations it may degrade the shape - e.g, a cube may get rounded corners). 60 = all the edges in the imported model will be preserved. Usually, a few (trial and error) rounds are necessary to find the best value for a particular model - there is no general rule to say which value is the best.
5. Repeat as many times as required.
6. Click **[End]**

Number of vertexes on polyline substituting imported spline

Some CAD applications use a definition of a spline curve that is different from the one used in Scia Engineer. Such splines are imported as polylines. This parameter specifies the number of vertexes on the substituting polyline.

Preview window

The view in the preview can be adjusted using the standard Scia Engineer mouse+key controls (shifted, rotated, zoomed in/out).

Three check boxes are related to the preview window:

OpenGL in preview window

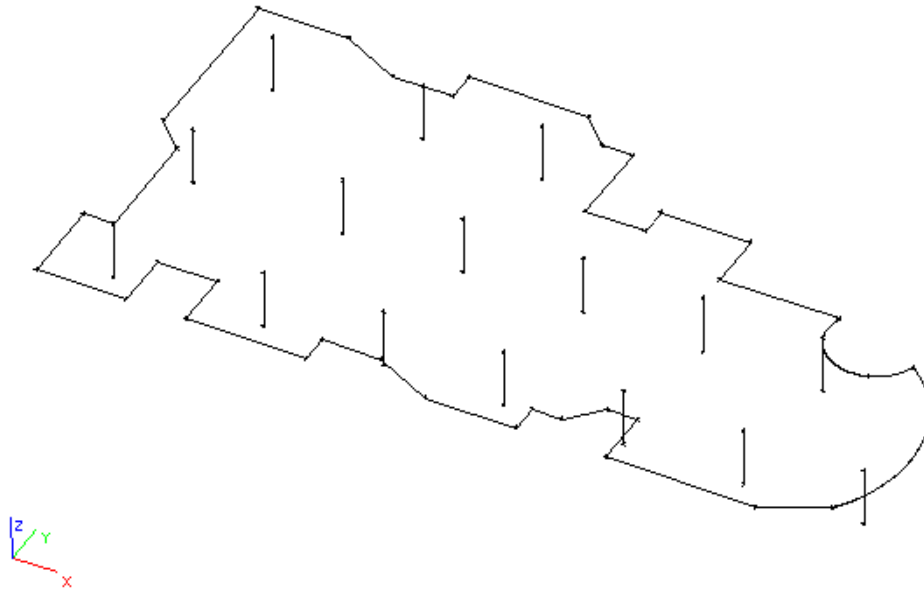
If ON, the parts of the model hidden behind other parts of the model are not drawn. Of OFF, the visibility is not considered and everything is displayed (which means faster response but may cause in some confusions).


OpenGL selection

If ON, the cursor selects the "visible" (the top most) part of the model (if two entities re located over each other). If OFF, the standard selection mode is active.

Show all objects

Some objects from the DWG/DXF that has no equivalent in Scia Engineer (e.g. text) cannot be imported to Scia Engineer. This objects are not shown in the Preview window. If you want to see them, select this option.



Note: Sometimes it may be convenient to import the drawing as lines and only later (i.e. in Scia Engineer graphical environment) transform the lines to 1D members using function Drawing a member in mode Select line ().

Import selected / Import all

This button start the import itself. Either entities selected in the Preview window or All entities from the Preview window are imported as lines or 1D members or slabs (depending on the adjustment of other import parameters). When the button is pressed, the property dialogue for lines or 1D members or slabs (depending on the adjustment of other import parameters) is opened on the screen and you may set the required properties.

Import into the Paperspace gallery

DWG, DXF and VRML files can be imported as a picture or as a drawing.

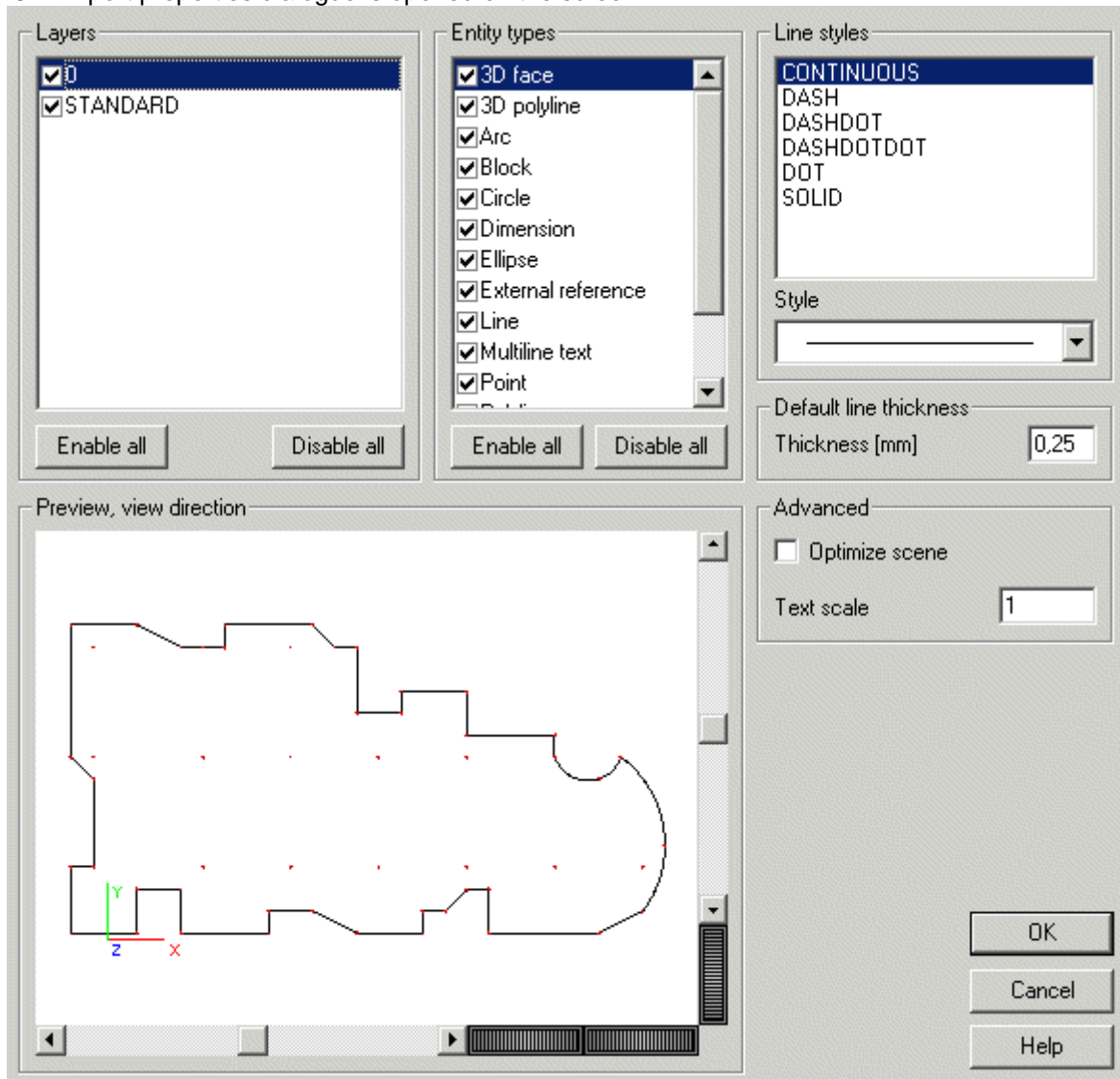
Importing a picture

This function imports the DWG, DXF and VRML file as a "standalone" picture. The picture can be placed anywhere to the drawing, resized, moved, etc.

Procedure to insert a picture from DWG or DXF file

1. Open the **Paperspace gallery**.
2. Open or create a drawing.
3. Start function **Insert picture > Insert picture from DWG or DXF file**.

4. Two file types are available:
 - a. AutoCAD R12, R13, R14, 2000 (*.dxf)
 - b. AutoCAD R12, R13, R14, 2000 (*.dwg)
2. Select the file and open it.
3. Import properties dialogue is opened on the screen.



5. Adjust the required properties (see below) and confirm.
6. Use the mouse to position the picture and the left mouse button to drag the size of the picture on the drawing.

Procedure to insert a picture from VRML file

1. Open the **Paperspace gallery**.
2. Open or create a drawing.
3. Start function **Insert picture > Insert picture from VRML file**.
4. Select the file and open it.
5. Use the mouse to position the picture and the left mouse button to drag the size of the picture on the drawing.

Layers

This list box contains the layers that were defined in the original DWG/DXF file. Only selected layers are shown in the preview window of the Import dialogue.

Entity types

This list contains available entity types. Only selected types are shown in the preview window of the Import dialogue.

Line styles

This window contains all the line styles that are used in the imported file.

Style

This box offer the styles of a line that can be used in the program. They can be assigned to individual line types from the original file.

Default line thickness

Lines in DWG/DXF files may have either specific thickness or default thickness. In order to avoid using unknown thickness lines, the import dialogue enables you to define the numerical value for the default thickness.

Optimize scene

If ON, the lines with the same properties are joined together and drawn at the same time.

Text scale

Specifies the scale of the text.

Preview, view direction

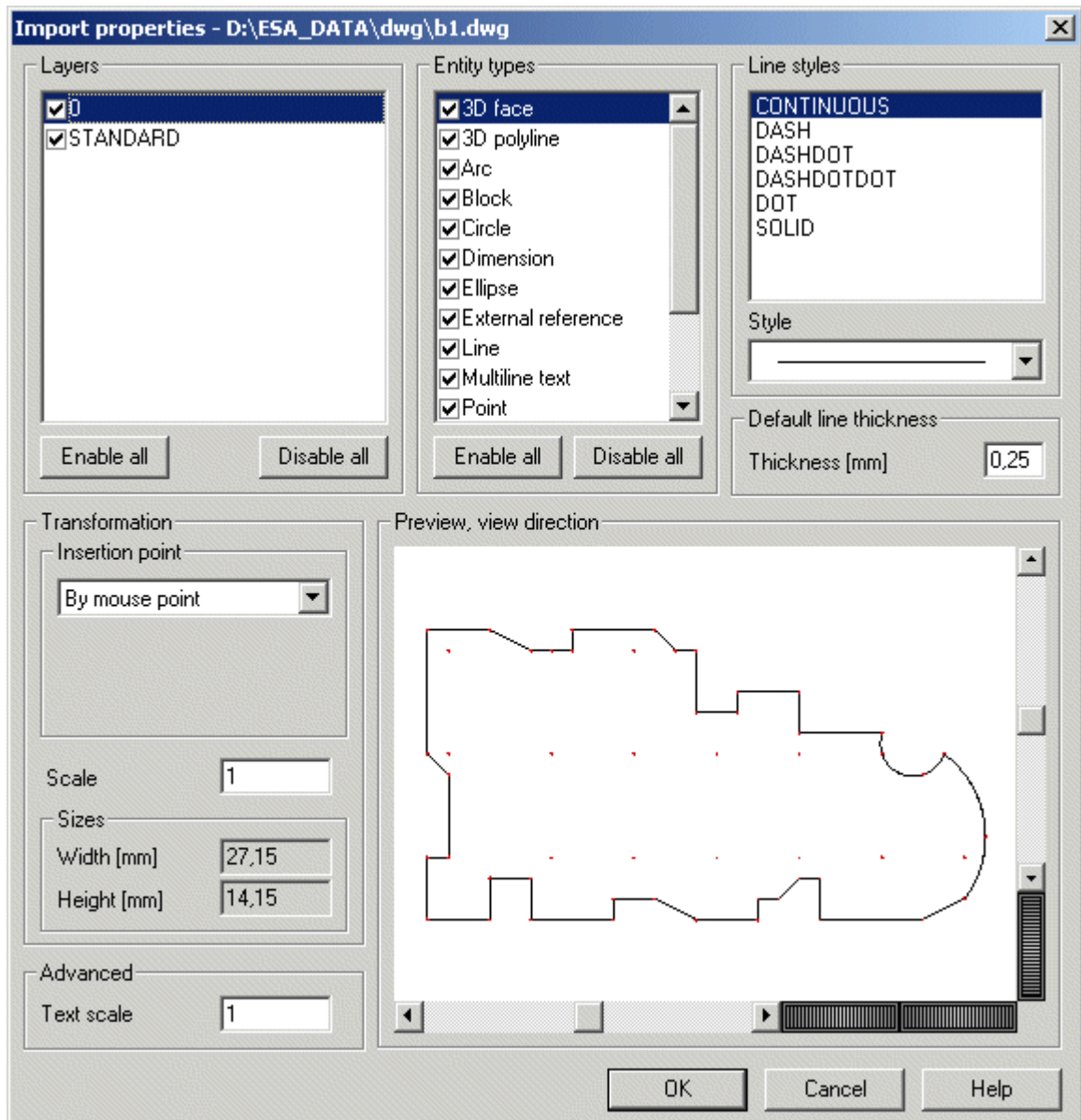
The view in the preview can be adjusted using the standard Scia Engineer mouse+key controls (shifted, rotated, zoomed in/out).

Importing a drawing

This function imports the DWG/DXF file as a drawing, i.e. it is added to the current drawing as a new part of it. The "size" of the imported drawing can be defined through parameter **Scale**.

Procedure to insert a drawing from DWG or DXF file

1. Open the **Paperspace gallery**.
2. Open or create a drawing.
3. Start function **Insert picture > Insert drawing from DWG or DXF file**.
4. Two file types are available:
 - a. AutoCAD R12, R13, R14, 2000 (*.dxf)
 - b. AutoCAD R12, R13, R14, 2000 (*.dwg)
2. Select the file and open it.
3. Import properties dialogue is opened on the screen.



5. Adjust the required properties (see below) and confirm.
6. Use the mouse to position the imported drawing to the current drawing.

Layers

This list box contains the layers that were defined in the original DWG/DXF file. Only selected layers are shown in the preview window of the Import dialogue.

Entity types

This list contains available entity types. Only selected types are shown in the preview window of the Import dialogue.

Line styles

This window contains all the line styles that are used in the imported file.

Style

This box offer the styles of a line that can be used in the program. They can be assigned to individual line types from the original file.

Default line thickness

Lines in DWG/DXF files may have either specific thickness or default thickness. In order to avoid using unknown thickness lines, the import dialogue enables you to define the numerical value for the default thickness.

Transformation – Insertion point

The imported drawing can be placed to a specified place in the current drawing (e.g. bottom left corner, centre, etc.). The meaning of individual options is clear from the name of the option.

Note: When option **By mouse two points** is selected, the input box **Scale** is hidden. The two points define the size of the imported drawing and the **Scale** parameter becomes redundant.

Scale

Specifies the scale for the imported drawing.

Text scale

Specifies the scale of the text.

Preview, view direction

The view in the preview can be adjusted using the standard Scia Engineer mouse+key controls (shifted, rotated, zoomed in/out).

Revit

The Scia Engineer finite element analysis allows for (through the structural CAD model) a direct analysis of plates, walls, frames and any kind of mixed structures of Autodesk® Revit® Structure 2010.

What can be exported and imported?

- Geometry (Revit builds an architectural/structural model and converts it into an analysis model) - Plates, walls, shells, 1D members, columns, bracings.
- Supports (Revit footings are converted into rigid supports in Scia Engineer)
- Loads (Revit inputs loads in Load Cases - these are converted into free loads in Scia Engineer).
- Combinations (combinations are possible but can be ignored in the export to Scia Engineer)

What are the installation requirements?

The link works as a Revit's plug-in, so you must have Revit installed - with or without Scia Engineer.

Scia Engineer user who have also Revit can make modifications in Revit and update them in Scia Engineer or vice versa.

How does it work?

Revit creates the model and it can be exported to Scia Engineer through function **Tools > External Tools > Send model to Scia Engineer**. There are two possible approaches.

1. Through a direct export to Scia Engineer. Scia Engineer is opened and the structure is visualized. The structure can be sent to Revit again, changed there and again updated to Scia Engineer, and over and over again.
2. Through a simple export to a file.

Scia Engineer checks the geometry, generates the FE mesh and performs the analysis. The structure can also be optimized and modified and then updated in Revit:

1. Function **Update from Scia Engineer** can be invoked: the structure in Revit is updated with the new dimensions.
2. After updating the user saves the model to a Revit file and that file is read into Revit.

Changes can be traced in Revit: modifications or added members are highlighted; deletions are listed.

For further info read chapters: [Revit import](#), [Revit export](#), [Revit update](#).

Revit import

The procedure to import data from Revit file

1. Start menu function **File > Import > Revit file**.
2. Browse for the required .r2s file.
3. Confirm the selection of the file.
4. If required by the import procedure, specify the national standard to be used.
5. Finish the import.

Note: The import function creates a new empty project and the specified file is imported into it.

Revit export

The procedure to export data into Revit file

1. Start menu function **File > Export > Revit file**.
2. Specify the required output folder.
3. Type the name of the file.
4. Complete the export.

Revit update

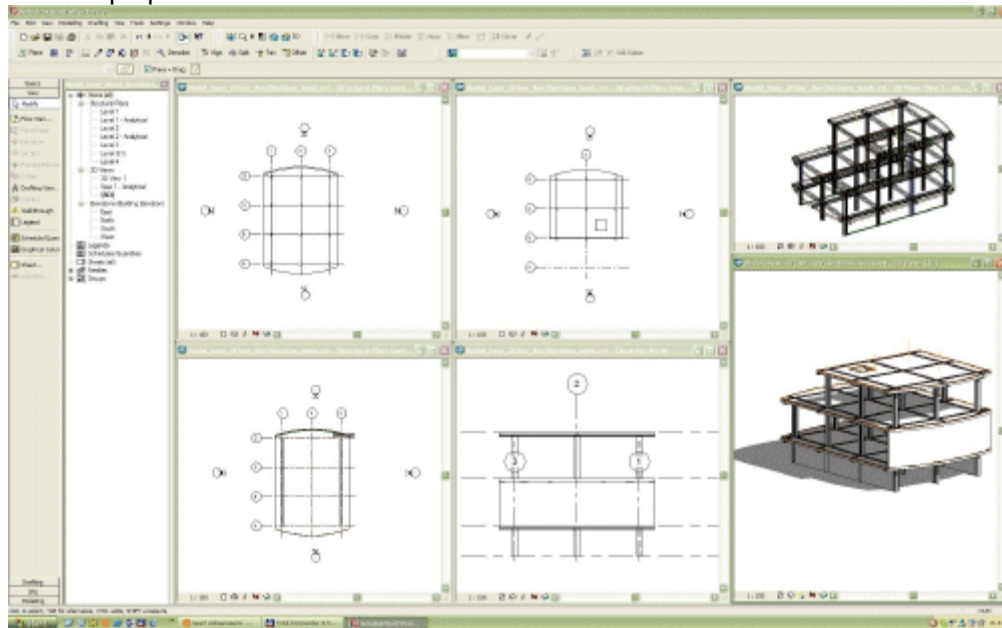
The procedure to update the project through Revit file

1. Open in Scia Engineer the project to-be-updated.
2. Call function **File > Update > Revit file**.
3. Browse for the file to-be-merged.
4. Confirm the selection of the file.
5. The project in Scia Engineer is updated by the data read from the file.

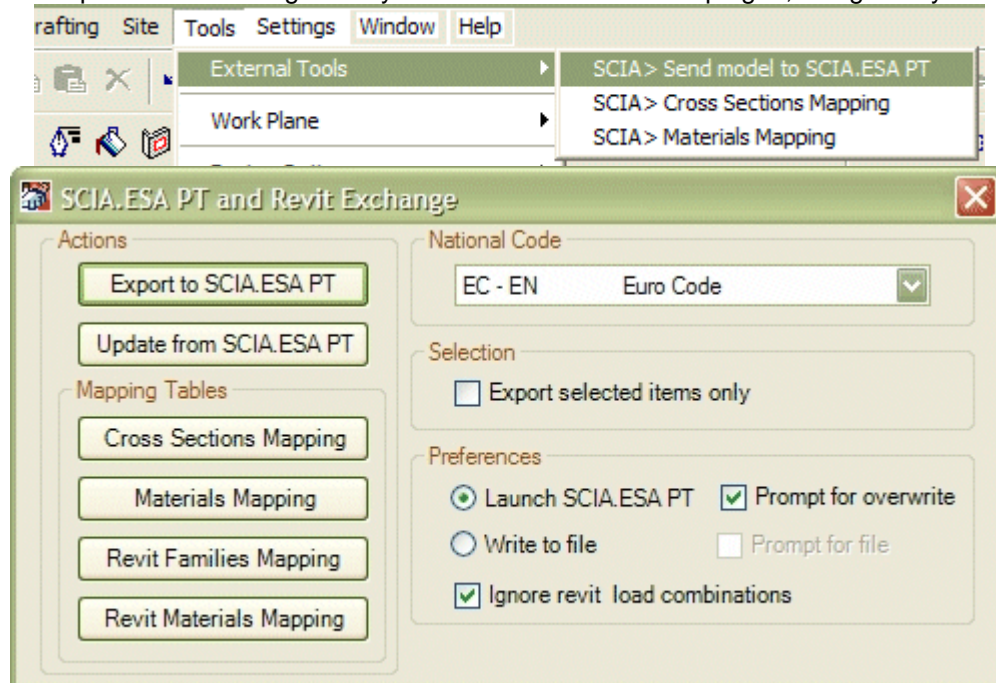
Sample roundtrip procedure

Step 1: Autodesk® Revit® Structure

A 3D model is prepared in Autodesk Revit Structure:



Then it is exported to Scia Engineer by the Revit Structure 3 or 4 plug-in, designed by SCIA:



The export features:

Either the complete model or only a selected part can be exported.

Revit combinations can be exported or ignored. If the latter is the case, new combinations must be defined in Scia Engineer.

The export can be done directly to Scia Engineer, on condition that Scia Engineer is installed on the same computer. The plug-in opens Scia Engineer – both applications will be open.

Modifications done in Scia Engineer can be then sent to Revit. When Revit function **Update from Scia Engineer** is called, the model is accepted in Revit and changes to the model can be traced in Revit Structure.

The export can also be done to an external file. This file with extension *.r2s can be sent to the construction partner. This partner is able to open the mode in Scia Engineer and save it again to the file with the same extension.

Mapping tables must be defined by the user to link the Revit Structure cross-section list to the Scia Engineer cross-section library. The same must be done for the materials defined in Revit Structure and for the Revit cross-sections and material families.

After this primary export, new modifications made in Revit can be re-sent to Scia Engineer.

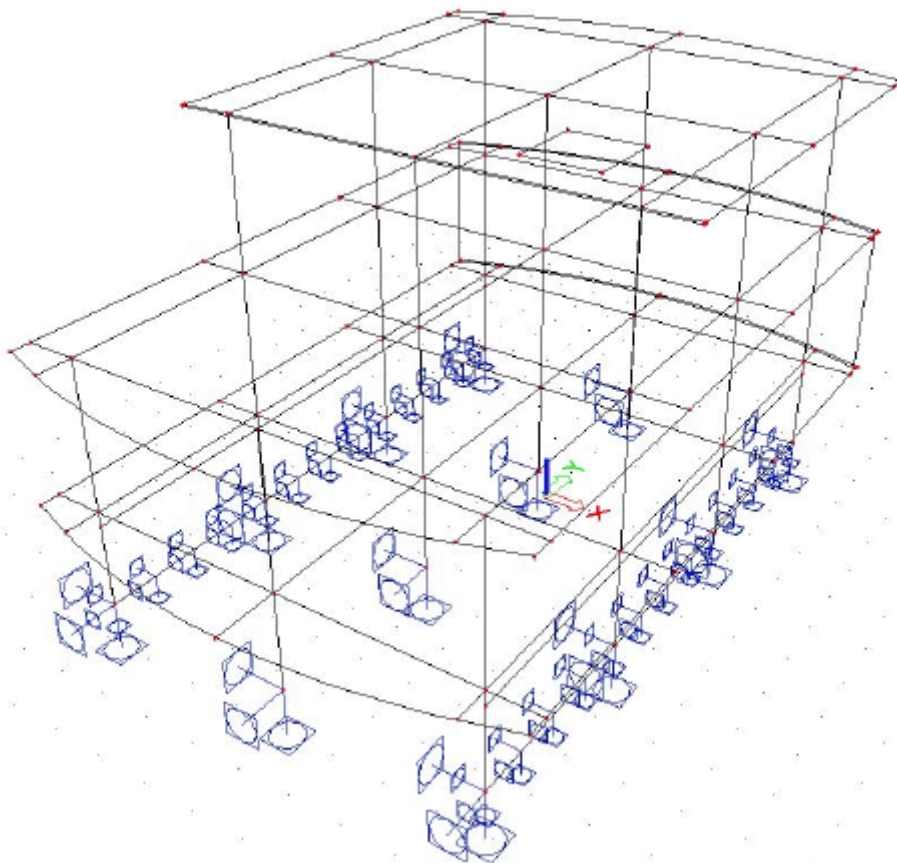
Step 2: Scia Engineer - finite elements analysis software

The model received from Revit Structure can be checked and corrected automatically for analytical anomalies and, if required, modified in Scia Engineer for further analysis.

