SCIA AutoConverter User Guide
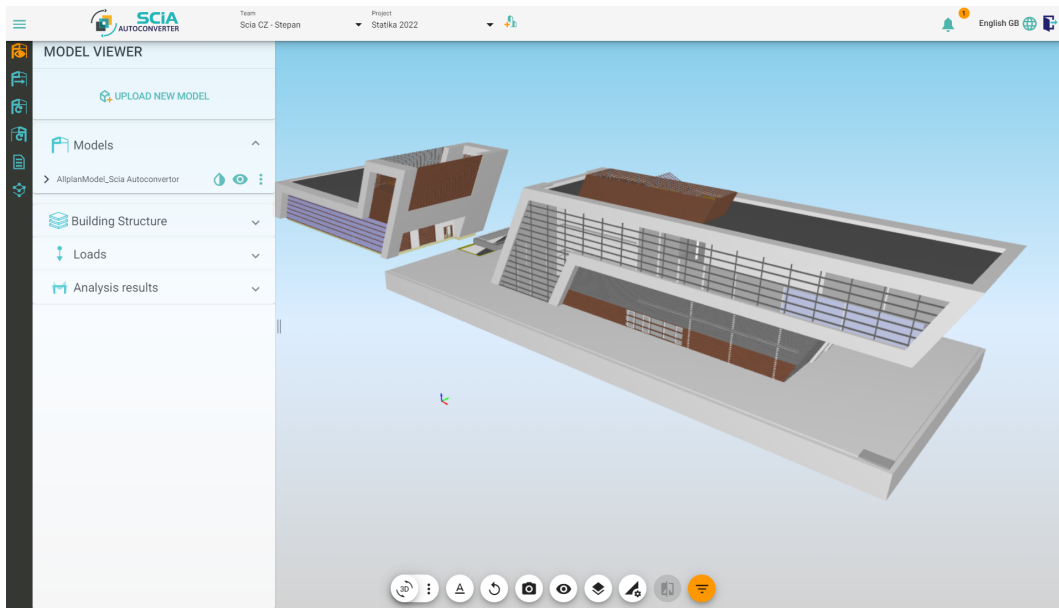
Version 1.0.6

SCIA

February 15, 2023

Contents

Figure 1: SCIA AutoConverter

1 Introduction

Welcome to SCIA AutoConverter user guide! This technical document will help you to understand the SCIA AutoConverter purpose, workflow and settings. SCIA Autoconverter is a web application designed for the conversion of structural models to analysis models, ready to import to various analysis software. The conversion is fully in the hands of structural analysis engineers and SCIA AutoConverter provides varied options for model creating settings. Model creation is an automated process that saves a lot of time for engineers. The application runs with connection to the Bimplus, a CDE platform provided by Allplan. SCIA AutoConverter also provides update workflows in both ways. Structural engineers can communicate selected changes done in the analysis model back to colleagues structural modellers, drawsmens or architects with the system of Tasks, tracked on Bimplus or also update the analysis model with selected changes done in the structural model in one of the revisions. Update of analysis model is done effectively and with the emphasis on reducing the remodelling necessary time to zero. Update is affecting only selected objects with changes, all additional data like loads or supports are kept. The update workflows are recommended in cooperation with the latest SCIA Engineer analytical software available.

2 Basic Overview of AutoConverter Interface

After your first login to SCIA AutoConverter you will see the application interface (Figure 2). Let's take a look at what you can find here. General overview of AutoConverter user interface consists of the following components:

1. **Top bar**

2. **Workflow selection bar**

3. **Workflow specific tab**

4. **3D scene**

5. **View settings**
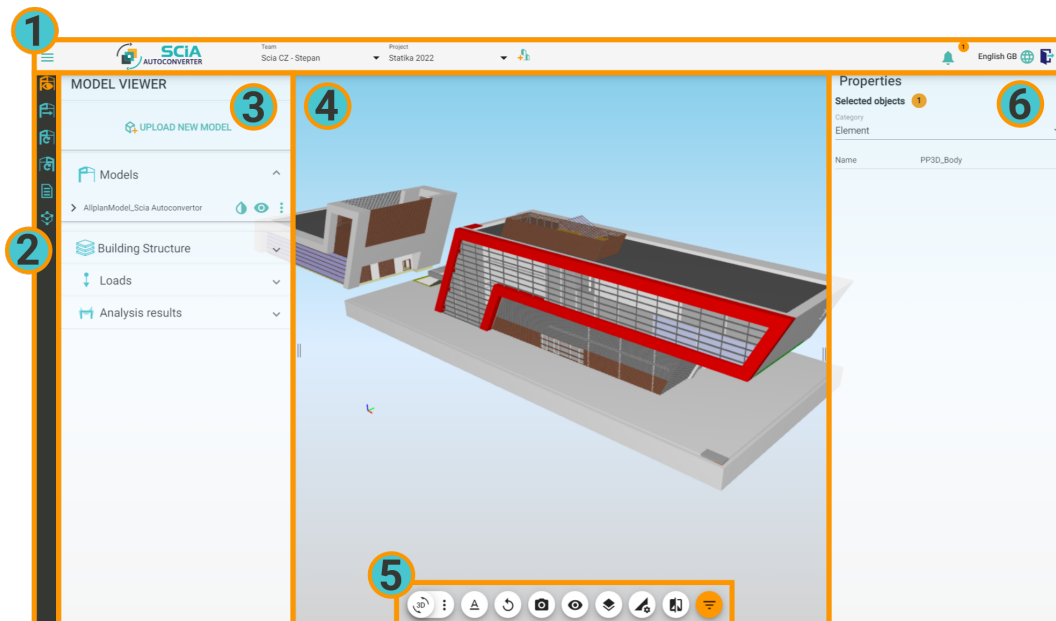
6. **Property tab**

Figure 2: SCIA AutoConverter user interface

1. Top Bar Top bar content is not changing based on the selected workflow and always stays the same. The top bar is used for general navigation in team and project, language settings and notifications. Let's look at the components one by one. (Figure 3)
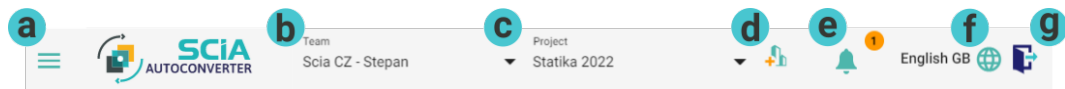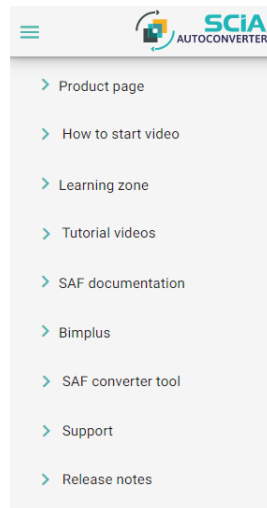
3

Figure 3: Top Bar description

Figure 4: Expanded hamburger drop down menu

(a) Hamburger menu

When hamburger drop down menu is expanded there are multiple options to go. (Figure 4)

Product page - link to SCIA Engineer website

How to start video - embedded video about quick start with SCIA Auto-Converter

Learning zone - link to SCIA Engineer learning zone

Tutorial videos - when expanded, there are numerous embedded tutorial videos with specific features in SCIA AutoConverter (all videos can be found also on SCIA Engineer YouTube channel)

SAF documentation - SCIA AutoConverter is supporting export to SAF format and this link leads to official SAF format documentation

Bimplus - link to Bimplus portal

SAF converter tool - link to SAF converter webpage (tool converting SAF files to Dlubal RFEM 5 excel input and in native Autodesk Robot files)

Support - when expanded, offers the option for reporting an issue, link to FAQ and contact information of SCIA Engineer

Release notes - provides up-to-date overview of released improvements (UserStory) and fixes (Bug)

(b) Team selection

Here you can select the Bimplus team you want to work in.

(c) Project selection

Here you can select the specific project you want to work in.

(d) Project creation

By clicking on the Project creation icon, you can create a brand-new project. The project is then automatically created on Bimplus.

(e) Notification bell

When the notification bell is in an active state (coloured circle with number inside), it means you have unread messages. By clicking on the notification bell, new messages will be shown. Messages can be marked as read with the orange tick button on the right bottom of the message. In case you have more unread messages, the latest message is at the bottom. Notification bell has two types of messages: (Figure 5)

Figure 5: Expanded notification bell
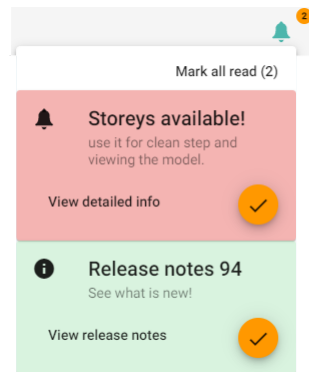
New release of SCIA AutoConverter - with every new release, a notification with release notes will be shown, by clicking on "View release note", complete release notes will be shown

General message - general message, usually informing about newly available functionality, the "View detailed info" contains a link to a tutorial video or to a dedicated webpage to the message.

(f) Language selection

Following languages are supported by SCIA AutoConverter: English GB, Czech, German, France, Portuguese

(g) Log out button

2. Workflow specific tab Workflow selection offers available workflows based on your licence. All available workflows are shown on Figure 6. From top to bottom there are:

Figure 6: Workflow selection bar

- **View Models:** Model viewer which allows both analysis and structural models in the 3D scene. Models can viewed and inspected in this regime.

- **Convert Structural to Analysis:** Shortly "S2A" is a process that guides you from the structural model (IFC on import) at the beginning to the analysis model at the end.

- **Update Analysis with Structural:** Workflow that guides you through the process of updating of the analysis model. The analysis model is updated with selected changes in the structural model between revision.

- **Compare Analysis Model Revisions:** SCIA AutoConverter will compare changes between analysis model revisions. Selected changes can be communicated via issues (tasks) with other stakeholders of the current project.

- **Reporting:** Reporting provides you with option for exporting a table with mapping between analysis objects and structural objects (relation table).

- **Import/Export of the SAF file:** Here you can export an analysis model stored in SCIA AutoConverter to SAF file (.xlsx) or upload your own SAF file to the project.

3. 3D scene For navigation in 3D scene, commonly used functions are available.

- For zooming use the middle mouse wheel.

- For moving the view hold the right mouse button + move

- For rotation of view hold the left mouse button + move

- For selection/deselection use point and click of the left mouse button.

- For multiple selection/deselection hold ctrl and left mouse click on multiple entities in the scene.

- For window selection/deselection drag with holding ctrl + left mouse button. (Both directions represent selection type "all inside the boundary rectangle")

4. View settings View settings at the bottom of the 3D scene are there for the general manipulation of 3D scene and selections and for displaying additional data. Features available you will find useful in all SCIA AutoConverter workflows. (Figure 7)
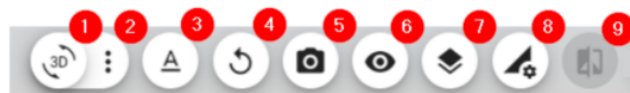
Figure 7: View settings of the 3D scene

1. Toggle 3D volumetric render of analysis model: When you have an analysis model (recognized or aligned) you can toggle ON and OFF volumetric render. Volumetric model is a 3D representation of an active (visible) analysis model in the 3D scene.

2. Additional settings for volumetric model render, scene light settings and colours of analysis entities: By clicking on 3 dots you can expand the additional settings. There is a setting for scene camera light, toggle for volumetric model transparency, colour of volumetric render of the analysis model and colour selection for analysis entities.

3. Label settings: By clicking on the "A" icon you can expand the label settings. In the label settings, you can set the visibility of various labels and their colour used in the 3D scene. Labels available are: Node labels, 1D member labels, 2D member labels, Load case labels, Result labels, LCS labels.

4. Reset scene: Reset scene will reset the view to "Side view", will turn all available models visibility ON and show all hidden elements.

5. Change camera orientation: Here you can select from predefined camera positions.

6. Hide objects (toggle): Hide object toggle turns ON and OFF "hide objects" mode. When the "hide objects" mode is active, all objects you click on in the 3D scene will be hidden and disappear from the scene. This feature is useful in inspecting the structural model, recognized model, aligned model and their relations. Hidden elements can be restored by clicking on the "Reset the scene" button (4) or toggling model's visibility OFF and ON.

7. Isolate objects (toggle): Isolate objects will isolate the active selection in the 3D scene. Isolated elements will still be highlighted in the 3D scene and all other active objects in the scene will become transparent.

8. Result setting: Here you can expand the setting available for result drawing. You can set various settings for drawing of available internal forces in the analysis model. (Result settings are valid only in cases when you have uploaded the analysis model with results.)

9. Invert selections: Invert selection icon is active only when you have a selection in the 3D scene. When you click on the "Invert selection" action button, all unselected elements will be selected and the previously selected elements will be deselected. This function is useful in "Clean model" in the workflow "Convert Structural model to Analysis model".
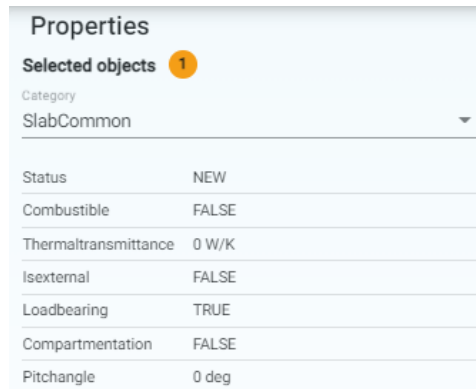
Figure 8: Property tab in the SCIA AutoConverter

5. Property tab Property tab (Figure 8) is shown when the structural or analytical object is selected in the 3D scene. The property tab can be resized by clicking and dragging on the edge approximately in the centre when active. In the property bar, you can inspect attributes and properties of selected objects in 3D scene. Properties are divided into categories based IFC property sets (for example property set WallCommon, SlabCommon etc. for structural objects, property sets for analysis elements are based on SAF format, more you can find on saf.guide).

