

# News



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## ❖ Introduction

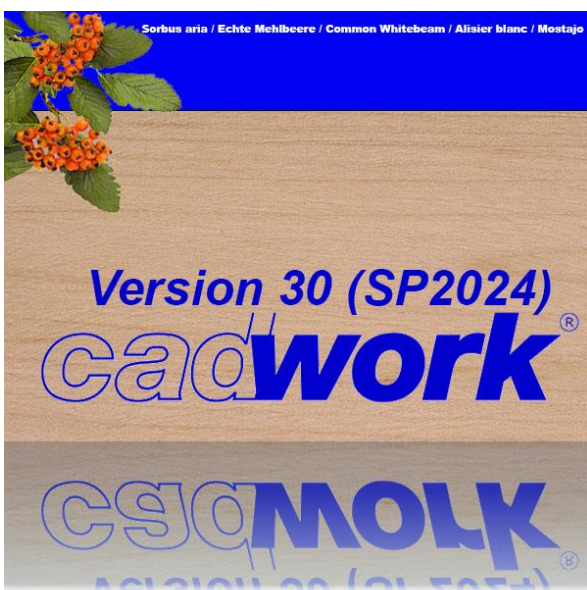
After many years, in 2024 we have finally succeeded in delivering the new version earlier in the year as well as moving the user meeting forward. The aim is to move that even closer to start of the year and into the first quarter. Release date depends on region.

In previous years, this was not possible for various reasons. One reason for this is the fact that the cadwork group and the number of customers has grown considerably. Added to this is the need to have the product available in more and more languages. In recent years, the complete delivery of a new version in all countries could therefore only be completed in the fourth quarter of a year.

Each country-specific delivery brings new customer feedback and entails additions to the development process, meaning that the entire development capacity cannot be available for the subsequent version from the start of delivery. The realization of all planned functionalities of the new version thus postponed the start of delivery to at least the second quarter.

Since the delivery of version 30, only a short period of time has been available for further development, testing and documentation. We have therefore decided to provide a service pack for version 30 in 2024 rather than a master release.

The functional scope of the service pack is less than in a master release. Nevertheless, there are a number of interesting new features, optimizations of existing functionalities and initial results on one of our core topics, the improvement of performance.



The version available from February 2024 is called *cadwork version 30 (SP2024)*. It is based on the version 30 database and is therefore backwards compatible with it and offers a high degree of stability thanks to the corrections and additions made in recent months.

*cadwork version 30 (SP2024)* is not installed via a setup as usual but is installed automatically for every maintenance customer via the Live Update. There is also no need for you to adapt init and/or layout files.

This year's early delivery of the new version is also linked to objectives that will have a positive impact on the performance of our software in the medium and long term. As the chosen approach does not tie up the entire development capacity, we have already been able to work on the cadwork version for 2025 since summer 2023. The development cycle for this version will therefore be extended, giving us the opportunity to further modernize the core and work on medium and long-term projects. For example, the connection to our HOOPS graphics engine will be re-implemented and we will focus intensively on the important challenges of performance and stability.

This version will be called *cadwork version 2025* and will be released at the beginning of 2025.

Due to the approach described for this year, the focus of the user meetings will also be slightly different. Of course, there will be enough space for a detailed explanation of the new features in *cadwork version 30 (SP2024)*. In addition, we will focus on functionalities that are not new, but are only known to a few customers or are not used to their full extent. And last but not least, we want to allow sufficient time to maintain the important personal exchange with our customers.

We would like to take this opportunity to thank all our maintenance customers. The regular mutual exchange provides us with an important basis for practical further development. Your maintenance contributions ensure that we can continue to develop our products and provide you with a competent and committed support and training team in your day-to-day business.

We would be delighted if you would continue to place your trust in us and hope that we will meet your requirements in terms of performance, quality and technical support at all times.

We wish you a successful start to working with the new version. The following documentation should help you. In addition, a [collection of videos](#) on various topics will soon be available in *cadwork version 30 (SP2024)*, which will be continuously expanded. Of course, we will also be happy to answer your questions personally.

Your  team.

## ❖ 3D

### ➤ General information

#### ▪ Update Graphic engine "Hoops Visualize"

We use the "Hoops Visualize" library from Techsoft to display the design in cadwork 3D. The library enables us to display the design in 3D in an appealing and high-performance way. Unfortunately, there were some performance problems in cadwork version 30, which we were able to resolve together with Techsoft last year. The performance in hidden line mode with dashed lines was so inadequate that the 3D file sometimes froze. With Service Pack 2024, you will receive an update that fixes these problems.

#### ▪ Update of the import library "Hoops Exchange"

##### • Importing IFC files

Techsoft supports us in importing IFC files with a library called "Hoops Exchange". We are now using the latest version of this library in Service Pack 2024, which can import large IFC files more quickly. The time gains are particularly large if many intersections have to be carried out due to the geometry description in the IFC file. As an example, we would like to mention IFC files that were output by "Tekla". Furthermore, in the latest "Hoops Exchange" library, the reliability and tolerance of faulty and "unclean" IFC files has been improved. Customers who frequently receive IFC files from various third-party programs know that the quality of the data is often not sufficient to read them into cadwork 3D. "Hoops Exchange" is looking for ways to read the "unclean" IFC files anyway, so that they can at least be used as a visualization in cadwork 3D.

##### • Importing SketchUp files

With the update of "Hoops Exchange", the import of SketchUp files in 2023 format is supported. In the last version, you could only import files up to the 2021 format.

- Calculation time of the wall shop drawing export

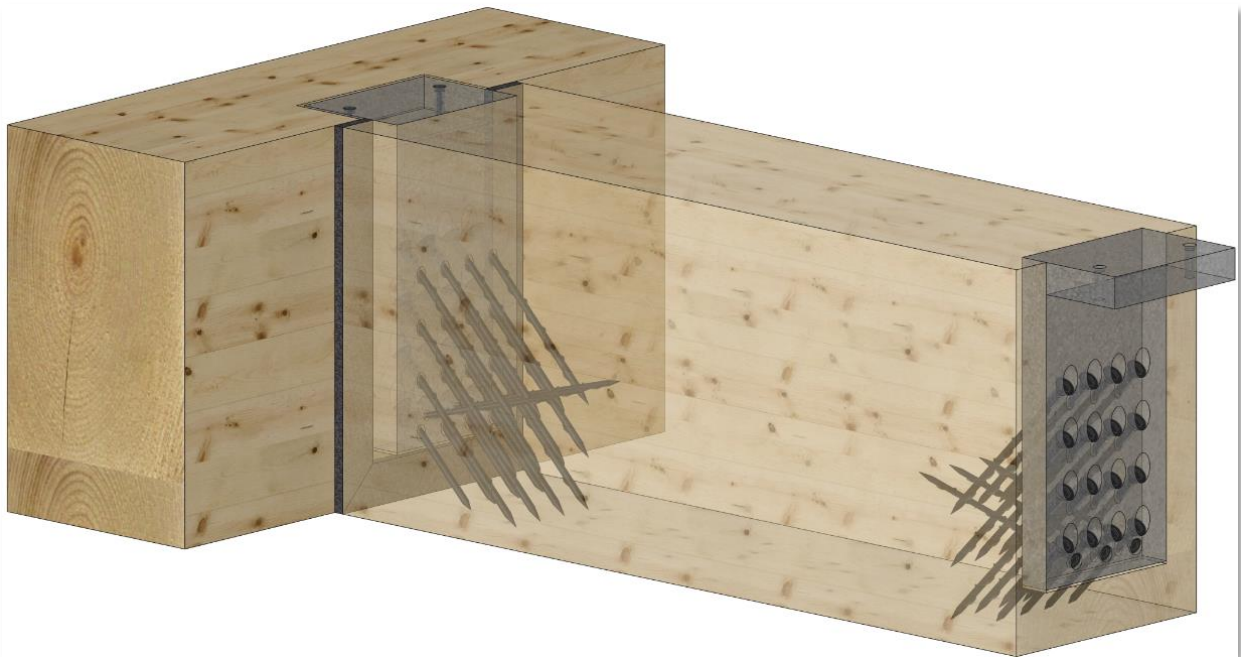
For Service Pack 2024, we have focused a great deal of attention on the performance of the wall shop drawing export. Especially exports from projects with a high level of detail and many drillings, connector axes and/or MEP axes repeatedly lead to long export times.

For this reason, we have examined these exports in detail and implemented various conversion measures and optimizations that lead to a significant acceleration of export times. The more complex the project, the greater the time saving. Nevertheless, the speed advantage also has a positive effect on smaller projects.

For future versions, we will continue along this path and focus a large part of our attention on optimizing and accelerating exports in order to further reduce export times significantly.

- Revision of the connector axis drilling visualization

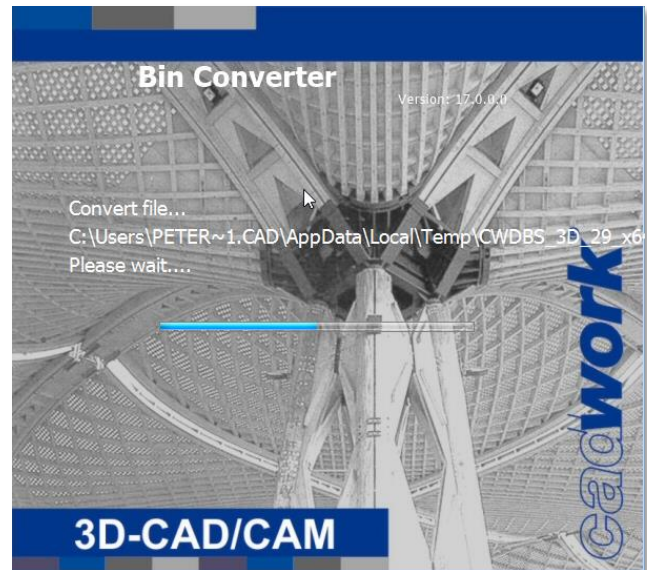
For Service Pack 2024, the visualization of the drillings of connector axes (CA) has been revised. This revision means that the holes of the CA are now rendered much faster. Especially for drillings that are cut off at an angle, only around a third of the time should be required for rendering. This means that files with many diagonally cut CA drillings in particular should now start up much faster.



- Starting up version 15

Up to and including cadwork version 15, a binary file format was used for sequential reading and writing of all data. Since version 16, an SQL database has been used to store the design data. This is a modern form of data storage that enables flexible expansion, offers greater data security and ensures permanent access to all data during program runtime.

Due to this change, 3D files from version 15 or older had to be converted for loss-free data transfer. For this purpose, a converter was installed upstream when importing such a file. The continuous further development of the database also requires further development of the converter.



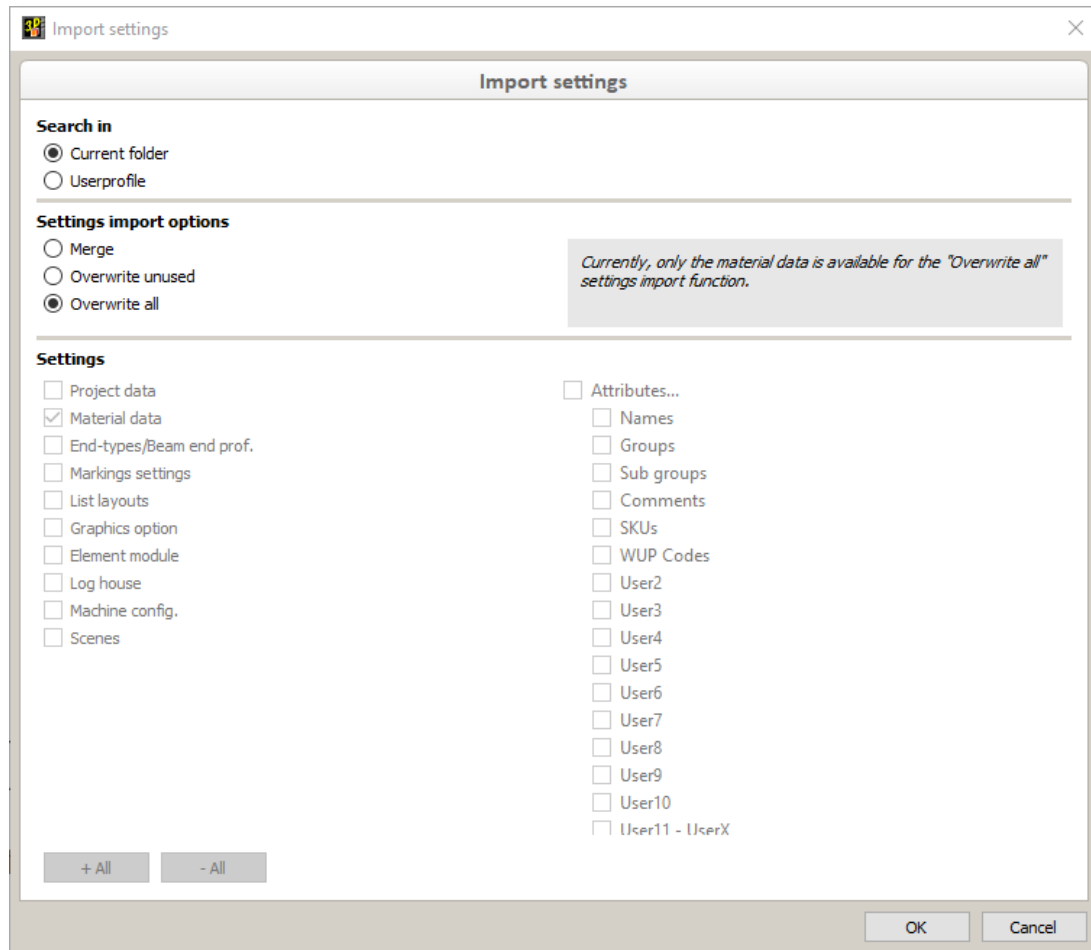
As the import of 3D files from version 15 or older is required less and less frequently, the adaptation of the converter has been discontinued from version 30 (SP2024). Therefore, only files from version 16 can be imported from now on. When attempting to import older files with cadwork 3D (SP2024), a corresponding message will appear. Older files must first be started and saved with an older version (e.g. cadwork 3D version 29).



➤ Windows/Top menubar

▪ *File -> Settings...*

With the Service Pack 2024, the new *Overwrite all* settings import option is available in the *Import settings* dialog.



The *Overwrite unused* settings import option, which was already available in previous versions, first deletes the existing settings from the 3D file and then imports all settings from the selected file.

Settings that are already in use are neither deleted nor changed with this option.

The new settings import option *Overwrite all* also deletes the existing settings from the 3D file and then imports the settings from the other file.

The settings that are in use are also overwritten with this option. This means that it is subsequently possible to completely adopt the settings from another file and apply them to the entire project.

This option is currently only available for material data. It is planned to extend this to the other settings in the next versions.



- *File -> Import...*  
 News on importing files is documented in the Right menu section under *Add -> Files....*  
 News on importing IFC files can be found in the IFC section.  
 News on importing Point clouds can be found in the Point cloud section.
  
- *File -> Export...*  
 The latest news on exporting files is documented in the Right menu section under *Export -> Files....*  
 News on exporting IFC files can be found in the IFC section.
  
- *Edit -> Activate all <Ctrl+A>*

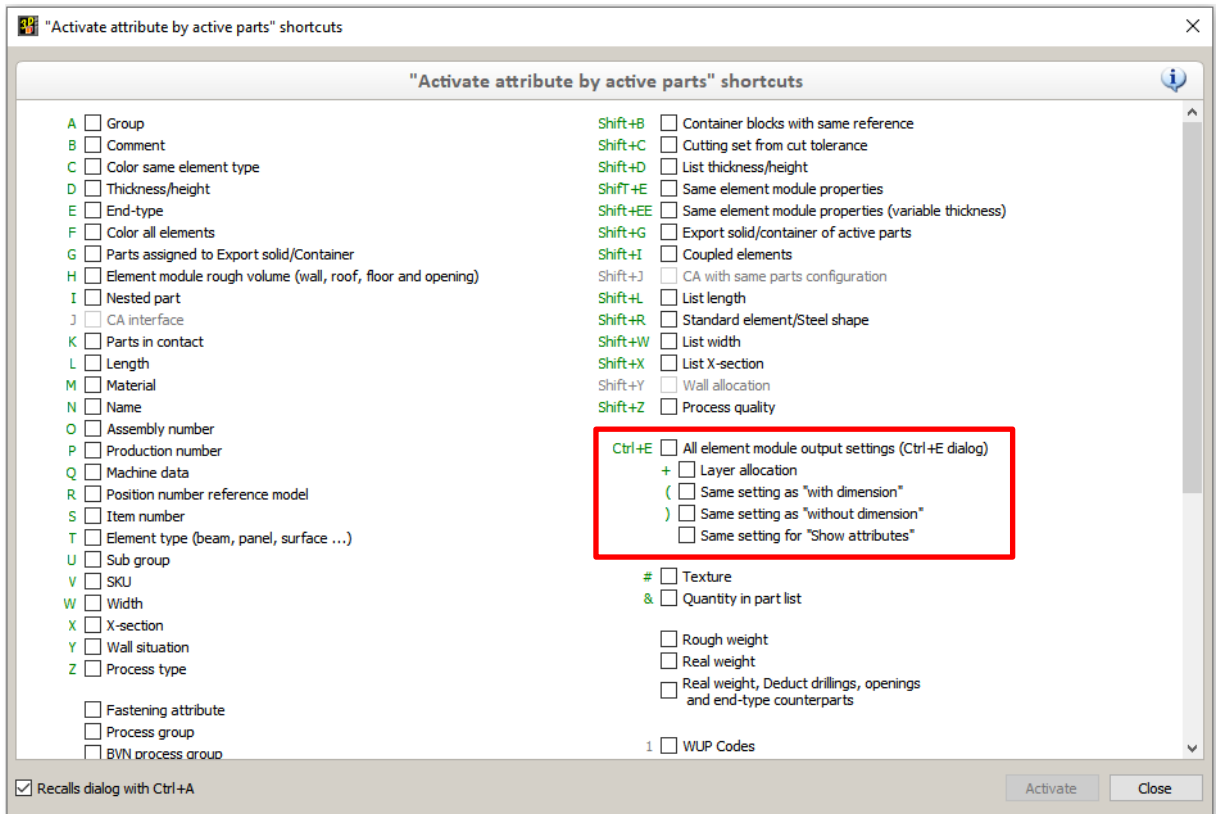
- *Info window*

This function can be used to activate all elements or elements with special properties very quickly. As with every activation process, the info area in the bottom right-hand corner is then updated with the information on the active components. If not all item properties have been calculated in advance, there may be a noticeable delay before further processing is possible, if there are a large number of active items. This time span has been shortened in the service pack, making it possible to work more smoothly.

It was also possible to achieve a speed advantage in large files if there are many items with the same item or production number. The calculation time for the number of active items with same item or production number has been significantly reduced.

Info	
Element	Group
Name	***
Group	***
Sub grp.	***
Building	***
Storey	***
No. production li..	***
No. item list	***
No. assembly list	***
Wall situation	
Color	***
Active elements	
Nodes	0
Axes	18
Lines	4
Surfaces	2
Volumes	29
Volume	15.076 m³
Weight	23223.0 kg
Beam run length	42.1776 m
Panel run length	11.7444 m
Running length pa..	3.6560 m
Panel surface	40.6 m²

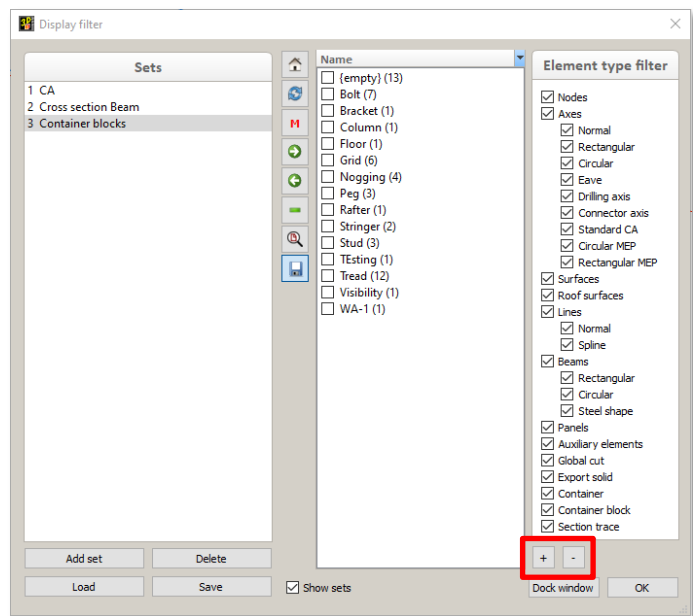
- *Shortcut key <Ctrl+E> - "All element module output settings (Ctrl + E dialog)"*  
 In Service Pack 2024, the output settings available in the Element module dialog (Ctrl+E dialog) are grouped together. The entire group can be activated with the new option *All element module output settings (Ctrl+E dialog)*.



In addition, the previously missing output setting "Show attributes" can be taken into account with the option *Same settings for "Show attributes"*. This makes it possible to activate elements with identical output settings for the wall shop drawing export.

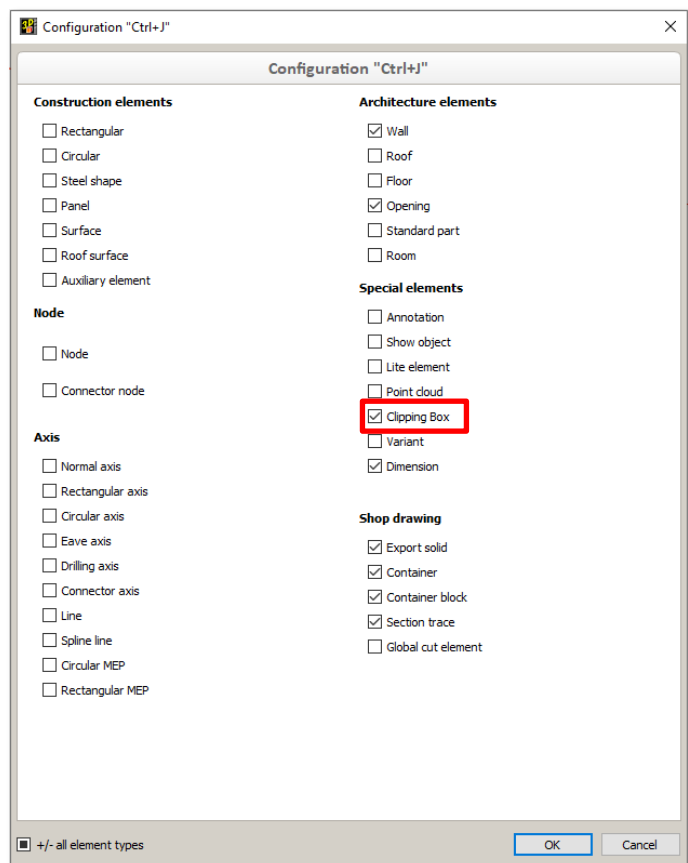
▪ *Window -> Display filter*

The + and - buttons are available for the element type filter to quickly select or deselect all element types. The position of these is now fixed so that both buttons are displayed even in small window sizes with an active scroll area.



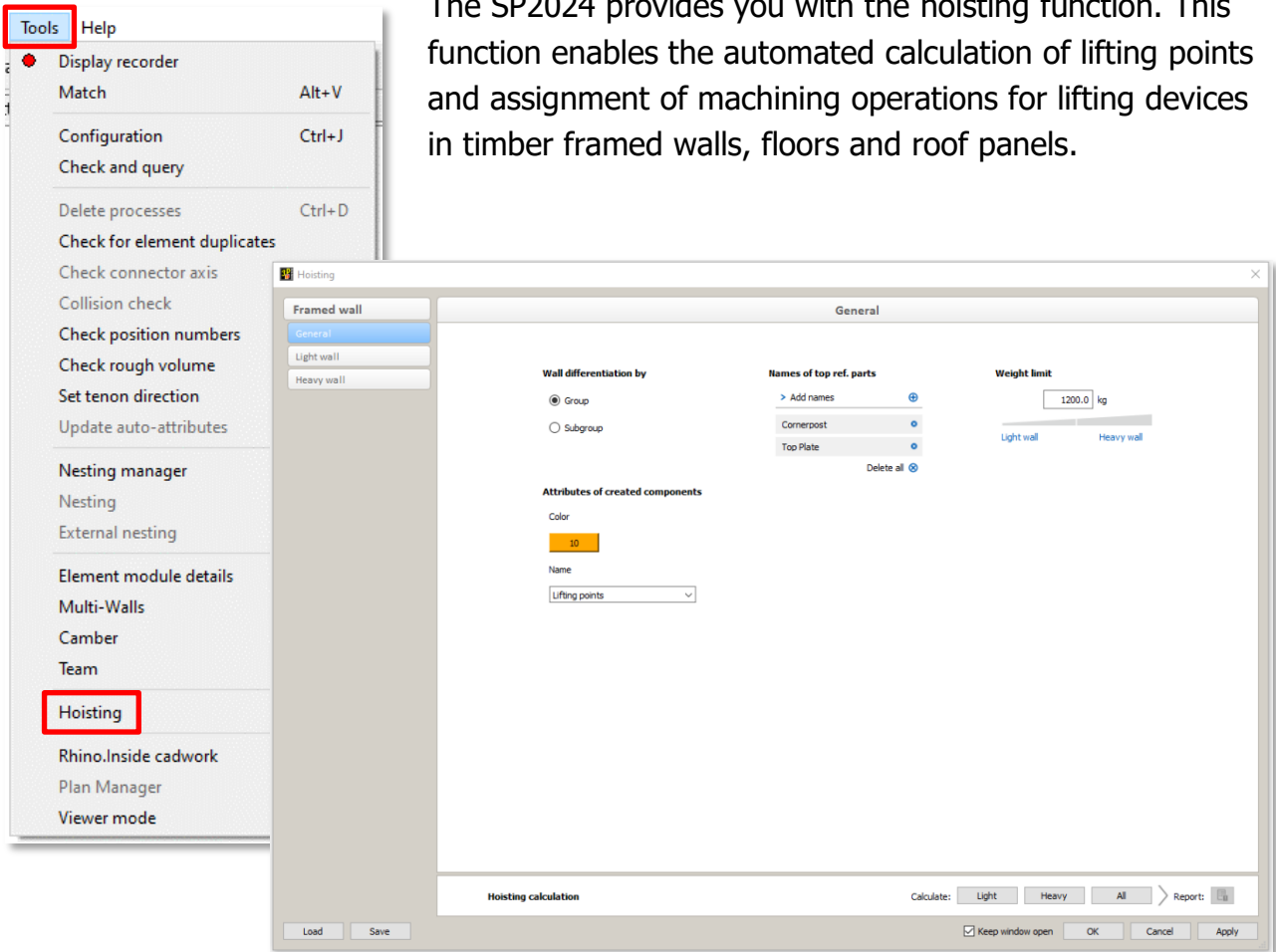
▪ *Tools -> Configuration "Ctrl+J"*

The clipping box is now available as a new special element for fading in or out using the <Ctrl+J> key combination. The clipping box makes it easier to work with point clouds.



Tools -> Hoisting

The SP2024 provides you with the hoisting function. This function enables the automated calculation of lifting points and assignment of machining operations for lifting devices in timber framed walls, floors and roof panels.



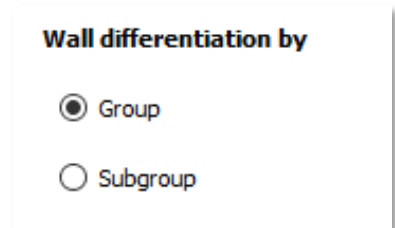
A *Light wall* and a *Heavy wall* option are available. The *Light wall* option places the lifting points in the top plate as example for mechanical lifting anchors whereas the *Heavy wall* option additionally uses the studs to spread the lifting load onto the panels. An example here would be lifting slings.

- *General*

All settings that can be defined independently of the wall type are defined in this tab:

- ◆ *Wall differentiation by*

As usual in cadwork, the walls, roof or floor panels are defined as a group or a subgroup with a wall rough volume. The rough volume axis alignment plays a decisive role in referencing, which is required to calculate the lifting point positions. The lifting points you define are distributed according to the materials centre of gravity.



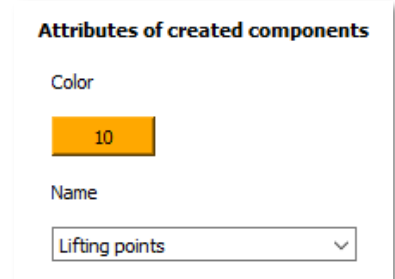
- ◆ *Attributes of created components*

- ▶ *Color:*

Color that the created lifting point components (Drilling axes, connector axes, markings,...) are getting assigned.

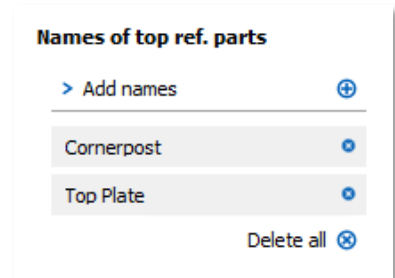
- ▶ *Name:*

Name that is assigned to the generated lifting point components.



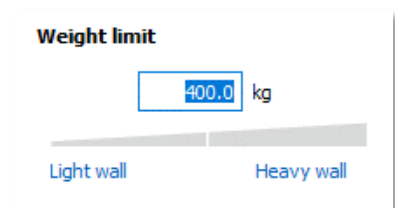
- ◆ *Name of top reference parts*

When calculating the lifting points, it is essential that reference parts are defined. Those top reference parts are used for lifting point placement. These parts serve as the basis for the positioning of machining operations for lifting points such as drilling axes, connector axes and end-types. Defining the reference parts ensures that the machining operations are carried out in the correct places.



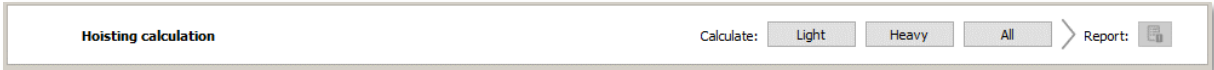
- ◆ *Weight limit*

Limit value for determining whether a wall is considered as "Light wall" or "Heavy wall". This differentiation is mainly used to assign the two option of lifting point configuration - top plate fixing only or use of stud to spread the load.



- *Hoisting calculation*

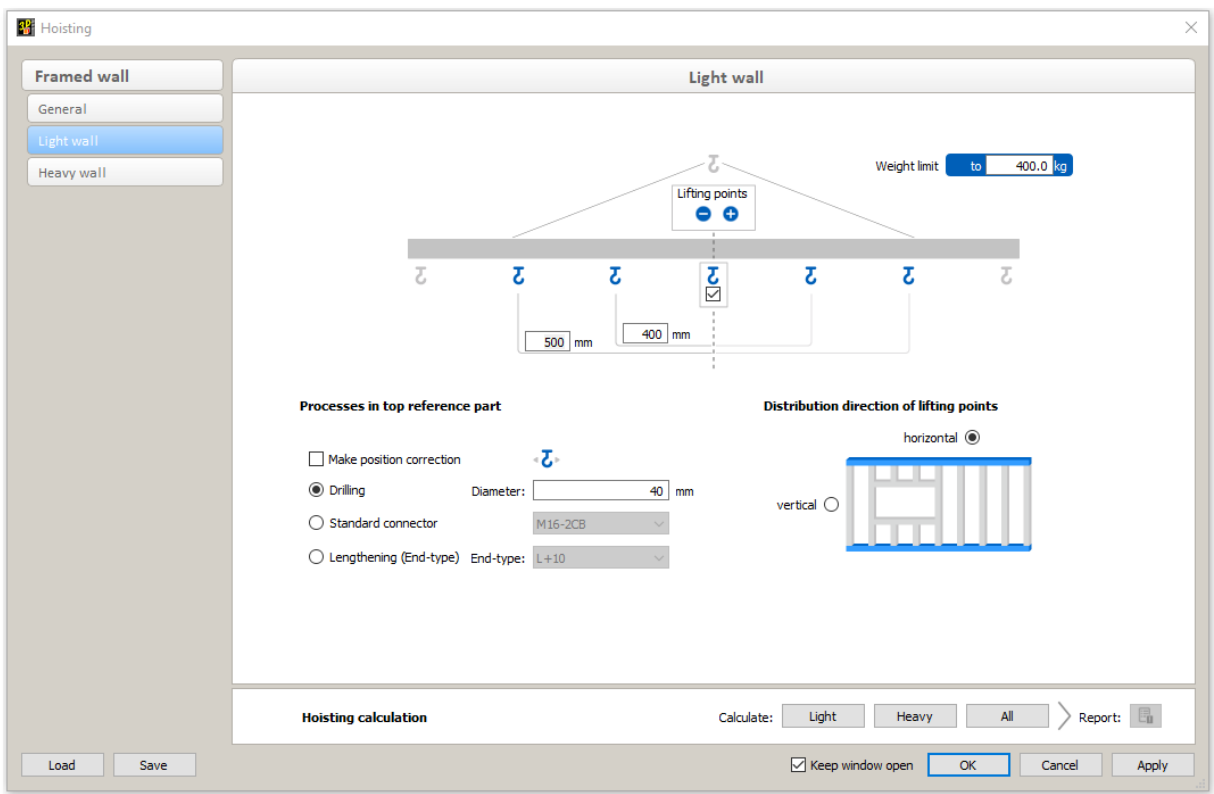
This dialog component is displayed on all three tabs (*General, Light wall* and *Heavy wall*) in the "Hoisting" dialog so that the calculation can be started at any time.



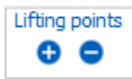
Use the *Light, Heavy* and *All* buttons to start the respective calculation for all active walls.

- *Light wall*

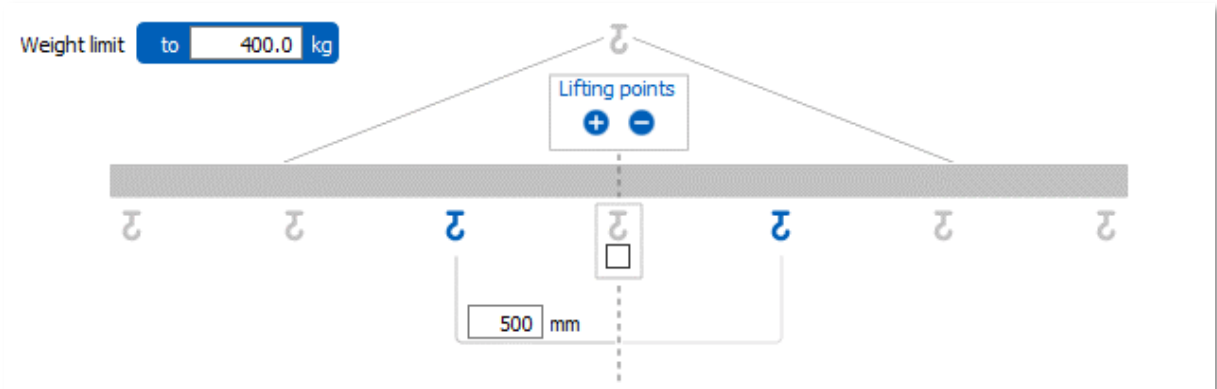
All walls below the entered weight limit are classified as "Light wall" and their lifting point calculation is carried out using the configuration in this tab.



◆ Lifting points



You can use the + and - buttons to add or remove additional lifting points.

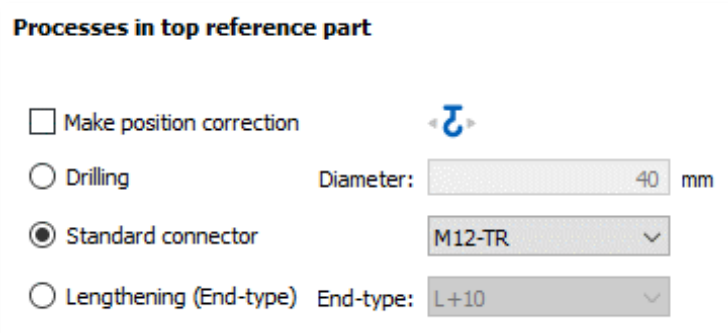


You must specify an offset for each additional lifting point. The offset is always calculated from the centre of gravity in the positive and negative axis direction.

