HiCAD

UNLIMITED CAD PERFORMANCE **DEVELOPED BY ISD**

What's new?

Version 2024

News Overview

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THE WORLD OF CAD AND PDM SOLUTIONS

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Discontinuations

Discontinuation of Windows® 7 and Windows® 8

Microsoft® has discontinued support for the Windows® 7 operating system in January 2020. For compatibility reasons, HiCAD 2020 SP2 and HELiOS 2020 SP2 were the last versions of our CAD or PDM system to support Windows© 7. HiCAD 2021 and HELiOS 2021 no longer run under Windows© 7, Windows© 8 and the corresponding server operating systems (Windows Server 2008 R2, Windows Server 2012 and older) are also no longer supported. If an attempt is made to install HiCAD 2021 or HELiOS 2021 on a computer with Windows© 7 or Windows© 8, a message appears.

Discontinuation of "old" HiCAD itemization

As of HiCAD 2019 the “old” itemisation, i.e. the itemisation that was used up to HiCAD 2017, will only be available for model drawings that were already itemized with these functions. From HiCAD 2021 onwards, only the “new” itemization will be supported. Please also read the information given in the Conversion of Old Itemisations topic.

Discontinuation of "old" OpenGL versions

From HiCAD 2021 on, only OpenGL version 4.3 is used in all HiCAD modules. Until now this was only the case with the module **HiCAD Point Cloud**. This means that HiCAD 2022 can no longer be run on computers without a separate graphics card. To avoid possible problems with onboard graphics cards, we recommend using a stand-alone graphics card.

Discontinuation of old figure format (FIG)

The following notes regarding FIG-FGA conversion are unnecessary if HELiOS is used in conjunction with the HELiOS Vault Server.

Since HiCAD 2017 we support FGA as figure format (before that FIG). From HiCAD/HELiOS 2021 or HELiOS 2021 as an update for HiCAD 2019/2020 onwards, we require that all figures stored with HELiOS have been converted to the new FGA format beforehand. To convert existing 2-D FIG files, the tool Converter_FIG_To_FGA.exe is available in the exe directory of the HiCAD installation.

If there are still unconverted FIG files in the HELiOS document database at the time of the database update, you will be informed of the outstanding conversion of these files before the database update. In this case, the conversion must be carried out before or at the latest directly after the update using Converter_FIG_To_FGA.exe.

Discontinuation of the "old" Create detail drawing function

With the release of HiCAD 2012, the previously valid workshop drawing functionality in Steel Engineering had been extended to a function for general drawing derivation. The previous functions for detail drawings in Steel Engineering were still available in the Detail drawing section of the Drawing menu. As of HiCAD 2022 (Version 2700.0) these functions are no longer supported.

Discontinuation of HELiOS 32 Bit, HiCAD Viewer 32 Bit and Office Interface 32 Bit

Since HELiOS 2022 (Version 2700.0) there is no 32 Bit version available for HELiOS and the HiCAD Viewer. The HELiOS interface for a 32 Bit Office was one of the few components that was still 32 Bit on these versions. Since Microsoft has also offered a 64 Bit installation of Office since Office 2010 and many add-ins for Office are now also available as 64 Bit versions, with HELiOS 2024 we will support an Office interface only for a 64 Bit Office. If you are still using the Office interface in conjunction with a 32 Bit Office, you must uninstall your Office version and reinstall it as a 64 Bit version as soon as you update to HELiOS 2024.

Discontinuation of CADENAS PARTdataManager

As of HiCAD 2022 SP2, the CADENAS **PARTdataManager** will no longer be supported. Thus, the functions **Insert main part, PARTsolutions (CADENAS program)** and **Import PARTsolutions part** will no longer be available from SP2 onwards.

Discontinuation of 3-D projection grid

As of HiCAD 2023, the **3-D projection grid function** is no longer available.

Discontinuation of the Zuken E3 interface

As of HELiOS 2024 (Version 2900.0) we will no longer support the Zuken E3 interface.

Discontinuation of the "old" Report Manager

From HELiOS 2024 onwards, the "old" Report Manager, i.e. the Report Manager up to 2022, will no longer be delivered with a standalone installation of the HELiOS Desktop. In a HiCAD/HELiOS installation or a HELiOS update of HiCAD, however, the "old" Report Manager is still included. From HiCAD 2025 onwards, only the "new" Report Manager as of 2023 will be supported.

Discontinuation of Part insertion v26 (PE)

The old **Part insertion** function in Plant Engineering has now been completely replaced by the new **Part insertion** function. Therefore, the function **Part insertion v26** in Plant Engineering is no longer available as of HiCAD 2024 SP1.

Discontinuation of Part type catalogue, 3-D

The function **Civil Engineering - Part type catalogue, 3-D** in the **Civil Engineering functions** docking window at **Civil engineering, general** contained functions that have now been replaced by new developments and are no longer required. The function is therefore no longer available as of HiCAD 2024 SP1.

Basics

Service Pack 1 2024 (V 2901)

Licensing

The **Mechanical Engineering Package** is a new HiCAD basic module. The range of functions corresponds to that of the **Mechanical Engineering Suite Premium**, but the number of parts is limited to 750.

Changing the size of the transparent toolbar

The size of the transparent toolbar can now be adjusted directly in HiCAD.

If you click on the  symbol with the right mouse button, a corresponding slider is displayed.

Part attribute masks modernised (SP1)