When object is selected you can inspect properties through categories, you can also use attribute selection by clicking on specific attribute in the property tab. All objects with the selected attribute and value will be selected in 3D scene. This is very useful for cleaning the structure in S2A workflow (step 2). When multiple objects are selected, you can use toggle at the top of the property tab "Show common properties" to see only attributes and values they have in common, the result is shown in one category "Shared properties". When toggle is turned OFF, properties for the last selected object are shown in the property tab.

3 IFC input recommendation

Models, that are being converter to analysis ones in SCIA AutoConverter are models stored on Bimplus - CDE platform by Allplan. The input for the AutoConverter is an IFC file. IFC is a file format defined and developed by Building SMART (see: Building SMART). IFC format provides various options for storing and transferring a lot of data (geometry data and attributes) defined in the authoring model application (Allplan, ArchiCAD, Revit, Tekla, Advance Steel and others). IFC model uploaded and processed on Bimplus is an input for SCIA AutoConverter. The resultant quality of the analysis model is tightly connected to the quality of an IFC model uploaded to SCIA AutoConverter (Bimplus). This chapter is therefore crucial for the best possible result.

3.1 Best Practices for IFC Creation

In this chapter, let's take look on IFC attributes and geometry precision. It is common that all modelling software are offering export to IFC files. In every single software, there are usually provided predefined settings for IFC export and possibilities to adjust the export rules manually (define new exporting template). SCIA AutoConverter can process IFC2x3 and IFC4.

Focus on the correctness of the following attributes during the export of the IFC. Check the steel cross-section profiles (IPE200, HEA240, L100x100x6 ...), material types (concrete, steel, aluminium, timber ...) and load-bearing ability of specific elements when possible. When these attributes are set correctly, it will speed up your workflow at the maximum level in the SCIA AutoConverter.

According to geometry precision, it is good to pay attention to following:

- Overlap of elements - Unwanted overlap of elements could lead to unwanted subregion creation in SCIA AutoConverter.

- Inaccuracy in modelling - slight change of slab thickness along the length could possibly lead to the recognition of multiple analysis elements instead of only one. The recognition algorithm can't manage variable thickness of 2D objects at the moment.

- Export of notches (and other detailed adjustments) - Consider the creation of notches during export. They are often automatically exported to IFC, but it is better to have less notches in 2D elements (walls, slabs etc.) for best outcome from the SCIA AutoConverter. The detail level

of exported IFC can be usually adjusted in the export setting in BIM authoring aplication.

Consider also the intent for receiving the IFC from your partner. For example in case there you know that full BIM workflow won't be used in the project you are working on, there is no need to work with the full IFC model (which is often really huge and even difficult to transfer) - in this case, it is the best way to ask only for load-bearing elements exported in IFC and upload this submodel for your need to SCIA AutoConverter.

4 Structural to Analysis workflow

Spatial workflow in SCIA AutoConverter. The goal is to create the analytical model which can be imported in your analysis software (SCIA Engineer etc.). The process is divided into 6 steps, set in reasonable order. In every step, SCIA AutoConverter user has the possibility to affect the final outcome - the analysis model - in various ways. For going through the S2A workflow we need to have uploaded the IFC model in SCIA AutoConverter first. For starting the workflow, please select "Convert Structural to Analysis".

4.1 Select Model

The general objective of the step: Select a model you want to convert to an analysis model. Model selection is done simply by making the model active in the scene. Activate the model by clicking on the "eye" icon in the left tab - part "Models".

Best practices:
Activate only one model at the time. Currently, working with multiple structural models at once is not supported. Already existing/stored analysis models in the AutoConverter can't be used in this workflow again. Often model structure contains multiple disciplines, activate only relevant disciplines (building, steel etc.) for streamlining the workflow (filtering to the load-bearing structure will be easier in the next step). IFC models often contain information about storeys (IfcBuildingStorey), use Building Structure for fast pre-selection of specific storeys. (Figure 9)
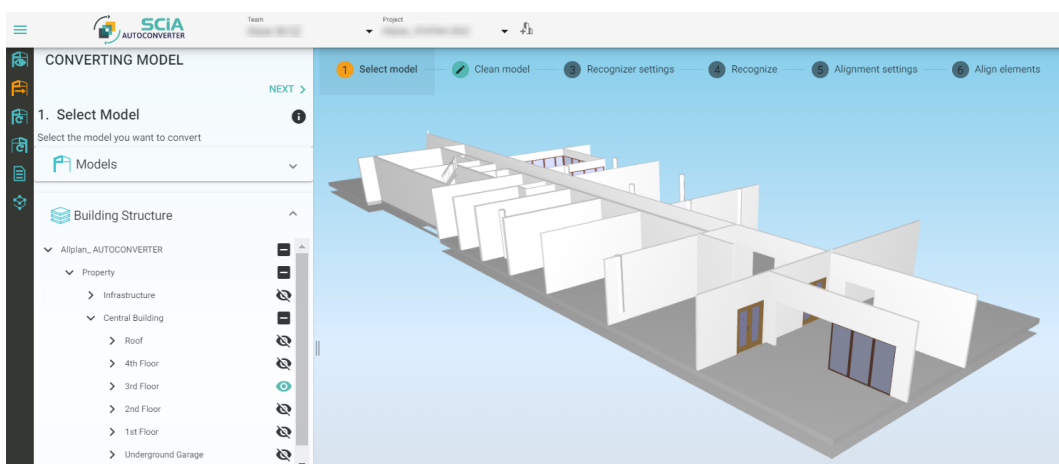

Figure 9: Building Structure

12

4.2   Clean the Model

The general objective of the step: Keep in the scene only the objects you want to convert to analysis elements. The clean step is crucial for all further steps. In most cases, we want to convert only the load-bearing structure or part of the load-bearing structure loaded (visible) in the SCIA AutoConverter 3D scene. To achieve that, you can use filtering options that are available. Filtering divides elements into the active and removed state. Toggle for switching between views (active and removed) you can find right above the View settings in 3D scene (Figure 10).

Figure 10: Filtering: Selected trees are ready to be removed

IFC types:
Together with disciplines from the previous step. IFC types (Figure 11) offers also a raw filtering approach. IFC types are available on the left, workflow-specific bar. Click on the "trash" icon on specific IFC type row for removing it from the scene. You can also click before removing complete IFC type out of the scene only on the IFC type to make a group selection. Group selection will be highlighted in the scene and you can check, if all the objects are really candidates for deletion.
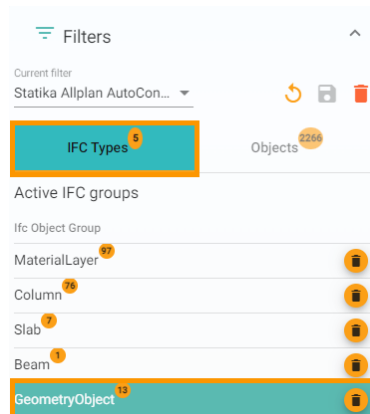
Figure 11: IFC types in left workflow bar

Filtering in 3D scene:
Filtering in 3D scene (Figure 10) can be easily done by selecting an object in the SCIA AutoConverter 3D scene. The selected element can be deleted by clicking on the "Trash" icon right above the scene settings component. You can extend selection using multiple selection by holding "CTRL" key + left click on more objects in 3D scene. Another option for selection is holding "CTRL" key + left click + drag and drawing a window inside the 3D scene. Every object with a centroid inside the selection window will be selected. The most effective way is to use attribute selection for filtering. Select an object in 3D scene and take a look at the Property panel on the right-hand side of the screen. By clicking on the selected property you will achieve a property selection. All objects with selected property and value will be selected in the 3D scene and highlighted.

Active/Removed views:
Everything you remove from the scene with the methods mentioned above can be seen in 3D scene when you switch between an active and removed view. Default one is active, switching is done by toggle above View settings, next to the thrash icon. Removed view will show in scene all removed element. It means you can check if all the elements that suppose to be removed are removed and in case, some element is considered load-bearing, you can easily move it back to active with "Keep selected for recognition" button (in active state "Remove selected from recognition") (Figure 12).

Figure 12: Active view with "remove" action and Removed view with "keep" action

Saving the filter:
The current state of your 3D scene - meaning the state of objects (Active or Removed) can be stored as a saved filter. Filter can be saved using the "save filters" icon as can be seen on Figure 13, left. You will be asked for naming the filter and the filter is saved. Once is filter saved, you can recall the filter from memory every time you will get back to the model. One model can have multiple filters stored. Filters can be recalled from memory, updated and saved again. Other actions available are "Reset filters" which will activate all removed objects and put the structure in a default state with no active filter. The next action is "Delete currently selected filter", this action will permanently delete the active filter. Be careful with deleting filters that might be useful. Filters are also used in other workflows, therefore it is wise to save them.
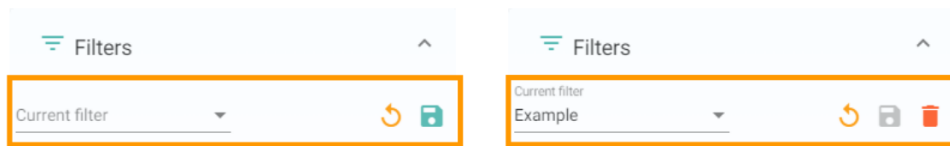

Figure 13: Creating, saving and recalling the filter

Best practices:
Before you start with cleaning take a look at available properties in the right property tab. Look for properties that will help you with the filtering. Examples of useful attributes for filtering: LoadBearing = YES, NO . With the LoadBearing attribute correctly set you can easily select all non-loadbearing elements and remove them. Material = MineralWool, EPS, etc. with the materials properly defined you can easily identify non-load bearing elements based on material and remove them.
Also, keep in mind that you can use inverted selection in case you need to select exactly opposite objects that you are allowed with the attribute selection. Use the BuildingStructure in case it is defined in your IFC file - Going throughout the building storey by storey makes all non-loadbearing elements easy to be removed from the scene.
During the filtering process, you can use also functions available in the "View

settings" of the 3D scene, especially the "Hide" and "Invert Selection" functions.

4.3 Recognizer Settings

General objective of the step:
In this step, you can define how the active objects in the scene should be handled by the recognizer. Recognize is a function responsible for converting the structural object into analysis objects. You can define and change the conversion groups - recognize as 1D, 2D or solid. You can assign cross-sections in case of steel manufactured profiles (or aluminium) are used. You can assign materials from the material library and define layers where possible. This step is also very important for the final model quality - correctly set dimensions, material and profiles will increase model quality and save you a lot of time. (Figure 14)
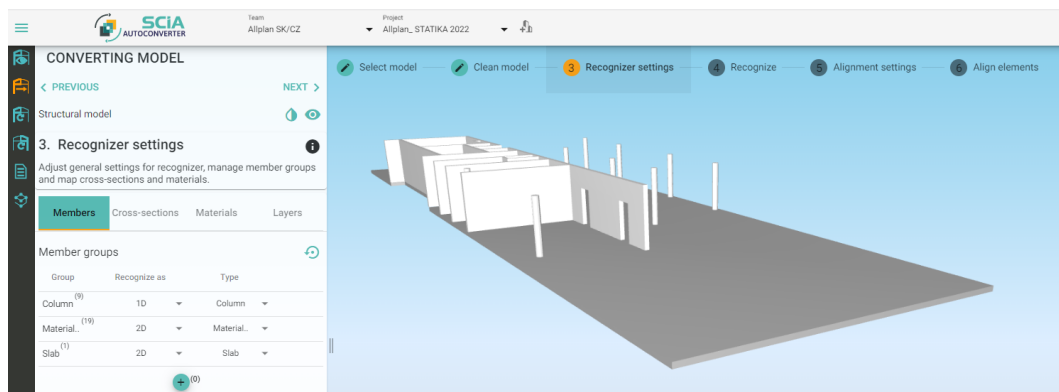

Figure 14: Recognizer Settings

Members:
In the member groups, you can decide how each object group will be recognized and converter to an analysis object. By default, several groups are created based on IFCBuildElement entity - typically Beam, Slab, Column etc. The groups are editable and elements can be moved to the newly defined groups and between already existing groups. This provides you with flexibility in case the IFC data are not according to structural reality. The group component is divided into 3 columns - Groups, Recognize as and Type. Each row represents a separate group. Every object in the 3D scene has to be assigned to only one group (Figure 15).
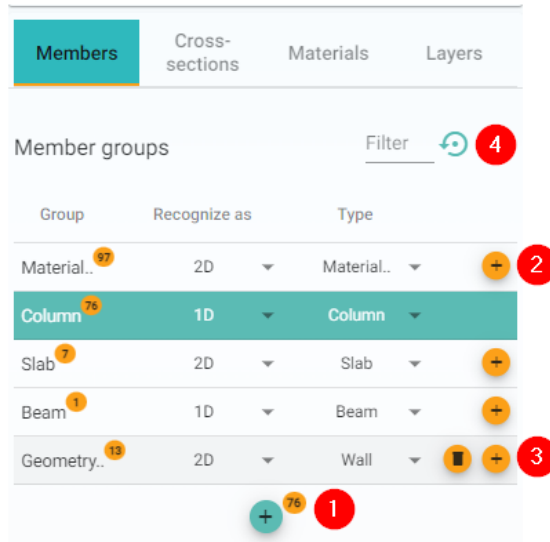
Figure 15: Member groups component with actions buttons

Columns meaning in member groups:

1. Create new group: New group can be created empty or with the selection of elements.

2. Add elements to an existing group: When you have a selection of elements you can reassign them to another existing group.