◆ Processes in top reference part

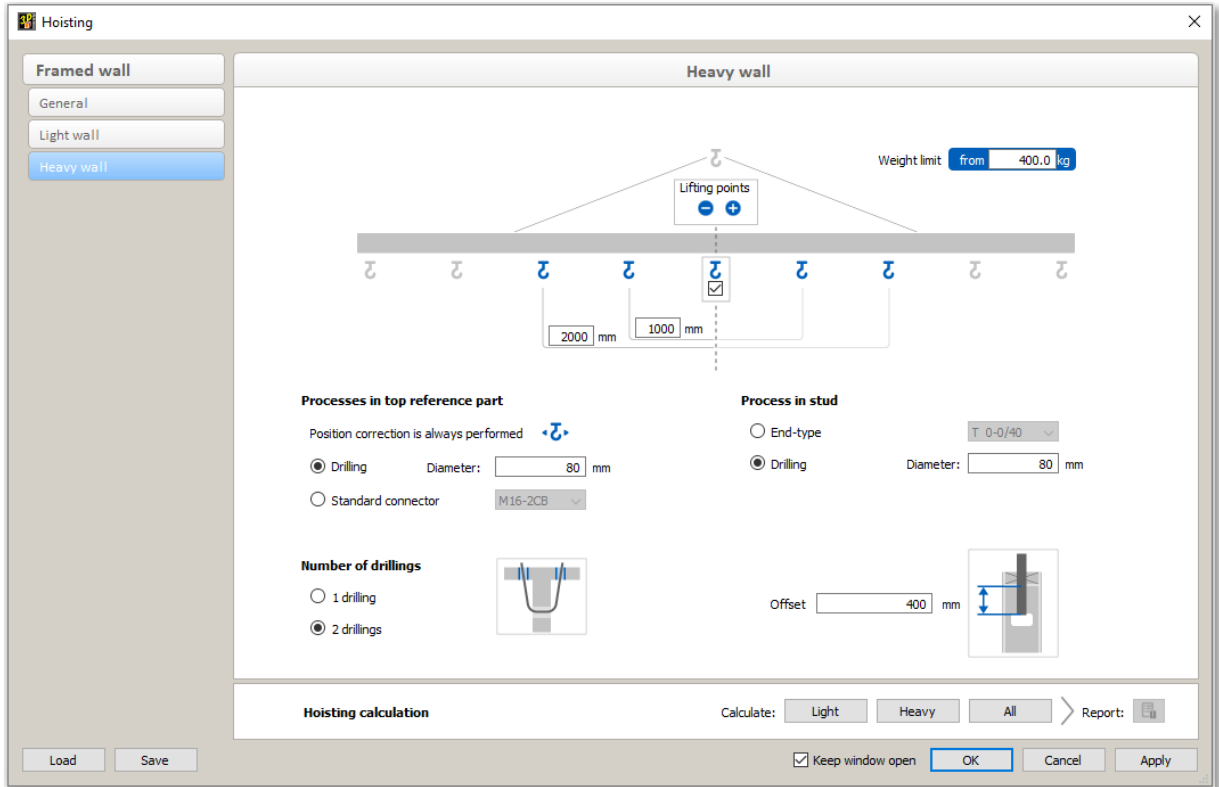
If the *Make position correction* checkbox is selected, an attempt is made to position the lifting point without colliding with the studs. Otherwise, the exact offset specified is used for positioning. A position correction is only made in the direction of the longitudinal axis of the wall.

The element capable of creating a process is generated during the calculation according to the active selection. Drilling axes with the defined diameter, connector axes previously added to standard element library as standard connector axis or end-types of the type "Lengthening" are available for selection.



Lifting point positions can be generated either horizontally or vertically for the type "Light wall". The prerequisite is that top reference parts can be determined in these directions. Vertical lifting point positioning is not possible for the type "Heavy wall".

- *Heavy wall*

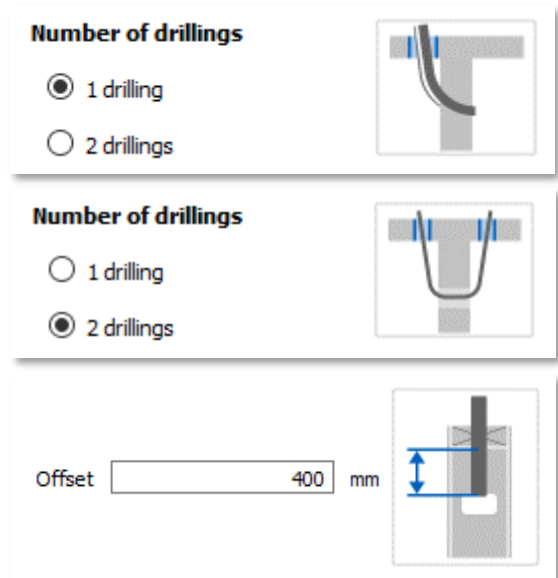


For walls that are heavier than the weight limit entered, the calculation is carried out using the configuration in the *Heavy wall* tab. Lifting point positions can also be added here and their offsets defined. In contrast to light walls, the position correction is always carried out. Standard connector axes or drilling axes can be calculated in the selected top reference part.

◆ *Number of drillings*

It is specified whether one or two drillings are to be drilled to accommodate the lifting device. For a single hole, this is positioned on the pull side of the handle. If two holes are drilled, these are positioned to the left and right of the identified stud.

In addition, holes or mortises are created in the vertical stud by the selected end-type, through which a sling can then be pulled.





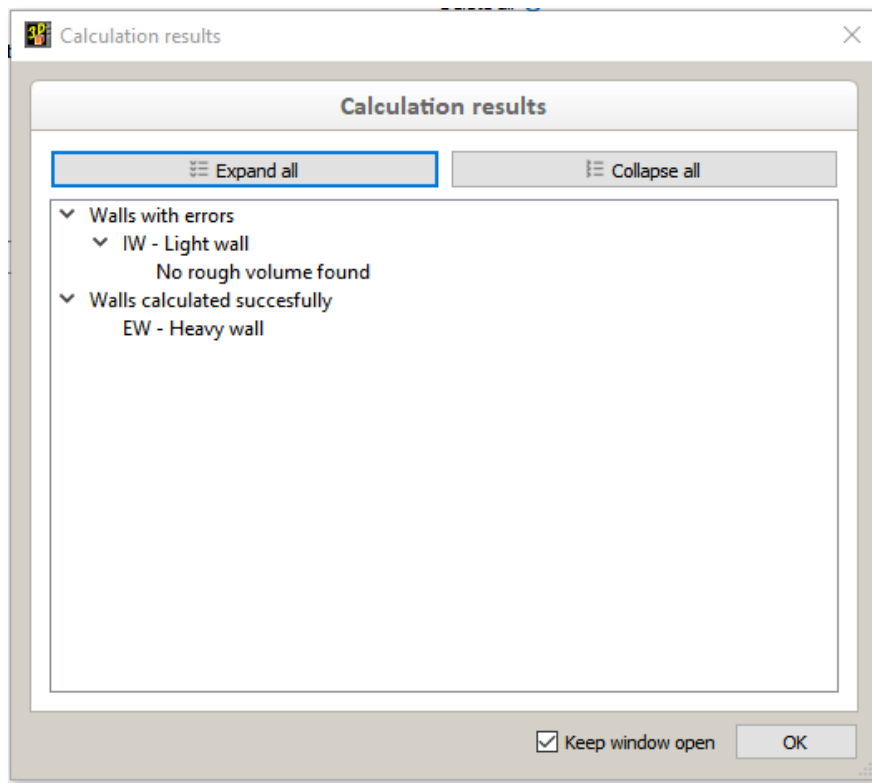
- *Hoisting calculation*

The *Light*, *Heavy* or *All* buttons are used to calculate the lifting points for the active walls, taking their weight classification into account.



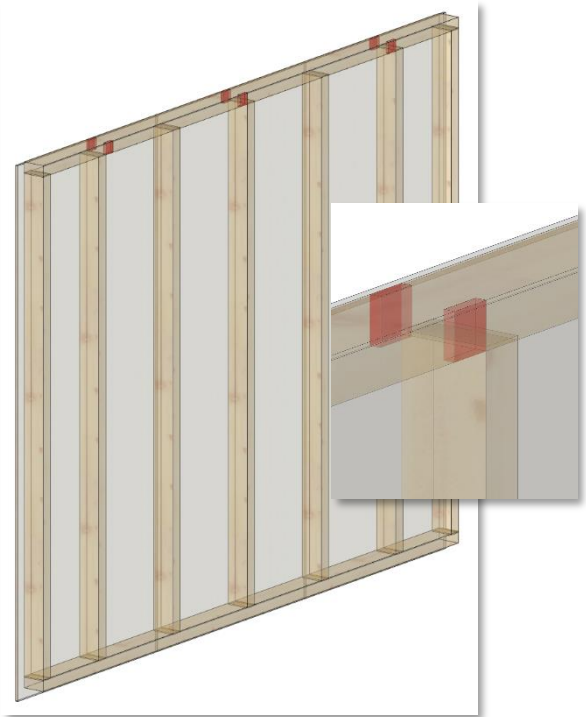
After the calculation, a calculation report is displayed.

The components created by the hoisting function were added to the walls with the properties set under *General -> Attributes of created components*.



- Lifting point examples, possible calculation results

- ◆ Light wall with 15 mm lengthening (End-type), horizontal distribution of lifting points, processes in top reference part with name "Top plate".



- ◆ Heavy wall with two drilling axes through the top reference part named "Top plate". The sling hole through the stud is generated with use of an end-type.

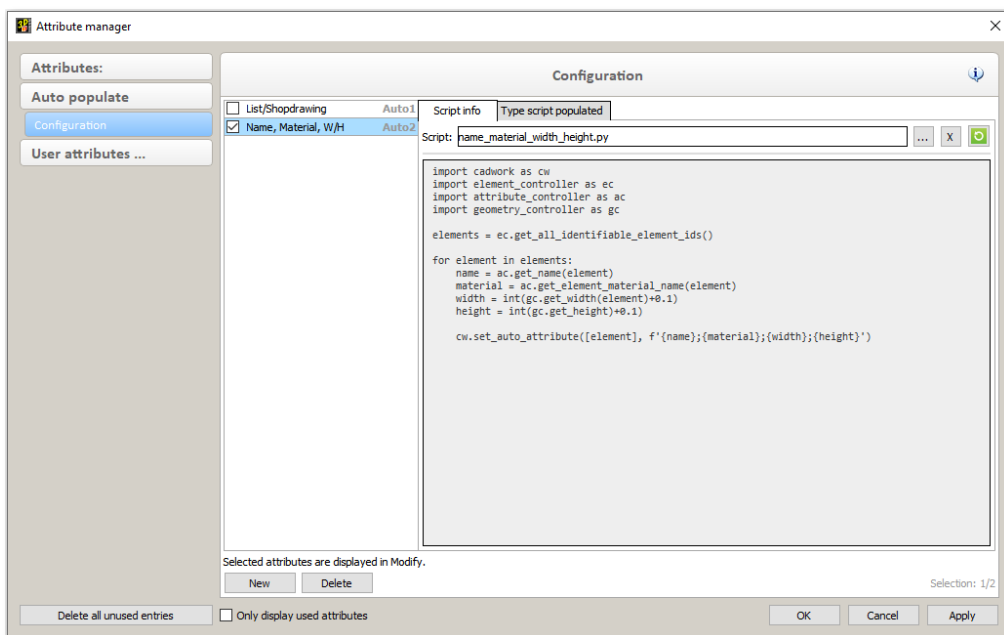
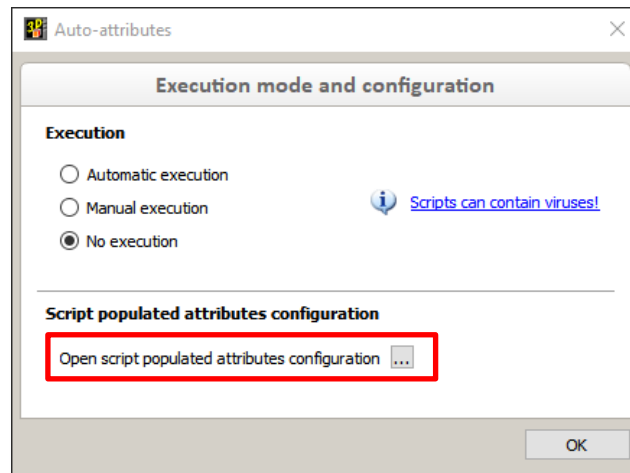
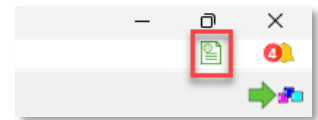


- Script-populated attributes

The script-populated attributes introduced in version 29 are becoming increasingly popular.

Especially when developing a script-populated attribute, it is often necessary to adapt and correct the script used. To do this, it is necessary to open the *Attribute manager* dialog under *Userprofile (user's settings) -> Timber... -> Attributes*, switch to the *Auto populate* tab and update the corresponding script using the button.

To simplify this process, the *Script-populated attributes* dialog has been revised and the *Script populated attributes configuration* section has been added. This makes it possible to switch directly to the *Auto populate -> Configuration* tab in the *Attribute manager* dialog and update the corresponding script or create new scripts.

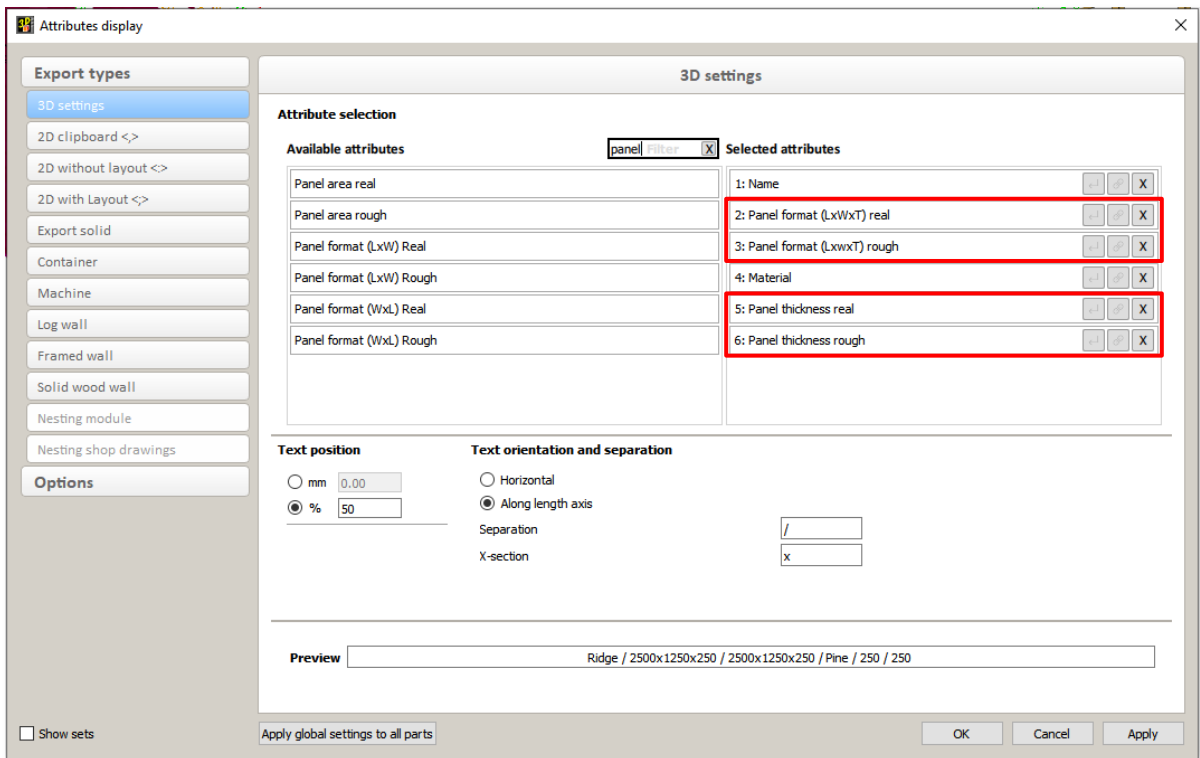


➤ Status bar

▪ *Attributes display*

The following new attributes are now available for the attribute display:

- *Panel thickness real*
- *Panel thickness rough*
- *Panel format (LxWxT) real*
- *Panel format (LxWxT) rough*



➤ Left menu

▪ Configuration of the *Graphics options*

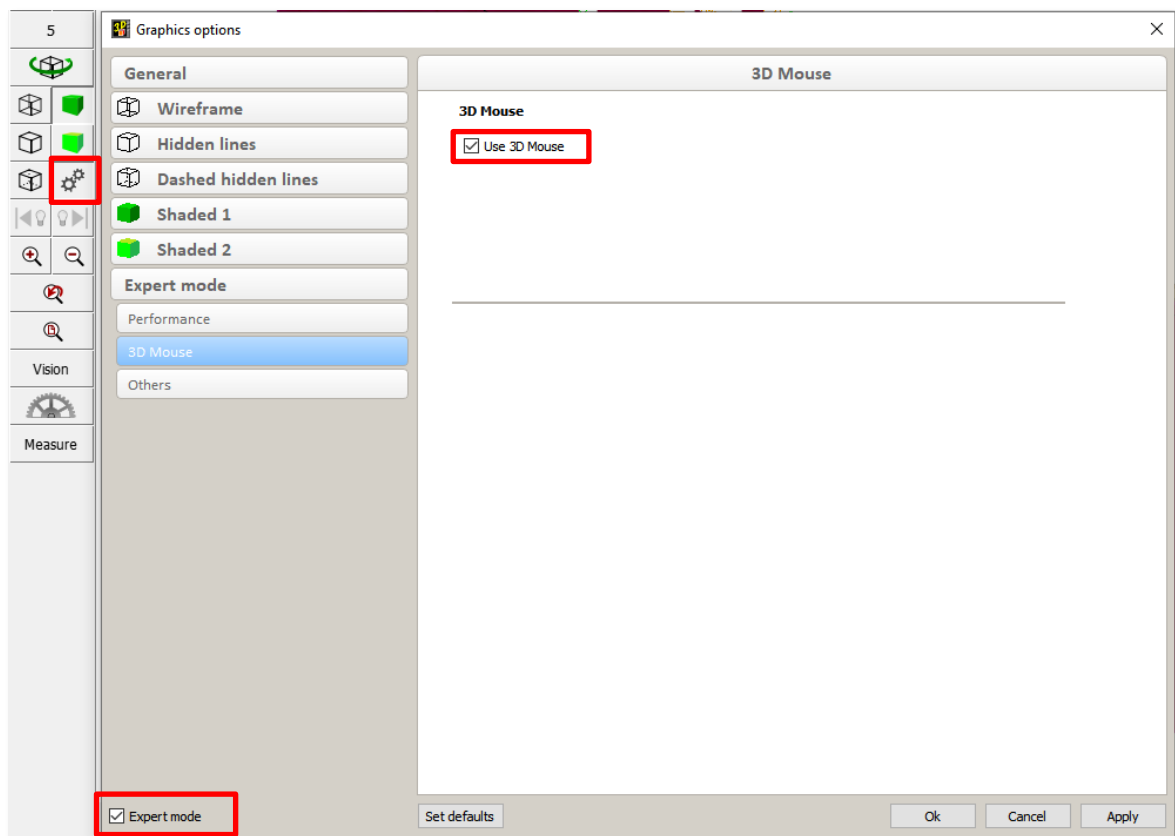
• *Expert mode -> 3D mouse*

The integration of SpaceMouse devices from 3Dconnexion has been improved with Service Pack 2024.

These improvements are currently being reviewed by the manufacturer 3Dconnexion. After successful testing and review, cadwork 3D is a certified software for 3Dconnexion 3D mouse. The certification will be completed in the next few weeks.



In order to use a SpaceMouse in cadwork, it must be activated once in the *Expert mode* tab of the *3D - Graphics options* dialog:



All other settings for the 3D mouse are made in the 3DxWare driver of the mouse. This requires the mouse to be connected and the latest 3DxWare software from 3Dconnexion to be installed.

The following changes have been integrated in this area:

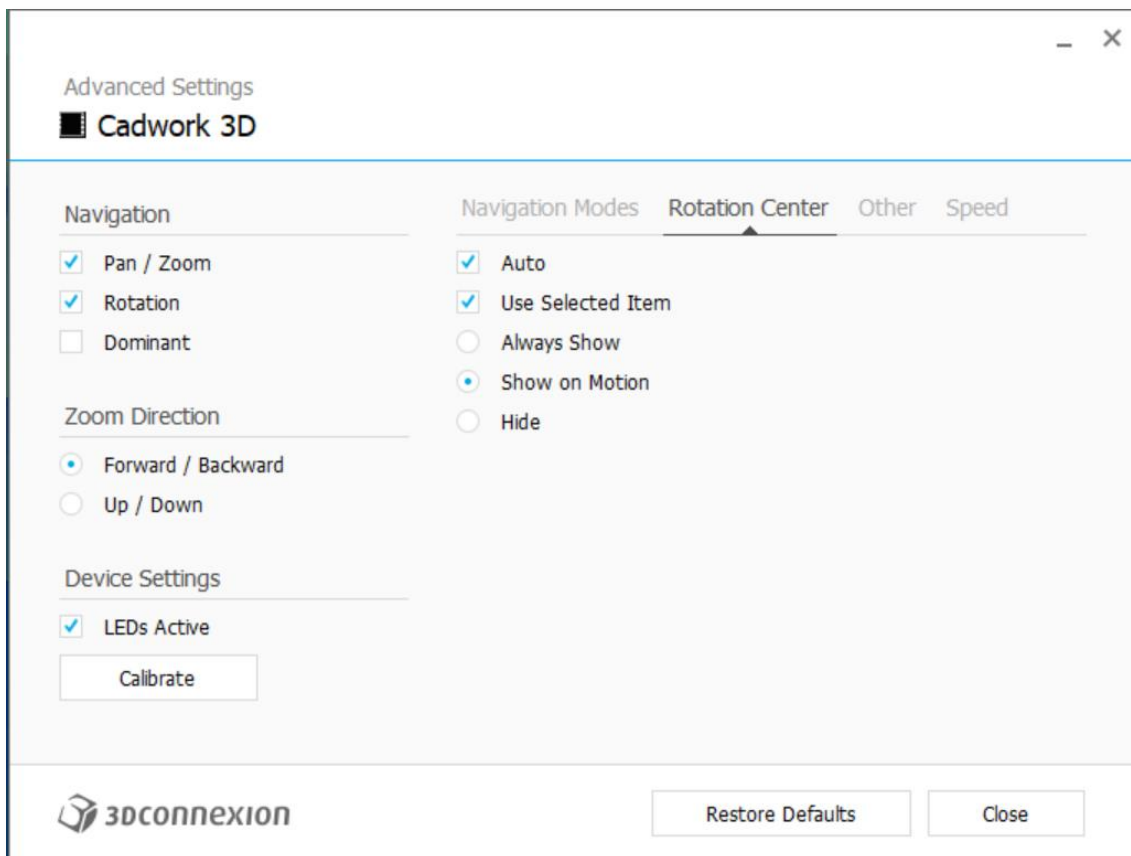
- ◆ New representation of the rotation point

We now use the symbol provided by 3Dconnexion for the rotation point instead of a generic point marker.



The new rotation point icon is a small representation of the 3Dconnexion logo.

During rotation, the settings set in the 3Dconnexion driver are used:



- ▶ *Use selected item*

If this option is active, the rotation point is placed on the active point. If no point is active, the rotation point is placed in the middle of all active elements.

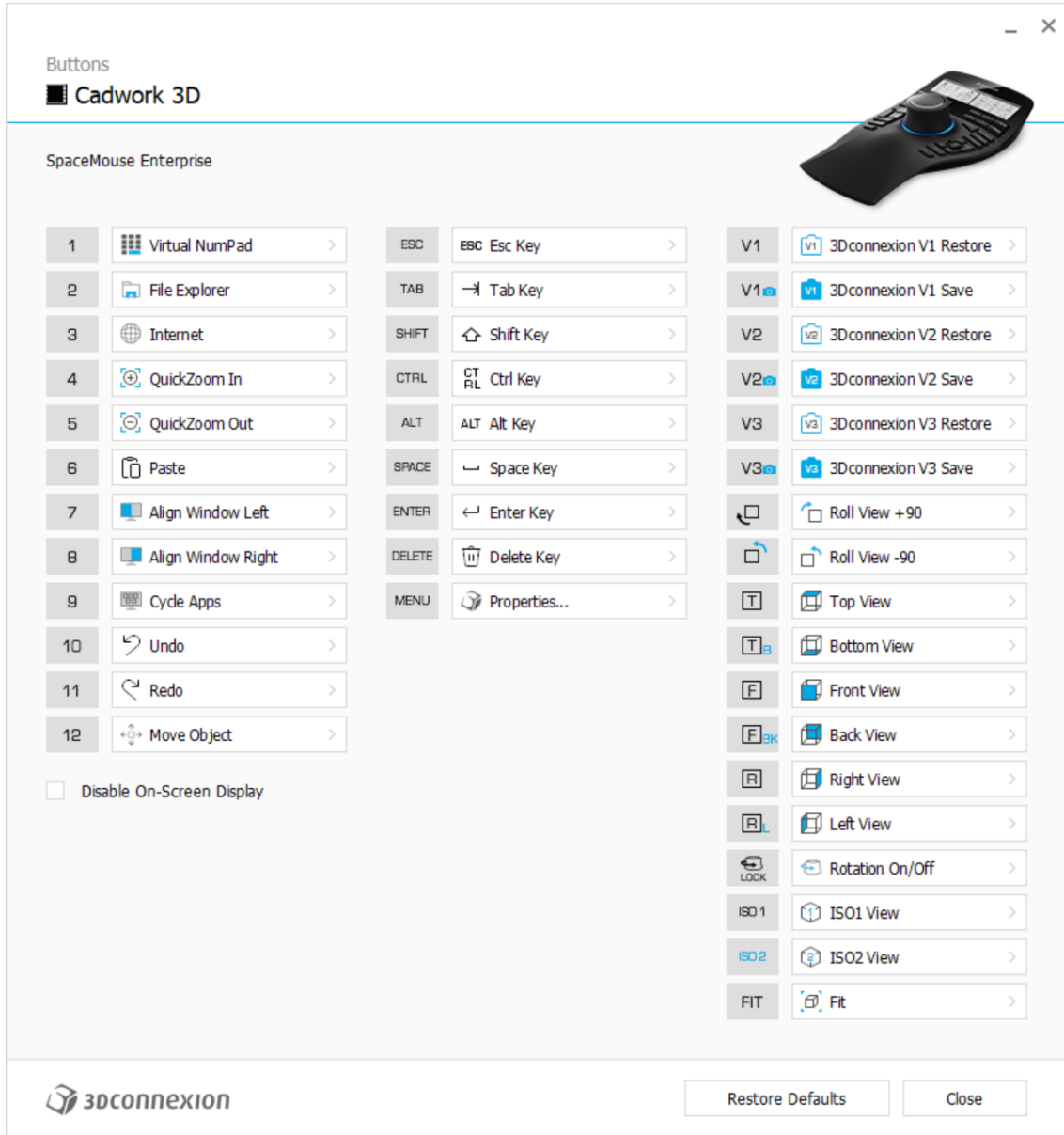
If no elements are active or the option is not activated, the following applies:

If all visible elements are in the image section, the rotation point is placed in the middle of the visible elements, otherwise the rotation point is placed on the first element in the middle of the image section.

- ◆ Navigation functions

All navigation functions of the SpaceMouse are now supported.

QuickZoom (zooms in/out to crosshair position), Fit (zooms to all visible elements), ISO views and saving and loading views in 3DxWare and many other functions are now also supported by cadwork 3D.



- ◆ Compatibility

The integration is only compatible with SpaceMouse devices from 3Dconnexion. Devices from other manufacturers are not supported.

The 3D mouse is an additional operating option for cadwork 3D, but it never completely replaces a normal mouse and keyboard.

- *Userprofile (user's settings)-> Timber... -> End-types... -> Dovetail*

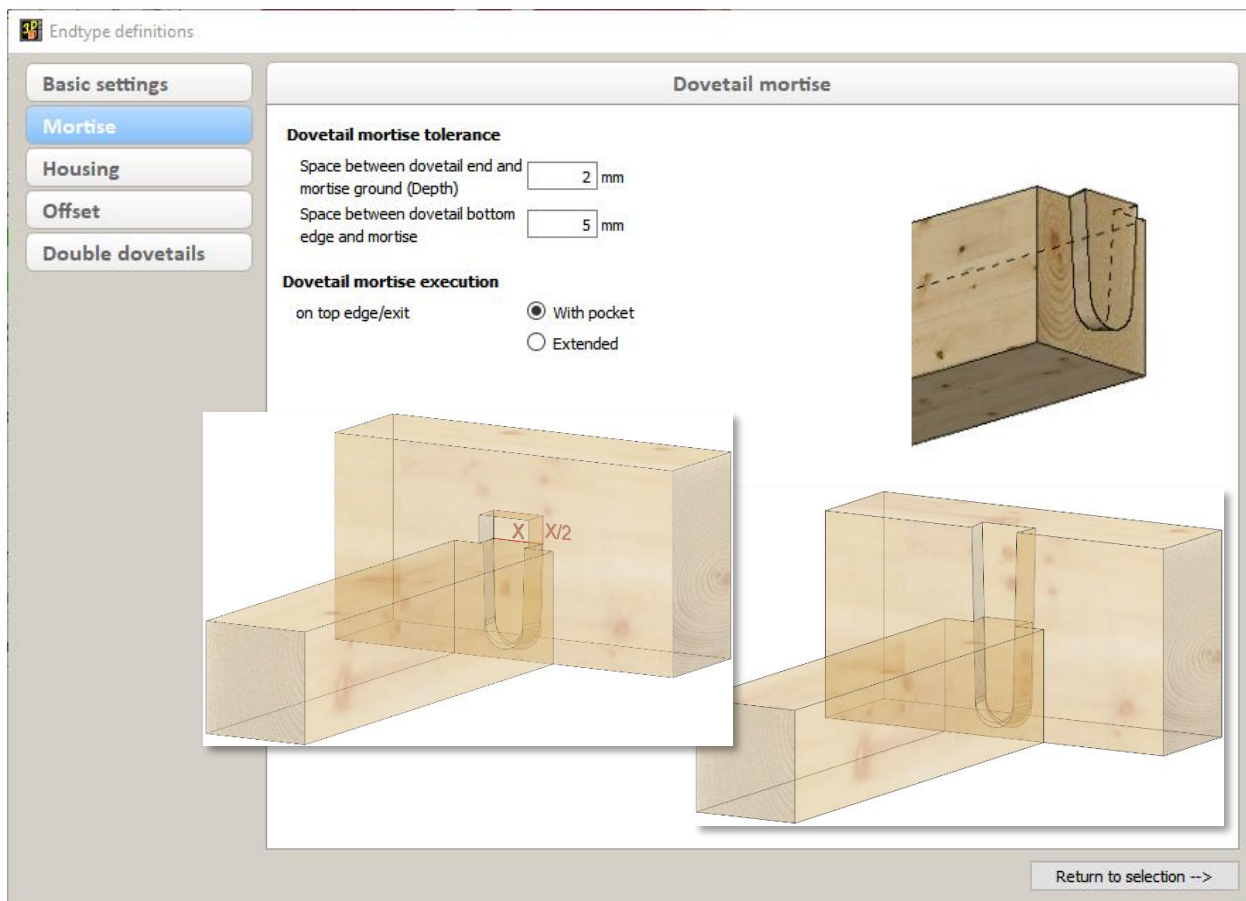
In some configurations the dovetail mortise must be enlarged so that a dovetail can be inserted into its counterpart. The dimensions required for this are usually only known on the machine. For this reason, the BTL, bvx and bvn machine interfaces provide the options "With pocket" and "Extended" for inserting the dovetail.

These options can be found in cadwork in the end-type definitions under *Modify -> Dovetail mortise -> Dovetail mortise execution*.

With the execution "Extended", the dovetail mortise is extended along the taper angle up to the top edge of the receiving part. Under certain circumstances this can lead to very long processing e.g. lintel to mullion connection.

With the execution "With pocket", a rectangular pocket is milled out above the dovetail mortise just high enough for the dovetail to be inserted and slid into place. How high and wide this pocket is ultimately produced is decided by the respective CAM system. The dimensions of this installation pocket on top of the dovetail mortise can therefore not be displayed realistically in the viewer mode, in the shop drawing export and in the individual control under machine export.

In cadwork, the height of the pocket corresponds to half its width.





▪ *Userprofile (user's settings)-> Timber... -> End types... -> Tenons*

For better usability, a new note has been added for the tenon in the end-type dialog. This message appears as soon as you have set that drilling axes are to be created that are not linked to the end-type. This is intended to reduce the risk of this option being set unintentionally. The advantage of non-coupled drilling axes is that they can also be modified in 3D afterwards, as they behave identically to drilling axes that have been created manually.

The major disadvantage, however, is that the drilling axes do not react to any changes to the tenon. If the part with the tenon is moved, copied or deleted, for example, the drilling axes remain unaffected. If you are not aware of this, drilling axes can quickly remain in the wrong positions in the project and/or be missing in other places. If this only becomes apparent later, it is very difficult to clean up the project. To avoid this problem, it is recommended to work with linked drilling axes for the tenon. However, if you are aware of how the non-linked drilling axes behave, there is of course no reason not to use them.

**Endtype definitions**

**Basic settings**

- Mortise
- Backcut
- Haunched tenon
- Housing
- Offset
- Double tenon
- Drillings/pegs**
- Tenon execution

**Drillings/pegs**

**Drillings/pegs**

- No drillings
- Single row drillings
  - Distance to end face (Y)  mm
- Grid drillings

**Quantity of drillings**

- One drilling
  - Centered
    - Third if angle is over: (centered if not)  deg
    - Defined location  mm
  - Two+ drillings
    - Quantity of drillings
    - Equally divided
    - Defined location
    - X1  mm X2  mm

**Drilling set based on**

- Tenon dimensions
- End face

**Drillings execution**

- Only drillings
  - Draw bore  mm Diameter  mm
  - Diameter  mm
- Drilling as drilling axis **!**
  - Link axis to end-type
- Drilling as std connector axis

**Caution!**

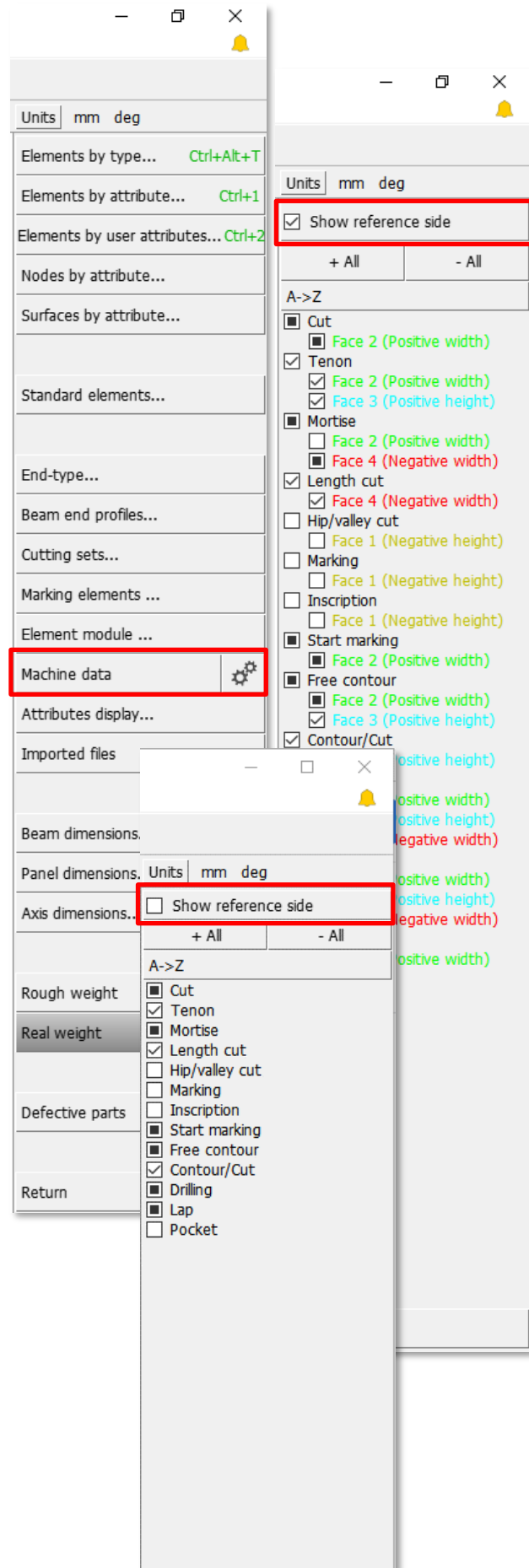
If you use the option **Drilling as drilling axis** without using the **Link axis to end-type** option, drilling axes are created that are independent of the end-type. These drilling axes can be subsequently modified in 3D, but they no longer react to modifications in the end-type dialog.

In order to avoid additional drillings being created with every change in the end-type dialog, those drilling axes are only created when attaching the end-type to a component initially, but not if you modify the end-type afterwards. If you still want to create the drilling axes again, you must first remove the end-type from the component and then attach it again.

If you want to create drillings that always react directly to modifications in the end-type dialog, use the option **Link axis to end type**.

## ➤ Right menu

- *Activate attribute -> Machine data*  
The function *Activate attribute-> Machine data* activates all parts for which machine data has already been calculated. The settings button can be used to differentiate which processes the parts to be activated should contain. If parts are already active, calculated processes are switched on for these parts. parts are activated or deactivated by switching them on or off. In version 30 (SP2024) there is an additional setting *Show reference side*. If you activate this option, the machine processes calculated in the design are further subdivided into the determined machining reference sides. This information is helpful if accessibility is restricted during production, for example machining from below (Side 1) on a panel that cannot be turned. The colors and numbers (Side 1, Side 2, ...) of the displayed reference sides correspond to those in the individual control of the machine export. The designations (negative height, positive width, ...) refer to the local axis system of the parts.

