

BIM-Connector® FBC

Contents

Application options	2
System requirements	2
Application guide	3
Program structure	3
Physical modell	4
Analytical model	4
Symbols of the Ribbon Bar	5
Symbols in the graphics window	7
Program operation via shortcuts	8
Operation basics	9
IFC-import	9
SAF-import	9
Adjust the Physical Model	10
Colour Material type	10
Floor allocation	11
Model structure	11
Norms / Specifications	11
Non-relevant components – Load transfer	12
Assign material	13
Adapt the Analytical model	14
Calculated thickness	14
Loads	14
Clean up geometry	15
Straighten	16
Join of component axes / component areas	17
Move members	21
Disassemble walls	22
Conversion	23
Check model	23
Export to the design programs PLT, B5 +, GEO	24
Selection of objects/components with the mouse	25
FAQ	26

Basic Documentation – Overview

In addition to the individual program manuals, you will find basic explanations on the operation of the programs on our homepage www.frilo.com in the Campus-download-section.

Application options

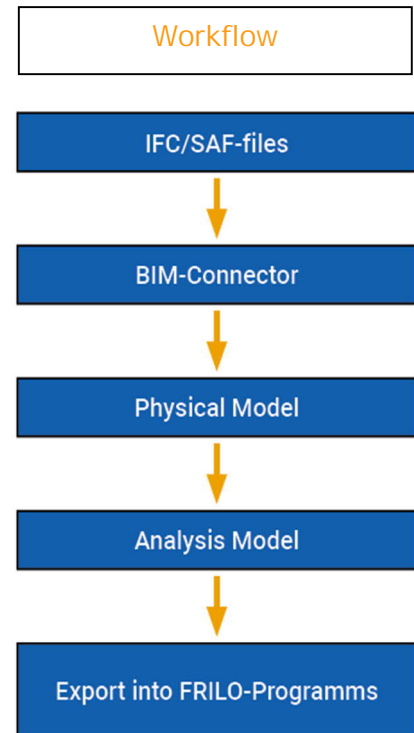
Structural engineers can use the BIM-Connector® with the FRILO programs to participate in an OpenBIM process. The BIM-Connector® enables the collaboration between architects and structural engineers, as well as structural engineers among themselves on a model basis. Any CAD models can be imported into the BIM-Connector® in the form of an IFC file (Industry Foundation Classes - IFC 2x3 and IFC4). The Physical Model can then be converted into an Analytical Model, which can then be transferred to the FRILO programs. It is also possible to read in SAF files (Structural Analysis Format) in the BIM-Connector®.

Of the many Model View Definitions from the IFC standard, the BIM-Connector® primarily processes the Reference View and the Design Transfer View (= volume representation). The Structural Analysis View (= Analytical Model in IFC format) is not processed because it is less developed than the SAF format. A Model View Definition (= MVD) is essentially a filtered view of the IFC in order to provide the project participants with the information relevant to them.

The BIM-Connector® was developed with the usual multi-storey building in solid construction in mind. Framework models will only be considered at a later point in time.

Your added value:

- Saving of time through direct import of IFC and SAF files
- Thus, more time for the actual structural planning
- Data loss is prevented by direct import
- Manual transmission errors are reduced
- Creation of high quality and precise analytical models
- Direct availability of all relevant data
- Optimized workflow between project participants.
- The export to the design programs PLT Slabs by Finite Elements, B5+ Reinforced Concrete Column and GEO Building Model (GEO from release 2022-1) are implemented.



System requirements

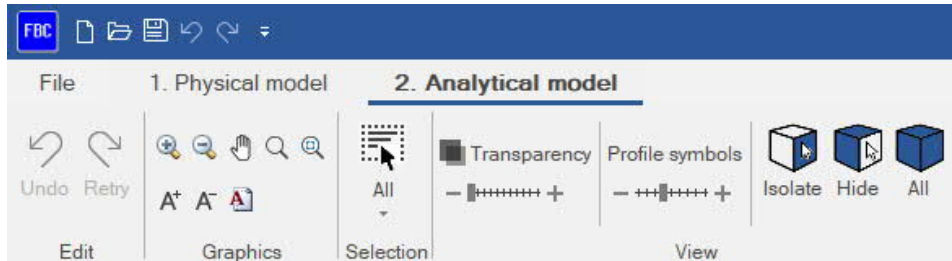
The BIM-Connector® can run on normal work computers. We recommend these system requirements for optimal use of the program:

Project size	Small and medium-sized models (e.g. single-family houses)	Big models (e.g. office complexes)
CPU	Intel Core i5, comparable AMD	Intel Core i7, comparable AMD
Random access memory	8 GB RAM	16 GB RAM
Hard disk	SSD	SSD
Graphic card	2 GB VRAM OpenGL compatible graphic card	4 GB VRAM OpenGL compatible graphic card
Screen resolution	HD/Full HD (1920x1080)	HD/ Full HD (1920x1080)

Application guide

Program structure

The menu items "File", "1. Physical model" and "2. Analytical model" are arranged in the ribbon.



File menu

Start with a new item:

Via File ▶ New (from IFC/SAF) → a new item based on a model link from an IFC or SAF file can be opened.

Start with an already saved position:

Via File ▶ Open an existing item can be reopened with all changes that have already been saved.

Saving an item:

Via File ▶ Save As you can save an item in the [FRILO Control Center FCC](#).

Note: From Release 2022-1, the BIM connector items are integrated into the FRILO Control Center. The associated IFC / SAF file is attached to this item as a zipped file and is thus integrated in the FRILO file management.

Menu items Physical model / Analytical model

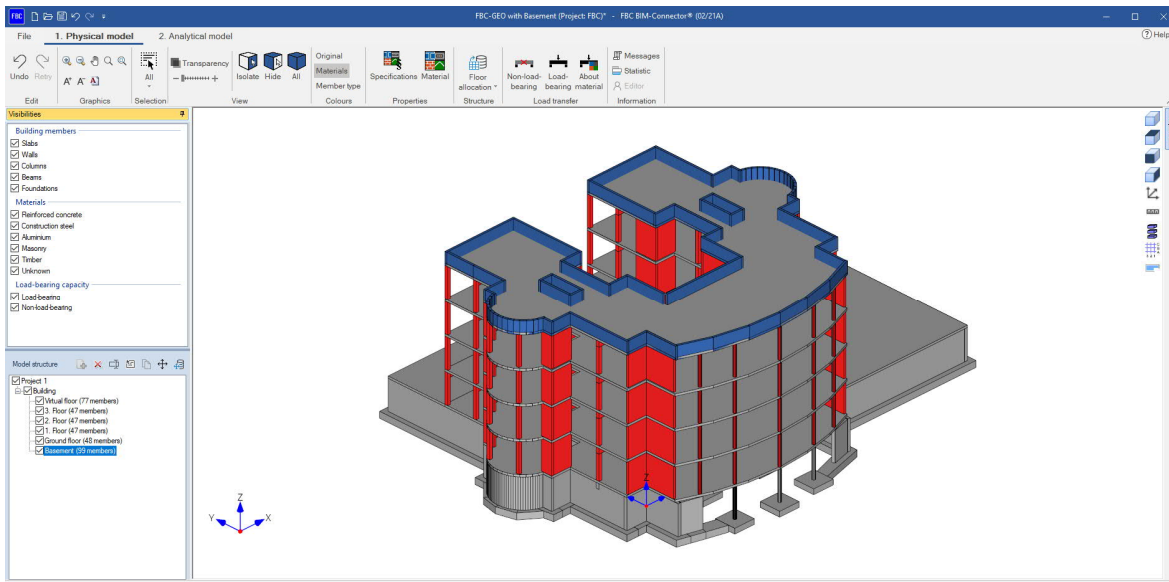
A click on "1. Physical model" or "2. Analytical model" shows the associated command buttons/icons of the so-called "[Ribbon Bar](#)". The program can be operated both via the command button and via the context-sensitive graphic (right mouse button).

Physical model

The physical model is a model that consists of volume elements. This can contain both load-bearing and non-load-bearing components.