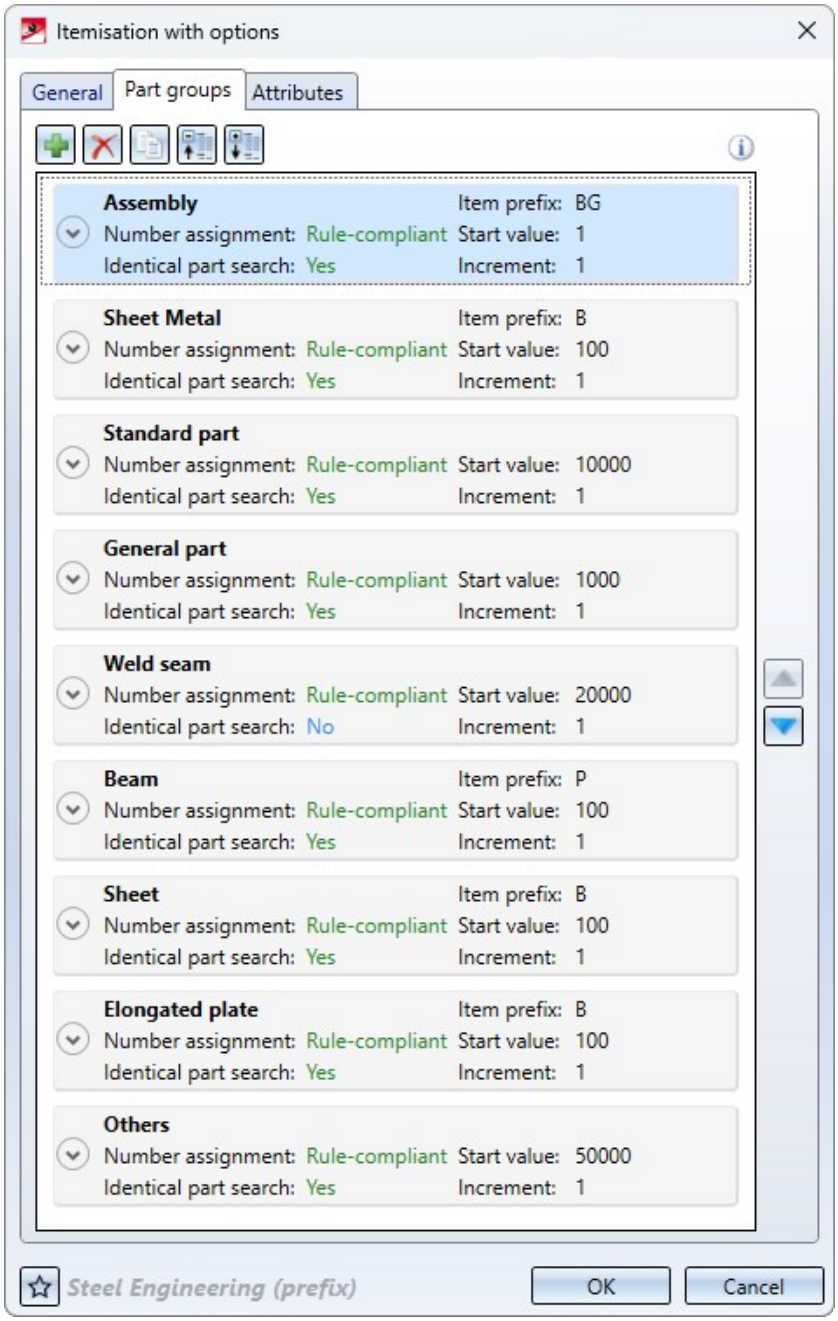
The attribute masks for parts and assemblies have been modernised. In addition, their size now adapts to the screen resolution.

The HDX files on which the masks are based have not changed as a result.

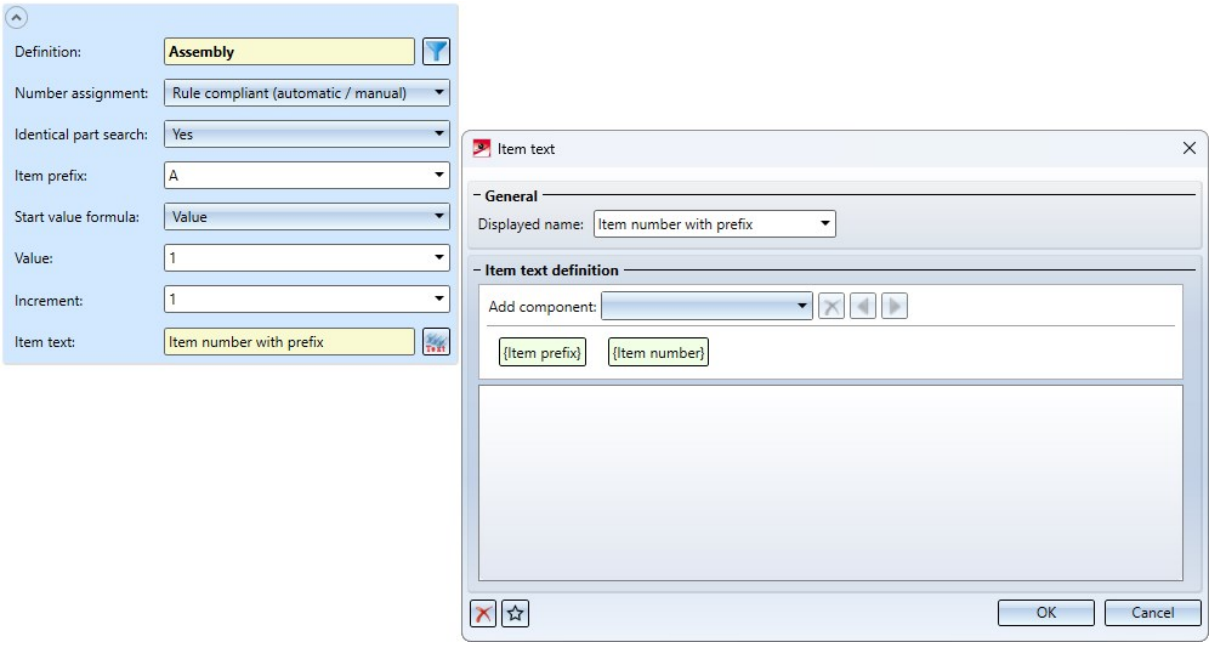
Automatic itemisation - numbering according to part groups with prefix

As of SP1, an item prefix can be defined for each part group in the parameter settings for **Itemisation**  , provided that the generation of item texts is activated on the **General** tab. A part is then only uniquely identified by item prefix + item number. The parts are then not only differentiated according to the item number as a number, but the item prefix is also taken into account during itemisation. This means, for example, that an assembly and a profile can have the same item numbers, but the prefixes are different.

To use item prefixes, you must load the ISD template **Steel Engineering (item text with prefix)** in the favourites, then adapt it if necessary and save it as a new template. Prefixes for assemblies (BG) and beams (P) as well as for sheet metal, sheets and elongated plates (B) are defined in this template:



Item number with prefix is permanently entered in the **Item text** field. This configuration can be changed by clicking on  . As with "normal" item texts, you can change the order of the components or add further components.



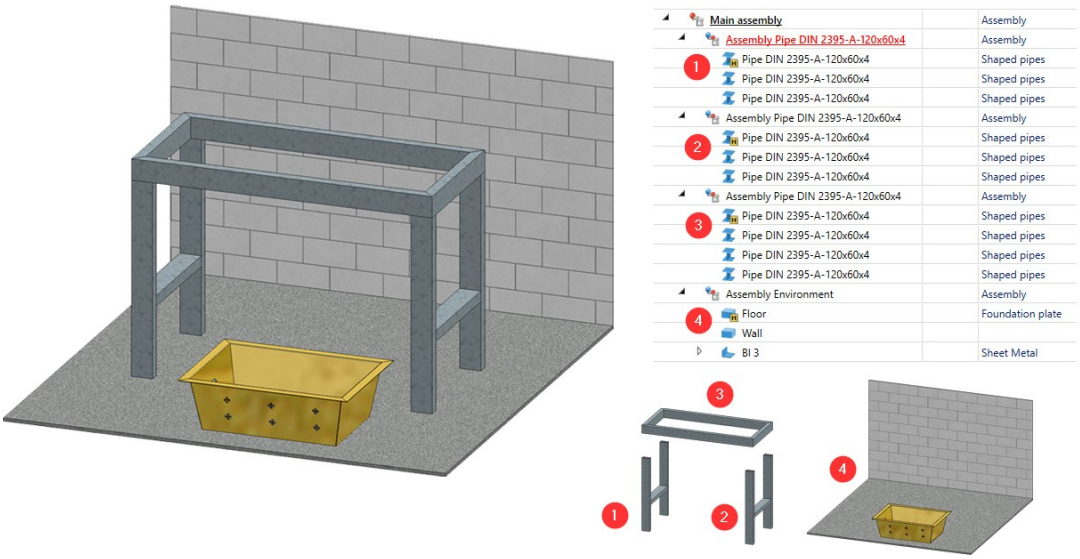
Please note that numbering by part group with prefix is not supported by the drawing management,