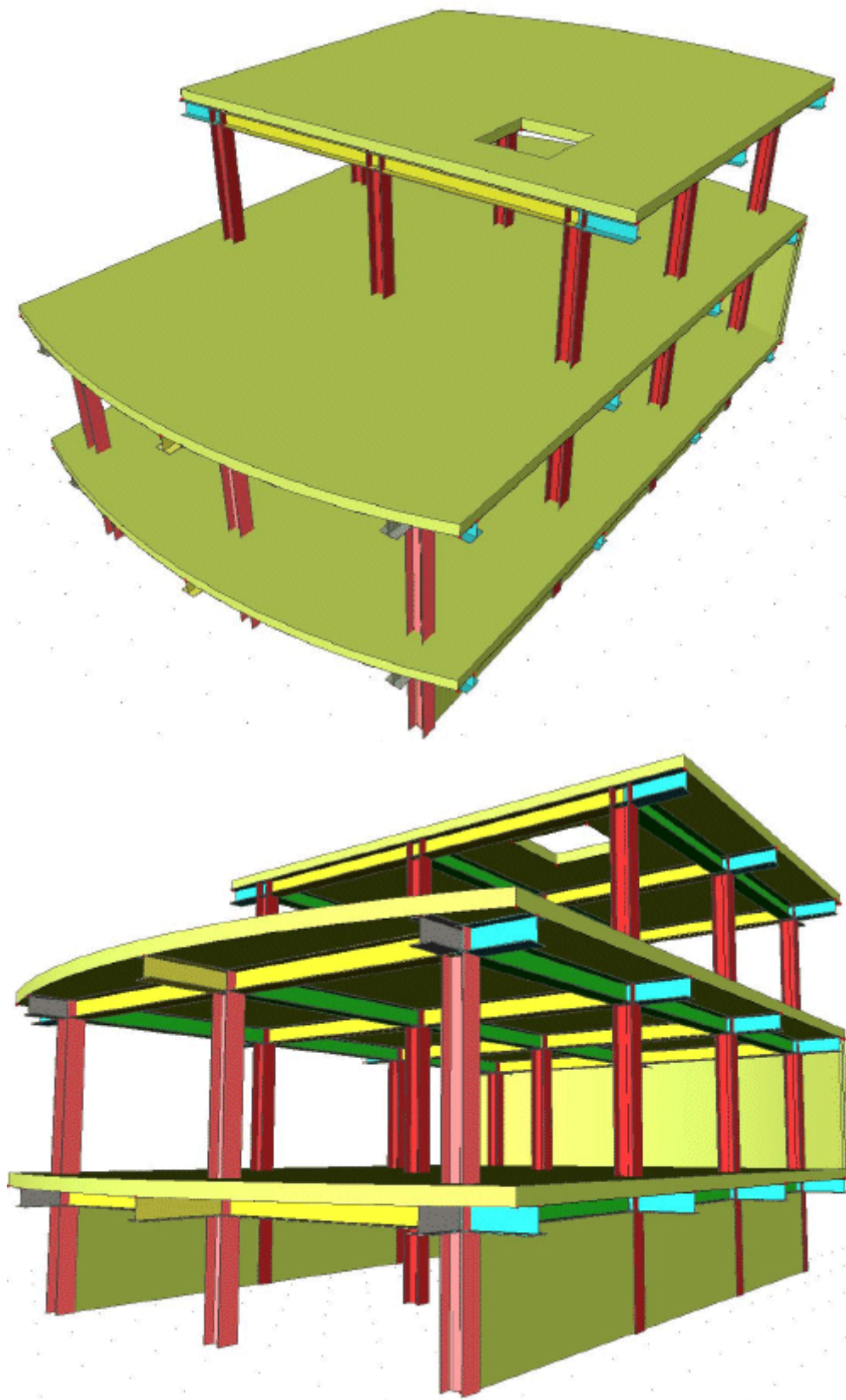
The received model can contain:

- Columns, beams
- Plates
- Holes
- Walls
- Curved slabs
- Loads & load cases
- Supports

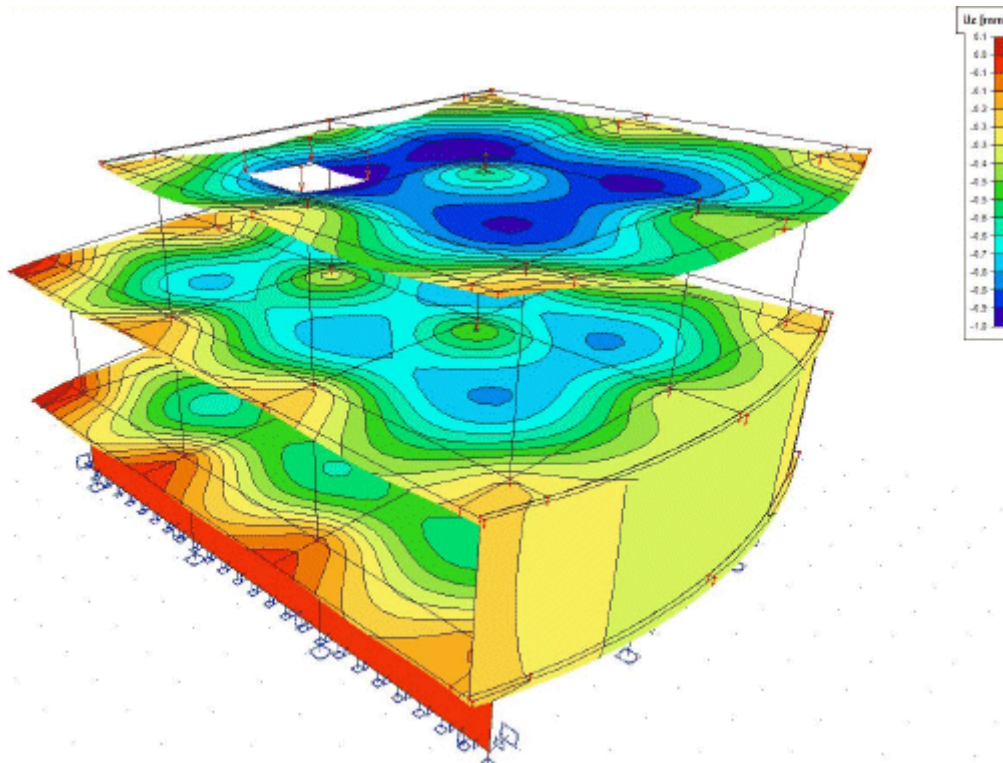
The structural model is automatically converted into an analysis model



At any time, the Scia Engineer Structural model can be reviewed in order to help with visual checks.

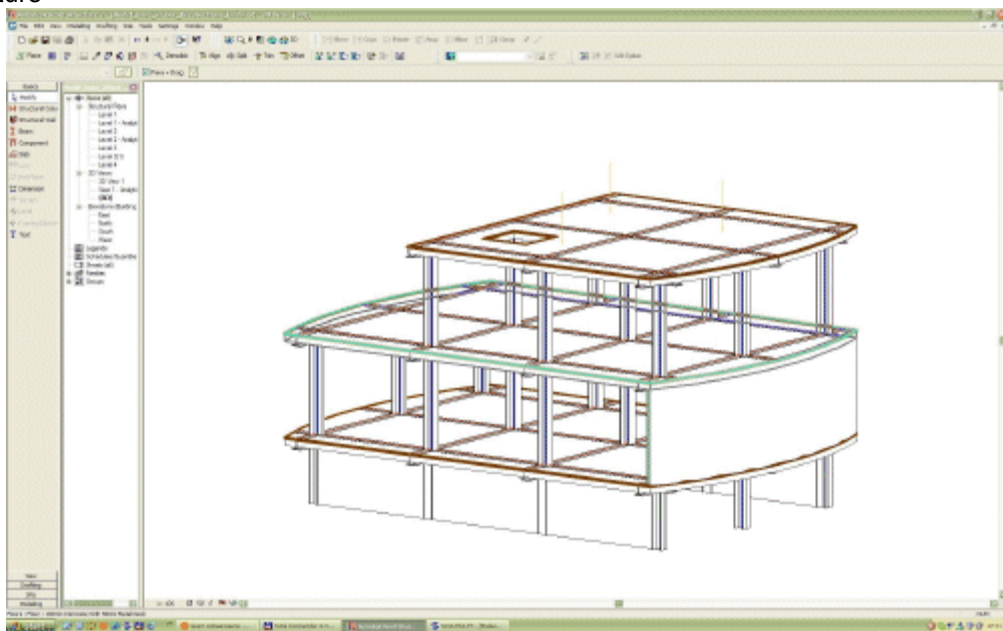


After the consistency of the structure is checked, the structure can be meshed and analysed and the results in the frame elements, plates and shells can be evaluated.



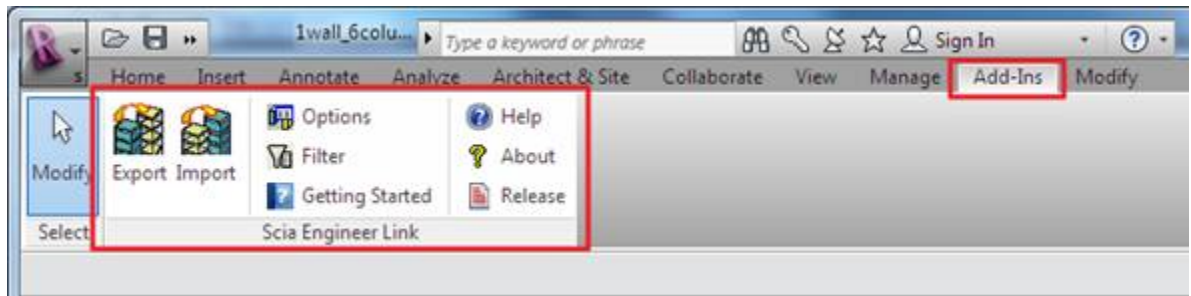
Step 3: Autodesk® Revit® Structure

After the analysis and optimisation the model is sent (and updated) back to Autodesk Revit Structure



Revit link

The Revit link is a tool which allows transferring the Revit project to the Scia Engineer. The link is implemented to the Revit structure applications (as a plug-in). It is displayed in the tab Add-ins. There are buttons and dialogues which provide link between Revit and Scia Engineer.



The supported versions are up to the Revit structure 2012. The setup can be found on the installation DVD of Scia Engineer or on the secured download web site for Scia users.

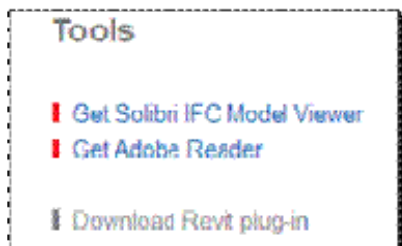
The file format which is used between Revit and Scia Engineer is R2S (Revit to Scia).

Installation and the setup of the Revit link

How to find the Revit link on the web

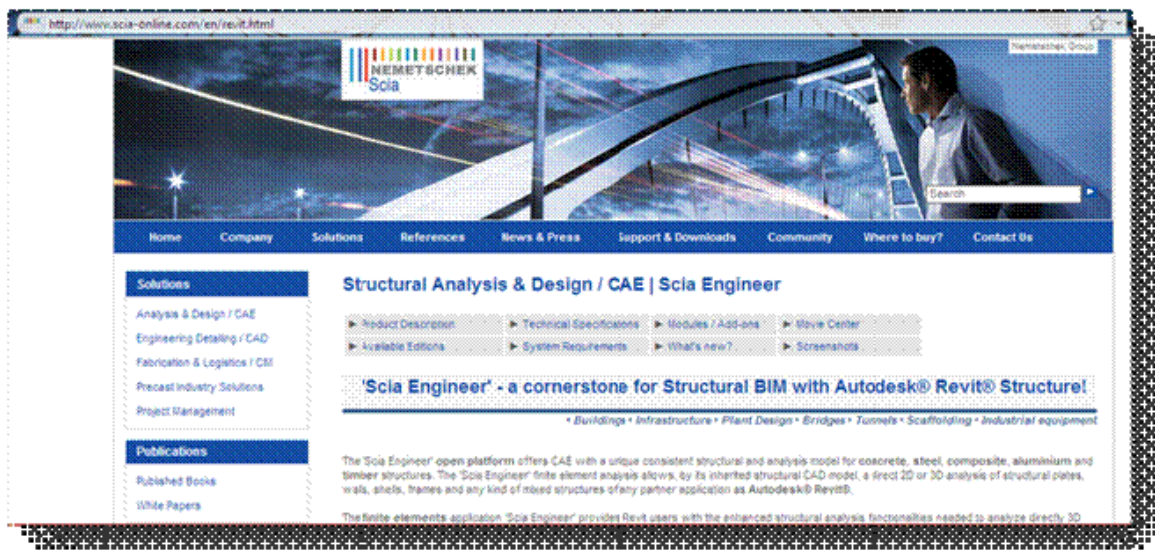
The secured download page is in Service Packs > Scia Engineer. The link can be also found in the setup of Scia Engineer.

Main menu Instalation&software Tools Download Revit plug-in



This tool will start the download web site for the Revit link.

It is <http://www.scia-online.com/en/revit.html>.



How to find the Revit setup on the installation DVD

The folder with the Revit link setup file can be found here:

➡➡➡ Setup DISK1 RevitLink revit_setup.exe



The process of the installation

Installation of the Revit link is a standard tool. During the process is required to define a Revit version and also the correct path where the Revit link should be installed. Then the installation can continue in a standard way.

The plug-in is automatically installed to the selected Revit structure in tab Add-ins.

The installation of the Scia Engineer application itself is not required. The installation of any additional Scia Engineer tool is not necessary.

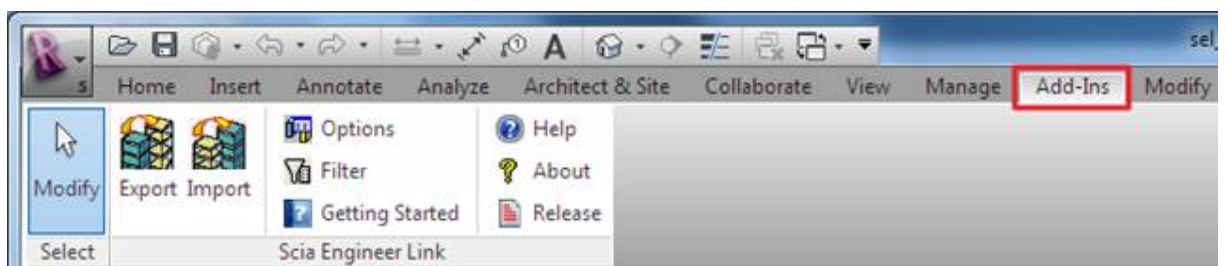
Note

- If there are more Revit structure versions on the computer user have to choose the one for the plug-in.



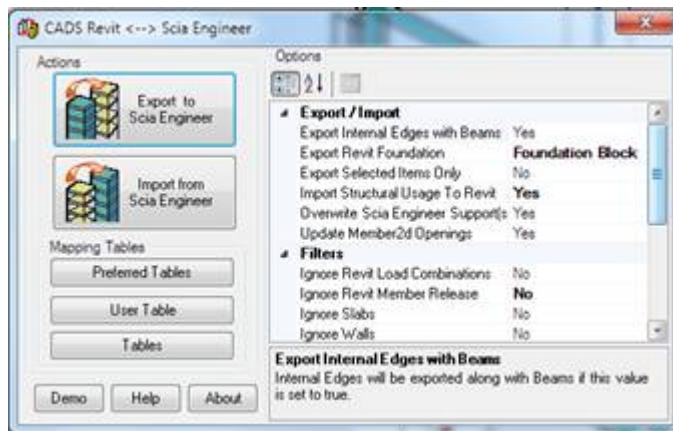
The Revit link options

The Revit link is installed to the Revit structure tab named Add-ins.



Options

There are several functions which affect what can be transferred to the Scia Engineer. These functions can be found in the Options dialogue.



The Revit link exports structural 1D members and 2D members with openings. The supports can be exported as a Foundation blocks or Standard supports.

The Revit link exports the whole or the selected part of the structure from the Revit project.

Launching of the Scia Engineer

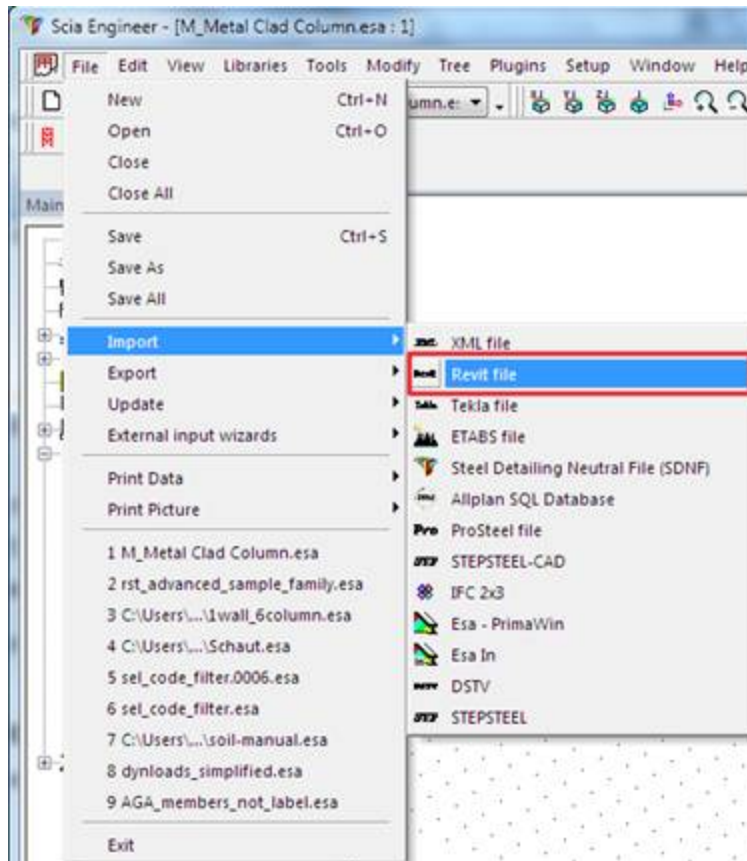
The export tool can launch Scia Engineer application automatically or save the project as a R2S format. The Scia Engineer version which is launched and national code for an exported project is defined by the Options.



If the Scia Engineer is not launched automatically then there are two possibilities:

- 1) The Export tool will ask for the path where the R2S file will be saved (option Prompt for file to YES).
- 2) R2S is saved automatically to the default folder. User is not disturbed at all (option Prompt for file to NO).

The R2S file can be imported to the Scia Engineer via the standard import tool.

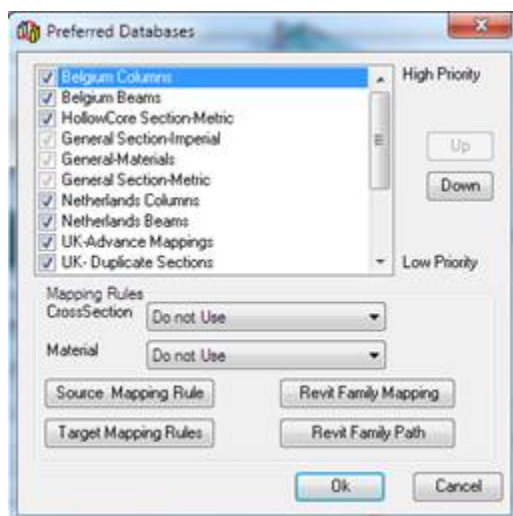


Mapping materials and cross-sections

The mapping between the Revit and the Scia Engineer is based on tables. There are three types of mapping tables – user-defined table, tables based on national codes and mapping rules.

Preferred tables

This dialogue defines which predefined tables should be used. There are mapping tables based on national codes and also user defined table in the list.



The top tables in the list have the higher priority. The mapping is done preferably by them. The checkbox can deactivate table for mapping. The buttons Up and Down is used for moving the selected table in the list.

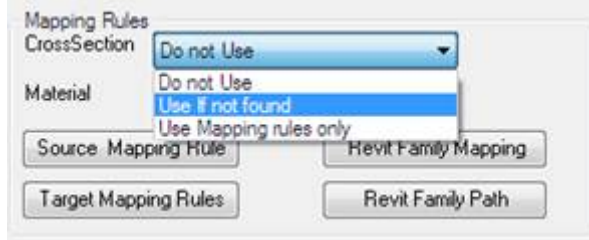
Mapping rules

Source and target rules

The mapping rules can be used when mapping by tables is not sufficient anymore:

- 1) There are too many items for mapping
- 2) There is no sufficient table and the mapping has to be done manually – some specific project or a national code.

It is used when the combobox is set to “Use If not found” or “Use mapping rules only”.



The mapping rule requires mapping item to item. Each rule defines the name in the Revit (Source table) and the name for the Scia Engineer (Target table). This is not based on libraries from the Revit or the Scia Engineer but on exact naming.

The example with the L section:

The source and the target table



The explanation of the name composition:



Note

- The star symbol can be used to map the section as a particular family in the Scia Engineer. This can define the section which is not in the Revit actual model but should be adapted later in the Scia engineer.

▶ IPE	European Beams	IPE	I section	TypeName	Depth	None	None	None	X
▶ UC*	none	UC254/254/132	I section	None	None	None	None	None	

Revit family mapping

The Revit family mapping dialog specifies the mapping of the section types to the Revit family names.



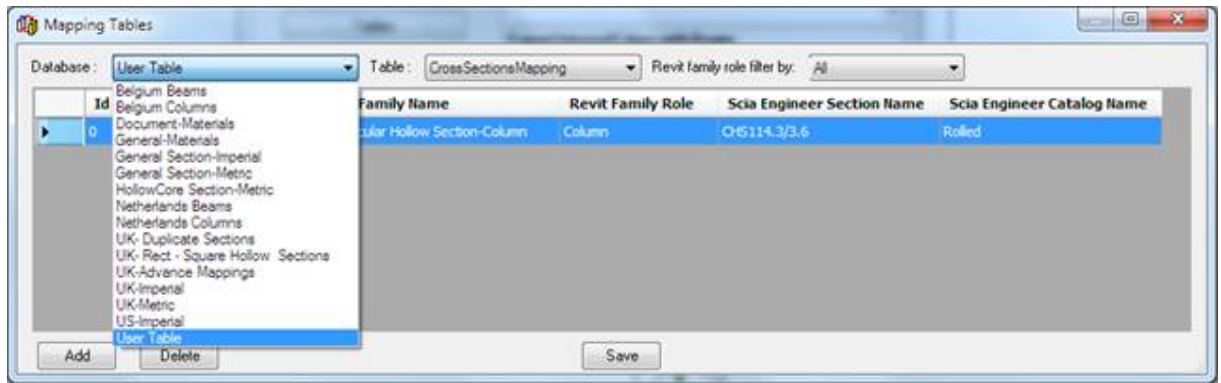
Revit family path

The Revit family path dialogue allows user to define the path of another Revit family which is saved on his computer.