3. Delete group: Existing group will be deleted and a new "Not assigned" group will be created - all elements have to be in the recognition group.

4. Restore defaults: It will reset the grouping to a default IFC structure-based state.

Member groups component: Member groups component and actions are visible in the Figure 15.

- Group: In Group column, there is the name of each group together with the number of entities assigned to the specific group. Group can be easily renamed by double left click on the name. After renaming, confirm the action by hitting "Enter" key.

- Recognized as: Recognized as determines, to which type (1D member, 2D member, solid) will be structural entity assigned in the group converted. In the drop down, there are four options.

- Unknown - means that the algorithm will determine whether an object is 1D, 2D or solid.

- 1D - The recognizer will convert elements assigned in 1D groups to analysis elements represent by centre lines (typically columns, beams, girders, purlins, rafters etc.).

- 2D - Recognizer will convert elements assigned in 2D groups to analysis elements represent by centre plane (typically walls, slab, shear walls etc.).

- Solids- The Recognizer will not convert elements assigned to this group. The original structural object will be kept and passed to the analysis software as a 3D solid. This can be useful for objects that are not loadbearing but could be of interest for e.g. defining loads. Or if you have complex objects like double-curved elements that can't be recognized to pass them at to your analysis software and use them as a reference for modelling them in your analysis software.

- Type: Offer various options for further specification of elements. The selected type is stored and will be transferred to your structural analysis application. Types can also affect the direction of recognition in cases where is not clear which direction is dominant. For example in case you have an element which you would like to have recognized as 1D, but it has no dominant dimension (in meaning l»a>b). Type then can be decisive in centre line placement. Column - centre line is primarily vertical. Beam - the centre line is primarily horizontal (in case of dimensions are not geometrically clear).

Cross-section:

Cross-sections are dedicated to 1D elements and their profiles (cross-sections). SCIA AutoConverter is capable of recognizing general cross-section described by polygons, parametric cross-sections (rectangle and circle) and provides also the option for mapping structural profiles to analysis profile library (HEA, HEM, HEB, IPE, UPE etc.). SCIA AutoConverter can also recognize several compound cross-sections (double-I, double channel, double angle etc.) and compound general cross-sections as well (see Recognizer chapter).

For setting cross-sections properly before the recognition (conversion of elements from structural to analysis ones). Follow the steps described below. First of all, you need to decide if this setting is relevant to your current project. In case you have no manufactured profile available in your project

(your project is purely from concrete elements or timber, you can skip this step and go directly to "Materials"). All elements are by default recognized as "Parametric" where possible and as "General" in all other cases.

In case you have manufactured profiles (Figure 16):

Figure 16: Cross-sections: Mapping process and overview

1. Find the attribute that is containing the cross section name like e.g. HE200A: Inspect elements is your structural model a find a property that contains data about cross-sections. (Figure 17)
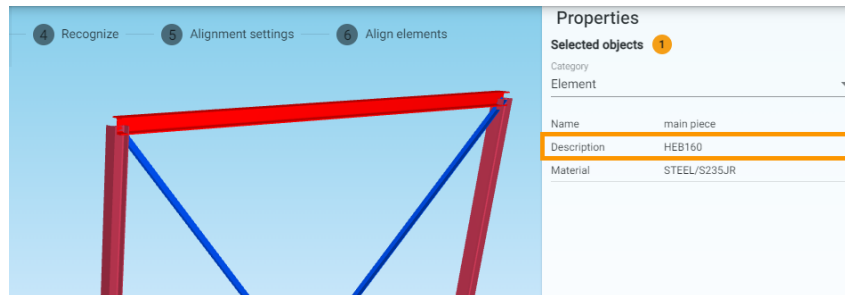
Figure 17: Finding right attribute for CSS mapping

2. Open a property dialogue and select the property that you have identified as the right one for cross-section mapping.

3. Property will appear in the property list. Selected can be one of more properties - In some cases, there might be more suitable attributes for mapping. In case more attributes are selected the order is important. The second (and every next attribute) are showing in the mapping table only rows for elements that have no value for the hierarchically higher attributes. This eliminates the issue with "double" mapping, but still provides you flexibility in profile mapping.

4. When the attribute is selected, the mapping table is shown. The table has two columns and one action. Column "Original CSS" represents the value from the structural model, "Analysis CSS" column represents the value from the steel profile library (default value is "General"), the action button is shown when the row is selected. The action button is used for the mapping to the steel profile library available in SCIA AutoConverter.

5. Other action buttons: There is an option for excel mapping. Import is for loading the mapping table inside SCIA AutoConverter, export is for exporting a mapping table from SCIA AutoConverter to an excel sheet. Excel mapping can save you time during the mapping. You can export the excel and manually do the mapping in excel, save it and import the excel into the SCIA AutoConverter. Or you can export the mapping table and import it in another project. With that you can use the mapping table accross different projects. In case you work with regular partners you can reuse your mapping table across multiple projects in SCIA AutoConverter.

6. Smart mapping toggle: Smart mapping toggle helps you make the mapping process faster. When toggled ON, it tries to find a logic in mapping you provided with the mapping dialogue. In case you map

"IPE_200" to "IPE200" profile from the library, Smart mapping will map automatically map all others "IPE_***" to "IPE***".

Mapping dialogue:

Mapping dialogue can be opened by clicking on the action icon in the selected row in the mapping table (Figure 16). In mapping dialogue, first select a type. Type "General" - the algorithm will recognize a shape and will create a general or parametric cross-section where possible. Type "Manufacture - rolled", "Manufactured - welded" and "Manufactured - cold formed" will guide you through the library tree to the final cross-section that is defined in the SCIA AutoConverter steel profile library. For example: "Manufactured - rolled" as Type, "I-section" as a Category, "HEB" as a Cross-section group and "HEB140" as the final library profile.

The special Type in the mapping dialogue is "Manufactured - user defined". This type of CSS provides you with the ability to be not limited only by the library available in the SCIA AutoConverter. Thanks to "Manufactured - user defined cross-section" you can define your own manufactured type of cross-section. This newly created cross-section is meant to be familiar to your analysis software. The result is your specified profile, with your defined name, formcode and the "Type" set to Manufactured (Steel checks might be applicable on the members with this cross-section afterwards).

How to define a "Manufactured user - defined cross-section? (Figure 18):

1. Specify a name. This should be a name known in your analysis software cross section library.

2. Set the dominant dimension. Usually, it is a height. This is necessary for the correct positioning of the profile and local coordinate system in relation with the bounding box of cross-section.

3. Set the type of profile you are now defining.

Figure 18: Manufactured user - defined cross-section

4. Select formcode when necessary. More about formcode can be found in the saf.guide - formcodes. Formcodes are important in the way, how the certain profile is defined in cross-section library in relation to the member's local coordinate system. For profiles with two axes of symmetry, formcodes are irrelevant. For profiles with one axis of symmetry there are two formcode options (typically asymmetric I sections - they can be placed with wider flange in local Z positive or local Z negative). Profiles with no axis of symmetry have typically four formcodes (L-sections for example).

Materials:

Mapping of the materials is in principle same as the mapping of the cross-section. The goal is to assign correct materials to members based on attribute mapping. When no action is taken in this step, all members are assigned with default material "MAT1" - this material is not suitable for analysis and manual change is necessary in your analysis software. It is recommended to map material where possible and thus save yourself time spend with manual reassignment.

How to map materials?

1. Select a property useful for mapping.

2. Property is shown in "Selected properties list". Same as for cross-section that might be more properties selected with the same rules.

3. Map the structural attribute to the material library using the action button that shows up after the selection.

For more detailed description of action buttons see chapter cross-section, functionality is the same.

Layers:

Layer are in principle same as material mapping and cross-section mapping. The core is in selecting useful property and mapping it to the desired value. Only difference with Layers is that there is no library and you can set the value as you wish. in case layers are not defined, a basic layer set is created based on the type of object available in SAF (`saf.guide`). For example StructuralCurveMember layer for all 1D elements, StructuralSuraceMember for all 2D elements etc.
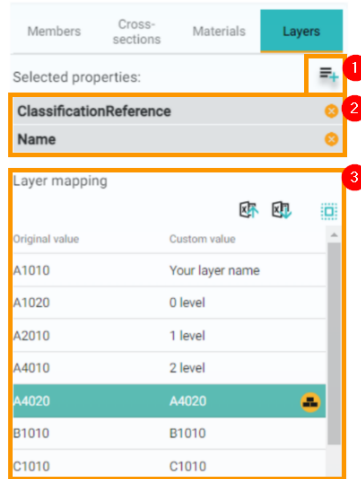
How to define layers? (Figure 19)

Figure 19: Layers definition

1. Select a property useful for mapping.

2. Property is shown in "Selected properties list". Same as for cross-section and materials there might be more properties selected with same rules.

3. Layers names are set as default to the same value as selected property has. You can change it to your own specified name by clicking on the action button a typing the new name in.

When you are satisfied with the step "Recognizer settings" you can proceed to the next step, the recognition.

4.4 Recognizer

The recognizer is the step where the conversion takes place from structural objects to analysis objects. Tasks related to object grouping and mapping are done thanks to the previous step and last thing we need to do is defining the general recognize settings (conversion to analysis objects) itselfs and steering the algorithm settings. General recognizer settings you can find under the gear wheel icon. General recognizer (Figure 20) settings consists of:

- "Recognition of 1D members" - algorithm rules for 1D conversion

- "Recognition of 2D members" - algorithm rules for 2D conversion

- "Use prefix" - rule for the naming of the nodes

- "Support for large models" - algorithm approach towards the recognition process based on model size and complexity

General recognize settings: (Figure 20)

Figure 20: General recognizer settings

1. Enable parallel member detection (toggle) - When turned ON, parallel members are detected and the algorithm creates compound cross sections out of them or general compound cross sections. Higher priority has the compound cross-section type (for example double I section, angle back to back etc.) than general compound cross section type. See Annexes for the list of supported compound sections, rules and further information about the general compound type.

2. Minimal 1D size - Set a rule for minimal 1D length size. All 1D members with lengths smaller than the set value will be placed in a special group after recognition called "Small 1D Members". You can then decide if you want to keep them in your analysis model or remove them. This setting is mainly useful for making sure that the cleaning of the structure was done properly and that there are no forgotten bolts, screws etc. in your analysis model.

3. Minimal 2D member size - All 2D elements with smaller are than set area will be placed in a special group called "Small 2D Members". After recognition, you can decide which elements you want to keep a which to delete. This might be useful in case you want to be sure all connection plates in steel connections are gone or you have complex 2D elements geometry. In some cases where 2D elements have inclined edges, they have areas with different thicknesses (recesses) or in another way not straightforward geometry they might be recognizer as more than one analysis element. With Minimal 2D member size, you can easily identify them.

4. Minimal 2D opening size: All openings with smaller area than defined will be placed in the separate group "Small 2D Openings" after recognizer. Thanks to that, you can easily get rid of all the small openings in your 2D analysis elements that are irrelevant to the current structural analysis.

5. Minimal 2D thickness - All 2D elements with smaller thickness than defined will be placed in the separate group called "Insufficient thickness". After recognizer, you can decide which elements to keep in the model and which to remove. This setting might be useful in making sure, that all connection plates, and e.g plasterboards are filtered out of the model.

6. Use prefix (toggle) - When turned ON, the naming of the nodes will have a prefix "N" (for example N101, N102, N103 etc.). When turned OFF nodes will be named only with numbers without the "N" prefix. Check out what is the right format for your structural analysis application. (For SCIA Engineer is recommended to have toggle turned ON.)

7. Enable "long duration" support mode (toggle) - This applies special order of processing of structural elements that is effective for huge models only. This processing is not performing well for small or medium size models. In case this needs to be activated, you will be asked for that after first recognizer run timeout. The timeout of the regular recognizer is 2 minutes.