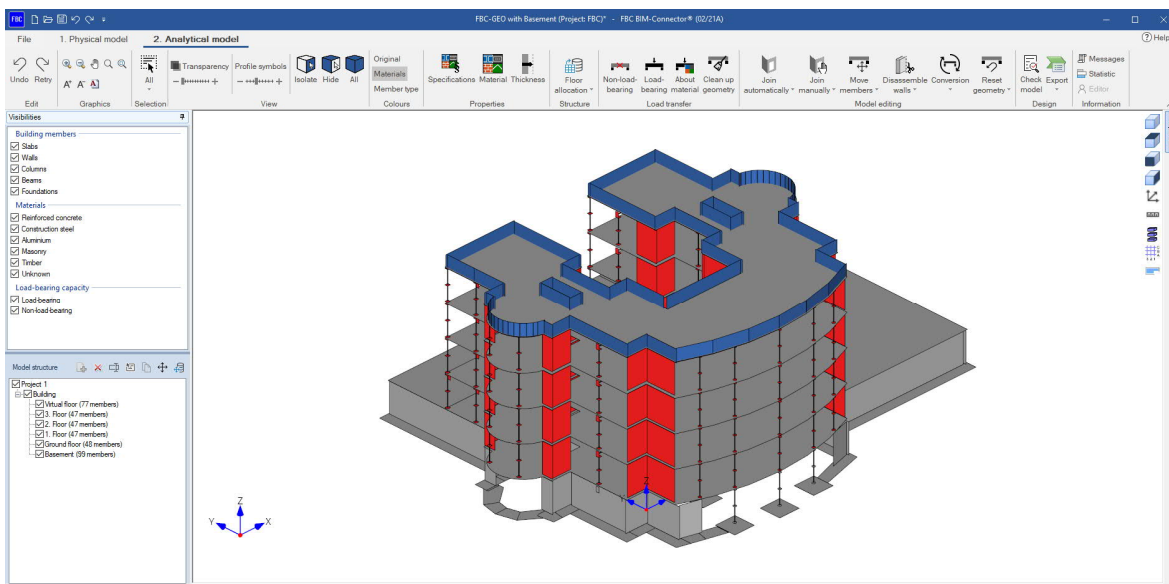
The physical model is opened / imported as an * .ifc file. This model is usually provided by the architect and usually has to be adjusted (remove objects that are not relevant for the statics, etc.)

The following figure shows a physical model.




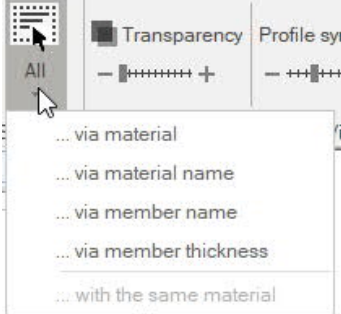










Analytical model













The Analytical model, on the other hand, contains all the components required for the static calculation. Thus, only the load-bearing components are displayed in the Analytical model. The model now only consists of bar and surface elements. The volume components were reduced to their component axes. The following figure shows an analytical model.




Symbols of the Ribbon Bar

The individual program icons are explained below:

 All	<p>Single or multiple components can be selected using the material, the material name, the component name or the component thickness.</p> <p>Click the upper half of the icon to select all objects. Click on the lower half to expand the submenu items. After selecting a submenu item, a dialog with further options is displayed.</p>	
 Isolate	<p>Reduces the display to the selected component.</p>	
 Hide	<p>Removes the selected components from the display.</p>	
 All	<p>Cancel all visibility settings again.</p>	
 Colours	<p>Colours the model according to the material type, material grade, component type, cross-sections and load.</p>	
 Norms	<p>Opens a dialog for selecting design standards and materials as default standards.</p>	
 Material	<p>Assigns a new material to all selected components.</p>	
 Thickness	<p>Assigns a new calculated thickness to all selected surface components (walls, ceiling slabs).</p>	
 Loads	<p>Assigns a constant load to all selected components (walls/ceilings).</p>	
 Floor allocation	<p>Allocates all selected components to a target storey or rearranges the entire building.</p>	
 Non-load-bearing	<p>Marks all selected components as non-load-bearing. Non-load-bearing components do not appear in the Analytical model.</p>	

 Delete	Deletes the marked components permanently from the model.
 Load-bearing	Marks all selected components as load-bearing .
 About material	Opens a dialog in which the load-transferring components can be specified using a material selection. In addition, if the "Component thickness" option is selected (check mark), components can be marked as load-bearing or non-load-bearing according to their component thickness.
 Clean up geometry	Removes negligible short supports and beams, as well as small openings and small slabs, from the model.
 Straighten automatically	<ul style="list-style-type: none"> ▪ Without options: projects all visible or selected slabs and beams up to an inclination of 2° in the XY plane. ▪ With options: projects all visible or selected slabs and beams in the XY plane, whereby components with a larger angle of inclination can also be taken into account. ▪ Manually: straightens the first selected slab/beam to the second selected slab.
 Join automatically	<ul style="list-style-type: none"> ▪ Without options: joins all visible or selected components with one another so that the component edges lie on top of one another. ▪ With options: Automatically joins all selected components, including neighbouring panels that are to be viewed statically as one panel.
 Join manually	<ul style="list-style-type: none"> ▪ Without options: extends the first selected part to the second selected part. a) surface component with surface component, b) line component with surface component and c) line component with line component. ▪ With options: joins selected components, including neighboring panels, which are to be viewed statically as one panel.
 Move members ▾	Moves the first selected component in the axis of the second selected component or aligns a member axis to an edge of a slab.
 Disassemble walls ▾	Decomposes all selected walls into strips or individual components. As a result, openings in walls can be taken into account when modeling the Analytical model. Furthermore, walls can be converted into beam elements.
 Conversion ▾	Converts walls into beams, walls into columns, or beams into walls.
	Restores the original wall geometry including all openings.
 Check model	Checks all or the selected components of the analytical model with regard to unfavorable component orientation.

 Design / Export	Exports the selected component to the assigned (selectable) FRILO dimensioning program.
Help	Here you can call up the help file or various workflow papers. You can also submit a support request if you have any questions about the product.

Symbols in the graphics window

- Isometric view [CTRL] + [NUM 1]
- Top view [CTRL] + [NUM 5]
- Front view [CTRL] + [NUM 2]
- View from the side [CTRL] + [NUM 6]
- Measures the distance between two corner points of a component.
- Shows/hides the coordinate system.
- Shows/hides the main dimensions of the building.
- Shows the building in an exploded view.
- Shows/hides the axes/design grid if they are included in the IFC/SAF file.
- Shows/hides the thickness range.
- Highlights the edges when selecting objects.
- Activates/deactivates the section plane function.



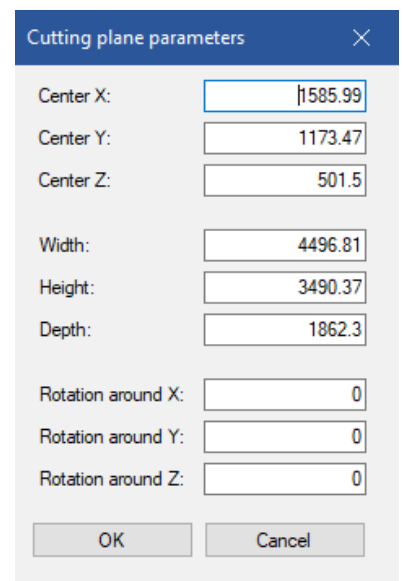
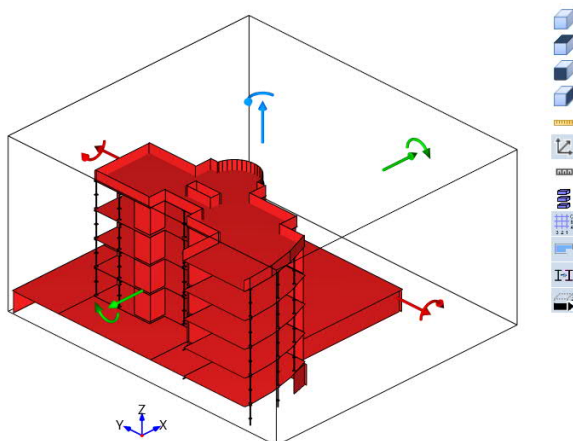
Section plane function

A click on the section plane symbol displays a rectangular editing box around the building. To switch to editing mode, one of the lines in the box must be clicked. This causes different arrows to appear on all sides of the cube. By moving the arrows in the plane of the cube or by rotating them around the direction of the arrows, a part of the model can be cut off.

This allows a view into the interior of the building, e.g. to visually check whether components are on top of each other.

