

Stade des Alpes - Grenoble
Etudes et Techniques Internationales (ETI)

Tutorial

BIM Toolbox

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. Scia is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2011 Scia Group nv. All rights reserved.

Table of contents

Contents

Table of contents	3
What, Why, When?	5
What is BIM toolbox?	5
Why should be used BIM toolbox?	5
When should be used BIM toolbox?	5
BIM toolbox service	6
Convert	6
General solid into beam/column.....	6
General solid into plate/wall	9
General solid into hollow core	12
Part of a general solid to a member	13
Beam/slab into general solid	17
Align	17
Align	17
Connect members/nodes	17
Check structure data	23
Evaluate	24
Clash check.....	24
Align	25
Master planes	25
Actions (align nodes of)	26
Display options	27
Advanced	27

What, Why, When?

What is BIM toolbox?

BIM (building information modelling) is a process of building data exchange during its life cycle. BIM toolbox is a tool that simplifies the work with data from architectural applications. It guides you from the import of a structural model to creation on the analytic model. Often after importing a model from another application a structure consists of unconnected general solids. That means a structural engineer needs to create Scia Engineer native members which can be connected together and used for the analysis. That all is possible with the **BIM toolbox** service.

Why BIM toolbox should be used?

Main reasons for using it:

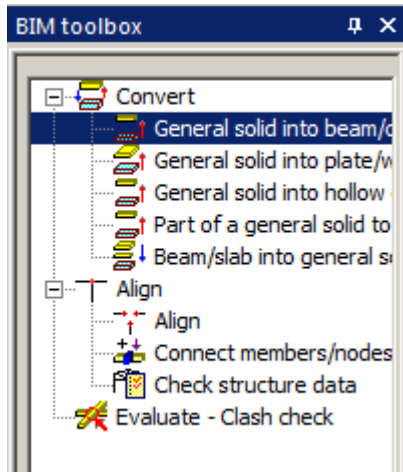
- Saves time – using functions **convert** and **align** is quicker than creating new entities from general solids and connecting unaligned entities manually.
- Saves work – the imported model created by an architect in other software needs modification for the purposes of finite element analysis - modification by BIM toolbox is the fastest way how create a correct analysis model.
- Keeps original structural model – in the same project both the original shape designed by an architect and the analysis model created by a structure engineer are kept. Then it is easy to compare if both models match.
- Creates analysis model in an easy way – the alignment and connect function creates analysis model very quickly from the imported structural shape.

When BIM toolbox should be used

BIM toolbox should be used if you need to create an analysis model from any imported file format in which the model is not connected properly for the purposes of finite element analysis, e.g. models imported from AllPlan and IFC files. In such a case you get the primary structural model. As you can see later some of the convert functions have been made primarily for the AllPlan interface.

BIM toolbox service

This chapter introduces the entire service in general and two functionalities (Convert and Align) in detailed.

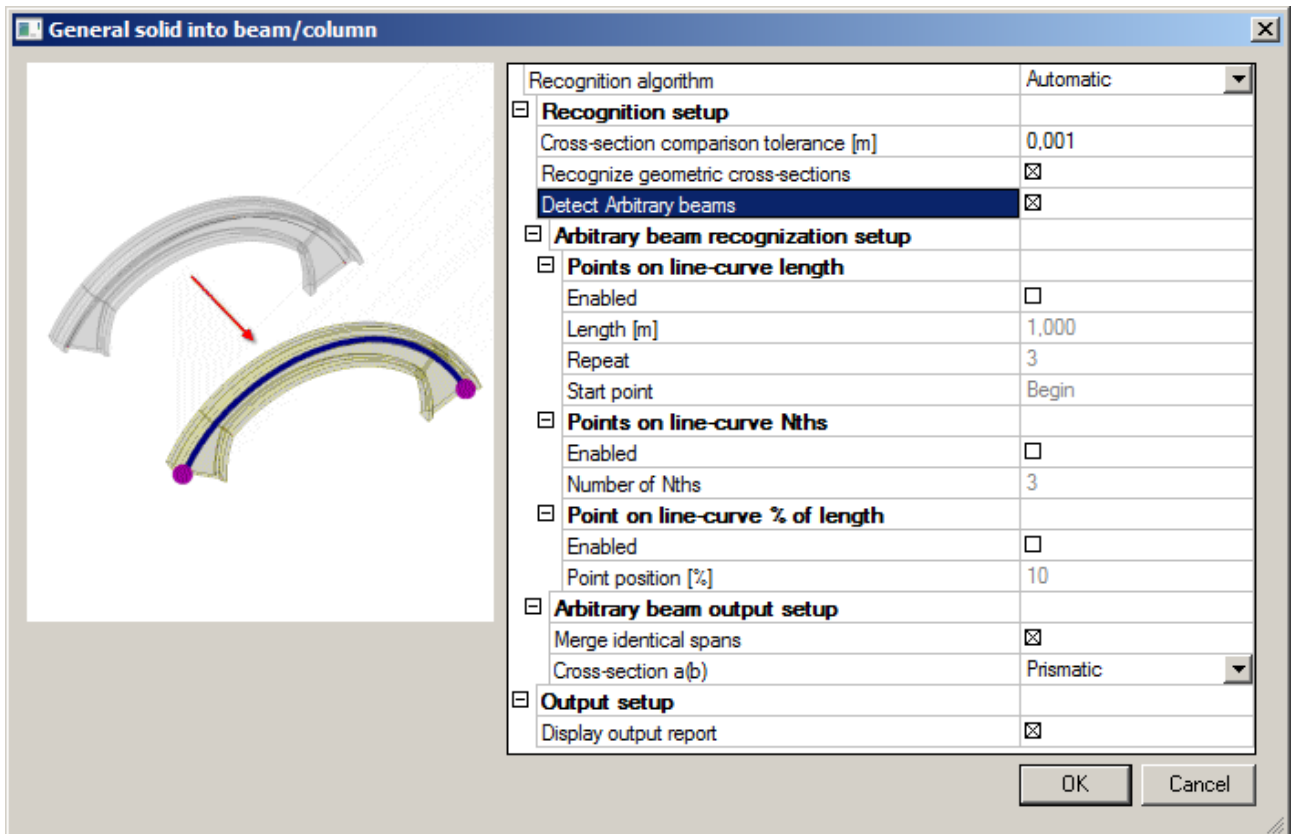


Convert

Not all entities imported from another file format can be imported directly as Scia Engineer native members. In these cases we import the elements as general solids which must be converted to the native members. This is possible by means of the functions described below. Sometimes the analysis shape cannot be the same as the shape of the general solid. Therefore, the proper shape is kept as the structure shape with the General mode.

General solid into beam/column

This function is for entities which have the character of a 1D member. Both prismatic and tapered ones are supported. There are two options for the recognition algorithm – **Automatic** and **Detect straight prismatic beams**. The main difference between the two options is the calculation of the member system line (see Reference Guide for detailed explanation).



Recognition setup

Cross-section comparison tolerance [m] – defines the maximal allowable distance of two points to determine if a new cross-section is created by the recogniser or if an already existing cross-section from the project database is used.

Recognize geometric cross-sections - if ON the recogniser tries to create a profile from the Geometric shape group. If OFF a general cross-section is created.

Detect arbitrary beams – if ON the algorithm tries to detect the variability of profile along the beam length and creates an arbitrary beam.

Arbitrary beam recognition setup

Points on line-curve length – recognises the shape of the cross-section in specified length intervals.

Points on line-curve Nths – recognises the shape of cross-sections in points which are defined in N-ths of the beam length.

Points on line-curve % of length - recognises the shape of cross-sections in places which are defined with percentage of the beam length.

Arbitrary beam output setup

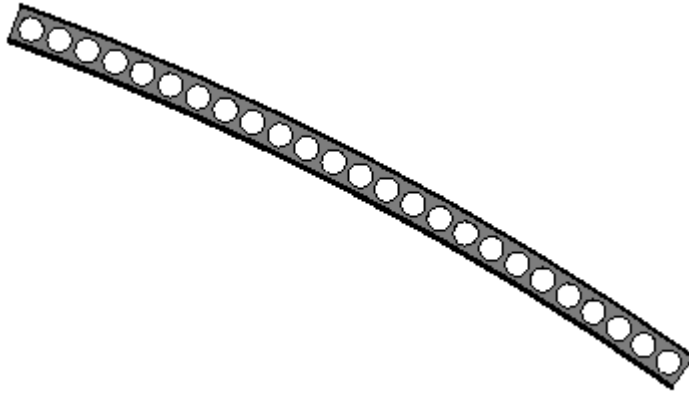
Merge identical spans – if ON all spans with identical cross-section are merged into one.

Cross-section a(b) – Prismatic – a span has assigned only one cross-section.

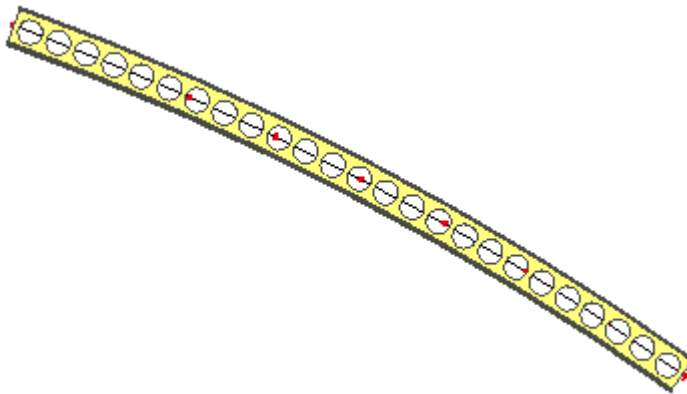
- **Two Css** – a span has different profile on each end.

Example 1

1. A curved beam which is imported to Scia Engineer as a general solid.



2. Run convert function with default settings.
3. A polyline beam is created with an I profile and web openings are preserved in structural shape.

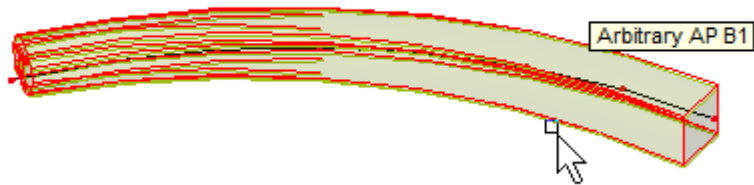


Example 2

1. A curved arbitrary beam is imported to Scia Engineer as a general solid.

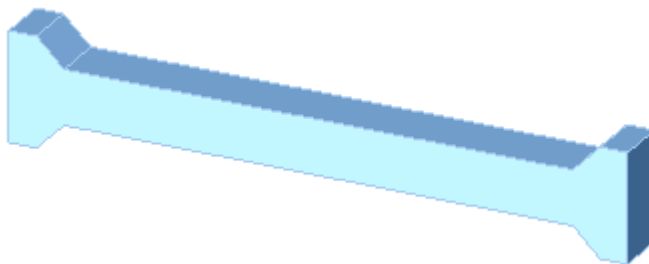


2. The convert function was run with following settings:
Detect arbitrary beams – yes.
Points on line-curve length – enabled.
Cross-section a(b) – two Css.
3. An arbitrary polyline beam is created.