Please note:

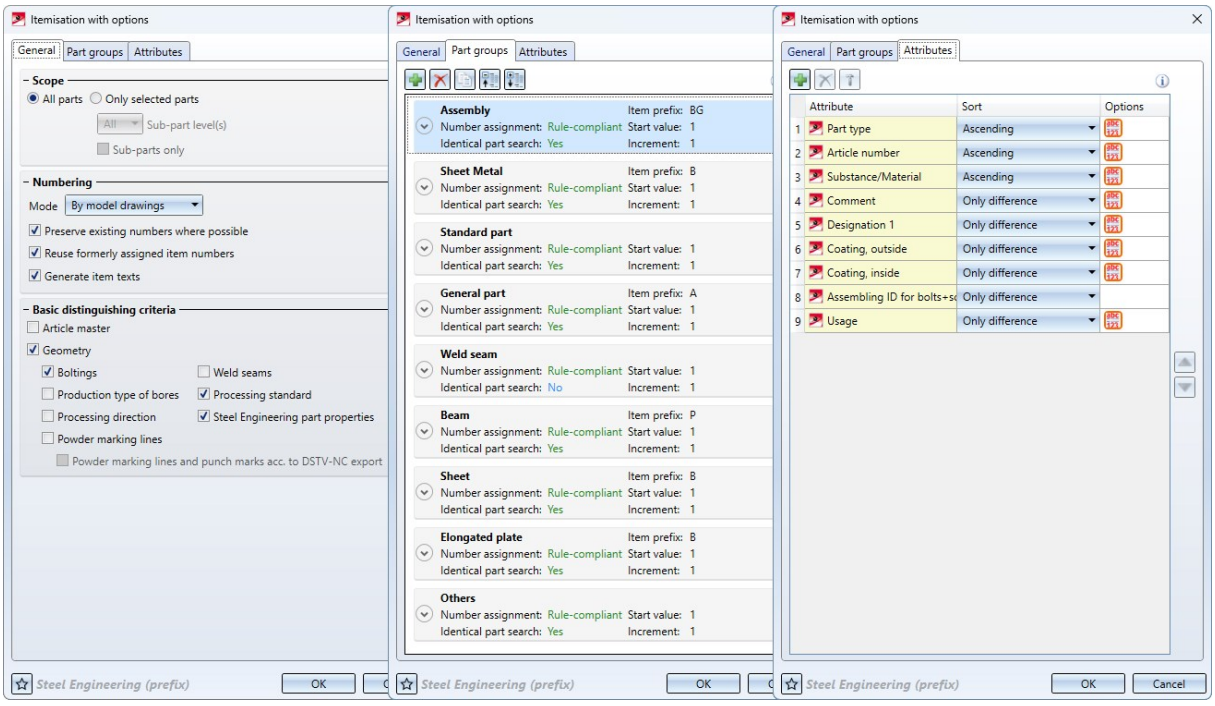
- The item prefix is assigned to the part attribute **$PPFX**.
- The annotation templates included with HiCAD only contain the item number without the prefix. If you also want to use the item prefix here, you must adapt the templates accordingly.
- In the bill of materials, the item number with prefix is displayed in the **Item text** column.

Example

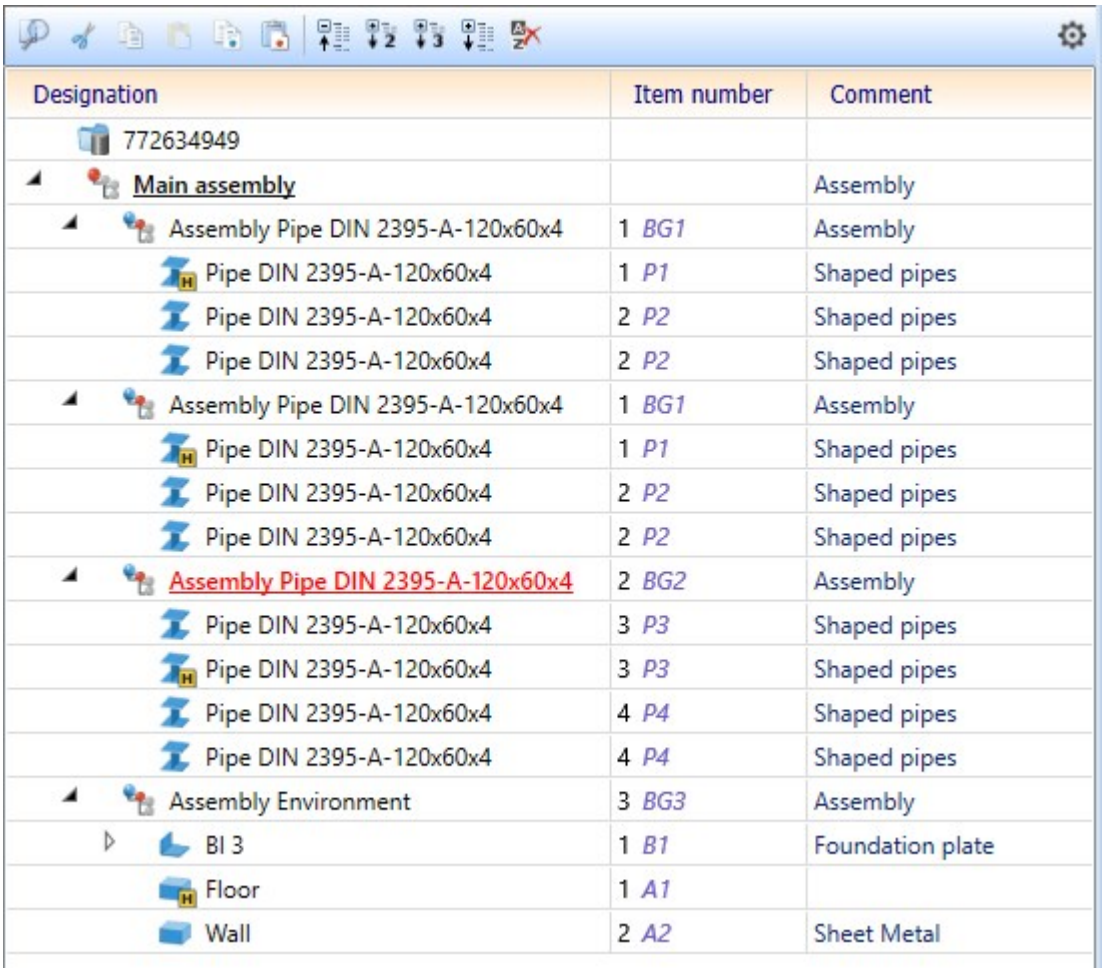
The drawing shown should be itemised with a prefix.