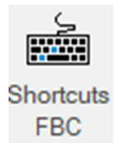
When the editing mode is activated, the other editing functions are deactivated. By double-clicking on the cube, a settings dialog for the section plane parameters can be opened.
















Clicking outside the cube exits the editing mode. Now you can continue editing the model.



Program operation via shortcuts

In order to be able to work even faster and more efficiently, "Shortcuts" (key combinations for quick access) were assigned to the most frequently used functions with the R-2023-1 release. A list of all saved shortcuts can be called up via the "Help" menu item and the "Shortcuts" button.



Keyboard shortcuts		
	Function	Keyboard shortcut
	FBC manual	[F1]
	TeamViewer	[F8]
	Abort a function	[ESC]
	Selection of components that are in the background	[CTRL] + [MOUSE WHEEL] scroll
	View: zoom	[MOUSE WHEEL] Scroll
	View: entire graphic	Double click in empty area
	View: move	Drag the mouse while holding down [MOUSE WHEEL].
	View: isometric	[STRG] + [NUM 1]
	View: top view	[STRG] + [NUM 5]
	View: from the front	[STRG] + [NUM 2]
	View: from the right	[STRG] + [NUM 6]
	Function: delete	[CTRL] + [DELETE]
	Function: non-load-bearing	[DELETE]
	Function: straighten	[S]
	Function: join (automatically)	[J]
	Function: join (manually)	[CTRL] + [J]
	Function: move (component to component)	[M]
	Function: disassemble walls (into wall strips)	[W]

Operation basics

In order to achieve good results, some basic working methods are shown below.

The program interface reflects the way the program works. First, an IFC/SAF file is read into the BIM-Connector®. The structure can then be viewed in the BIM-Connector® and edited accordingly. It can then be transferred to a program.

IFC-import

The basis for working with the BIM-Connector® is a well-structured architectural model that has already been reduced to the shell. A shell model is generated from the architectural model, in which the terrain has already been hidden and, ideally, windows and doors as well as expansion elements or insulation are no longer available. We therefore recommend that you give the architect a corresponding hint, because the quality of the Analytical model depends very much on the IFC file on which it is based.

Notes on IFC models

The best results are obtained when the model:

- only includes the shell (e.g. IFC Design Transfer View, IFC Reference View)
- is well structured
- Applicable component types are used (IfcWall, IfcBeam, IfcColumn, IfcSlab, IfcFooting, IfcRoof, IfcMember)
- A material is assigned to the components (at least the material name)

- 1.) Select File → New (from IFC/SAF)
- 2.) The Physical model of the imported IFC file of the architect opens

SAF-import

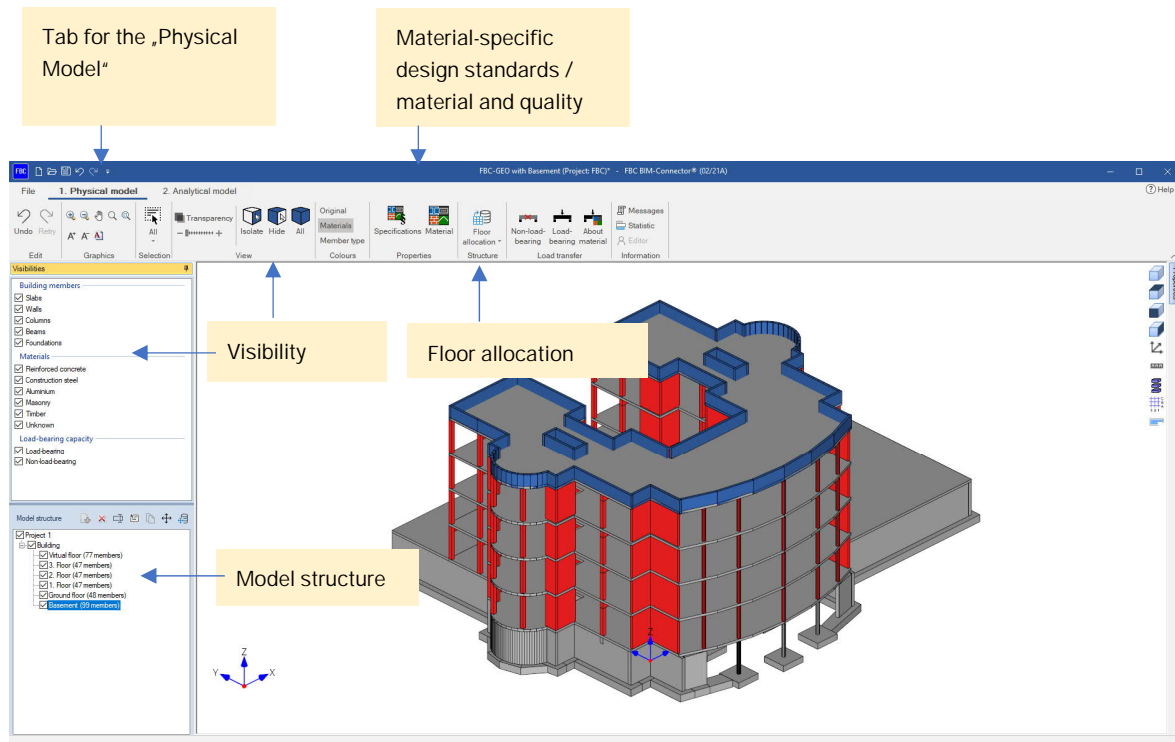
SAF is an Excel-based file format that was developed to improve collaboration between structural engineers. SAF files are used to exchange data between the structural analysis software from different manufacturers. The following manufacturers have already implemented the SAF format or are in the process of implementing it:

FRILO, SCIA, Graphisoft, Allplan (BIMPlus), RISA, Strusoft, Axis VM, Dlubal, Sofistik, SCAD, LIRA land and Radimpex. More detailed information must be obtained from the relevant software manufacturer.

SAF files transfer ready-made analytical models. The components are imported as bar and surface elements and the component axes are already connected to one another.

- 1.) Select File → New (from IFC / SAF)
- 2.) The Analytical model of the imported SAF file opens

Adjust the Physical Model



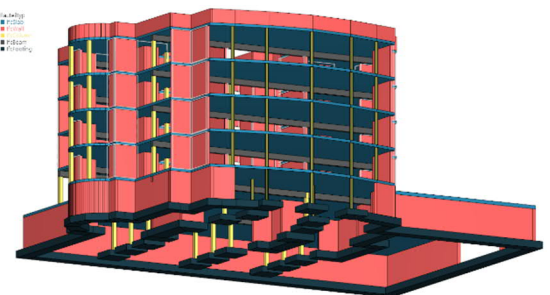
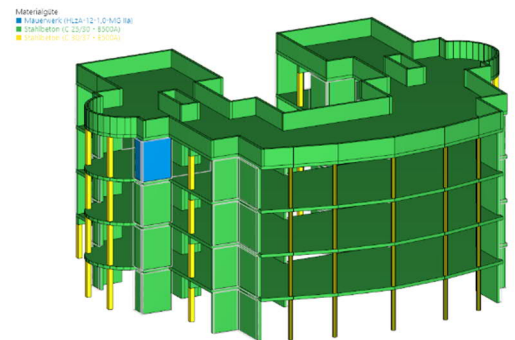
Colour Material type

Via the ribbon group "Colour" the model can be coloured according to the material type, the material grade, the component type, the cross-sections and the load.

The colour display according to the material quality makes it easy to check whether all components have been assigned a material quality and whether this corresponds to your own specifications.

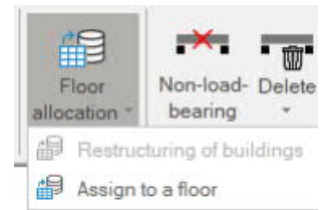
The colour representation according to the component type makes it easy to check whether the correct IFC component type has been assigned to the components. The BIM Connector currently supports the following IFC component types:

- IfcWall
- IfcBeam
- IfcColumn
- IfcSlab
- IfcFooting
- IfcRoof
- IfcMember



Floor allocation

The floors for calculation can be automatically restructured using the "Floor allocation" function. From the previous view from "top to bottom", as is usual for architects, the view from "bottom to top", which is usual for structural engineers, can be generated.



Note: *The Model structure can be generated automatically by the program using the "Floor allocation" function. It is advisable to use the Floor allocation first and then to manually adapt the Model structure generated afterwards. Individual components can be moved to another floor using the "Floor allocation" function or with a right-click "Assign to another floor". A clearly structured model achieves better results with the "Join automatically" function.*

Model structure








The building can be structured in a meaningful way using the Model structure. Buildings and floors can be added, deleted and renamed. Components can also be moved to other floors or entire floors to another building.