8. Restore defaults - recognizer settings is stored in SCIA AutoConverter and it is loaded every time you get back to the model. In case you want to get back the default settings, just click on the "Restore defaults" button.¨

9. Done - All settings are confirmed by clicking on the "Done" button.

Grouping of elements that do not satisfy conditions set above: (Figure 21) In case some of the recognized objects do not satisfy the recognizer settings, then it is placed in separate groups. The above-mentioned groups for specific conditions are "Add to analysis model" (1) and "To be deleted" (2) group. Below the two main groups, there is one group for each condition in settings - Small 1D Members, Small 2D Members, Small 2D Openings and Insufficient Thickness.
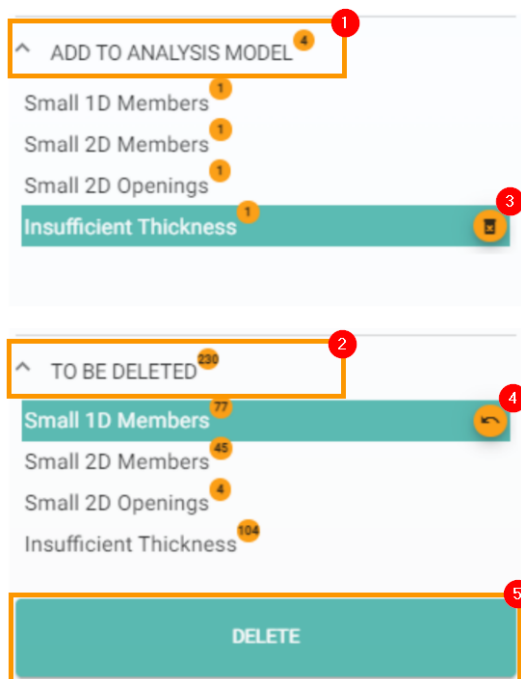

Figure 21: Post-recognition grouping

Elements can be moved between the main groups using the action buttons (3) and (4) that are available when the whole group is selected or one or more elements from a specific group is selected. That means you can move all elements at once between groups or you can use selection in the 3D scene for moving only selected objects (openings for "Small 2D Openings", 1D members for "Small 1D Members" etc.). When you are done with the regrouping between

"Add to analysis model" and "To be deleted" group press the "Delete" button (5). The delete button will remove all elements that are placed under "To be deleted" and all objects under "Add to analysis model" will be kept. In normal case, you delete all the "To be deleted" objects as they have passed your threshold.

Note: Recognizer can be run also with a selection of elements. Selection can be done the usual way - object selection, window selection and attribute-based selection. If you decide to recognizer only a selection you are defining the selection also for further steps (Alignment setting, Align elements).

4.5   Alignment Settings

Now you have successfully converted your active objects in the scene. In the model view settings, you can see now two models: The structural model (1) and the Recognized model (2). Besides the already known toggle buttons for visibility (3) and transparency (4), you can see also the new "Delete recognized model" (5) and "Save recognized model" (6) on the Figure 22.
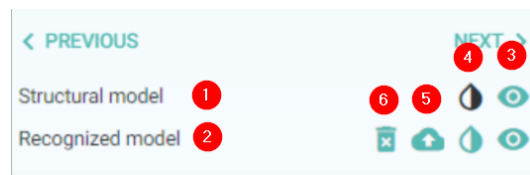

Figure 22: Recognized and Structural model

The general purpose of alignment settings is to prepare everything for the alignment process. The alignment process will connect elements together and before it happens there is a possibility to adjust the centre lines and centre planes positions for analytical elements and set if and which analysis eccentricities you would like to take into account. Also, in the tab "Element behavior" you can enforce the master planes and prevent elements from movement in the alignment process.

System lines:
System lines component works very similar as "Members" in the "Recognizer setting" in therm of creating groups and moving elements between groups. In addition, there are rules that 1D analysis elements can be reassigned only to 1D group, 2D element to 2D group. Solids are excluded from grouping and there are kept a strictly geometrical representation. Every row in System lines

groups represents one group, groups are based on "Recognizer settings" groups defined by "Recognized as" and "Type".
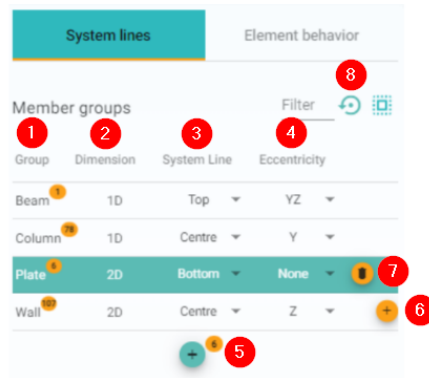
Columns in System lines component: (Figure 23)

Figure 23: System lines component

1. Group: Name of a group. Group can be renamed the same as a Member group in "Recognizer settings".

2. Dimension: Unchangeable field, indicates analysis representation of the group. Typically 1D or 2D.

3. System line: Drop down selection field. Provides you with the option with changing the system line position. List of system line positions is presented in Annex.

4. Eccentricity: Drop down selection field. In case you want to take into account eccentricities in your structural analysis, select the direction from the drop down. Available values are shown in the Annex.

5. Create a group: Action button which creates a new group with selected entities. In case, you have in selection both 1D and 2D elements, two groups will be created.

6. Add to group: When selection is done SCIA AutoConverter offers you groups where the object can be reassigned.

7. Delete group (hover state): On hover, you can delete the group. All objects will be moved to either new or existing group "*D unassigned".

8. Restore defaults: In case you want to get back to the original state, click on the restore defaults.

Element behavior:

Element behavior component is empty by default. The component is used to freeze elements (both 2D and 1D) along the specific axes. 2D elements can be frozen along the local Z axis and 1D elements can be frozen along the Z, Y or both. Freezing the elements will prevent them from moving in the alignment algorithm and all other elements will tend to be connected to them. They will define so-called master planes.
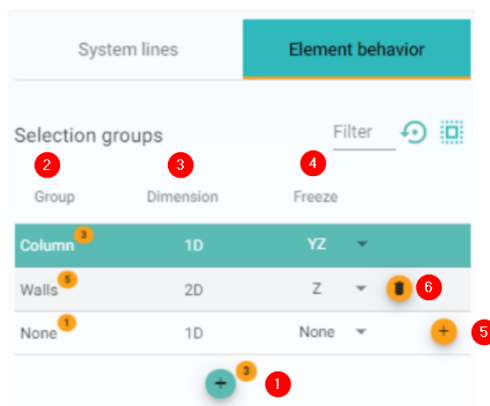
Element behavior component: (Figure 24)

Figure 24: Element behavior component

1. Create a group: Select elements and click on the action button to create a group. Groups will be created based on the type of analysis entity (1D or 2D).

2. Group column: Name of the group is shown. Groups can have custom names.

3. Dimension column: Unchangeable value, show the analysis type of entities in the group.

4. Freeze: Drop down allows you to choose in which direction should be element frozen - movement in alignment will be prevented. See Annex for available values for freeze.

5. Add to another group: By clicking the action button, you can move entities along the existing group from one to another. Also, objects selected in the scene can be moved this way under existing groups.

6. Delete group: Group will be deleted and entities won't be frozen anymore.

30

Best practices:
When you want to change the system line of one beam only, the right approach is to select analysis objects in the 3D scene, create a new group and set the correct system line position. The same approach is working for eccentricities.

In element behavior, we are usually working with only few elements that we are choosing for freezing. Freezing means that these elements will not move in the defined direction. It will create a so-called masterplane and all elements in relation with this frozen element will move to that masterplane. Typical situation for using this freeze option:

- A slab is moved to the plane of a subregion as this subregion has a much bigger thickness. Then you freeze that slab and you will see that the subregions are moved to the plane of this slab.

- There are different levels for slabs on one floor and you want to keep this difference in height. Then you freeze these slabs.

- There are beams moved to the level of other beams close to them. Then you freeze one member in each plane to keep them both.

You can experience a decrease in the quality of the aligned model when there are too many elements in the freeze state. In that case, it is useful to reset freeze to defaults and start over again and freeze less elements.

4.6   Align Elements

The main purpose of this step is to connect elements to each other and obtain a valid structural analysis model. Connection is done based on alignment settings and can be done in 3 modes - Align all at once, Partial alignment based on selection and most advance Alignment per groups. As input is taken the recognized model you have created in the workflow so far. General settings (Figure 25) steer the algorithm and understanding the settings is crucial for the alignment result.

General Alignment Settings:
General alignment settings are a set of values, toggles and options that will adjust the alignment algorithm rules and have the direct impact on alignment outcome - which members will be connected and which members won't and if the post-processing objects will be created or not (for example intersection objects, subregions, ribs, etc.). The complete alignment settings you can see in Figure 25. Alignments settings consist of several parts, two main ones are General alignment settings and Align connection settings.
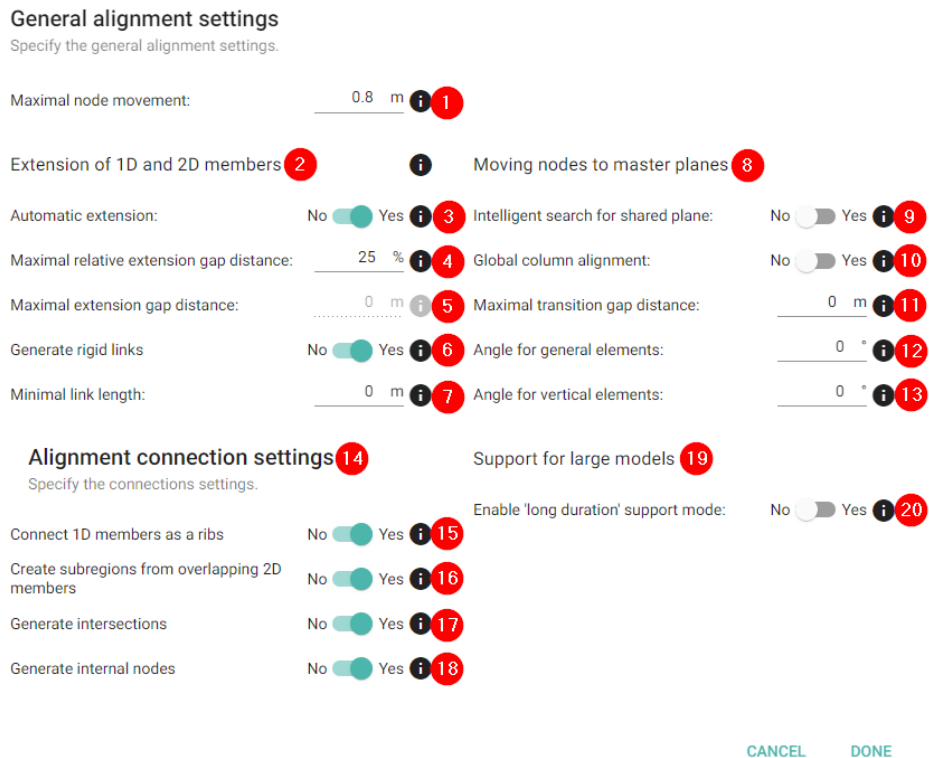
31

Figure 25: Alignment general settings - expanded

Description of setting values and options: (Figure 25)

1. Maximal node movement: Sets the maximum value for the movement of nodes during the alignment procedure. This value is superior to all other settings and values.

2. Extension of 1D and 2D elements: This part of settings determines behaviour of 1D and 2D objects and their extension during the alignment. Values focus mostly on the structural gap between boundary boxes of 1D elements and faces of 2D elements. Structural objects with overlapping or touching boundary boxes (1D elements) and touching of overlapping masses (2D elements) will be connected automatically - there is no structural gap between them.

3. Automatic extension (toggle): Defines how the maximal bridged gap should be calculated - either as relative value to the length of each 1D element and relative to the thickness of each 2D element (4) or as hard set values in meters (5)

4. Maximal relative extension gap distance: Sets maximal relative gap between recognized member boundary surfaces to be bridged by extension relatively in percentage. This value applies to both ends of the member and the original length of each member is used as its own reference.

5. Maximal extension gap distance: Sets maximal absolute gap between recognized member surfaces to be bridged by extension. This value applies to both ends of the member.

6. Generate rigid links (toggle): Turn ON or OFF rigid link creation in the alignment process.

7. Minimal link length: Field is active only when Rigid link creation is toggled ON. When the length of the extension in the alignment of 1D elements is equal or bigger than set value, the rigid link is created instead of the element extension.

8. Moving nodes to master planes: Settings in this category affect other movements than an extension of the centre line for 1D and edge extension for 2D.

9. Intelligent search for the shared plane (Toggle): This toggle defines the position of the plane of analysis elements, typically rafters and purlins. When toggled ON master plane is shifted towards the contact plane between rafters and purlins. When turned OFF, master plane is defined by rafters usually (members with the dominant dimension of cross-section boundary box)

10. Global column alignment (toggle): When turned ON, physically separate structures will share the vertical master planes.

11. Maximal transition gap distance: Set value that should be bridged during alignment in the movement of 1D elements into common master planes. Value is in meters.

12. Angle for general elements: Set value in degrees (up to 10°). Value helps you neglect small discrepancies in structural models and helps you align elements in straight segments.

13. Angle for vertical elements: Set value in degrees (up to 10°). Value helps you neglect small discrepancies in structural models and helps you align vertical elements in straight segments.

14. Alignment connection settings: This part of settings is used for post-processing of the aligned model. These settings do not affect the overall model geometry.

15. Connect 1D members as ribs (toggle): When toggled ON ribs (Structural-CurveMemberRib in SAF) are automatically created. Ribs are separate objects linked to the parent slab. Some structural analysis software allows the creation of ribs afterwards directly when using them, in that case, you can keep the toggle turned OFF and create ribs there.

16. Create subregions from overlapping 2D members (toggle): Usually used for mushroom heads above columns or for local stiffening of foundation slabs. When toggled ON, subregions (StructuralSurfaceMemberRegion in SAF) are automatically created from overlapping elements. Subregions are separate analysis objects linked to the parent slab.

17. Generate intersections (toggle): When turned ON, internal edges (StructuralCurveEdge in SAF) are created in logical places on 2D analysis elements. Some structural analysis software allows the creation of internal edges afterwards directly when using them, in that case, you can keep the toggle turned OFF.