We use the ISD template Steel Engineering (prefix). On the **Part groups** tab, we set all start values to 1 (for illustration purposes) and define the prefix **A** for the **General part** group.



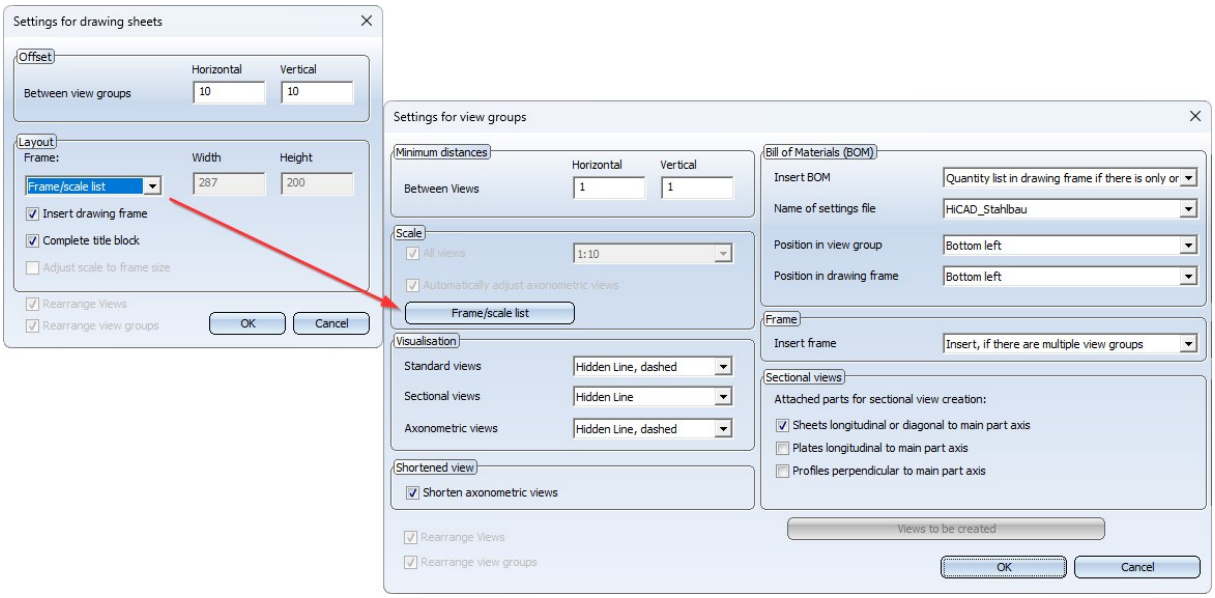
The following figure shows the result of the itemisation.



Designation	Item number	Comment
772634949		
Main assembly		Assembly
Assembly Pipe DIN 2395-A-120x60x4	1 *BG1*	Assembly
Pipe DIN 2395-A-120x60x4	1 *P1*	Shaped pipes
Pipe DIN 2395-A-120x60x4	2 *P2*	Shaped pipes
Pipe DIN 2395-A-120x60x4	2 *P2*	Shaped pipes
Assembly Pipe DIN 2395-A-120x60x4	1 *BG1*	Assembly
Pipe DIN 2395-A-120x60x4	1 *P1*	Shaped pipes
Pipe DIN 2395-A-120x60x4	2 *P2*	Shaped pipes
Pipe DIN 2395-A-120x60x4	2 *P2*	Shaped pipes
Assembly Pipe DIN 2395-A-120x60x4	2 *BG2*	Assembly
Pipe DIN 2395-A-120x60x4	3 *P3*	Shaped pipes
Pipe DIN 2395-A-120x60x4	3 *P3*	Shaped pipes
Pipe DIN 2395-A-120x60x4	4 *P4*	Shaped pipes
Pipe DIN 2395-A-120x60x4	4 *P4*	Shaped pipes
Assembly Environment	3 *BG3*	Assembly
Bl 3	1 *B1*	Foundation plate
Floor	1 *A1*	
Wall	2 *A2*	Sheet Metal

Frame/scale list

The **Frame/scale list**  function has been removed from the **Drawing> Itemisation/Detailing > Derive...** menu and integrated into the automatic drawing derivation dialogue. The **Frame/scale list** option can now be selected in the **Drawing sheets** dialogue window under **Frame**. If the option is selected, the corresponding dialogue window can be started in the **View groups** dialogue window with the **Frame/scale list** button.

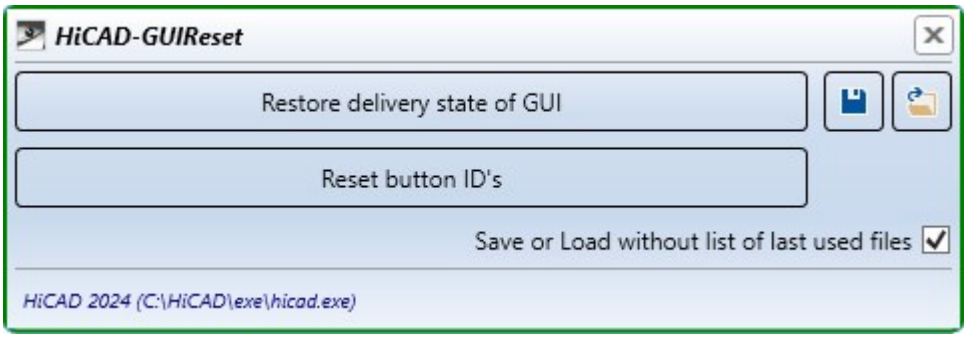


In the **Frame/scale list** dialogue window, you can then define separately for each intended use, e.g. for assemblies, I-beams, sheet metal, etc., which drawing frames should be used in combination with which scales.

By integrating the frame/scale lists into the drawing derivation dialogue, these are also taken into account when updating drawings.

HiCADGUIReset - List of documents used

The tool HiCADGUIReset, with which the entire HiCAD user interface can be reset to the default state after the HiCAD installation, has been extended.