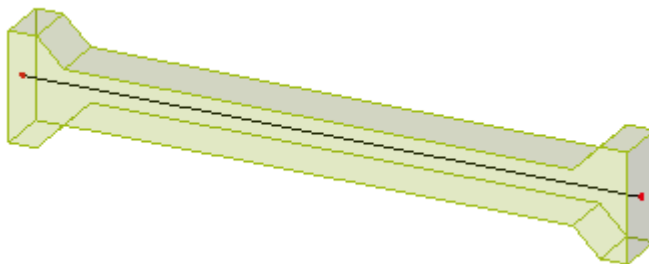


Example 3

1. A straight arbitrary beam is imported to a project.

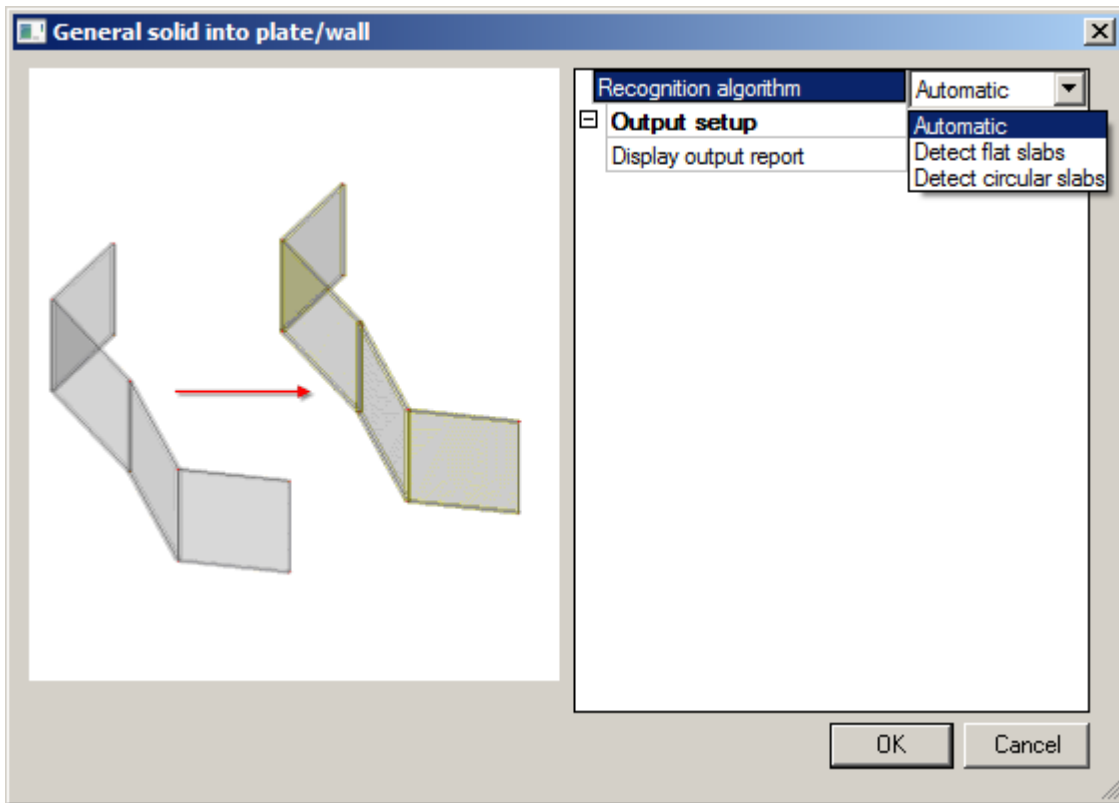


2. The convert function was run with following settings:
 - Detect straight prismatic beams.**
 - Detect arbitrary beams** – yes.
 - Points on line-curve length** – enabled with **Repeat** -5.
 - Cross-section a(b)** – two C_{ss}.
3. An arbitrary beam with 5 spans is created.



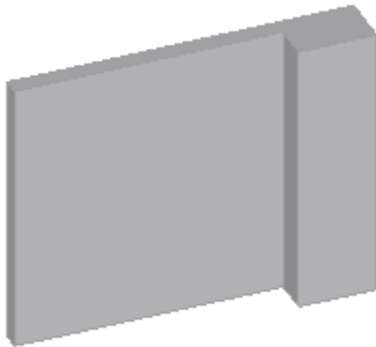
General solid into plate/wall

This function is for entities which have the character of a 2D member. Only slabs with a constant thickness are supported. There are three options for the recognition algorithm – **Automatic**, **Detect flat slabs** and **Detect circular slabs**. The main difference between them is the calculation of member system line. **Automatic** – detects all possible flat slabs in a solid (see Example 1). **Detect flat slabs** – creates only flat slabs in a plane of a solid (see Example 1). **Detect circular slabs** – converts a curved solid into a circular slab (see Example 2).

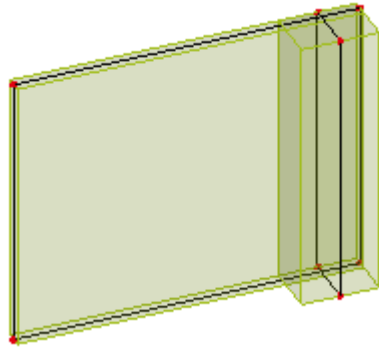


Example 1

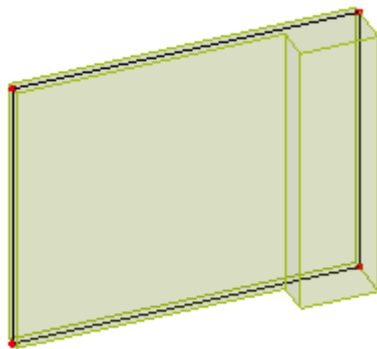
1. A non-prismatic wall is imported to Scia Engineer as a general solid.



2. A result for **Automatic** algorithm is creation of two perpendicular walls.



3. A result for **Detect flat slabs** algorithm is a creation of a prismatic wall.

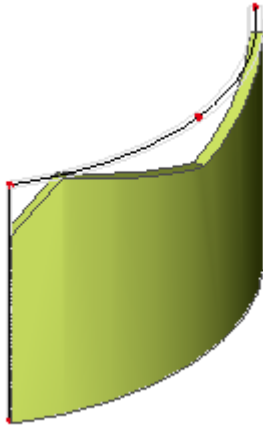


Example 2

1. A curved wall with variable height is imported to a project.



2. The result for **Detect circular slabs** algorithm is a Scia Engineer native circle arc wall. In the picture there is seen that the original shape is kept for structural shape.

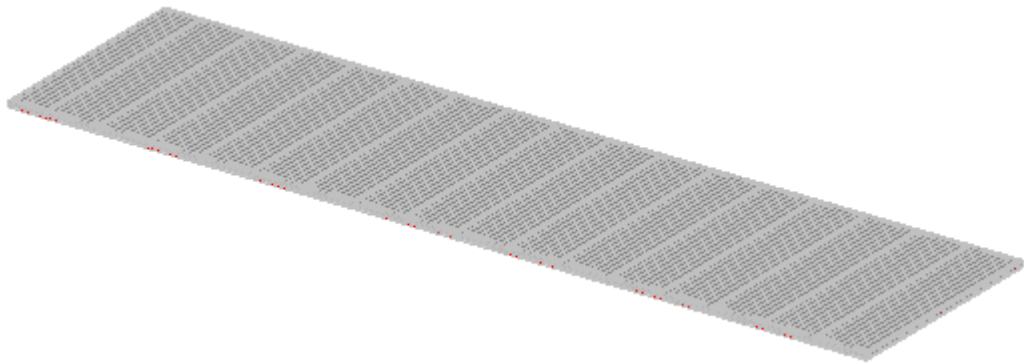


General solid into hollow core

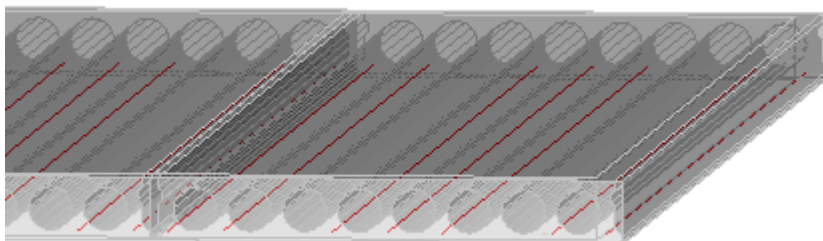
This function is mainly for AllPlan interface for an entity which has a character of hollow core 2D member. The result in Scia Engineer is a prefab slab with beams inside.

Example

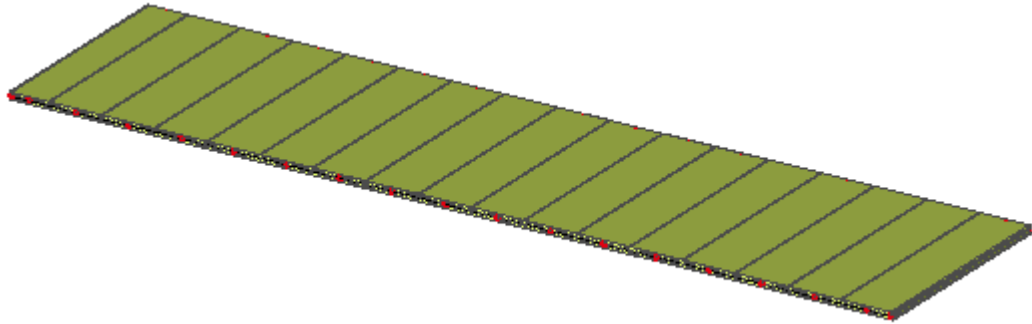
1. An imported solid from Nemetschek AllPlan.



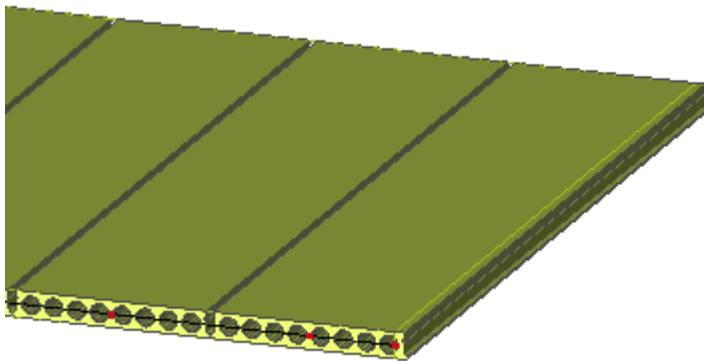
Detailed view at a hollow core slab part.