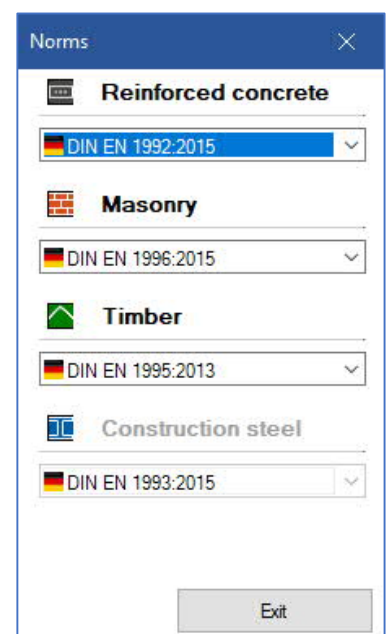
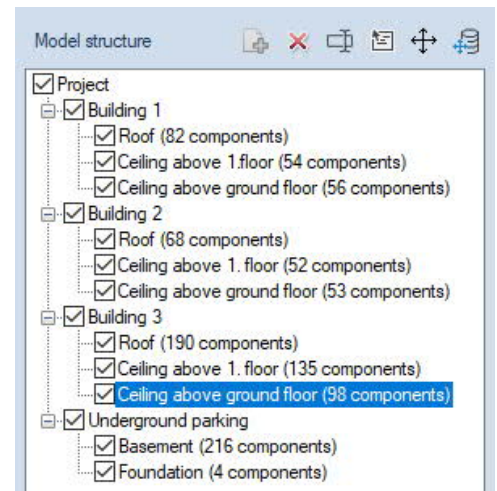
Add a new structure:

To do this, you first have to click on the "Construction project" (first entry in the structure tree), then the +icon for a new building is activated.

Add a new floor:

To do this, you first have to click on the "Building", then the icon for a new floor becomes active.

-  Add a new building/floor
-  Delete building/floor
-  Rename project/building /floor
-  Floor data dialog
-  Copy building (with the same structure)
-  Moves the floor to another building
-  Moves components to another floor



See also [FRILO-Insight-2021-2, Restructuring buildings.](#)

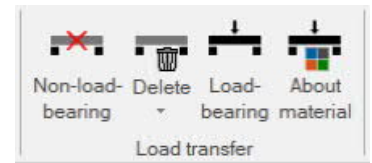
Norms / Specifications


The corresponding design standards can be specified using the "Norms" function.

Non-relevant components – Load transfer

After the IFC/SAF file has been imported, the Physical model should be cleaned of all components that are not relevant for the calculation. The quickest way to do this is to use the "Load transfer > About material" function in the "Load transfer" function group. Here, all components relevant for load transfer can be set to "load-transferring" and all components that are not relevant for load transfer to "non-load transferring".

The checkmark for the component thickness is set by default. Components with a small component thickness can be identified quickly and set to "non-load-bearing".

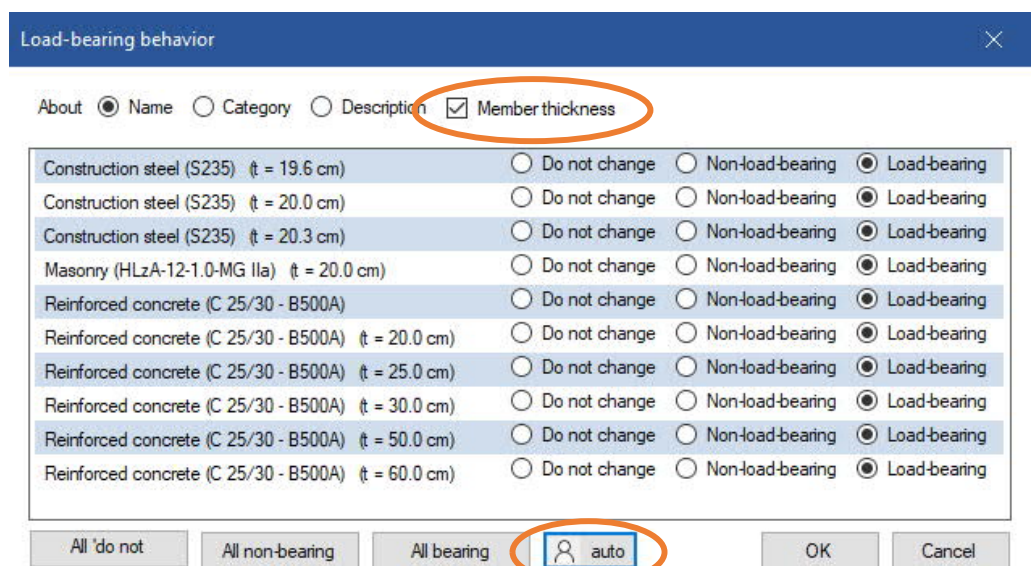


Click on the symbol  in the lower area to let the program automatically determine the load-bearing capacity and the component thickness via the material name.

Note:


In the building model (GEO), a strip or single foundation is automatically applied under the last load-bearing component. This means that strip and single foundations contained in the IFC file must be used as non-load-bearing components.

All previously marked objects can be set to non-load-bearing / load-bearing using the "non-load-bearing" or "load-bearing" icons.



The "Delete" icon can be used to permanently remove all selected objects from the model. It is planned to use this function to remove objects such as wall layers, floor structure, etc. from IFC files that have not been cleanly reduced to the shell.

Note:

By deleting components, the performance of the programme can be significantly improved. In particular, the loading of items is considerably accelerated. Deleted elements can be restored via the corresponding button , but this requires saving and re-loading the item. Furthermore, the elements are read in in their original form.

Note on IFC files:

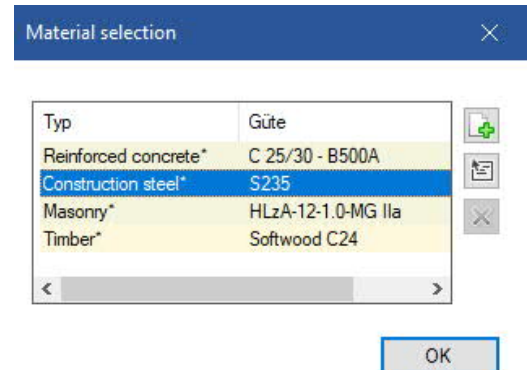
The load-bearing capacity can also already be read out of the IFC file. For this, the attribute "static load-bearing" must be assigned to the objects in the corresponding CAD program.

Assign material

The next step is to assign the correct materials and their material grades to the components. With the help of the various [selection and display options](#) (visibility, model structure, etc.), for example, all supports can be selected in next to no time.

With the help of the "Material" function (Properties), all selected columns can be assigned the appropriate material with one click.

Behind the button for the material you will find a list of all materials created for this item. The materials highlighted in yellow and marked with an asterisk are the standard materials. The standard material grade is used for all components that have not been explicitly assigned a different material grade. The standard materials cannot be deleted.



Adds a new material/material grade.



Opens a dialog for changing the selected material/material grade.



Removes the selected material/material grade.

If a material grade has already been assigned to a component and the dialog is subsequently opened again, the assigned material grade is displayed selected in blue.