18. Generate internal nodes (toggle): When turned ON, nodes logically moved to centre lines and planes of different analysis elements during the alignment process are listed there as internal nodes. This toggle is not related to new object creation, it only stores the reference. Some structural analysis software allows the creation of internal nodes to be referenced afterwards directly when using them, in that case, you can keep the toggle turned OFF.

19. Support for large models: This setting should be used only in cases when large and geometrically complex models can't be aligned and the alignment ends with timeout.

20. Enable "long duration" support model (toggle): Adjusts the alignment data processing. Usually, you won't need to use this toggle. In case you have huge models, you will be asked by SCIA AutoConverter to turn it ON after the timeout of the first alignment attempt (timeout is in approximately 2 minutes).

Partial alignment

Now we have finished the settings we can press the alignment button and the whole model will be aligned. There are situations that you want to align step by step. This is possible by selecting recognized objects in the 3D

scene. You will see that the button Align is changed in a Button with Partial Alignment. Once done you can change the settings and select another group of recognized objects. Etc. If you make a mistake you have to start all over again and also if you changed the settings inbetween you have to redo them. There is a more advanced way of doing this by using the Alignment per groups, see next chapter.
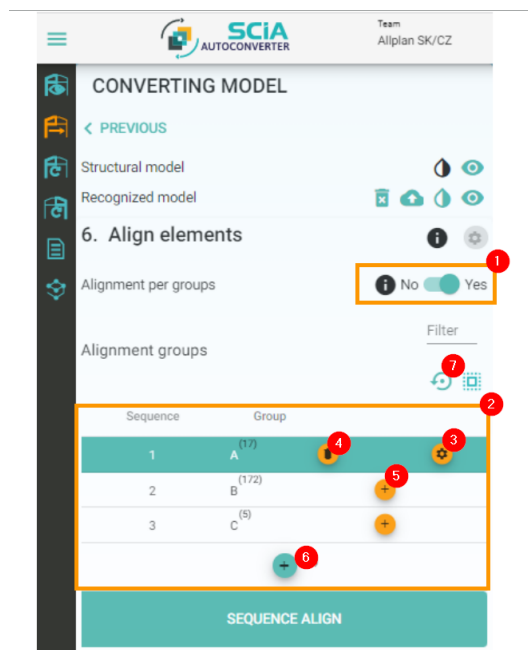

Figure 26: Alignment per groups setup

Alignment per groups (Figure 26)
Alignment per groups is an alignment procedure where recognized elements are grouped in groups and aligned in sequence one group after another. You can activate the feature by toggling ON Alignment per groups toggle (1). The grouping component is activated (2), and each group has its own alignment settings (3), which allows you to steer the alignment procedure in a very detailed way and you can take into consideration the specific needs of individual parts of your structure. Work with the groups is consistent with other components as well - you can create and delete groups (4), move elements between groups (5), create new groups (6) and reset all to default with the reset button (7). The order of the groups plays a significant role in this step, order can be changed simply by "drag and drop" of the group to desired position. When you are satisfied with the distribution of recognized analysis elements in the groups, the order of groups and the Alignment settings for every group you can hit the alignment button.

35

Alignment is done in the hierarchical order of groups - 1 to n group, one after another. You can also align only selected groups, click on group to select it (or use "ctrl + left mouse click" to create a multiple group selection). For the best result order groups in a way that the 1st group has the highest maximal node movement and this setting is descending group by group. It means that the highest movement is set to the 1st group and the lowest to the last group used. Alignment per groups can significantly improve the quality of the analysis model. Typically examples are: Aligning Beams and Columns in one group and Braces in the next group, Aligning the subsection of the model based on needs for different settings for maximal node movement (case shown on Figure 26) or Aligning fist walls + slabs and 1D elements in one or more separate groups.

Best practices:
Pay attention always to the maximal node movement value. Correctly set value reduces significantly the number of unwanted connections.
Check the tooltips in the alignment settings, with visual examples in there you will be able to understand each value properly. (Check the tooltips also in other parts of SCIA AutoConverter)
When dealing with precast structures focus on the Extension of 1D (and 2D members) and on Generate rigid link settings.
When dealing with steel structures, focus on the Extension of 1D (and 2D members) and add on the Maximal transition gap distance.
If you are not satisfied with the outcome of alignment, try to set Angle for general elements" to circa 3° and check the result again.
When you have a model with subregions, you might need to freeze the main slab in cases when slab aligned as subregions has a bigger thickness.
When you have an unwanted overlap of objects in the structural model and subregions are turned ON in settings, you can use them for the detection of such places and delete them in your structural analysis software in case the overlap is unwanted.
When you want to create subregions in Alignment per groups, please put elements meant for subregions to the last group of alignment, otherwise, they won't proceed to the final aligned model.

Notes:
In case you have beams, and you intend to connect them as ribs. For beams with width as the dominant dimension of cross-section, check the position of ribs in your analysis software. Ribs do not have their own local coordinate systems, they are inheriting LCS from the slabs to which they belong and therefore they might be rotated!

4.7 Saving and Downloading the Model

When the model is successfully aligned, it is time to save the model and import it into your analysis software. For saving the model click on the cloud (Figure 27) icon in the model section on the left. After clicking on the cloud icon, the saving dialogue will appear. In saving dialogue you can choose between two options for how to save the model, use either save to Bimplus option or save to .xlsx file in SAF format.
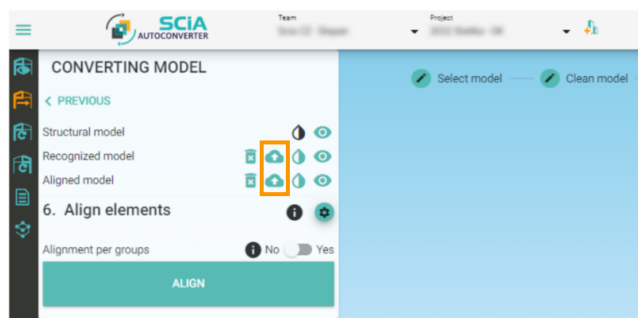

Figure 27: Save icon in the model section

Saving to Bimplus:

The saving to Bimplus option will save the analysis model on Bimplus servers. After saving, model will instantly be accessible in the SCIA AutoConverter and in the Bimplus. The model can be downloaded to your analysis software via import dialogue dedicated to Bimplus. This approach is recommended to SCIA Engineer users and it brings benefits related to revision handling and update workflows described in chapters 5 and 6.

How to save the model to Bimplus? (Figure 28)

1. National code: Select the national code you intend to use in your analysis software. National code can be usually changed in your analysis software afterwards (SCIA Engineer).

2. Export destination: In the drop down select "Bimplus".

3. Team: Select the team in which you want to save the model. Usually, the same team where you have the structural model you have just converted into the analysis one.

4. Project: Select the project in which you want to save the model. Usually, the same project where you have the structural model you have just converted into the analysis one.

5. Create model: Check the box if you want to save the model as a new one. Usually used after the first conversion from structural to the analysis model.

6. Analysis model name: Give analysis model the name. Name can be changed on Bimplus afterwards.

7. Click on the Save button.

Figure 28: Save on Bimplus

How to save a revision on Bimplus? (Figure 29)

1. Create a revision: Check the box for saving a model as a revision of an already existing analysis model. Usually used in workflow in chapter 5 (Update of an analysis model).

2. Analysis model name: Select the right analysis model from the drop down.

3. Revision name: Give a name to a revision you are about to create. ("Name" does not need to include the number, revisions are numbered automatically in the SCIA AutoConverter)

4. Revision description: You can list known changes or any other general text in this field.

Figure 29: Save model as a revision on Bimplus

5. Click on the Save button.

Saving to the SAF (.xlsx) file: (Figure 30)
Saving to the SAF file provides you with the option of saving to the SAF file (.xlsx) on your local drive. SAF file is an open format dedicated to the transfer of structural analysis models. More about SAF you can find here. SAF can be imported into your analysis software, list of analysis software supporting SAF is presented here. SAF file can be viewed and edited by Microsoft Excel or any other software capable of opening the .xlsx files.

How to save a model as a SAF file? (Figure 30)

1. National code: Select the national code you intend to use in your analysis software. National code can be usually changed in your analysis software afterwards (SCIA Engineer).

2. Export destination: In the drop down select "SAF".

3. LCS of cross-section: In drop down, select the LCS of cross-section used in your analysis software. The LCS of cross-section is related to the way how are cross-section stored in local libraries of analysis software. For more info open the tooltip in SCIA AutoConverter (SCIA Engineer uses "1.ZYX").

4. Version: Select the SAF version you want to export to. Check first which SAF version is supported in your analysis software.

5. File name: Give the file the name.

6. Click on the Save button.

Figure 30: Save model to SAF file

5   Update of Analysis Model based on changes in Structural Model

Update of Analysis Model based on changes in Structural Model provides you with an option on how to handle changes that are occurring along the design process of the structure. This workflow is designed for situations when you receive a changed model at the time when you are already in structural design process inside the SCIA Engineer. We guarantee this workflow in combination with SCIA Engineer, for other structural analysis software we can't guarantee as it heavily depends on their implementation of the SAF format which we can't control.

This workflow gives you the option for comparing two selected revisions of structural model, reviewing changes and selecting specific changes you want to use for an update of your analysis model. The update process is done with your specifically selected (accepted) changes and all other relevant data

40

are untouched. For example, when there is a new opening on the floor slab, you can update your model with this new opening and all the loads applied on the slabs will be updated and kept according to the change. Main purpose is to keep the process of making changes transparent and reduce the amount of manual work needed for handling the changes in the analysis model.

5.1 Uploading of the revision of the Structural model

The revisions of the structural model (*.ifc files) can be uploaded in the "Models" by clicking on three dots at the end of the line with the model you want to upload a new revision. After clicking three dots, select the option "Update model" and select an IFC file on your local hard drive. The model will be uploaded and processed in the SCIA AutoConverter. When the model is uploaded and processed you will see a "Clock" icon in the model row. Use the clock icon to activate specific revision in the 3D scene. Revisions of only structural models can be created this way. Make sure you are always uploading a revision to the correct parent model. (Figure 31)
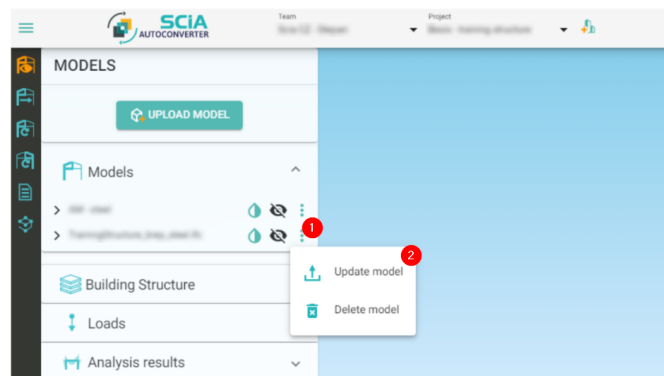
Figure 31: Update structural model revision

5.2 Compare Structural Model Revisions

The workflow icon can be found on the left workflow bar, third from the top (1 - Figure 32), right below the Convert Structural to Analysis workflow. Compare Structural Model Revisions is the first step of the workflow dedicated to the model selection and determining what should be compared between which revisions. The purpose of this step is to set correct comparison conditions and proceed to comparison.
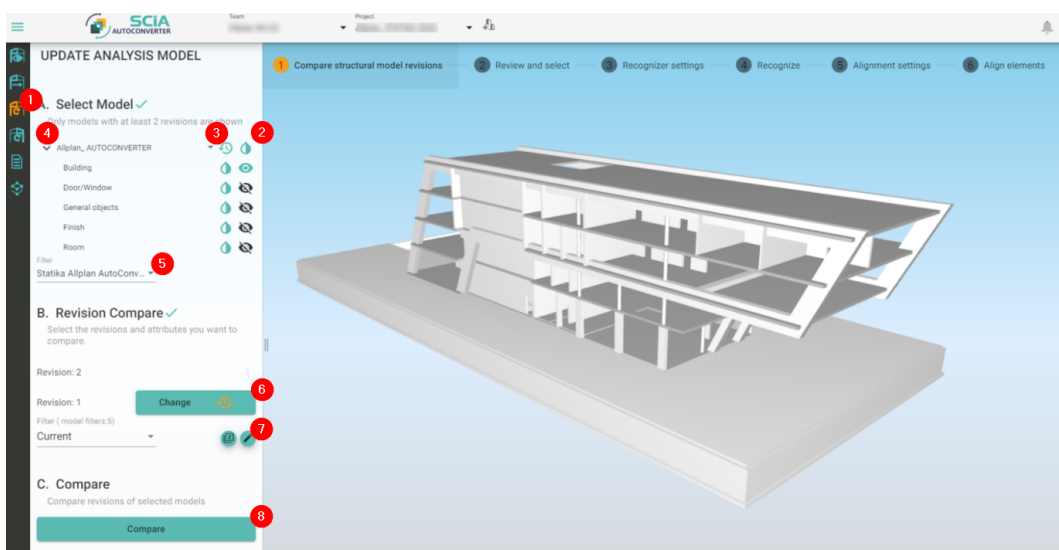
Figure 32: Compare Structural Model Revisions

How to set the model and comparison details? (Figure 32)

1. Select the "Update existing Analysis model from Structural model" workflow

2. Select the structural model with at least two revisions. Models without at least two revisions are not offered in the drop down.