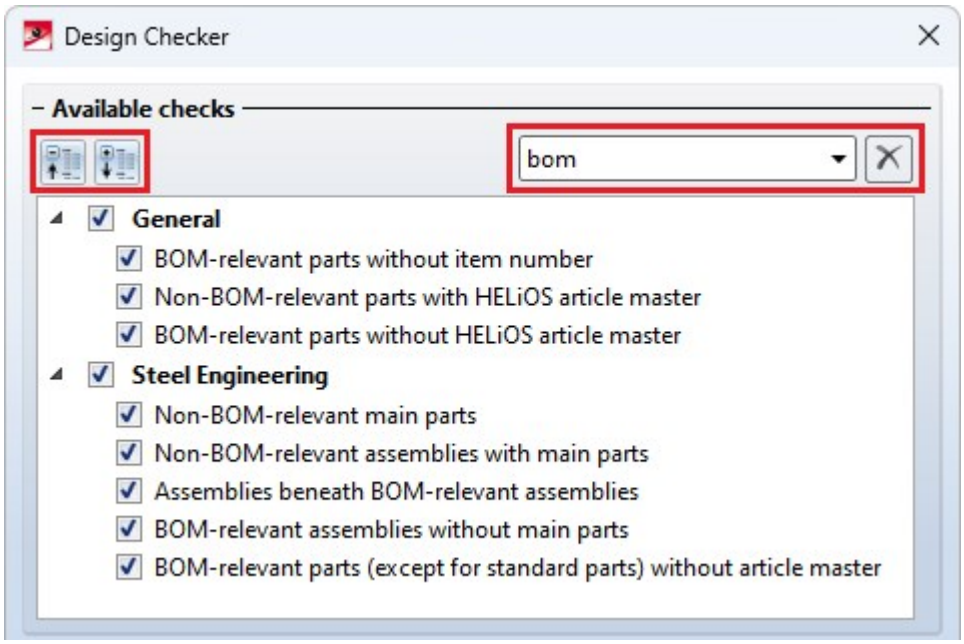


New is the checkbox **Save or Load without list of last used files**. By activating or deactivating this checkbox, you determine whether the list of recently used documents should also be saved or loaded when saving and loading the GUI settings. The checkbox is activated by default, i.e. the GUI settings are saved and loaded without the list of recently used documents.

Design Checker with improved usability

The Design Checker has become more user-friendly with Service Pack 1:

- When selecting/deselecting tests, the usual Windows multiple selection with SHIFT and CTRL is also supported.
- Use the symbols  or  to collapse or expand all test groups in one step.
- If you enter a search term in the search field, HiCAD lists all tests whose name contains the specified search term.

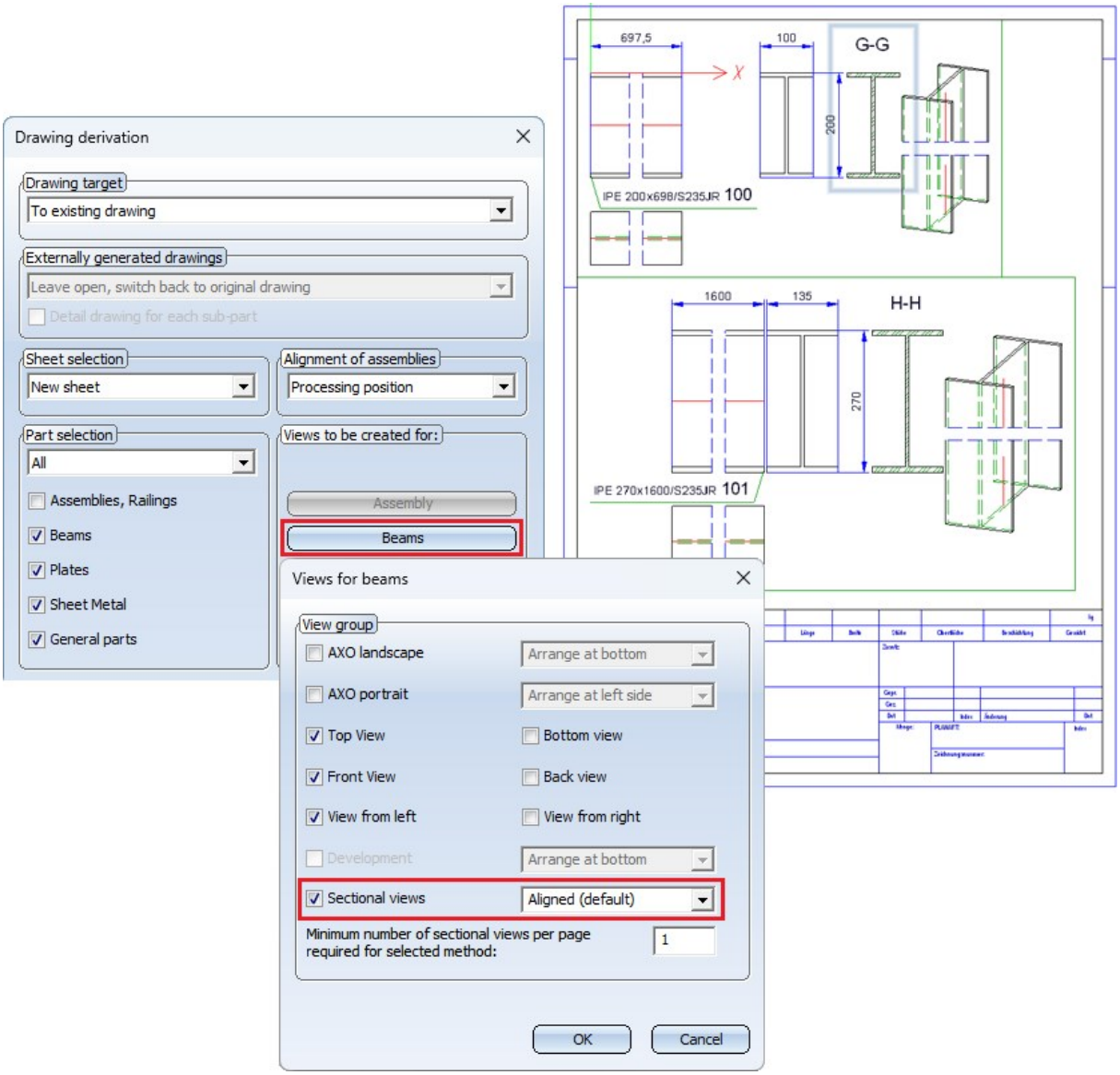


Summarising the functions for 3DFindit

The function was divided into 3DFindit (Basis) and 3DFindit. These have been merged into one function, 3DFindit, and the title bar now shows whether it is the free (basic version) or paid version (full version).