Note on IFC files:

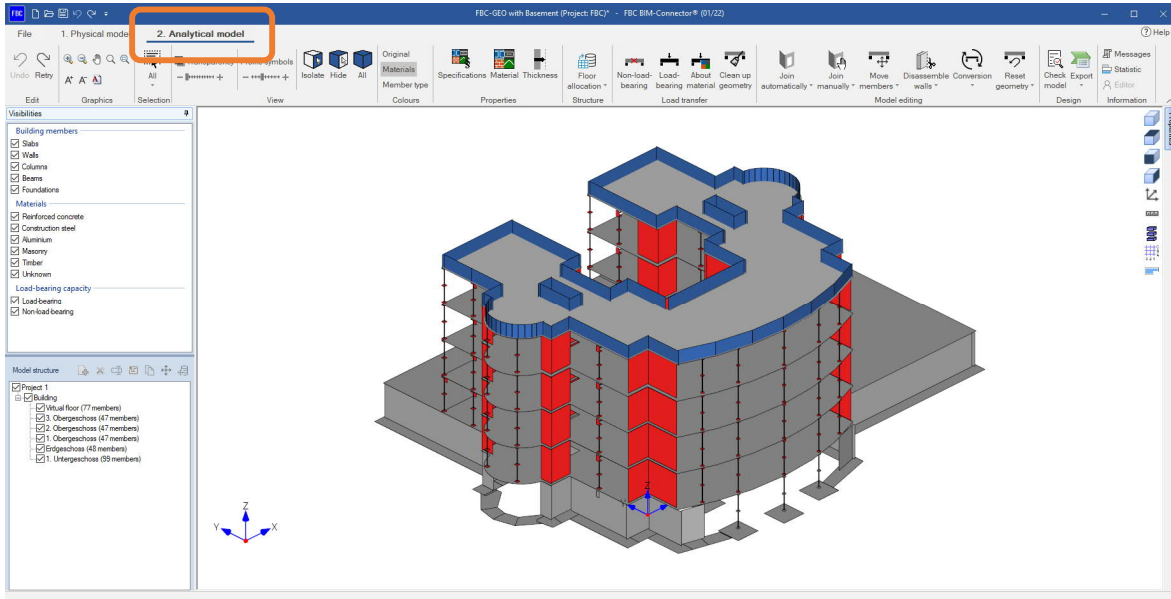
The materials and material grades can already be read out of the IFC file. For this, one of the material designations listed in the following table must be stored in the corresponding CAD programme in the field for the material (see also workflow paper).

Material	Name component	Specific material designation
Reinforced Concrete	„concrete“	Naming scheme: Cxx/xx bzw. LCxx/xx e.g. C25/30 or LC20/22 (all common material grades)
Masonry	„masonry“ „brick“ „blockwork“ „clinker“ „aerated concrete“ „poroton“ „bisotherm“	
Steel	„steel“	Naming scheme: Sxxx e.g.. S235 (all common material grades)
Timber	„timber“ „glulam“ „osb“	
(Aluminium)	„aluminium“	non

If the correct materials have been assigned to all components and the model has been cleared of the non-load-bearing components, you can switch to the Analytical model. Only the load-bearing components are shown there.

Adapt the Analytical model

Via the tab "2. Analytical model ", the model is finally prepared for the component design.



Calculated thickness

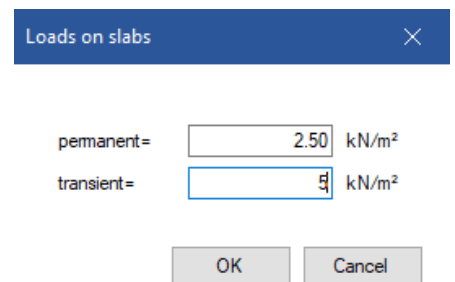
Using the "Thickness" function, surface elements (walls, floors) can be assigned a new calculated thickness.

Loads

To assign a load, the component must first be selected. Then click on the "Load" function.

Assign area loads to a floor slab:

A constant permanent and variable load can be assigned to floor slabs via the "Loads" function.

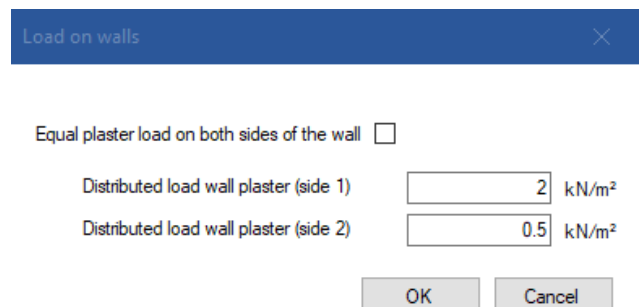


Assigning area loads for walls:

By selecting one or more walls, a surface load for the wall plaster (on both sides) can be added to them. This function can also be found in the GEO and PLT under the "wall properties".

If the option "Equal plaster load on both sides of the wall" is activated, the entered load value always refers to the inside and outside of the wall surface. I.e. the entered load value is multiplied in the GEO by the total wall area (inside and outside) and added to the dead weight of the wall.

For this reason, half of the load, i.e. the averaged value, must be entered in the input field.



Example for wall loads:

Plaster load on both sides:

Outside 3 kN/m²

Inside 1.5 kN/m²

$(3 \text{ kN/m}^2 + 1,5 \text{ kN/m}^2) / 2 \text{ wall sides} = 2,25 \text{ kN/m}^2 \text{ per wall side (input value)}$

Plaster load on one side: Outside plaster 3 kN/m²
 3 kN/m² / 2 wall sides = 1.5k N/m² per wall side (input value)

If there are different load values per side, it is advisable to enter the load value separately. The option can be deactivated for this purpose. Two input fields then appear. One for the inside and one for the outside of the wall.

Clean up geometry

The "Clean up geometry" function can be used to clean up negligible small openings, short columns, beams and small slabs.

The function searches for the components based on the dimensions (length / width / height) or cross-sectional areas.

The search can be started with the "Start" button. The following dialog then appears:

Problem area	Member types	Member names	Member thicknesses	Floor	Description
Component irrelevant	Wall	Wandschicht: 3.1.1-DG	6.0 cm	DG	Component fulfills conditions for neglect.
Component irrelevant	Wall	Wandschicht: 3.1.3-DG	10.5 cm	DG	Component fulfills conditions for neglect.
Component irrelevant	Wall	Wandschicht: 3.1.4-DG	11.5 cm	DG	Component fulfills conditions for neglect.
Component irrelevant	Wall	Wandschicht: 3.1.2-DG	20.0 cm	DG	Component fulfills conditions for neglect.
Component irrelevant	Wall	Wandschicht: 6.1.3-DG	10.5 cm	DG	Component fulfills conditions for neglect.
Component irrelevant	Wall	Wandschicht: 6.1.1-DG	4.0 cm	DG	Component fulfills conditions for neglect.
Component irrelevant	Wall	Wandschicht: 7.1.1-DG	8.0 cm	DG	Component fulfills conditions for neglect.
Component irrelevant	Wall	Wandschicht: 6.1.4-DG	11.5 cm	DG	Component fulfills conditions for neglect.

The components that meet the criteria for the entered dimensions are displayed and can be removed by selecting the corresponding line and pressing the "Clean up" button.

Note: Via Ctrl. + A all lines can be selected at once. It is advisable to clean up the model of irrelevant elements first and only then to use the Join automatically/manually function.

Straighten

Diagonal beams and slabs could not be transferred to the GEO building model up to R-2022-2. For this reason, a function was developed to project components (beams and floor slabs) into the horizontal plane (xy plane).

The "Straighten" function is divided into the areas "Automatic straightening", "Straightening with options" and "Straightening manually".



Straighten automatically

If you click on the upper part of the button, "automatic straightening" takes place. If no component is selected, all beams and slabs are projected into the xy plane up to an angle of inclination of 2°.

If beams and slabs are explicitly selected, then these components are always projected into the xy plane, regardless of the angle of inclination.

The pivot point depends on the nearby slabs. If the algorithm finds an adjacent slab or, in the case of a beam, a slab above or below it, the point closest to a slab is always searched for and used as the pivot point. If no slab is found, the highest point of the component to be straightened is automatically used as the pivot point.

Straighten with options

Under "Straightening with options" the inclination angle up to which the components are to be straightened can be adjusted manually for more inclined beams and slabs.

Straighten manually

The slab/beam selected first can be straightened to the second selected slab using the "Straighten manually" function.

Join of component axes / component areas

After deriving the Physical model for the Analytical model, the components are not yet joined together. This is due to the fact that the components are reduced to their component surfaces/axes, which means that half the component thickness is always lost. The "Join automatically" function in the "Model editing" group is suitable for establishing a connection between the components. Depending on the size and complexity of the building, it is also advisable to use the "Join" function only for the floor under consideration, as a great deal of data is processed with this function.

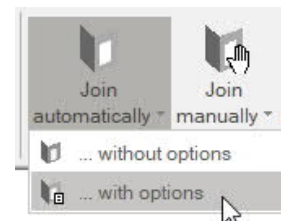
Note: The "Join" function achieves better results with a clearly structured building that has been cleared of non-load-bearing and irrelevant elements.

During the joining, all selected/visible components are combined in pairs.

No components are moved in the process. The components are only lengthened in their area and axis until they meet another component that is in the vicinity. The plate contour is aligned with the walls and beams.