3. Select the active revision in the scene. Usually, it s the last revision uploaded.

4. By clicking on the action arrow, you can expand the disciplines of the model available. Set the activity of disciplines as you wish for the comparison. Usually, same discipline activity is set as in "Structural to Analysis workflow"

5. Select the filter (drop down): Select the filter you want to use for revison comparison. Usually, it is the same filter as was used in converting the model from structural to analysis one.

6. Select the second revision you want to compare. (First revision for comparison was selected in step (3))

7. Property filter: Property filters, in general, provide you with the option to determine which attribute will be compared. Filters are explained below in detail.

8. When everything is set, you can click on the Compare button and proceed to the next step.

How to set and store property filters? (Figure 33)
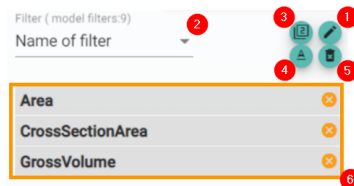
Figure 33: Setting the property filter

1. Pencil icon: Click on the pencil icon for opening the attribute selection window (Figure 34). Select specific attribute(s) you would like to compare exclusively. You can add one property or more (1), when you are done, click on Done button (2).
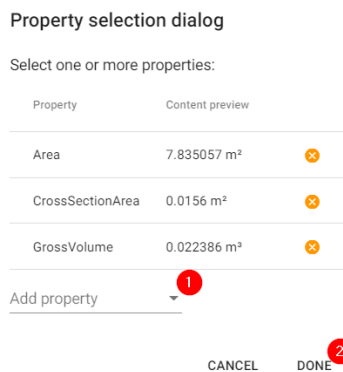
Figure 34: Selecting specific attributes for comparison

43

2. Filter (drop down): Select the already existing and saved filter in the drop down selection.

3. Clone icon: Click on the clone icon for creating a copy of the existing property filter. Existing filters can be copied, adjusted and saved as new with the clone command.

4. Rename the filer: Clicking on the "A" icon you can activate the renaming regime. When activated, rename the filter in row (2) a click on the "Diskette" icon for confirming the action after renaming.

5. Thrash bin icon: Clicking on the thrash bin icon will delete the currently selected property filter.

6. List of selected attributes: In the list, you can see selected attributes from step (1). By clicking on the cross icon you can remove specific attributes from the filter. This action is automatically saved.

Best practice:
In case the workflow icon is in the disabled state, check your subscription level.
Nothing to select in part "A. Select the model"? Check if you have a structural model with at least two revisions uploaded.
When you are about to update the already existing structural model, make sure you are uploading the model as a revision to the correct parent model, team and project.
Use the same activity of disciplines for revision comparison as you used for conversion from structural to analysis.

5.3   Review and Select

After hitting the Compare button, SCIA AutoConverter will process the revision compare according to your settings. The comparison result is represented in the left workflow toolbar in a list, visually highlighted in the SCIA AutoConverter 3D scene and also available in the table overview. In orange colour are presented the Modified properties, light green is for Deleted objects, dark green is for New objects and blue is for Modified objects. The main purpose of this step is to review the result of the comparison, select relevant changes for you as a structural engineer and proceed with selected changes to the next steps of the update. (Figure 35)

- Orange - Modified properties. Example: Changed the thickness of the slab and its volume.

- Light green - Deleted objects. Example: Removed column.

- Dark green - New objects. Example: Newly added column.

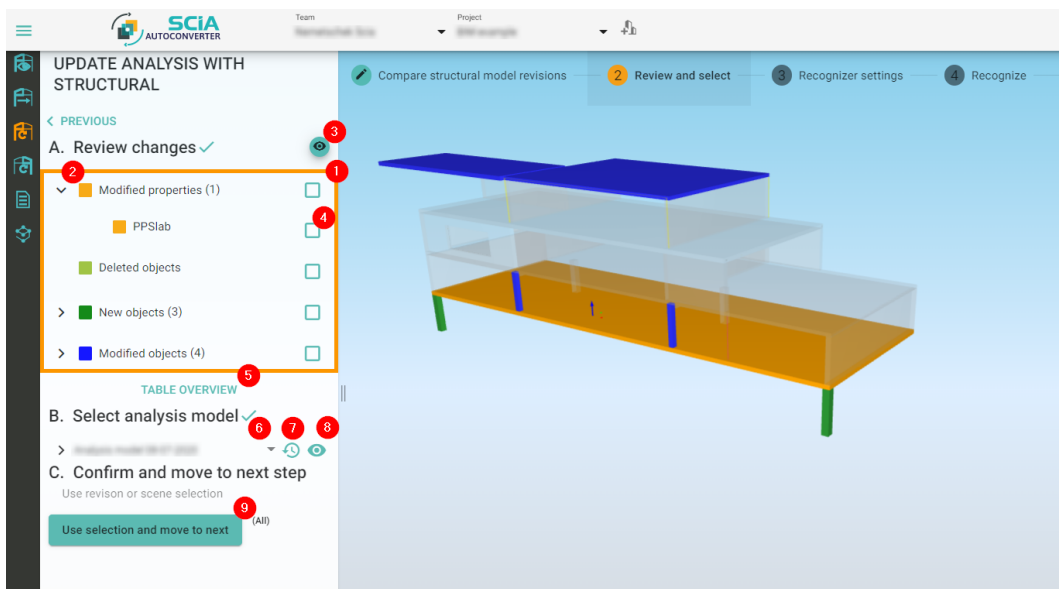- Blue - Modified objects. Example: Newly added opening in the slab.

Figure 35: Review and select changes

Review and select overview:

1. Changes review part: Here you can see all the changes after the revision comparison. Changes are logically filtered in this view, for example, new opening in a slab is represented in Modified objects (Blue) and not presented as a changed property of a slab (Volume). For a full overview open (5) Table Overview.

2. Each part can be expanded. With that, you can see specific changes one after another.

3. Toggle the transparency of the structural model for a better overview of where are changed, deleted and new objects located.

4. Select changes by clicking on the selection box. Selected changes will be used for the update of the selected analysis model (6). Changes can be selected in the 3D scene and also in Table Overview. When no change is selected, everything is used for an update.

45

5. Table Overview: Click to open the table overview (Figure 36).

6. Analysis model selection (drop down): Select the analysis model you want to have updated.

7. Select the revision of the selected analysis model: Select which revision you would like to use for the update. Usually, the last revision is updated with the latest changes.

8. Visibility of analysis model (toggle): Toggle ON and OFF visibility of analysis model.

9. Use selection and move to next step: Once you selected changes for update and the right analysis model you can proceed to the next step.

Table Overview:

The Table Overview will provide you with all details about what was changed. Selection done in Table Overview works the same was as a selection in Review and in the 3D scene. The Table Overview consists of the following components you can use. (Figure 36)

Figure 36: Table Overview

1. Progress bar: Once is Table Overview opened, please wait until the table is fully processed.

2. Table headers:

   • Name - entity name

- Member type - slab, column etc.

- Type of change - New, Deleted, Modified object or property (based on grouping mentioned above)

- Attribute - what attribute was changed (if any, not relevant for New and Deleted objects)

- Previous attribute value - the value of an attribute in lower revision

- New attribute value - new attribute value in the higher selected revision

3. Select all check boxes: It will mark all the check boxes, use when you want to proceed with all changes or most of them.

4. Select specific change: It will mark only specific changes as changes you want to use for the update of your analysis model.

5. Filter: Write down keywords in the Filter field to reduce the number of rows based on keywords. Every row where the keyword is present will be shown, for example, "NetVolume" if you are interested in volumetric changes. (Attributes names are IFC specific and "NetVolume" might not be presented in your IFC, check the Structural model attribute structure first.)

6. Export to excel (.xlsx): You can download the complete table on your hard drive in .xlsx format. Filter (5) used is not taken into account and the complete table is always downloaded.

Once you have selected specific changes and analysis model you would like to have updated, you can click on "Use selection and move to next".

Notes:
In case compare button is in the inactive state, check if you have selected two revisions (Figure 32, bullet (6)).

5.4   Recognize, Align and Update

The following steps, including saving the model are the same as for Structural to Analysis model conversion described in chapter 4. For saving is recommended to save the model as a revision on the Bimplus, look at chapter 4.7. Settings are taken over from the Structural to Analysis workflow run, so in step 3, you should only check if settings fit and in case you have new objects, changed materials or cross-section check the Members, Cross-section and

Materials in step "3. Recognizer settings" and adjust them or map them if necessary.

The recognition and alignment, in this case, work the way that newly recognized are only selected changes, the rest of the structure is kept from the aligned model and in the alignment process are newly recognized objects connected to the already existing structure. All secondary objects like loads and supports are kept even for changed and modified objects and you do not need to model them again in SCIA Engineer.

In alignment, you can use all alignment modes (align model, partial alignment and alignment per groups) as mentioned in chapter 4.6.

6 Compare revisions of Analysis model

Compare revisions of analysis model workflow is comparing selected revision of analysis model. Thanks to this workflow you can easily track changes done in your model and create tasks for your colleagues and partners (Bimplus task) or export tasks in excel or BCF files in case you want to go beyond the Bimplus CDE cooperation.

6.1 Compare Revision

The first step is to select the model, revisions, general comparison scope and click the compare button. (Figure 37)
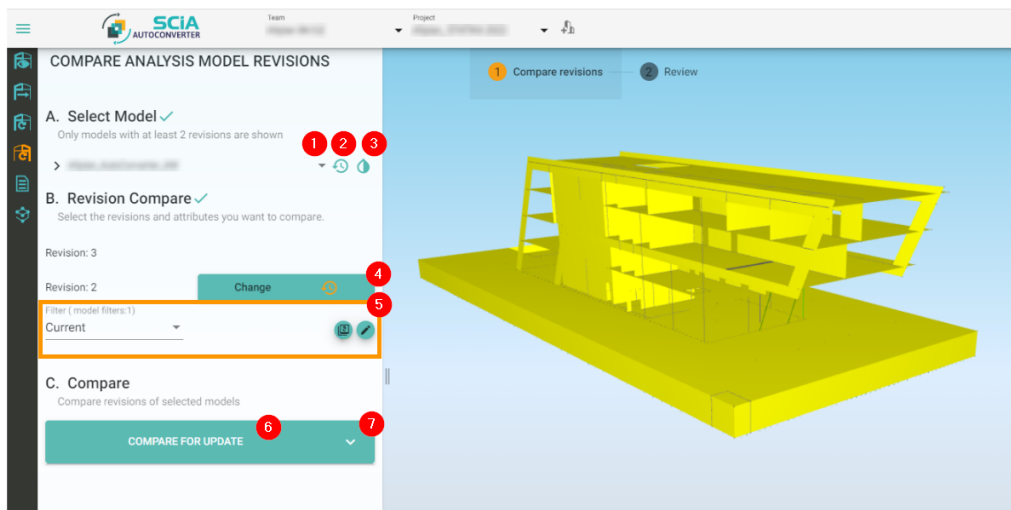
Figure 37: Compare revision

1. Select the model: Select the analysis model you want to use in the analysis model comparison. Only models with at least two revisions can be selected for model comparison.

2. Select the revision of analysis which you would like to use in the revision comparison.

3. Model transparency (toggle): Turn ON and OFF transparency of the selected model (revision) in the 3D scene.

4. Change revision: Select the second revision used in the revision comparison. The order of selected revisions does not play a role, revisions

are always compared the way the newer revision selected is compared to the older revision selected. (Same as in Chapter 5)

5. Property filter: Property filter provides you with an option to select only specific attributes to be compared in the comparison process. The property filters work the same way as described in Chapter 5.2.

6. Compare for update button: Click on the "Compare for update button" to run the revision comparison. This comparison is designed for tracking the changes and creating tasks.

7. Arrow, Full compare: Click on the "Full compare button" for running the full comparison. Full compare compares the whole SAF files, changes like newly created internal edges, modified lists of internal nodes of members, changed structural eccentricities etc. are included in this comparison. The result of this comparison is available in Table Overview only. Complete SAF files behind analysis models are being compared. This comparison you can use in case you are interested in tracking very detailed changes in the analysis model.

Notes: When you have no available models in model selection (1), please check if you have in the project analysis model with at least two revisions.

6.2   Review and Create Tasks

Once you Compared for update, the results are shown in the 3D scene. The result is presented based on what was changed, newly added or deleted, for example, "Modified cross-section", "Area" or "Lenght" of 1D member. Changes are also differentiated by colour. (Figure 38)

- Orange - Modified properties. Example: Changed cross-section (HEA300 -> HEA400)

- Light green - Deleted objects. Example: Removed column (analysis element).

- Dark green - New objects. Example: Newly added opening in slab or wall.