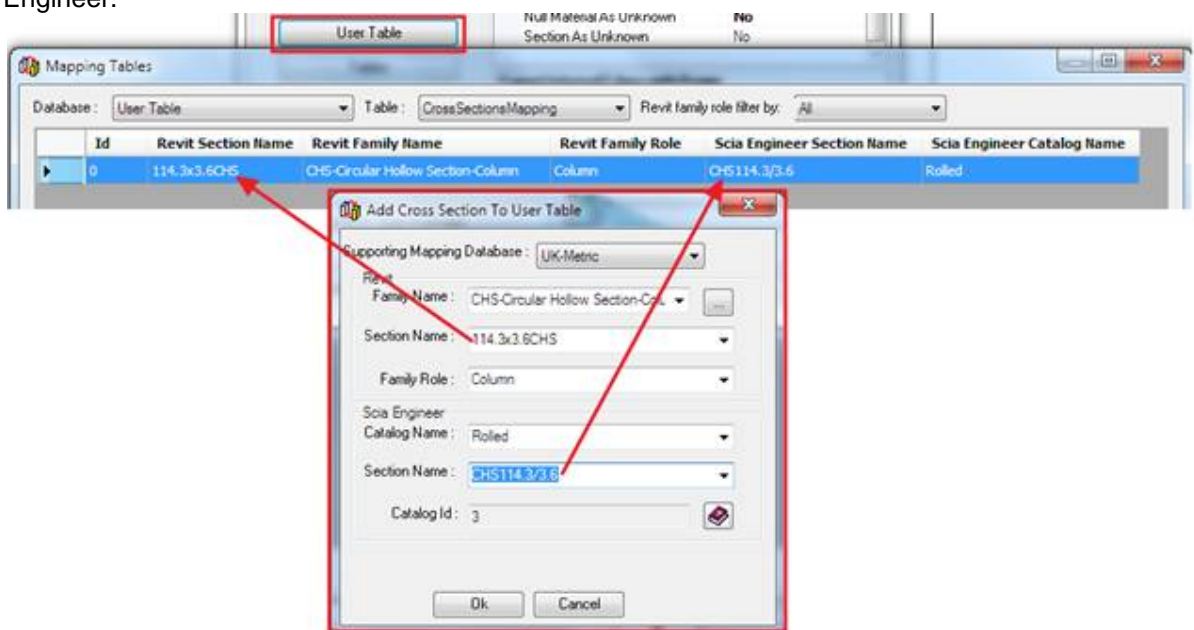


Tables

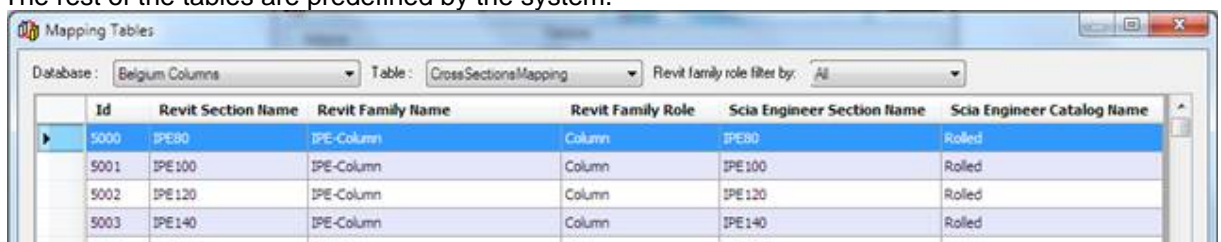
This is a dialog which shows the whole database of mapping. These tables are based on the libraries taken from the Revit and Scia Engineer.



User table is clearly user defined. The new mapping rule can be added by the dialogue button “Add”. User selects the section name in the Revit and the corresponding name in the Scia Engineer.

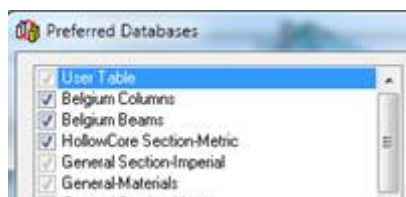


The rest of the tables are predefined by the system.



Note

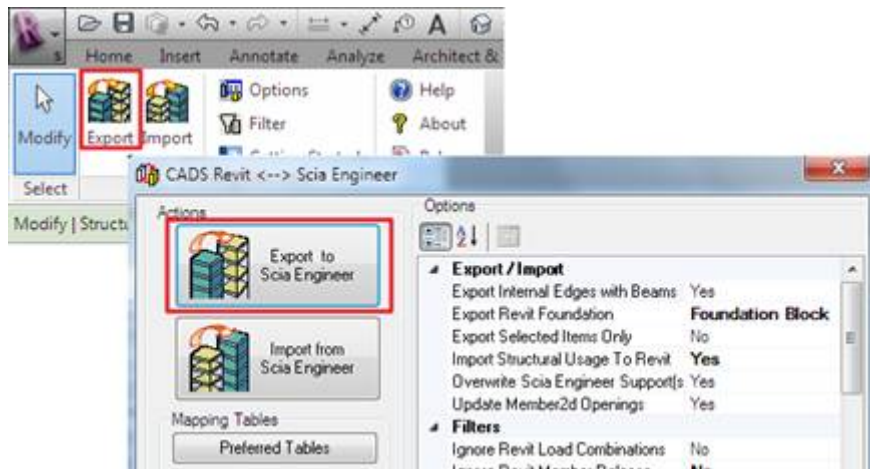
- If the user table contains rules with the higher priority then it must be moved in the list to the top. This table will be used in advance for exporting.



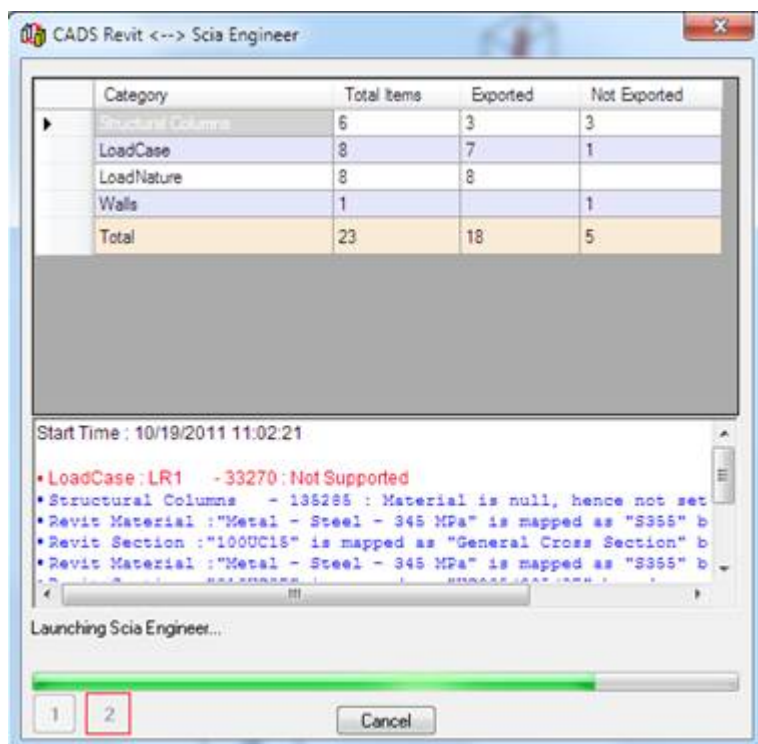
Exporting the structure from Revit to Scia Engineer

The model in the Revit structure has to be saved before it is exported to the Scia Engineer.

The export button is in the Add-ins tab and in the Options dialogue. Both start exporting the structure from the Revit structure to the Scia Engineer.



The dialogue displayed during export shows the number of items which were exported and their status. The information about the whole process is displayed in the log part. The exporting status is also displayed on the progress bar.



The log can be saved to the text-file when the export is finished.

```

Start Time : 10/19/2011 11:02:21

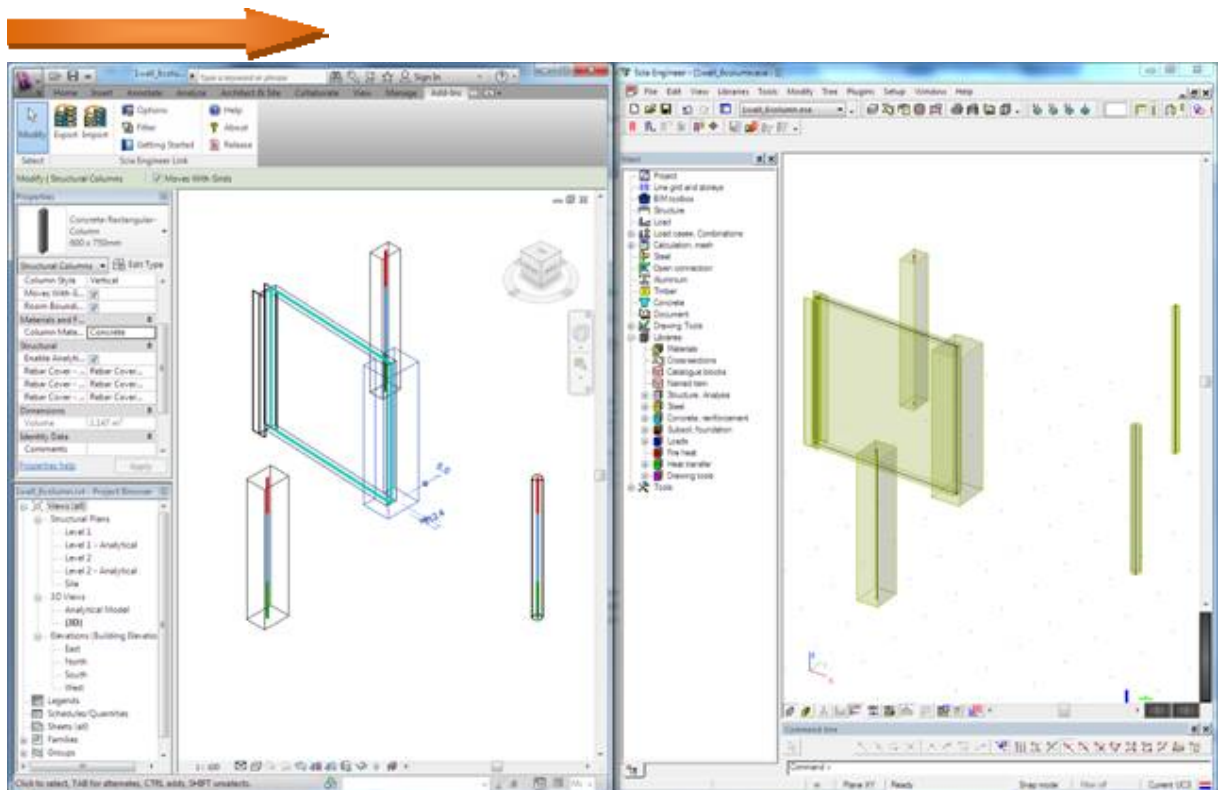
•LoadCase: LR1 - 33270 : Not Supported
•Structural Columns - 125288 : Material is null, hence not set.
•Revit Material : "Metal - Steel - 345 MPa" is mapped as "S355" based on user mapping.
•Revit Section : "I100UC18" is mapped as "General Cross Section" based on user mapping.
•Revit Material : "Metal - Steel - 345 MPa" is mapped as "S355" based on user mapping.
•Revit Section : "I310UC97" is mapped as "UC305/305/97" based on user mapping.
•Structural Columns - 125276 : Material is null, hence not set.
•Revit Material : "Metal - Steel - 345 MPa" is mapped as "S355" based on user mapping.
•Revit Section : "CHS193.7x8" is mapped as "CHS193.7/8.0" based on user mapping.
•Structural Columns - 125239 : Material is null, hence not set.
•Wall# - 125701 : Material is null, hence not set.

Exported Cross Section(s):
•"CHS193.7x8" is mapped as "CHS193.7/8.0"
•"I100UC18" is mapped as "General Cross Section"
•"I310UC97" is mapped as "UC305/305/97"

End Time : 10/19/2011 11:06:35

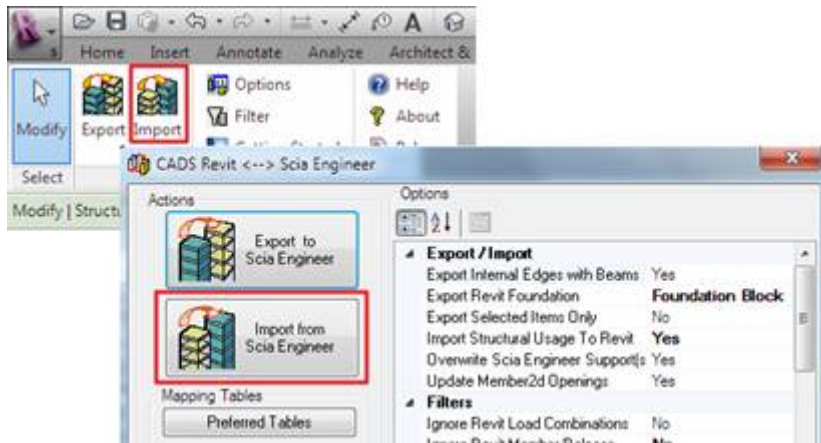
Total Time : 00:04:14.3149584
    
```

Scia Engineer is opened automatically if it is set in the option dialogue. Newly exported structure is displayed in the model window.



Import the structure from Scia Engineer to Revit

The import button is in the Add-ins tab and also in the Options dialogue. Both start importing the structure from the Scia Engineer to the Revit structure.



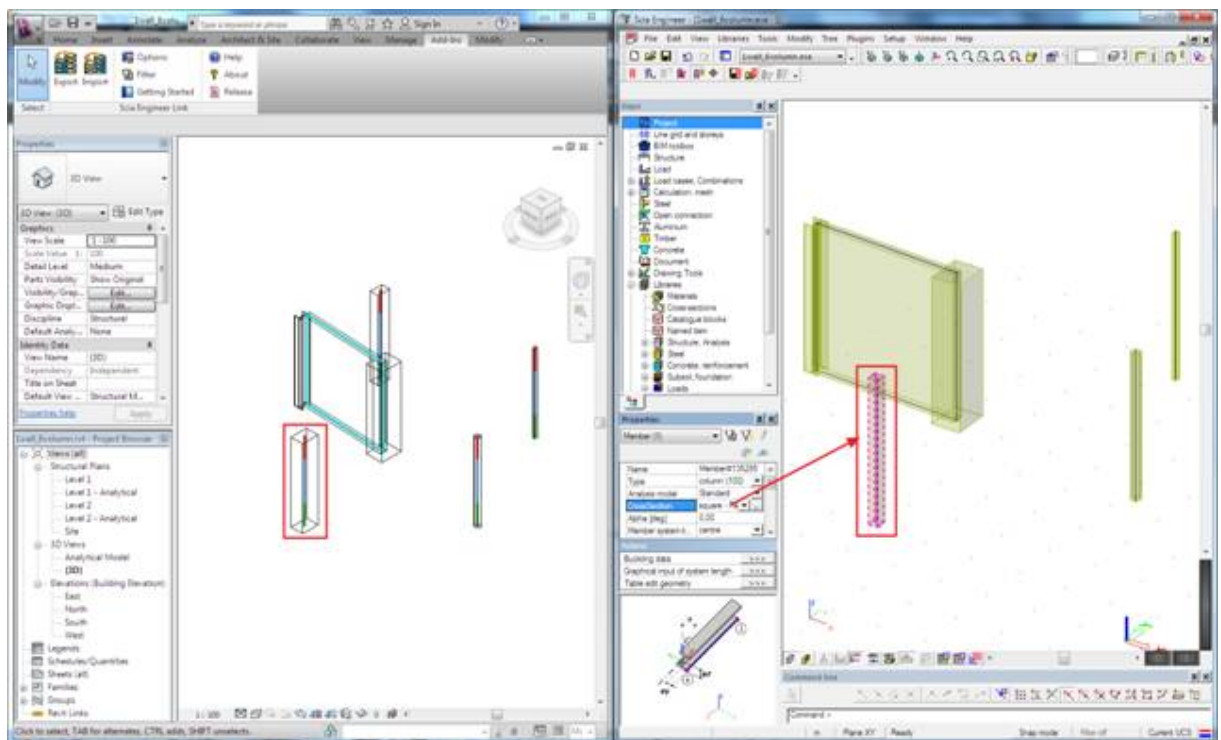
The Revit link is mainly developed for exporting the data from the Revit structure to the Scia Engineer. Only some smaller changes made in the Scia Engineer can be imported back to the Revit.

This functionality has its limitations. The Revit API doesn't allow deleting item in the Revit when it is imported for the Scia Engineer. It is possible only to modify what is already exported.

The possible modifications are clearly described in the next four examples.

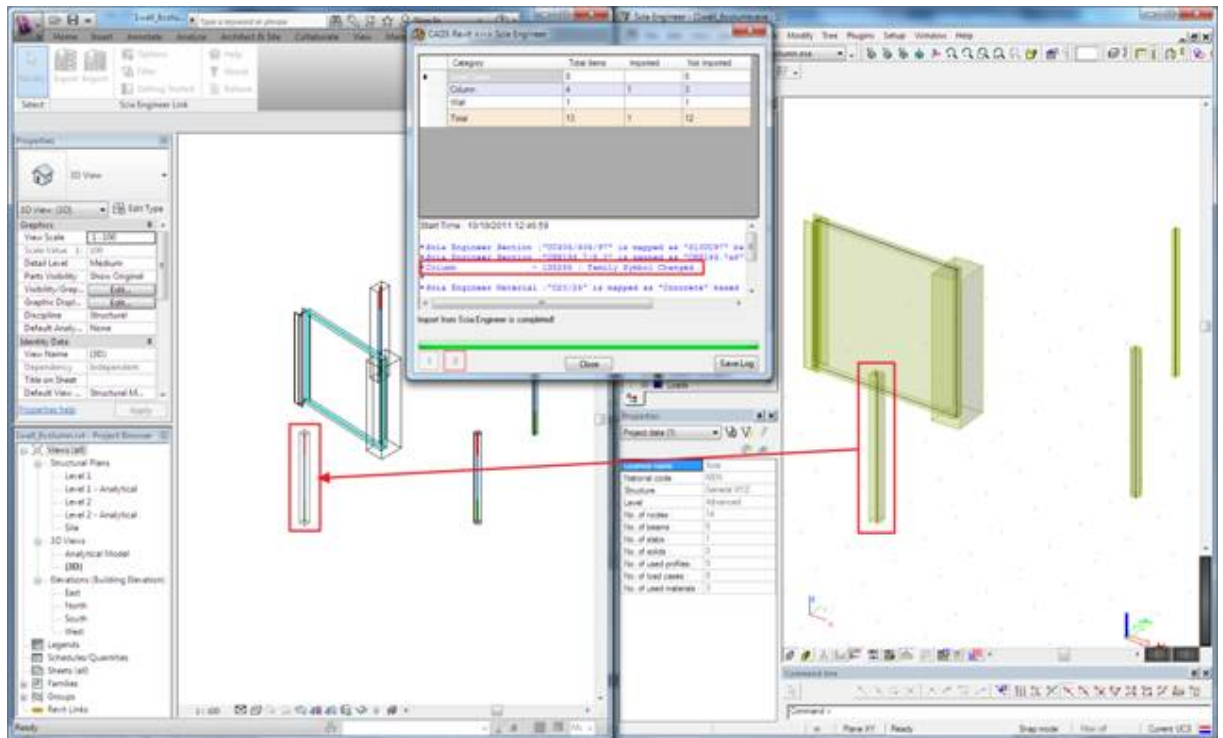
Example no.1 - modification

Adapt the column cross section in Scia Engineer and import it back to Revit.



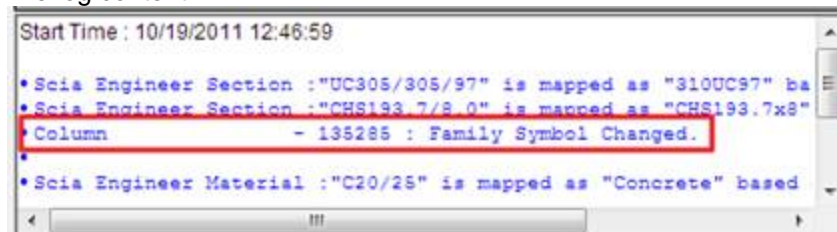
The default cross-section

Cross-section is adapted to square



Updated cross section in Revit

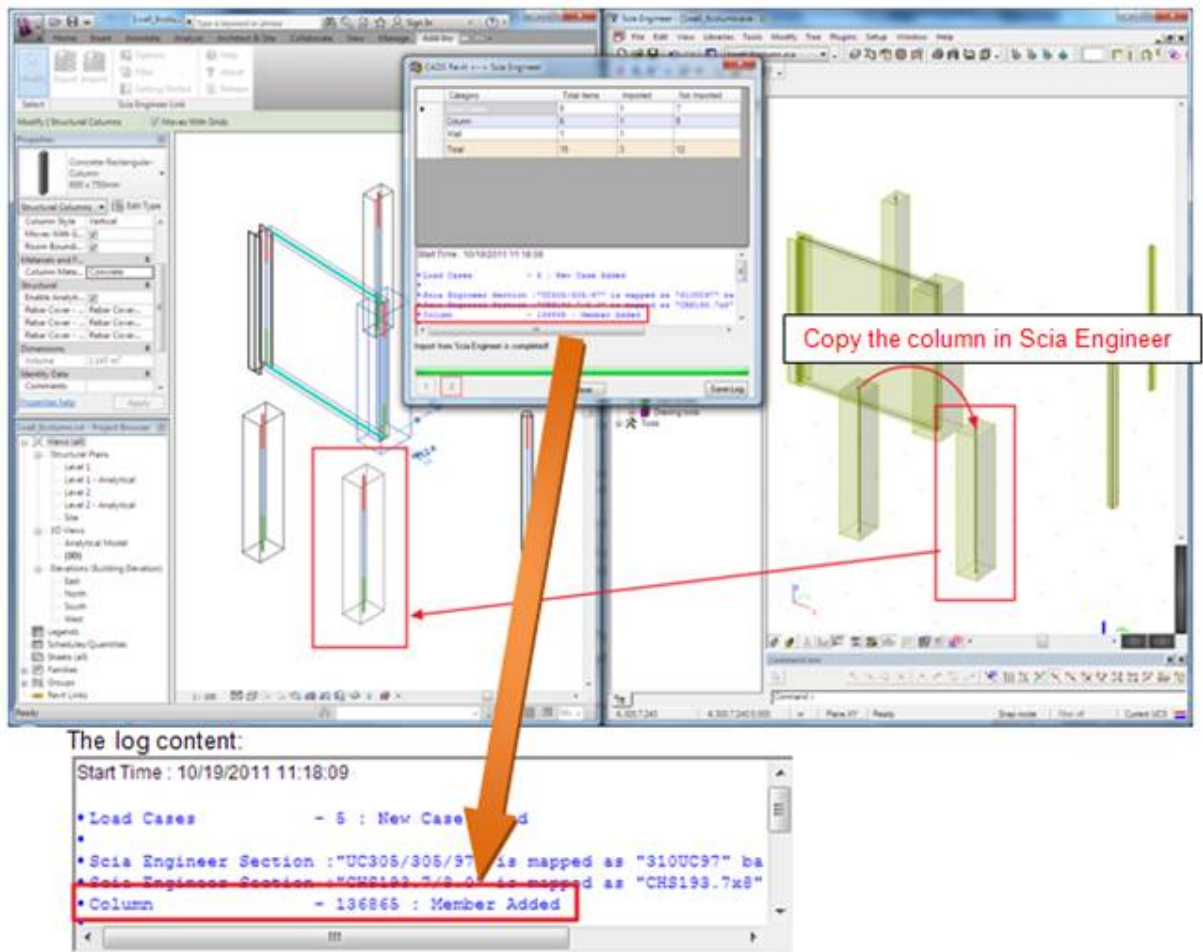
The log content:



The result: The Revit cross section is modified after the import.