Drawing derivation - sectional view of the beam cross-section

As is already the case for assemblies and sheet metal, a sectional view can now also be optionally generated for beams from SP1 onwards. This displays the beam cross-section. It is displayed without dimensions.



Old workshop drawings (before HiCAD 2024 SP1) remain without a view of the profile cross-section when updating.

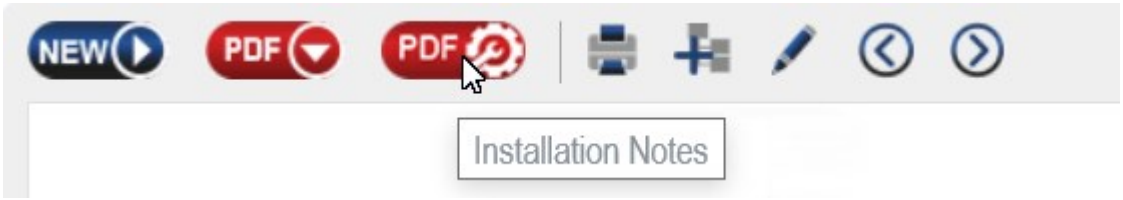
SpaceMouse®

In the Configuration Editor under **System settings** > **Miscellaneous** there is the new setting **End SpaceMouse mode by moving the mouse**. If this setting is activated, SpaceMouse mode must be ended by a mouse action.

Major Release 2024 (V 2900)

Installation Notes

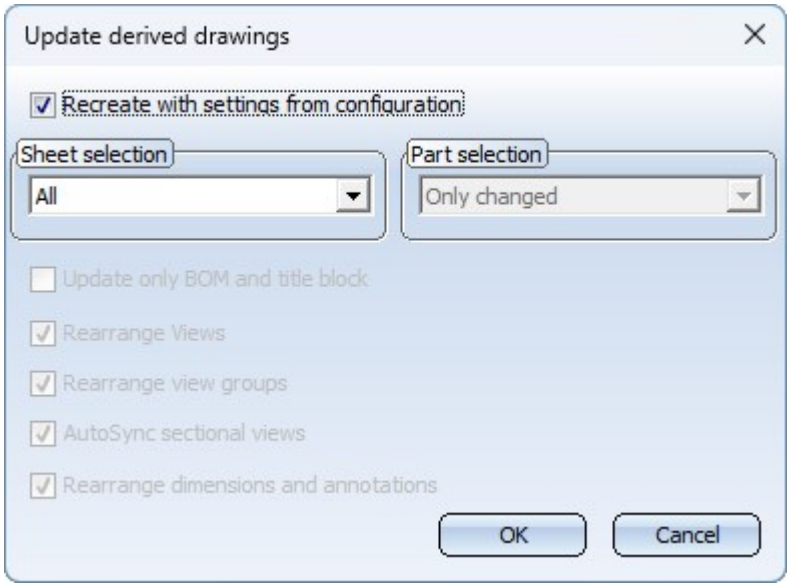
The PDF file with the **Installation Notes** can now also be opened directly via the toolbar of the online help.



Update derived drawing

The **Update derived drawing**  function was previously used to update/add to existing drawings after changes to the model or to close gaps in drawings by updating them, for example when deleting unnecessary views.

As of HiCAD 2024, it is also possible with this function to update existing drawings using the current parameter settings in the Configuration Editor. For this purpose, the **Recreate with settings from configuration** has been added to the dialogue window of the function.



If the checkbox is active, then the drawings are recreated with the current parameter settings for the different usage types. This simplifies, for example, the configuration of derived drawings.

When the checkbox is active, all setting options of the dialogue window are inactive except for the sheet selection.

1.

Drawing derivation in external drawings

- For drawing derivation in external drawings it is now possible to set in the Configuration Editor whether these drawings can be processed or not. For this purpose, the checkbox **Allow processings in external drawings** is available under **Automatic drawing derivation > Production drawing**.

  With the ISD default setting, this checkbox is inactive.

Access to 3Dfindit

3Dfindit by CADENAS is the visual search engine for 3-D CAD, CAE and BIM models. It enables access to CAD catalogues of well-known component manufacturers for design and engineering.

For direct access to the search engine, two new functions are available in HiCAD at **Drawing > Insert Part > Exp.**:

- **3Dfindit (Basis)**
  The free version offers you access to the available 3D catalogues that have a download contract with CADENAS (as of 06/2023, there are approx. 900 manufacturers).
- **3Dfindit**
  The paid version gives you access to the DIN/ISO/EN standards and to the available 3-D catalogues that have a download contract with CADENAS (as of 06/2023 there are approx. 900 manufacturers).