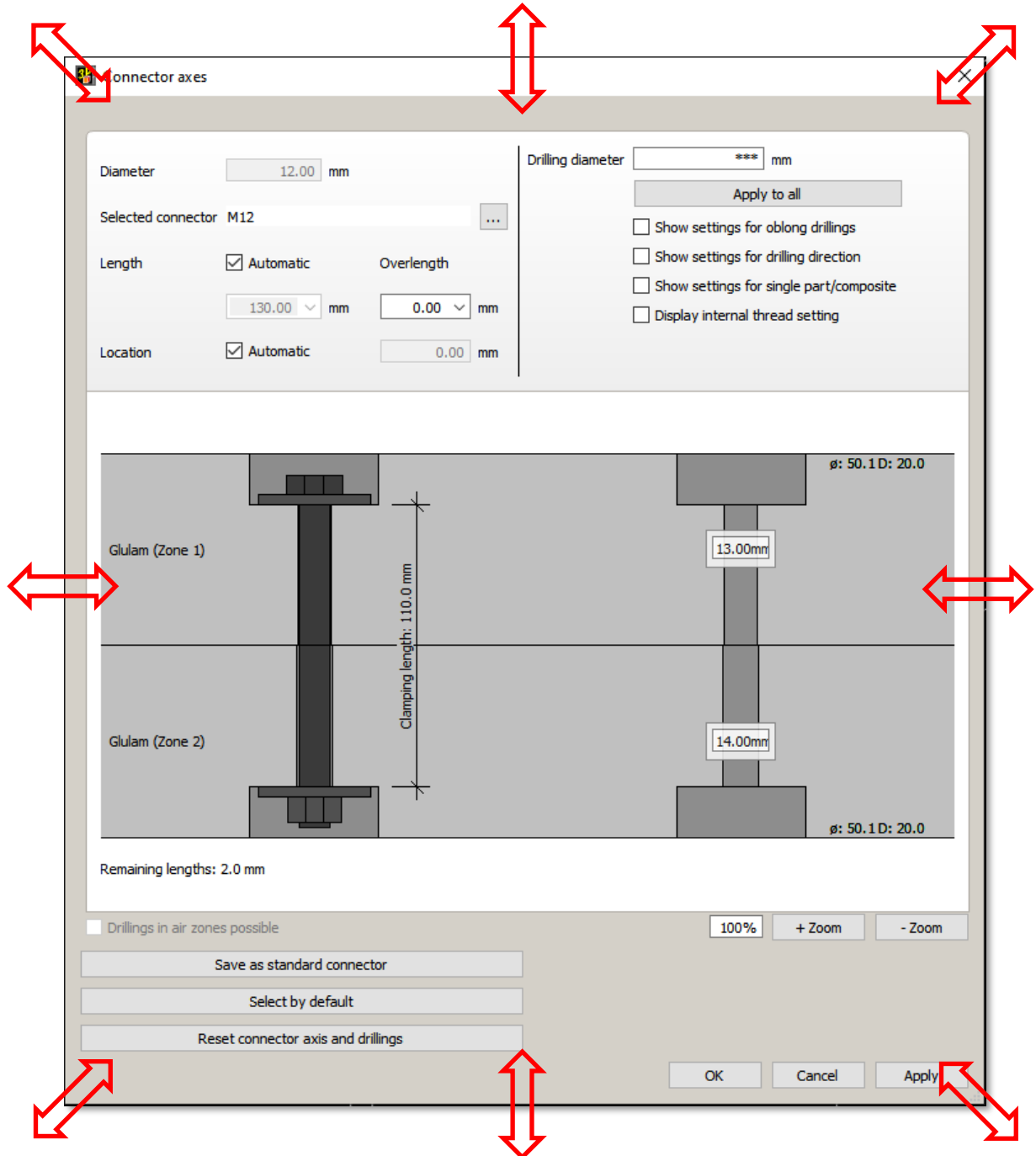


- Add -> Axes... -> Connector axes

The option introduced in version 28 to automatically determine the axis length in the 2D plane using the keyboard option <E> Active elements is now active by default as soon as the function with active elements is called.

```
(L)1. Point (M)Cursor (R)Return I:Intersection M:Mid-point X:X auxiliary line Y:Y auxiliary line D:Distance E:Active elements V:Element in foreground  
Add connector 1 point and length
```

The size of the connector axes dialog could previously only be adjusted at the bottom right-hand corner. Especially when working with multiple screens and different screen resolutions, it may not have been possible to reach this corner. The dialogs can now be resized across all edges and corners. This means that the size can be adjusted at any time, regardless of the current position of the dialog on the screen.

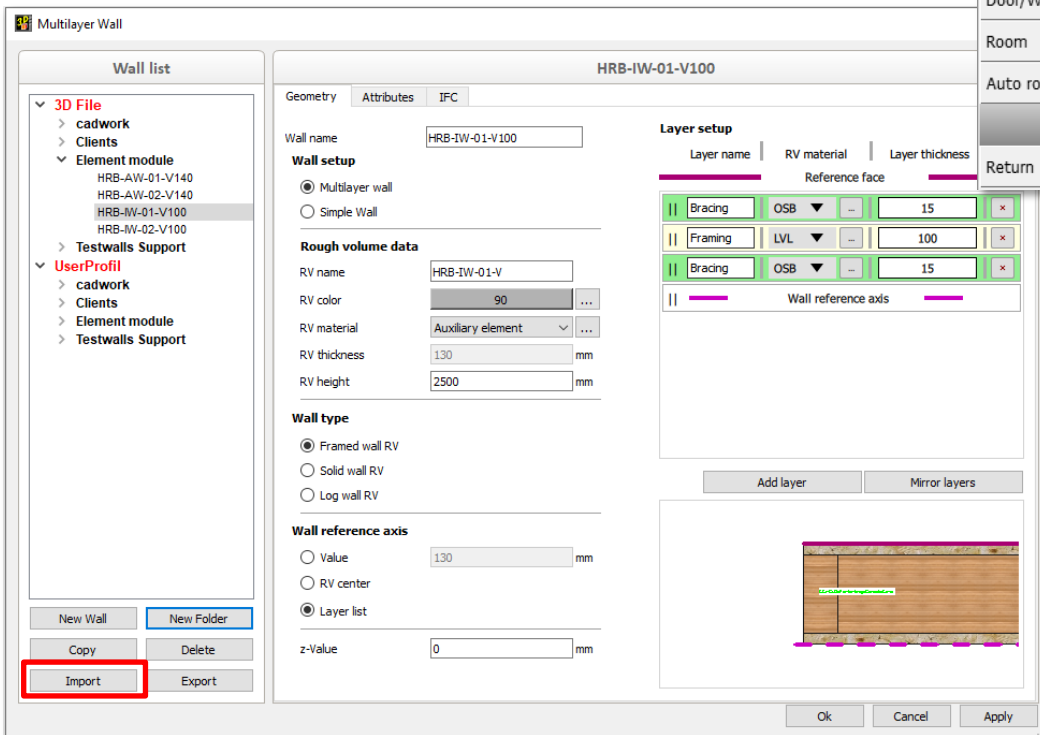


In order to be able to recognize possible collisions with the bolt at an early stage, wooden plugs are made slightly transparent.



- **Add -> Architecture -> Wall**

With cadwork version 30 (SP2024), this menu item is displayed in purple. This means that this menu item will be removed in future versions. In the future, the use of multilayer walls will be changed. It is already possible to convert 2dc walls into multilayer walls. The "Import" option is available for this purpose in the configuration dialog for multilayer walls.



- *Add -> Architecture -> Multilayer wall*

Refinements have been made to the "Multilayer walls" introduced in version 30. This affects the display of multilayer walls in cadwork 3D as well as working with the dialog for defining multilayer walls.

Previously, the wall layers in the wall rough volume were displayed as planes.

From Service Pack 2024, each layer is displayed as a volume. The transparency defined for the wall rough volume also affects the display of the wall layers.

If wall layers are shown, they are displayed in the set transparency and the wall rough volume is displayed completely transparent. If wall layers are hidden, the wall rough volume is displayed with the set transparency.



Version 30 without transparency



Version 30 (SP2024) without transparency

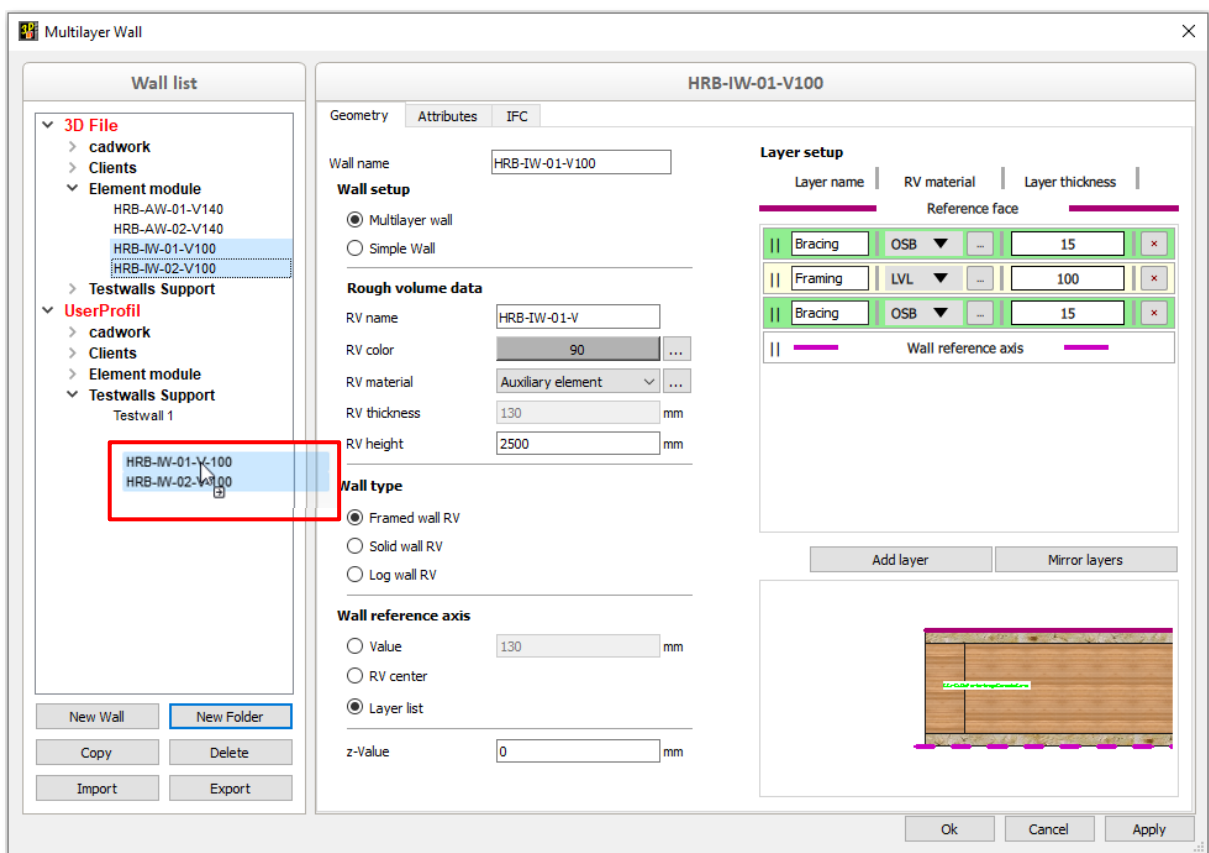


Version 30 (SP2024) with transparency

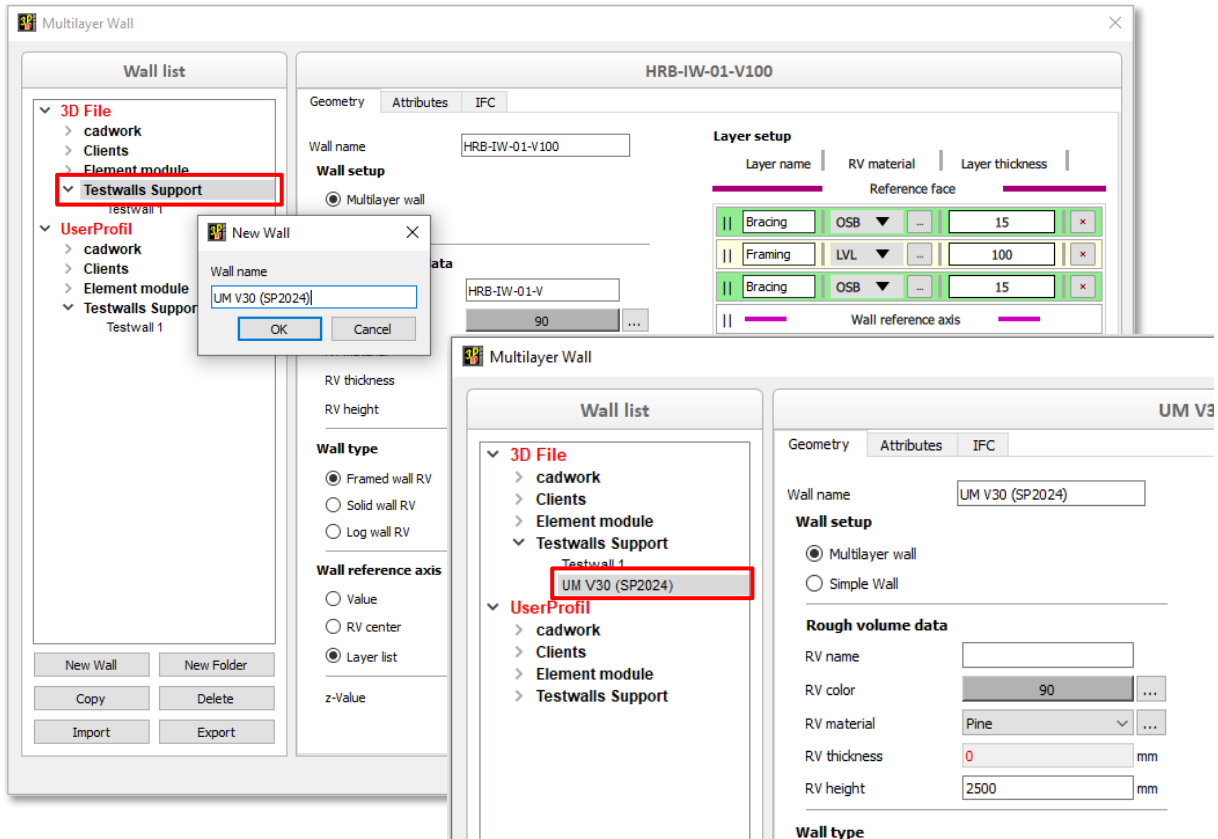
The changes in the dialog affect the behaviour of the left-hand area in which the list of existing multilayer walls is displayed. Two improvements have been made here.

When copying multilayer walls from *3D File* to *UserProfil* via "drag and drop", it was previously the case that the wall had to be dragged exactly to the *UserProfil* or to a folder in the *UserProfil* when moving it.

This has been adjusted for easier handling. The mouse button can now be released at any point in the *UserProfil* tree. The copy in the *UserProfil* is created exactly at this point in the *UserProfil*.



Furthermore, it was previously the case that when a new folder was created, it was always created on the top level of the 3D file. Now, the active folder or active wall is always taken into account during creation and the new folder is created at the corresponding position.



- *Add -> Files...*

- *IFC (\*.ifc)...*

News on importing IFC files can be found in the IFC chapter.

- *Revit (\*.rvt, \*.rfa)*

Revit has again released a new version (Revit 2024) in 2023. We have expanded our interface so that we now support this version. This means that currently Revit versions 2014 to 2024 can be imported.

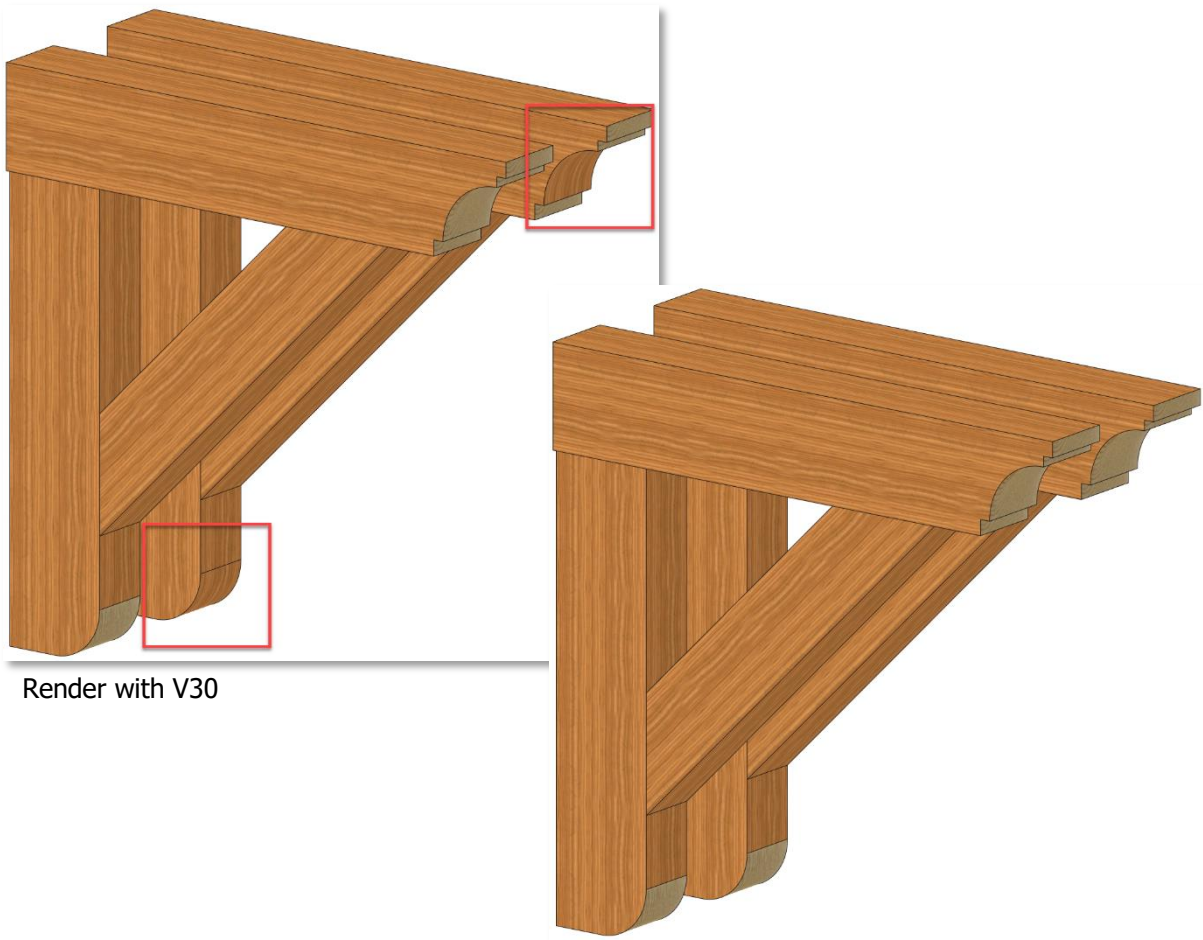
We import Revit files in \*.rvt and \*.rfa format with the help of an external tool. We will provide the next Revit version, which is expected to be released in 2024, as soon as the external tool supports it. Based on our experience in recent years, we expect this to happen at beginning of 4<sup>th</sup> quarter in 2024.

- *Point cloud (\*.pts, ...)*

News on importing Point clouds can be found in the Point cloud chapter.

- *Copy/Move -> Symmetry*

For each rendered representation, the different textures must be used correctly parallel and transverse to the wood grain. In some situations, the more complicated geometry of beam end profiles led to the longitudinal texture being displayed on end grain surfaces when moving and copying symmetrically.



Render with V30

Render with V30 (SP2024)

With the help of the updated "Hoops Visualize" graphic engine, we were able to eliminate this problem in Service Pack 2024. The end grain texture is also displayed for each end grain surface.



- *Modify -> Other attributes...*

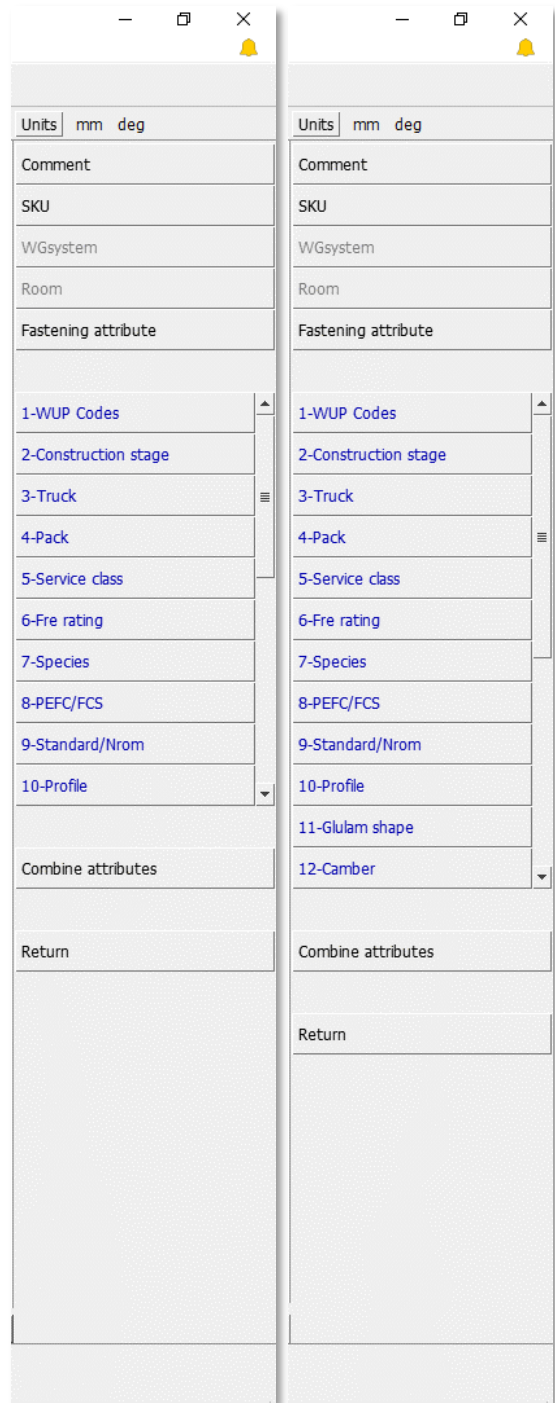
Since the number of user attributes is no longer restricted, they are increasingly being used to structure the design.

This means that the number of user attributes used is generally much higher than 10, as was previously possible.

In Modify, up to version 30, a maximum of ten attributes were displayed in the list. To access the remaining attributes, the scrollbar that was then available had to be used.

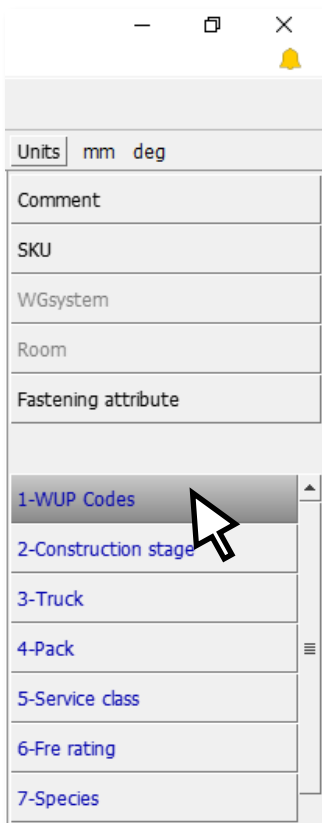
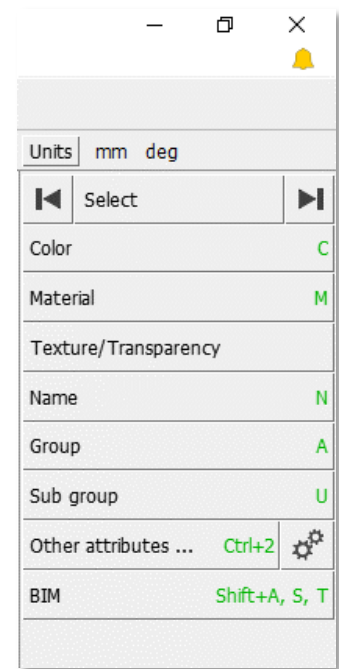
With service pack 2024, the entire available space in the right-hand menu is used to display defined user attributes. The space for the display depends on the screen resolution used.

A scrollbar is only displayed if the number of user attributes used cannot be displayed in the available space. This can be used to access the other attributes.

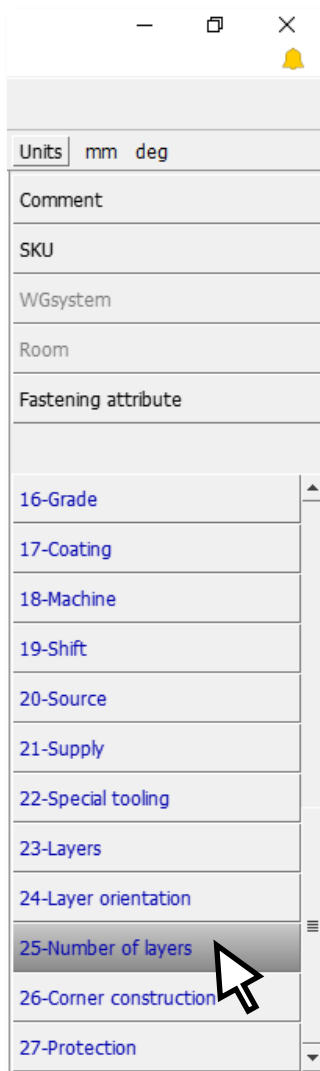


The increased use of user attributes means that they need to be modified more often. An additional shortcut key has therefore been provided in Service Pack 2024 to enable quick access to the modification of user attributes.

After modify has been called up via the <1> shortcut key, the <Ctrl+2> key combination can be used.



The user is then at the top of the selection list of user attributes (user attribute 1).



The attribute to be modified is quickly selected via a further keyboard entry. For example, entering <2> + <5> in quick succession places the cursor on user attribute 25. A (L) mouse click activates this attribute for modification.

▪ *Export*

- *Machine -> ... -> Individual control manager*

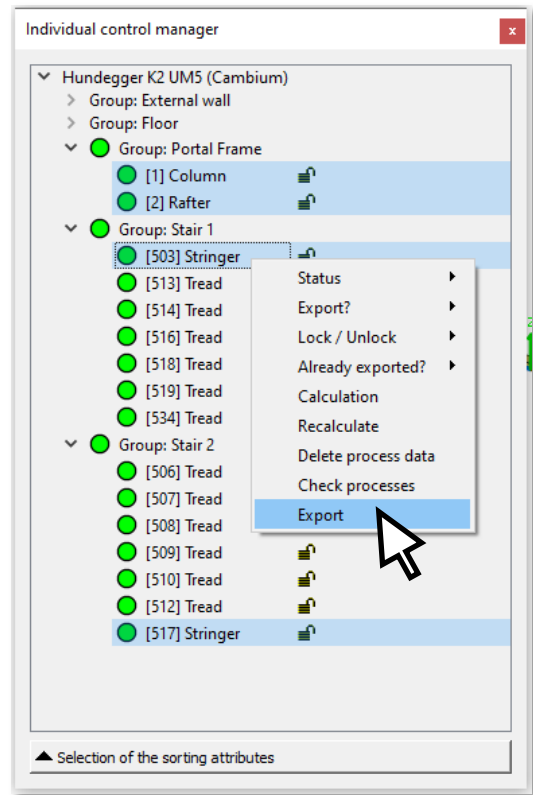
◆ Selective export of machine data

In the individual control manager, it is possible to select several names within the tree, provided you are in the menu immediately before the machine individual control.

To make a selection, hold down the <Ctrl> key while clicking on the individual names in the individual control manager with the left mouse button (L).

Then right-click (R) to open the context menu. Select the new entry *Export* (only available after calculation) and the export dialog will open.

The export that is then started will only transfer the selected items to the corresponding machine file. This makes it easy to transfer individual parts and groups independently of their sorting in the individual control manager.

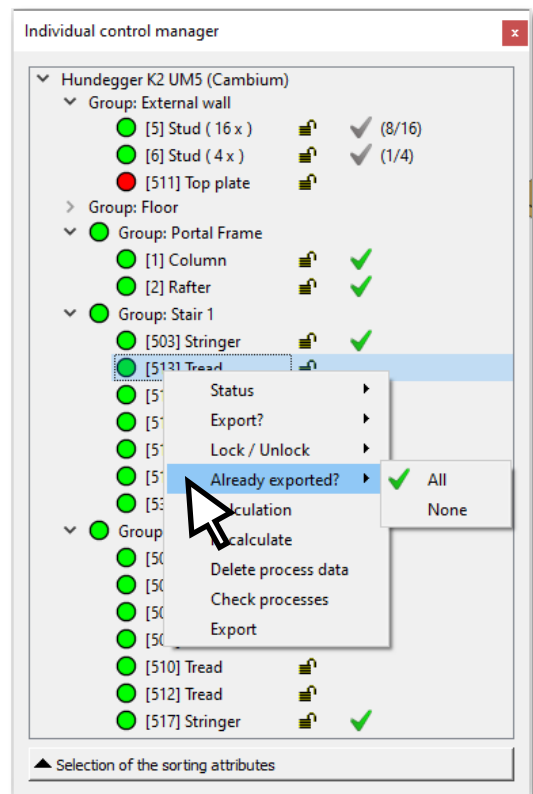


◆ Exported parts

As soon as certain items have been exported to a machine file, the corresponding entries are marked with a green tick in the individual control manager.

This marking can also be set or removed manually. To do this, you will find the entry *Already exported?*.

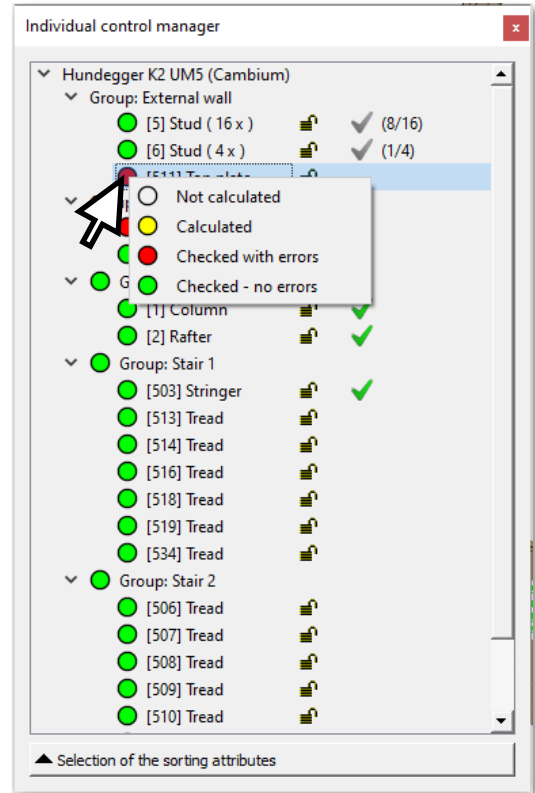
If only a partial quantity of an item that contains several components has been exported, a grey check mark appears here, along with an indication of how many components of this item have already been exported.



- Machine -> ... -> Individual control / Individual control manager

- ◆ Change component status

To change the status of a component, you can now also click directly with the left mouse button (L) on the circular status symbol. This opens the selection menu for the various statuses such as *Not calculated*, *Calculated*, *Checked with errors*, *Checked - no errors*. If the processing check reports errors for a component, you can change the status to *Checked - no errors* after a successful visual inspection.



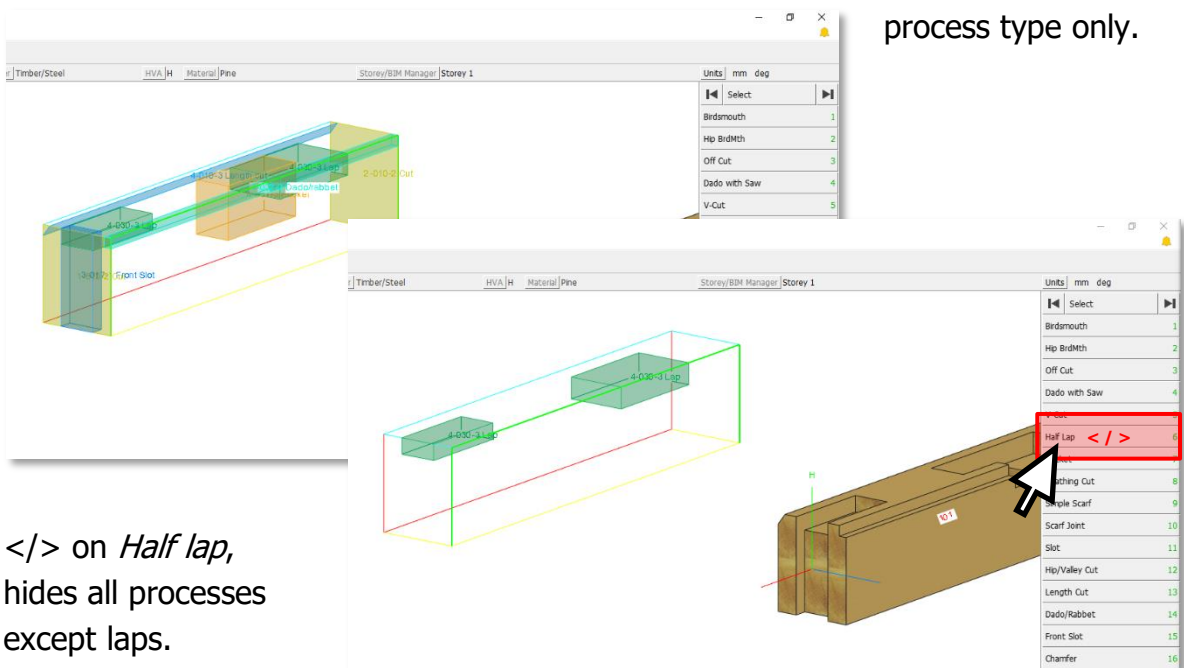
- ◆ Shortcuts for calculated processes selected in the individual control manager

In the individual control manager, you can perform the following actions for the processes currently selected in the tree structure using the usual shortcuts:

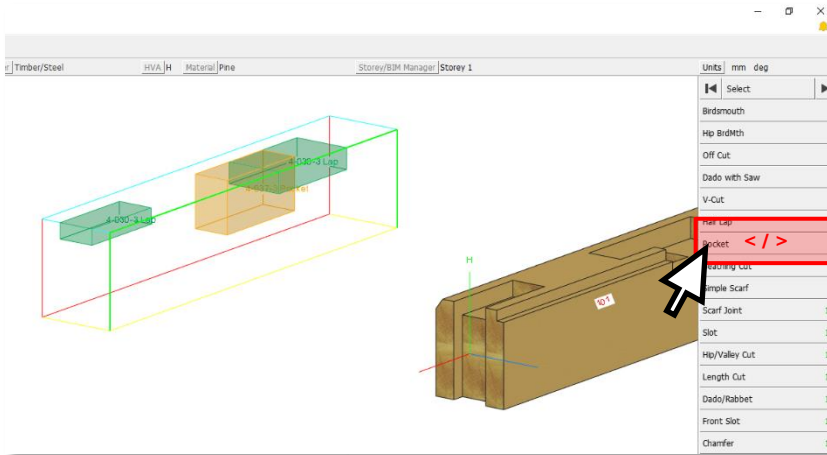
- ▶ </> Hides all but the selected processes
- ▶ <\*> Hides the selected processes
- ▶ <J> Hides all processes
- ▶ <Delete> Deletes the selected processes

- ◆ Combining shortcuts for visibility

As soon as the mouse pointer is positioned on a process button in the right-hand menu, the shortcuts </> or <\*> can be used to show or hide processes for this process type only.

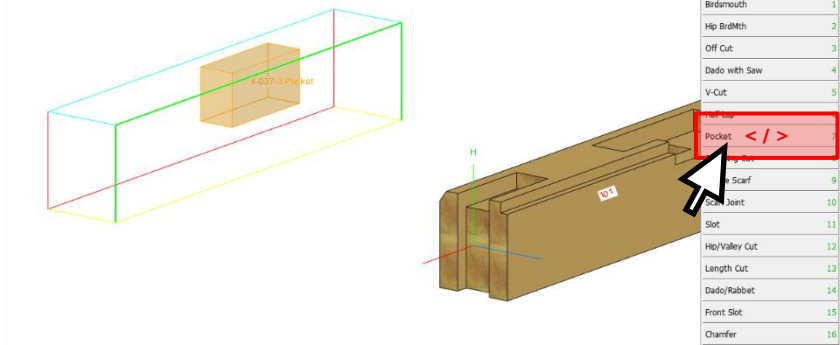


</> on *Half lap*, hides all processes except laps.



From version 30 (SP2024), pressing the </> key now on the *Pocket* button will also show all pockets in addition to the laps.

Only repeated execution of the </> key on the Pocket button will only show pockets.

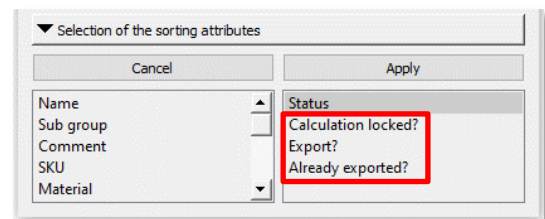


With this procedure, any combination of processes can be added with </> and hidden with <\*>. Only with a double execution: </>+</> or <\*>+<\*> are only the processes of this process type shown or hidden again.