Example no.2 - copy

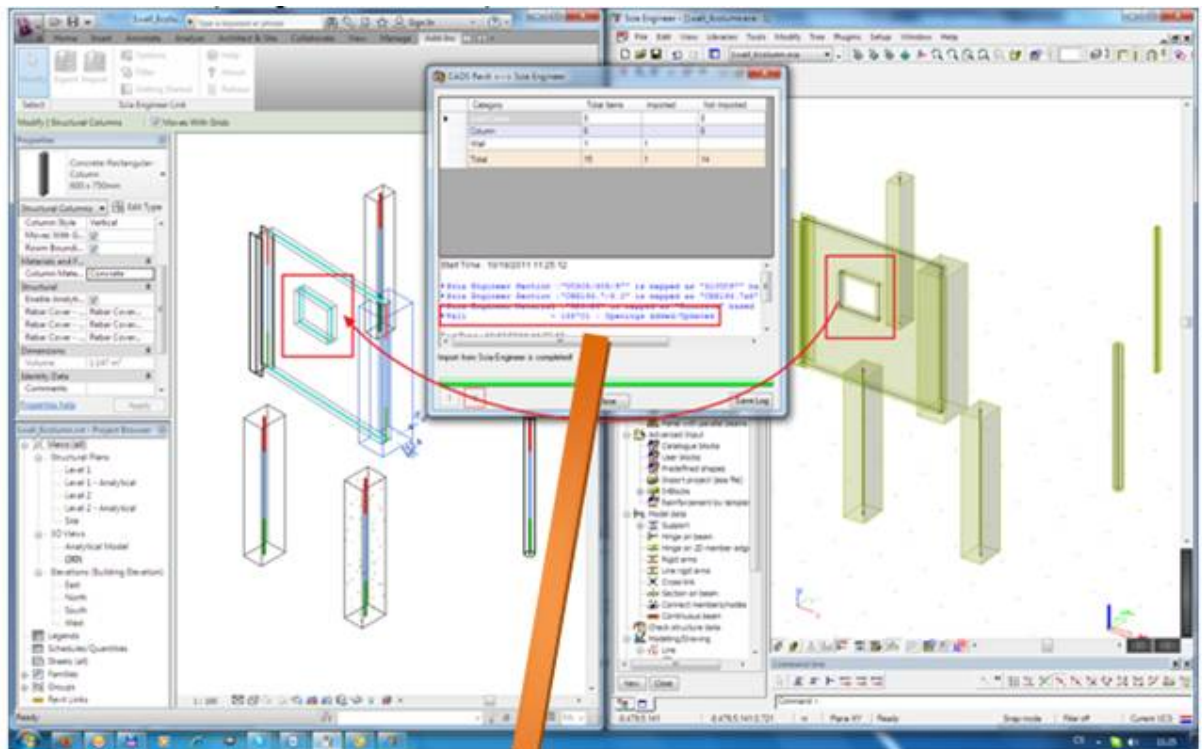
Copy exported column in Scia Engineer and import it back to Revit:



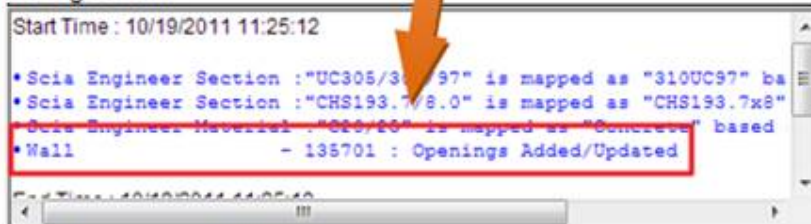
The result: The Revit structure adds one more column after the import.

Example no.3 – create new on exported

Create new opening in the wall and import it back to Revit:



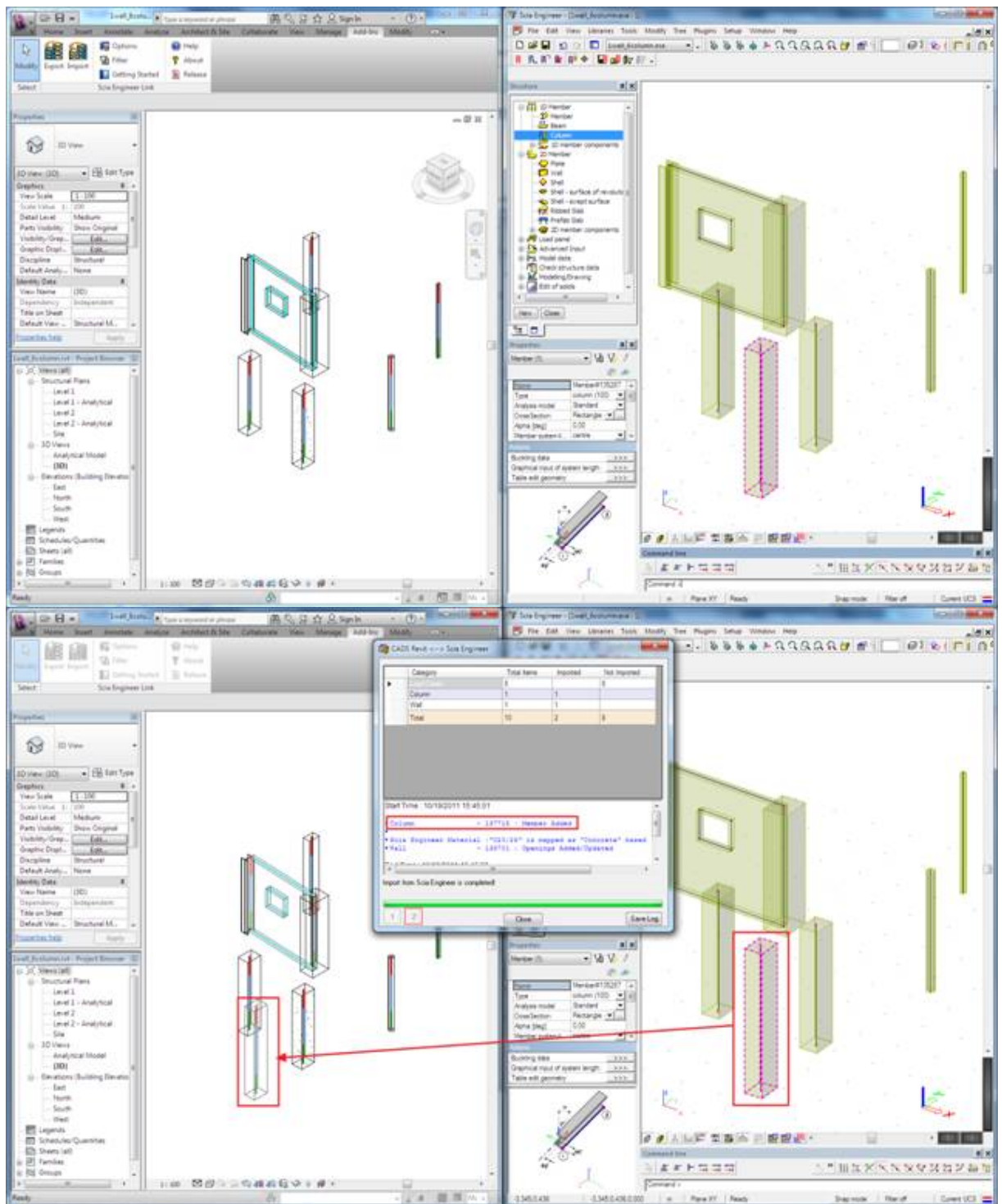
The log content:



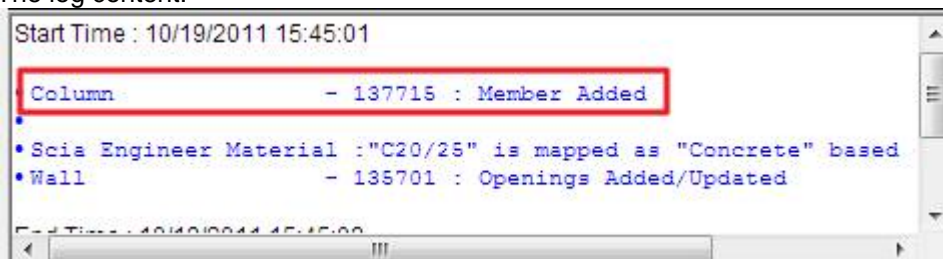
The result: The Revit structure creates new opening in the wall after the import.

Example no.4 – create new item

The Revit link is not the tool for importing the whole new structure from the Scia Engineer to the Revit structure, but in the simple cases is possible to add a new item:



The log content:



The result: The Revit structure creates new column after the import.

When something is not exported

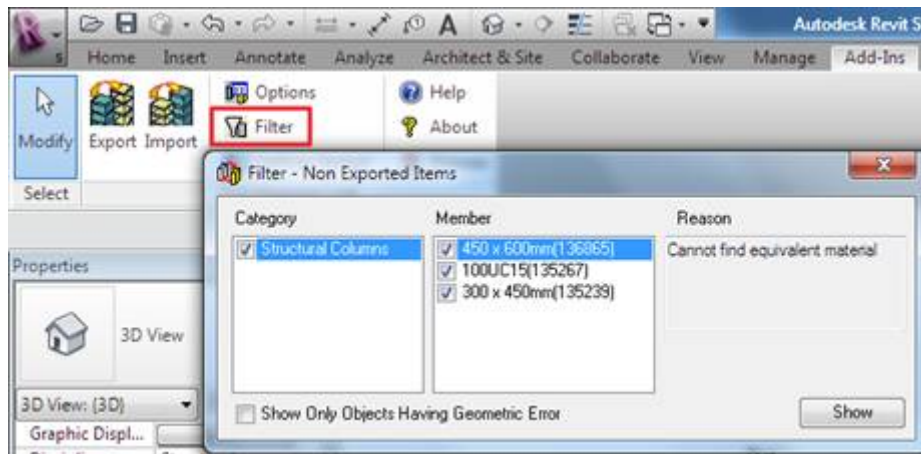
This can be caused by a several reasons.

The cross section or material is not correctly mapped

This information can be found in the log.

```
•Revit Material : "Metal - Steel - 345 MPa" is mapped as "S355" based on user mapping.
•Revit Section : "100UC15" is mapped as "100UC15(135267)" based on user mapping.
•Structural Columns - 135276 : Material is null, hence not set.
•Revit Material : "Metal - Steel - 345 MPa" is mapped as "S355" based on user mapping.
•Revit Section : "CHS193.7x8" is mapped as "CHS193.7/8.0" based on user mapping.
•Structural Columns - 135239 : Material is null, hence not set.
•Walls - 135701 : Material is null, hence not set.
```

Not exported items can be selected and showed by the tool Filter.

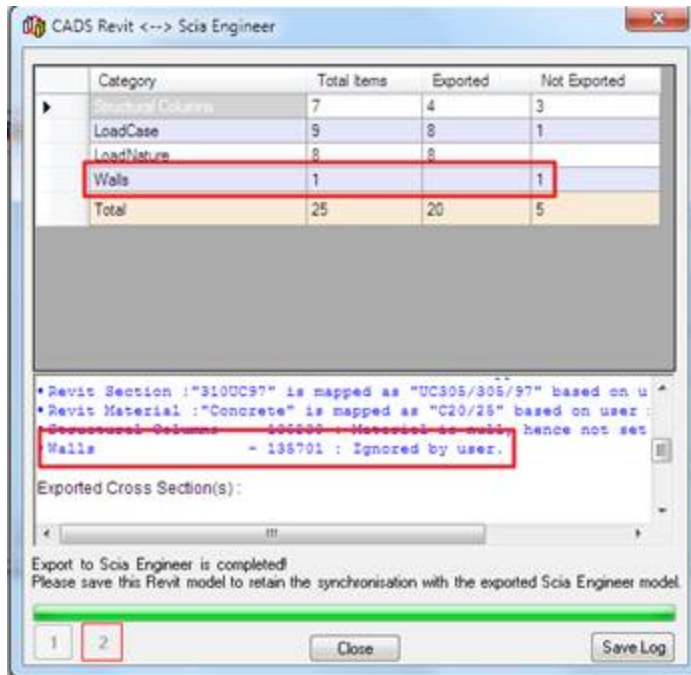


- ➔ There is information why the item was not exported.
- ➔ Checkbox displays only objects with a bad geometry.
- ➔ Show button selects and zoom the item in the Revit model window.

The dialogue Option ignore the entity

There is a possibility to ignore 2D member during the export. When this functionality is used the 2D member is not exported to the Scia Engineer.

This can be found in the log:



The object is not supported by the Revit link

The exportable objects are the structure objects. If the model contain any architectural entity this won't be exported to the Scia Engineer.

If the model is created in the Revit architectural then the model must be edited so the objects are switched to the structural ones.

The object is not supported by the Scia Engineer

Some objects are specific so they don't have any corresponding entity in the Scia engineer. Some of those are not yet implemented.

Format IFC (Industry Foundation Classes) allows for transfer of projects and models between applications which are used in a building industry. The objective is to simplify the work.

In which way can you import and export by means of IFC?

At first we would like to explain you the IFC format terminology. Each member (entity) can be described in IFC format in several ways. This is called the representation of an entity. We support following representation types:

SweptSolid – import and export of straight 1D and flat 2D members and export of steel connection plates.

Brep – import and export of all entities which we allow to import and export.

CSG – only for import, the result is always a general volume.

Clipping – import and export, the result is shown in a structural model.

AdvancedSweptSolid – import and export of concrete reinforcement.

SectionedSpine – only for export of arbitrary and haunched beams.

MappedRepresentation – import and export for particular cases.

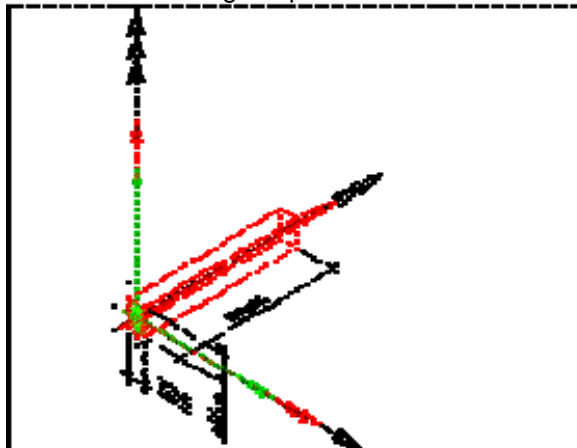
GeometricCurvedSet – export of linegrids.

SweptSolid

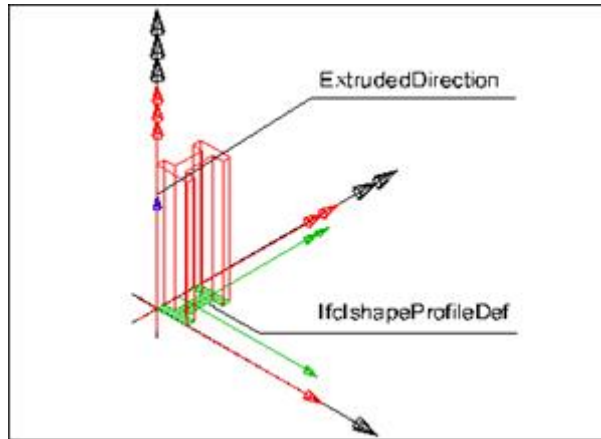
SweptSolid – means that a profile (or an area) is defined and it is extruded along an axis.

Members defined with SweptSolid representation are converted into Scia Engineer as native 1D or 2D members.

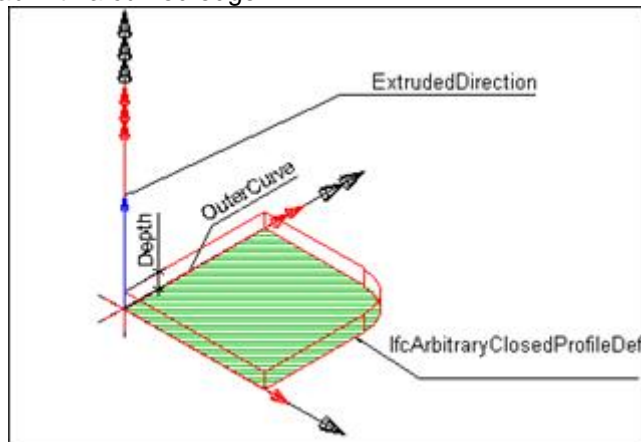
Example of a beam with a rectangular profile



Example of a column with an I profile



Example of slab with a curved edge

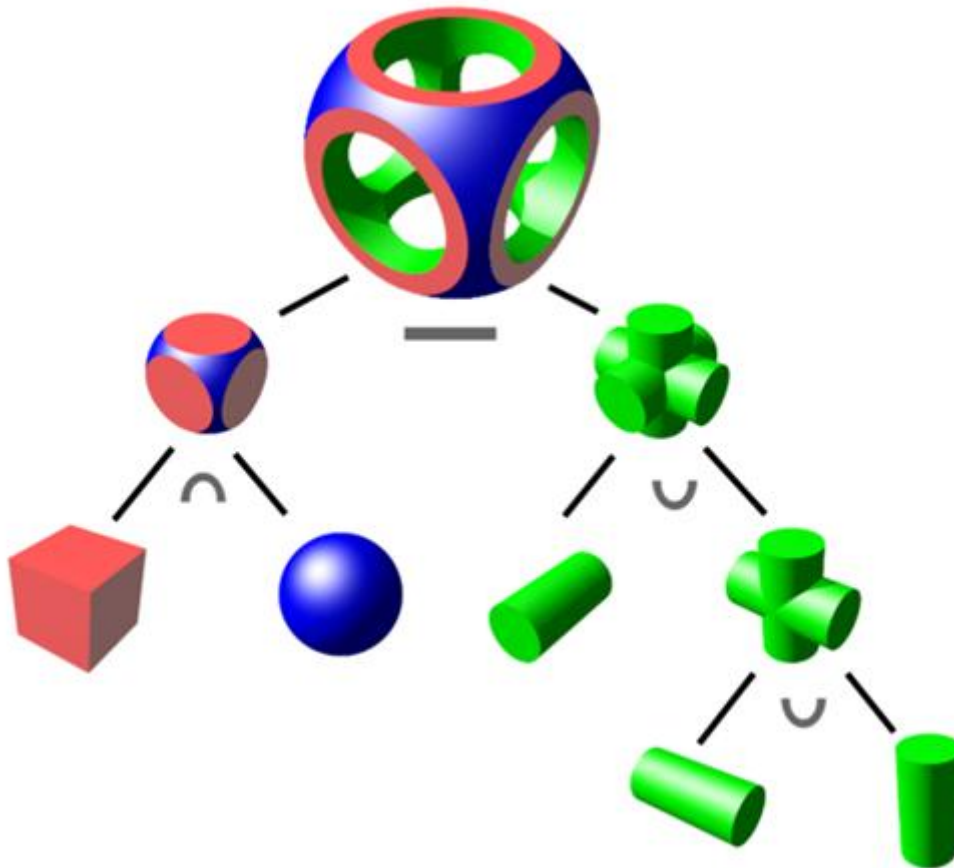


Brep

Brep (=boundary representation) – means that an entity is described with vertexes (points) which are connected with lines to create a volume. These ones are imported like general volumes (solids) to Scia Engineer. In this case it is possible to use an option for automatic recognition of members. This function tries to convert the general volumes to beams, columns, walls or slabs.

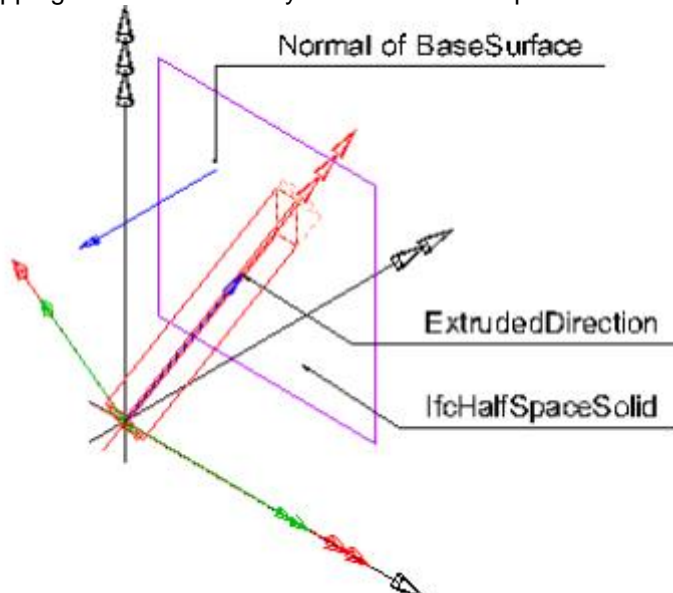
CSG

CSG (= constructive solid geometry) – allows Booleans operations (union, difference and intersection) between solids.



Clipping

Clipping – means that entity is defined as SweptSolid with a cut-out by plane or other way.

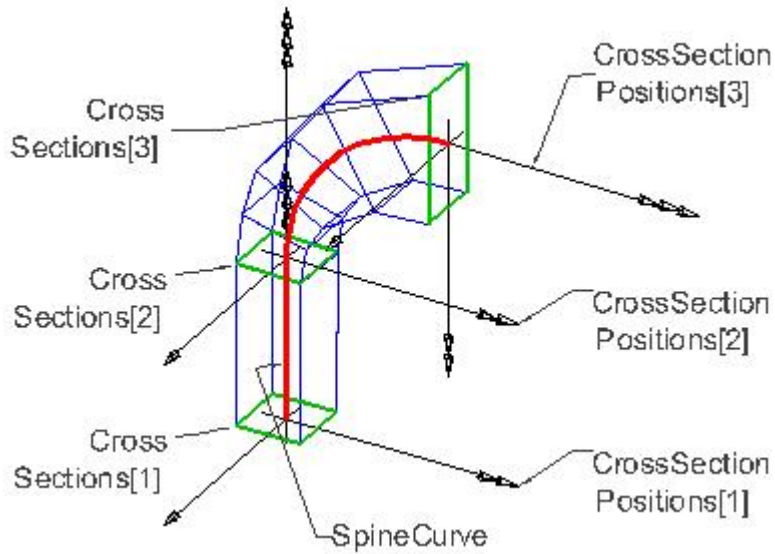


AdvancedSweptSolid

This representation defines the geometry by means of diameter and centre line. The entity profile is always a circle.

SectionedSpine

This representation allows defining of two or more different profiles for a beam or a column.



MappedRepresentation

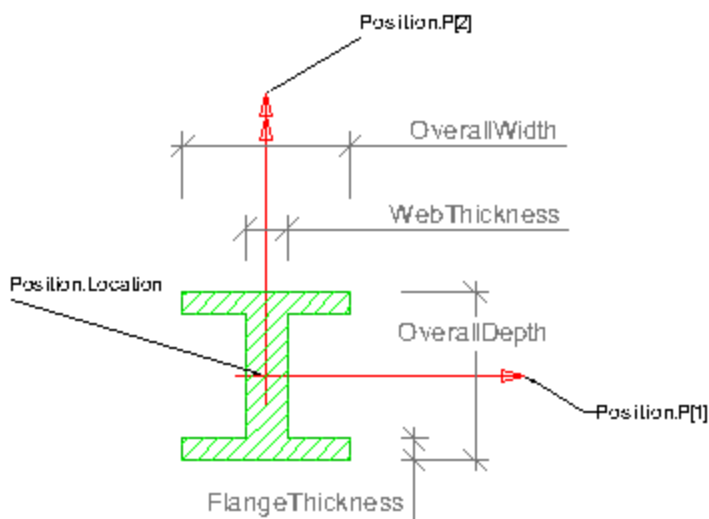
Allows using of one defined geometry for more entities.

Beam and column profiles

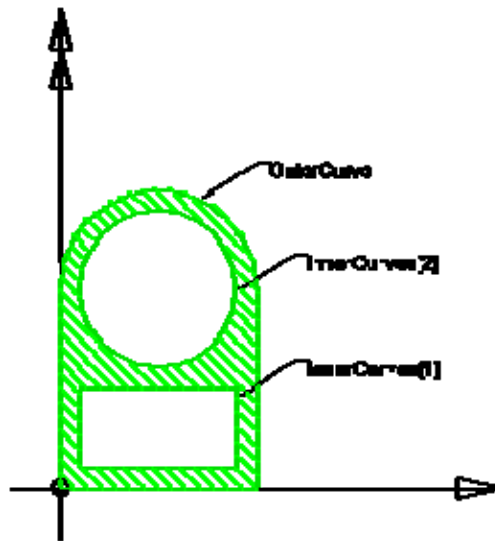
After the import beams and columns have one of the following profile types (cross-section):

- cross-section from the Profile Library - if the same name is found in Scia Engineer database,
- cross-section from Geometric shapes - if the profile is defined by parameters in the IFC file,
- a general cross-section.