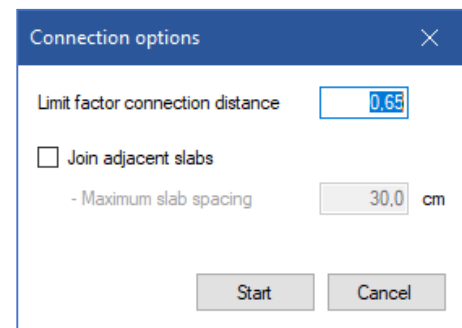
If the components are not joined as desired, a user-defined value can be entered for the "Limit factor connection distance" which you will find when "Join automatically with options".

The elements merged into a pair are only connected if their distance is small and the user-defined distance limit is not exceeded. It is advisable to set the limit factor greater than 0.50. If the value of the limit factor is too high, if there are many components to be connected, this can lead to the undesired effect that elements that do not belong together are blended.



Joining automatically with options / adjacent slabs

If two adjacent slabs are to be used statically as a common slab, the "Join automatically with options" function is available. The option "Join adjacent slabs" can be selected here so that they are also automatically joined. It is necessary to specify a maximum horizontal slab distance at which the slabs can still be joined. The distance between the slabs must be entered in centimetres. This function only applies to slabs that lie in the same horizontal plane. For slabs with different slab thicknesses, thickness ranges are automatically created when joining.



Note: If not all components were connected to one another with "join automatically", it is advisable to select the components that are not connected and to connect the components using the "Join automatically - with options" function. The limit factor can be increased accordingly.

Joining manually

Manual post-processing is sometimes necessary when joining the components. The "Join manually" function is available for this. With this function, the first selected component is always joined to the second selected component or the first selected components to the last selected component. In contrast to the "Join automatically" function, no limit distance is used for "Join manually". This means that components can also be connected which are far apart and which were not recorded by the "Join automatically" option. In contrast to the "Join automatically" function, the merging of adjacent slabs, which are vertically offset to one another, is possible with the "Join manually" function.

This function can also be used to influence the order in which the components are to be connected to one another.

Note: Several components can be selected using the "Ctrl" key.

Note: In the GEO Building Model, only one ceiling can be modeled per floor. In order to obtain a correct floor division in the GEO, slabs "must" be connected within a floor in the BIM-Connector or set to non-load-bearing.

Joining types / components

The "Join" function is offered both for surface components such as walls and ceiling slabs, as well as for line components such as columns and beams, which are represented in the static calculation by surfaces and frameworks.

Three types of joining are supported:

1) surface component with surface component,

- Walls with one another: walls that extend to their component axis (example below) or walls that are at a certain angle to one another can be joined.
- Ceilings with one another: ceilings with the same component thickness and ceilings with different component thicknesses can be joined. In this case, a thickness area is created. So that the floors can be joined, they must run parallel to each other.
- Walls and ceilings

2) line component with surface component

- Beams with walls: the wall must be at a certain angle to the beam. Walls that are in the extension of the beam axis cannot currently be joined with beams.
- Columns can be joined with floor slabs

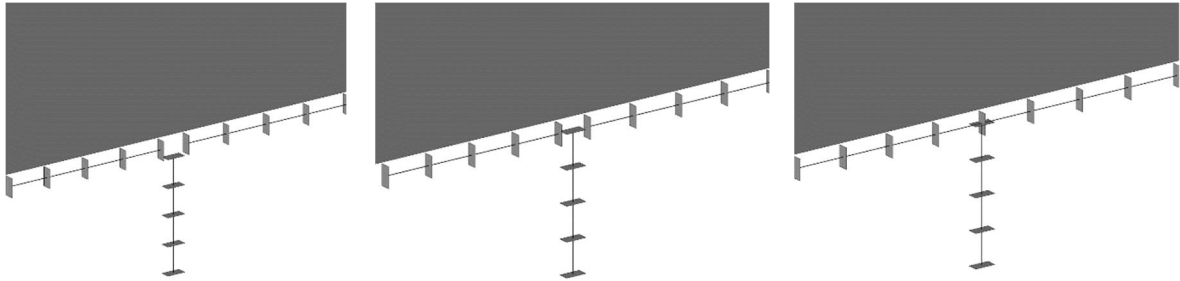
3) line component with line component.

- Beams with columns/columns with beams (example below)

Examples

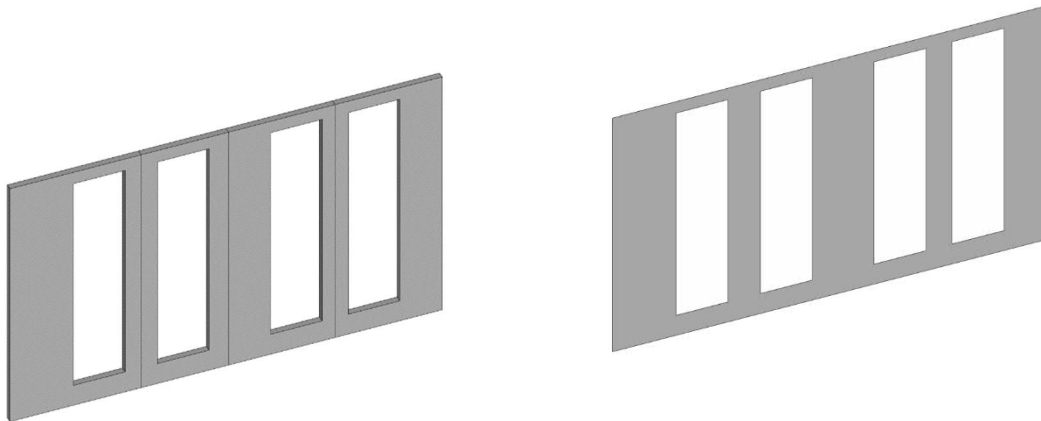
Joining line component with line component

You can see the Analytical model of a ceiling slab with edge joists (picture on the left). The components are not joined and can be joined using the "Join manually" function. The column can first be extended up to the beams (middle picture). To do this, the column is selected first, then one of the two beams. Then the beams have to be extended up to the support (picture on the right). Therefore, the two beams are selected first, then the column. The components are always extended to the last selected component.



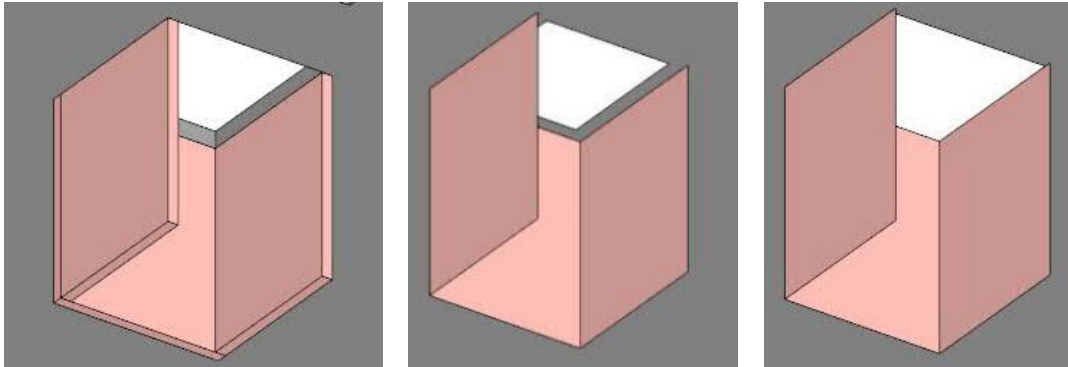
Joining of walls

Two wall elements can always be selected by pressing the "Ctrl" key and joined using the "Join manually" function. In the left picture four individual wall sections can be seen, in the right picture a continuous wall is shown.



Relate opening to wall axis or beam axis

The following picture shows the physical model of a floor from below. This rests on three walls. The analytical model for the same situation can be seen in the middle picture. It becomes clear that the opening is not related to the axes of the walls. This can be adjusted later using the "Join manually" function. To do this, the opening must first be selected, then the corresponding wall to which the opening is to be related with the "Ctrl" key pressed. Then the function is selected and the opening is related to the axis of the wall (picture on the right).