As usual, all processes are visualized by pressing <J>. Pressing <J> also interrupts the combining of visibilities.

◆ **Sorting and combining**

In addition to *Status*, the attributes *Calculation locked?*, *Export?* and *Already exported?* can now be used to sort and combine parts in the individual control manager.



◆ **Info window in the individual control**

The *Subgroup* attribute has been added to the info window of the individual control. The subgroup is often used as a further distinguishing attribute for production.

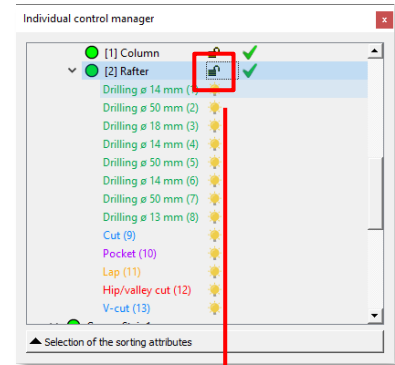
Machine type	Hundegger K2 UM5 (Cambium)
Machine calculation set:	Standard_Hundegger K2 UM5 (Standard)
No. production list	524
Quantity	1
Name	Stud
Group	Internal wall
Sub group	Iw - Light wall
Material	PTNE
Real width	45 mm
Real height	90 mm
Real length	2400 mm
Rough width	45 mm
Rough height	90 mm
Rough length	2400 mm
current processes:	8

- ◆ Shortcut key for locking the parts in the individual control

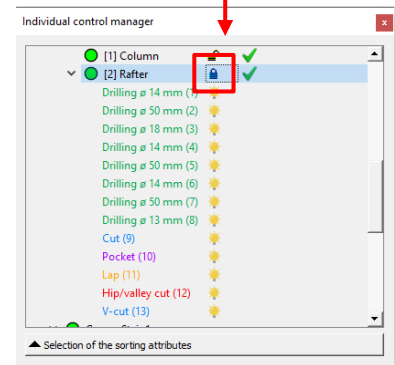
Locking a part in the individual control manager (lock symbol) prevents the machine data from being changed. Locking is useful after successfully checking the generated machine data. As of service pack 2024, you can use the `<AltGr+L>` or `<Ctrl+Alt+L>` (US keyboard) shortcut key to *Lock* or *Unlock* a part in the individual control.

This means you can if desired only use shortcut keys to check position by position in the individual control check:

- ▶ `<C>` Trigger processing check
- ▶ Check result
- ▶ `<AltGr+L>` or `<Ctrl+Alt+L>` (US keyboard) Lock component
- ▶ `<Page down>` Show next position...



`<AltGr+L>`



If a part is locked, you will receive an error message when executing an action that is locked as a result.

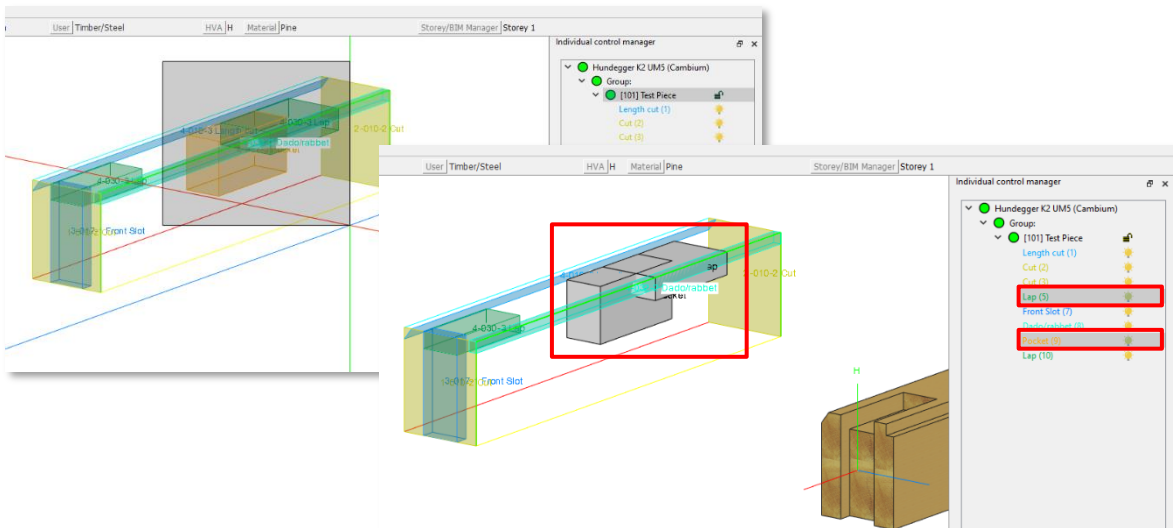
The current element is locked for machine data modification!  
`<Return>` to continue

In addition, functions that are not permitted are greyed out.



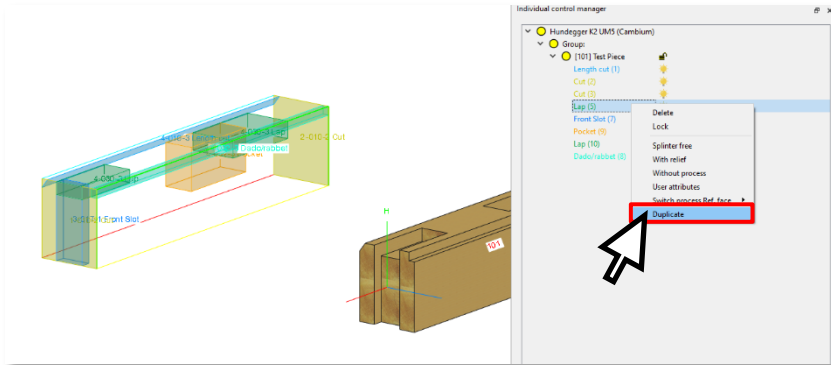
- ◆ Lasso in individual control

From version 30 (SP2024), machine processes can be activated with the lasso. As usual, the machining processes must be captured completely with the white lasso, whereas the red lasso enables activation by incomplete selection of the processes.



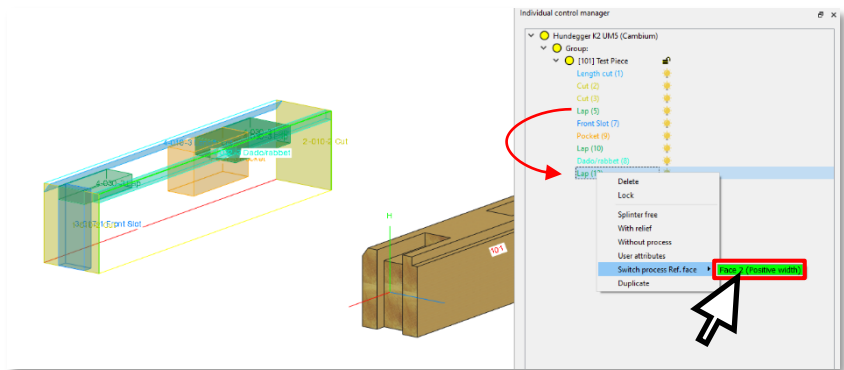
◆ Duplicating processes

The *Duplicate* function has been added to the menu for processes in the Individual control manager. This duplicates the selected process operations.



In a subsequent step, the duplicated processes can be modified.

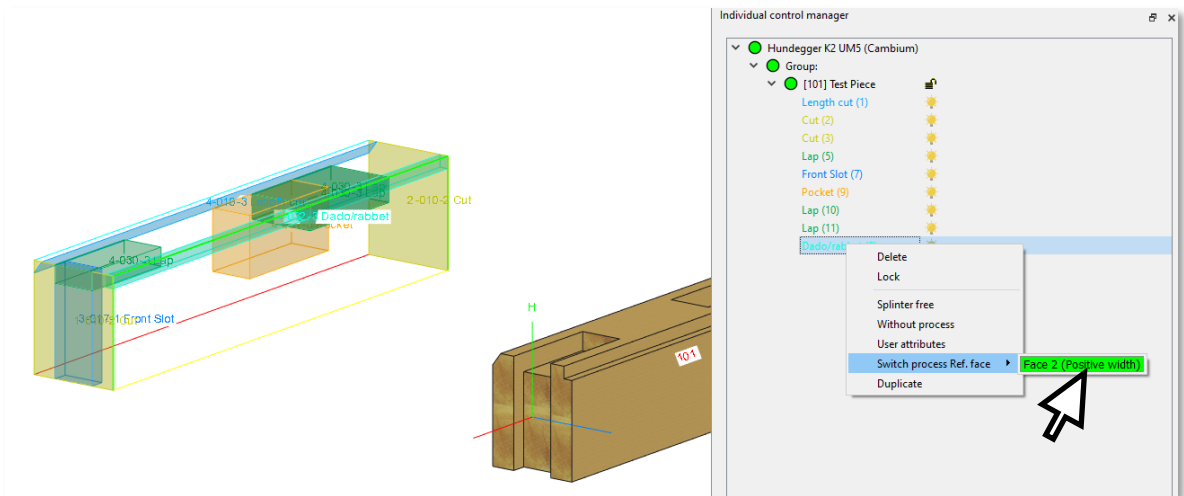
For example, it is possible to clean out rounded corners of a lap by first duplicating the lap and then assigning a different reference side to the duplicate process.



◆ *Switch process Ref. face* for Dado/rabbet

The *Switch process Ref. face* function is also available for Dado/rabbets, making it possible to process a dado/rabbet from a different side.

As with the other processes, this function can be used via the context menu in the

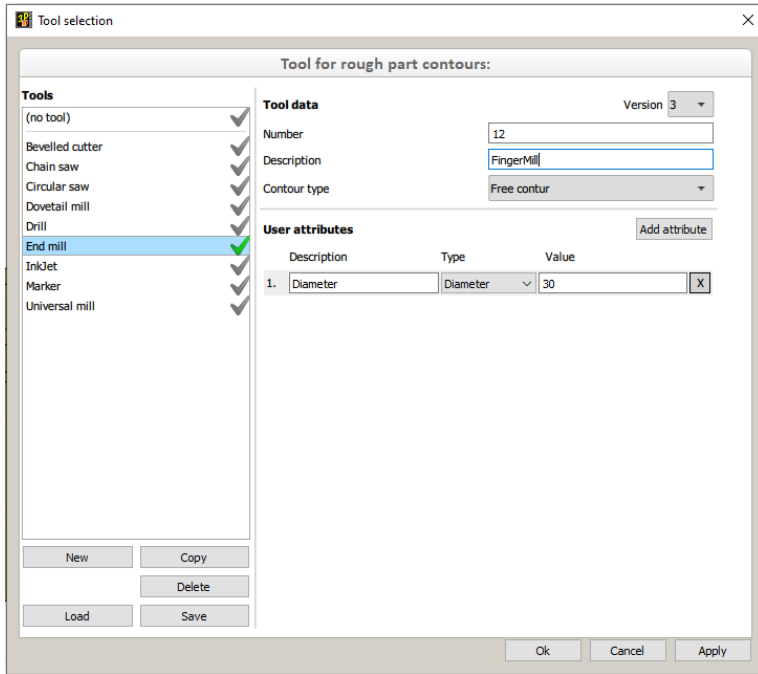


Individual control manager and via the <ALT+R> shortcut key with processes selected in the main window.

◆ Tool settings for contours



A default tool can be stored for the available contour types. The tool query will no longer appear when manual processing is re-inserted.

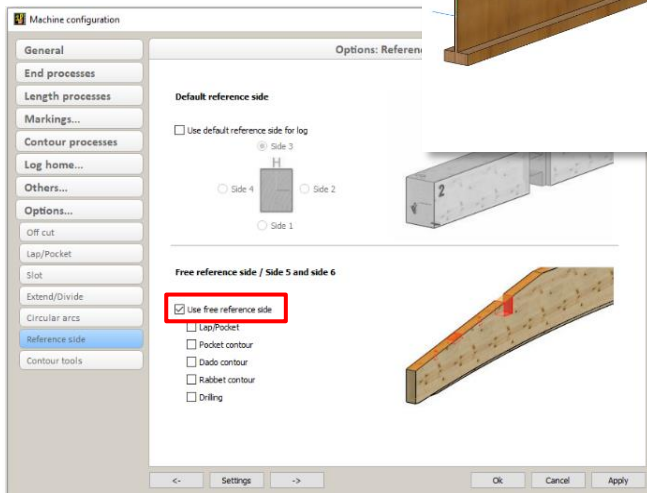


• Machine -> Calculation

◆ Profiles as a silhouette



The cross-section profile of finished profiles (circular beams, I-beams, hollow sections, steel sections, etc) can be calculated as a silhouette from the end face (Ref. 5) and transferred to BTLx as a silhouette (from BTLx2.2) or outline without processing. This information can be used in CAM systems for better collision control.

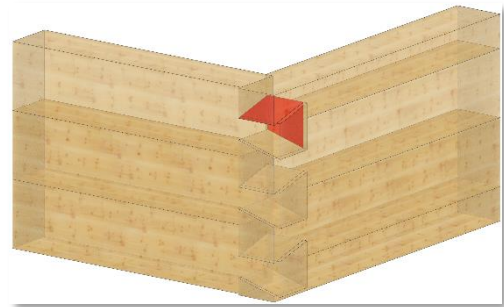


To do this, in the machine configuration dialog activate the *Calculate outline as Silhouette* option under *Options -> Silhouette* as well as *Use free reference side* option under *Options -> Reference side*.



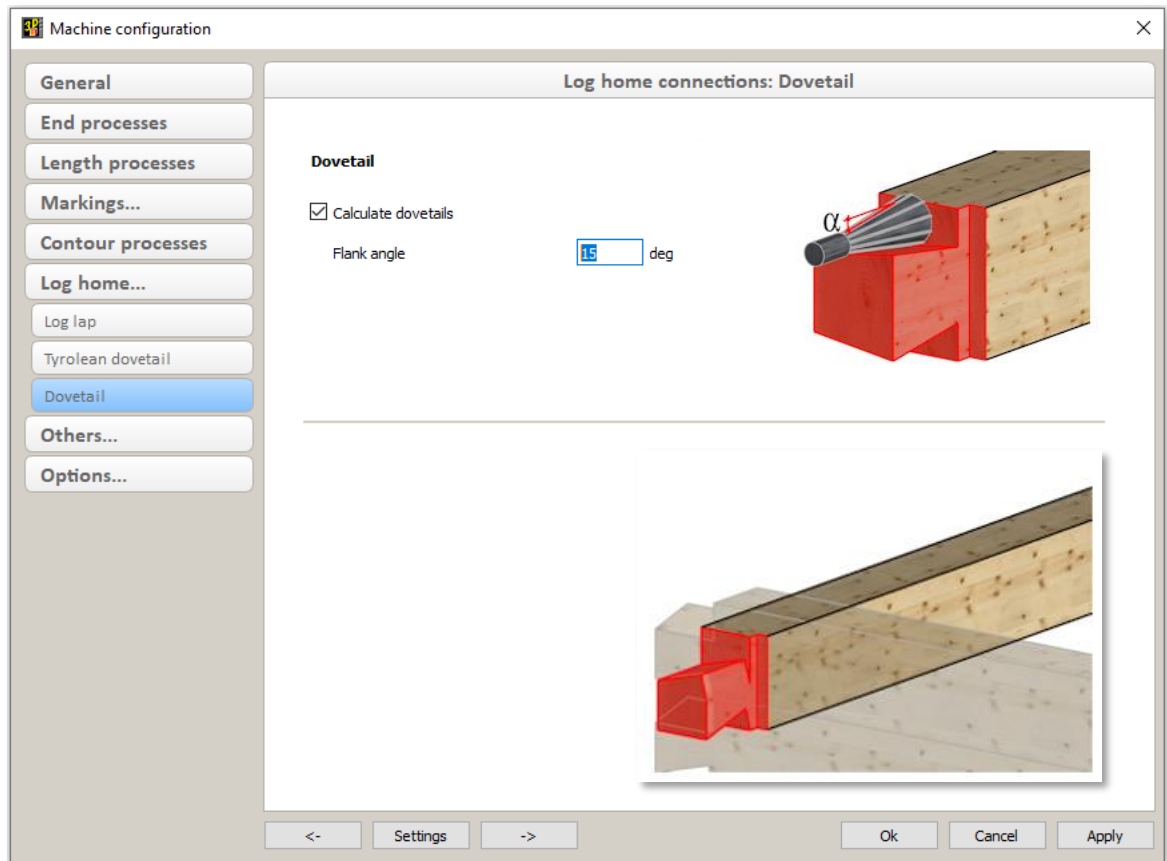
◆ Dovetail Flank angle

The European dovetail is a special form of log home corner lap joint that is designed so that it can be produced using standard tools. This joint consists of a partial hip cut and a birdsmouth.



The angles of inclination of the flashings and the notch must correspond to the set angle. As a rule, the angle is 15°.

In the case of an American dovetail, a special tool with the set angle as the flank angle is required. The connection consists of a partial hip cut and an acute-angled notch.



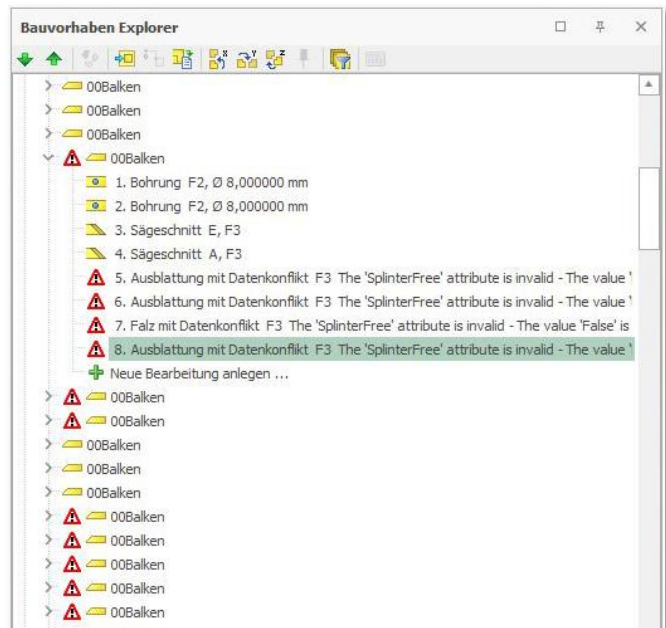
For both shapes, the flank angle set here must correspond to the angle in the configuration dialog *Cut/Solder* -> *Log home...* -> *Dovetail settings dialog*.

- *Machine -> ... -> Export*

- ◆ Hundegger - Consistent BVX2.1 transfer

The BVX2 interface for transferring machine data to Hundegger machines with CAMBIUM® has undergone continuous development in recent years. Specifications have also been changed in the process. Nevertheless, the machine control was quite tolerant and continued to read in BVX data correctly, sometimes with outdated descriptions.

With the latest CAMBIUM® versions, Hundegger has started to check the BVX data received -very strictly for conformity. Components that do not fully comply with the specification are no longer accepted. We have therefore scanned the BVX export for inconsistencies and corrected existing errors. With the current cadwork versions, an export to the latest CAMBIUM®- versions should work smoothly.

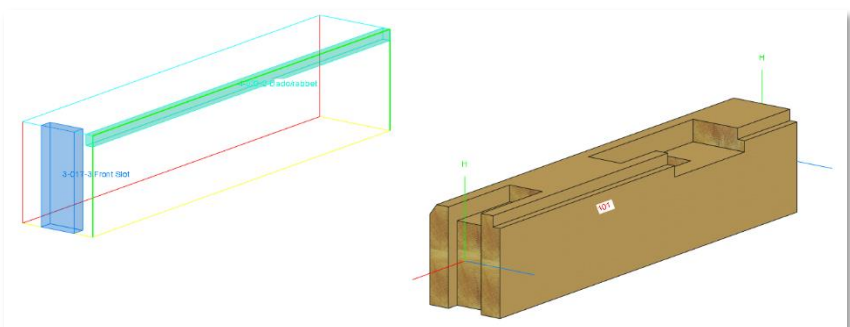


Faulty parts in CAMBIUM®

If you import an old bvx file into CAMBIUM® and find that components are classified as faulty, you must export this file again with a current cadwork version.

- ◆ Dado/rabbet and front slot in BTLx

In the BTLx interface description, there are no dado/rabbet and front slot operations. Although these are calculated in cadwork, they are exported as a lap or pocket.



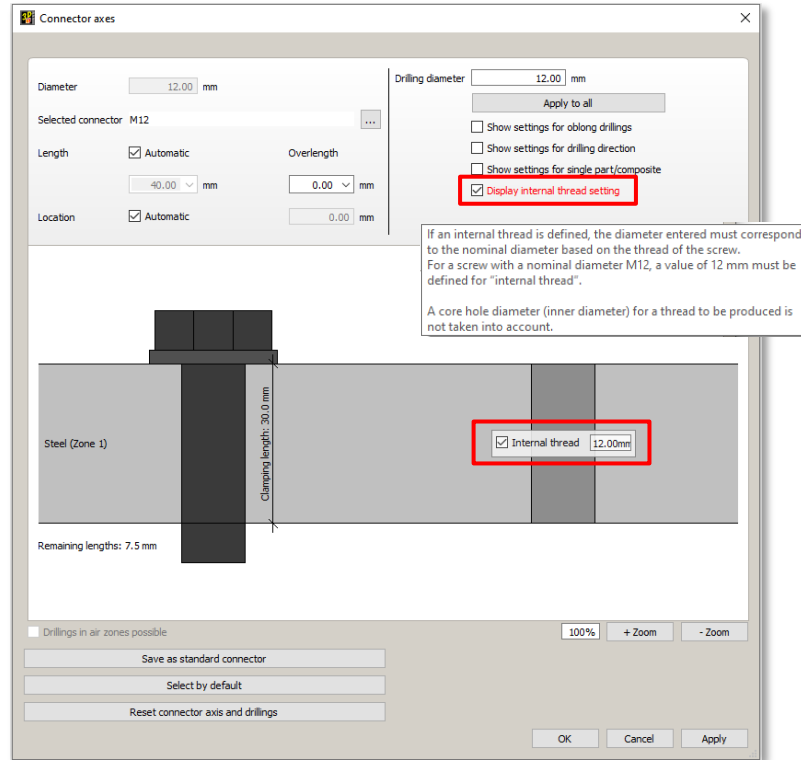
For BTLx exports to the CAM system Lignocam, the processes analysed as dado/rabbet or front slot are exported with an additional attribute:

for dado/rabbet `<UserAttribute Name="LC_PROCESS_TYPE" Value="AsDadoRabbet"/>`

for front slot `<UserAttribute Name="LC_PROCESS_TYPE" Value="AsFrontSlot"/>`.

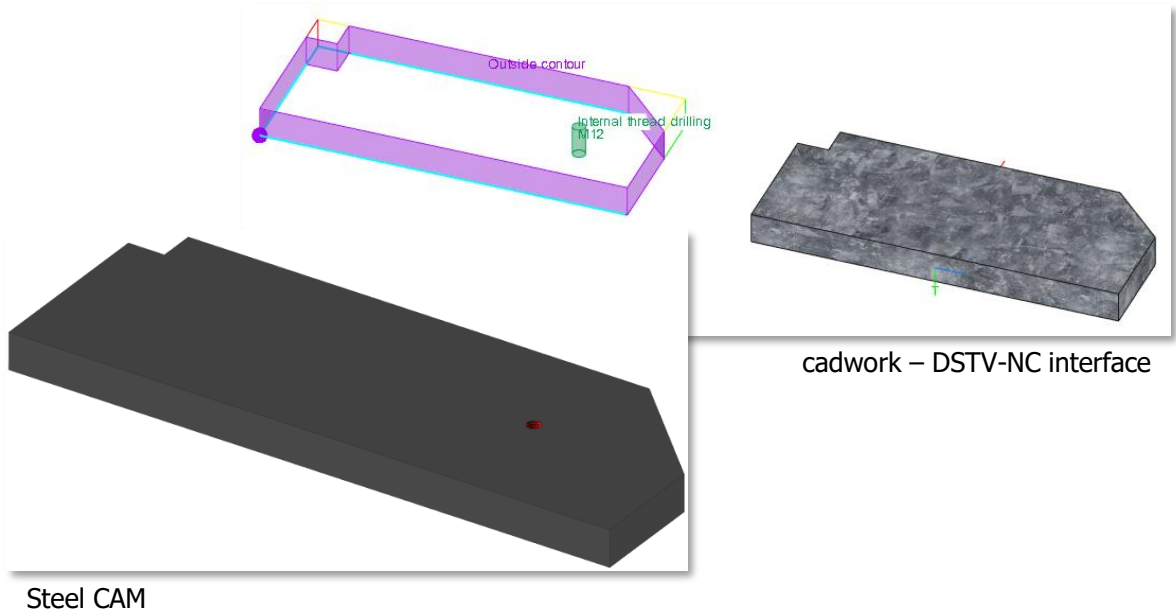
This means that the strategy tables for dado/rabbet and front slot can continue to be used in Lignocam.

- ◆ Export internal thread to DSTV-NC steel  
Since version 30, the setting *Display internal thread setting* in the connector axes dialog can be used to define for each zone independently whether the associated drilling should have an internal thread.



This information is exported for Service Pack 2024 via the DSTV-NC interface. Please note that the nominal diameter must be specified in the case of an internal thread.

The corresponding machining operation receives the designation *internal thread* from the machine data analysis and the nominal diameter (here M12) is displayed instead of the bore diameter.



- *Machine -> Panel prefab...*

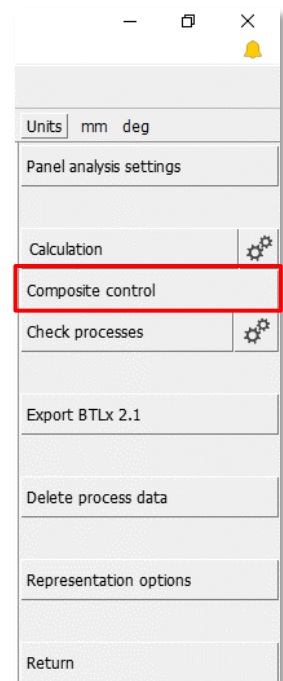
In version 30 (SP2024), the menu item *Individual control* has been renamed to the more appropriate term *Composite control*.

- ◆ Open composite dialog

When opening the *composite control* in panel prefab, the composite dialog is now always opened in version 30 (SP2024). It is no longer relevant which part of an element is activated or selected. The first layer of the selected element is always opened in the composite dialog.

However, it is essential that the machine data has been calculated so that all information is available for display in the composite dialog. If the data is not available, you will receive an error message and the composite control will not open.

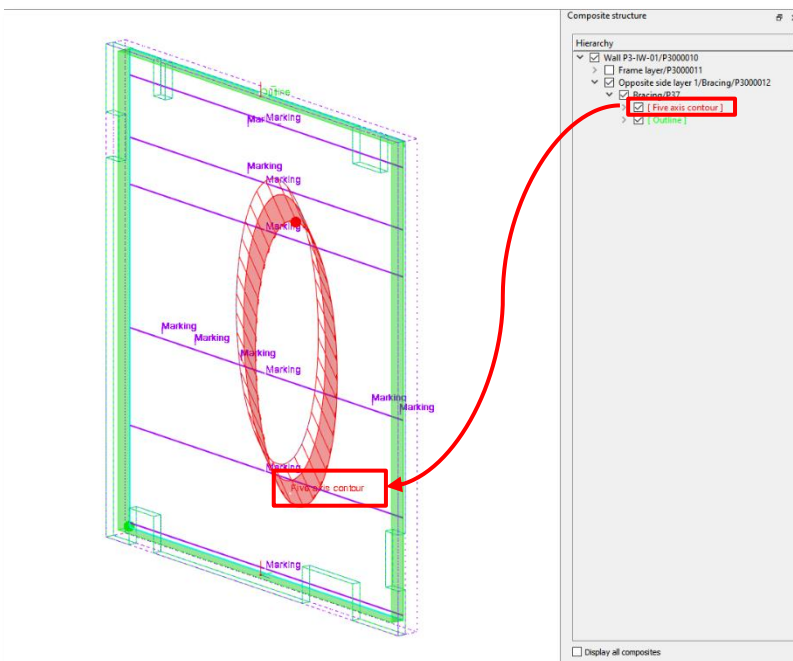
The dialog can be closed and reopened with the short cut key <Q>.



Ctrl+R:Reference face color    Q:Composite Dialog    Ctrl+Alt+L:Lock part

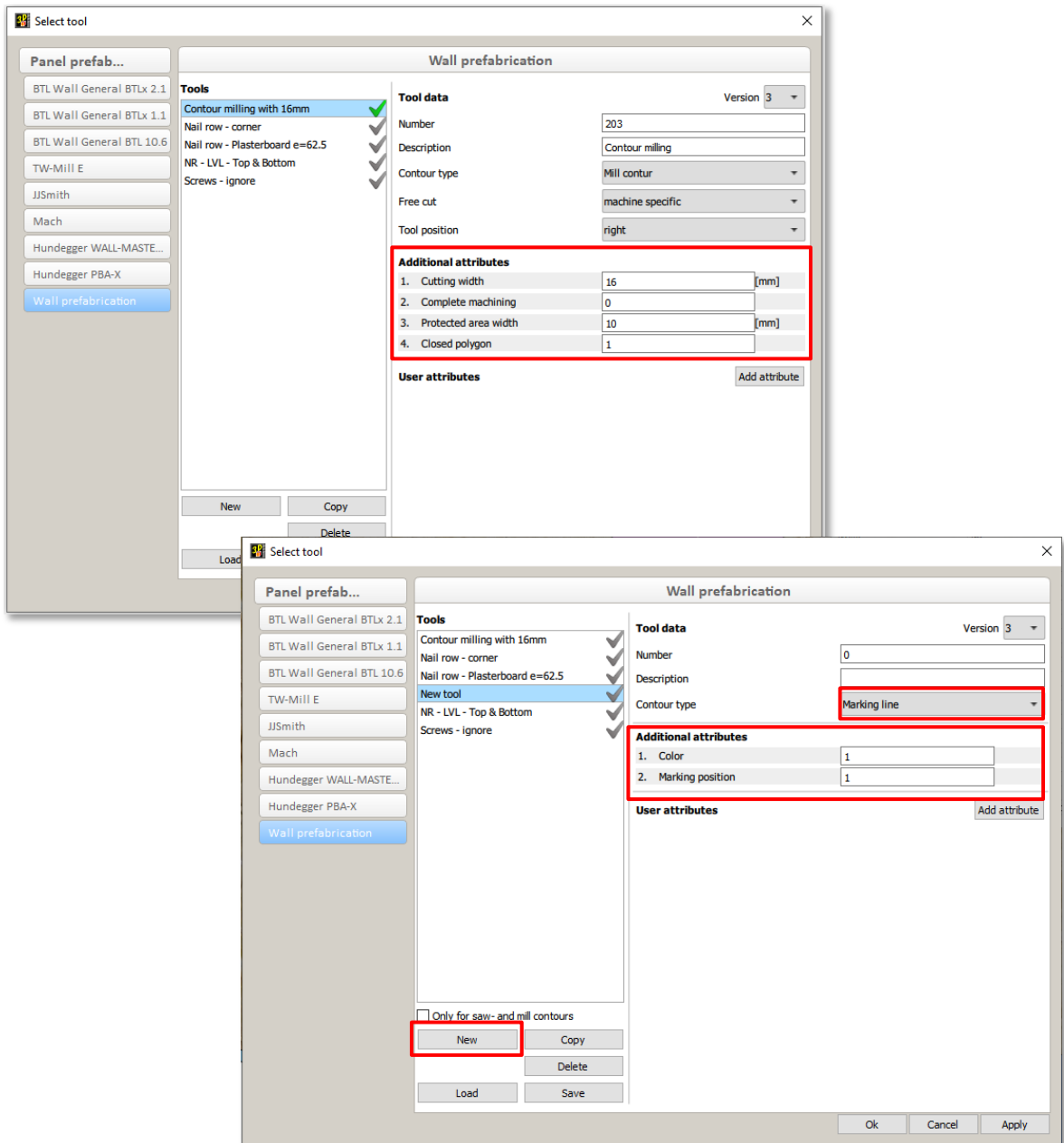
- ◆ 5-axis contour in panel prefab

5-axis contour processing is enabled in Service Pack 2024 for machine exports in panel prefab production. The processing can either be picked up manually in a layer (frame layer, reference side or opposite side layer) on an additionally displayed part or it is calculated automatically as process of a and can then be moved to a layer.



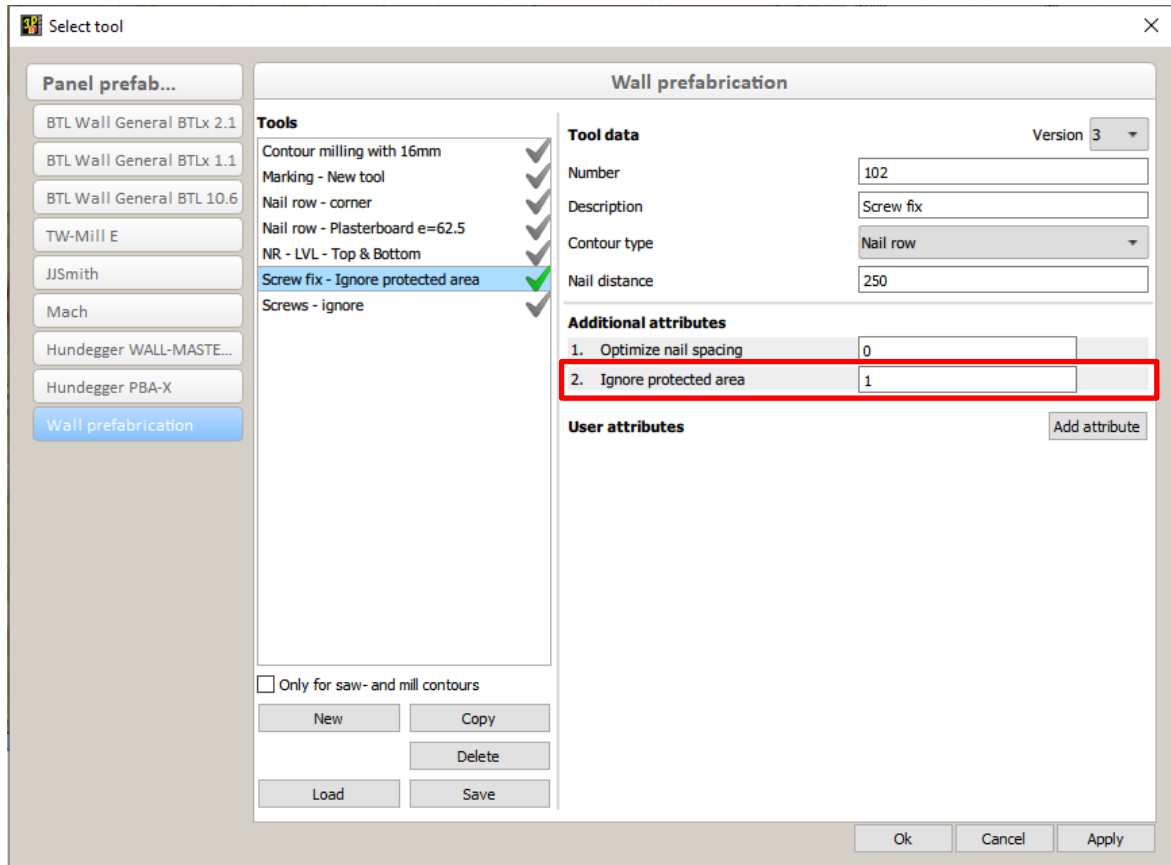
### ◆ Panel prefab tools

Tools are used for manual processing with the *Rectangular MEP* axes on the composite or for *cutting, gluing* and *insulation* in the panel analysis settings and can be created and configured in the tool dialog. Depending on the contour type, further properties are available in the *additional attributes* area. To create a new tool, the tool data for the desired processing type previously had to be loaded from the Userprofile using tool files or existing tools had to be copied. It was no longer possible to change the contour type at a later date. With cadwork version 30 (SP2024), you have the option of creating tools for manual processing in the tool dialog with the *New* button and subsequently changing the *contour type* and thus the type of processing, for example from *nail row* to *saw line*. The available *additional attributes* of the selected contour type are automatically adjusted.