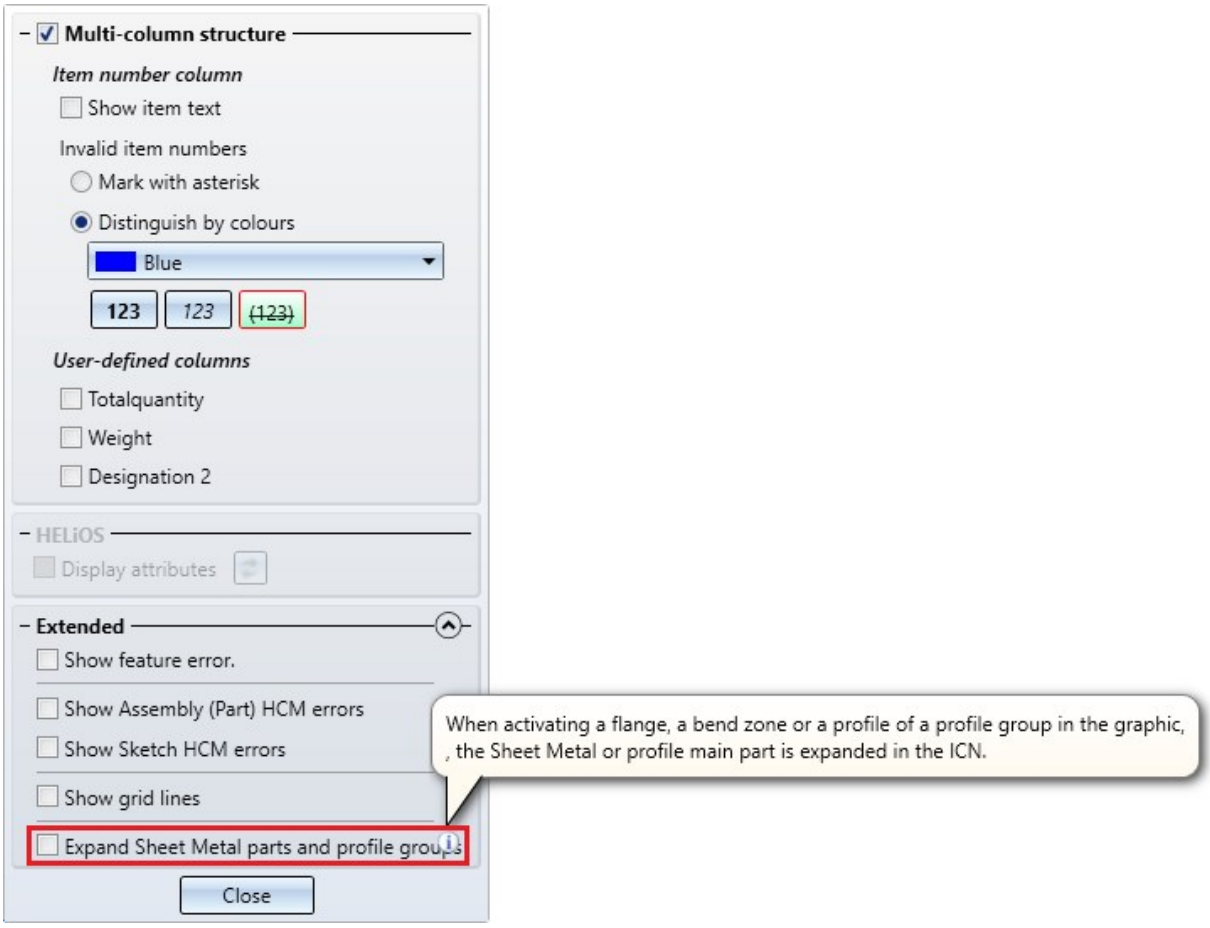
The previous functions

- parts4cad (Basis),
- parts4cad and
- bimcatalogs

are covered by the new functions and are therefore no longer available from HiCAD 2024.

Sheet metal parts and multi-part profiles in the ICN

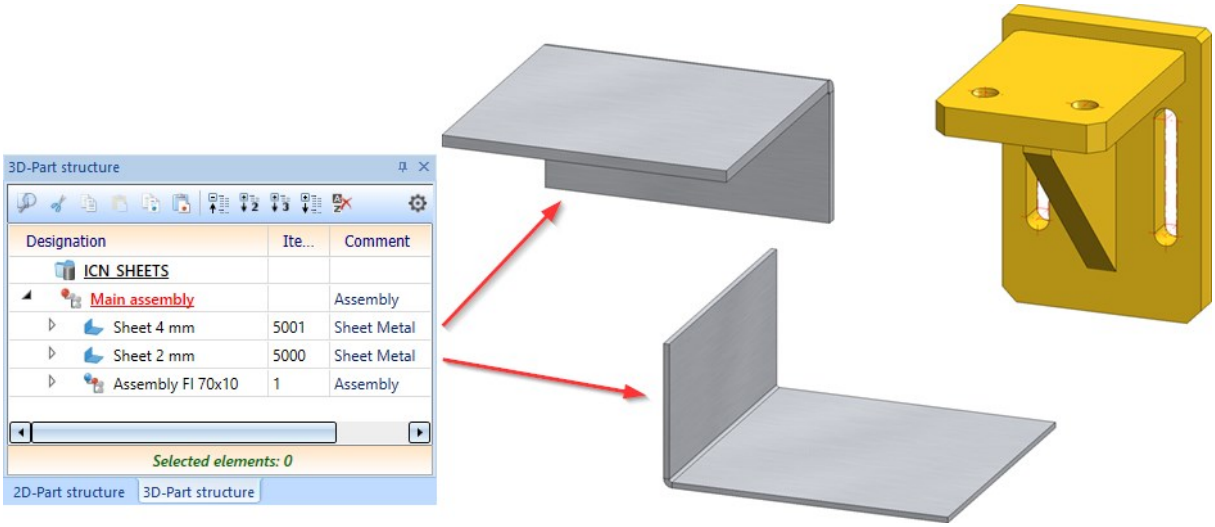
The behaviour when selecting a bend zone/flange of a sheet metal plate or a profile belonging to a profile group in the drawing has changed. Previously, when selecting a corresponding part in the design, the structure of the sheet metal part or profile was always automatically expanded in the ICN. As of HiCAD 2024, this behavior can be defined in the ICN settings.



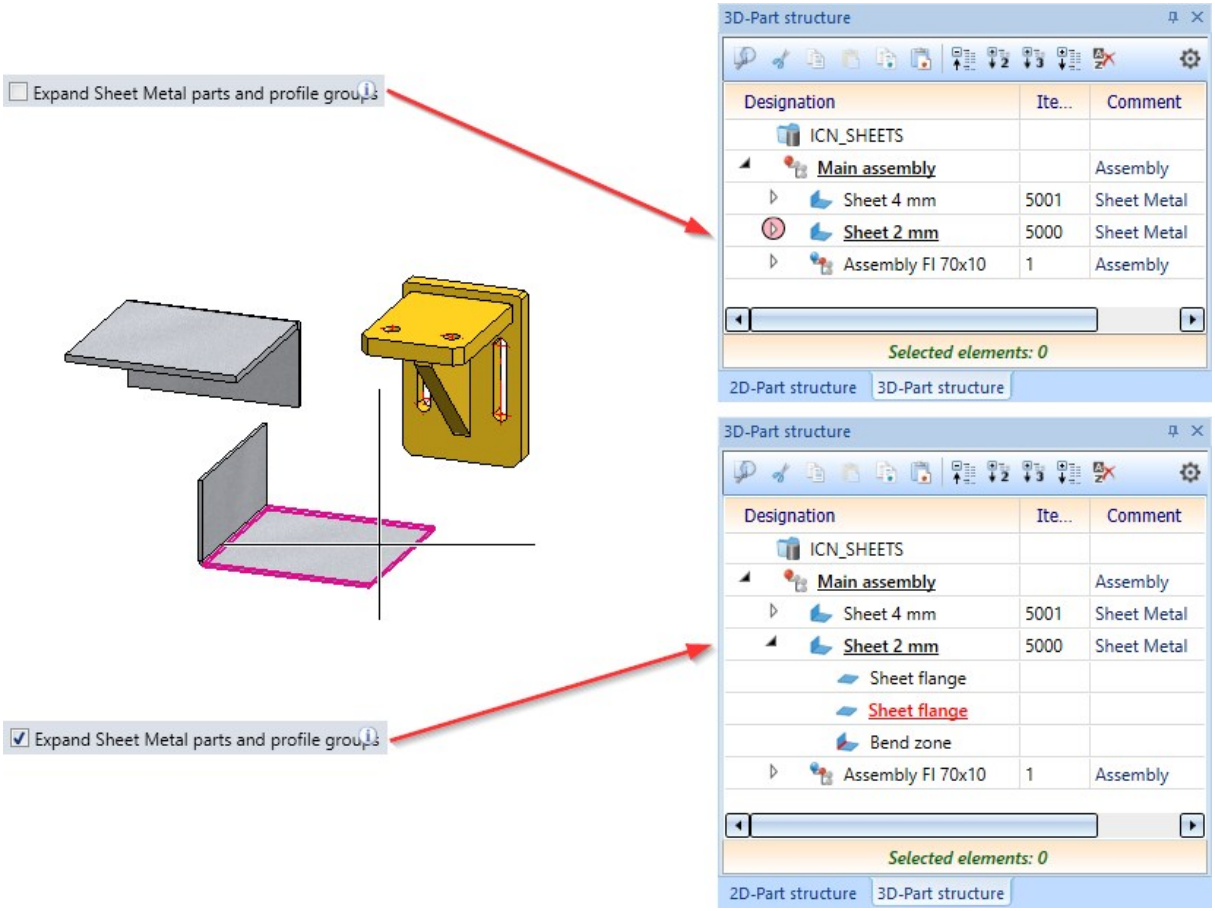
With the ISD default setting, the checkbox is inactive, i.e. the structure of the sheet metal part or profile is not expanded automatically, but only up to the corresponding main part. This is indicated in the ICN by the  symbol.