2. After running the function – a whole prefab slab.

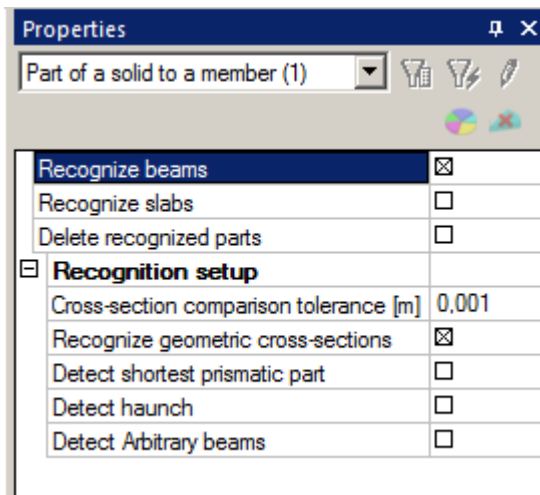


A detailed view of the beams with the hollow core profile.



Part of a general solid to a member

For complicated solids which have a character of a group of more members. This allows for converting only a part of a solid (manually defined) into Scia Engineer native 1D and 2D members. First, it is necessary to choose if the part is to be converted into a beam or a slab. Only one of the options **Recognize beams** or **Recognize slabs** can be selected.



Recognize beams – if ON a beam will be generated from the selected part.

Recognize slabs – if ON a slab will be generated from the selected part.

Delete recognized parts – if ON the original part of a solid will be removed. If OFF the solid is not changed after converting of the selected part into a member.

Recognition setup

Cross-section comparison tolerance [m] - defines the maximal allowable distance of two points to determine if a new cross-section is created by the recogniser or if an already existing cross-section in the database of the project is used.

Recognize geometric cross-sections - if ON the recogniser tries to create a profile from Geometric shape group. If OFF a general cross-section is created.

Detect shortest prismatic part – if ON, only the prismatic part of arbitrary solid is detected for conversion (see fig.1).

- If OFF, the part of solid for converting is taken from the beginning (see fig.2).

Fig.1

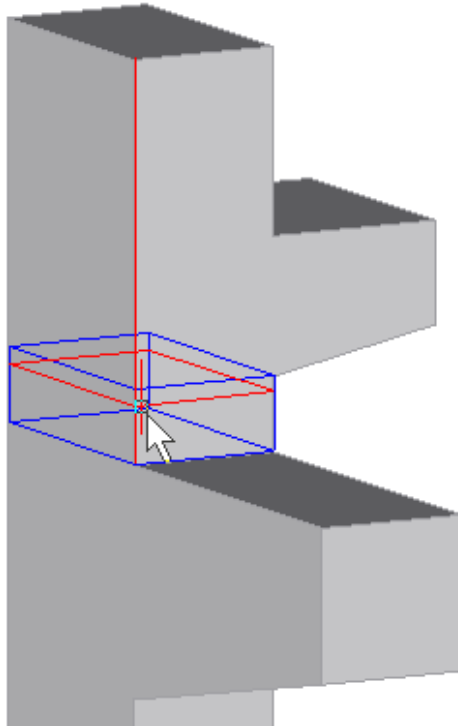
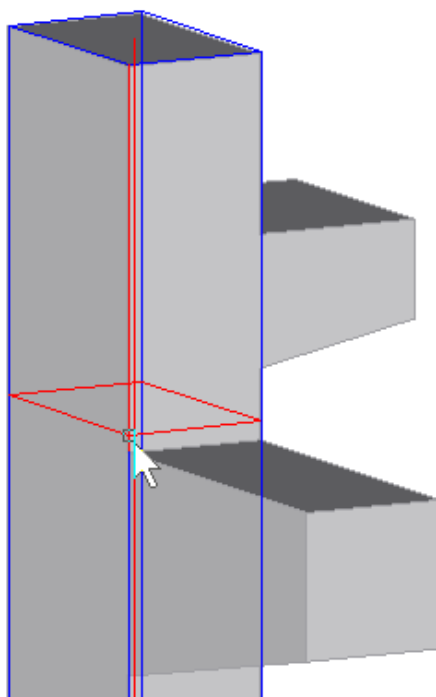


Fig.2



Detect haunch – if ON the algorithm tries to detect the variability of profile and creates a haunched beam (see the following example).

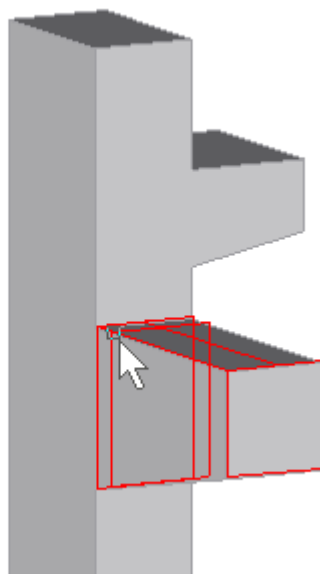
Detect arbitrary beams - if ON the algorithm tries to detect the variability of profile and creates an arbitrary beam.

Example 1

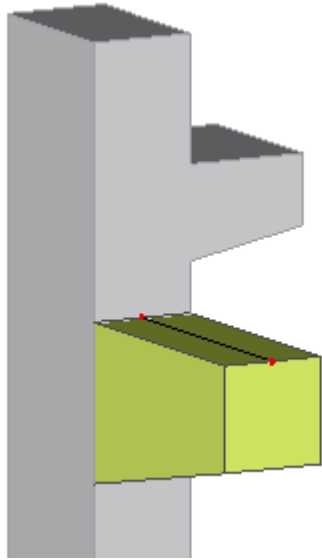
1. A beam with corbels is imported to Scia Engineer project as a solid.



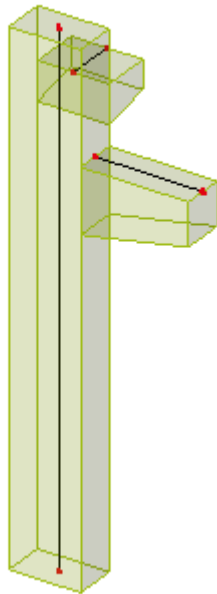
2. The direction of beam depends on the edge which is select.



3. The result is a haunched beam.

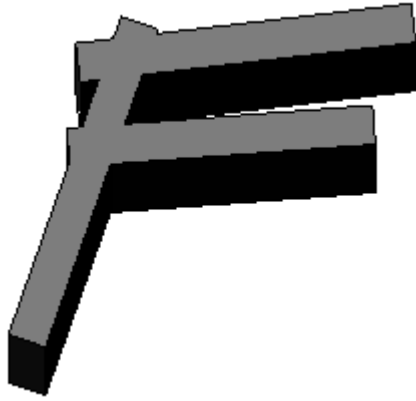


4. The beam with corbels after conversion.

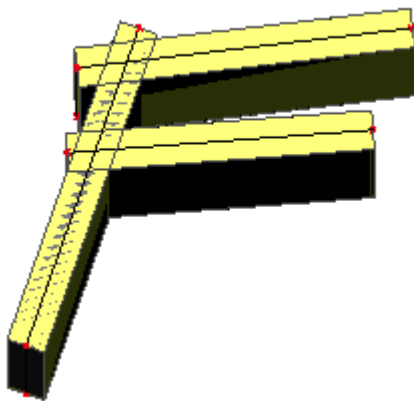


Example 2

1. A solid with a character of 2D members.



2. A result is three walls.



Beam/slab into general solid

Structural shape of native 1D and 2D members is converted into general solids. If the entity has an opening, subregion or haunch the result shape is made without any information about these add data.

Align

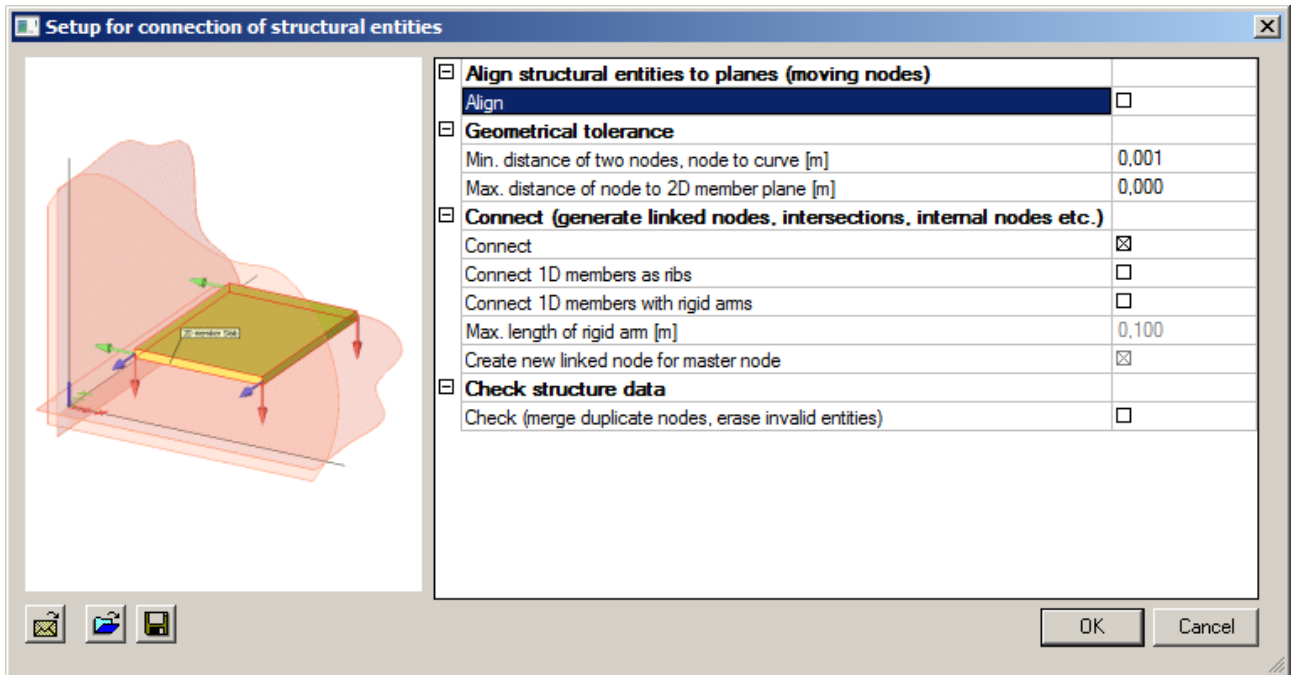
Lots of imported models are not connected, for example after an import from the IFC file. The reason is that some IFC files with the coordination view schema contain information only about the volumes of entities. 1D member axes and 2D member system planes are generated during the import to Scia Engineer.