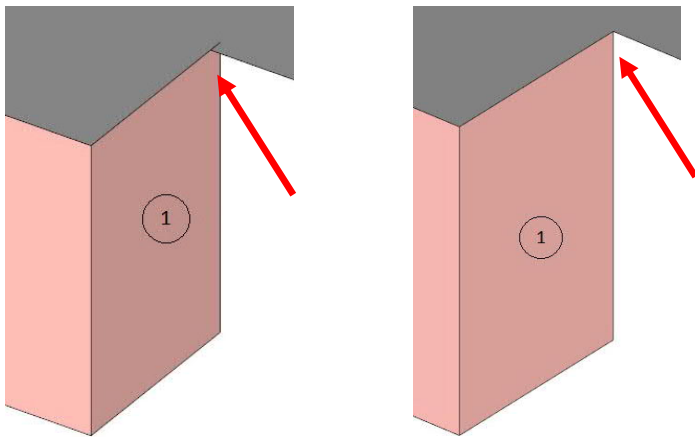
Note: components lying in the background can be selected by holding down the "Ctrl" key and using the mouse's scroll wheel.



Fig. : With a "right click" on an opening, openings (both in walls and ceiling tiles) can be removed.

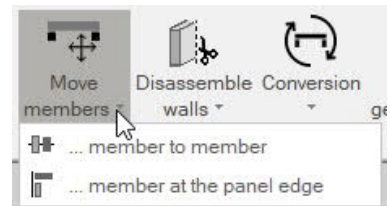
Align the slab edge with the end of the wall

During the automatic joining, the panel contour was not related to the wall end of wall 1. In some cases, however, this may be desirable. The function "Join manually" ▶ "Align panel edge to wall end" is available for this purpose. This function can be found in the drop-down menu of the "Join manually" function. In order to adapt the geometry accordingly, the plate must first be selected, then the right edge of wall 1. Then the function can be carried out. The edge of the plate is aligned with the exact end of the wall that is closest to the click position when selecting the wall.



Move members

When moving members, first click on the first component that is to be moved, then press the Ctrl key and then click on the second component. Then select one of the two move functions:



... Member to member

You can use this function to move the component selected first into the axis of the second component selected.

... Member at panel edge

The component selected first is moved to the edge of the slab/panel selected second.

When clicking the (second) slab, please pay attention to the click position. The edge of the slab to which the (first) component is moved is determined by the click position in the (second) slab selection - it is therefore moved to the edge of the slab/panel that is closest to the click position.

Tip: Zoom in on the corresponding components (mouse wheel, zoom functions under "Graphics" in the top menu). You can rotate the system with pressed right mouse button.

Disassemble walls

With the function "Disassemble walls" and "... into individual members", either all walls can be edited at once or only the selected walls can be edited in different ways according to their openings. This means that existing openings can be taken into account in the Analytical model.

Click the top half of the icon to disassemble all of the walls at once.

Click on the lower half of the symbol to expand the submenu items and to split the previously marked / clicked wall into wall strips or individual components.



Disassemble walls into wall strips

This function splits the wall into strips according to the openings. In the program Slabs with Finite Element PLT and Building Model GEO, no support is modeled at these points.



Disassemble walls into individual members

This function enables the walls to be automatically disassembled into individual components. Areas above openings are automatically converted into beam elements. When handing over to PLT / GEO, the area above an opening is passed over as a downstand beam. In the dialog that appears, you have the option of removing narrow pillars and small openings with this function.

This function enables walls to be converted into beam elements. For example, if an attic was modeled as a wall in the CAD program, it can be converted into beam elements.

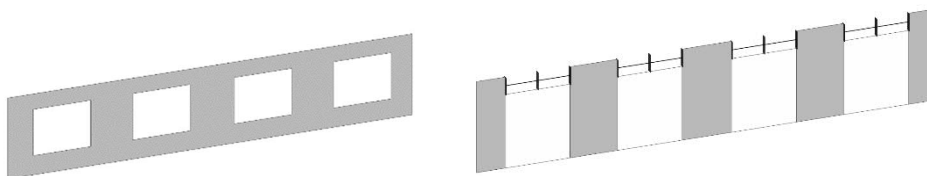
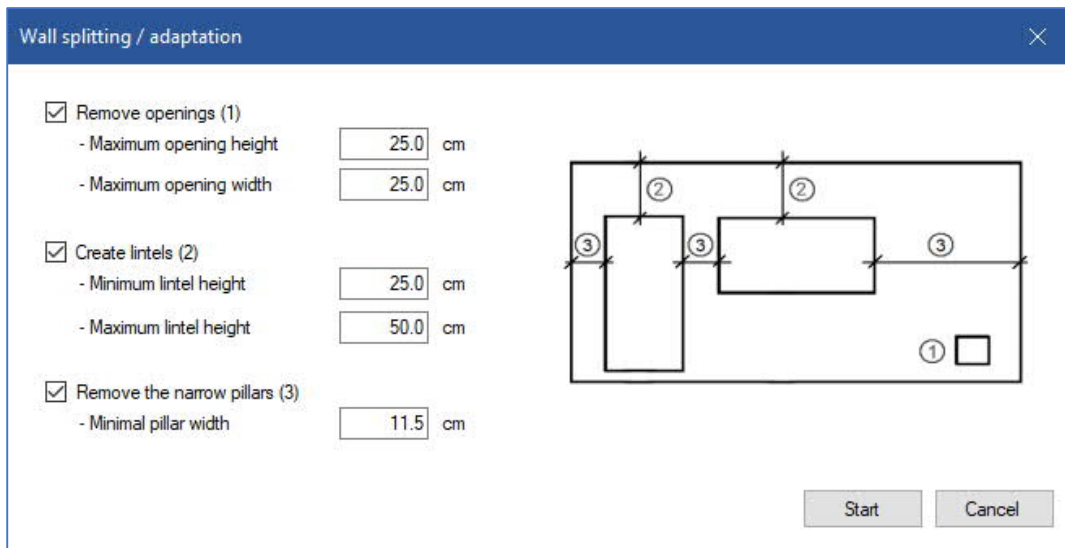


Fig. : A wall with 4 openings (left) was disassembled into wall strips/individual components (right).

The "Undo" function restores the original wall geometry.

Conversion

For the conversion function, the corresponding component must first be selected, then the function must be used. Various conversion functions are available:

The "**Conversion into beam/girder**" function enables walls to be converted into beam elements. For example, if a parapet was modeled as a wall in the CAD program, it can be converted into beam elements.

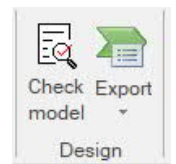
The "**... wall into column**" function enables walls to be converted into columns.

The "**... beams/girders into wall**" function enables beams to be converted into walls.



Check model

This function can be used to check the model for an unfavorable component orientation. The model can be checked for alignment in the vertical direction and for alignment in the horizontal direction. The check model function should be used before exporting.



Vertical direction

When checking in the "vertical direction", the components of one floor are compared with those of the floor below. It is checked whether the deviation of the component axes to be compared lies in the range defined by the minimum and maximum distance. The component axes between wall-beam, wall-wall and beam-beam are checked. If the option "Check columns only against other columns" is selected, a check is made as to whether the component axes of columns are on top of each other across floors.

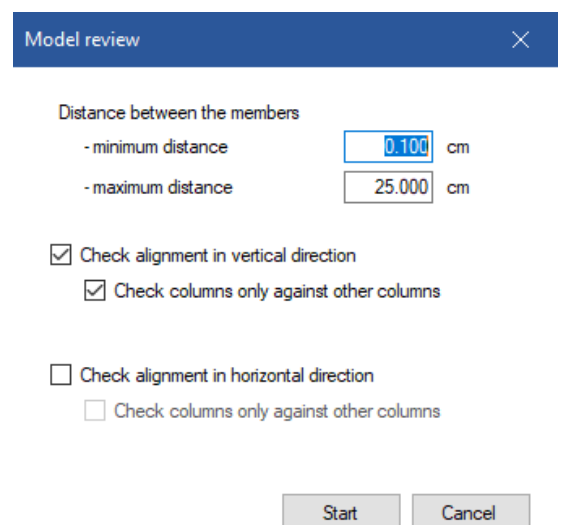
Horizontal direction

When checking in the "horizontal direction", on the one hand the distances between the components in relation to the slab are examined. So: slab-wall, slab-beam, slab-column.

In addition, the component axes within a floor are compared with one another. It is checked whether the deviation of the component axes to be compared lies in the range defined by the minimum and maximum distance. In contrast to the vertical inspection, the axes within a floor are compared with one another. The axes between wall-beam, wall-column, beam-column, wall-wall, beam-beam are checked.