- Blue - Modified objects (geometrically). Example: Extended beam (change in Lenght) or cut slab (change in Area)
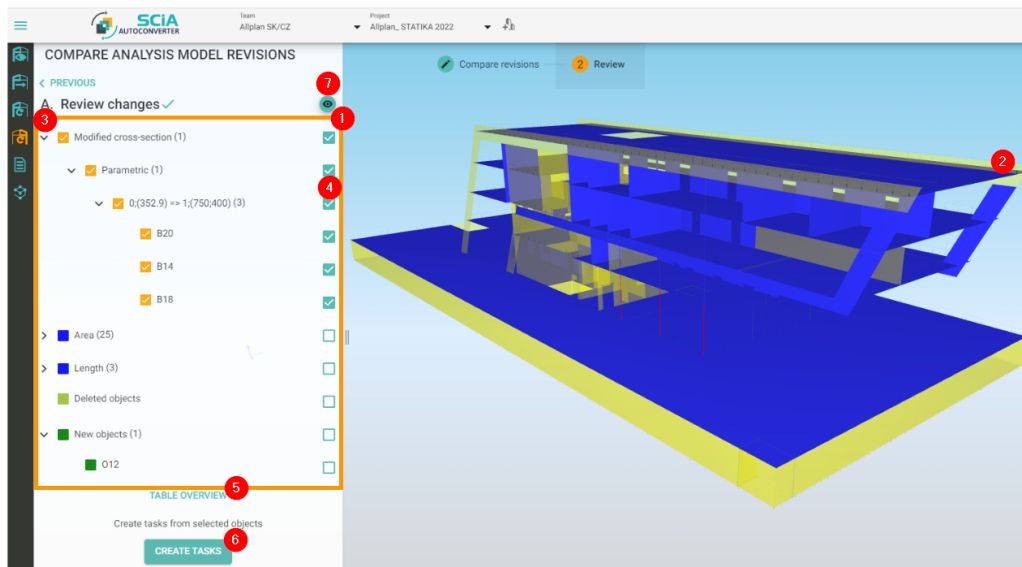
Figure 38: Review changes

Review of changes:

1. Result of revision comparison: The result of the revision comparison is presented in the left workflow bar.

2. Result displayed in 3D scene: The result of the comparison is available also in the 3D scene (including colours indicating the type of change).

3. Expand changes: You can expand each type of change to a deeper and more specific description of each change. Expansion can be done on multiple levels deep, where in the last level are shown names of all analysis objects affected by the specific change.

4. Select change box: Click on the selection box for selecting changes you want to be used for task creation. For example, when you click on a change related to the change of dimension of the parametric cross-section (4), all members affected by a change are selected automatically (B20, B14 and B18). In case you want to communicate a change related only to B14, selected only B14 in the last level of the change tree.

5. Table Overview: Click on "Table Overview" for opening a new window with a detailed overview of all changes. (Figure 39)

6. Create task: For creating a task click on the button. Window with task specification will appear. (Figure 40)

7. Analysis model visibility (toggle): Toggle visibility of the selected analysis model in the scene. When the model is set to visible, it is always transparent.

Table Overview: (Figure 39)

Figure 39: Table Overview

1. Table headers:

   - Type of change - indication for what was changed
   - Attribute - attribute affected by a change
   - Previous attribute value - attribute value for an older revision in comparison
   - New attribute value - attribute value in newer selected revision
   - Affected objects - list of analysis objects names affected by a change

2. Selection check box: You can select changes also here, in Table Overview, for task creation.

3. Create a task: Click on create a task button for opening a task creation window. All selected changes by (2) will be presented in a task.

Task creation: (Figure 40)
Task creation provides you with an option for how to communicate specific changes to your colleagues and partners. Each task has its own description,

priority, responsible person and date until it should be resolved. All tasks are available on Bimplus immediately once created in the SCIA AutoConverter. Further work with the task is handled on Bimplus only (editing a task, changing a state, adding notes and views etc.) A task can be exported to excel or a BCF file from Bimplus for further distribution.

Figure 40: Task creation

1. Task name: Task name is predefined based on the type of change. Click in the text field for editing the name.

2. Responsible person (Drop down): Select a responsible person for a task. List of people in the drop down is based on people with access to the project.

3. Task description: Task description is predefined based on the type of change. Click in text field for editing the task description.

4. Add people to CC: You can type in an email of a person who should be informed about a task.

5. Remove people from CC: Remove an email address from a CC by clicking on the cross button.

6. Time frame (Calendar selection): Set a date until when the task should be resolved.

7. Task priority (Drop down): Set the right priority for a created task - High, Medium, Low. Setting the right Priority will help your colleagues in prioritizing the urgent tasks first.

8. Save button: Click on "Save" for creating a new task.

Note: Once a task is created, the screenshot of the current view on the analysis model is taken and it is part of a task that is visible in the Bimplus viewer. The persons that are mentioned in the task (the responsible and the people in the cc) will automatically receive an email that a task has been created for them. There is a link in the mail that directly launches Bimplus viewer and loads the right project and model. Here you can use the Taskmanager to see all tasks appointed to you and you can process them. See the next chapter for more details.

6.3   Tasks on Bimplus

Tasks created in the SCIA AutoConverter can be viewed, further edited and exported to BCF or excel files using the Bimplus. For viewing the tasks on Bimplus portal, follow the steps below.

1. After logging into the Bimplus portal, using the same credentials as for SCIA AutoConverter. Open the project menu. (Figure 41)
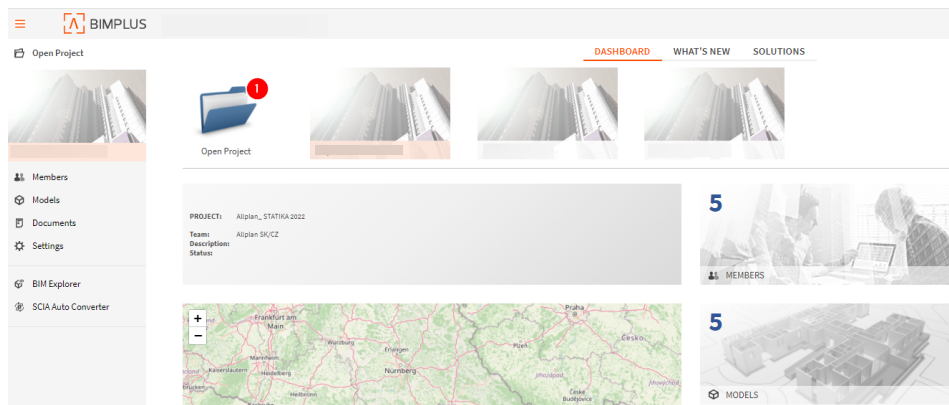

Figure 41: Open project on Bimplus

2. In the project menu, select the right team and project you want to work in. (Project, where is stored model with the created issue). (Figure 42)
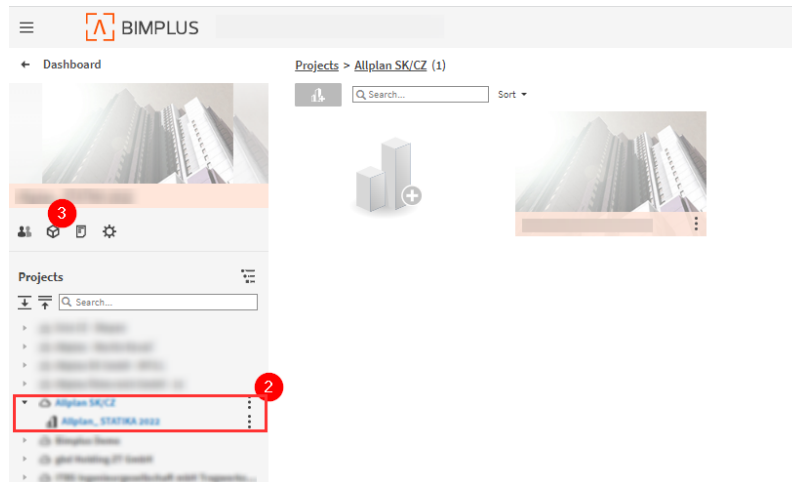
Figure 42: Open right team and project on Bimplus

3. Then click on the model icon to see available models in the project. (Figure 42)

4. Double-click on the structural model which is the issue related to opening the model in Bimplus viewer. (Figure 43)
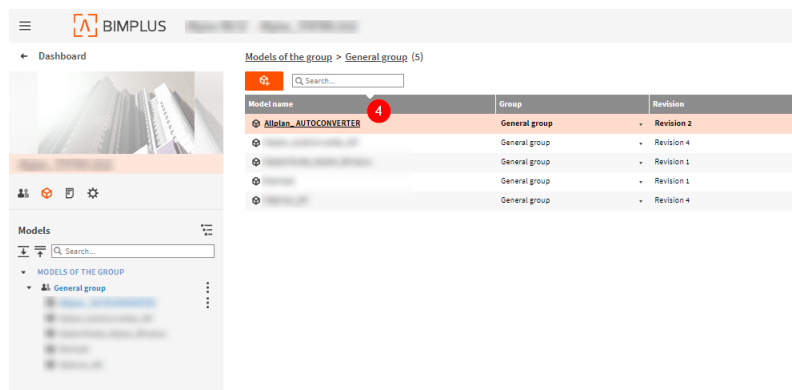

Figure 43: Open the model on Bimplus

5. Once in the viewer, click on the Issue manager icon in the right tab. (Figure 44)
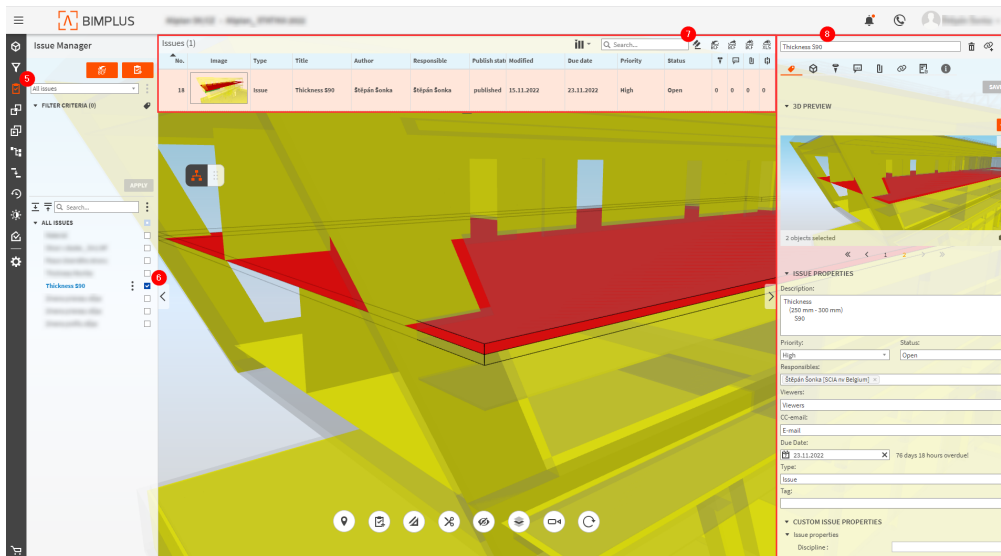
Figure 44: Bimplus viewer with opened issue

6. Select the issue from the issue list. (Figure 44)

7. On the top there are listed activated issue, with possibility to export issue to BCF and XLS format. (Figure 44)

8. In the right issue property bar you see your previously inputted data like description, priority etc. These fields can be edited here as well. (Figure 44)

7 Team management on Bimplus and support

In this chapter are described basic operations related to SCIA AutoConverter on the Bimplus portal. For more detailed information about Bimplus features see Bimplus website.

Assignment of license and adding your colleagues to the team:

Team management and assignment of SCIA AutoConverter licenses is done on Bimplus. From SCIA AutoConverter you can go to Bimplus using the link in the hamburger menu (see chapter Introduction) or through the direct link here Bimplus portal. When you are logged in, make sure that you have activated your team. Active team and project you can see at the top bar, next to the Bimplus logo. To change the active team, see Figure 42.

1. On the Bimplus dashboard, click on the "Team members" in the left tab and "Subscription and users" page should be opened in a new tab of your browser. (In case the option is missing, double-check that you have the right team active.) (Figure 45)
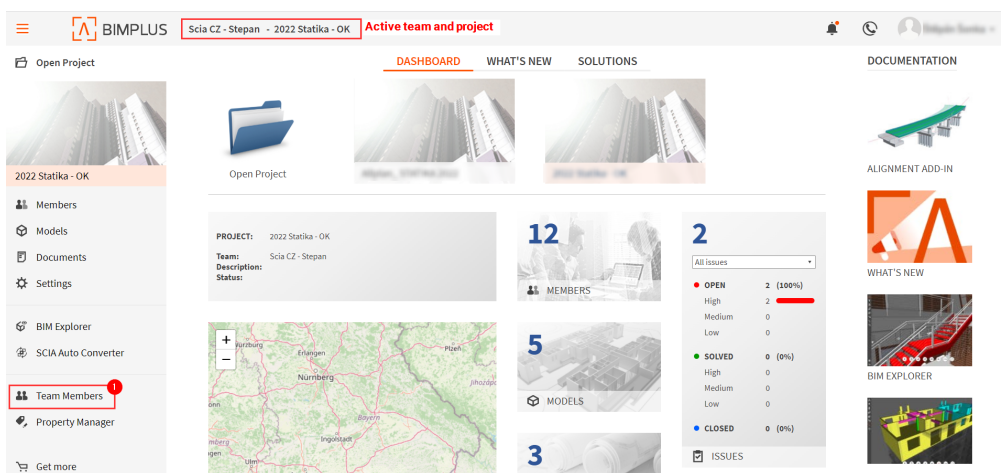