Align

See the detailed description in the next chapter.

Connect members/nodes

The first part of this dialog is **Align**. In fact it is the same functionality as the function **Align** which is described in the separate chapter. The reason for doubling of this function is keeping the original functionality simultaneously with the new one. It does not have all options as the new **Align** itself. The next group is Geometrical tolerance which is also in the dialog **Setup > Geometry/Graphics**. If a user changes the values in one dialog, the value is automatically changed in the second one. Some types of connection from the group **Connect** can be done with separate functions. But this is the only place where all connect functions are together. The last part **Check structure data** calls the same dialog as the command in the next sub chapter.



Align structural entities to planes (moving nodes)

<input type="checkbox"/>	Align structural entities to planes (moving nodes)	
<input type="checkbox"/>	Align	<input checked="" type="checkbox"/>
<input type="checkbox"/>	Master planes	
<input type="checkbox"/>	Planes of parametric input	<input checked="" type="checkbox"/>
<input type="checkbox"/>	GCS parallel planes	<input checked="" type="checkbox"/>
<input type="checkbox"/>	UCS XY planes	<input checked="" type="checkbox"/>
<input type="checkbox"/>	UCS XY parallel planes	<input type="checkbox"/>
<input type="checkbox"/>	Storey planes	<input type="checkbox"/>
<input type="checkbox"/>	Section planes	<input type="checkbox"/>
<input type="checkbox"/>	Line Grid planes	<input type="checkbox"/>
<input type="checkbox"/>	Beams LCS planes	<input type="checkbox"/>
<input type="checkbox"/>	Align to slab planes globally	<input type="checkbox"/>
<input type="checkbox"/>	Max. distance between parallel master planes [m]	0,400
<input type="checkbox"/>	Max. angle between master planes [deg]	1,00
<input type="checkbox"/>	Parametrize the structure by master planes	<input type="checkbox"/>
<input type="checkbox"/>	Limits	
<input type="checkbox"/>	Max. distance between master plane and node to be aligned [m]	0,510
<input type="checkbox"/>	Max. total displacement of node [m]	0,510
<input type="checkbox"/>	Keep original shape of the model (by eccentricities)	<input checked="" type="checkbox"/>

Master planes – planes which are used to move the entities or nodes into.

Planes of parametric input – this option is available only if at least one coordinate of a node is defined by means of a parameter. Then the plane which contains this node is determined as a master plane and other nodes are aligned to this plane.

GCS parallel planes - if ON, entities are aligned in order to fit into any of parallel planes with GCS.

UCS XY planes – if ON, entities are aligned in order to fit into any XY planes of defined UCSs.

UCS XY parallel planes – if ON, entities are aligned to fit into a plane which is parallel with the XY plane of defined UCSs.

Storey planes – if ON, entities are aligned in order to fit into any of defined storey planes.

Section planes – if ON, entities are aligned in order to fit into any of defined section planes.

Line Grid planes – if ON, entities are aligned in order to fit into any plane of the defined line grids.

Beams LCS planes – if ON, 1D members are extended or shortened along their normal axis or moved into LCS plane of other member.

Align to slab planes globally – if ON, 2D members are extended in their plane.

Max. distance between parallel master planes [m] – defines the maximal distance of parallel planes. If the distance of a node is lower than this value, no new parallel plane is created in the node.

Max. angle between master planes [deg] - defines the maximal angle between master planes. If the angle between the existing plane and distance of a node is shorter than this value, then no new parallel plane is created in the node.

Parameterize the structure by master planes – this option is available only if at least one coordinate of a structure node is defined by means of a parameter.

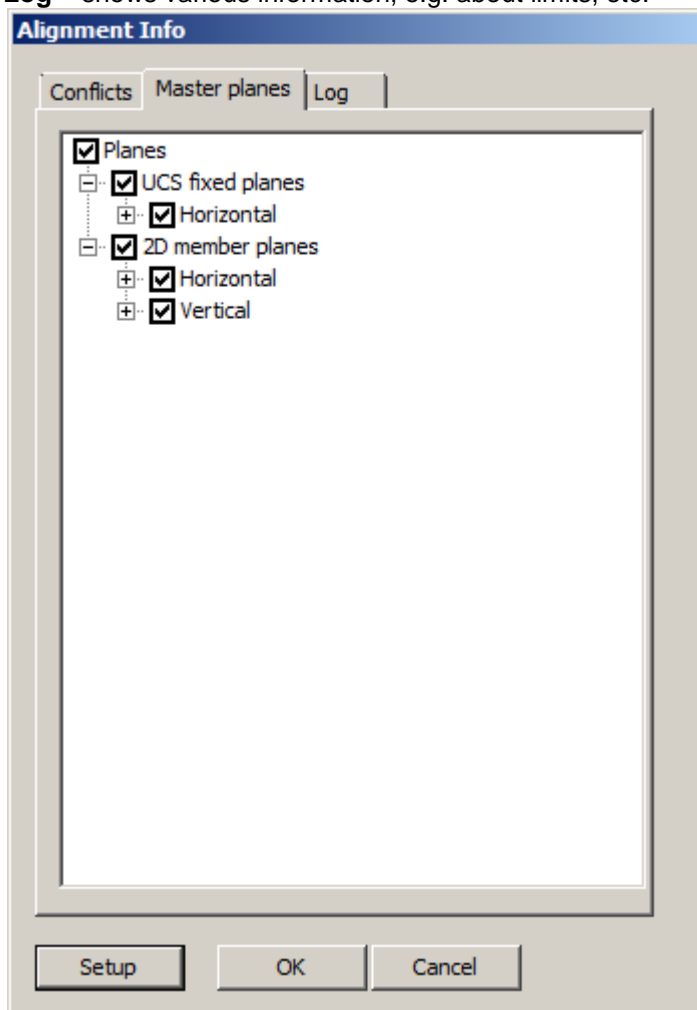
 **Note:** LCS planes of 2D members are always used for alignment.

Alignment info dialog

Conflicts – displays a list of entities and nodes which are in conflict with another one.

Master Planes - if any plane is selected it is displayed in blue contour in the graphical window. By means of check box you can select or unselect a particular plane or an entire group of planes.

Log – shows various information, e.g. about limits, etc.



Setup – runs Setup for connection of structural entities dialog again.

OK – confirms the selected Master planes.

Cancel – cancels the function.

Limits

Max. distance between master plane and node to be aligned [m] – defines maximal distance when a node is aligned into the master plane. If the distance is bigger the function is not performed.

Max. total displacement of node [m] – defines maximal total displacement of a node when the node can be moved to the master plane. If the total displacement is bigger the node is not moved. This value must not be lower than the value specified above.

Keep original shape of the model (by eccentricities) – if ON, the original shape remains in the original location by means of a defined eccentricity.

Geometrical tolerance

The values from this group are used for all geometrical operations.

Min. distance of two nodes, node to curve [m] – defines the minimum distance when two nodes are considered as separate nodes. If the distance is lower than the defined value, the nodes are considered as one.

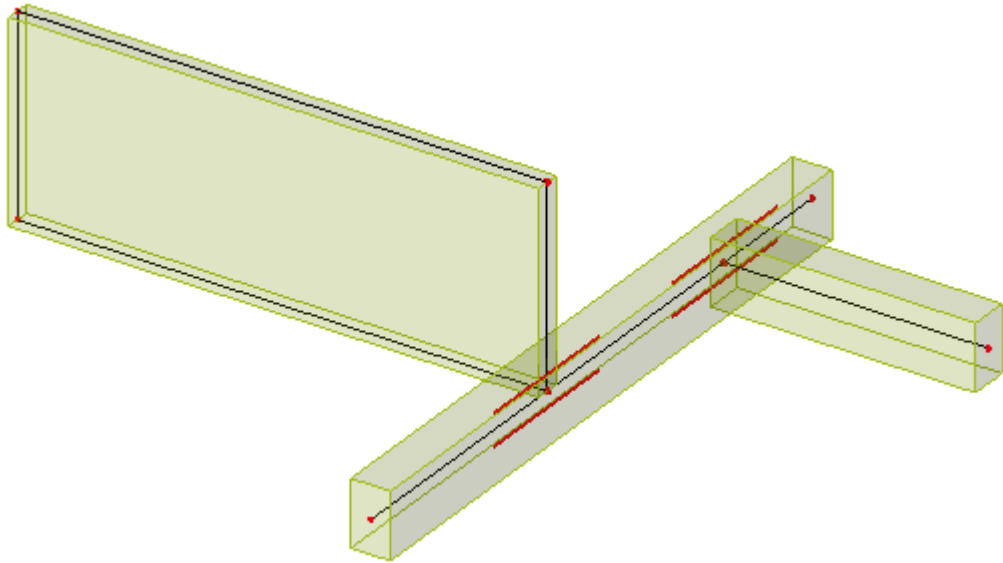
Max. distance of node to 2D member plane [m] – defines the maximum distance of a node from 2D member system plane. If the node has greater distance from 2D member of which it is a part, the geometry of 2D member is evaluated as invalid.

Connect

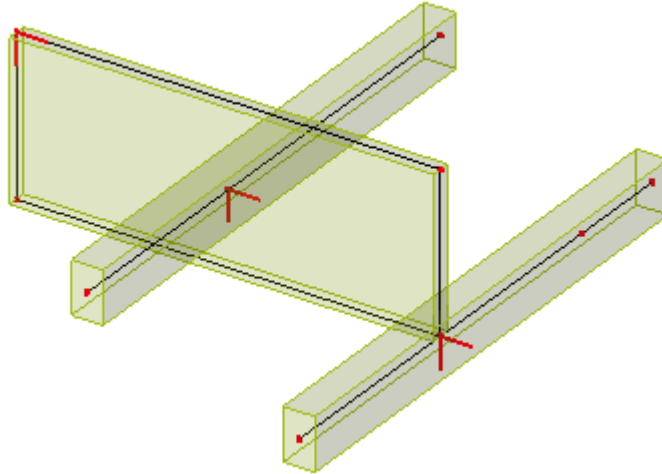
This group contains functions which allow for connecting members by different ways, such as linked nodes, intersections, rigid arms or joining beams as ribs to a slab.