- ◆ Additional attribute *Ignore protected area* for manual nail rows

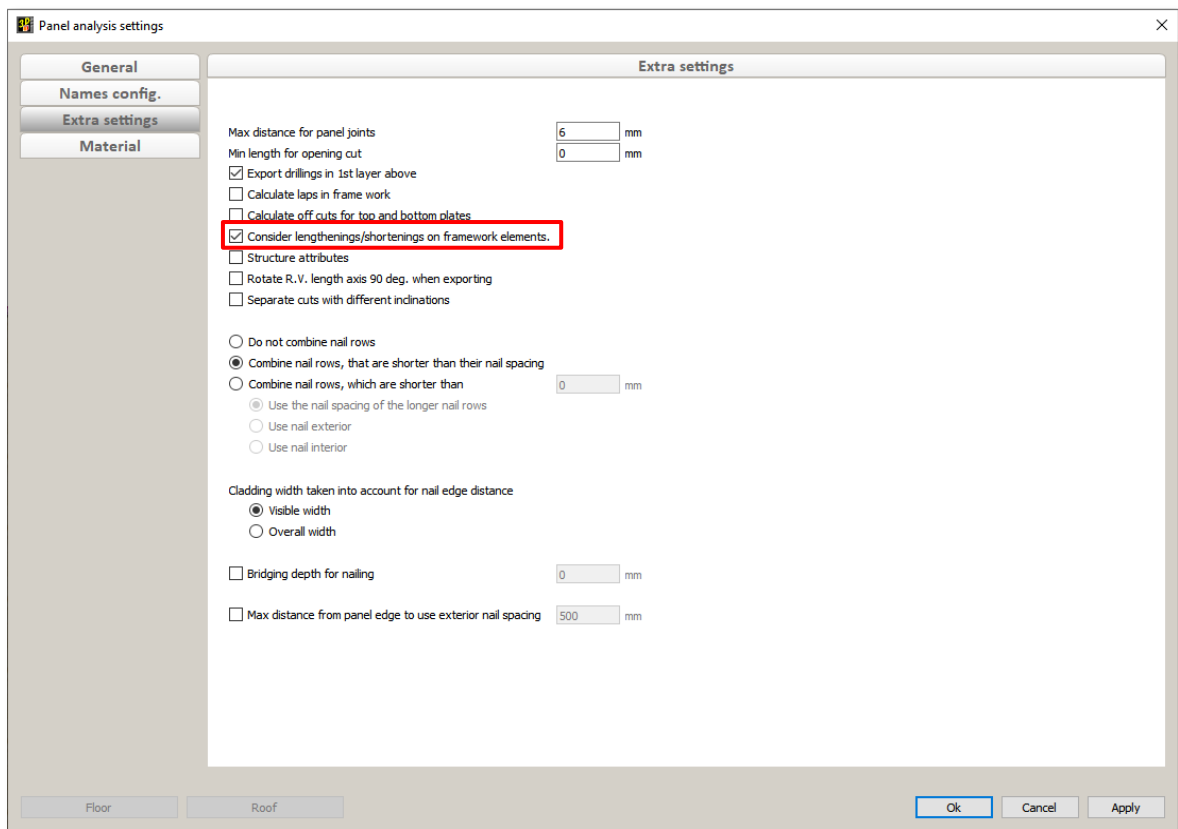
For manually defined nailing rows using the *Rectangular MEP axis*, cadwork version 30 (SP2024) provides the additional attribute *Ignore protected area* in the tool dialog.



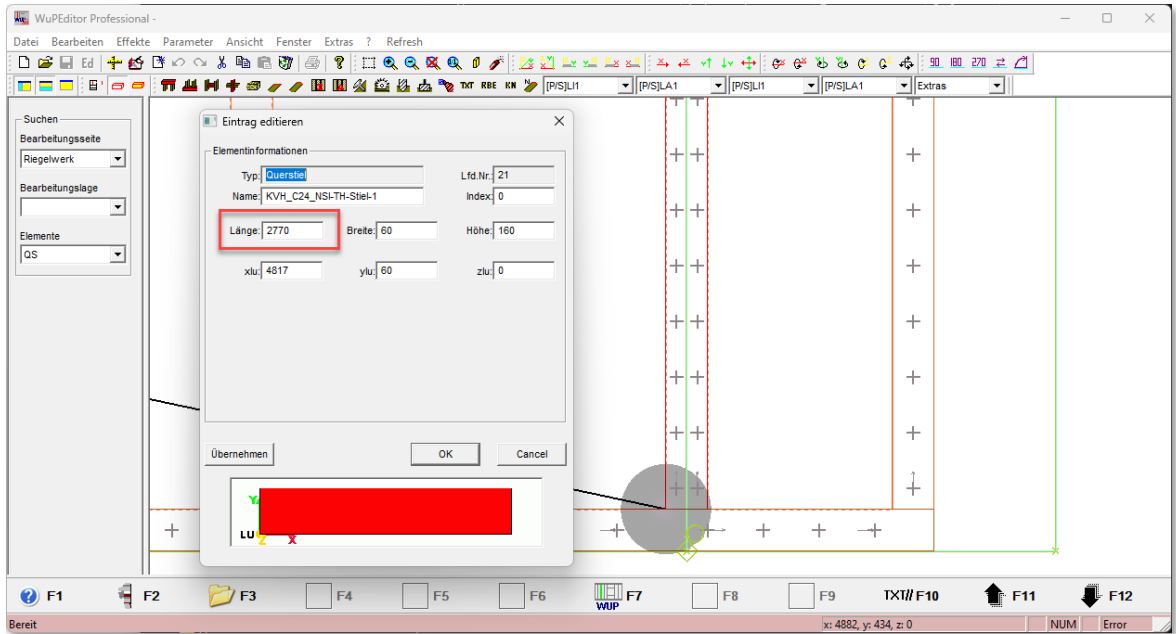
If the value 1 is set for this option, the nailing row is not interrupted by a protected zone surface, even if the option *Interrupt nailing in protected zone* is set in the *Panel analysis settings* -> *General*.

If the value 0 is set for this option, the row of nails is interrupted at protected zone surfaces as before.

- ◆ Consider lengthening/shortening end-type on elements in the framework  
Up to now, the lengthening/shortening end-type has not been taken into account when determining the length of the framework elements. As a result, framework elements are not exported in their true length.  
In version 30 (SP2024), you can specify whether the lengthening /shortening end-type should be taken into account when exporting the framework elements.  
For this purpose, the new option *Consider lengthening/shortening on framework elements* has been added in the *Panel analysis settings -> Extra settings*.

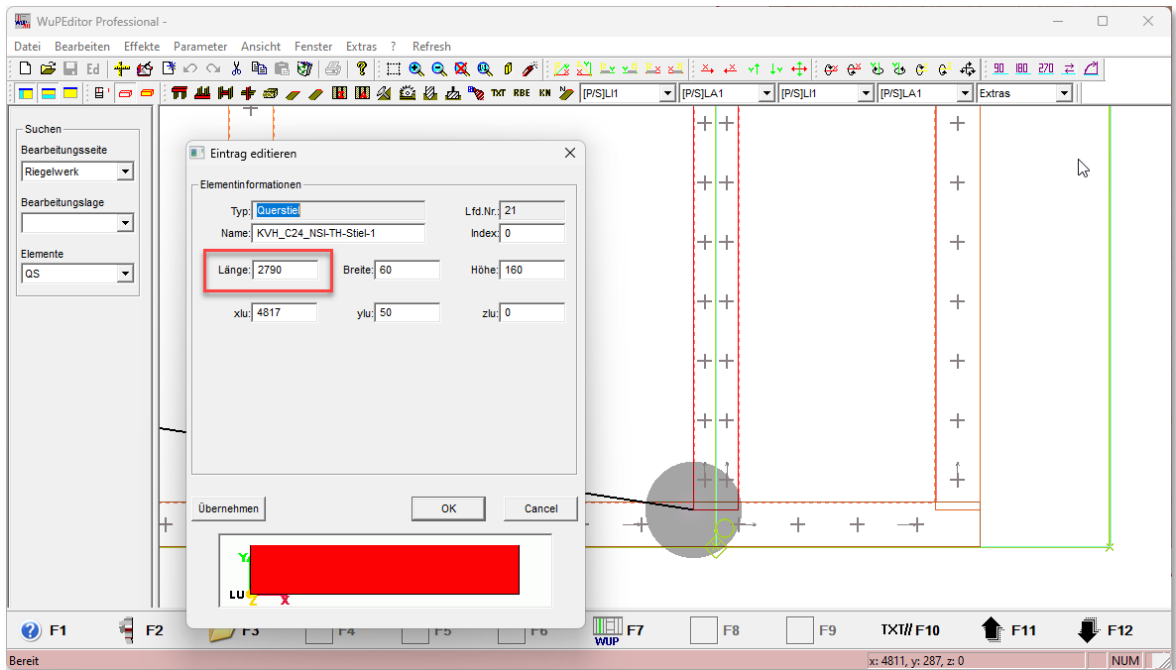


Weinmann WUP export *without* consideration of lengthening/shortening end types on elements in the framework:



\*.wup file without end-type consideration loaded in Weinmann WupEditor Professional

Weinmann WUP export *with* consideration of end type lengthening/shortening of elements in the framework:



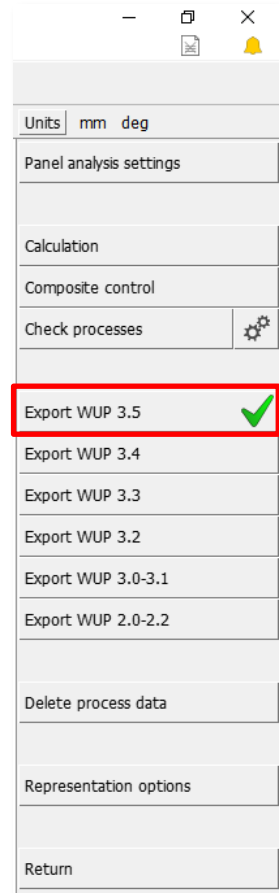
\*.wup file with lengthening/shortening considered loaded in Weinmann WupEditor Professional



- ◆ Weinmann WUP 3.5 version

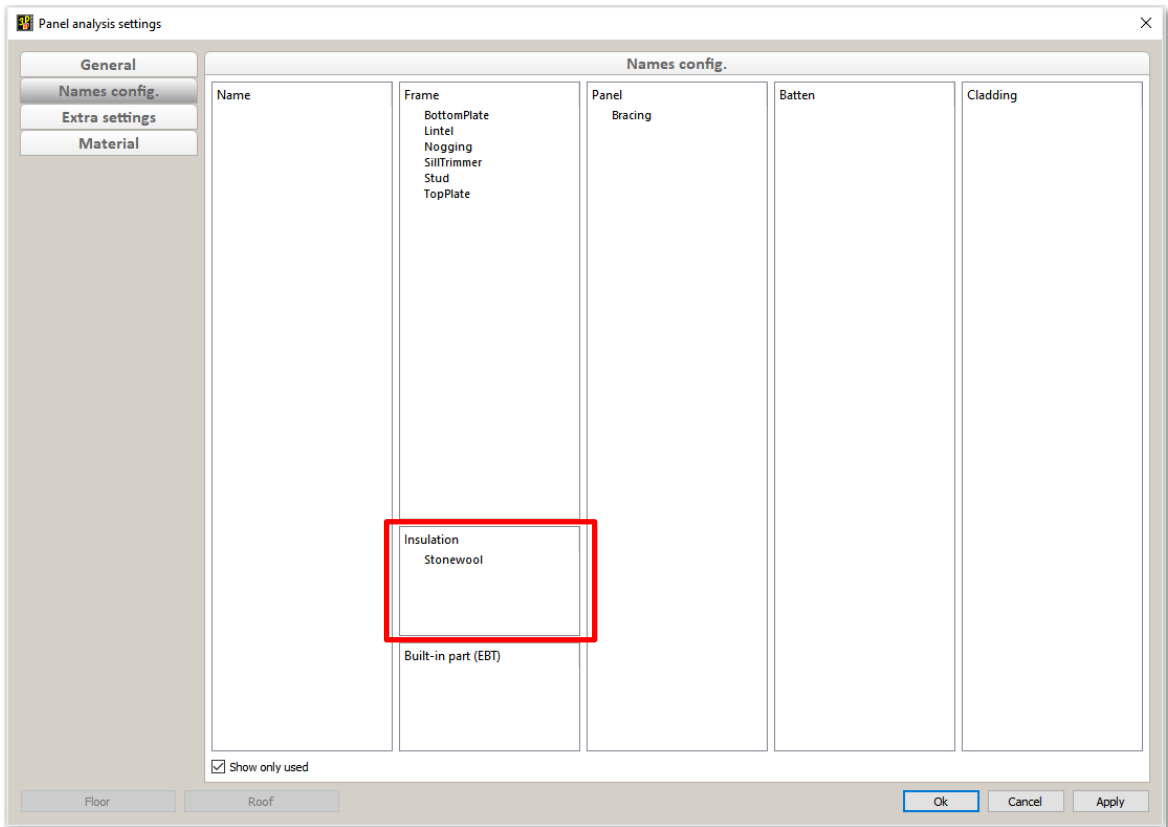
In cadwork version 30 (SP2024) the export Weinmann WUP 3.5 is available. When exporting with this interface version, the previous keywords for the machining operations lap (FRZ, FRY) and drilling (BOZ, BOY, BOX) are replaced by the keywords PAF PP and PAF MP respectively. The default value 0 is assigned as the tool index so that the machine takes over the selection of the tool.

A function key has been added to the *Machine* category for this output.

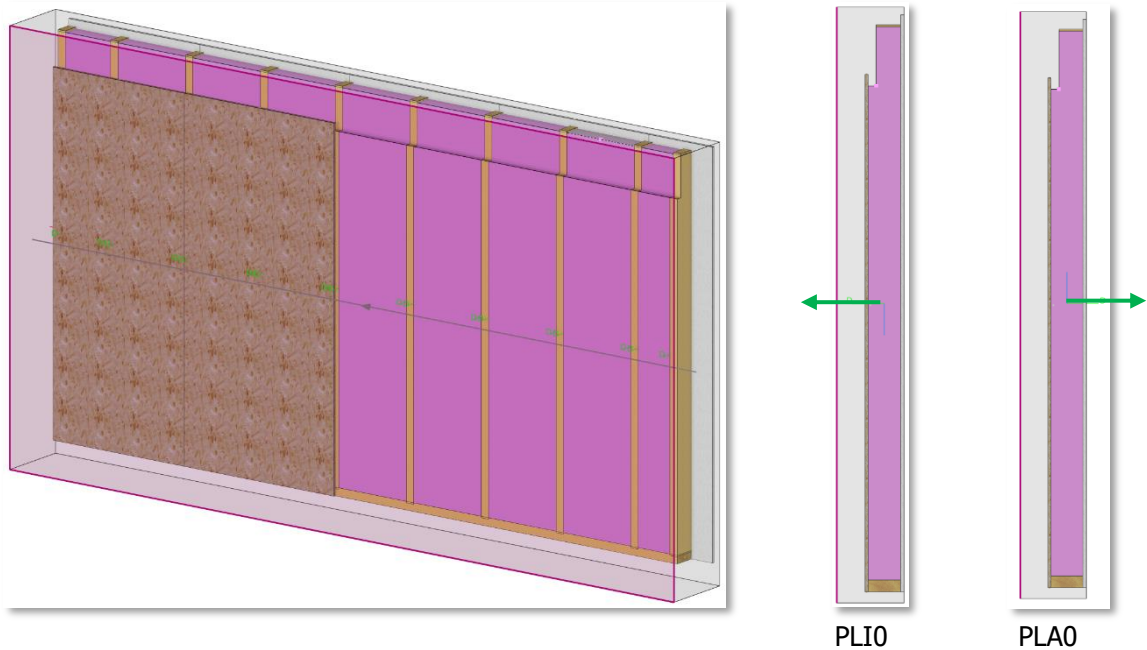


- ◆ Insulation in the Panel analysis settings

In version 30 (SP2024), it is possible to assign insulation elements that lie completely within the framework to a separate layer 0. For this purpose, the *Insulation* column has been added in the *Panel analysis settings* -> *Name config.* Elements that are assigned to this category and lie completely within the framework are exported in the reference side layer 0 or in the opposite side layer 0.



Whether an insulation element is assigned to reference side layer 0 or opposite side layer 0 can be determined by the alignment of its height axis.



All other elements that lie within the framework and are not assigned to the frame column in *Panel analysis settings* -> *Name config.* are exported in a higher reference or opposite side layer.

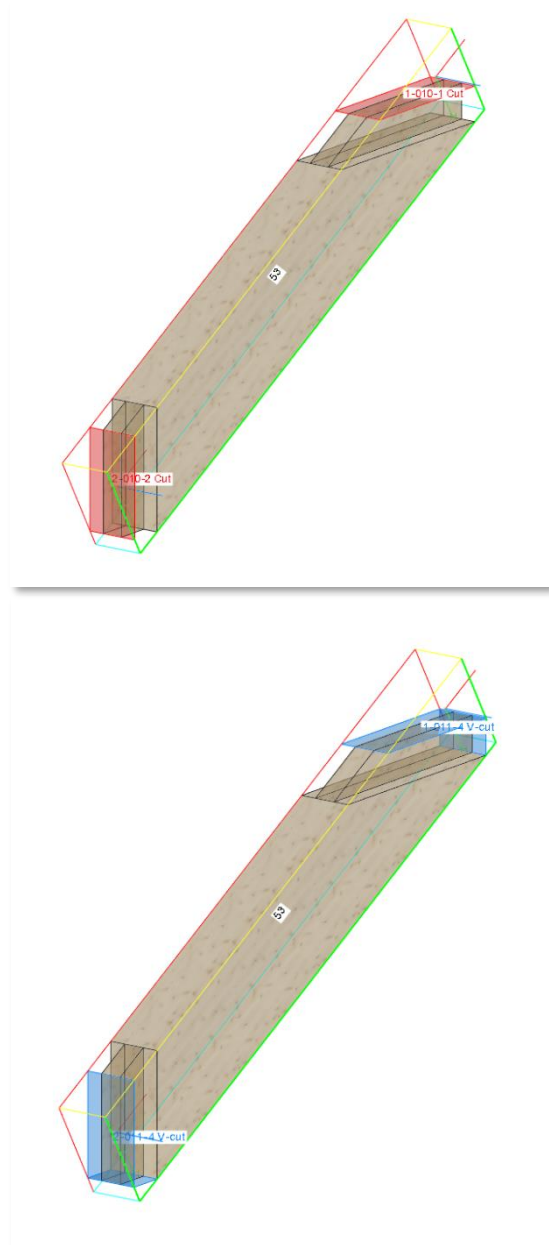
If you do not add any elements to the *Insulation* column in the *Panel analysis settings* -> *Name config.* tab, all elements that lie completely within the framework are exported in reference side layer 0 (PLI0) for the Weinmann WUP 3.4 export as before.

If you export with the new Weinmann WUP 3.5 export, it is necessary to configure the insulation in the *Panel analysis settings* -> *Name config.* in order to create a layer PLI0 or PLA0.

- Machine -> Others... -> Docking saw*

Tenons cannot be milled on cross-cut systems. Instead, an off cut was previously calculated at the end of the tenon. For some docking saw systems (e.g. Stromab), these sections were problematic for tenons on bevel cuts, as the off cuts plane extends beyond the rough part.

In version 30 (SP2024), a further off cut is added so that the part is always cut within the rough part. If the docking saw system supports *V-cuts*, the two cuts are combined into one V-cut. Tenons without backcut are cut along the tenon, tenons with backcut are cut at right angles.



The *Activate attribute -> Docking saw*

F-key function activates parts with right-angled off-cuts in order to filter them and decide whether they should be produced manually or on a docking saw system. Previously, it was not possible to execute the function without first calculating machine data.

As of version 30 (SP2024), this function key can also be used without calculated production list numbers, without a machine license and without calculated machine operations. If machine data is already available, it is used to determine the relevant components. However, the calculation time of the function is extended if no machine data is available and again if no production list numbers have been calculated, as in this case each component must be examined individually.

- *Shop drawing -> Piece-by-piece*  
 The text with the macro no. *1107#nesting volume number* from the layout file specifies the assigned numbers of the nesting volumes for the shop drawing export. Previously, the macro was only supported for element type panel. With service pack 2024, element type beam is also exported with their assigned nesting volume number.

X-sec corr. pos. height: 1031#X-secCorrPosHeight  
 Nesting vol. number: 1107#NestingVolNo  
 Volume: 4009#Volume

Macro in piece-by-piece - 2dc layout

Nesting vol. number: 2001000, 2001001

Pos **201** Name: **Floor panel**  
 Quantity: **6**

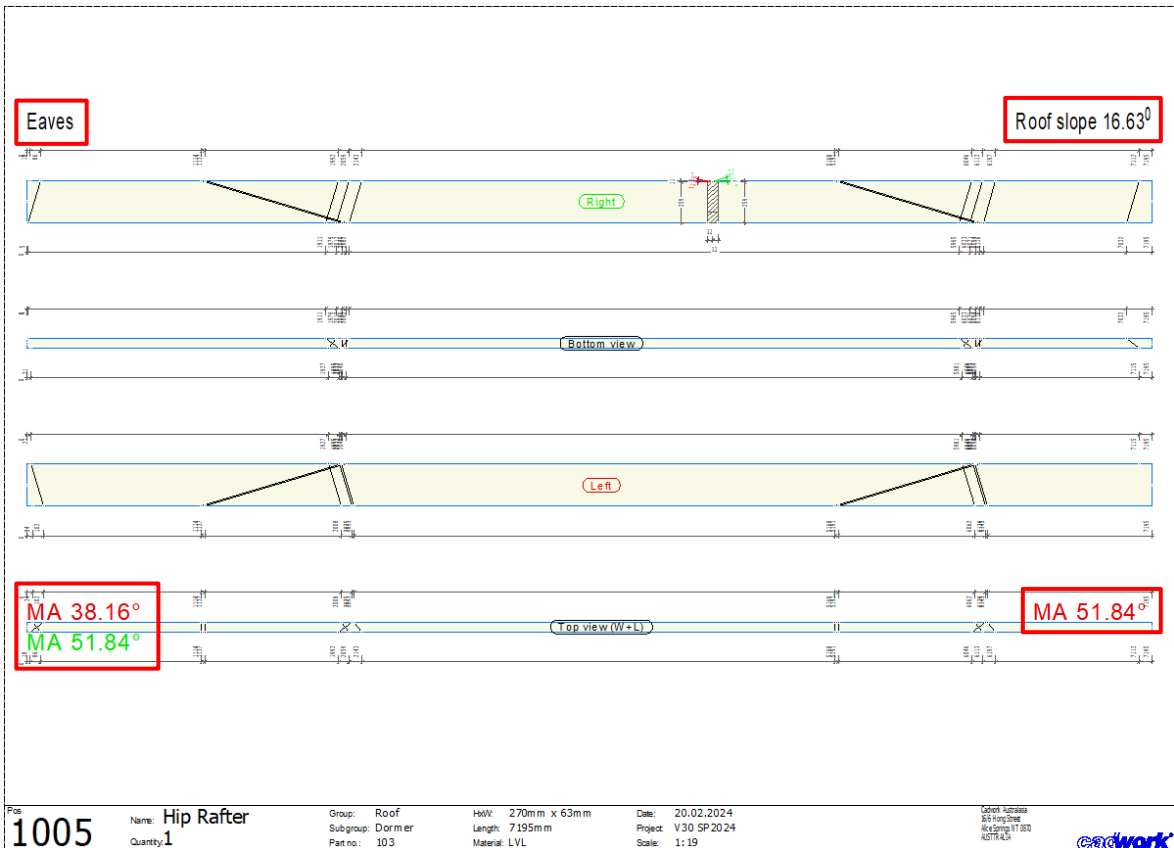
Result exporting element type

Nesting vol. number: 2000001, 2000002, 2000003, 2000004, 2000005, 2000006

Pos **12** Name: **Tread**  
 Quantity: **6**

Result exporting element type

Service pack 2024 introduces an additional function for the new shop drawing export of hip/valley rafters according to the method "Tracing lines all rough part edges". Information on the roof slope, machine angle and reference to the location of the eaves is provided within the " Tracing lines all rough part edges method " method.


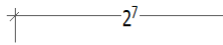



The marking for the eaves is at the top left of the shop drawing export, while the note for the roof slope is at the top right. The values for the machine angle are traditionally located on the ridge and on the eaves of the hip/valley rafter.

It should be noted that the positioning of the notes is fixed and cannot be positioned individually.

To be able to customise the text formatting like font, color and size of the Eave and Slope information the text macro `6209#` has to be used. This is a new macro in SP2024 and must be added manually to your existing piece-by-piece layout file as text *Eaves / Slope* with the name `6209#` at the location shown below. Otherwise, default formatting will be used.

### Setting for Hip/Valley Rough part dimensioning method

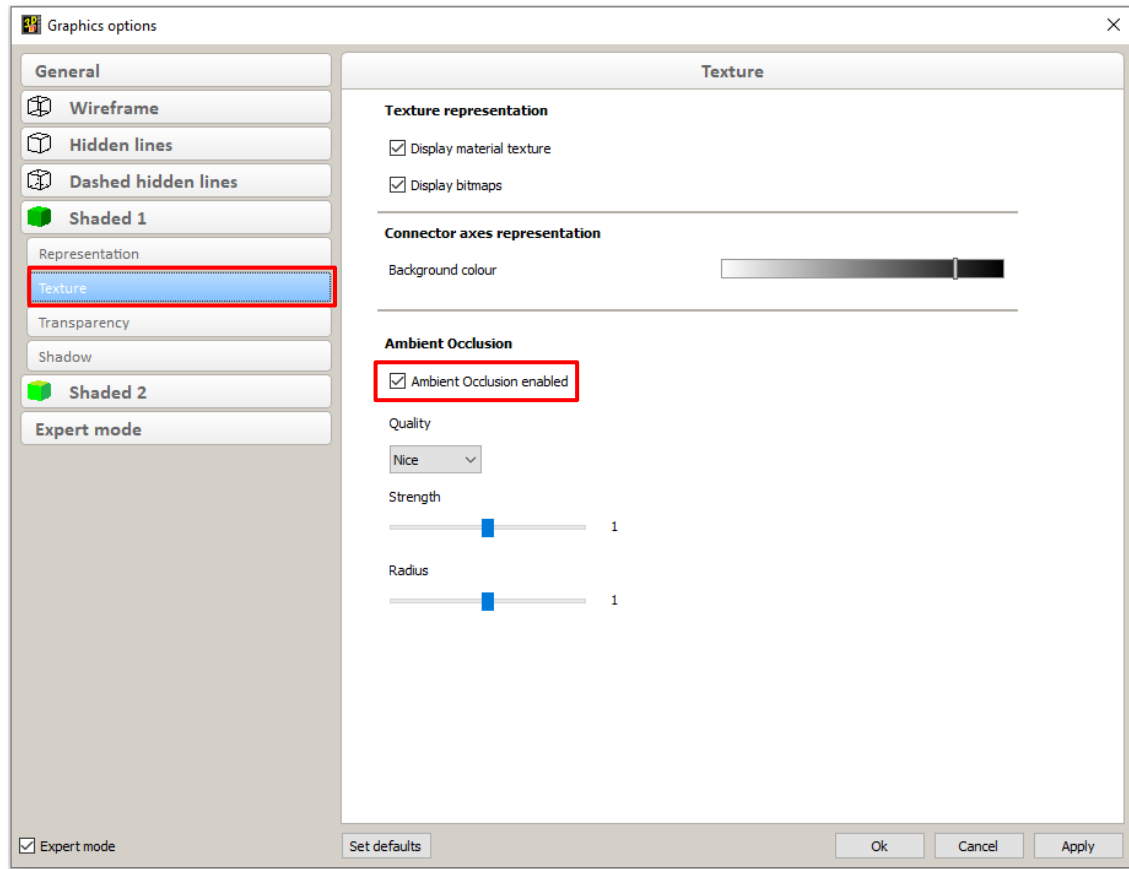
<p style="color: red; font-weight: bold; margin: 0;">Left</p> <p style="color: green; font-weight: bold; margin: 0;">Right</p>	<p style="font-size: 8px; margin: 0;">- Color definition of texts and dimensions of the rough part method related to the left and right side of the component (viewing direction from eaves to ridge).</p> <p style="font-size: 8px; margin: 0;">- The content and presentation of the text can be changed as desired and is output accordingly in the floor plan.</p>
<p><b>Lengthwise dimensioning</b></p> <p>The color of the dimensions is output according to the color definition of texts and dimensions of the rough part method (see above). All other properties are taken from this dimensioning</p>	
	
<p><b>All other dimensions</b></p> <p>The color of all other dimensions is always output in color 8 (white/black) and cannot be set. All other properties are taken from this dimensioning</p>	
	
<p><b>Angular dimension</b></p> <p>The color of the dimensions is output according to the color definition of texts and dimensions of the rough part method (see above). All other properties are taken from this dimensioning</p>	
	
<p><b>Text representations</b></p>	
<div style="border: 1px solid #ccc; border-radius: 10px; padding: 5px; margin-bottom: 5px; text-align: center;">View description</div> <p style="margin: 0;"><b>Machine angle</b></p> <p style="margin: 0;">Plumb</p> <div style="border: 2px solid red; padding: 2px; margin: 0;"><b>Eaves / Slope</b></div>	<p style="margin: 0;">Definition of view description for floor plan and top view. The content of the text should not be edited</p> <p style="margin: 0;">Color definition is not considered</p> <p style="margin: 0;">Color definition and content is considered</p> <p style="margin: 0;">Color definition is considered</p>

New in Version 30

This text marco has been added to most recent default layout files supplied by cadwork, which can be downloaded via Userprofile (user's settings) -> Manage catalog/userprofile in the Piece-by-piece - Shop drawings register

- *TIF file (\*.tif)*

To create appealing rendered and shaded representations, the option *Ambient Occlusion enabled* can be selected in the Graphic options dialog, which can be called up via the left-hand menu.

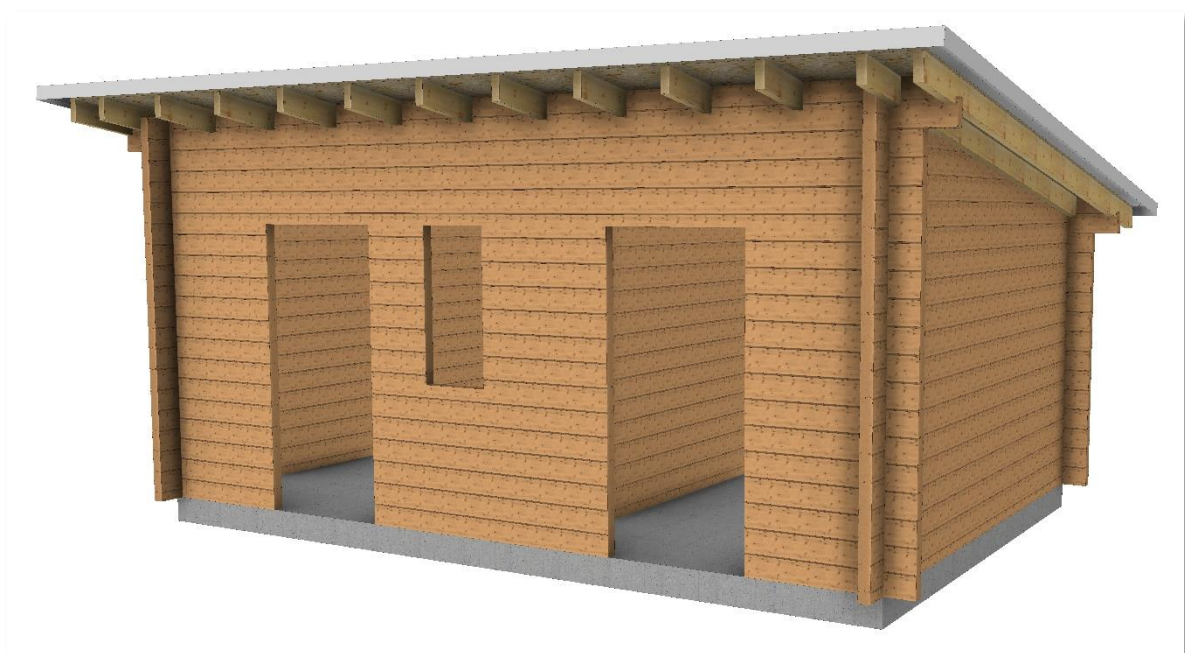
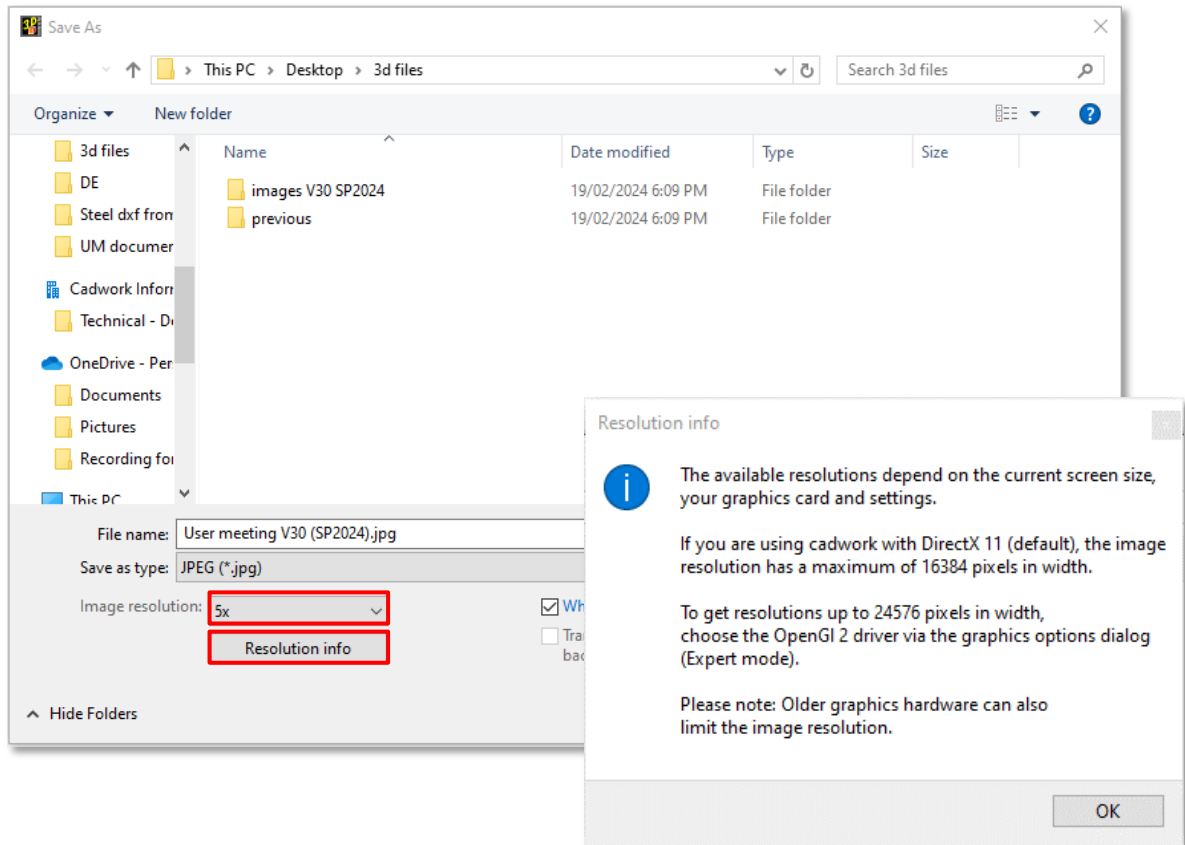


The respective display can be exported as a tif or jpg file via *Export -> Image (\*.tif, \*.jpg)*. In previous version when using the *Ambient Occlusion enabled* option, the resolution factor has to be set to 1 for error-free export. This significantly limited the resolution of the generated image.

This restriction was removed in Service Pack 2024, so that higher-resolution images can now also be exported.

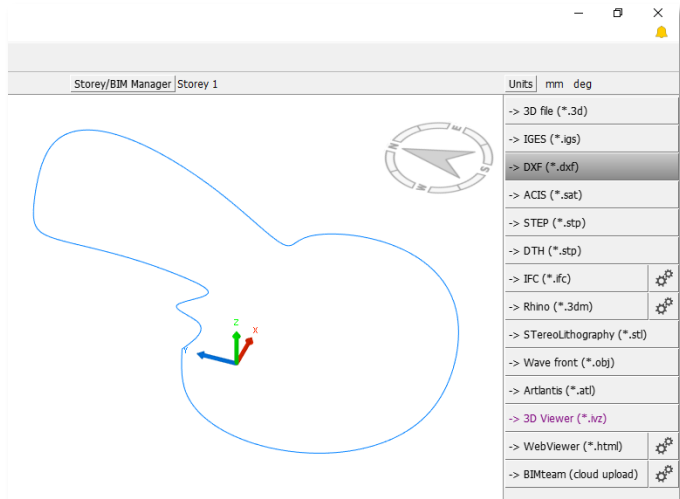
With regard to the possible resolution factor, however, there are still restrictions resulting from the graphics card and graphics driver used.

You can obtain detailed information on the restrictions via the *Resolution info* button in the image export save as dialog.



## Files... -> DXF (\*.dxf)

The export to DXF files has been improved so that spline lines can now also be taken into account. The export from 3D is done in the DXF format AC1027 for AutoCAD 2013 to AutoCAD 2017.



- Files... -> IFC (\*.ifc)

News on exporting IFC files can be found in the IFC chapter.



## ❖ IFC

### ➤ IFC import

#### ▪ Certification for IFC 4 - Reference View

cadwork has received certification for the export of IFC 4 - Reference View from buildingSMART. This means that the software complies with the international standard IFC 4 and can therefore export models according to the specifications of buildingSMART.

This certification ensures smooth interoperability and the exchange of models between different software applications that use the IFC standard.