Figure 45: Bimplus dashboard - Team members

2. In the "Subscription and users" page you can manage the licenses of products and their assignment and people's access to the team as well. On top, you can browse the teams you are a member. (Figure 46)
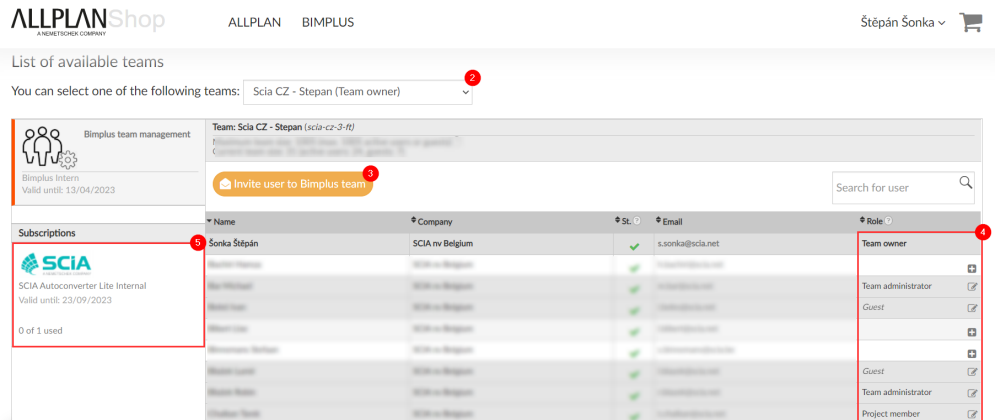
Figure 46: Team management

3. You can invite people to your team. In case these people do not have an Allplan account, they will be automatically asked to create one in the invitation email. (Figure 46)

4. In the list of collaborators in the team, you can manage rights. Click on the plus icon to add colleagues from your organisation to the team, or the pencil icon to edit rights. (Figure 46)

5. In the left tab, you can see your active subscriptions, click on the specific SCIA AutoConverter subscription for the assignment. (Figure 46)

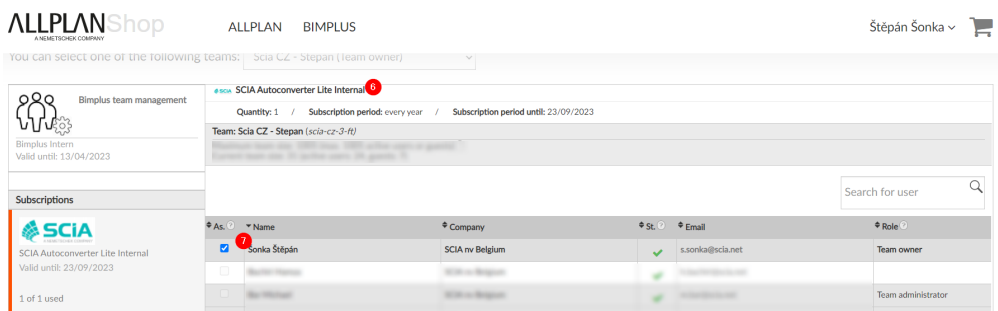6. Here you can see your selected product for assignment. (Figure 47)

Figure 47: License assignment

7. Use the check-boxes for licenses assignment for a specific member of the team. Figure 47

Notes: In case you have a possibility to create a company alias (email), you can add this alias to your team, assign the SCIA AutoConverter license

58

to it and share it with your colleagues.

Colleagues with Admin rights have access to all projects in the team automatically.

In case you run out of space in the team, consider either a team extension or removal of someone from the list in "Team members".

It is not necessary to own a Bimplus team for being a team member in a team owned by someone else.

Assignment of your colleagues to the specific project:

Adding your colleagues to a specific project can be done on the project level as well. Collaborators added to a specific project will be seen in "Team members" with the status "Project member" and they will have access to selected projects only.

1. On the Bimplus Dashboard click on "Open project". (Figure 48)

Figure 48: Open project

2. On the left, select the right team and project where you want to add your colleague in. (Figure 49)
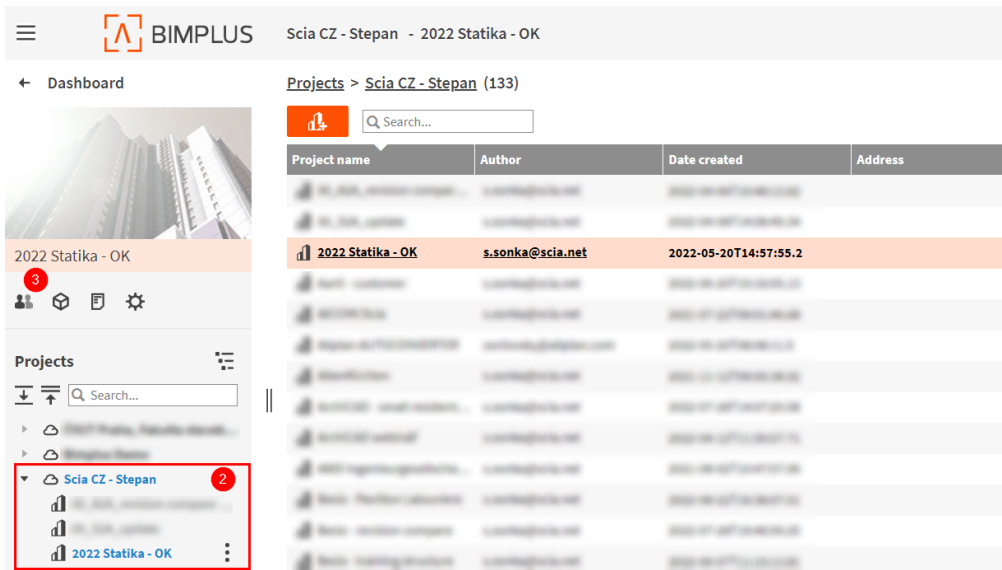
Figure 49: Select the right team, project and open "Administration"

3. Click on the "Administration of project members" icon. (Figure 49)

4. Click on "Invite member" for adding a new collegue to the project. (Figure 50)

Figure 50: Click on "Invite member"

5. Fill in the email, set the member role and click on the "Invite" button. (Figure 51)

60

Figure 51: Fill in the member email, rights and click on invite

8   Import and export of the SAF file

Import, and export of the SAF file you can use when you want to upload an existing SAF file from your local hard drive to the SCIA AutoConverter or when you want to export an already stored analysis model from the SCIA AutoConverter to a SAF file (.xlsx file).
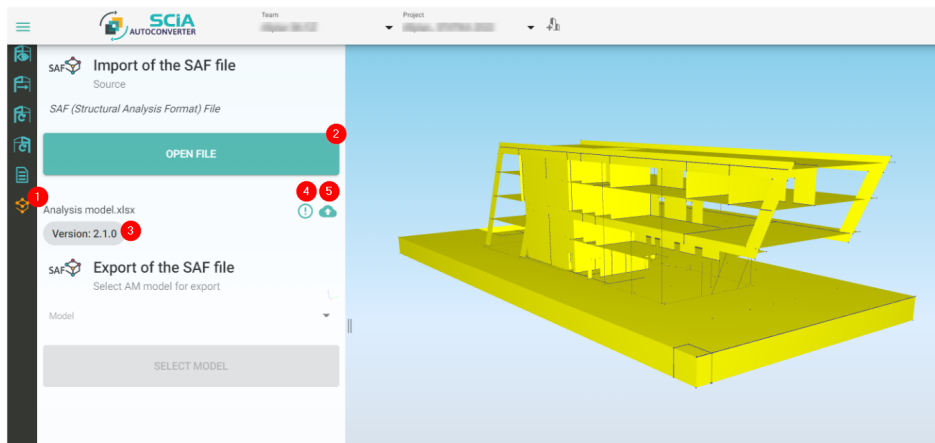
Import of a SAF file: (Figure 52)

Figure 52: Import of a SAF file

1. Import/Export of SAF file: The position of the Import/Export of SAF file in the left workflow bar.

2. Open file: Click on the "Open file" button for selecting a SAF file from your drive you want to visualise and save to the SCIA AutoConverter.

3. SAF name and version: Here you can see the name of SAF file active in the 3D scene and the version of the loaded SAF file.

4. Validation: Click on the "Exclamation mark" for expanding the validation window. In the validation window, you can track down the possible issues in the SAF file.

5. Save to Bimplus: Click on the cloud icon for saving the model in the SCIA AutoConverter. Model can be saved as a new one or a revision of the already existing model.

Export of a SAF file: (Figure 53)

1. Select an analysis model already saved in the SCIA AutoConverter.

2. Click on the "Save to SAF file" for opening a SAF export dialogue.
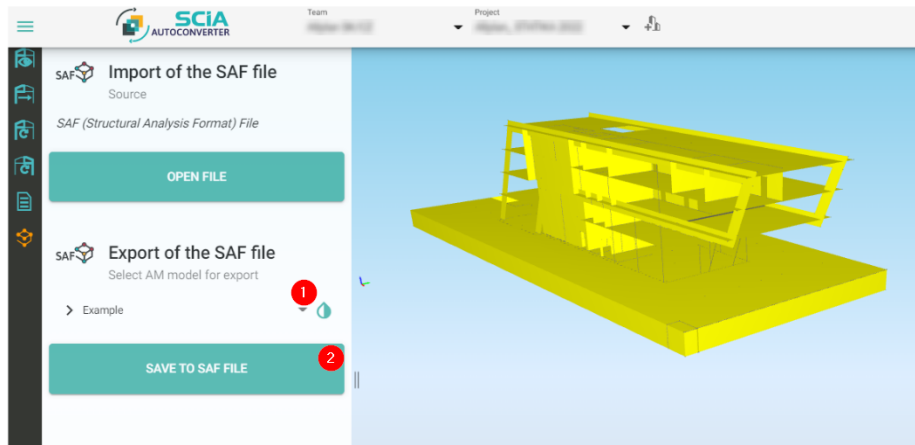

Figure 53: Export of a SAF file

9 Summary

SCIA AutoConverter is an online tool for structural engineers, providing them with possibilities to automate the analysis model creation process. The conversion from structural to analysis allows SCIA AutoConverter users to effectively remove non-loadbearing elements from the structure, redefine the role of elements in the model (beam, girder, slab, wall, solids etc.) if necessary and map steel profiles and materials to structural analysis profile and material libraries (SCIA Engineer).

Selected elements are automatically converter to analysis objects during recognition according to the set recognizer settings. Analysis elements not fulfilling the set recognizer conditions can be removed or kept, the decision is up to the structural engineer. Alignment settings allow SCIA AutoConverter users to make all necessary preparation (system line positioning, eccentricities setting, creating master planes in element behaviour etc.) for the best possible alignment result. Alignment can be done in various ways, everything at once with one settings, partial alignment based on selection or alignment per groups where every group can have its own settings for the best alignment result for complex structures.

The final model can be saved to the SCIA AutoConverter or saved to SAF file (.xlsx). SAF is now a widely used format for the transfer of analysis models between various stakeholders and software, therefore it is the right format for SCIA AutoConverter. See Who supports SAF?. Automatic conversion saves structural engineers time spend by remodelling the model from scratch.

Available for SCIA AutoConverter users are also update workflows. First mentioned in the SCIA AutoConverter user guide is the workflow "Update Analysis model based on changes in Structural Model". This workflow allows users to compare the structural model revisions, selected relevant changes for the structural engineering domain and update with them the already used analysis model. During the update process, additional model data like loads and supports are kept. This saves a significant amount of time with endless manual adjustments of the model.

The second update workflow is heading the other way around, in case SCIA AutoConverter users are making necessary changes in the analysis model, revisions of analysis models can be compared as well, and selected changes can be communicated back to the author of the BIM model via Bimplus task (BCF files, .xlsx files etc.). All changes done are trackable and have clear statuses which provides structural engineers with a better overview

and control of the whole process.

All the main workflows are heavily supported with SAF import/export options, SAF viewer, tutorial videos, release notes and a lot of self-explaining tooltips for the best possible outcome structural engineers can get.

10   Annexes

In annexes, you can find more information about specific parts of SCIA Auto-Converter application. This information is outside the user guide scope, but still very important and will help you to understand the processing behind the SCIA AutoConverter functions.

10.1   Parallel Member Detection

Parallel member detection general condition:

- Distance between centre lines of elements is smaller than 25 centimetres.

- Overlap in the length of elements merged together in at least 80%.
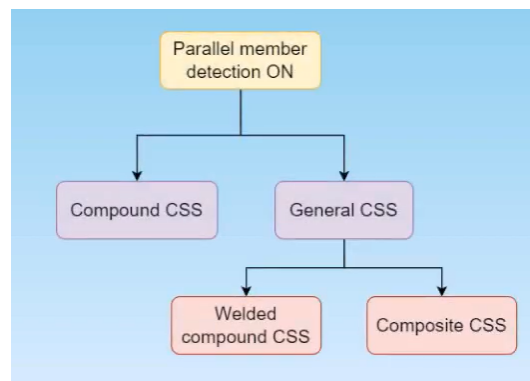
Diagram of processing: (Figure 54)

Figure 54: Parallel member detection diagram

- Compound cross-section - when possible, compound cross-section is created with priority. Members have to fulfil the general condition mentioned above, steel profiles have to be mapped to library profiles and compound type has to be supported. Supported compound types are mentioned below.

- General cross-section - when profiles are not mapped or compound type is not supported, then a general compound cross-section is created.

- Welded compound - general compound cross-section consisting of multiple mapped steel profiles

66

- Composite compound - general compound consisting of more shapes or profiles and more than one material

Supported compound cross-section types:

- Double I

- Double channel

- Starred angle

- Double angle

- Double pipe

All available compound cross-sections in SAF: SAF compound sections

10.2 System Line Positions

List of system line positions 1D:

- Centre

- Top

- Bottom

- Left

- Right

- Top left

- Top right

- Bottom left

- Bottom right

List of system lines positions 2D:

- Top

- Centre

- Bottom

List of eccentricities for 1D :

- None

- Y

- Z

- YZ

List of eccentricities for 2D:

- None

- Z