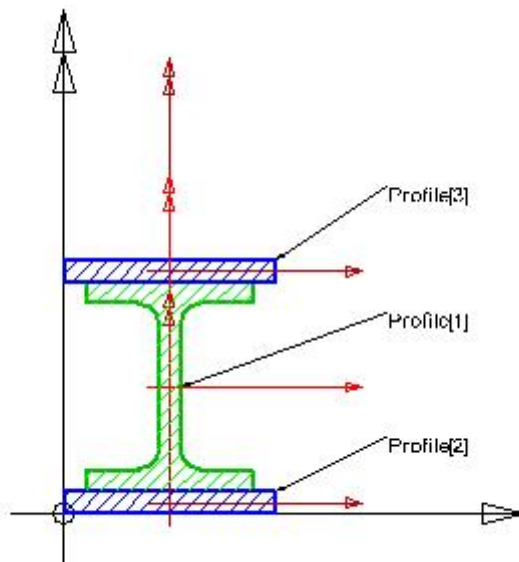
Parametric profile example



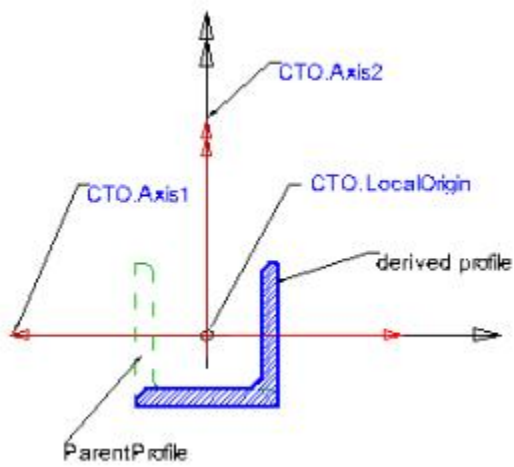
General profile example



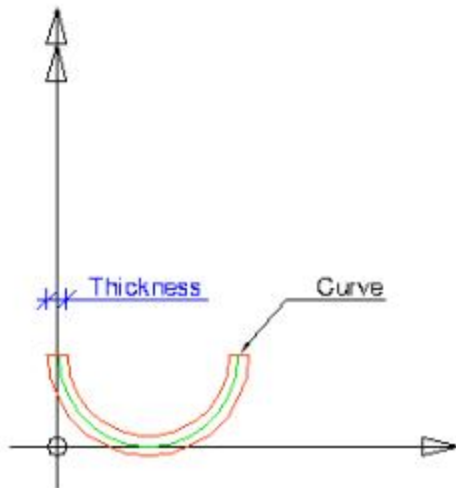
Composed profile example



Derived profile example



Thin-walled profile example



What is possible to import and which way

- flat walls and slabs with openings and a constant thickness
- flat walls and slab with recesses and a constant thickness
- circular arc walls without openings and recesses (with openings and recesses only in the structural shape)
- slabs with circular arc edges
- 1D members with openings
- shells, curved 1D members, beams with a haunch and arbitrary beams – only as general volumes
- steel connection parts as plates and bolts – in the current version only as general volumes
- concrete reinforcing bars as free bars if the geometry is described in IFC file
- IFC predefined attributes for beams, columns, members, walls and slabs

What is possible to export and which way

- flat walls and slabs with openings
- flat walls and slabs with subregions where the subregions are defined as recesses
- circular arc walls and slabs with circular arc edges
- 1D members with openings
- shells and curved 1D members – only with boundary representation
- 2D members with cut-outs – only with boundary representation or without cut-outs
- parameters with their values which are defined as an attribute of 1D or 2D members as member property
- IFC predefined attributes for beams, columns, members, walls and slabs
- beams with a haunch and arbitrary beams – with SectionedSpine or boundary representation
- steel connections as plates and bolts
- concrete reinforcement of 1D members and free bars as reinforcing bars
- concrete reinforcement of 2D members as reinforcing mesh

- storeys as building storeys
- circular and rectangular 2D line grids and 3D rectangular line grid as more 2D line grids
- colour of entity by layer

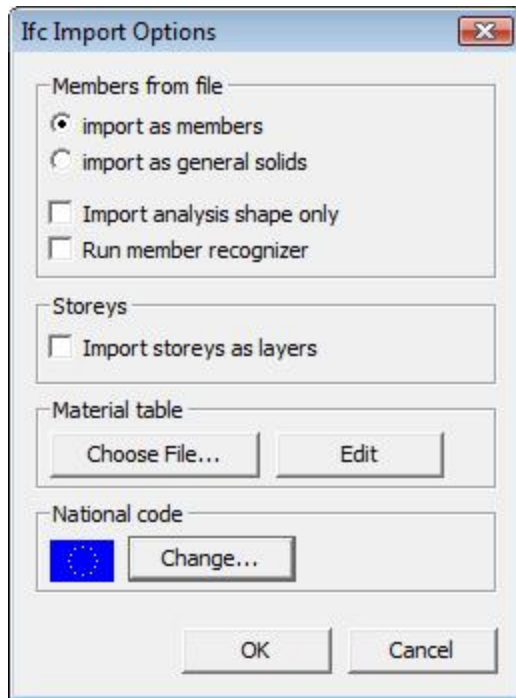
Import

Procedure

1. Open Scia Engineer.
2. Run **File > Import > IFC 2x3**.



3. Select an IFC file in dialog Open and confirm with **[Open]** button.
4. Choose the way which a model should be imported. Confirm with **[OK]** button to complete the import.



Options:

- Import as members
It means all members defined with SweptSolid representation will be imported like Scia Engineer native members (both 1D and 2D members).
- Import as general solids
All members will be imported as general volumes.
- Import analysis shape only
Only member shape will be imported without clipping on members.
- Run member recognizer
This option runs a member recognizer which will try to convert brep members (general solids) to Scia Engineer native members.
- Import stories as layers
If an IFC file contains some building storeys and the option is ON then each storey is a separate layer in Scia Engineer. The layer name is the same as the storey name.
- Material table
If material names in the IFC file are not in accordance with code names it is necessary to define a material conversion table. If no table exists default materials are assigned. Button [Choose file...] allows for the selection of the file with the material table. Modifications can be made through button [Edit]. The file has *.con file extension and it is a plain-text file, for example:

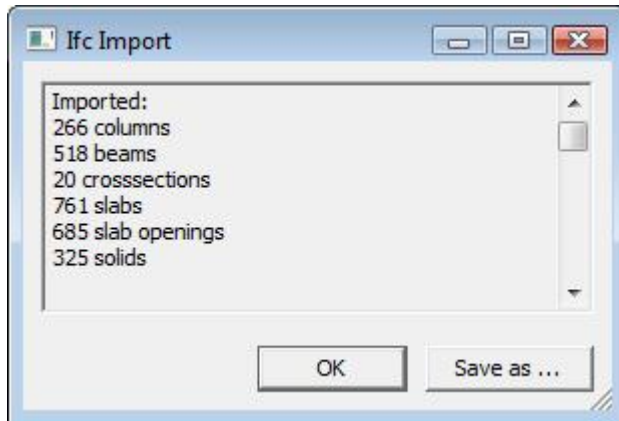
```
[materials]
;
Concrete1=C12/15
Concrete2=C25/30
```

The first name is name of material in the IFC file and the second one is the code name of material used in Scia Engineer. It is necessary to respect all characters.
- National code
The code in which the imported project will be open must always be selected.

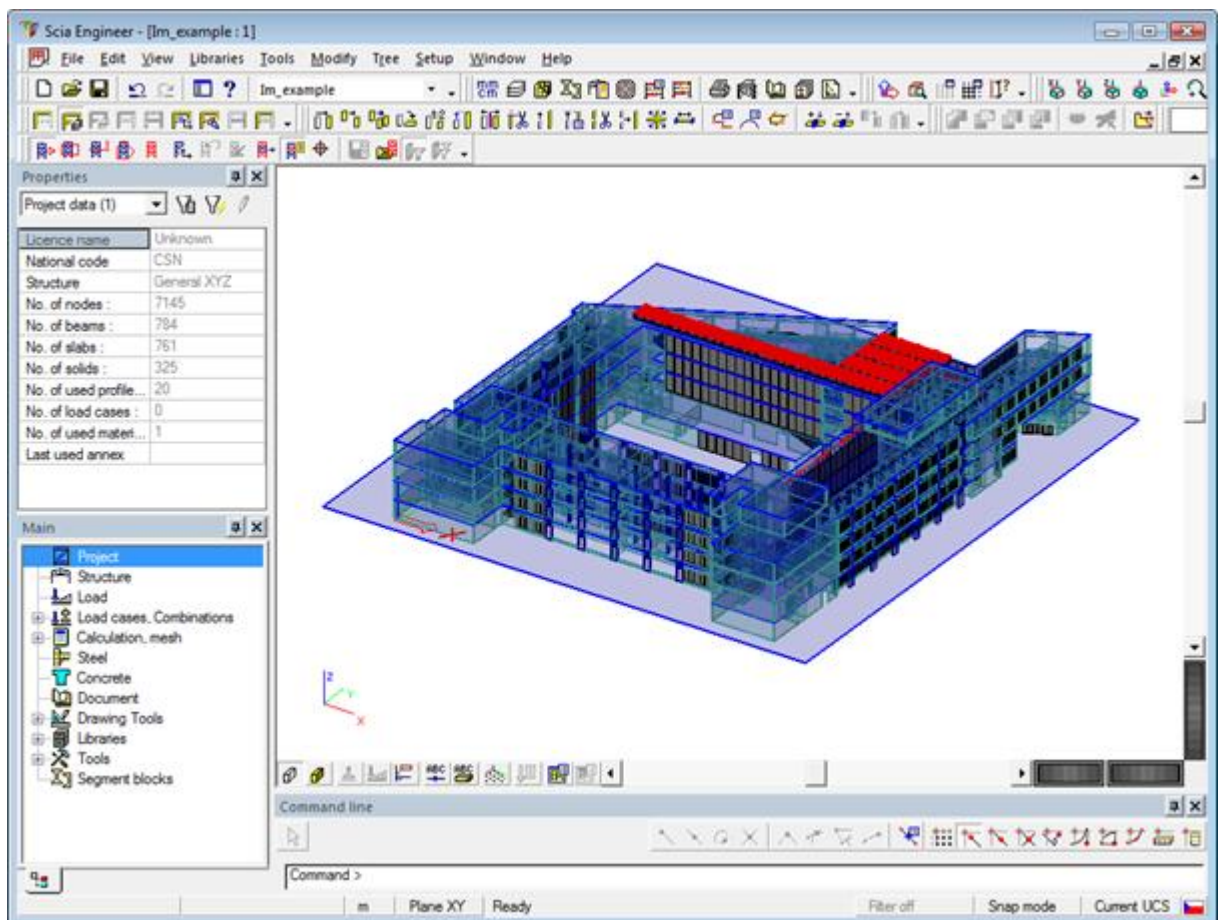
5. A dialog with the progress of import is displayed.



6. A dialog with the import report is displayed. To finish the import press the **[OK]** button. To save the import report press the **[Save as...]** button.



7. In order to creating a proper analysis model it is necessary to use Connect and Align functions.

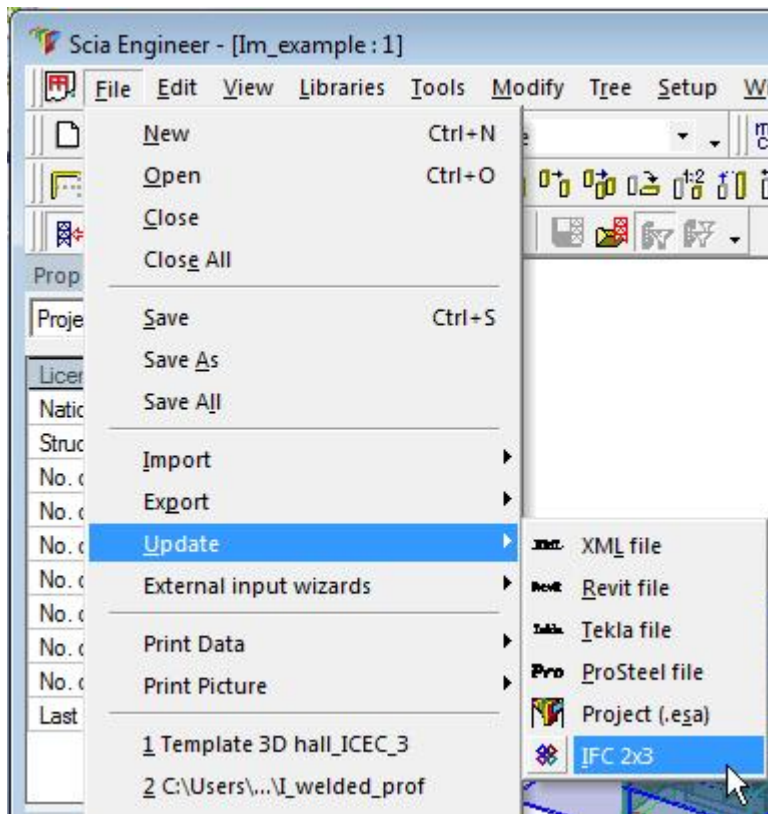


Update

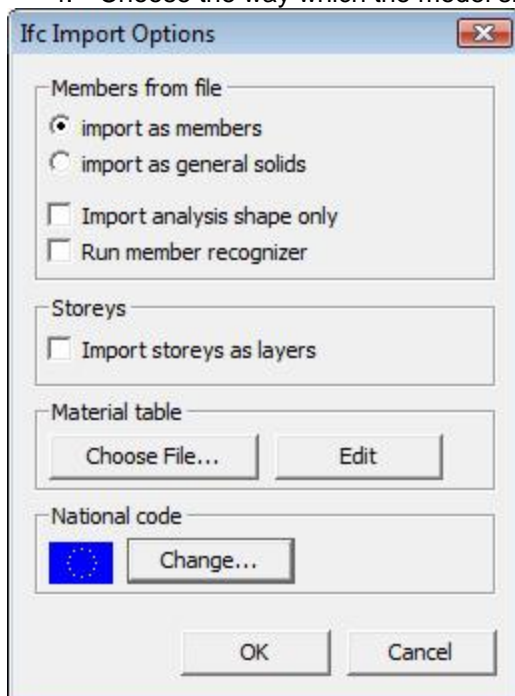
Procedure

1. Open Scia Engineer project which has been updated.

2. Run **File > Update > IFC 2x3**.



3. Select the IFC file which is to update the project and confirm with **[Open]** button.
4. Choose the way which the model should be imported. Confirm with the **[OK]** button.



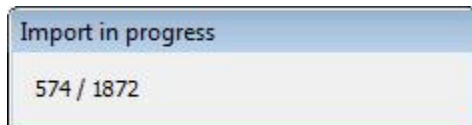
Options:

- Import as members
- Import as general solids

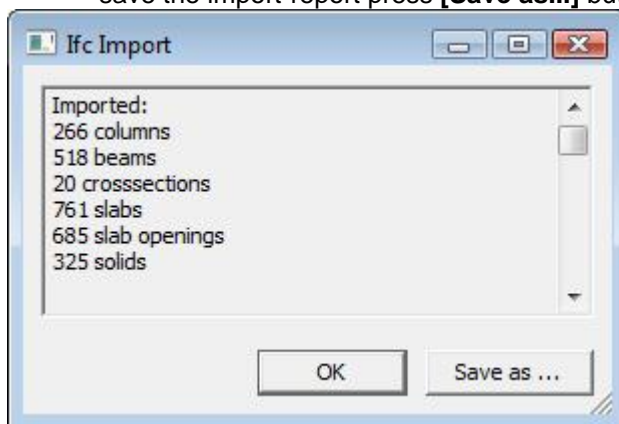
- Import analysis shape only
- Run member recognizer
- Import stories as layers
- Material table
- National code

Note: For the explanation of options see the previous chapter.

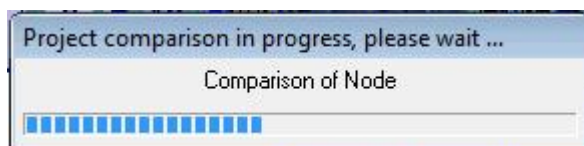
5. A dialog with the progress of import is displayed.



6. A dialog with the import-report is displayed. To finish the import press **[OK]** button. To save the import-report press **[Save as...]** button.



7. The import is finished and comparison of the two projects (the original one and changed one) is launched. The procedure is similar to ESA-ESA Update.



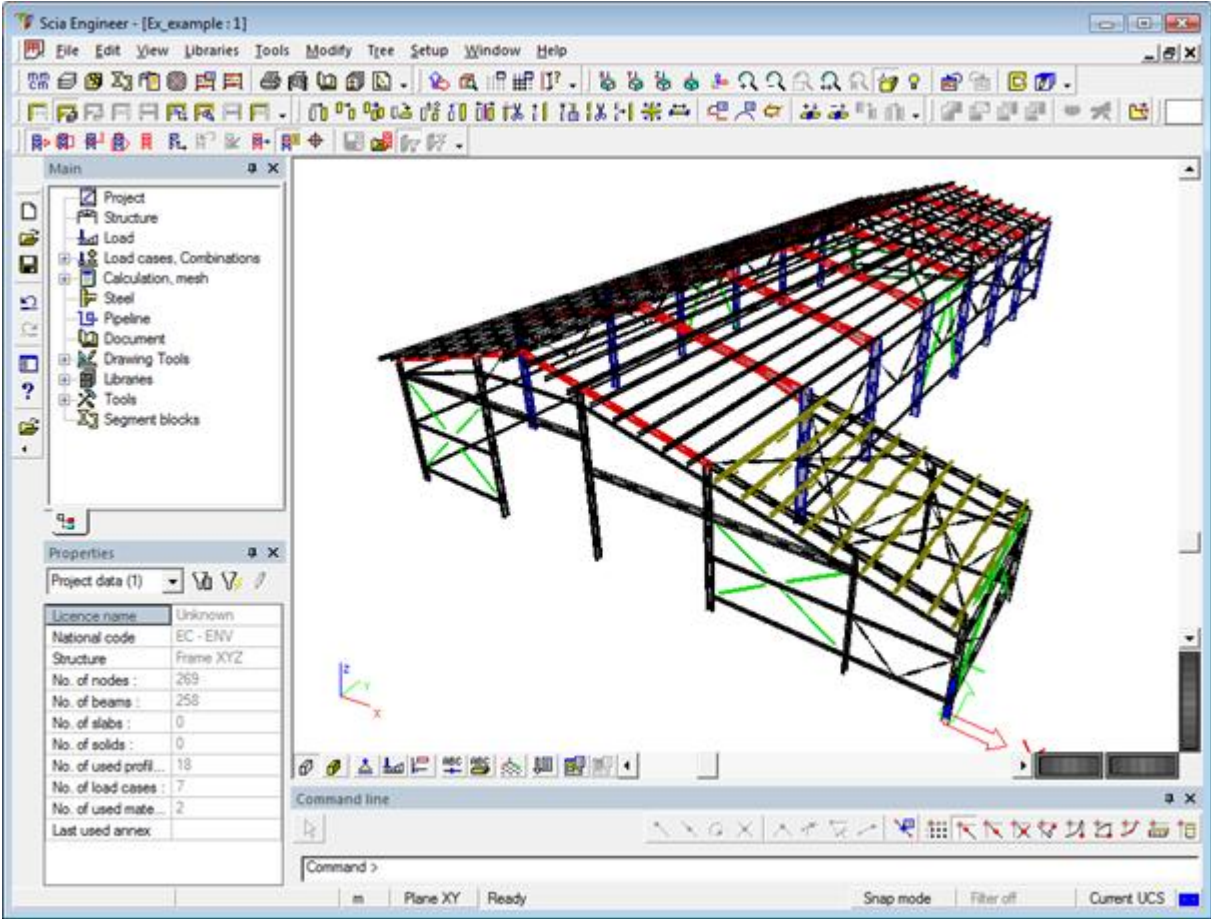
Note: Though Update function helps to save your time, it is necessary to respect some disadvantages of IFC, such as:

- the IFC format describes only member surfaces. That means if you have the structure connected and aligned after update it will be detected like changes.
- All member nodes are generated during the import, i.e. during update the nodes are in groups Deleted entities and New entities.

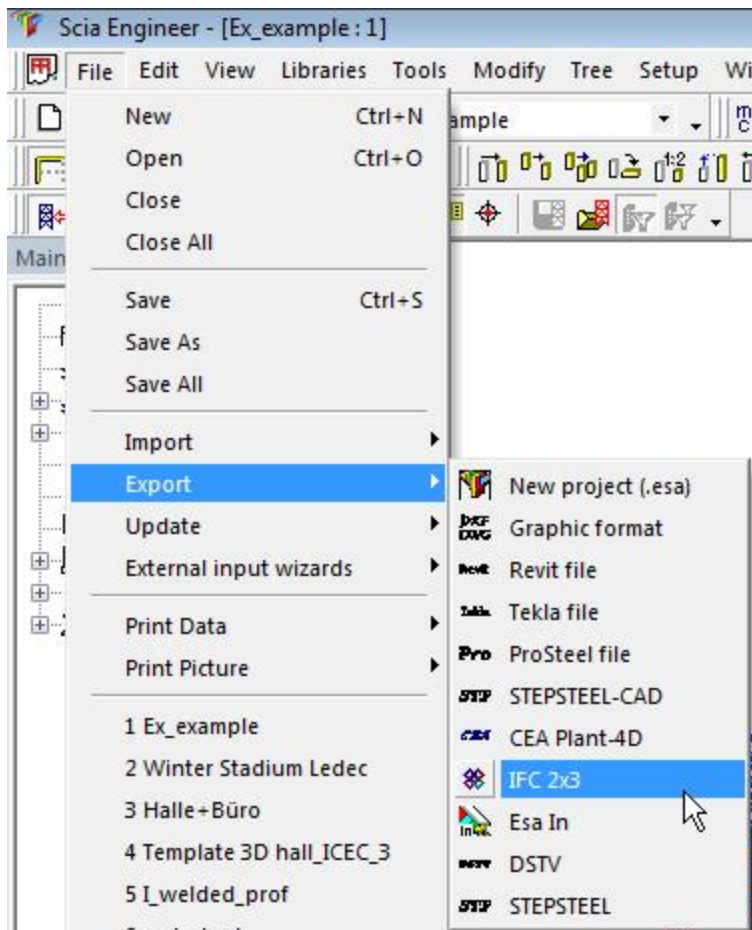
Export

Procedure

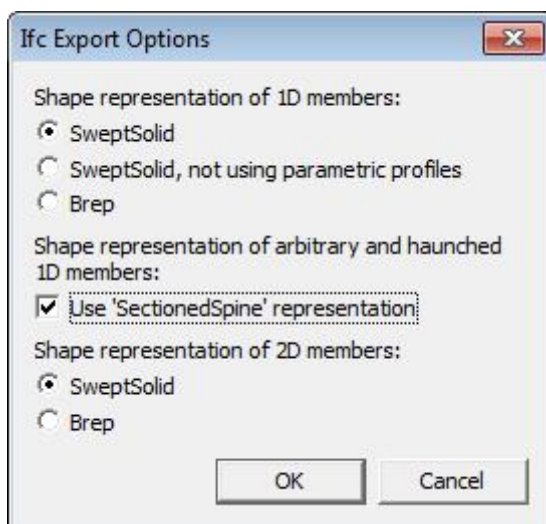
1. Open Scia Engineer file.



2. Run **File > Export > IFC 2x3**.



3. Write a name of IFC file in dialog **Save As** and confirm with **[Save]** button.
4. Choose the way which the model should be exported. Confirm with **[OK]**.



Options:

- SweptSolid
All members that can be exported as a profile (area) extruded along an axis and cross-sections that can be described with parameters will be written this way to the IFC file.

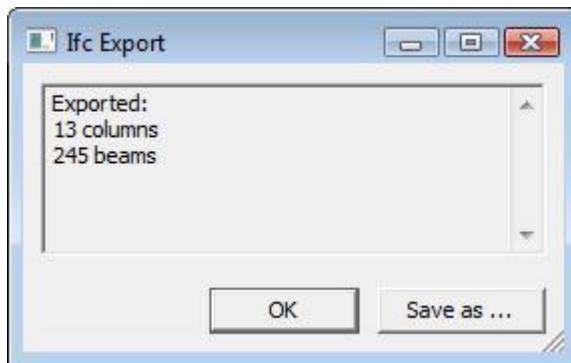
- SweptSolid, not using parametric profiles
The same as previous option except cross-sections. All cross-sections are exported as general profile.

Note: Not all applications are able to read parametric profile.

- Brep
All members are exported with boundary representation, i.e. no profiles are defined, etc.
- Use SectionedSpine representation
If ON, all beams with a haunch and arbitrary beams are exported by means of the SectionedSpine representation. If OFF, all beams and columns are exported with prismatic profiles.

Note: Not many applications support this representation type.

5. A dialog with the export-progress is displayed.
6. A dialog with the export-report is displayed. To finish the export press **[OK]** button. To save the export report press **[Save as...]** button.