If the option "Check columns only against other columns" is selected, a check is made as to whether the columns are parallel to each other.

Note: For the sake of clarity, the horizontal inspection of large buildings with several floors can also be carried out separately for individual floors.



Since only one slab can be modeled and transferred to the GEO Building Model and PLT slab program per floor, the "Check model" function also searches for unconnected slabs within a floor.

Export to the design programs PLT, B5 +, GEO

Export to B5 +

By selecting walls and columns, these components can be transferred to the [B5+](#) reinforced concrete column program.

Export to PLT

By selecting a slab, a single slab can be transferred to the PLT slab program.

Export to GEO

Individual floors can be exported to [GEO](#). Various export options are available:

- If one slab has been selected, it is transferred to the [PLT](#). All components that lie within the slab contour are also transferred. Components that lie outside the slab contour are not transferred.
- If two or more slabs have been selected, they are transferred to the GEO. All components that lie within the slab contour and any additionally selected components that can also lie outside the panel contour are also transferred.
- If no component was selected, the entire building is transferred to the GEO - including components that are outside the slab contour.
- Only visible floors/buildings are transferred.

Note: After the export, the entire building model should be

- completeness of the components
 - component dimensions
 - positioning
 - static effect
 - material allocation
 - support
 - bedding module of the base slab
- be checked!*

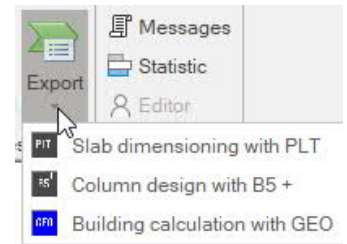
Note: After the export, use the various 3D views for visual inspection.

Note: In order to obtain a correct floor division in the GEO building model, slabs within a floor must be connected in the FBC or set to non-load-bearing.

Note: Since no foundations have to be entered as individual components in the GEO Building Model, the individual and strip foundations should be marked as non-load-bearing components.

Export of buildings with differences in height or multiple building towers with one common basement

For a building with height differences, the slabs should be connected to each other before exporting it. Slabs with the same slab thickness can be connected in FBC via the "Connect manually" function. The first selected slab is automatically moved to the height of the second selected slab and connected to it at this height level. Even slabs with different slab thicknesses can be connected using the "Connect manually" function. In this case, a thickness region is also created for the first selected slab, however. After exporting the data to the Building Model, you should take a detailed look at the transition area. If necessary, an additional beam and/or a slab joint must be inserted here. See also [FRILO Insight 2/2022](#), page 8.



Selection of objects/components with the mouse

To assign a material to components, for example, you must select the relevant components. To do this, you can, drag an area over the components to be selected in the graphic with the mouse. The selected components are now highlighted in color.

Tip: You can switch off the "visibility" of components, materials and properties using the options area on the left, so that only those elements that you want to change/edit are displayed in the graphic for selection. You can also hide, edit, add or delete individual floors and buildings via the "model structure".

Multiple selection with a click of the mouse

To select several individual elements, hold down the "Ctrl" key and click the desired elements with the mouse.

The function groups [Selection and View](#) in the ribbon bar offer further convenient options.

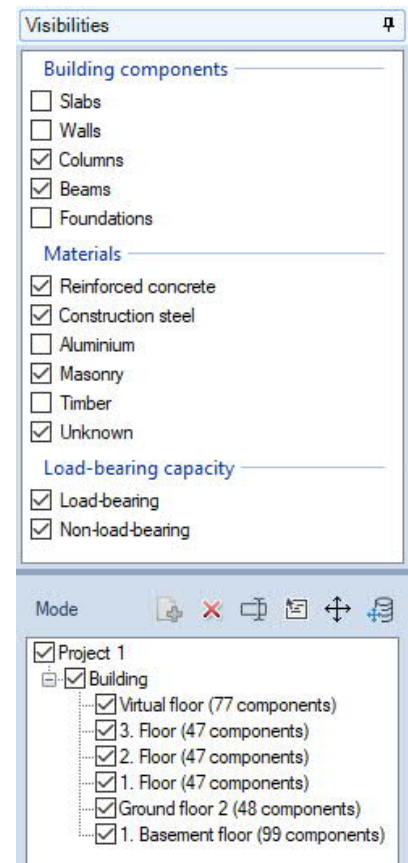
Note: components lying in the background can be selected by holding down the "Ctrl" key and using the mouse's scroll wheel.

Rotate building

The building can be rotated by holding down the right mouse button. The central axis of the building is set as the pivot point. If a component is selected, the pivot point is reset. The selected component is set as the new pivot point for the entire building until the selection is canceled or a new component is selected.

Zooming

As an alternative to the zoom functions (magnifying glass symbols in the top menu), you can also zoom using the mouse wheel.



FAQ

Which FRILO programs are linked to the BIM-Connector®?

In the 2021-1 release, the BIM-Connector® is connected to the PLT Slabs by Finite Elements and the B5+. Reinforced Concrete Column. Since Release 2022-1 the GEO Building Modell interface is also implemented.

Do you want to connect other programs to the BIM-Connector®?

In the following releases the DLT+ Continuous Beam, Framework RSX and the masonry programs will also be connected.

Which file formats can the BIM-Connector® read in?

The BIM-Connector® can read in both IFC files (IFC 2x3 and IFC4) and SAF files.

Which settings when exporting the IFC models from a CAD program favor the import into the BIM-Connector® FBC?

The best results are achieved when the generated IFC file

- only includes the shell (e.g. IFC Design Transfer View, IFC Reference View)
- is well structured
- applicable component types are used (IfcWall, IfcBeam, IfcColumn, IfcSlab, IfcFooting, IfcRoof, IfcMember)
- a material is assigned to the components (at least the material name)
- based on volume geometries that are not too complex (e.g. separate volume description for each wall instead of a single volume for all walls)

How is an analytical model derived from the physical model?

The components are reduced to their system axes. Volume elements are reduced to bar and surface elements.

Can slabs with different thickness ranges be transferred to the PLT/GEO?

Slabs with different component thicknesses can be connected using the function "Join manually"; this slab with its thickness range is transferred to the PLT/GEO programs.

Can openings in walls be taken into account?

Walls can be disassembled into strips. When exporting to the PLT panel program, no support is generated at these points. The openings in the FBC are extended over the entire height of the storey.

The wall can also be disassembled into individual components. A beam is then automatically created over an opening.

Why are wall openings not transferred to the GEO Building Model / PLT Slabs program?

In order for wall openings to be transferred, the walls must first be edited using the "Disassemble walls" function. Walls can be disassembled into strips or individual components. In the area above an opening, either a beam or no support is transferred to GEO/PLT.

Note: Walls in which openings have been modeled up to the ceiling must also be edited.

Can the BIM-Connector® also read IFC analytical models?

No, not yet. At the moment only IFC files with physical elements and SAF files can be read.

Why are some components not read in or not displayed exactly like the original?

When reading in complex component geometries, we undertake a certain abstraction. If the original geometry is too complex, noticeable deviations can occur here. As a rule, however, the deviations are minimal. We are constantly trying to refine and improve our algorithms. However, the focus is on the components, which we can also measure.

Why are items missing from my file?

If components are missing, the components were either modeled with unsupported component types or the "IfcBuildingElementProxy" element type was used. IfcBuildingElementProxy elements are geometric placeholders with no information on the function. For example, if a wall was modeled from IfcBuildingElementProxy elements, "the wall does not know that it is a wall". If the wall were to be converted into an IfcWall component, it would also be recognized by the BIM-Connector® as a wall and thus displayed. The BIM-Connector® currently supports the following IFC component types:

- IfcWall
- IfcBeam
- IfcColumn
- IfcSlab
- IfcFooting
- IfcRoof
- IfcMember

Why does the FRILO BIM-Connector® need higher system requirements than the other FRILO programs?

The BIM-Connector® processes a lot of CAD data. As a result, it is very resource-hungry and requires certain system resources to smoothly display complex models

Can component thicknesses be adjusted in the BIM-Connector®?

Currently, the calculated thickness of ceiling tiles and walls can be adjusted. Advanced settings will follow.

Why is a different floor distribution / an incorrect floor distribution transferred to the GEO building model?

Since only one ceiling can be modeled per floor in the Building Model, the BIM-Connector® can only transfer one ceiling per floor. Unconnected slabs are each pushed into a separate floor. Therefore, before exporting, several slabs should be joined to one another within one storey or set to non-load-bearing.