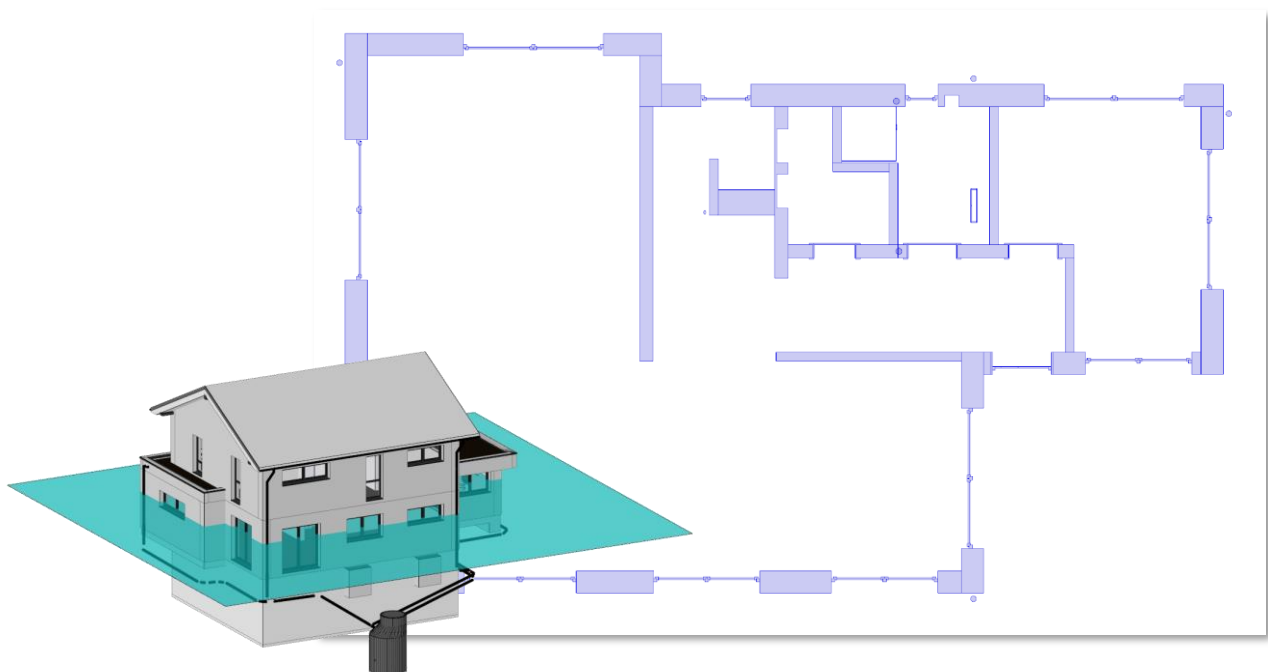
Vendor	Product	Schema	Exchange Requirement	Import / Export	Status	Started	Completed
cadwork Informatik	cadwork 3D	IFC4	Structural Reference Exchange	Export	Finished	2019-10-07	2023-07-13

#### ▪ Exchange objects

A current Hoops version has been integrated into Service Pack 2024, which promises improvements in terms of both display quality and performance/stability.

It is now also possible to visualize exchange objects in 2D planes. The elements are displayed as an intersection in the plane.

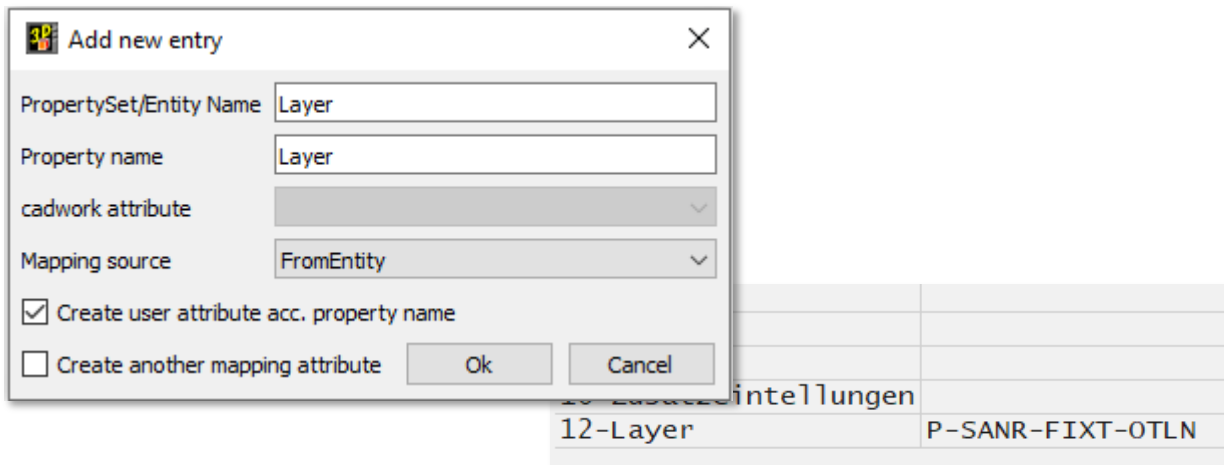
Currently, the points of the visualized elements cannot be selected and used for construction.



- IFC Property Mapper and layer mapping

You can use the property mapper to transfer/"map" attributes from the IFC file to the imported and converted elements in the 3D file.

Previously, this mapping only worked for the properties of the IFC file, but now the information of the IFC layer can also be transferred for the elements of the 3D file. Like the properties, the IFC layer is also created as a cadwork attribute.



➤ IFC export

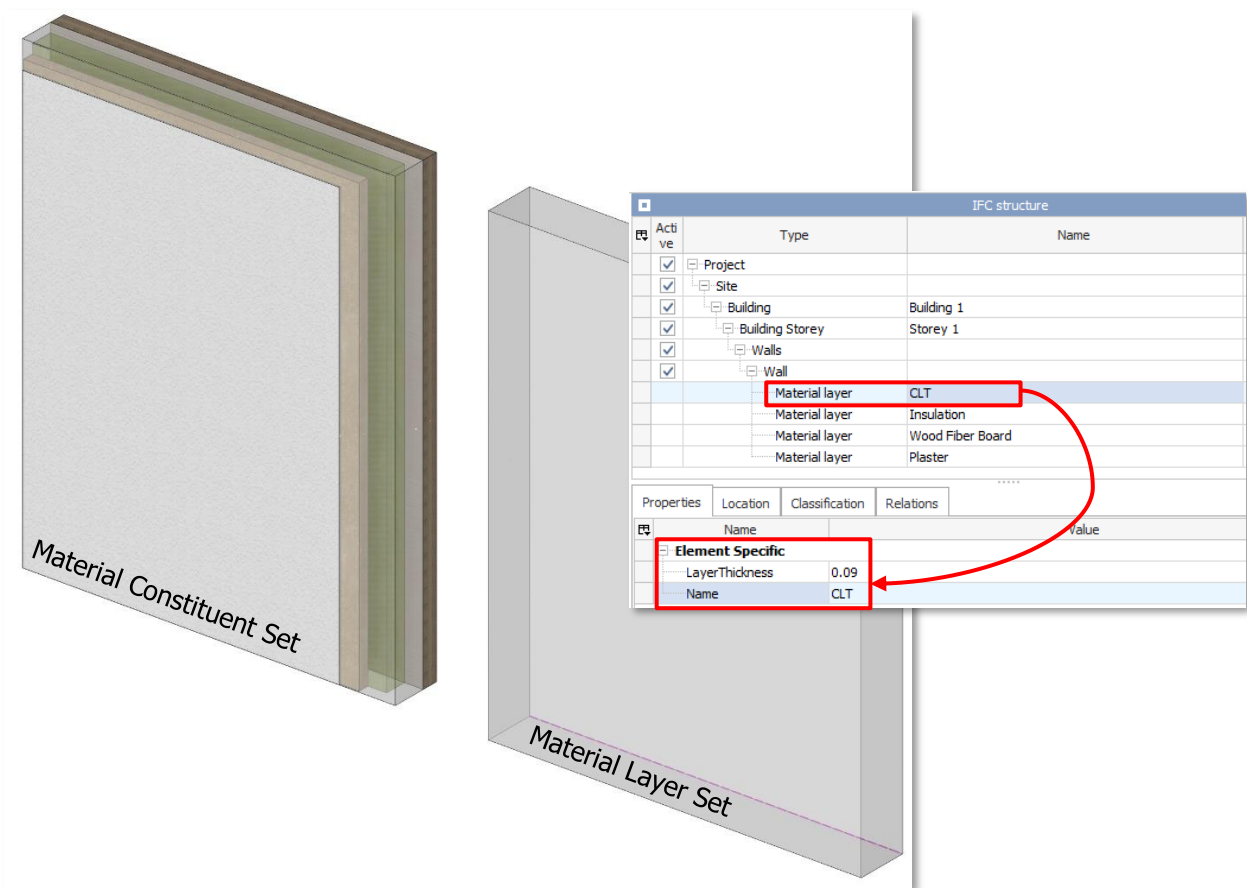
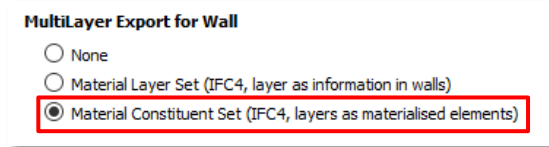
▪ Export of multilayer wall - IfcMaterialConstituentSet

We already introduced the option of exporting multilayer walls to IFC files in version 30.

With the service pack 2024, you can export these multilayer walls not only with the

IfcMaterialLayerSet, in which the layer definitions of a wall are included as informative text in the IFC file, but optionally also using the IfcMaterialConstituentSet.

With this option, you can respond to the requirements of the project partner in an even more differentiated way and thus optimally support the subsequent planning steps.

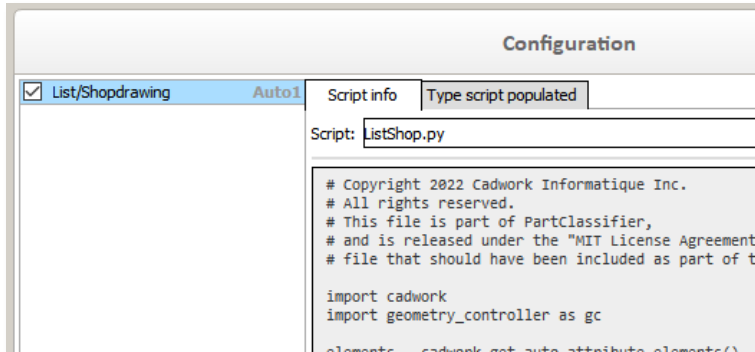


Wall imported on the left as a Material Constituent Set: The individual layers are available as materialized volumes including the wall rough volume.

On the right, the same wall as a Material Layer set: the wall rough volume is generated as a volume, the information of the defined wall layers is defined as attributes/properties.

- *Script-populated attributes in the export*

Attributes filled with scripts are taken into account when exporting in the cadwork3dproperties property set.



This means that attributes and properties generated or influenced by scripts are exported in the standard property set of cadwork.

This is particularly relevant for users who create complex attributes in their models using Python scripts. The automatically generated information is now also retained when exporting IFC files.

16-Grade	
17-Coating	
18-Machine	
19-Shift	
20-Source	
21-Supply	
22-Special tooling	
23-Layers	
24-Layer orientation	
25-Number of layers	
26-Corner construction	
27-Protection	
List/Shopdrawing	Shop Drawing

Properties	Location	Classification	Relations	.....
Name	Value			
<b>Element Specific</b>				
..... Guid	3ItC1BLrDDffU27Z0waRnm			
..... IfcEntity	IfcMember			
..... PredefinedType	NOTDEFINED			
<b>BaseQuantities</b>				
<b>Cadwork3dProperties</b>				
..... List/Shopdrawing	Shop Drawing			

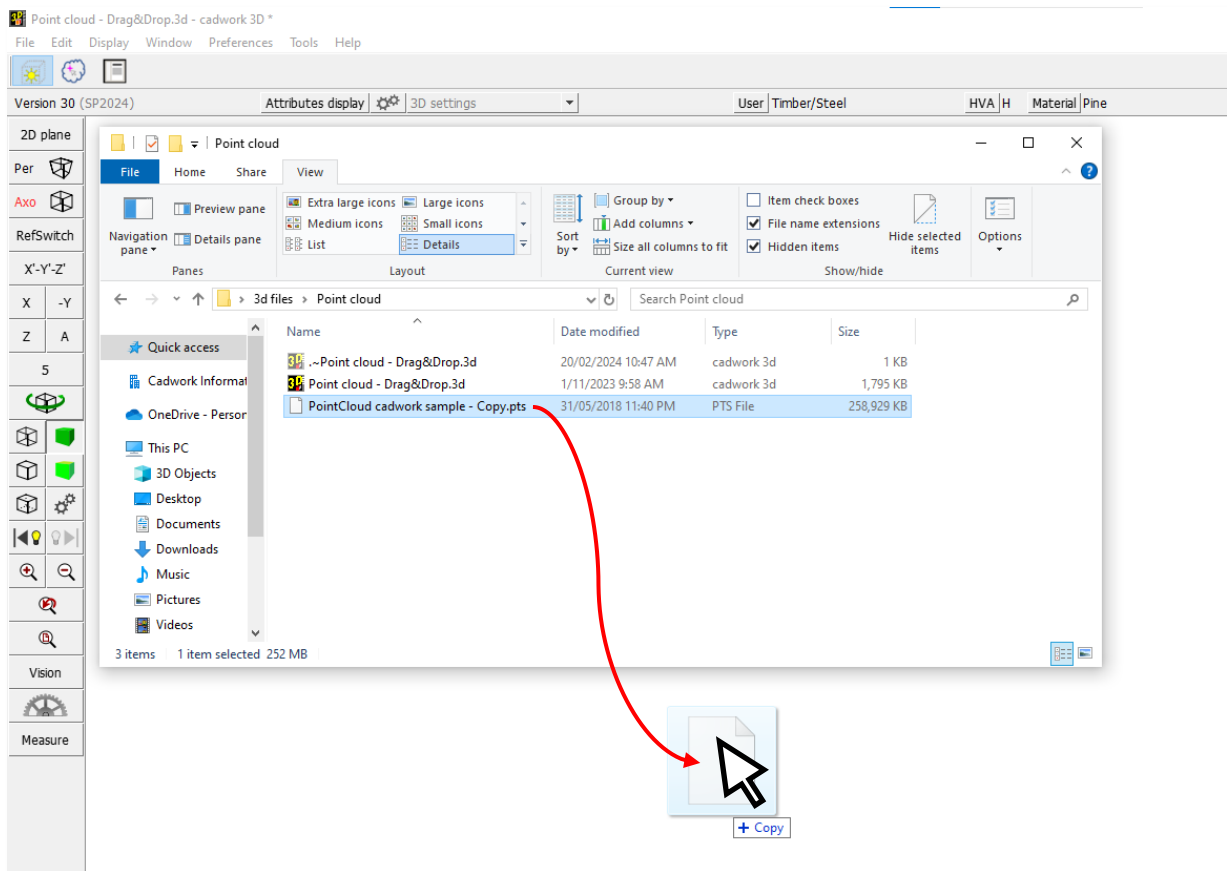
## ❖ Point cloud

### ➤ General information

As usual, you can only use the Point cloud interface via the corresponding license. on the following pages the new features are described.

### ➤ Import

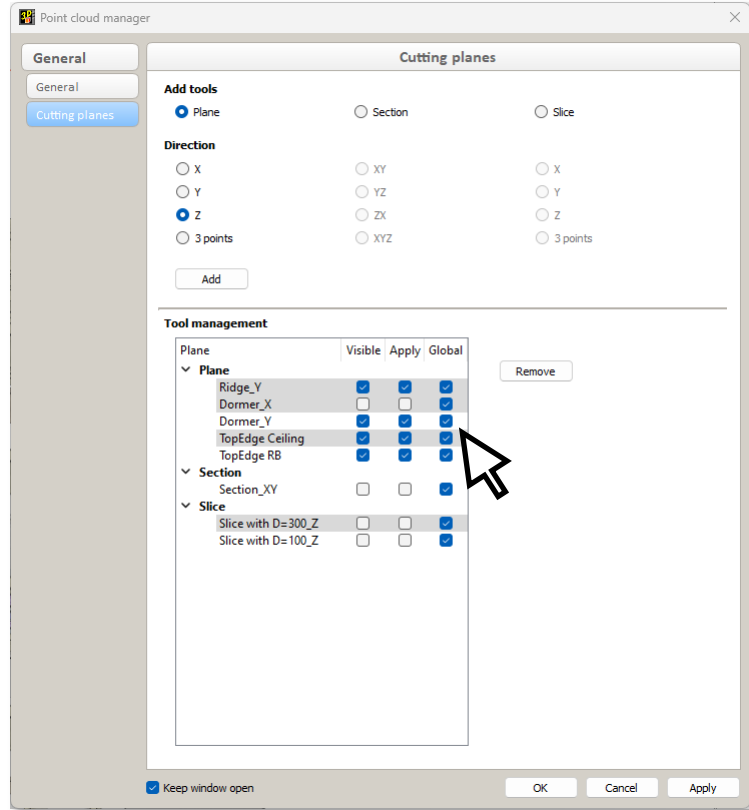
In Service Pack 2024, you now have the option of dragging and dropping a Point cloud directly into your 3D file. Nothing else has changed in terms of import behaviour. The usual options via the menu or function keys are still available to you.



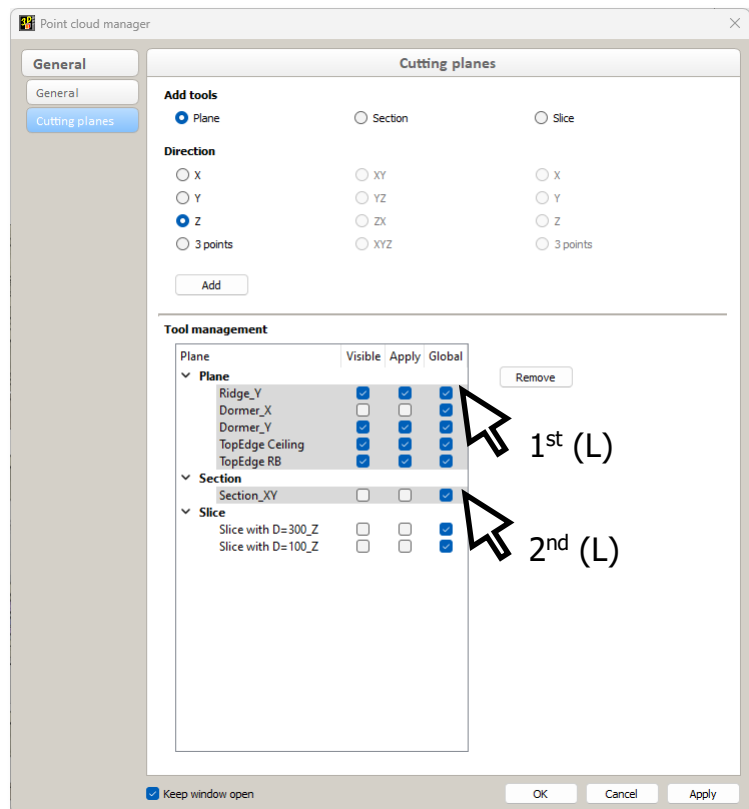
➤ Point cloud Manager

The Point cloud Manager has been expanded to include the multiple selection function. Here you have the option of activating and controlling multiple sections.

You can use <Ctrl>+(L) to individually pick planes, sections and slices with the mouse and add them to your selection.



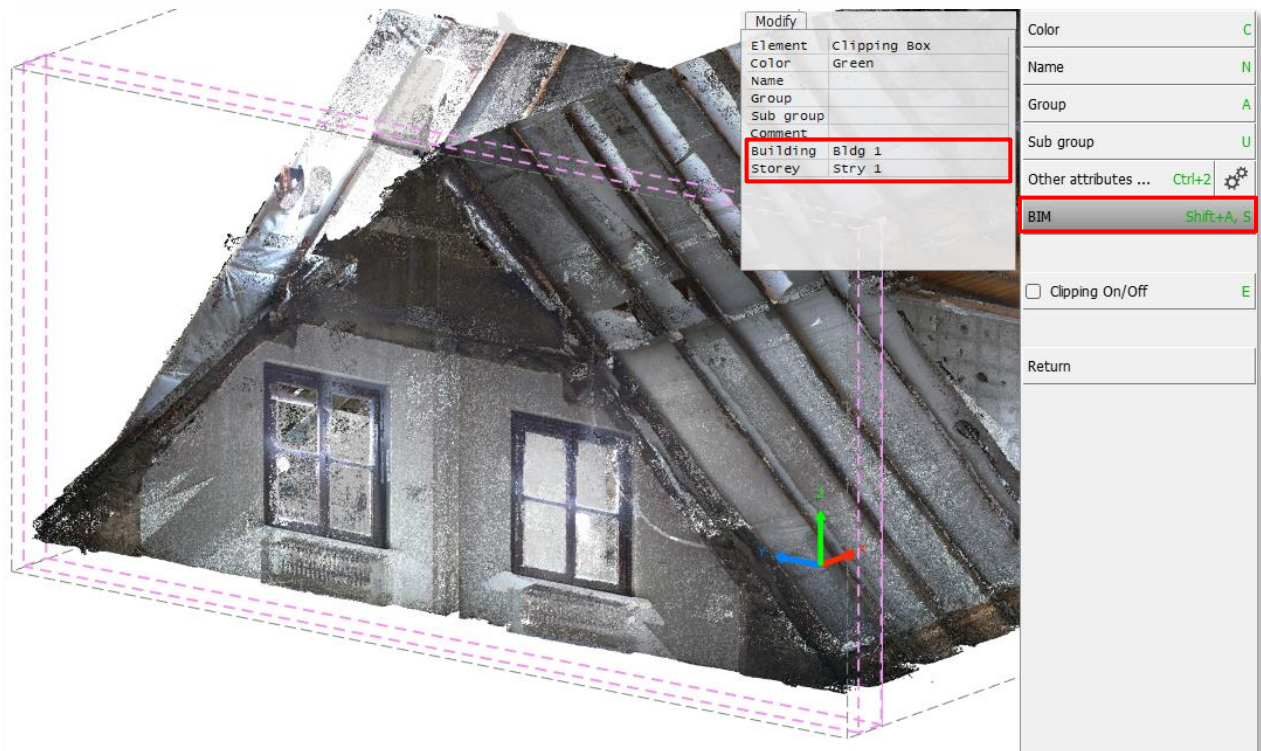
You can use <Ctrl+Shift>+(L) to select an entire portion of the list. Pick first and last Plane, Section or Slice with (L) and it adds everything in between to selection.



## ➤ Clipping box

The Zoom All behaviour of the clipping box has been adjusted so that when the clipping box is active, right-clicking (R) on *Zoom All* zooms to the geometry of the clipping box and not the geometry of the bounding box.

Furthermore, the BIM attributes have been added to the clipping box. You now have the option of controlling the visibility of the clipping box via the storeys.



## ❖ Plugins

The further development of functionality in cadwork is largely based on requests and suggestions that we receive from valued customers.

Customer requests also include requirements that cannot be implemented at short notice or are so specific that they cannot be incorporated into the general range of functions. In both cases, the cadwork API can be helpful, which can be used to implement a specific functionality as a plugin. Although the plugin is not fully integrated into the cadwork functionality, it can use large areas of the existing functionality, access the data and manipulate it.

The API offers the great advantage that its use can be learned by any cadwork user. After becoming familiar with the cadwork API, plugins can be developed independently to implement company-specific functionalities and thus customize cadwork.

### ➤ General

With cadwork version 30 (SP2024) it is possible to connect cadwork 3D directly to the execution of an API plugin. This is done by starting the 3d.exe with the argument `"/PLUGIN=PluginName"`. This can be implemented using a batch file \*.bat file, for example.

As soon as cadwork 3D is started in this way and the 3D file has been loaded, the plugin defined in the argument is automatically started. It does not matter whether it is a plugin developed in C++ or Python. However, it is important that the specified plugin can be found under the corresponding name in the regular plugin directory in the userprofil\_30 folder (... \userprofil\_30 \3d \API.x64). The following lines with the PATH and START instructions show how a batch file for starting cadwork with automatic execution of a plugin can look.

```
PATH=C:\Program Files\cadwork.dir\exe_30\pplib.x64;C:\Program Files\cadwork.dir\exe_30\LxSDK.x64\bin\dlls;  
C:\Program Files\cadwork.dir\exe_30\LxSDK.x64\bin;
```

```
start /wait /max C:\Program Files\cadwork.dir\exe_30\3d.x64\3d.exe "C:\Users\Users\Documents\TestFile.3d"  
/PLUGIN=example_plugin
```

As can be seen, certain paths to the cadwork installation directory are defined first. This is necessary so that they can be found by cadwork 3D. The drive names and paths must be adapted to the cadwork installation on the system used.



➤ cadwork API - Help and discussion forum

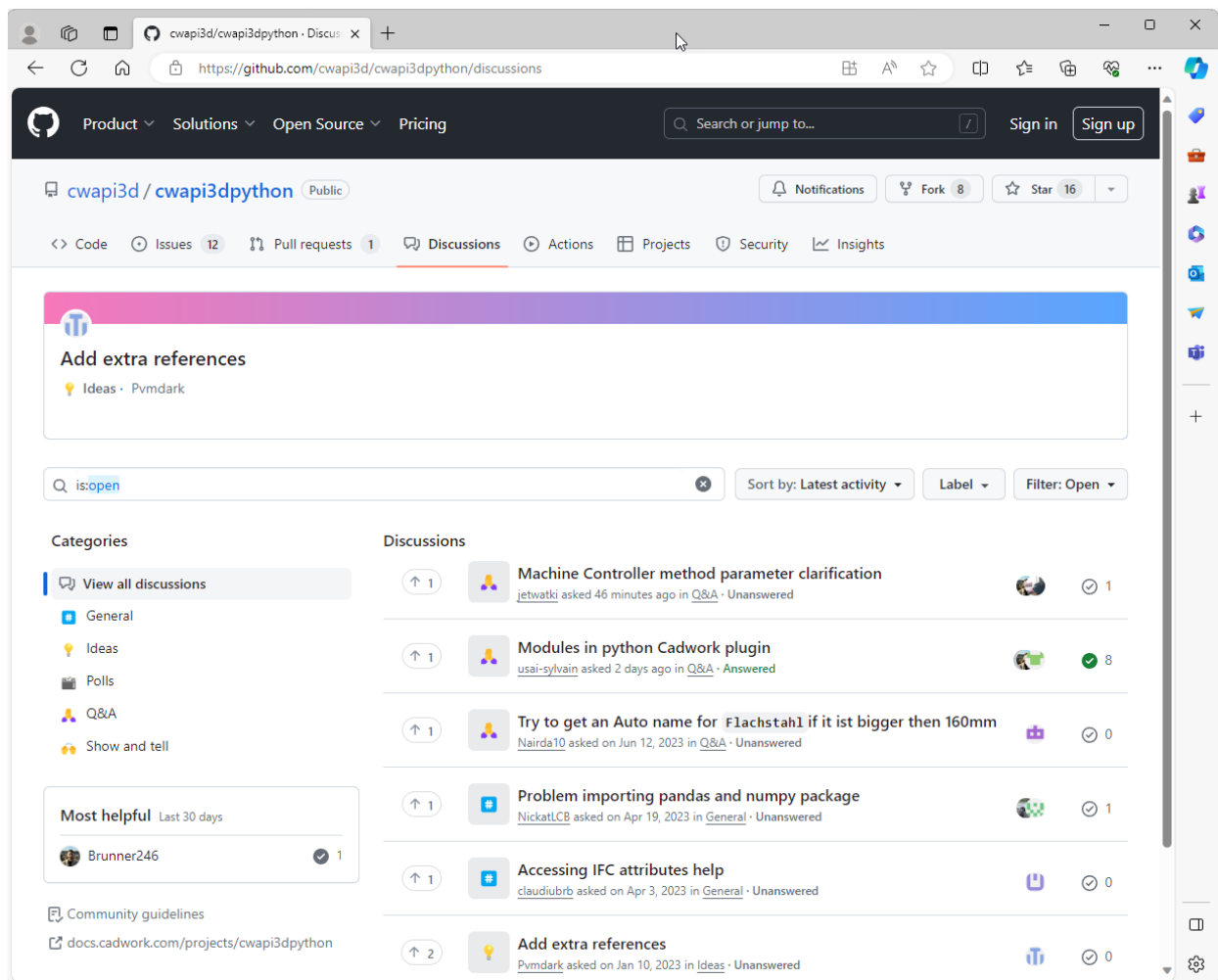
We would like to take this opportunity to point out that there is a possibility on the "GitHub" platform to ask questions about the use of our API and to exchange information with other users. Please use the following link:

<https://github.com/cwapi3d/cwapi3dpython/>

A one-time registration is required to use the platform. You will find the "Sign up" button in the top right-hand corner.

In the "Discussions" category, you can ask general questions about the use. To do this, you can open a new discussion using the "New discussion" button. You will then be shown various categories for this discussion. As soon as you have created a discussion, it is visible to other users. You can also participate in existing discussions at any time.

If you notice any misconduct when using our API, you can report this in the "Issues" category. You can also submit requests for new functionalities here.



➤ BCF Manager – Connect to BIMcollab platform

The extension of the BIM tools through the integration of the Software Development Kit (SDK) from BIMcollab as a plugin represents a step in the further development of the BIM tools in cadwork 3D. This extension enables a seamless connection to BIMcollab, a cloud-based platform that optimizes collaboration and communication in the field of BIM. In order to use the plugin, users need to create an account with BIMcollab. The first time they use it, they will be asked to create a login. On the cadwork side, the *IFC Import/Export* and *BCF Manager* licenses are required.

The BIMcollab SDK plugin allows users to access BIMcollab functions and resources directly and integrate them into their existing BIM workflows. By utilizing these SDK functionalities, users can make their workflows more efficient as they can now interact directly with the BIMcollab platform without having to switch between different applications.

This BCF Manager not only provides the ability to connect to the BIMcollab cloud, but also supports the use of local BCF XML files. This means that projects can be flexibly stored and edited both in the cloud and locally, depending on individual user requirements and preferences.

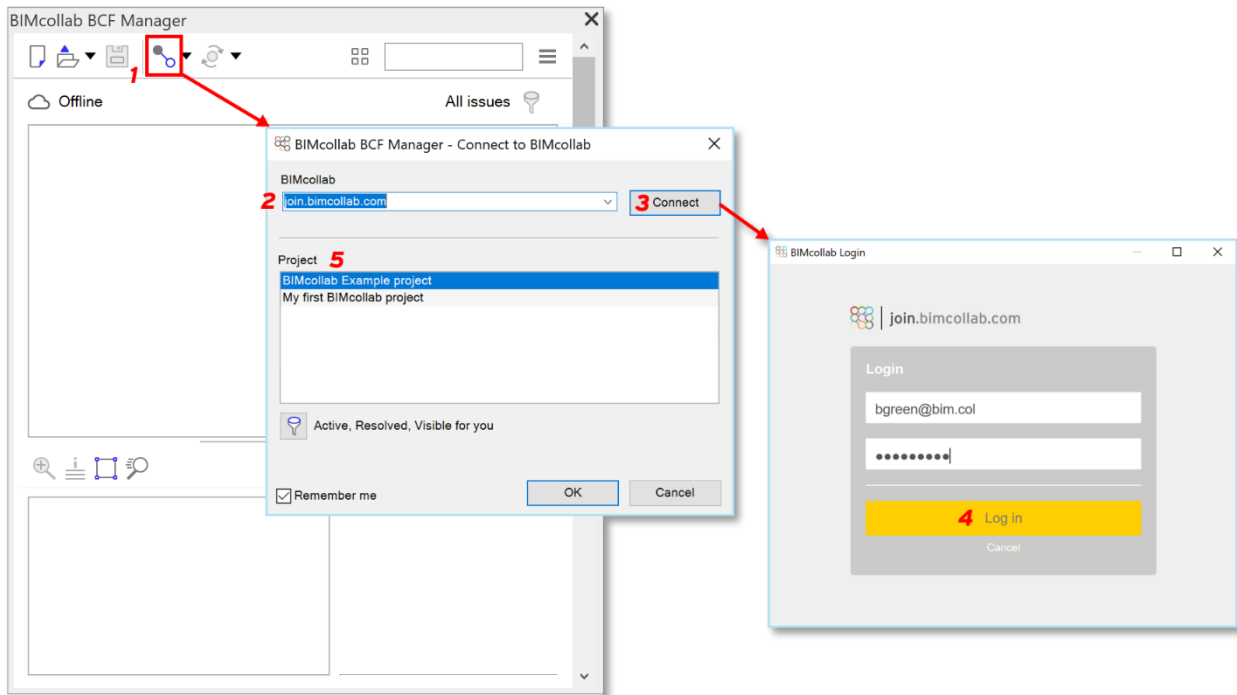
This extension removes obstacles that previously hindered smooth communication and coordination between different project participants.

▪ Connect with the cadwork BCF Manager or BIMcollab Zoom

1. Open the cadwork BCF Manager or BIMcollab Zoom and click on the "Connect to BIMcollab platform" icon.



2. In the "BIMcollab" field, enter the following information:
  - a. [join.bimcollab.com](https://join.bimcollab.com) to connect to your free BIMcollab project.
  - b. the URL of your private WebRoom (e.g. [BIMcollab](#) ) to connect to the private WebRoom.
3. Click on "Connect"
4. A new window will open. Enter your login details and click on "Log in".
5. Select the project to which you want to establish a connection and click OK.



Source: <https://helpcenter.bimcollab.com/portal/en/kb/articles/connect-from-bcf-managers-or-bimcollab-zoom>

- BIMcollab help center

Under the following link you will find the help center portal to report issues/bugs or log wishes/requests:

<https://helpcenter.bimcollab.com/portal/en/kb/articles/edit-and-comment-on-issues#BIMcollab Zoom and the BCF Managers>

## ➤ Joinery Tool Center (JTC)



The plugin is updated independently of the annual cadwork versions. This has led to the release of JTC version 3 in December 2023, which contains further improvements and changes.

- Changes to the design

Based on customer feedback, small adjustments were made to the design and structure.



- New "Step back" function

Previously, processes created by the JTC had to be deleted manually. Now it is possible to undo individual processing steps. The "Step back" function <AltGr>+<Z> can be used to undo any action performed in the plugin without changing previous calculation results.

- Hardware configurator database

The hardware configurator database has been adapted.

Manufacturer configurations can now be selected and set directly.

We have also added many typical furniture fittings to the selection

options. You can change and save these according to your own requirements. You can also add your own catalog elements to the hardware configurator.



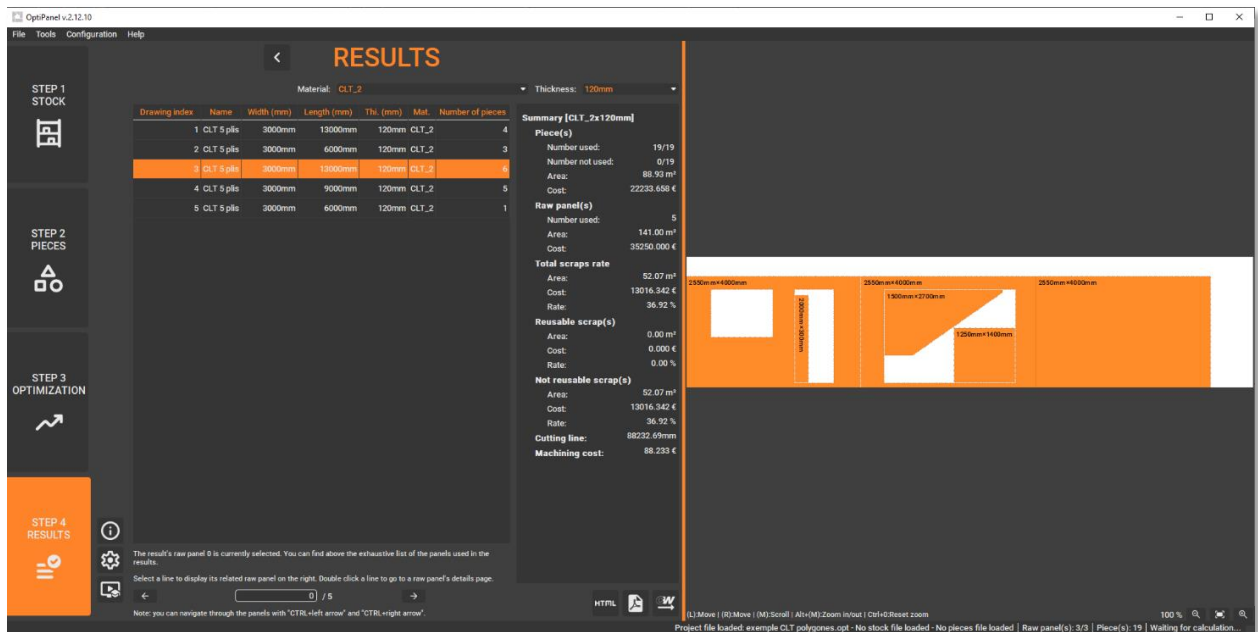
- Save & load default settings

The settings selected by the user in the various dialogs can be saved and loaded as default settings. Defining default settings speeds up working with JTC, as the dialogs are always opened with the user-specific settings.