Connect – if ON, all intersecting entities are connected; this allows for transferring of loads and internal forces between them. Entities are connected in the following way:

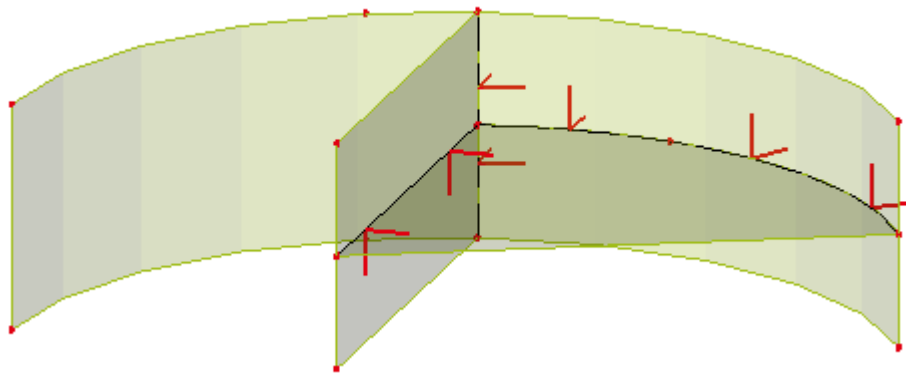
- a) 1D member + node – linked node



- b) 2D member + 1D member – internal node (point intersection)



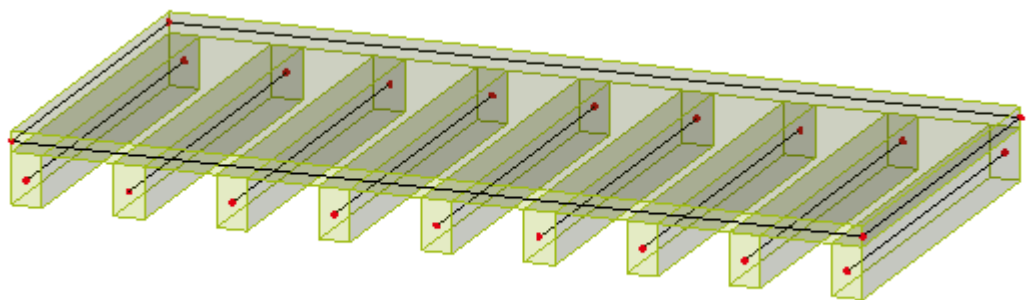
- c) 2D member + 2D member – internal edge (line intersection)



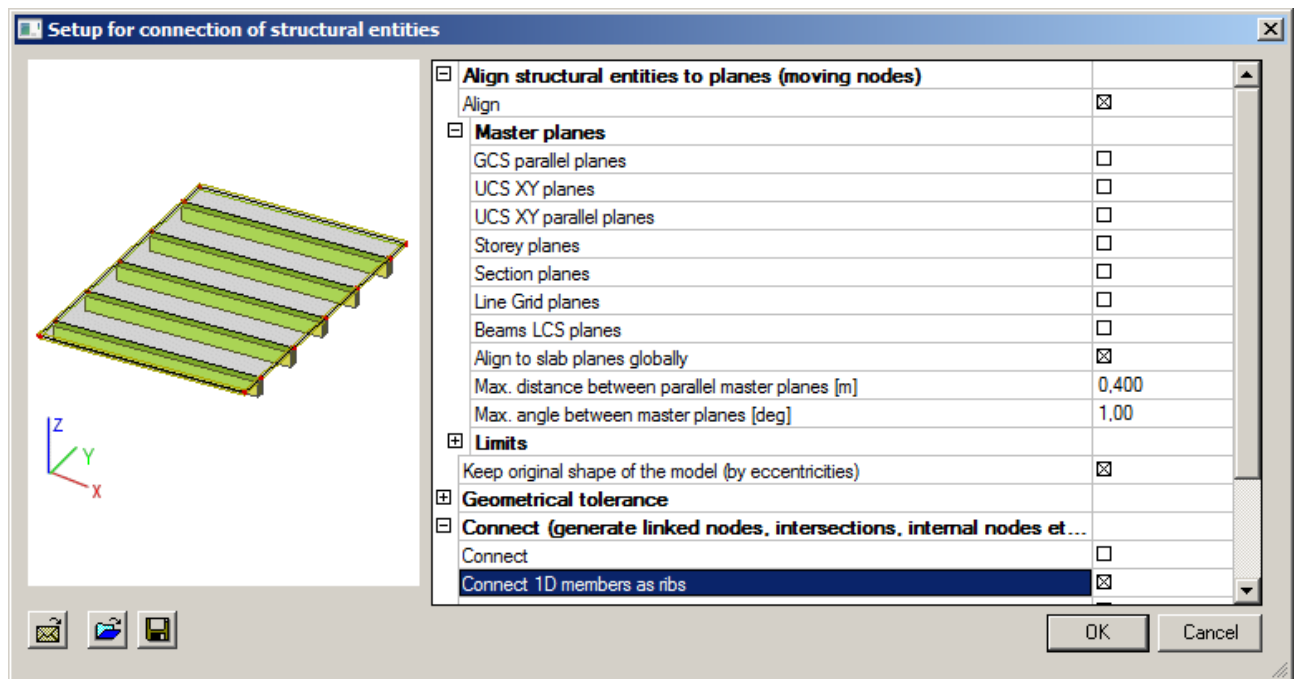
Connect 1D members as ribs – if ON, all selected beams that have the axes in the system plane of the slab are connected to the slab as plate ribs. If the axis of the beam is not in the system plane of the slab it is necessary to use function Align to slab planes globally first.

Example

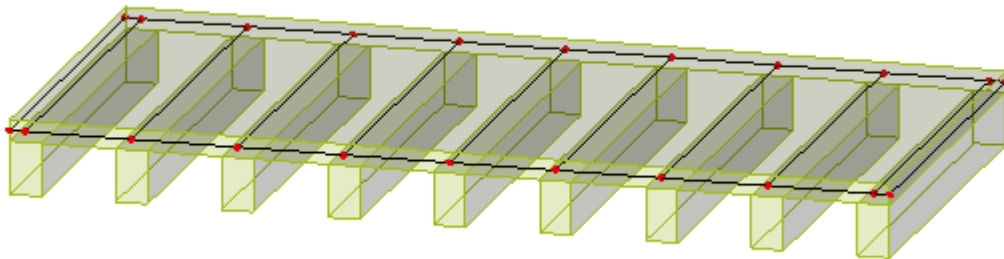
1. There are a plate and separate beams in the project.



2. Run **Connect members/nodes** function.
3. In **Setup for connection of structural entities** dialog select **Align > Master planes > Align to slab planes globally** and **Connect > Connect 1D members as ribs**. Check if the limit values are correct.



4. Confirm the dialog.



Connect 1D members with rigid arms – if ON, all the nodes of 1D members whose distance is shorter than the values defined in the next option will be connected with a rigid arm.

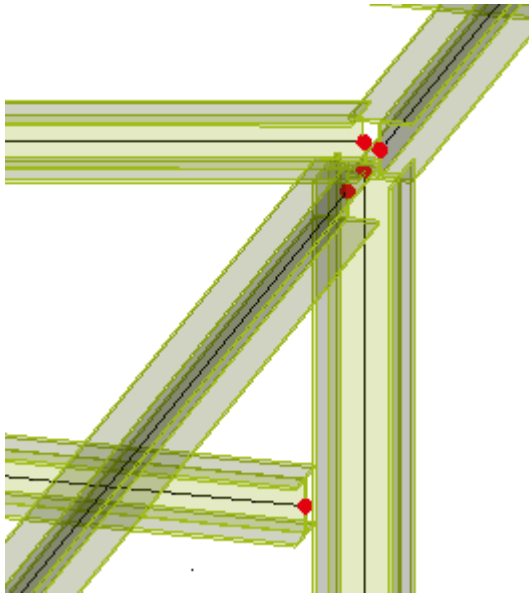
Max. length of rigid arm [m] – defines the maximum distance of two nodes or a node and an axis of the beam for which a rigid arm will be created.

Create new linked node for master node – if ON, a linked node will be created on the beam which has a node closer than specified by the value for the maximal length of the rigid arm. The linked node is always the master node of the rigid arm.

Example

This option is for situations when the user does not want to change the length of 1D members but he needs to analyse the model. The rigid arm entity connects the structure and transfers internal forces between beams.

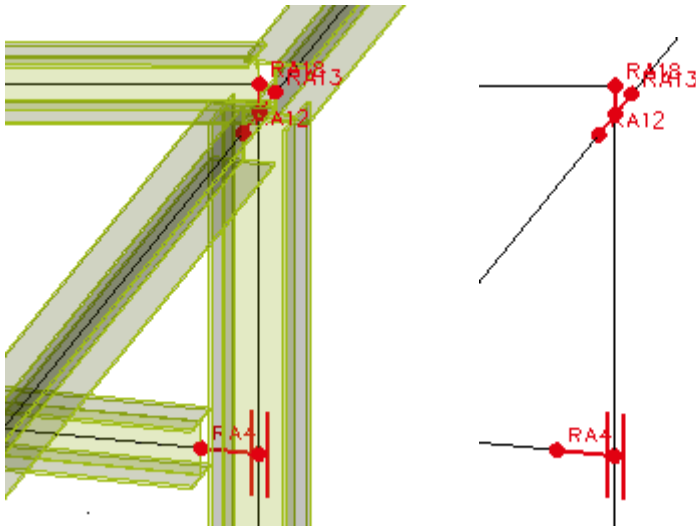
1. There is imported an unaligned frame structure into the project.



2. Select the option for connecting member with rigid arms and specify the maximal length of the rigid arm.

<input type="checkbox"/> Connect (generate linked nodes, intersections, internal nodes etc.)	
Connect	<input type="checkbox"/>
Connect 1D members as ribs	<input type="checkbox"/>
Connect 1D members with rigid arms	<input checked="" type="checkbox"/>
Max. length of rigid arm [m]	0,200
Create new linked node for master node	<input checked="" type="checkbox"/>

3. Confirm the dialog.
4. In the following pictures you can see how the nodes were connected.



Check structure data

Check – see the subchapter below.

Check structure data

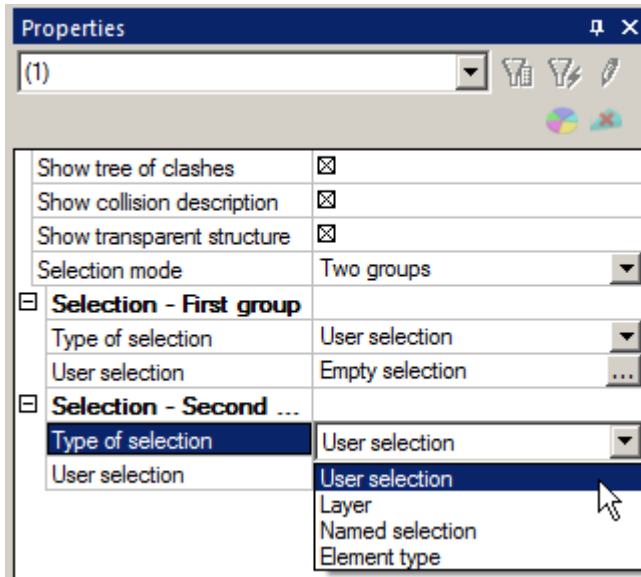
It is possible to call this function from various locations. It is here because it is appropriate to check model geometry more often if you use geometrical operations. It checks lots of things such as duplicate nodes, free nodes, members with zero length and undefined nodes, duplicate beams or slab, positions

of additional data, etc. For more detailed description of this function see Scia Engineer Reference Guide.