7. The IFC file is written.

SDNF

SDNF Interface

Import

Project data

Project data

Basic data | Functionality | Loads | Combinations | Protection

Data

Name: ProjectId

Part: StructureId

Description: ClientId

Author: EngineeringFirmId

Date: 21Dec98

Material

Concrete	<input checked="" type="checkbox"/>
Material	C12/15
Reinforcement m...	B 400A
Steel	<input checked="" type="checkbox"/>
Material	S 235
Timber	<input checked="" type="checkbox"/>
Material	C14
Other	<input checked="" type="checkbox"/>
Material	
Aluminium	<input type="checkbox"/>

Code

National Code: EC - EN

National annex: EC-EN

OK Cancel

Project data which are loaded from the SDNF file:

- Name is "Project Id"
- Part is "Structure Id"
- Description is "Client Id"
- Author is "Engineering Firm Id"
- Date is correctly loaded from *.sdf file

1D members

Member types:

- column is imported as column

- types beam and brace are imported as beam

Profiles

- if the same name of profile is found in Scia profile database or if a user conversion table is defined

Material

- if the same name of material is found in Scia material database or if a user conversion table is defined

Member properties:

- cardinal points are correctly set as "Member System At" property
- eccentricity – ez,ey is read from file
- correct position and orientation of begin and end node of member

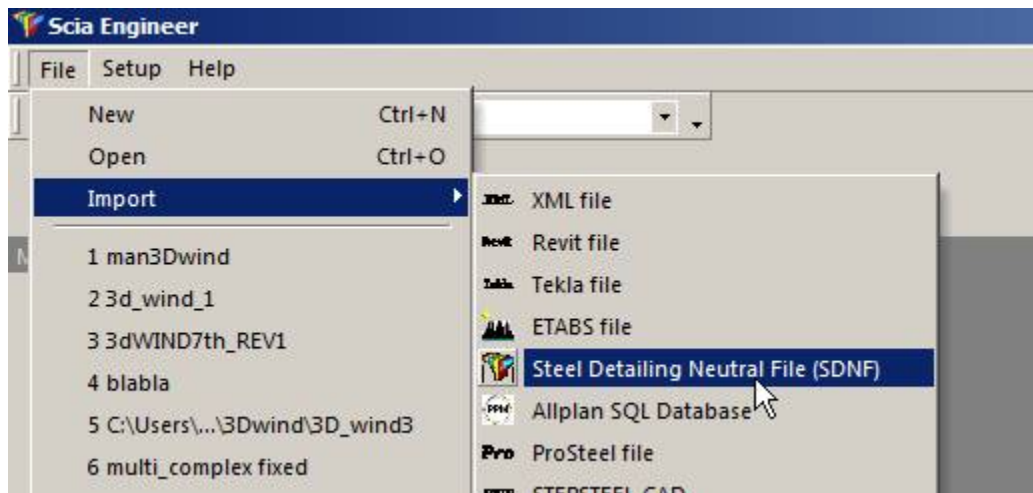
Add data

Supports/hinges

Supports and hinges are automatically generated from the defined releases in the file. The conversion is based on certain rules and not always the result is as what the user expects.

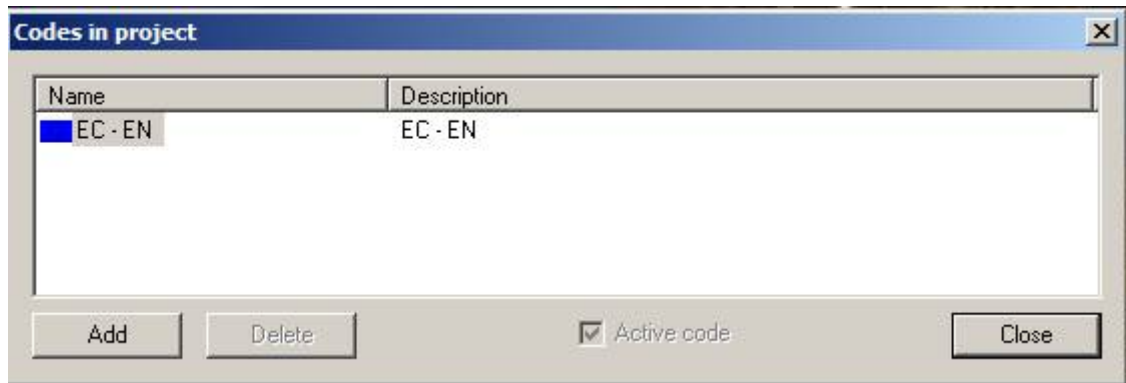
Procedure:

1. Run File > Import > SDNF file

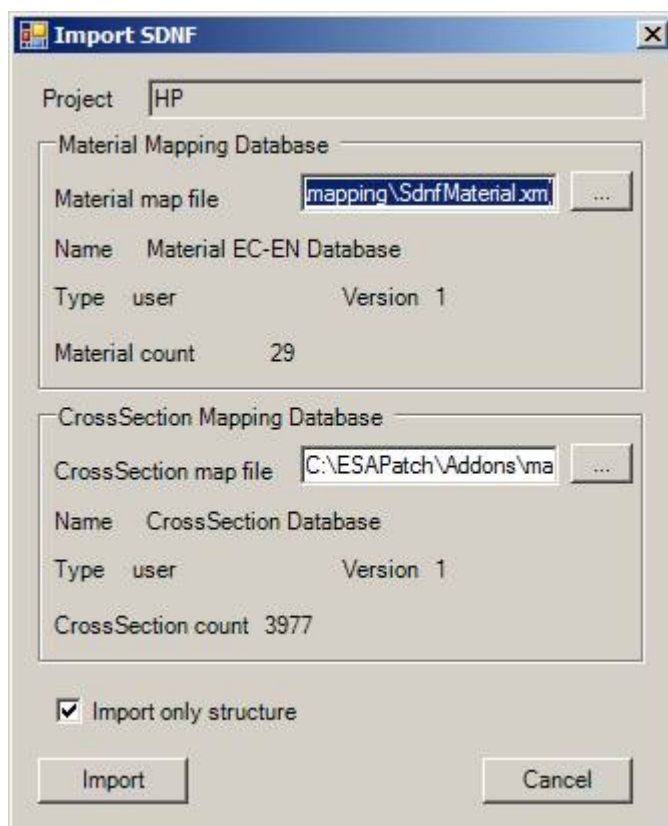


2. Choose .sdn file and confirm dialog Open. The version of SDNF is recognized by a record inside the file.

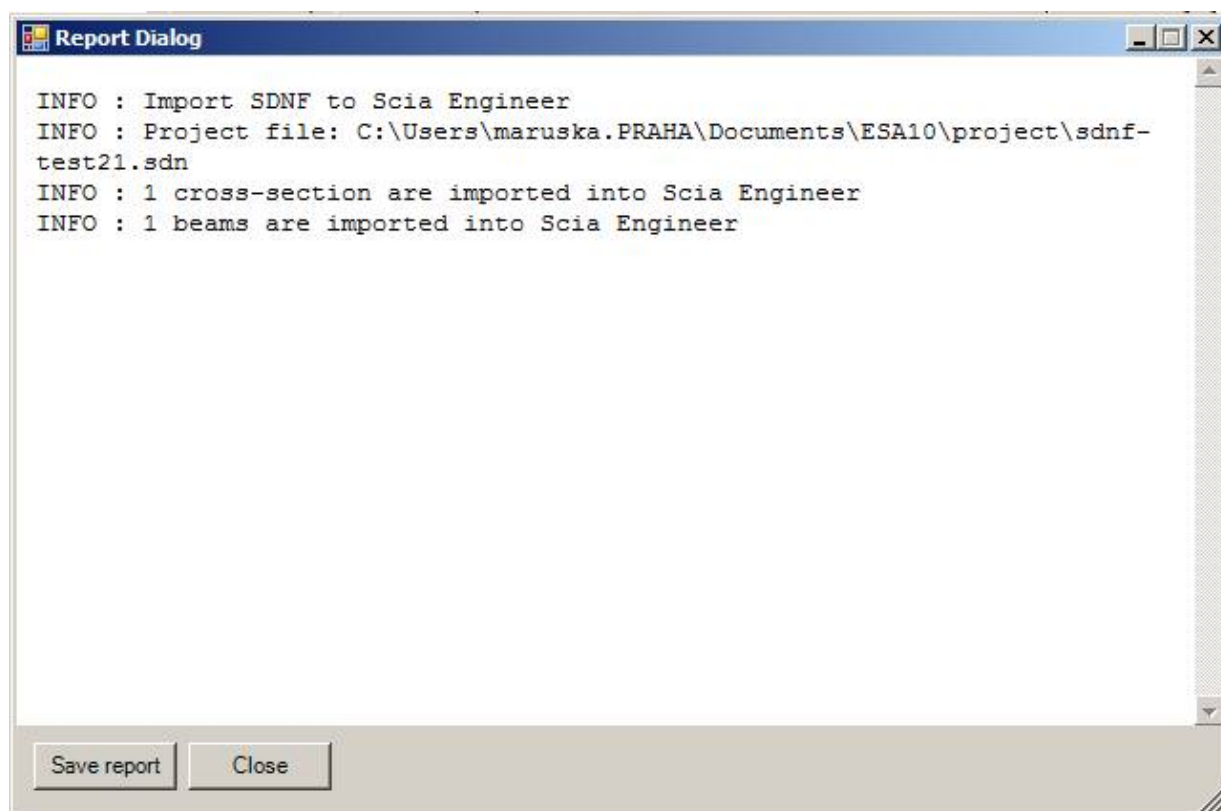
3. Choose a code



4. Import dialog is displayed. It is possible to select user defined tables for profile and material conversion. At the bottom of dialog there is a check box **Import only structure**. If ON, only 1D members with properties will be imported. If OFF the releases defined in the file will be taken in account.



5. After the import a report is shown summarising how many entities were imported.



6. A project is opened.

Export

Project data

Project data which will be saved to the file:

- Name is "Project Id"
- Part is "Structure Id"
- Description is "Client Id"
- Author is "Engineering Firm Id"
- Date is correctly written to *.sdn file

1D members

Member types

- column is exported as column
- type beam is exported as beam
- others types are exported as brace

Profile names and material names are written to the file. No conversion table is possible for export

Member properties

- member system is set as correct cardinal points number

- alpha of cross-section and/or LCS rotation
- eccentricity – ez,ey is written as cross-section offsets
- correct position and orientation of begin and end node of member

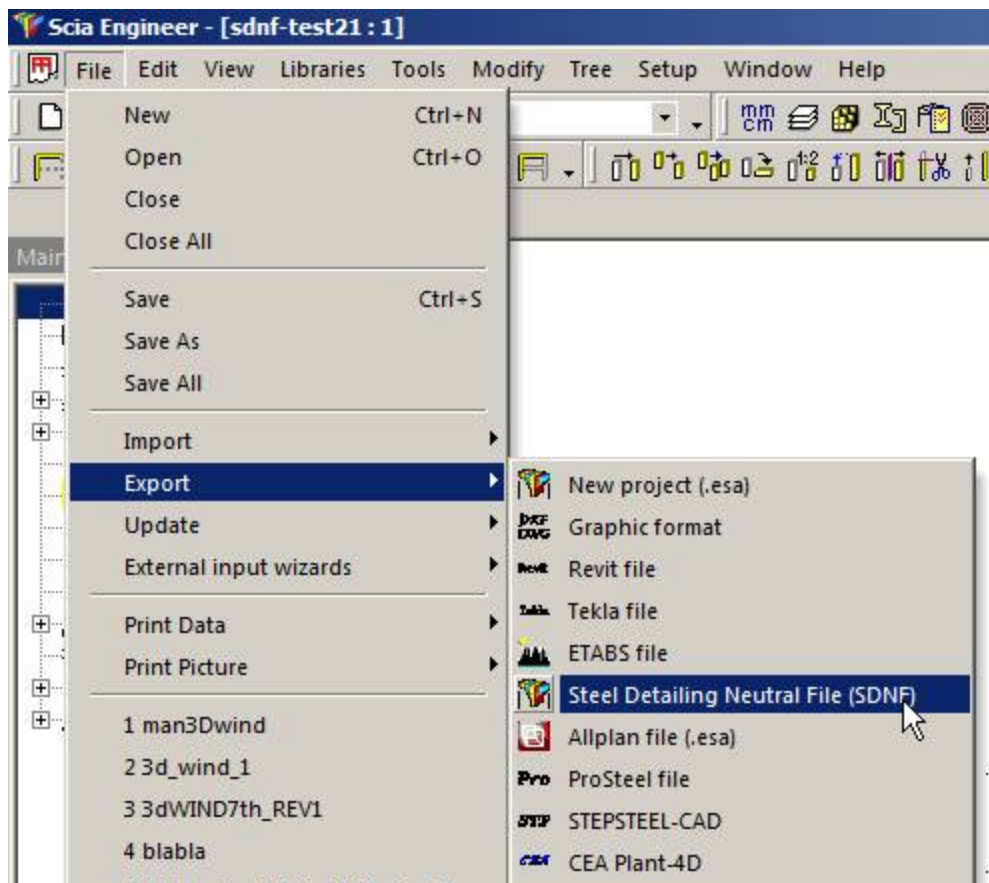
Add data

Supports/hinges

- both is exported as member releases to the file

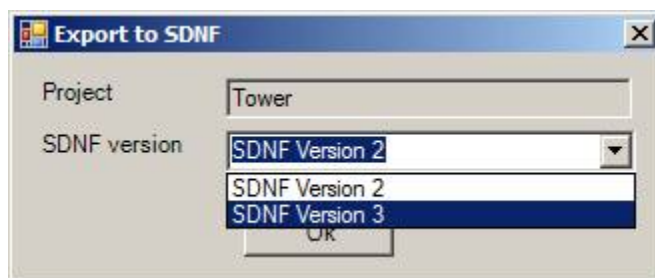
Procedure:

1. Run File > Export > SDNF file

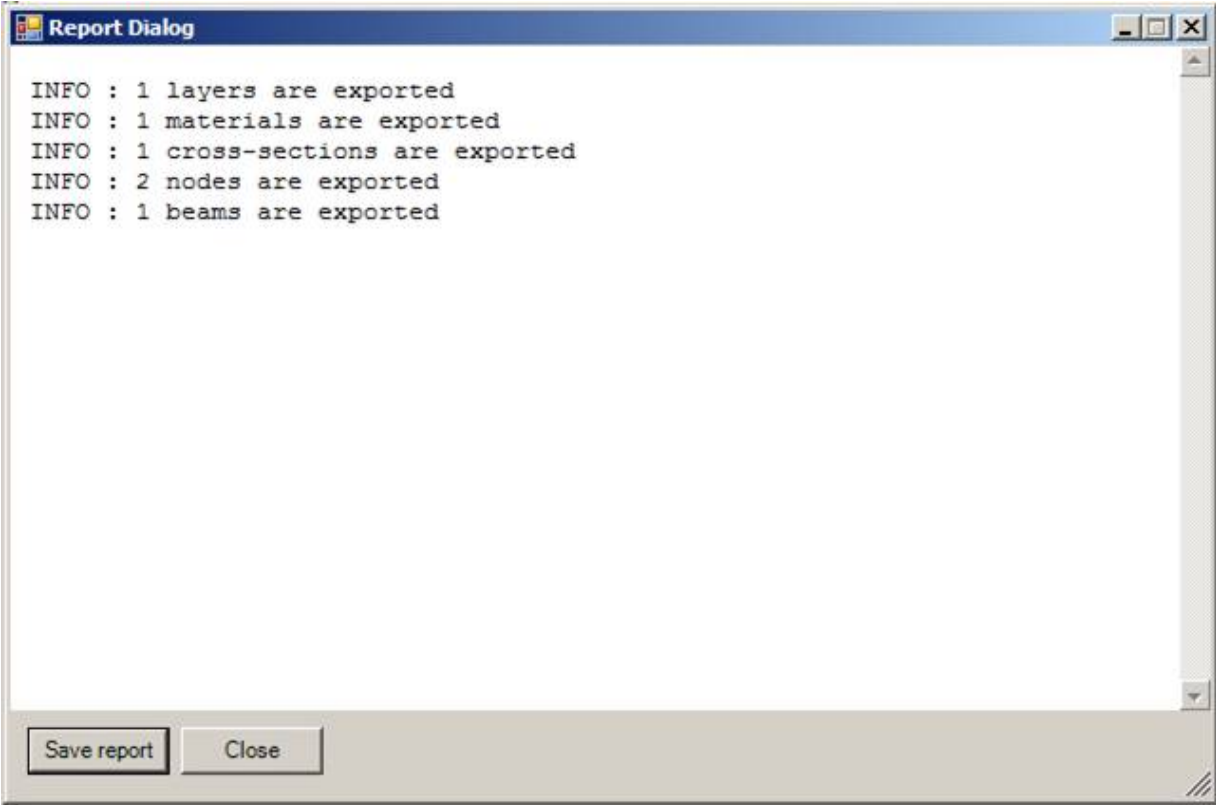


2. Define a file name and confirm dialog Save as.

3. Export dialog is displayed. The user can choose the version of SDNF format and confirm with [OK].



4. Export report dialog is opened.



4. A file with sdn file extension is created.

DSTV

DSTV

Scia Engineer allows for both [import](#) and [export](#) of files in DSTV format.
The interface is based on the format according to:
Schnittstellenkonvention Statik-CAD, Köln : Deutscher Stahlbau-Verband, 1993

Note: Only steel frame structures (i.e. structures composed of 1D members) can be imported/exported through DSTV format.

Note: More information about the DSTV format can be found in a separate SCIA manual titled Scia Engineer ASCII Import/Export Manual.

DSTV import

The import is based on the file with extension SC2. Parts *B00, *B01, *B02, *B03 and *B04 of this file are read into Scia Engineer.

The contents of the .SC2 file (origin : analysis)

Part	Description
*B00	General information (units, delimiters, etc.)
*B01	Topology of 1D members
*B02	Nodal co-ordinates
*B03	Names of cross-sections
*B04	Properties of 1D members

Note: More information about the DSTV format can be found in a separate SCIA manual titled Scia Engineer ASCII Import/Export Manual.

The procedure to import data from DSTV file

1. Start menu function **File > Import > DSTV**.
2. Browse for the required .SC2 file.
3. Finish the import.

Note: The import function creates a new empty project and the specified file is imported into it.

DSTV export

Scia Engineer generates files with extension SC1, SC2, SC3.

The contents of .SC1 file (origin : CAD)

Part	Description
*A01	Assembly definition

The contents of .SC2 file (origin : analysis)

Part	Description
*B00	General information (units, delimiters, etc.)
*B01	Topology of 1D members
*B02	Nodal co-ordinates
*B03	Names of cross-sections
*B04	Properties of 1D members

The cross-section names written in this file can be the original Scia Engineer names or the corresponding DSTV names. In order to have the DSTV names in the output file, you have to select the option "Use DSTV name conversion for section name". A dialogue with this option appears on the screen during the export procedure.

The contents of .SC3 file (origin : analysis)

Part	Description
*C01	Assembly definition

Note: More information about the DSTV format can be found in a separate SCIA manual titled Scia Engineer ASCII Import/Export Manual.

The procedure to export data into DSTV file

1. Start menu function **File > Export > DSTV**.
2. Specify the required output folder.
3. Type the name of the file (i.e. the base name without extension).
4. Confirm the input.
5. During the export procedure a dialogue opens on the screen.
6. Decide whether to "Use DSTV name conversion for section name" or not.
7. Complete the export.

Pro Steel

The exchange of data between Scia Engineer and ProSteel programs is achieved through a special format developed by companies KIWI software and SCIA. It is a XML-based format and the files use extension .PRO.

The format has been designed to support the following entities:

- straight steel 1D members,
- rolled cross-sections,
- connections: (i) end plate and (ii) frame connections for strong axis.

In order to use this feature, the user must have installed ProSteel v.17 or later.

Both [import](#), [export](#) and [update](#) functionalities are available for this format.

Pro Steel import

The procedure to import data from Pro Steel file

1. Start menu function **File > Import > ProSteel**.
2. Browse for the required input file.
3. The import dialogue opens on the screen.
4. Set the required options – see below.
5. Confirm with [**Import**].

Import from ProSteel dialogue

"graphical window of the dialogue"	<p>It displays the structure to-be-imported. The user may preview here the data from the file-to-be-imported. In addition, the window makes it possible to review the result of the automatic connection of 1D members.</p> <p>The window supports standard features of Scia Engineer graphical windows:</p> <ul style="list-style-type: none"> (i) pop-up menu with a set of zoom, print, save etc. functions, (ii) [Ctrl] + [Shift] + right-click and drag to zoom in and out the drawing, (iii) [Shift] + right-click and drag to move the drawing, (iv) [Ctrl] + right-click and drag to rotate the drawing.
Auto-connect members	<p>If ON, the import procedure automatically tries to connect entities of the imported model in such a way that the result is a correct analysis model.</p> <p>If OFF, the 1D members are imported in their actual positions, which does not guarantee that the imported model can be analysed by means of FEM algorithm. Some modifications and tuning of the imported model may be necessary to satisfy the prerequisites of the finite element method.</p>
Max. distance	If the distance of two points in the imported

	<p>structure is smaller than the value specified here, the program tries to "merge" the points into one.</p> <p>If the distance of two points is greater than the value specified here, the two points are treated as separate points and the connection algorithm does not try to "merge" them.</p>
Display report	If ON, the program issues a report about the import.
Refresh image	<p>This button performs in fact two actions:</p> <p>(i) it starts the connection algorithm which connects the 1D members, so that the result of the import is a correct FEM analysis model,</p> <p>(ii) it refreshes the image in the graphical window of the dialogue.</p>
Import	<p>This button imports the data into the main environment of Scia Engineer.</p> <p>Note: If the option Auto-connect members is ON and the button [Refresh image] has not been clicked yet, the button [Import] is disabled. The reason is that the user should review in the graphical window the result of the connection algorithm, so that he/she may get the chance not to accept it.</p>
Cancel	This button enables the user to abandon the current action.
CAD model	This disabled item has no meaning in the current version.

Note: The import function creates a new empty project and the specified file is imported into it.

Pro Steel export

The procedure to export data into ProSteel file

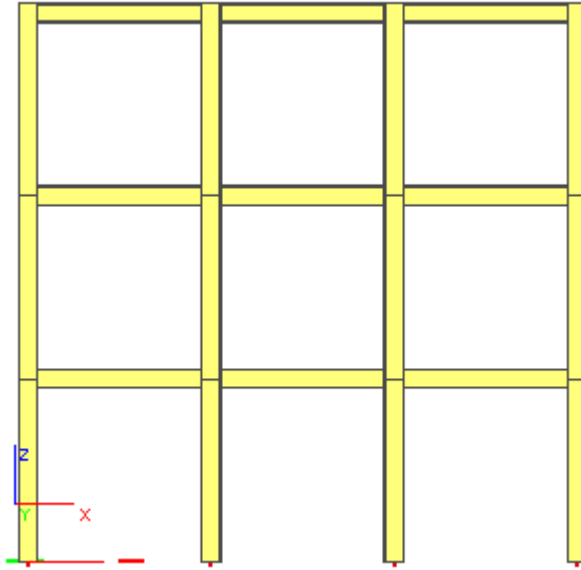
1. Start menu function **File > Export > Pro Steel file**.
2. Specify the required output folder.
3. Type the name of the file (i.e. the base name without extension).
4. Confirm the dialogue.
5. During the export procedure another dialogue opens on the screen.
6. Select if you want to export the analysis or structural model – see below.
7. Complete the export.

Structural versus analysis model

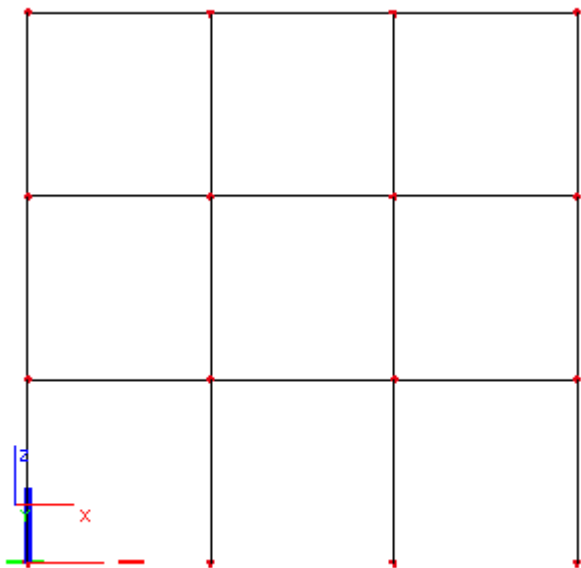
Scia Engineer enables the user to export either the analysis model (used for FEM calculations) or the structural model (used for drawings).