## ❖ Tools

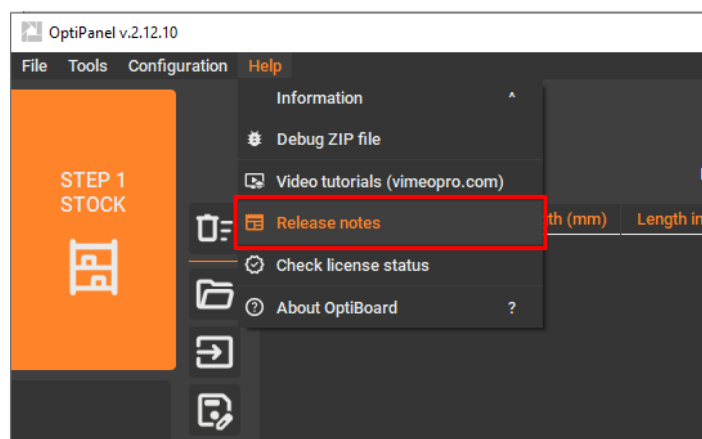
### ➤ OptiPanel

Version 2.10.x of the OptiPanel panel nesting optimization was made available for the delivery of cadwork version 30. Version 2.12.x will be delivered with Service Pack 2024, if required licence bit is active.



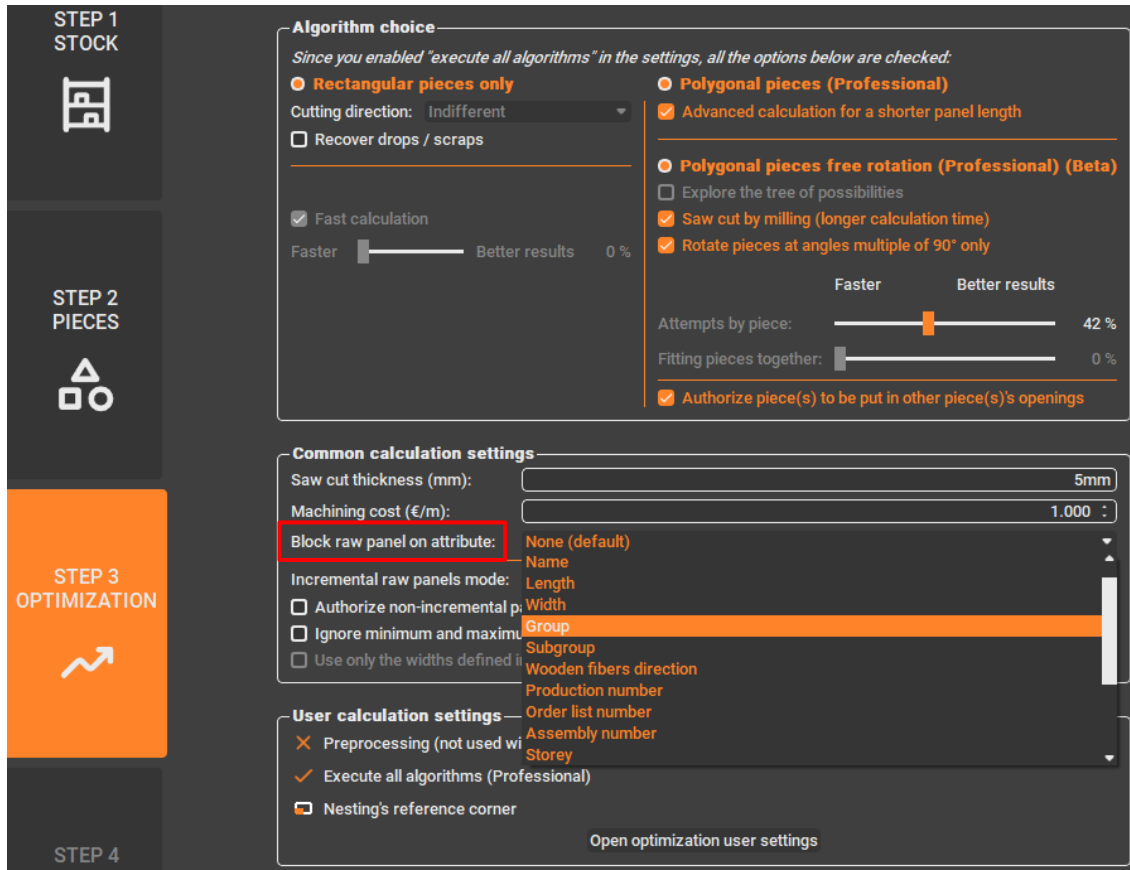
OptiPanel – 4 step nesting optimization OptiPanel – Step 4 Results

Numerous bug fixes and improvements have been implemented during this year. You can obtain an overview of all bugfixes and improvements at any time in OptiPanel itself via *Help -> Release notes*.



Below a list with a summary of key improvements:

- It is possible to undo or redo up to 50 steps. Previously, there was a limit of ten steps.
- The optimization algorithm for polygonal parts and free rotations has been updated and offers improved performance and new parameters.
- Support for additional attributes to keep panels with identical attributes together (block raw panel on attribute).



- It is possible to define an edge distance per edge for each nesting volume to position the panels to be nested.
- In the table of nesting volumes, it is possible to edit the values for the maximum length and the increment for the automated creation of new nesting volumes.
- The material costs can be predefined for lineal meters and square meters.
- It is possible to import the nesting volume list from any OptiPanel project file.
- There is a warning message if OptiPanel cannot process imported geometry information.
- In the nesting volume and panel tab, previously selected nesting volumes and panels can be duplicated using the *Duplicate* button.
- Duplicating panels may only be used if OptiPanel is used without a connection to cadwork. When importing panels from cadwork, duplicating these panels leads to an incorrect reimport of the optimization results back into cadwork, as the number of panels is no longer consistent.

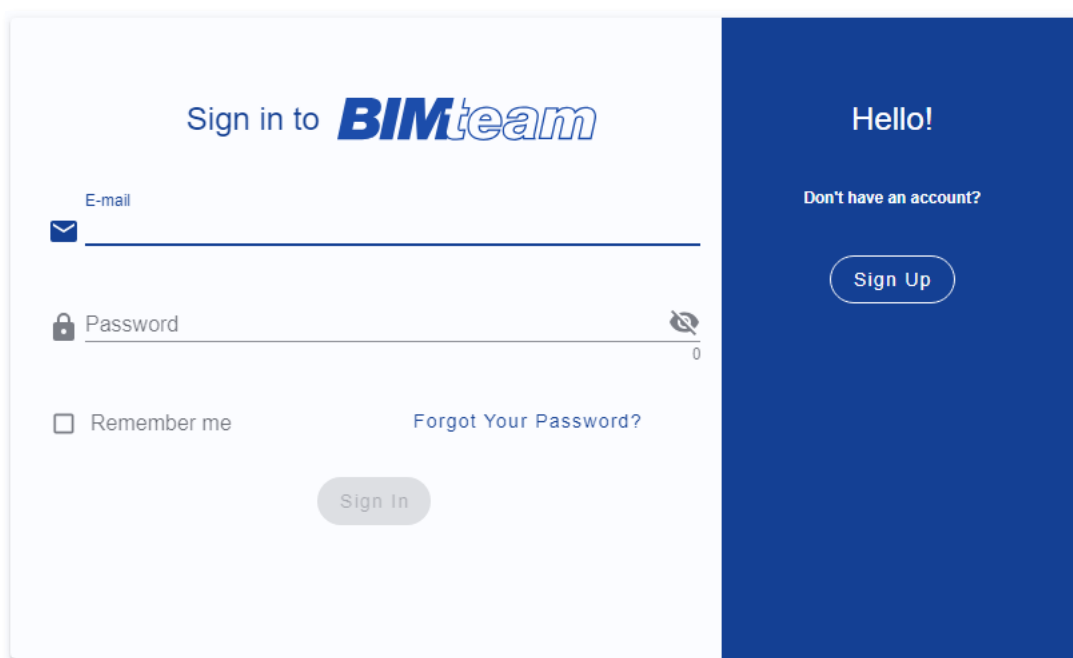
➤ BIMteam – cloud platform

▪ General information

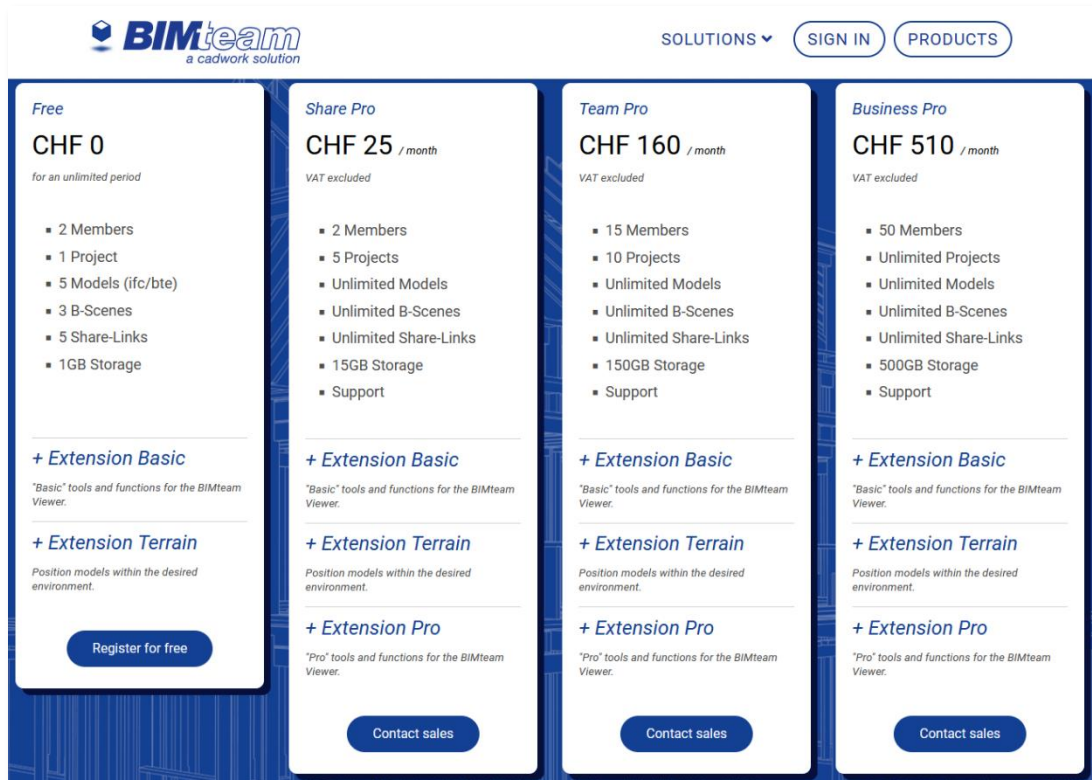
Version 23 introduced the WebViewer based on HTML5 files as a platform-independent visualization option for cadwork 3D models. The WebViewer is a helpful tool for providing 3D models to customers or partners. Over time, however, we have realized that the functionality has become increasingly limited due to manufacturer and web browser restrictions. The background to this is that HTML files can be circulated with malicious scripts and therefore represent a considerable security risk. HTML files exported from cadwork 3D are also very large, which makes sending them by email very cumbersome. For these reasons, we introduced BIMteam with version 30. The local WebViewer export will continue to be supported, but further developments of the viewer will only be developed for BIMteam.

▪ Overview

BIMteam is a cloud platform with an integrated WebViewer and constantly evolving features. You can use the platform to collect and manage your project-relevant files in one place. BIMteam is made available on a server infrastructure managed by cadwork. The platform can be accessed via any web browser. Register to test the platform free of charge at <https://bimteam.cadwork.ch/>. You can find a step-by-step video on our YouTube channel. BIMteam is already being used by some of our clients, mainly in the civil engineering field. It is still in the development phase as new functions are constantly being added. You can find an overview of the current status of the BIMteam solution on the following pages.



To be able to use BIMteam and its range of functions, you must first register, which is free of charge. BIMteam offers packages that can be purchased on a monthly basis as required. The "Free" package can be used free of charge and is available to every cadwork user after registration. There are four other packages ("Tiers"), each of which allows more users, storage space and projects to be managed. Your selected tier can be extended with additional functionalities, e.g. for the WebViewer, if required. A detailed list of the tiers and their extensions can be found [here](#).



- Structure

- Organization

BIMteam allows you to create and manage multiple organizations. Organizations can contain projects, users and payment information for corresponding license packages. Users within an organization can be assigned to different teams. Access authorizations are managed by administrative users.

- Projects

You can store associated files under the projects. BIMteam not only supports cadwork files (\*.3d, \*.2d or \*.cwl), but also files that are important for the project, such as \*.ifc, \*.dxf or \*.dwg or \*.pdf. These files can be made available to customers and other stakeholders using shared links. The viewer opens in the browser as usual via the link. The administrator can set which authorizations the individual users or teams have.



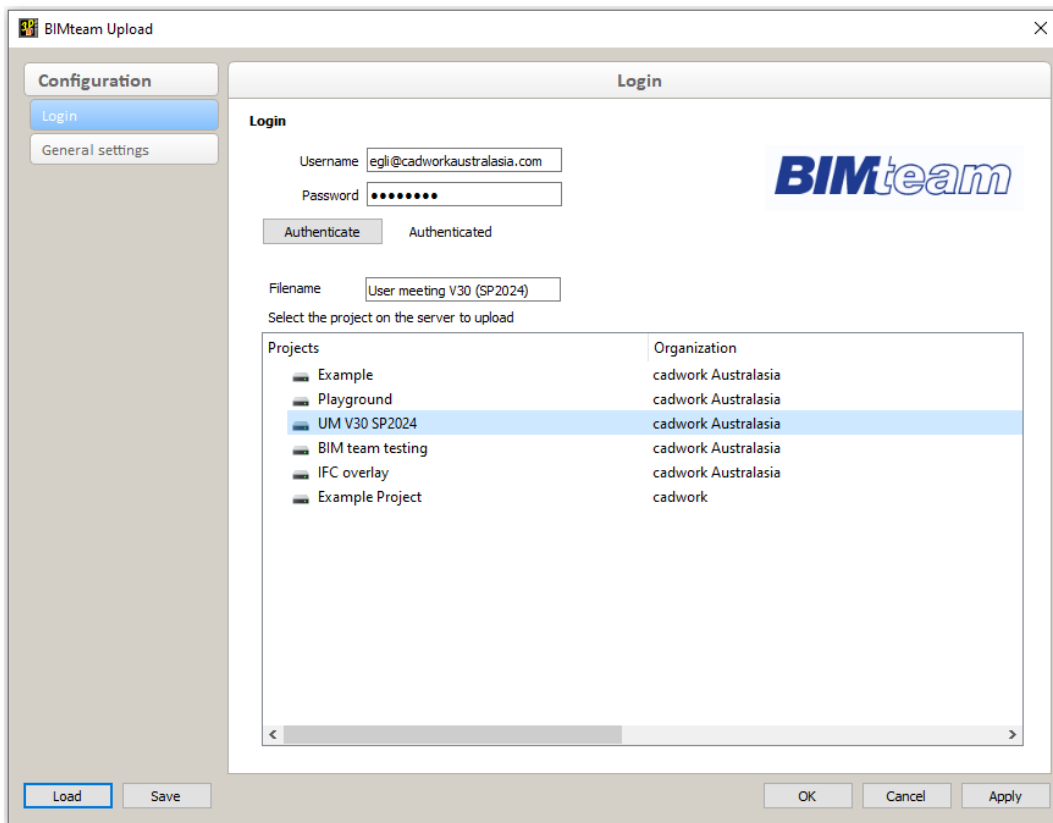
- Exporting viewer files from cadwork 3D to BIMteam

This point describes the function of exporting from the 3D as a viewer file to the platform, i.e. the area of the platform that is similar to the local WebViewer.

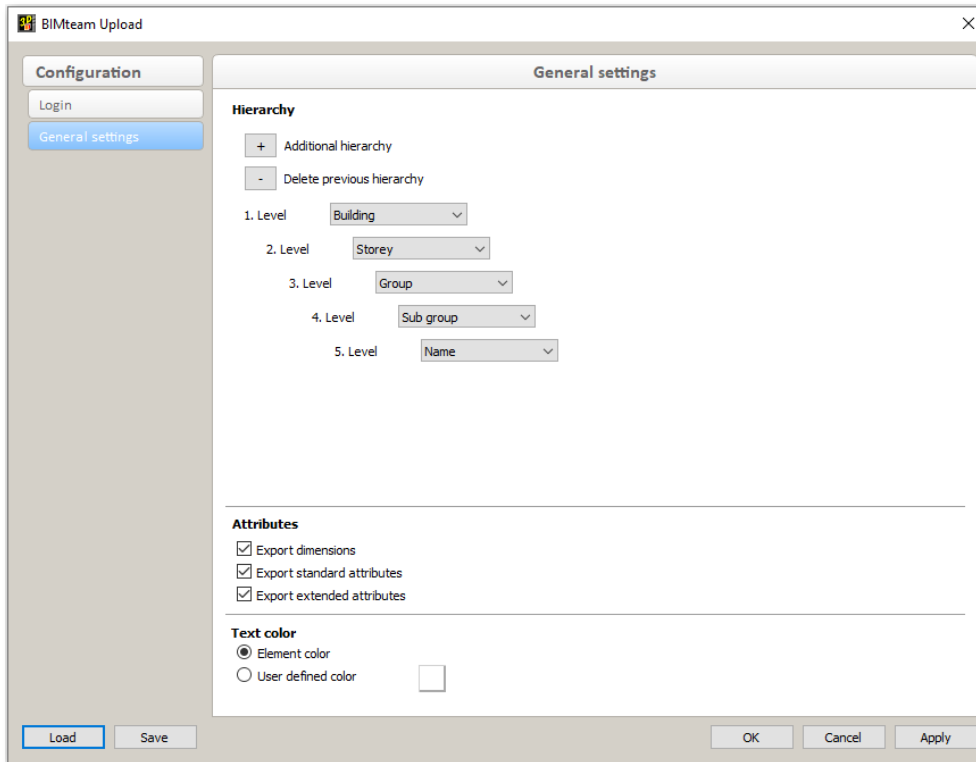


Viewer files can be uploaded directly from the cadwork 3D to a project on the platform. This is done via *Export -> Files... -> BIMteam (cloud upload)*

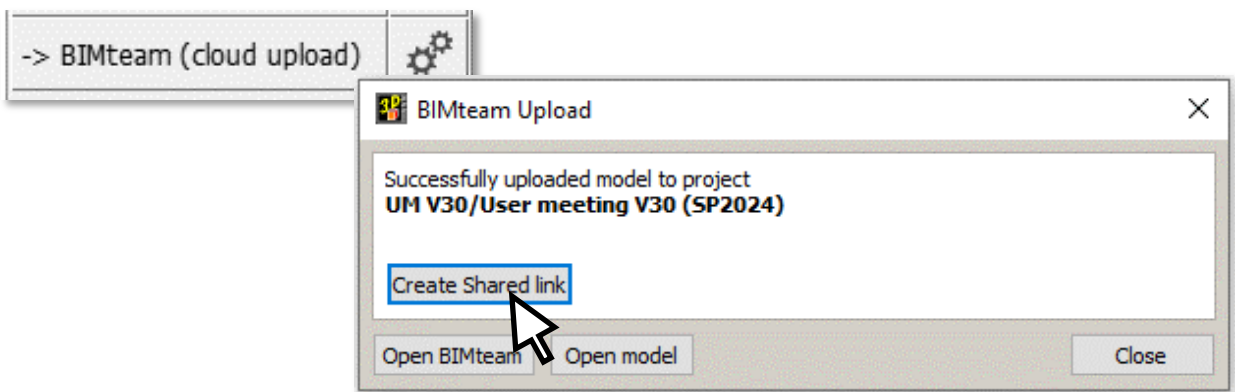
Under *Login*, you can log in to the BIMteam server and select the project to which you want to upload data. The associated organization is now also displayed in the selection.



Similar to the already familiar WebViewer export, you can also define a hierarchy for the exported file under *General settings*.

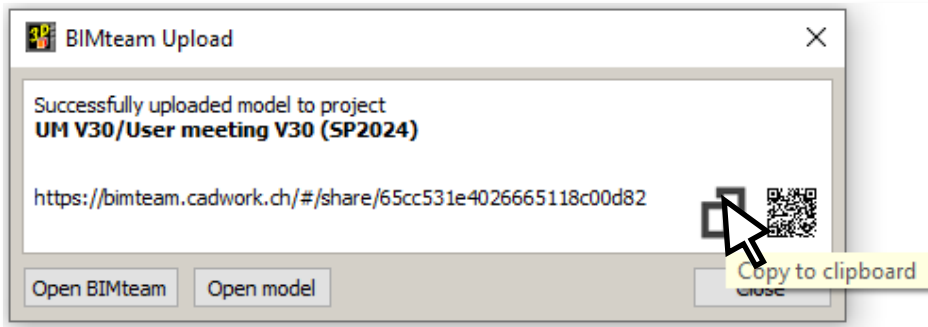


Click on *BIMteam (cloud upload)* to upload the file in \*.bte format to the selected project.



After the upload, a new window opens in which you have the following options: You can generate a link to send directly in the dialog without having to switch to the platform. After clicking on the *Share link* button, the link will be displayed. This can be saved to the clipboard using the copy symbol and pasted into an email, for example, using <Ctrl+V>. The *Open BIMteam* button takes you to the project overview on the BIMteam platform and the *Open model* button opens the project in the viewer.

Click on the QR code to generate the corresponding QR code in the clipboard. You can insert this into any file with <Ctrl+V>.



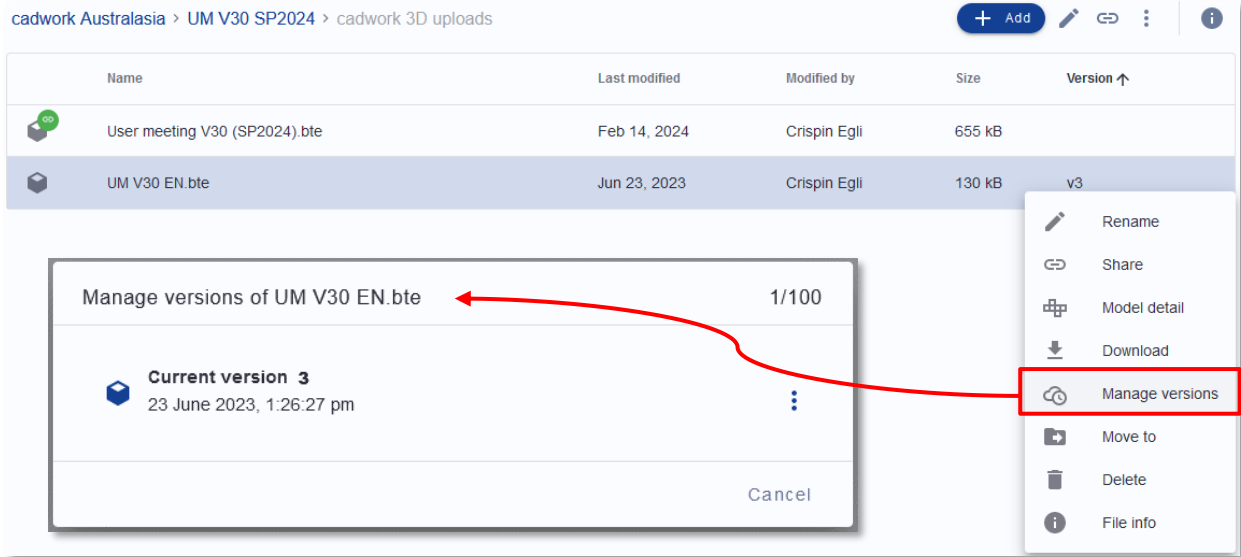
In the BIMteam web browser view, you will find the uploaded \*.bte file in a subfolder of the project called cadwork 3D Uploads. The viewer files can also be opened in the integrated viewer here by double-clicking. The same functions are available in this viewer as in the local WebViewer files.



- File versions

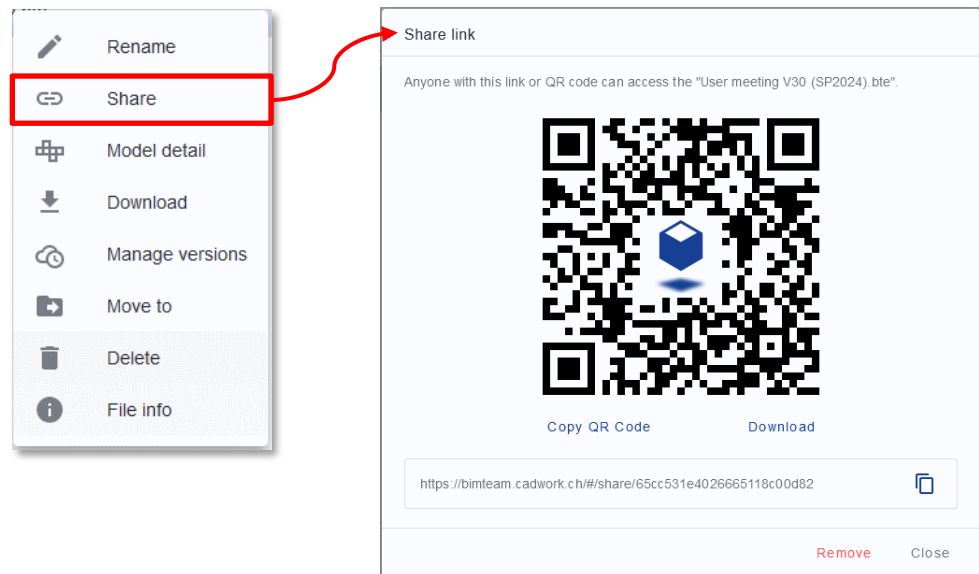
If the export is triggered again in the same file, the link does not change. Instead, a new version of the file is uploaded. This means that the link can always be used to open the current version of the viewer file.

The versions can be viewed and managed on the platform by right-clicking (R).

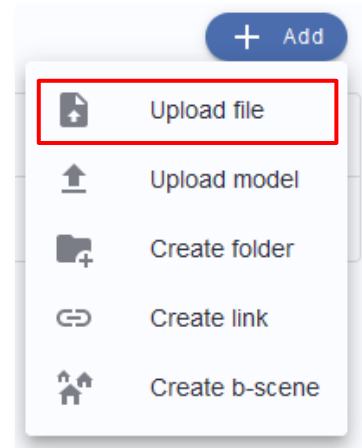


- Sharing a file on the BIMteam platform

To be able to share your project data, you need to generate a share link via the context menu on the BIMteam platform. You can access this context menu by right-clicking (R) on your file. Anyone who has access to this link will be directed to the file via the viewer.



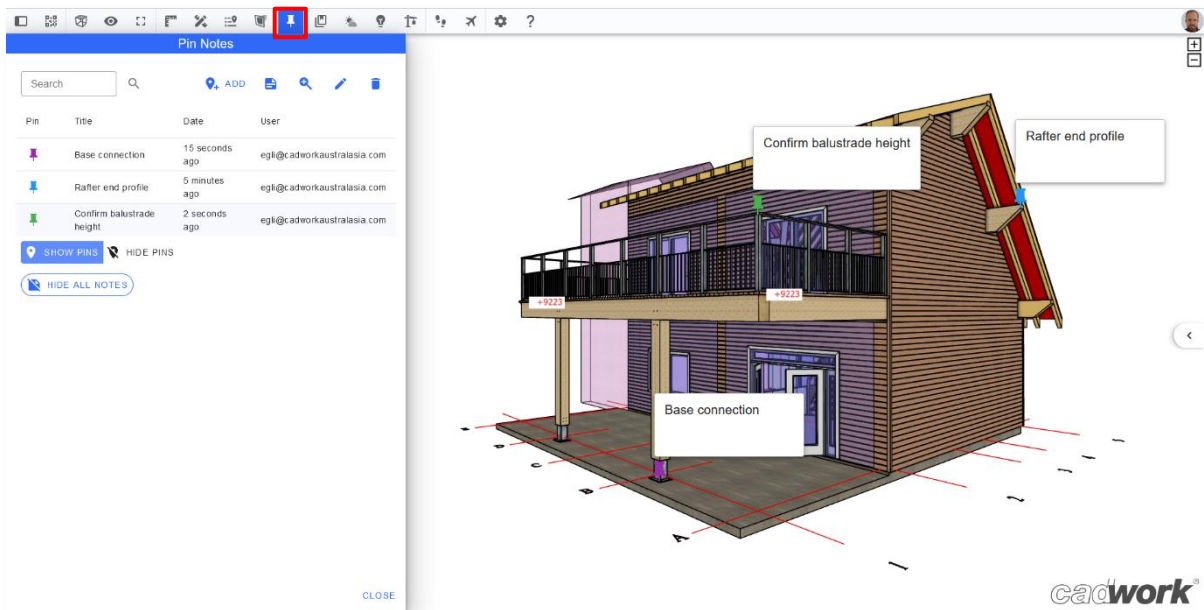
- Via *Add -> Upload file*, files such as cadwork 3D or 2D files, graphics, PDF or other file formats can also be uploaded via the BIMteam web browser portal. Many standard formats can be viewed via the integrated WebViewer. These include most graphic formats or the PDF format. Depending on your authorization, you can view, delete, upload or download any files.



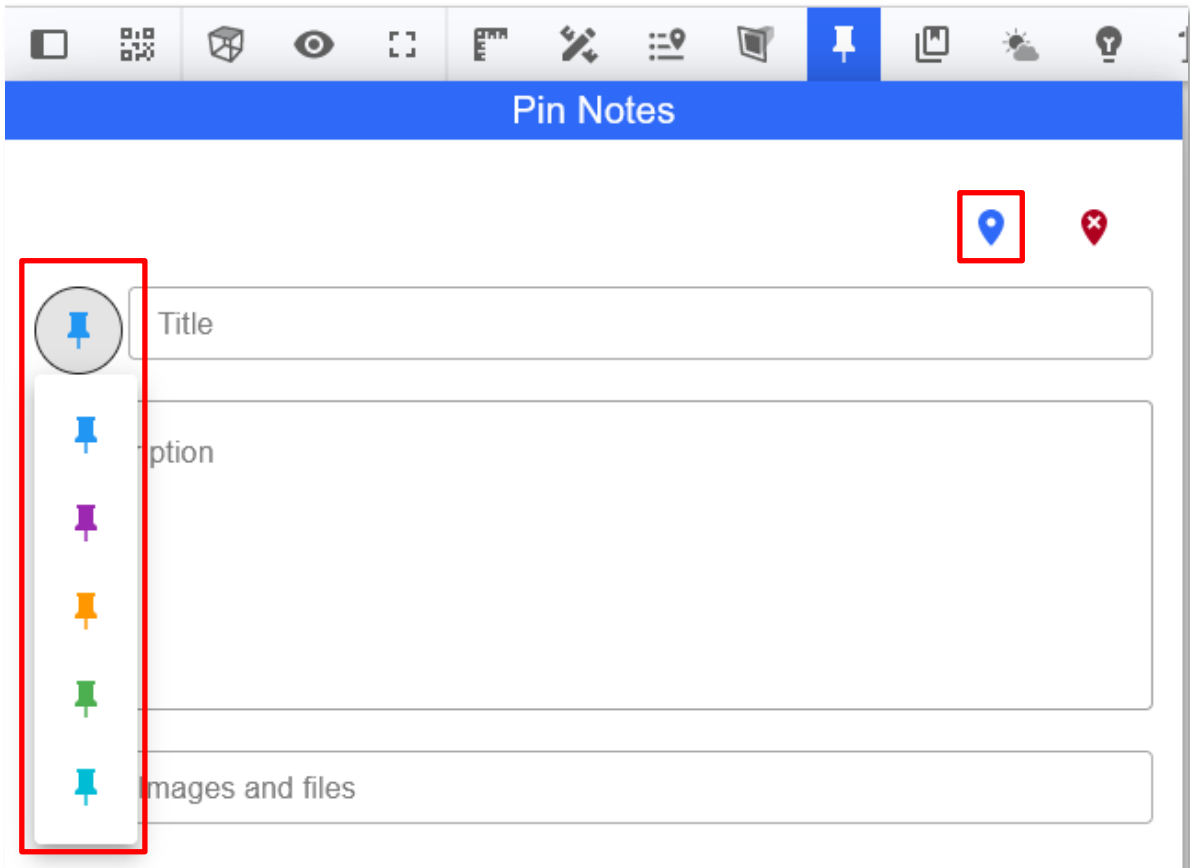
- New features since the release of version 30

- Pin notes (*Part of "Pro" Extension*):

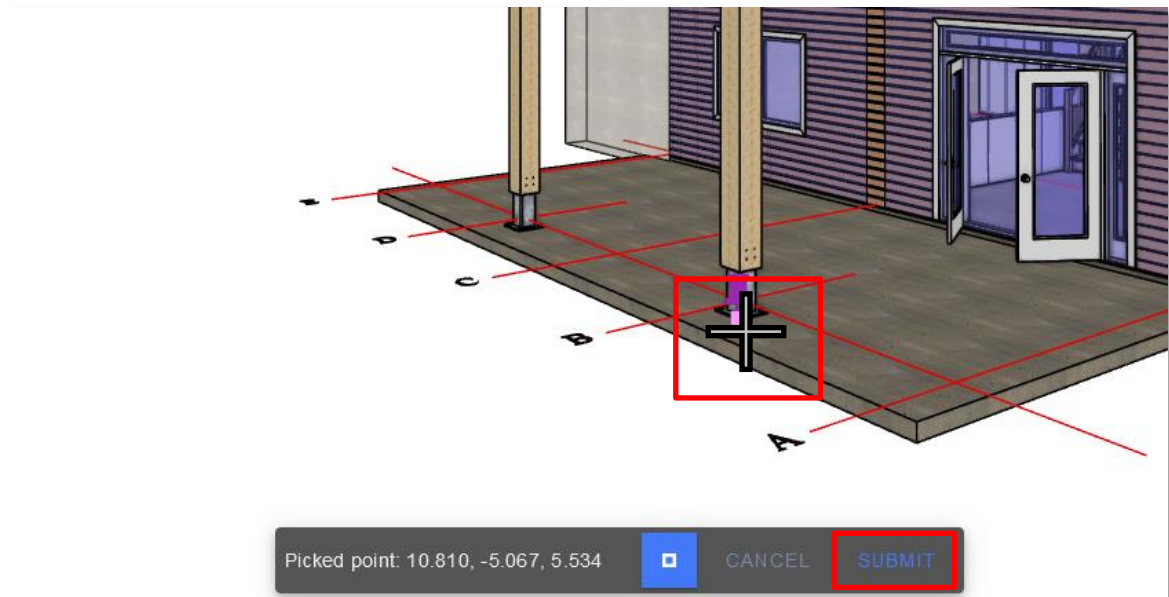
Enhance the presentation of your 3D model using pins to draw attention to specific areas. The pins can be added in five different colors, so you can differentiate between different points of interest.



Each pin can be given a title, an optional description and left-click (L) on *images and files* with files.

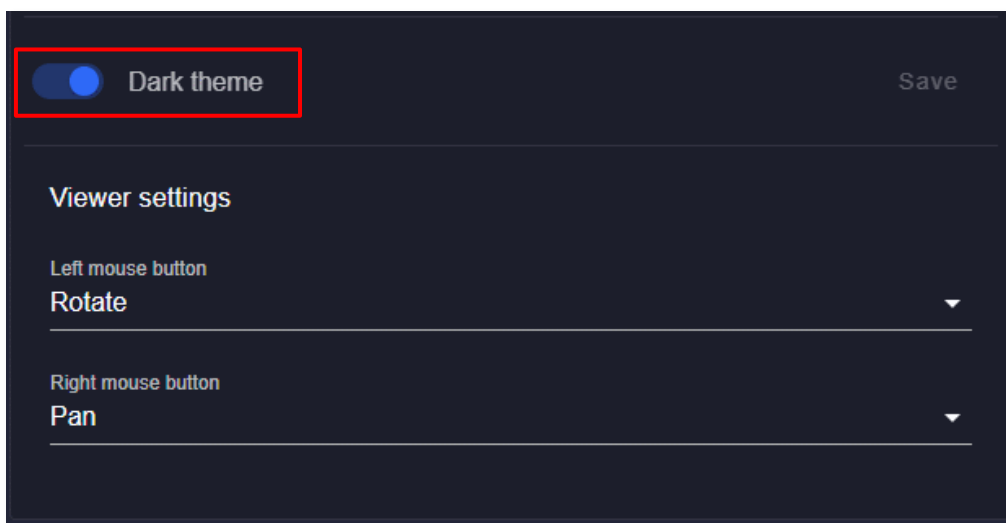


By clicking on the *Location* button, each pin must be assigned a "Location", which is selected with a point in the 3D model and defined with *Submit*.