Evaluate

Clash check

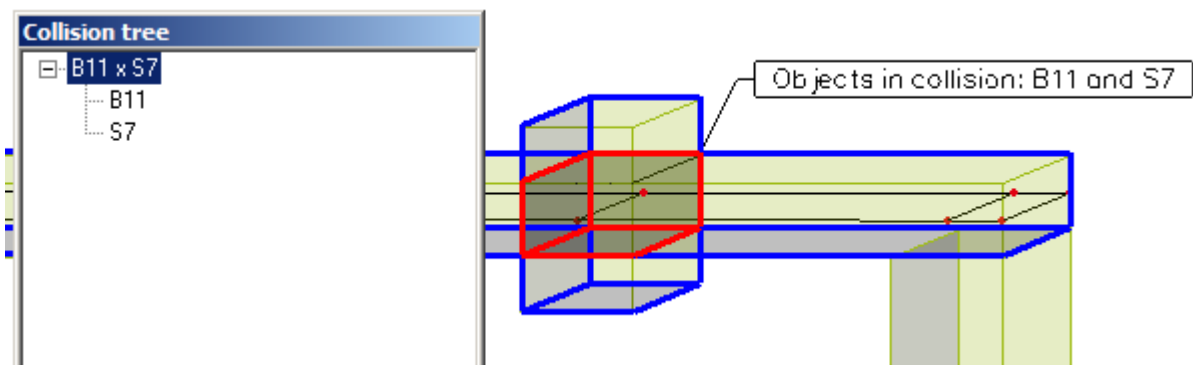
This function checks whether two or more entities (solids, 1D and 2D members, reinforcements) intersect each other. Then new solid is created in the intersection of the entities; this new solid is called collision solid.



Show tree of clashes – if ON, a dialog with a list of all collisions between entities is shown. If you click on the collision of two entities you will see a blue contour around the colliding entities and a red contour around the collision itself – see the picture below. If you click only on the concrete member you will see only the member with blue contour.

Show collision description – if ON, a label of collision is shown in the graphical window.

Show transparent structure – if ON, rendering of structure is set to transparent.



Selection mode – One group – a selection of entities by one criterion,
Two groups – a selection of entities by other criterion than in group one.

In each group it is possible to define the following Type of selection:

User selection – a list of selected elements for clash check.

Layer – all entities in one layer are selected for clash check.

Named selection – a selection by means of named selection

Element type – a selection by type of entities (1D members, 2D members, general solids or free bars).

Action buttons

Run clash check – runs the check and creates all collision objects.

Delete collision objects – removes all created collision objects.

End clash check – performs the function.

Align

This function can do the same as the original **Align** function and has some new extra functionality. New options for displaying of alignment-results before you perform the action itself are available.

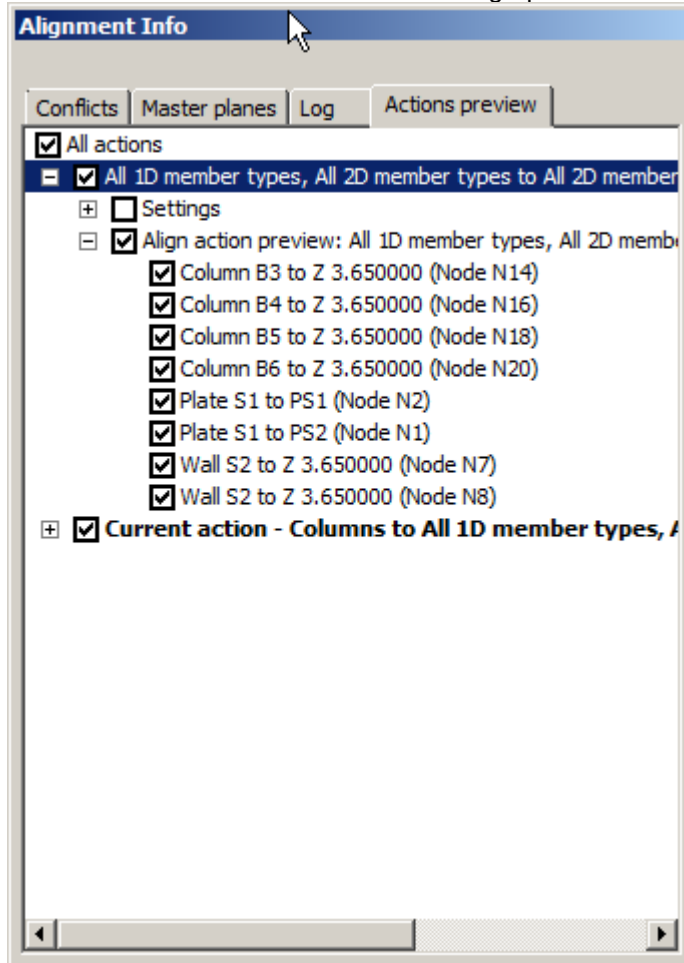
Show alignment info – if ON, Alignment Info dialog is shown. There are four tabs:

Conflicts – shows a list of all entities which are in a conflict with others.

Master planes – shows a list of all master planes which will be used for alignment. If you click any master plane or group of master planes you can see it displayed with a blue contour in the graphical window.

Log – shows various information, e.g. about limits, etc.

Actions preview – shows a list of all selected actions. If you click any action or group of actions you can see the results with a blue contour in the graphical window.



Live preview – if ON, you can see the results of selected options in the graphical window before you perform the function itself.

Master planes

UCS XY planes - if ON, entities are aligned to fit into XY planes of defined UCSs.

Storey planes – if ON, entities are aligned to fit into defined storey planes.

Section planes – if ON, entities are aligned to fit into defined section planes.

Line Grid planes – if ON, entities are aligned to fit into planes of defined line grids.

LCS planes (1D members) – if ON, entities are aligned to fit into XY or XZ planes of 1D members.

This option has to be ON if you want to use actions for 1D members.

Extend 1D member planes – LCS planes of 1D members are extended to infinity.

LCS planes (2D members) – if ON, entities are aligned to fit into XY plane of 2D members. This option has to be ON if you want to use actions for 2D members.

Extend 2D member planes – LCS planes of 2D members are extended to infinity.

Limits

Max. distance between parallel master planes [m] – defines the maximal distance of two parallel planes. If the distance of a node is smaller than this value, no new parallel plane is created in the node.

Max. angle between master planes [deg] - defines the maximal angle between master planes. If the angle between the existing plane and distance of a node is shorter than this value, then no new parallel plane is created in the node.

Correct angle of planes similar (within limit) to

GCS main planes – If ON, entities are aligned to fit into XY GCS parallel planes.

UCS XY planes - if ON, entities are aligned to fit into XY parallel planes of defined UCSs.

Actions (align nodes of)

This group of options specifies which structural type of members should be aligned with other structural type of member.

All 1D and 2D members to

All 1D member types – all 1D and 2D member are aligned to fit into LCS planes of 1D members.

All 2D member types - all 1D and 2D member are aligned to fit into LCS planes of 2D members.

Beams to

Columns – only all beams are aligned to fit into LCS planes of columns.

Walls – only all beams are aligned to fit into LCS planes of walls.

Plates – only all beams are aligned to fit into LCS planes of plates.

All 1D and 2D members – only all beams are aligned to fit into LCS planes of 1D or 2D member.

Columns to

Beams – only all columns are aligned to fit into LCS planes of beams.

Walls – only all columns are aligned to fit into LCS planes of walls.

Plates – only all columns are aligned to fit into LCS planes of plates.

All 1D and 2D members – only all columns are aligned to fit into LCS planes of 1D or 2D members.

Plates to

Plates – only all plates are aligned to fit into LCS planes of other plates.

Walls - only all plates are aligned to fit into LCS planes of walls.

All 1D and 2D members – only all plates are aligned to fit into LCS planes of 1D or 2D members.

Walls to

Beams – only all walls are aligned to fit into LCS planes of beams.

Walls - only all walls are aligned to fit into LCS planes of other walls.

All 1D and 2D members – only all walls are aligned to fit into LCS planes of 1D or 2D members.


Max. node-to-master plane distance [m] – defines maximal distance when a node is aligned into the master plane. If the distance is bigger the function is not performed.

Max. total displacement of node [m] – defines the maximal total displacement of a node when the node can be moved into a master plane. If the total displacement is bigger the node is not moved. This value must not be lower than the value specified above.

Display options

Highlight master planes – if ON, the contour representing the master plane is in bold.

Highlight nodes with plane – if ON, all nodes which are aligned to fit into a plane are displayed in bold in a colour which is set for the preview.

 **Note:** Live Preview must be enabled for highlighting of nodes.

Preview shows master planes – if ON, master planes are displayed also for Live Preview.

Preview shows nodes – if ON, nodes which are aligned to fit into a plane are also displayed in Live Preview.

Preview colour – specifies the colour of Live Preview.

Advanced

Parameterize the structure by master planes – this option is available only if at least one coordinate of a structure node is defined by means of a parameter.

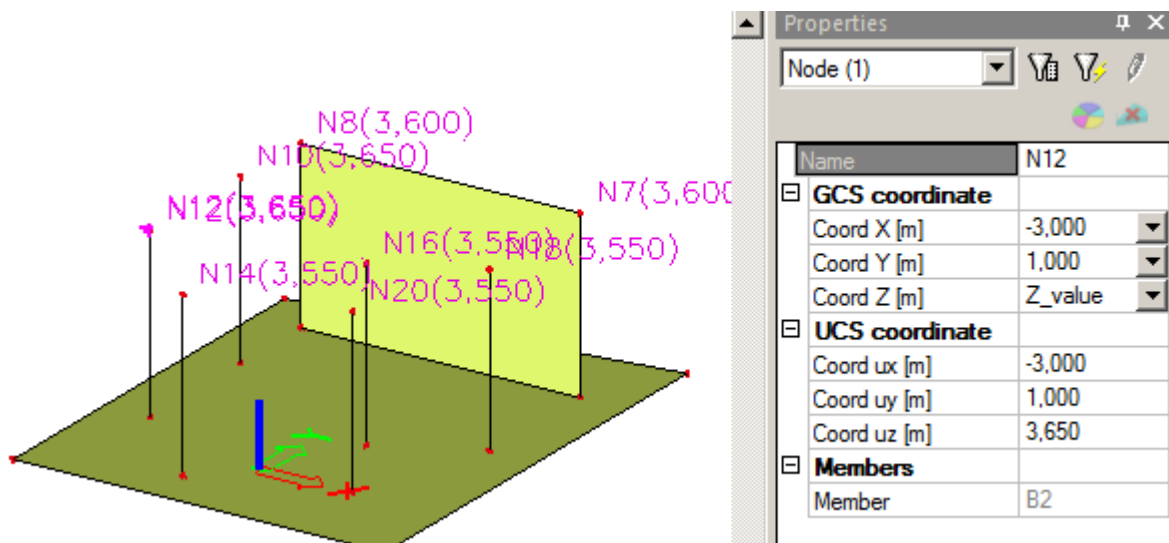
Keep original shape of the model (by eccentricities) – if ON, the original shape remains in the original location by means of a defined eccentricity.