The difference between the models can be best demonstrated on a simple example of a planar frame.

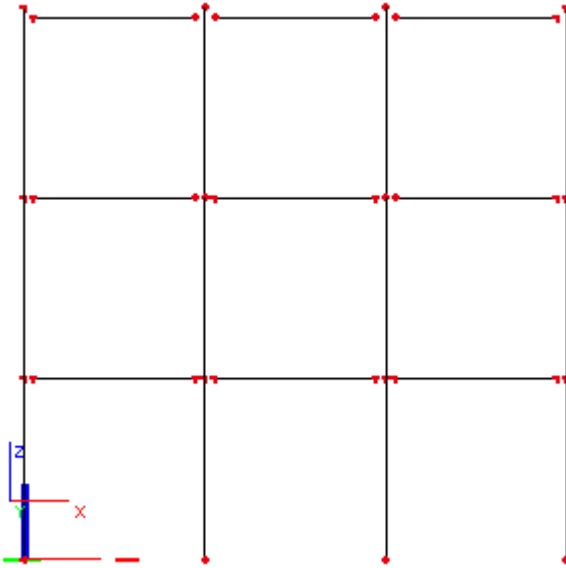
Let us have a simple frame:



The analysis model is the shape idealised for the needs of FEM analysis. It is an ideal axial model of the structure.



On the other hand, the structural model (shown in fact in the first picture) shows the real position and connection of individual structural members. Consequently, the "axial" representation of such model is different from the idealised FEM model.



The difference (and also problem during the import into calculation programs) is that the 1D members and columns do not intersect each other. This problem must be sorted out (somehow) after the import of the structural model into a FEM-analysis program.

Pro Steel update

Once a project created (or at least processed) in Scia Engineer has been exported to ProSteel format and handled outside of Scia Engineer, it is likely that some changes have been made to the project data.

In general, this may be a cause of troubles as someone must review both variants of the model (project in Scia Engineer and the third-party-processed file) and decide what to do.

In order to facilitate this task, Scia Engineer offers the **Update** function that compares the two variants and merges them together according to user-defined rules.

The procedure to update the project through ProSteel file

1. Open in Scia Engineer the project to-be-updated.
2. Call function **File > Update > ProSteel**.
3. Browse for the file to-be-merged.
4. The **Update from ProSteel** dialogue is opened on the screen – see below.
5. Select the type of changes that you want to accept.
6. Confirm with **[Update]**.
7. If some new entities were added by the third-party, the [Import from ProSteel dialogue](#) is opened on the screen.
8. Fill in the required options and perform necessary actions.
9. The update is completed.

Update from ProSteel dialogue

Group Update

This group contains a list of types of items that are compared during the update operation. If some items are not relevant for the particular update operation, they are disabled. Only the enabled items have been diagnosed by the update algorithm as relevant ones.

Added members

If the ProSteel file contains some new members (in comparison with the currently open project), the user may use this checkbox to decide whether the added members should be merged (item is ON) or ignored (item is OFF).

Deleted members	Analogous to the above. If ON, the members deleted from the merged file are deleted from the current project. If OFF, any deletions made in the ProSteel file are ignored.
Changed geometry	If ON, all the changes of geometry are taken into account. If OFF, the changes of geometry are ignored. The geometry remains as it is in the currently open project.
Changed cross-section	If ON, any changes of cross-sections are taken into account. If OFF, the changes of cross-sections are ignored. The cross-sections remain as they are in the currently open project.
Added connections	Analogous to above – but this time related to connections.
Deleted connections	Analogous to above – but this time related to connections.
Changed connections	Analogous to above – but this time related to connections.

Other controls


Display report	If ON, a report is generated about the update operation.
Button [Update]	This button performs the update (taking into account the adjustments made in group Update).
Button [Cancel]	This button cancels the whole update operation.

Graphical window

It displays the structure to-be-updated and shows the found changes. The changes are displayed in colour. Each type of change uses a specific colour that is indicated in the group **Update** of the dialogue.

The window supports standard features of Scia Engineer graphical windows:

- (i) pop-up menu with a set of zoom, print, save etc. functions,
- (ii) **[Ctrl] + [Shift] + right-click** and drag to zoom in and out the drawing,
- (iii) **[Shift] + right-click** and drag to move the drawing,
- (iv) **[Ctrl] + right-click** and drag to rotate the drawing.

 Note: The behaviour of the update-function may differ in some details from the description given here as the interface is still under development on both sides, i.e. Scia Engineer and ProSteel.

StepSteel - analysis model

The interface provides for the exchange of data with StepSteel and is based on the following references:

(1)

Standardbeschreibung Produktschnittstelle Stahlbau

Teil 1 : Empfehlungen für den Anwender

DSTV – Arbeitsausschuss EDV

April 2000

(2)

Standardbeschreibung Produktschnittstelle Stahlbau

Teil 2 : DatenModell

DSTV – Arbeitsausschuss EDV

April 2000

(3)

Standardbeschreibung Produktschnittstelle Stahlbau

Teil 3 : Implementierungsbereiche und Konformitätsanforderungen

DSTV – Arbeitsausschuss EDV

April 2000

The implemented EXPRESS-scheme is PSS_2000_04.exp.

Scia Engineer allows for both [import](#) and [export](#) of StepSteel files. In addition to the analysis model, also the structural model can be imported and exported – this is [described in separate topics](#).

Note: Only steel frame structures (i.e. structures composed of 1D members) can be imported/exported through StepSteel format.

StepSteel - import of analysis model

The procedure to import data from StepSteel file

1. Start menu function **File > Import > StepSteel**.
2. Browse for the required input file.
3. Finish the import.

Note: The import function creates a new empty project and the specified file is imported into it.

Note: More information about the StepSteel format can be found in a separate SCIA manual titled Scia Engineer ASCII Import/Export Manual.

StepSteel - export of analysis model

The procedure to export data into StepSteel file

1. Start menu function **File > Export > StepSteel**.
2. Specify the required output folder.
3. Type the name of the file (i.e. the base name without extension).
4. Select the output level – see below.
5. Adjust parameters relating to section names – see below.
6. Complete the export.

Available implementation levels

Implementation level	Entity group	Abbreviation	Note
Reading structure 'Statische Einlesen'	analysis Struktur	General data Analysis data I	1(IN) Not available for export
Writing structure 'Statische Schreiben'	analysis Struktur	General data Analysis data I	1(OUT)
Reading analysis 'Statik Einlesen'		General data Analysis data I Analysis data II	2(IN) Not available for export
Writing analysis 'Statik Schreiben'		General data Analysis data I Analysis data II	2(OUT) Two options available for export: with and without results.
Writing design 'Entwurf Schreiben'		General data Design data	3(OUT)

Note: More information about the StepSteel format can be found in a separate SCIA manual titled Scia Engineer ASCII Import/Export Manual.

Section names

The cross-section shape is recognised from the name of the cross-section. The user may adjust a few parameters that influence the applied conversion of cross-section names.

Use DSTV conversion for section names	If ON, the DSTV convention is used for cross-section names.
Use mapping table as conversion for section names	If ON, the mapping table is used to convert the names of cross-sections.

StepSteel - structural model

The [import](#) and [export](#) of structural model is an extension to the [import and export of the analysis model](#).

Note: Only steel frame structures (i.e. structures composed of 1D members) can be imported/exported through StepSteel format.

StepSteel - import of structural model

The procedure to import data from StepSteel-CAD file

1. Start menu function **File > Import > StepSteel-CAD**.
2. Browse for the required input file.
3. Finish the import.

Note: The import function creates a new empty project and the specified file is imported into it.

Note: More information about the StepSteel format can be found in a separate SCIA manual titled Scia Engineer ASCII Import/Export Manual.

StepSteel - export of structural model

The procedure to export data into StepSteel-CAD file

1. Start menu function **File > Export > StepSteel-CAD**.
2. Specify the required output folder.
3. Type the name of the file (i.e. the base name without extension).
4. Complete the export.

Note: More information about the StepSteel format can be found in a separate SCIA manual titled Scia Engineer ASCII Import/Export Manual.

CEA Pland-4D

Scia Engineer allows for export of project data into CEA Pland-4D file. This option is available only if the corresponding software is installed on your computer.

Tekla

Note: This import/export capability is not directly related to Scia Engineer functionality. Nevertheless, it can prove useful to Scia Engineer users. Please, consider this chapter to be only a brief introduction and contact SCIA support team for more information and/or guidance.

Note: The interface works with Tekla Structures versions 15 and 16.

If IFC format is not a solution for you to export to and from Tekla Structures, you can use the STEP Steel format. Alternatively, export can be made through Tekla Structure API link.

STEP Steel format

Tekla to Scia Engineer

1. Export a project by **StepSteel export** option in Tekla Structures to e.g. "Tekla_output.stp".
2. Open "Tekla_output.stp", and replace the text 'FILE_SCHEMA(('DATA_SECTION_SCHEMA_3_99'))' with 'FILE_SCHEMA(('PSS_2000_04'))' in the header, then save it again.
3. Now import this output-stp ("Tekla_output.stp" into Scia Engineer.
4. Make necessary modifications, analyses, checks, etc.
5. Export the structure into StepSteel CAD format (not levels 1-3), e.g. to "output01.stp" (see chapter [StepSteel - export of structural model](#)).
6. Change the header back: So now replace 'FILE_SCHEMA(('PSS_2000_04'))' with 'FILE_SCHEMA(('DATA_SECTION_SCHEMA_3_99'))'.
7. Then save it as "output01e.stp".

Scia Engineer to Tekla

1. In Scia Engineer you have to generate a new STP-File (e.g."Esa_output.stp". Choose the option: Stepsteel export CAD (not levels 1-3) (see chapter [StepSteel - export of structural model](#)).
2. Open "Esa_output.stp", and replace the text 'FILE_SCHEMA(('PSS_2000_04'))' with 'FILE_SCHEMA(('DATA_SECTION_SCHEMA_3_99'))' in the header, then save it again.
3. Now import this output-stp ("Esa_output.stp" into Tekla Structures.

Tekla Structure API link

The link is based on Tekla Structure API supports the export of:

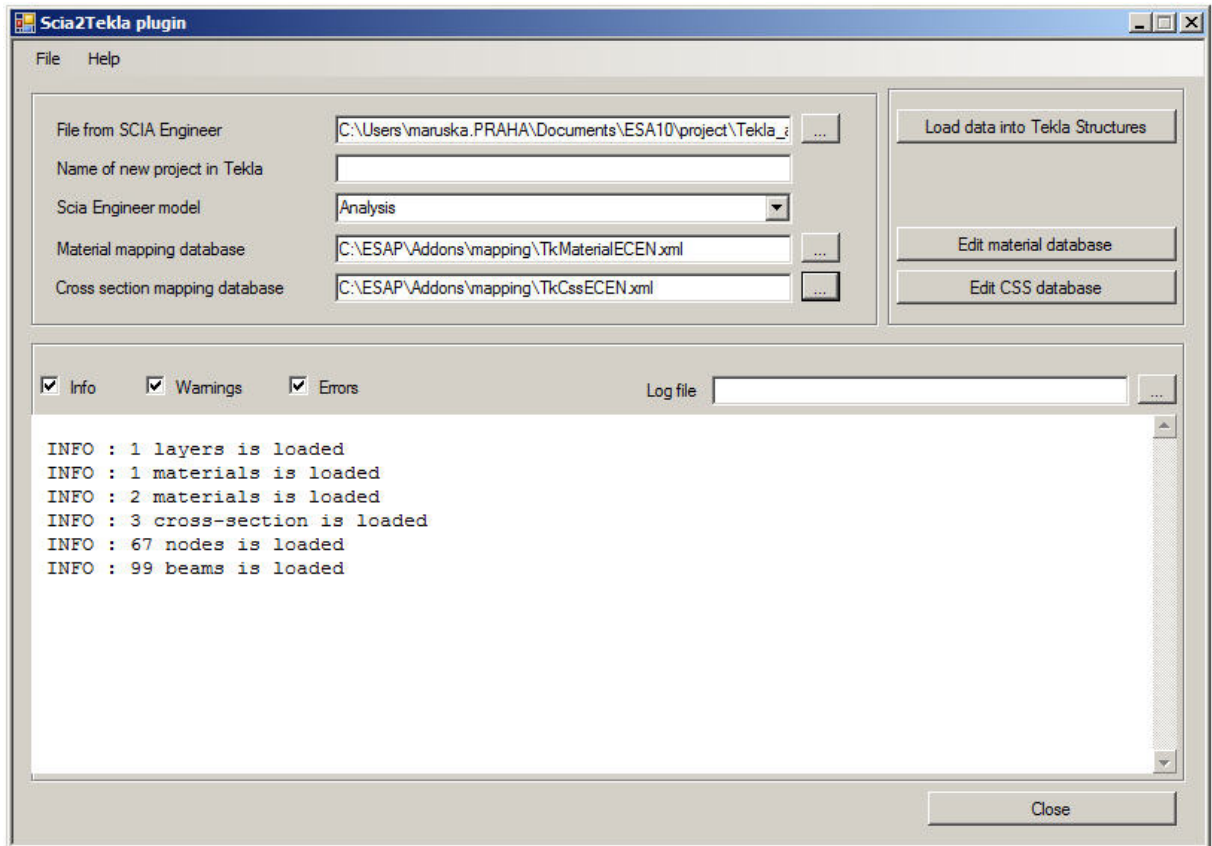
- Geometry of straight beams and columns (beginning- and end-nodes)
- Materials using the mapping database
- Cross-sections due to mapping database or geometrical profiles (except twin profiles)
- Eccentricity, ey,ez
- Member system-lines
- welded cross-section,
- hinges.

General procedure

1. Call **File > Export > Tekla file**.
2. File with .s2t extension is created.
3. Then it is needed to run **Tekla plugin – Scia2Tekla.exe**.
4. Load .s2t file, material and cross-section mapping database - otherwise the default profile (rectangle 10x10) is used.
5. Press button **Load Data into Tekla Structures**.
6. In Tekla Structures the project is imported.

Note: Tekla Structure has to be running before the step 5 is done.

Export dialogue



Menu File

- New – opens a new dialogue Scia2Tekla plugin without any settings (clears previous settings)
- Open – opens a dialogue for to select an existing *.sts (Scia to Tekla settings) file.
- Save – saves the user settings in the dialogue to *.sts (Scia to Tekla settings) file.
- Save as – saves the user settings in the dialogue to *.sts (Scia to Tekla settings) file.
- Close – closes the plugin dialogue .

Dialogue items

File from Scia Engineer

It loads s2t file which should be imported to Tekla Structure.

Name of new project in Tekla

It is possible to define a name of Tekla model.

Scia Engineer model

Either the Analysis or the Structural model can be selected for export.

Material mapping database

It loads XML file which contains conversion table for materials

Cross section mapping database

It loads XML file which contains conversion table for profiles

[Load data into Tekla Structures]

It runs an import of defined file to Tekla Structures

[Edit material database]

It runs a dialogue for defining and editing of conversion table for materials

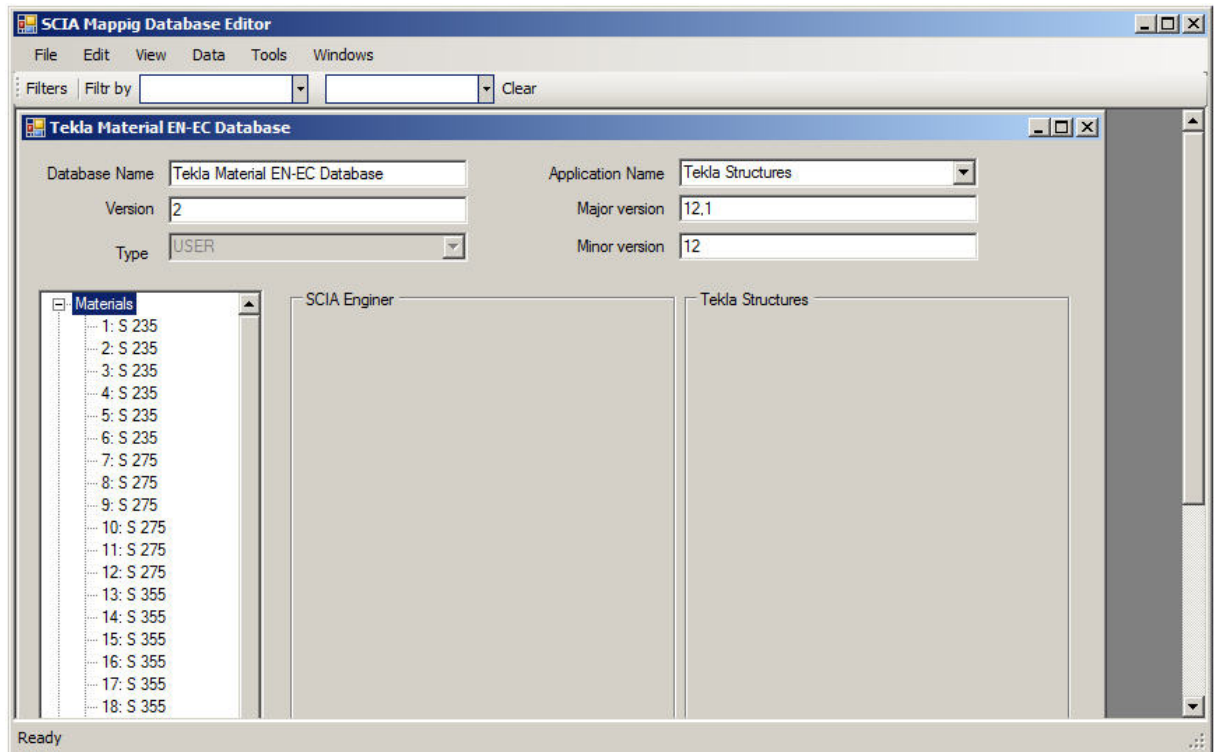
[Edit CSS database]

It runs a dialogue for defining and editing of conversion table for cross-sections

[Close]

It closes the dialogue.

At the bottom of dialogue is a window where a log is written. Above this window it is possible to choose what kind of information should be displayed. On the right side it is possible to define a text file to which the log will be saved.

Mapping database editor

Mapping databases are stored in XML.

The mapping database contains information about versions of Tekla structures for which it was tested.

Two working modes are available: Administrator (SCIA employee) and User:

- Administrator can create the master database,
- User can create only "user database".

Google Earth

Google Earth

Note: This import/export capability is not directly related to Scia Engineer functionality. Nevertheless, it can prove useful to Scia Engineer users. Please, consider this chapter to be only a brief introduction and contact SCIA support team for more information and/or guidance.

The export of your structure to Google Earth can be done in two steps.

1. The model of the structure must be exported to IFC format (see chapter IFC export).
2. The IFC file must be converted into .KML format. This format can be then opened in Google Earth.

Tip: A free tool lfcStoreyView can be used to make the conversion from .IFC to .KML format.

Interface with ETABS

Why and how to use?

There are lots of reasons why the interface between Scia Engineer and ETABS may be . Some users can need to use curved members others can need to use a steel check of hall. The next ones can want only to work with more customized environment. All this we can to provide to users. But they have got more projects in ETABS format and they surely do not want to lost them with changing of software and this is the reason why the interface with ETABS was developed. User can import project to Scia Engineer and to finish her/his work there. This interface cannot import everything at this moments but it is clear then a development of the interface will be continue in next versions of Scia Engineer.

What is better in Scia Engineer?

It knows:

- Curved beams.
- Curved walls.
- Lots of possibility to create shells.
- Better displaying of structure.
- Different types of results.
- More possibilities of reinforcement of concrete structures.
- More possibilities of checks (steel, concrete,..).
- Steel connections etc.
- More type of supports – line, surface,..
- Variable load on slabs.
- Free loads.
- Variable thickness of 2D and subregions.
- Haunches and arbitrary beams.

e2k file

The exchange of data between ETABS and Scia Engineer is done through the e2k file. This type of file can be created and also read-back in a standard version of ETABS. Scia Engineer offers the possibility to import and export this type of file in its standard **File > Import** and **File > Export menu**.

Import

This chapter explains what data are read during the import of e2k file. At the end of the chapter you will find the procedure how to make the import of e2k file to Scia Engineer.

Project data

The screenshot shows the 'Project data' dialog box with the following details:

- Basic data tab:**
 - Data:** Name: Etabs-Scia Engineer Sample Project; Part: -; Description: ETABS VERSION 9.14; Author: My Name; Date: 02. 10. 2009.
 - Structure:** General XYZ
 - Project Level:** Advanced
 - Model:** One
- Material section:**

Concrete	<input checked="" type="checkbox"/>
Material	C12/15
Reinforcement m...	B 400A
Steel	<input checked="" type="checkbox"/>
Material	S 235
Timber	<input type="checkbox"/>
Other	<input type="checkbox"/>
Aluminium	<input type="checkbox"/>
- Code section:**
 - National Code:** EC - EN
 - National annex:** EC-EN

Name

Name is filled in with the **Model Name** defined in e2k file.

Author

Author is filled in with the **Company Name** defined in e2k file.

Description

In Description there is the **Etabs version** which the e2k file was exported from.

Units

- Units are calculated in accordance with Scia Engineer units and the entire structure is imported in basic units (kN/m).

Nodes

- Names of nodes are generated during the import.

Beams and columns

- Beams are imported as 1D members with the type set to **beam**.
- Columns are imported as 1D members with the type set to **column**.
- Braces are imported as 1D members with the type set to **beam**.
- Line members with "none" profile are imported as 1D members with very a small cross-section and they are inserted to a special layer named **NullBeam** which is marked as **CAD only**. That means it is not taken in account during calculations.
- Properties: member system, type, cross-section.

Cross-sections

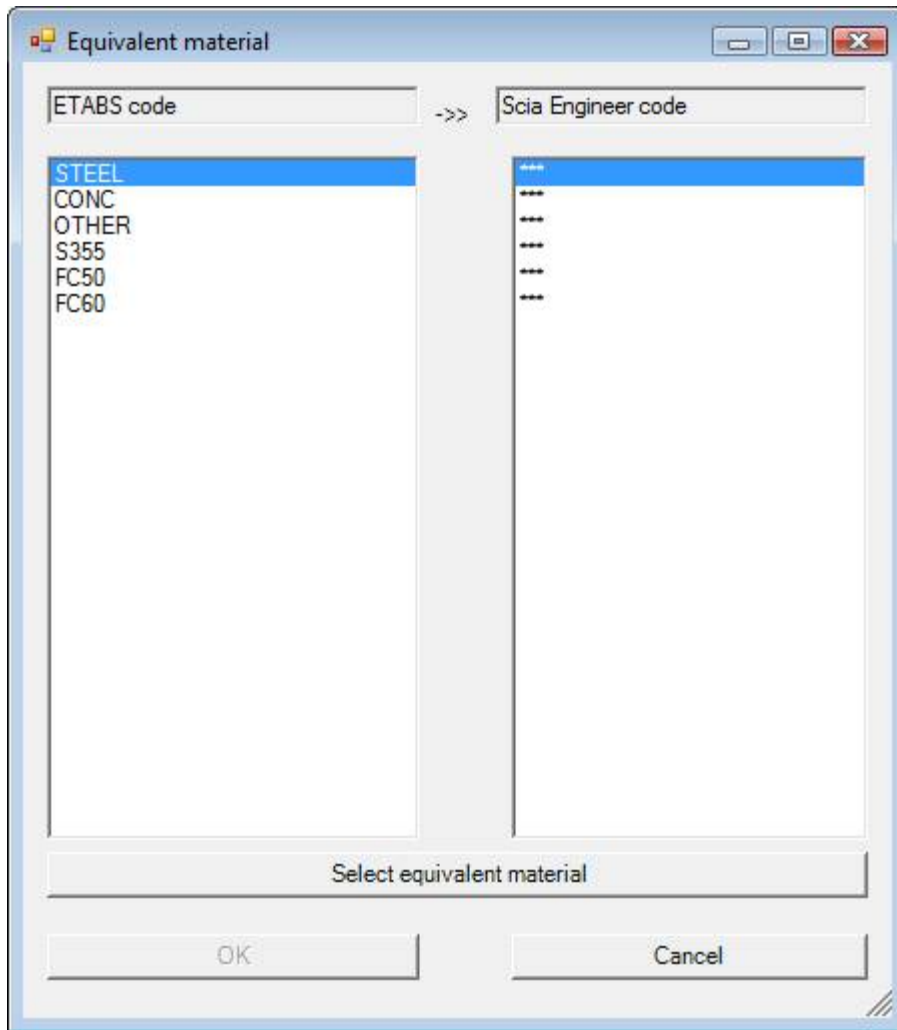
- Cross-section conversion table (a default one is distributed with Scia Engineer and users can create their own by means of tools distributed with Tekla-plugin).
- Geometric profiles (rectangular and circular shapes).
- If a profile is defined as auto-select section in ETABS, the first one from the auto-select section list is taken in account as the profile of the member during the import.