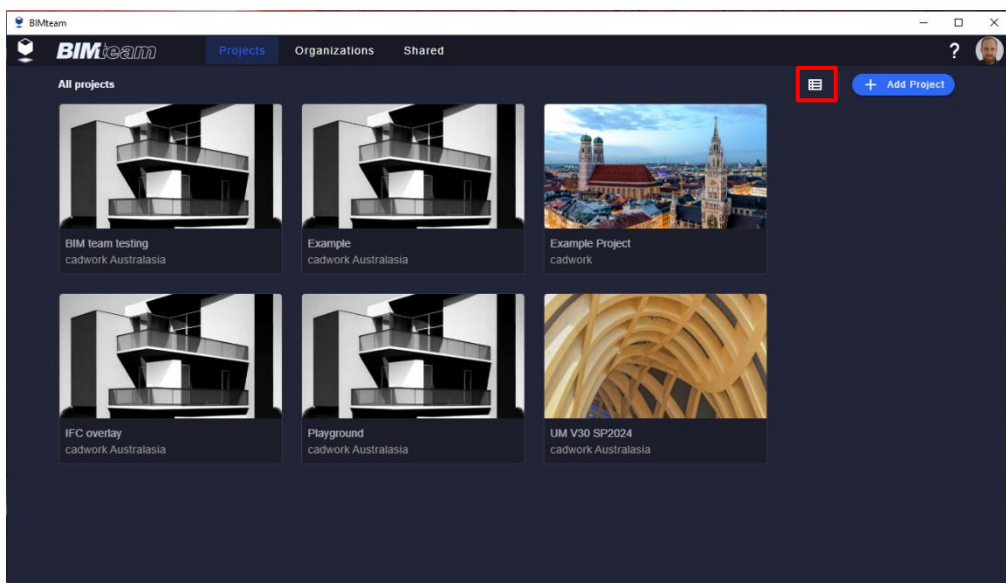


Appropriate access rights are required to create pins. Once pins have been created, you can share them with other people.

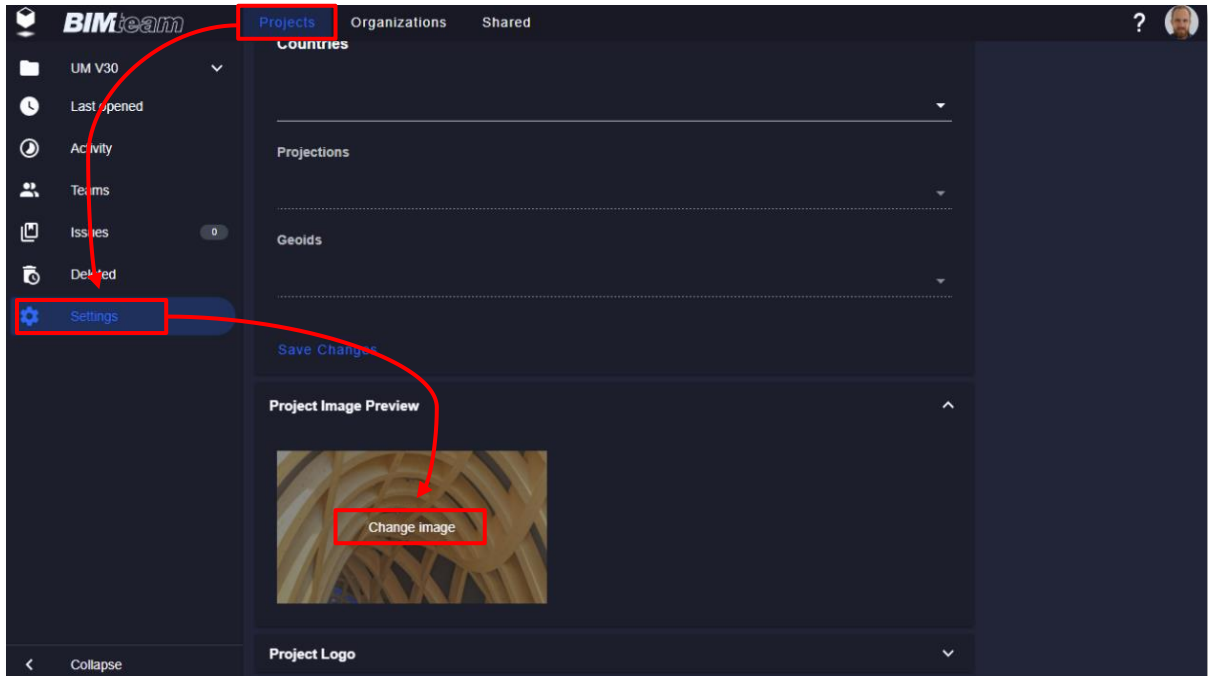
- BIMteam app  
The BIMteam app for Windows is now available. Discover the extended functionality with exclusive support for the file formats \*.dwg, \*.dxf and \*.dgn. The application can be downloaded from our website [www.bimteam.com](http://www.bimteam.com) under the menu item "Download".
- Dark and light theme for BIMteam viewer  
You can customize the appearance of the platform by choosing between a light and a dark theme in your profile settings. This setting is now also applied to the BIMteam viewer.



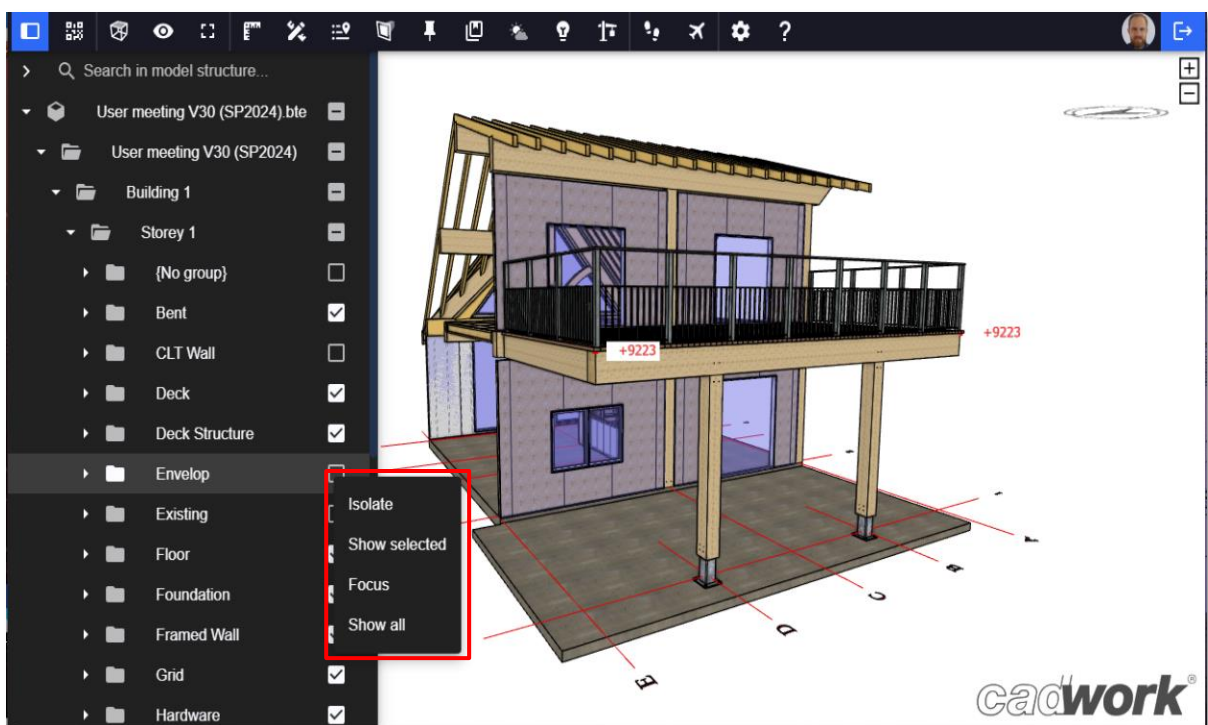
- New grid layout with project image in the *Projects* overview  
In the project overview, you can choose between two different layouts using the layout button: *Grid* or *List* layout.



To specify a project image, go to the project settings and select an image to be displayed in the grid layout.

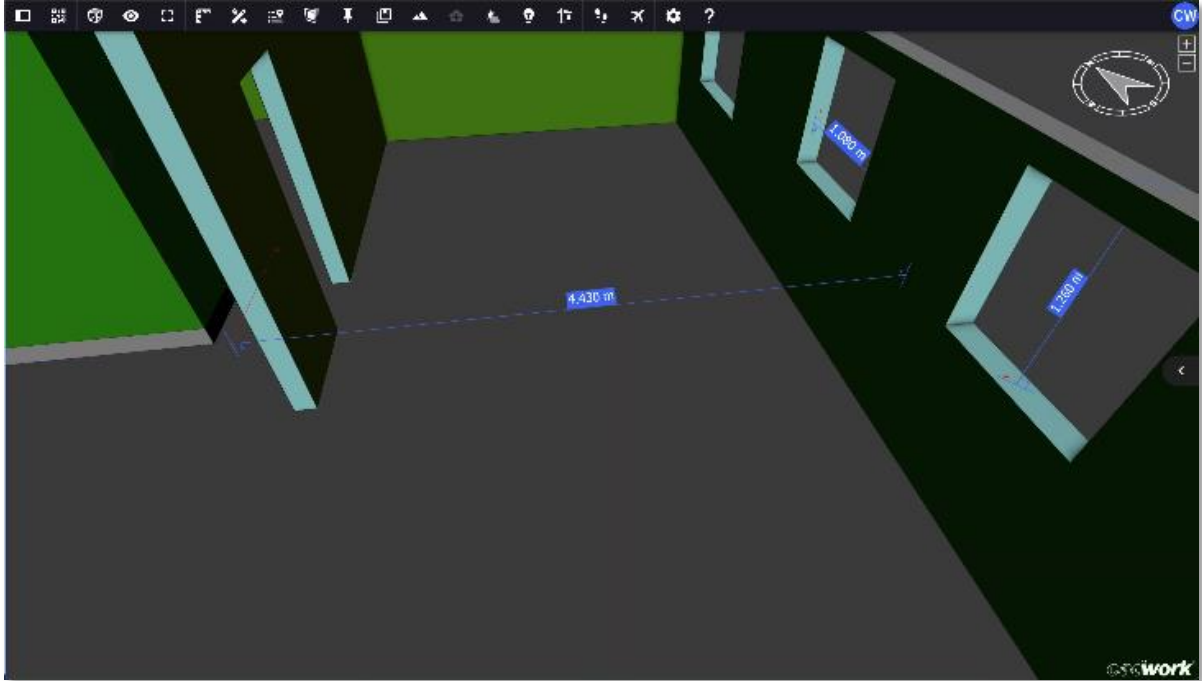


- Improved model structure and property sets for the BIMteam viewer  
 The updated model structure and property sets result in shorter loading times and improved user-friendliness. The design has been improved so that navigation is now even more intuitive. With a right-click (R) on a hierarchy level, you can select all elements within a level and make them reappear with *Show all*.

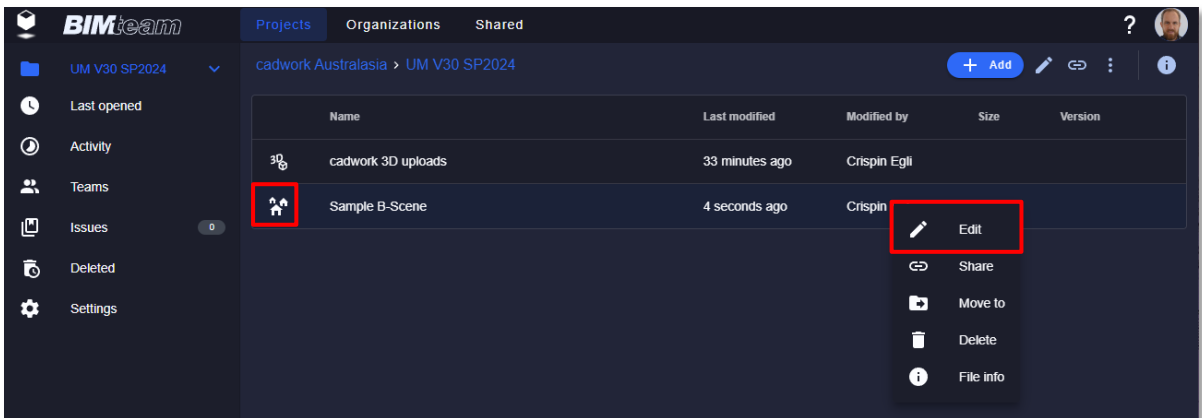


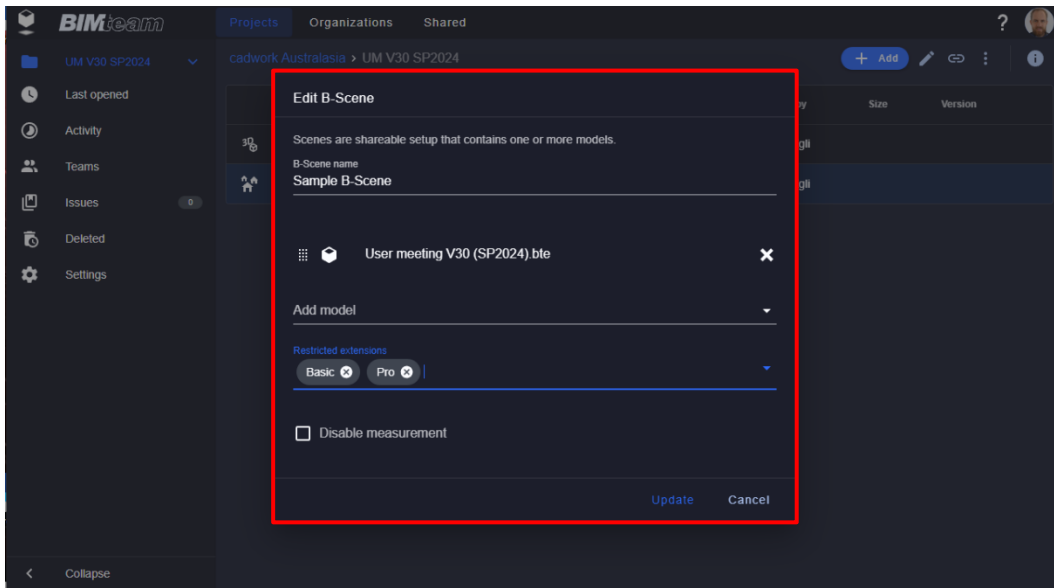


- Dimensioning tool - Normal measurement (*Part of Basic extension*)  
The dimensioning tool, which can be used to measure the normal distance between two surfaces has been added to the BIMteam viewer.

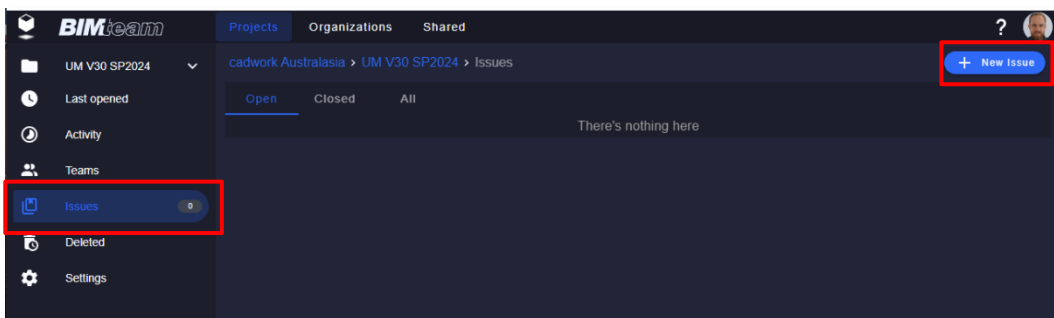


- B-scene editing  
With a B-scene, up to 50 models can be combined in one "scene".  
B-scenes can now also be edited after initial creation. To do this, right-click (R) on the desired B-scene and select the *Edit* option in the drop-down menu to add, delete or arrange the models in a desired order.

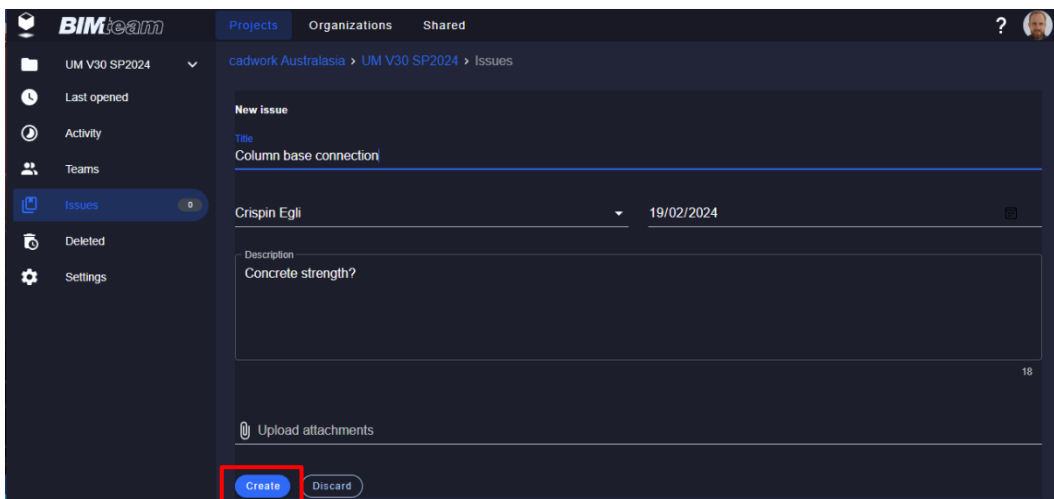




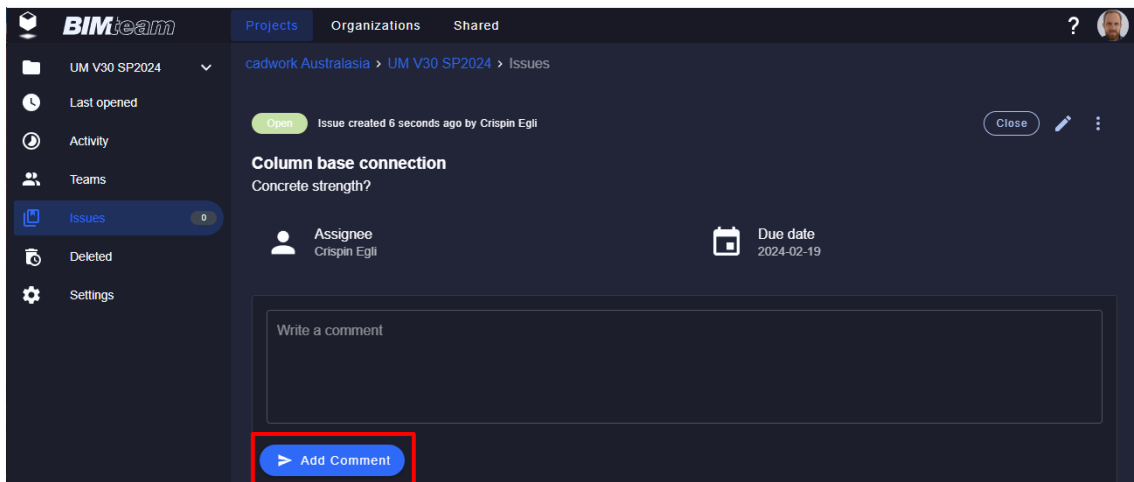
- New "Issues" feature (Part of "Pro" extension)  
You can now create issues within a project and discuss them with your team members. You will find the issues in the left-hand menu bar in your project. When you open it, you will get an overview of all open issues.



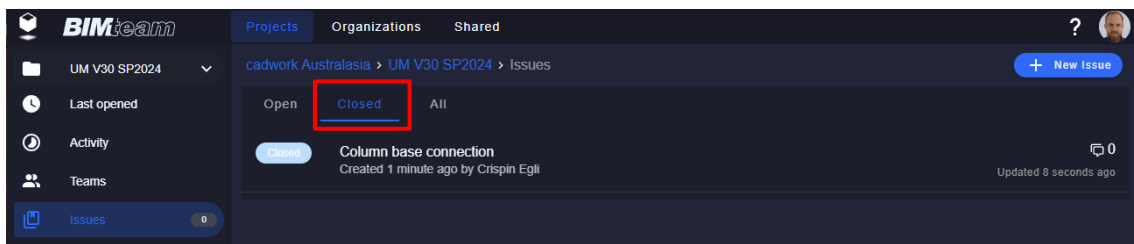
You can create a new issue by clicking on the *New issue* button and adding a title and an optional description.



Once created, you and other team members can add comments.



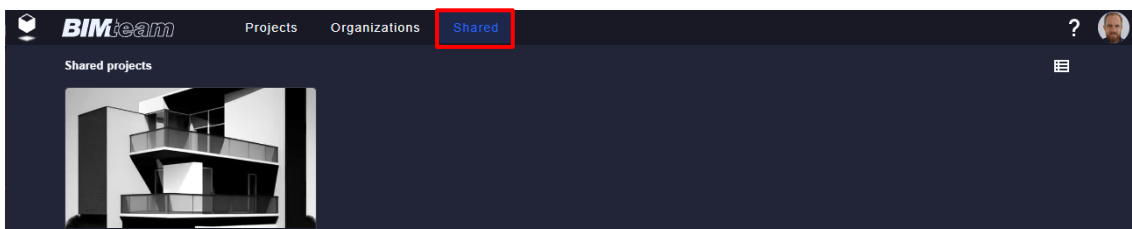
Once an issue has been completed, it can be closed and automatically moved to the *Closed issues*.



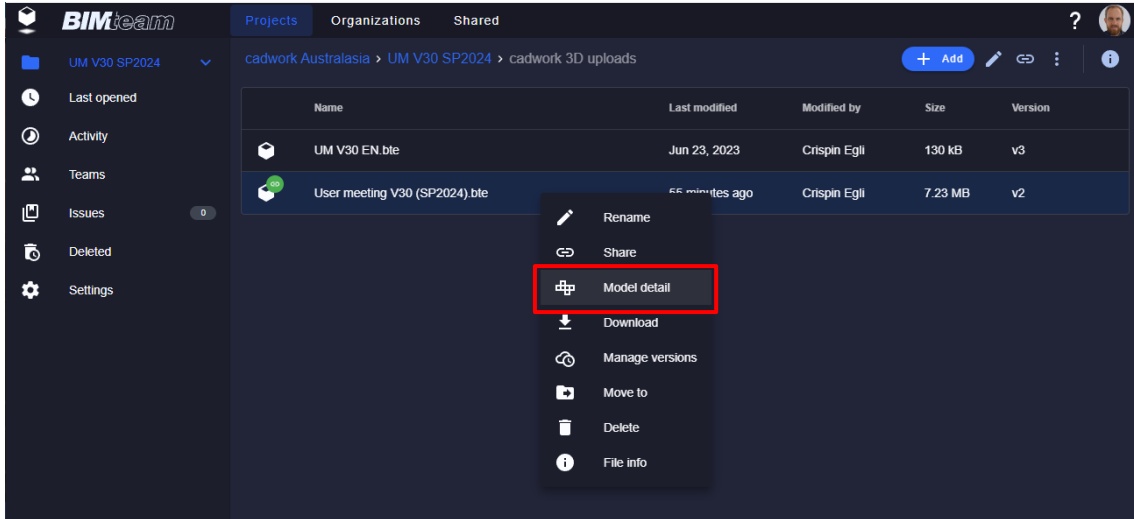
To be able to create and edit issues, you need at least Editor- permissions.

- New *Shared* button

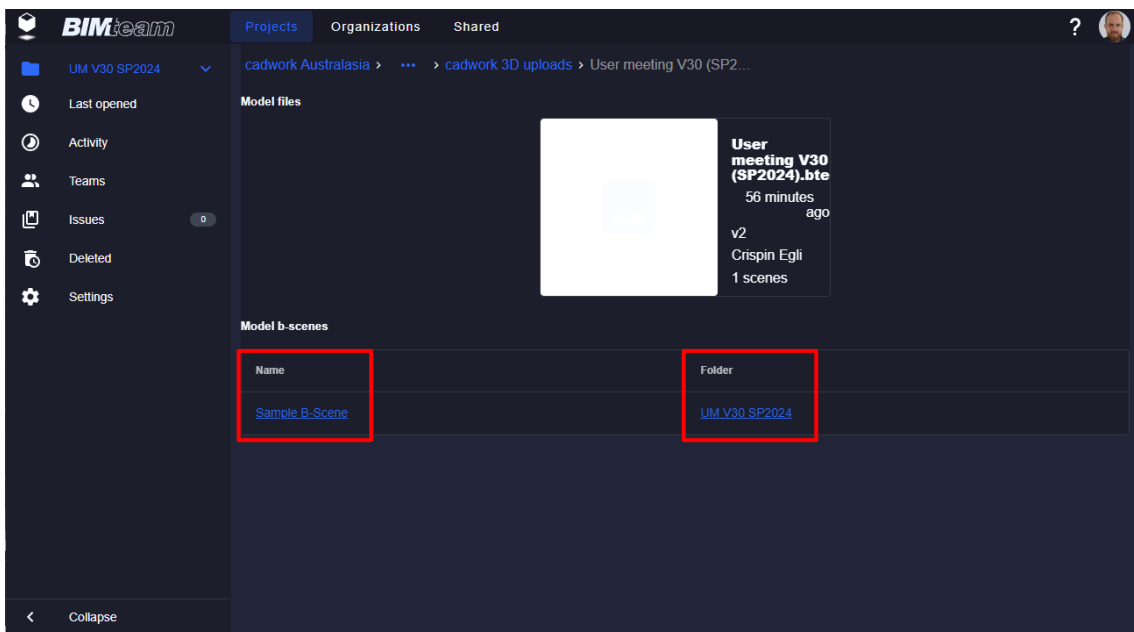
The *Shared* button takes you to a list of projects. These projects are visible because someone has shared a model or a B-scene with you via a link. If you double-click on one of these projects, you can navigate to the shared model. As you have received a share link, your access to the individual models within the shared projects is restricted.



- B-scenes linked in the model detail  
If you right-click (R) on a model and select *Model detail* from the drop-down menu, you will get a list of B-scenes that use this model.

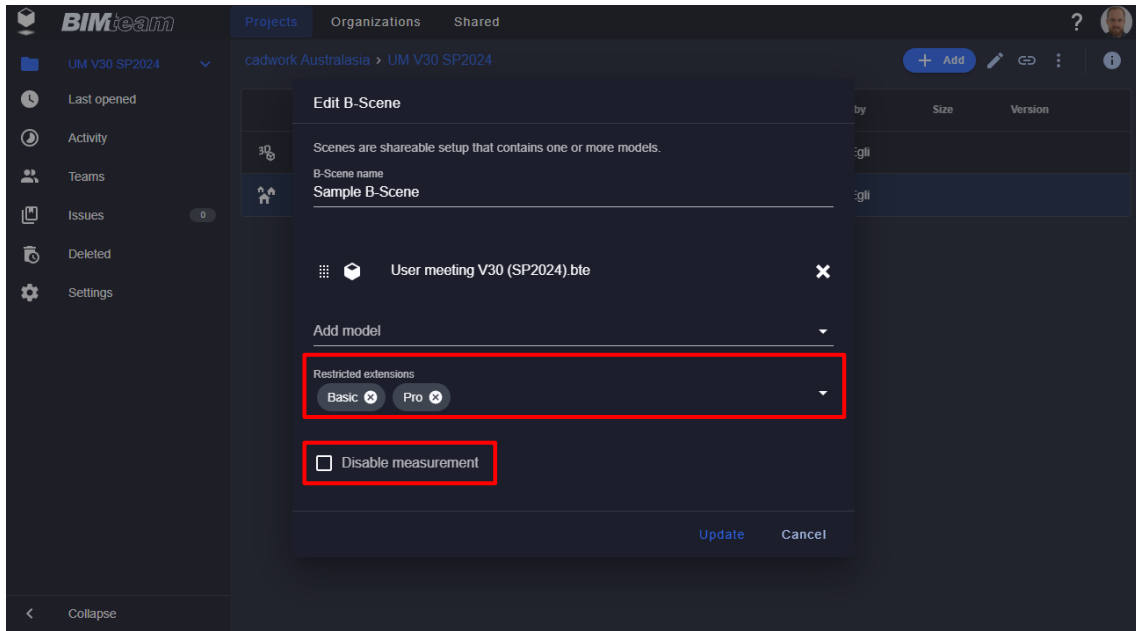


In our latest version, you have the option of opening the corresponding B-scene directly from the model detail. Alternatively, you can also navigate to the folder of the B-scene by clicking on the folder link.



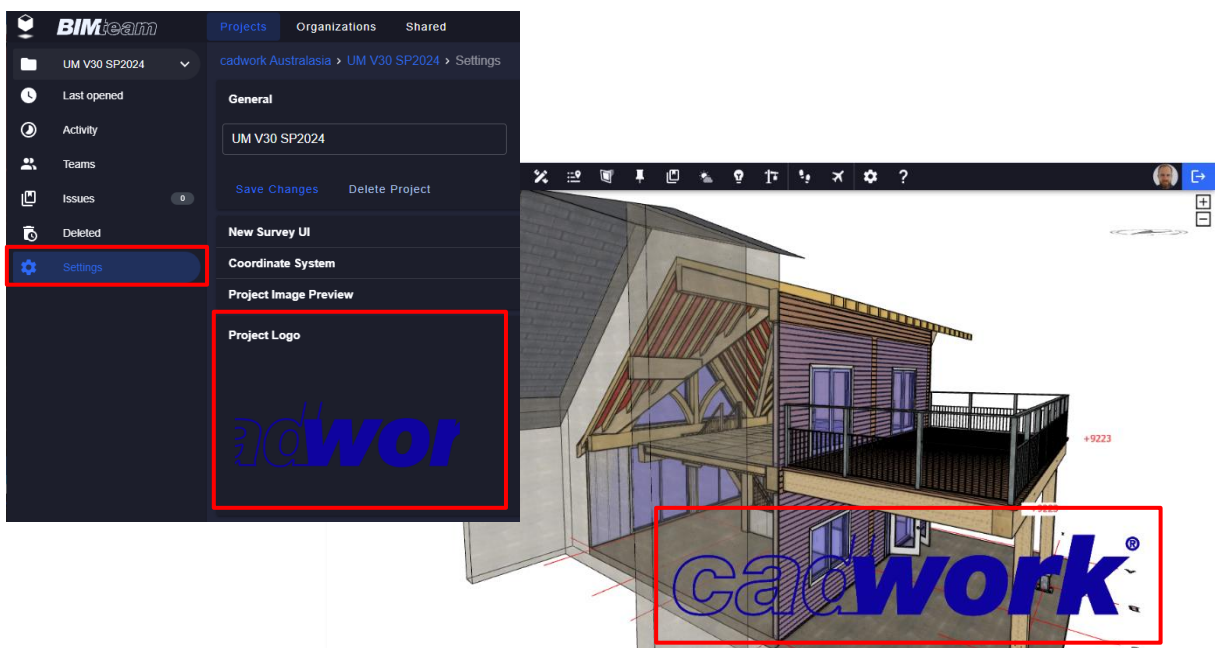
- Activating or deactivating viewer extensions and measurement function for certain B-scenes

When you create or edit a B-scene, you can now restrict some of the tools that appear in the Viewer toolbar. Select *Extensions* from the *Restricted Extensions* drop-down menu, the extensions you select will be disabled for that specific B-scene. You can also decide whether you want to activate or deactivate the measurement function.



- Customize the viewer logo (Part of "Pro" extension)

To customize your project, you can integrate a logo into the BIMteam viewer. This logo is displayed in the bottom right-hand corner (see illustration) and can be configured in the project settings under *Project logo*. This function is only available if you have purchased the "Pro" extension.



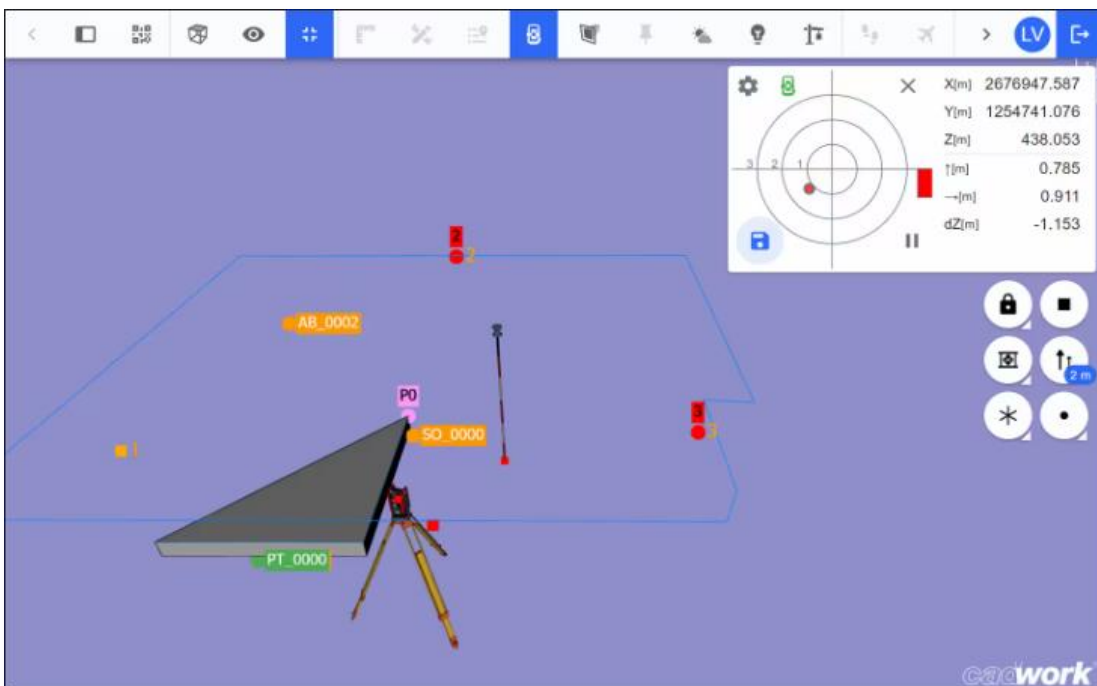
- Improved point selection with "crosshair" function for tablets (*Part of "ToField" extension*)

We have improved point selection with a new "crosshair" feature designed specifically for tablets. This improvement provides a more accurate and user-friendly



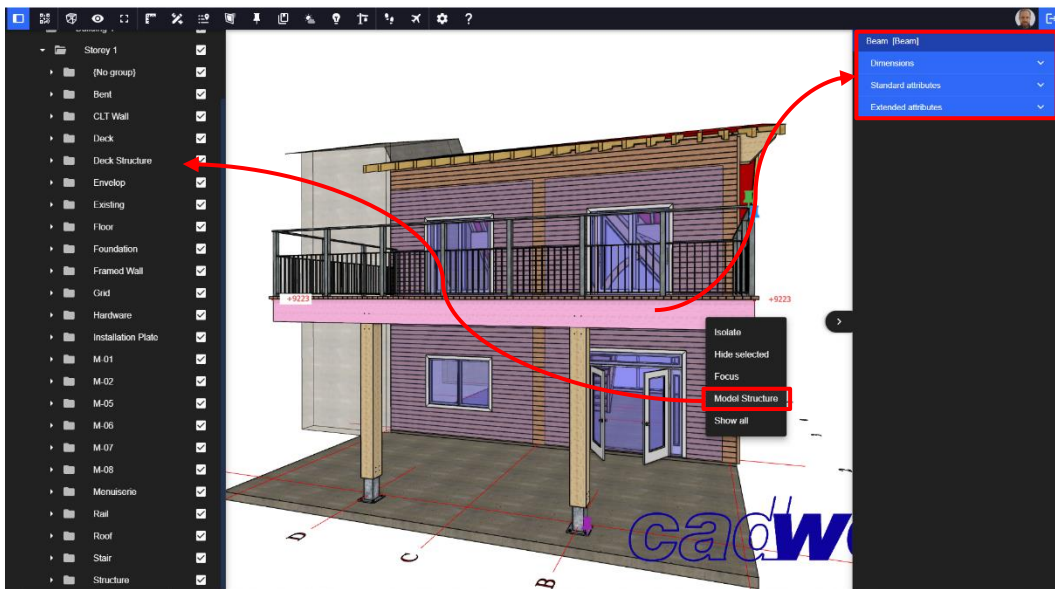
way to select points on the screen.

- New Surveying UI (User interface): (*Part of "ToField" extension*)  
Simplified interface, optimized workflows for an improved user experience  
Our new user interface combines routines with optimized workflows and instant accessibility.



- Introduction of Polyline: Improved stakeout functionality  
 We have extended our stakeout functionality by point, line and area to include stakeout by polyline.  
 This function is only available in the new user interface.

- Model structure - Behaviour refined  
 When you select an element (L), it is no longer automatically displayed in the model structure. Instead, the element and folder names have been added to the top of the properties window to provide clear navigation hints. If you want to find the element in the model structure, simply right t-click (R) on it and select *Model Structure* from the drop-down menu.



- Improved search functionality for the BIMteam viewer  
 The search function now not only highlights the elements you are looking for, but also opens them directly.



- Multiple selection in the BIMteam viewer  
With the familiar combination of <Ctrl> (<CMD> on Mac) and model selection, you can now activate multiple elements. This function is available both in the model structure and in the 3D model as well as a combination.



Selection in 3d model



Selection in model structure



## ❖ Appendix

### ➤ Keyboard options

#### ▪ <Ctrl+Tab>Reverse activation

If elements are already active, the function swaps the activation for all visible elements. All active elements are deactivated and the elements that are not active when the function is called are activated.

### ➤ Function keys F1-F12

#### ▪ Activate attribute

- Docking saw plus(see page 51)
- Activate parts – mitre off cuts only (see page 51)
- Activate parts – mitre and bevel off cuts(see page 51)

#### ▪ Machine

- Panel prefab... -> Weinmann WUP 3.5