Offset master planes – defines an offset for drawing between a master plane contour and edge of entity which defines the master planes.

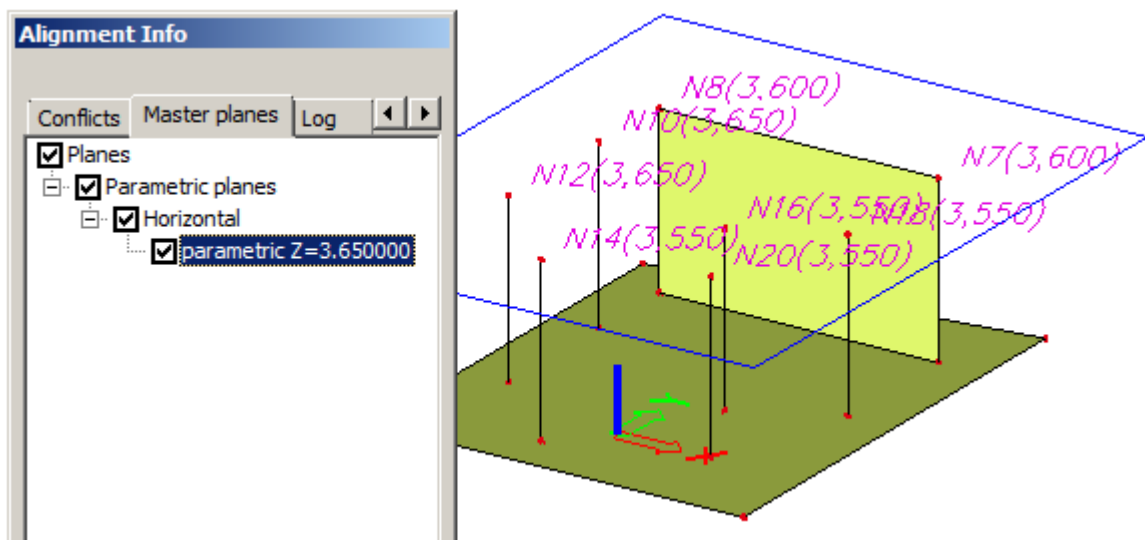
Planes of parametric input – this option is available only if at least one coordinate of a node is defined by means of a parameter. Then a plane which contains this node is determined as a master plane and other nodes are aligned to this plane.

Example 1

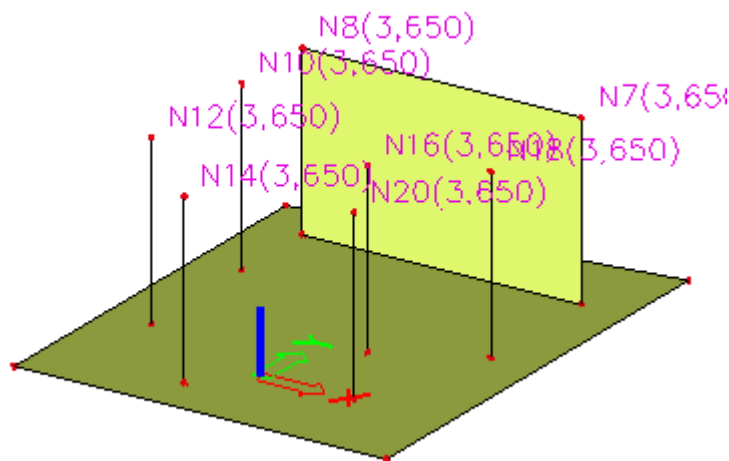
1. In the following example there is a node (N12) with Z coordinate defined by a parameter.



2. Select Planes of parametrical input as a master plane.



- All nodes are aligned into the plane with the Z coordinate 3.6m.



- If the option Parameterize the structure by master planes is selected, the Z – coordinate of all nodes in the plane has defined the proper parameter.

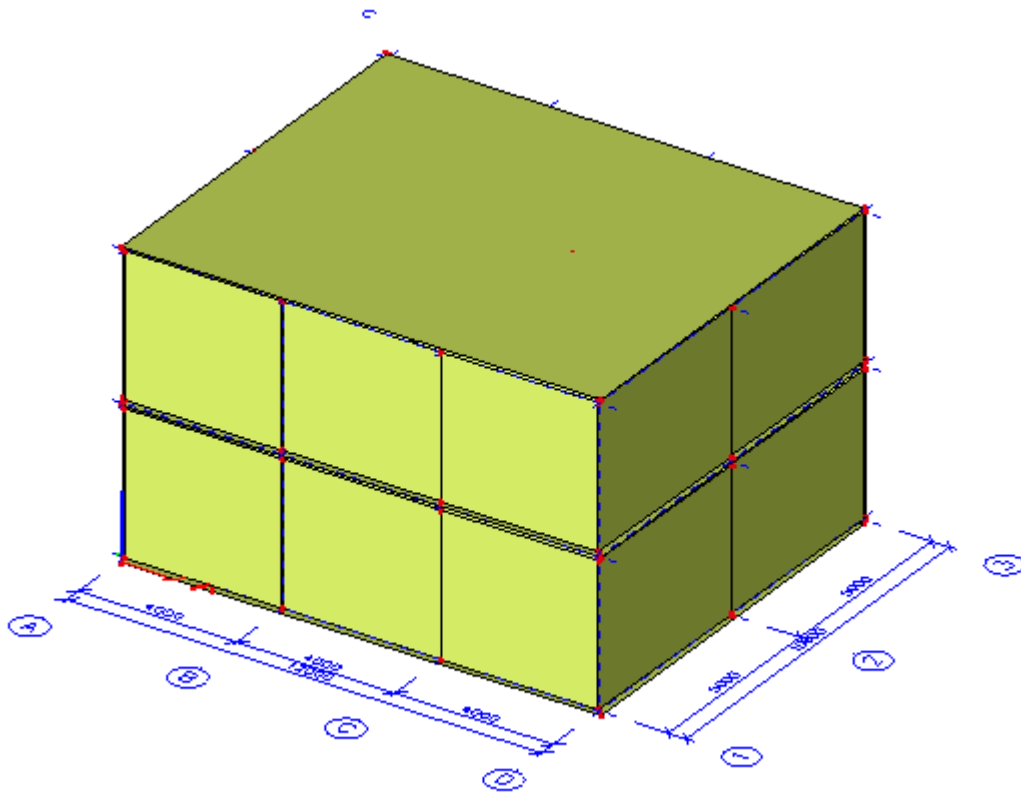
Properties

Node (1)

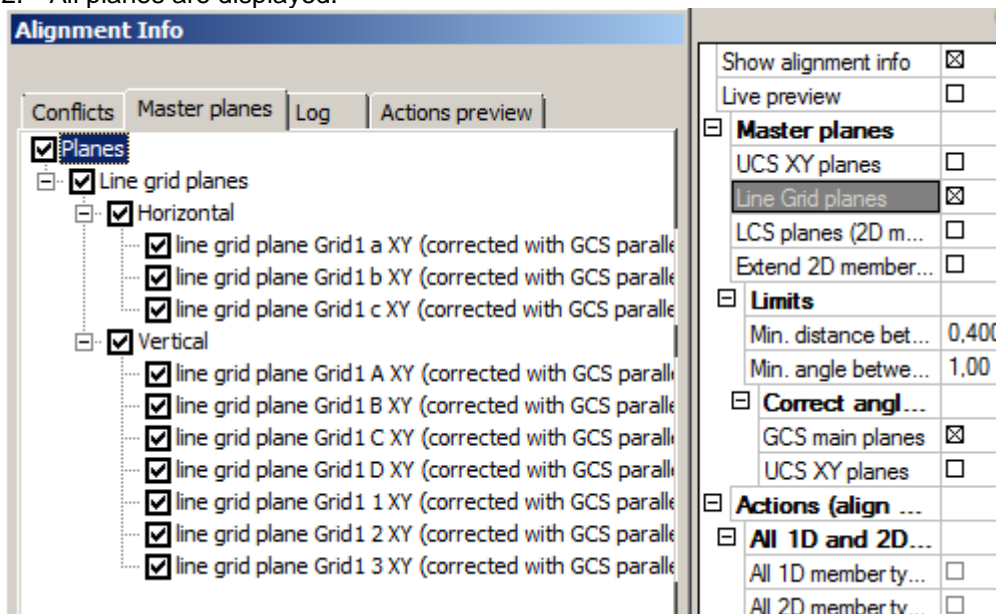
Name	N7
GCS coordin...	
Coord X [m]	3,000
Coord Y [m]	6,000
Coord Z [m]	Z_value
UCS coordin...	
Coord ux [m]	3,000
Coord uy [m]	6,000
Coord uz [m]	3,650
2D members	
2D member	S2