Walls and slabs

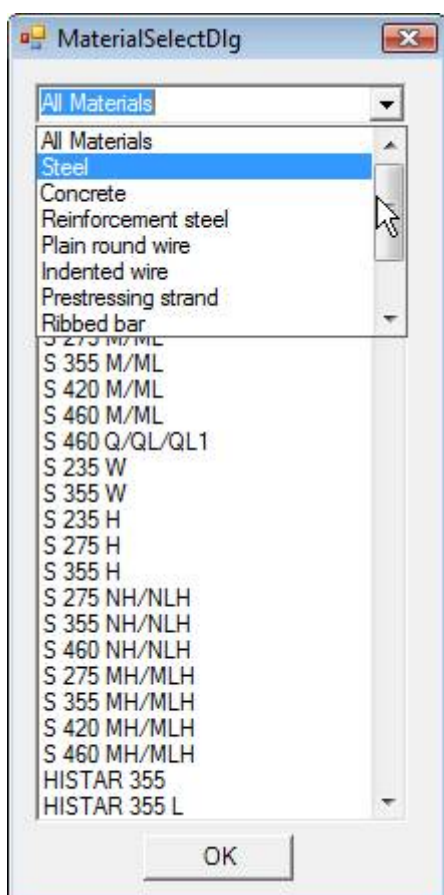
- Walls are imported as 2D members with the type set to **wall**.
- Slabs, decks and planks are imported as 2D members with the type set to **slab**.
- Ramps are imported as 2D members with the type set to **slab**.
- Area members with a "none" profile are imported as very thin 2D members that are inserted to a special layer named **NullSlab** and marked as **CAD only**. They are not taken in account during calculations.
- Openings.
- Properties: thickness, material, type.

Materials

- User defines a conversion table during the import. A dialog for the defining of material conversion appears on the screen during the import:



- On the left side there is a list of all materials defined in the imported e2k file. On the right side there will be a list of Scia Engineer materials which user assigns to each one from the ETABS file.
- Click button **[Define equivalent materials]**. A new dialog with a list of Scia Engineer materials for selected code is displayed.



- At the top part of the dialog there is a combo box where you can choose a type of material, such as steel or concrete, or all materials.
- Select which one you want to work with and confirm with **[OK]**.
- If all materials are defined a button **[OK]** is available in the **Equivalent material dialog** and the user can continue with the import.

Supports

- ETABS has got only point supports and these can be imported to Scia Engineer.
- Each restraint (Fx, Fy, Fz, Mx, My, Mz) is imported correctly as rigid or free. Springs or flexible supports are not supported.

Hinges

- Beam releases in ETABS are imported as hinges to Scia Engineer. In many cases it will be necessary to edit these hinges in Scia Engineer because their philosophy in Scia Engineer is different from the definition of releases in ETABS.
- Hinges on 2D member edges are not supported.

Loads

- Point load in point
 - Forces Fx, Fy, Fz, and moments Mx, My, Mz in global and local coordination systems are supported.
 - They are imported as native Scia Engineer forces in node.

- Point (repeating) loads on beam

Both forces and moments in global or local coordination systems are supported.

Repeated loads are imported as several separate point forces on beam. It is not possible to import them as native Scia Engineer repeated forces.
- Line loads on beam

Both forces and moments with a uniform or a trapezoid distribution in global or local coordination systems are supported.

Load with the trapezoidal distribution is imported as several line loads. Each part of trapezoidal load is an independent load in Scia Engineer.
- Surface load on slab

ETABS has got only a constant surface load which is supported for import to Scia Engineer.
- Temperature

Point temperature load is not imported because Scia Engineer has no point temperature implemented.

Line temperature load is imported only on 1D members.

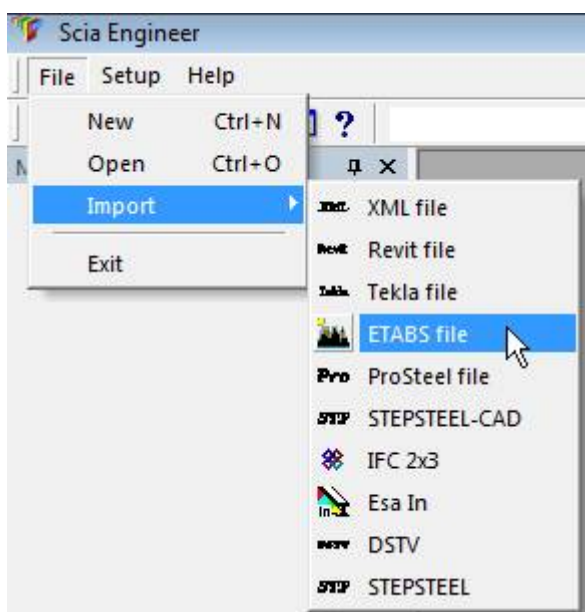
Surface temperature load is imported only on 2D members.
- Loads (except temperature) on line beam with a none profile are imported as free point or line loads with selection – select. In the selection there is only the closest member.
- Loads (except temperature) on area with none profile are imported as free surface loads with selection – select. Only the closest member is in the selection.
- Load cases

All static linear load cases are imported with their name, action type and load type to Scia Engineer.
- Load combination

All linear combinations are imported with their lists of load cases.

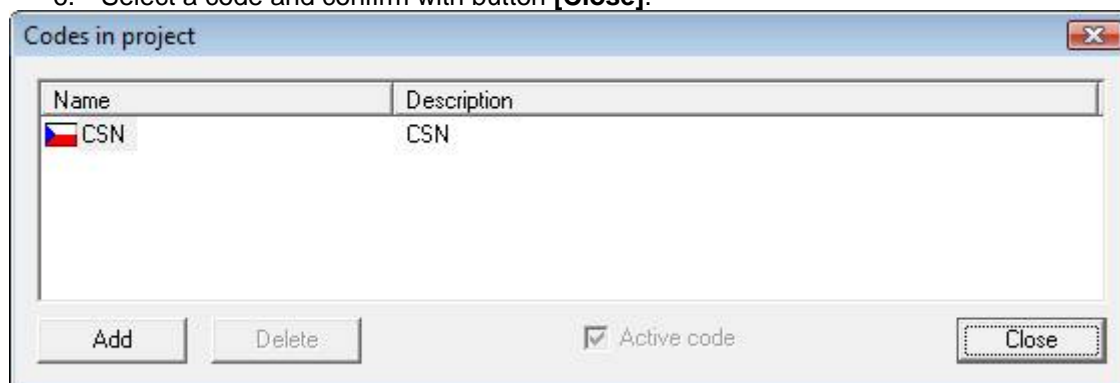
Procedure

1. Call function **File > Import > ETABS file**.



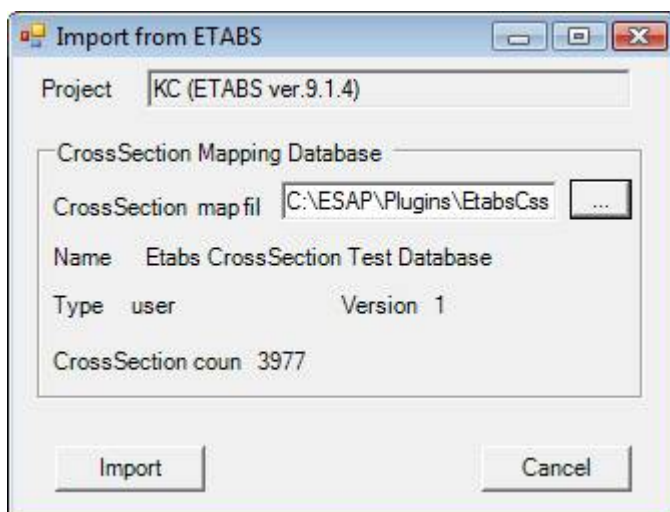
2. Dialog **Open file** is opened on the screen.

3. Select the required e2k file for import.
4. Confirm with **[Open]**.
5. Dialog **Codes in project** appears.
6. Select a code and confirm with button **[Close]**.

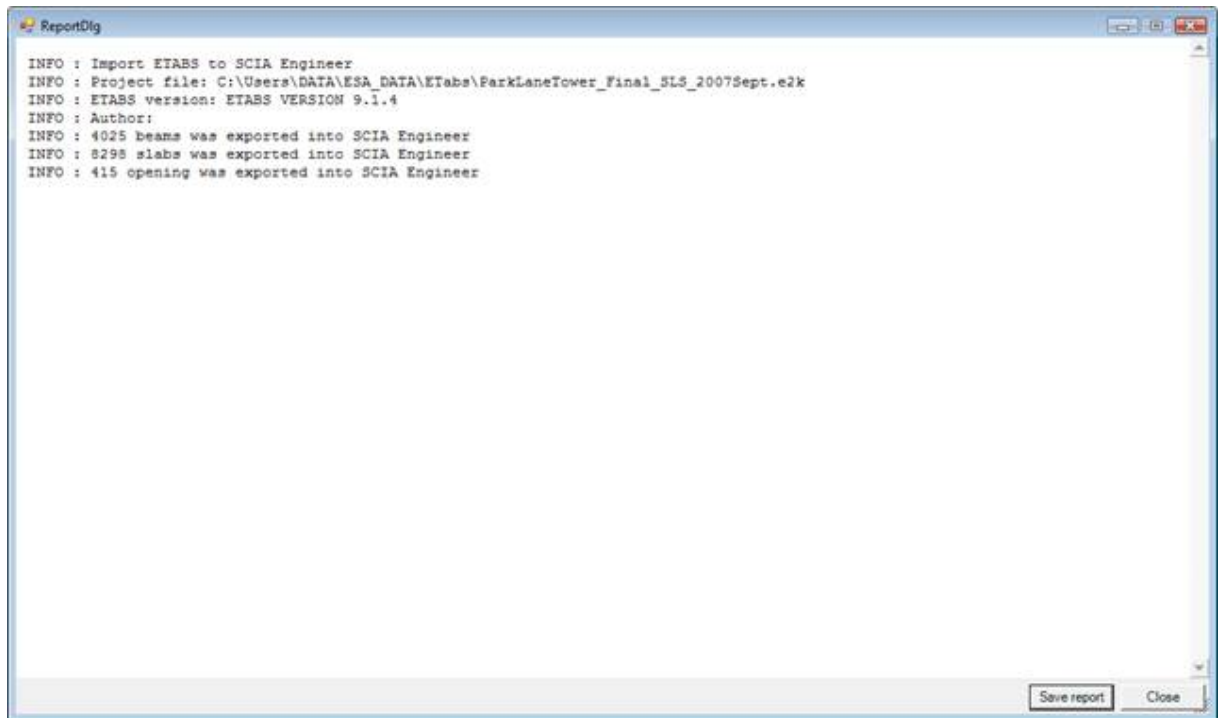


7. The **Import from ETABS** dialog is opened.

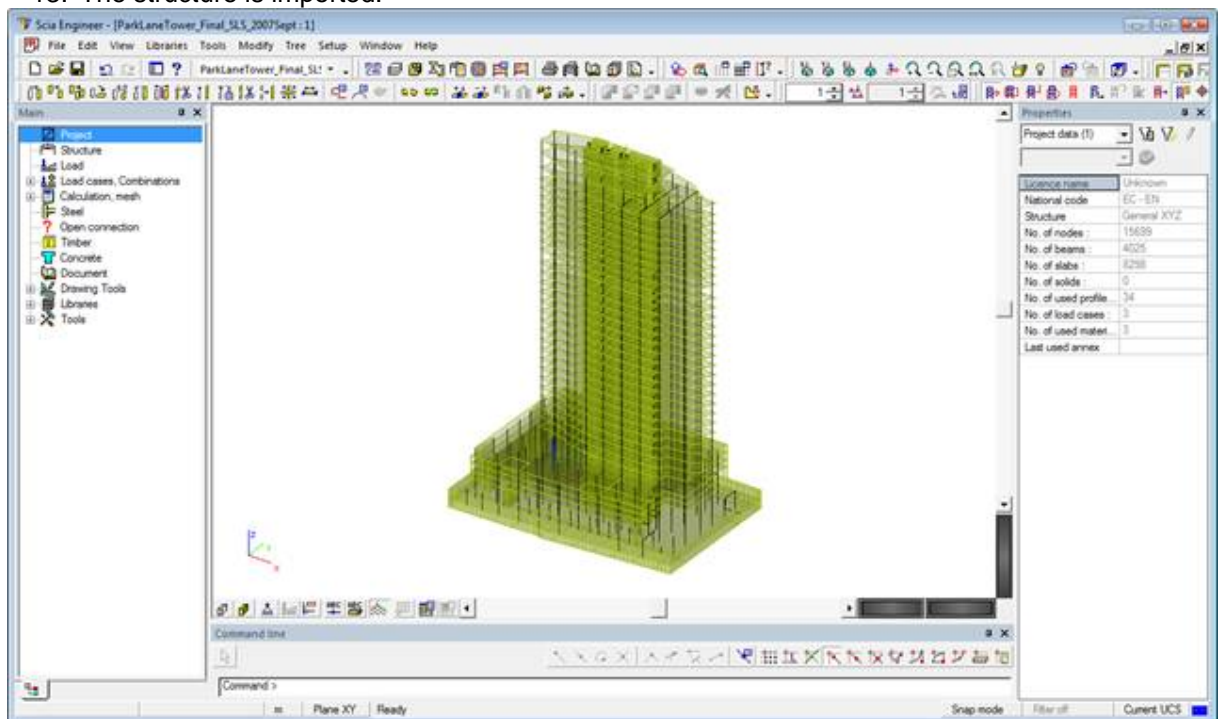
The ETABS version where e2k file was exported from is shown here. The user can choose the required user-defined cross-section mapping table or to keep the default one which is distributed with Scia Engineer.



8. Press button **[Import]**.
9. The dialog for material conversion defining is displayed.
10. Define the materials and confirm with **[OK]**.
11. The import is in progress. A big structure can take some time to import.
12. The log is shown.



13. The structure is imported.



Export

Storeys and line grid

- If storeys are defined in Scia Engineer project then they are exported to e2k files. It is important to have defined each member in the only storey. If any member is in no storey or in two or more storeys the member will be assigned to basic one or to the lower one. A basic line grid is generated during an export. No line grid from Scia Engineer is exported to e2k file.

Note: The export of storeys is supported from Scia Engineer 2011

Units

- A structure is always exported in basic units (kN/m).

Materials

- Each material is correctly written to an exported file.

Cross-sections

- For export of cross-sections the default (or user) conversion table is used. It is the same table which is used for import.

1D members

- Straight

For export only straight members without haunches, arbitrary shapes, openings, etc are supported. Polyline members are exported as several members. Member properties as angle, profile, material, member system and member type are taken in account. Columns and beams are exported as column and beams, others types are exported as brace to e2k file.

- Curved

No curved members are supported for export. All curved members are exported as straight.

2D members

- Flat

Only flat members with straight edges and openings and constant thickness and without subregions are supported. Member properties as material and member type are taken in account. Member types for export are only wall and slab. If a flat member has got a curved edge, the edge is exported by means of straight lines.

- Shells

Shells are not supported for export. They can be exported only as flat members to e2k file.

Supports

- Point in node

For export there are supported only point supports in nodes with rigid or free restrains. If a support with a flexible stiffness is defined it is exported as free.

- Others

Point supports on a beam or an internal node, line supports on beam or 2D member edge and surface supports are not supported for export and nothing will be exported to e2k file.

Hinges

- Hinges on beams are exported as beam releases to e2k file.
- Hinges on 2D member edge are not supported.

Loads

- Point in node

Point forces F_x , F_y , F_z , and moments M_x , M_y , M_z in a node in global and local coordination system are supported for export to e2k file.

Point force in an internal node is not supported.

- Point on beam

Both forces and moments in global or local coordination system are supported.

Repeated ones are exported as several point forces on beam.

- Free point

Free point forces or moments are not supported.

- Line on beam

Both forces and moments with uniform or trapezoidal distributions in global or local coordination system are supported for export.

- Line on edge

Line forces on 2D member edge are not supported.

- Free line

Free line forces are not supported.

- Surface

Only uniformly distributed surface load is supported for export to Scia Engineer.

- Free surface

Free surface load is not supported.

- Temperature

Only constant thermal loads on 2D members and 1D members are supported.

If the thermal load is defined as variable it is exported to e2k file as constant load with top delta z+.

- Load cases

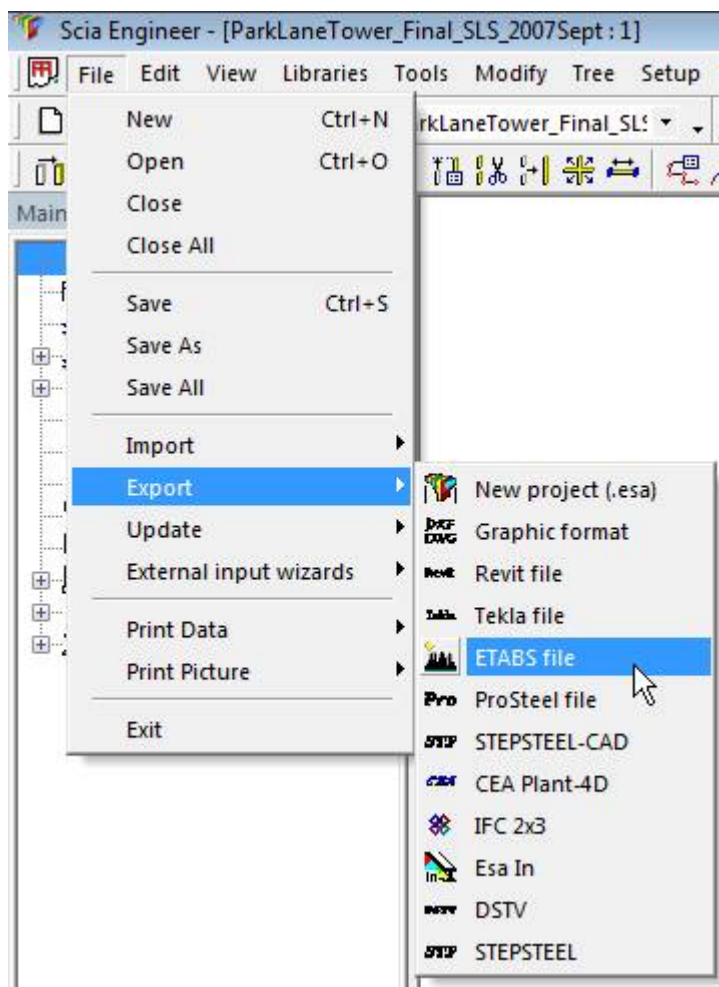
All load cases are exported to e2k file with their action type and load type.

- Load combination

All linear load combinations are exported to e2k file with a list of load case.

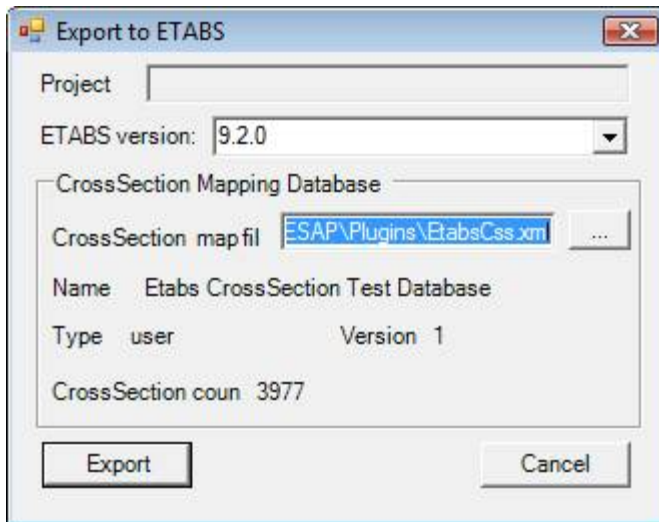
Procedure

1. Call function **File > Export > ETABS file**.

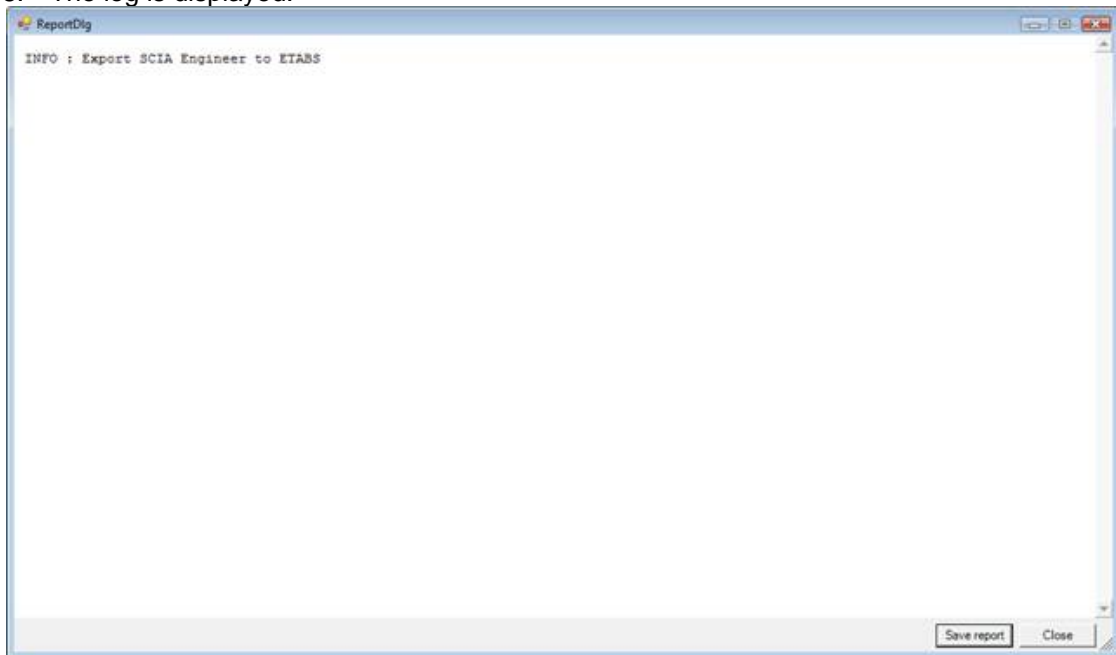


2. Dialog **[Save as]** is opened.
3. Define the name of the exported e2k file.
4. Confirm with **[Save]**.
5. Export dialog is opened.

It is possible to choose a supported version of ETABS where e2k file should be exported to. The user can choose a user cross-section mapping table or keep the default one which is distributed with Scia Engineer.



6. Press button **[Export]**.
7. The export is in progress.
8. The log is displayed.



9. The structure is exported to e2k file.

ECtools

ECtools© is a design software for Reinforced Concrete and Unreinforced Masonry Structures that aims to facilitate the practicing structural engineer or the design office in the correct application of the several clauses of Eurocodes 2, 6 and 8. It utilises the analysis results of SCIA Engineer© and can be used in the following cases:

- * Seismic design of Reinforced Concrete Buildings
- * Checking/Reviewing the Design of Reinforced Concrete Buildings
- * Seismic Assessment and Retrofitting of existing Reinforced Concrete Buildings
- * Seismic design of Composite buildings with Concrete Cores (The cores)
- * Seismic design of Unreinforced Masonry Buildings
- * Seismic Assessment and Retrofitting of existing Unreinforced Masonry Buildings

The procedure to export the project to ECtools

- 1) Run function: Plugins > ECtools > ECtools.
- 2) Type the name of the file and confirm with OK.
- 3) The file (a XML file with corresponding definition (DEF) file) is created.