Example 2

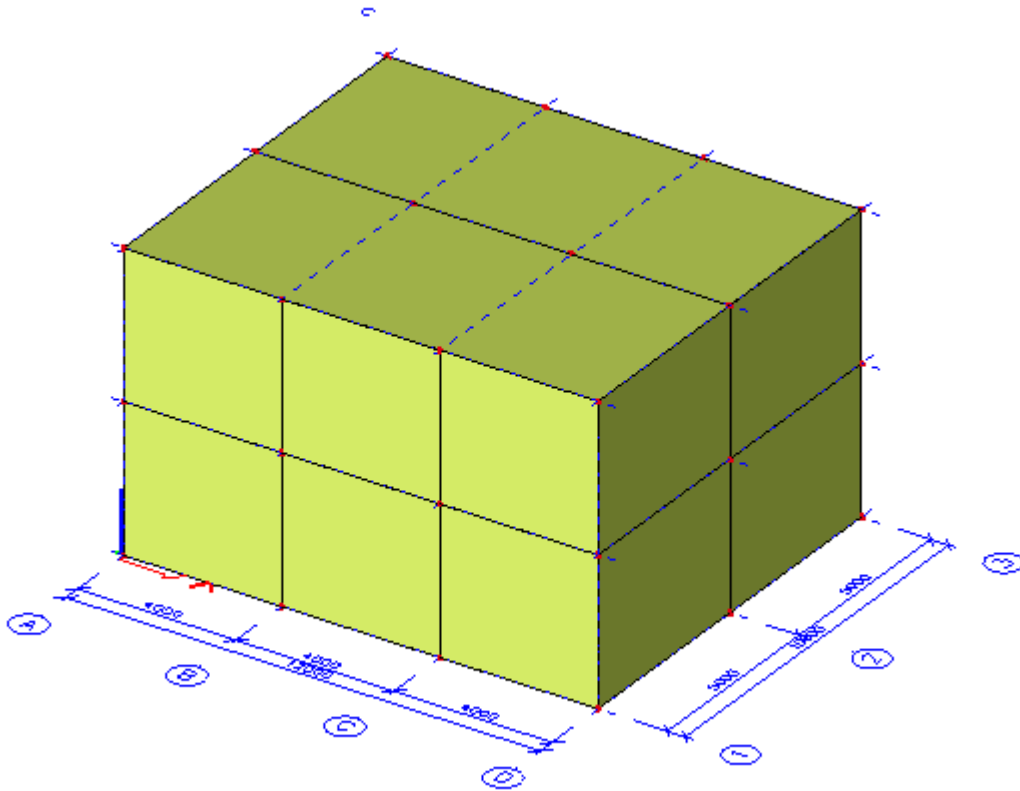
1. For a project where a line grid is inputted select option Line grid planes.



2. All planes are displayed.

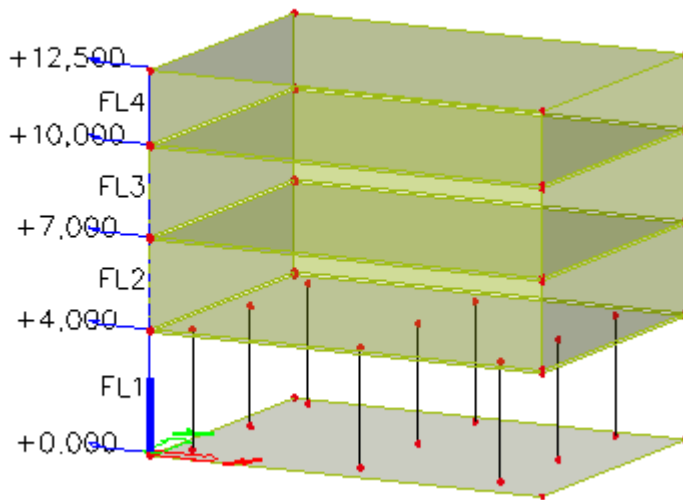


3. The structure is aligned.

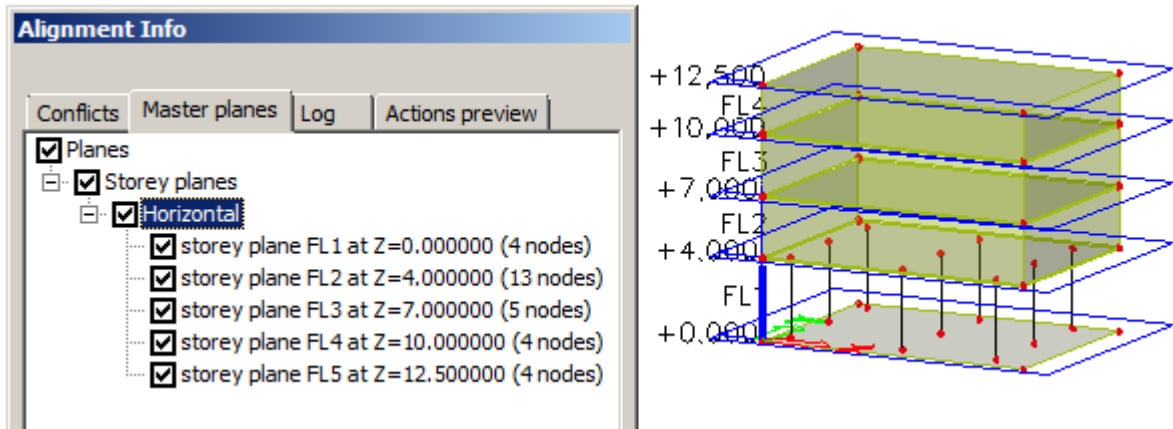


Example 3

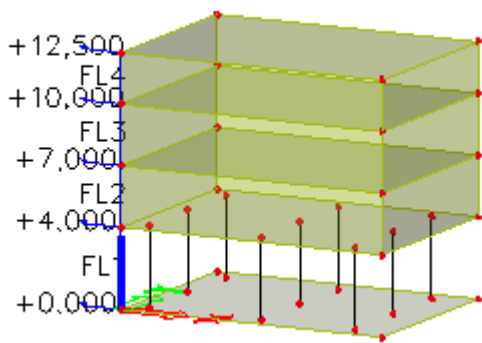
1. Storeys are defined in an imported project.



2. Select the option Storey planes.

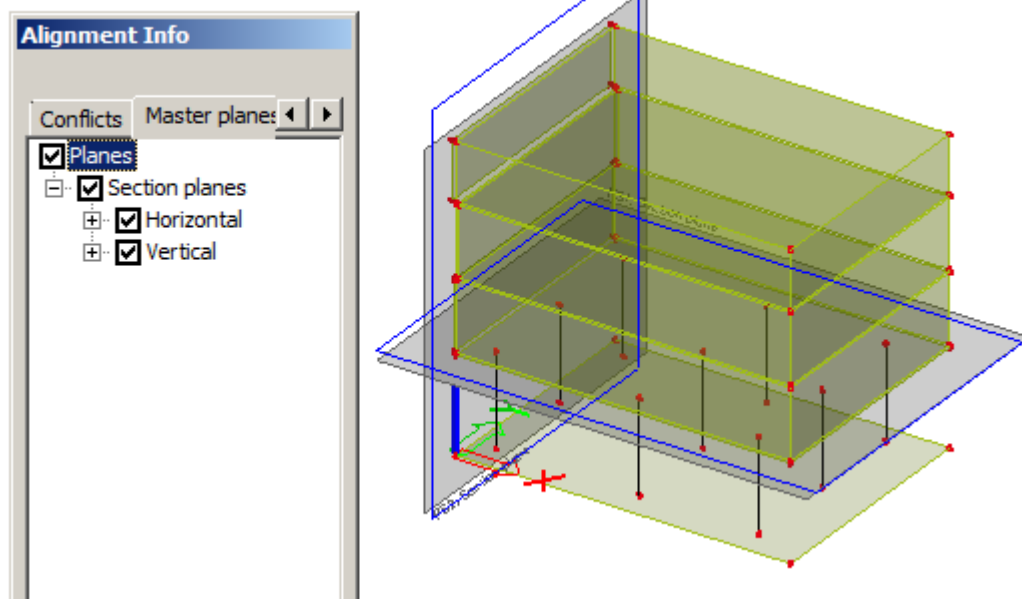


- The structure is aligned horizontally according to the defined storeys.

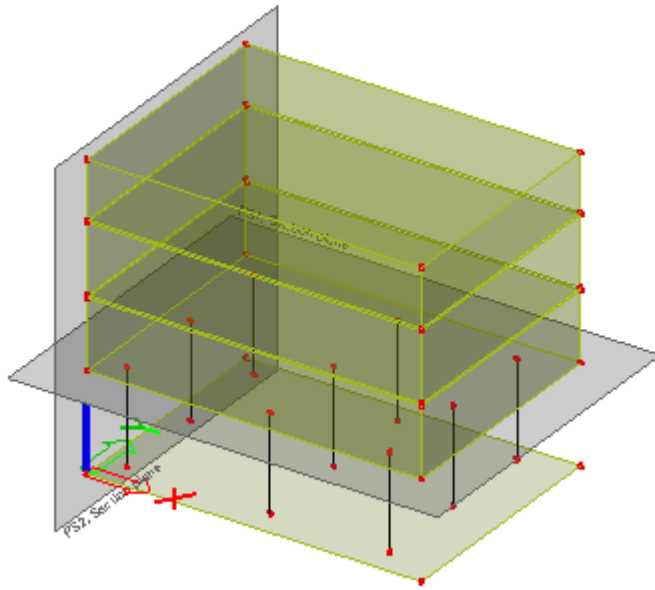


Example 4

- Section planes are defined in the project.
- Select Section planes as master planes.

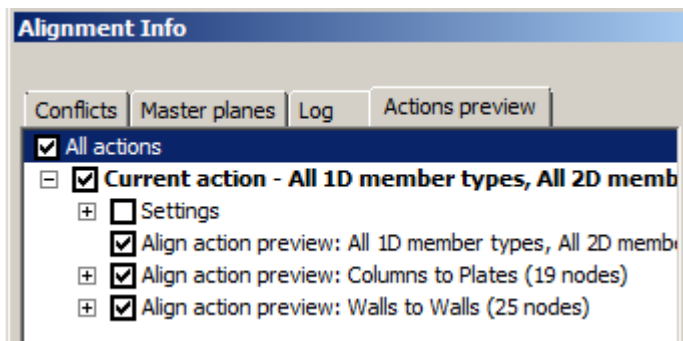


- Run Align function. Elements fitting into one of the planes are aligned and connected.

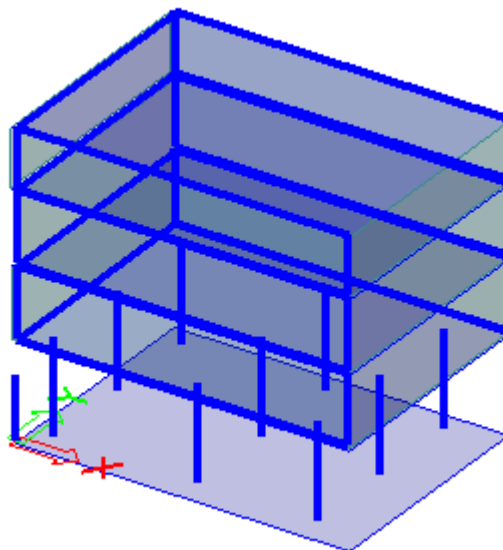


Example 5

1. Select LCS planes for both, 1D and 2D members.
2. Select some actions, e.g. Columns to plates and Walls to walls.



3. Select Live preview.



4. Confirm the action with [Run Align] action button.

- All walls are connected together and columns extended to plates. The rest of structure is not aligned.