Example:

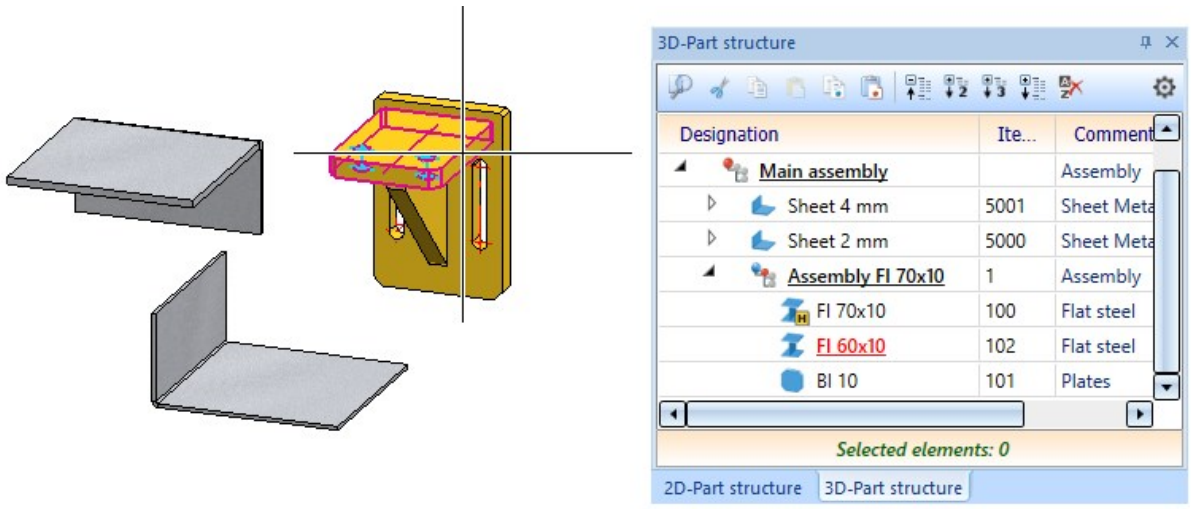
The following drawing consists of a main assembly with 2 sheets and another assembly.



If the checkbox is inactive in the ICN settings, the structure is not expanded when the tab of one sheet is selected.



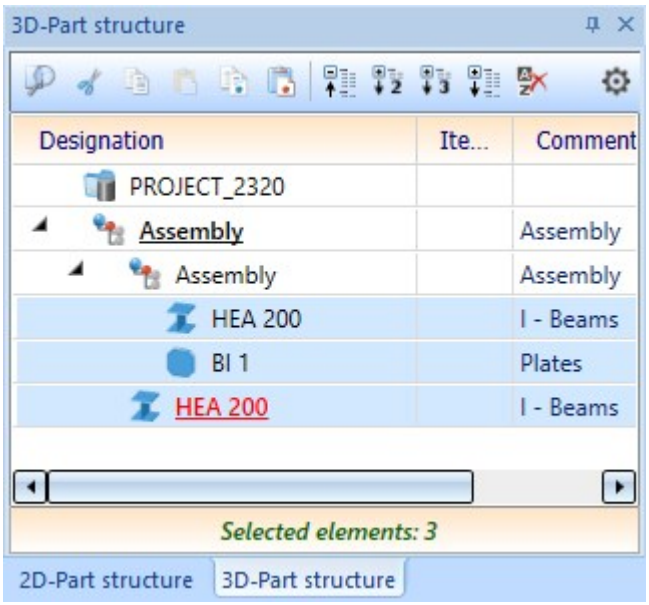
If you select a part of the assembly Fl 70x10, then the structure - as before - is always expanded - independent of the ICN settings.



This setting applies only to sheet metal parts and to profiles of a profile group.

Display of the number of parts of a multiple selection in the ICN

If several parts are selected for processing (multiple selection), then the number of selected parts will now be displayed below the window with the part structure.

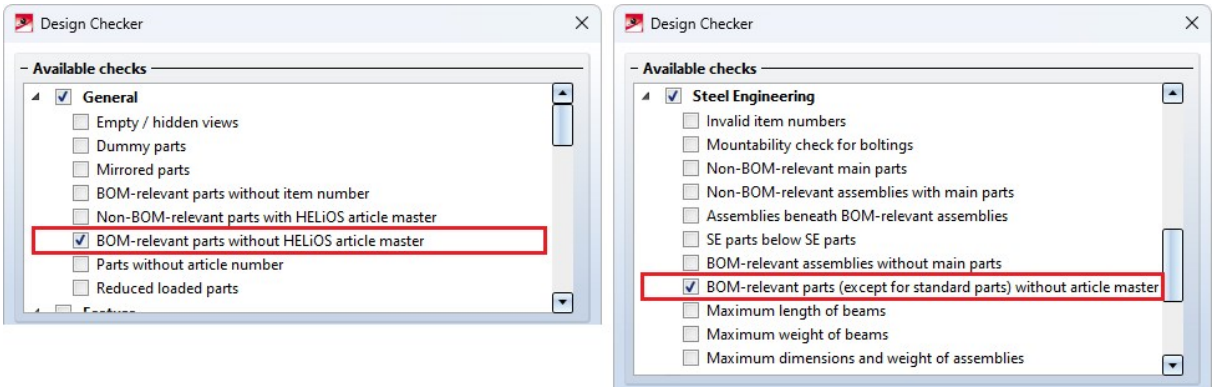


Sorting parts in the ICN

By clicking on a column heading, the sorting of the tabular display of the part structure can be changed in the ICN, e.g. descending by item number or ascending by comment. Up to now, when HiCAD was closed, the selected sorting was reset to default settings. As of HiCAD 2024, the sorting that was last active when HiCAD was exited will be automatically set when HiCAD is restarted.

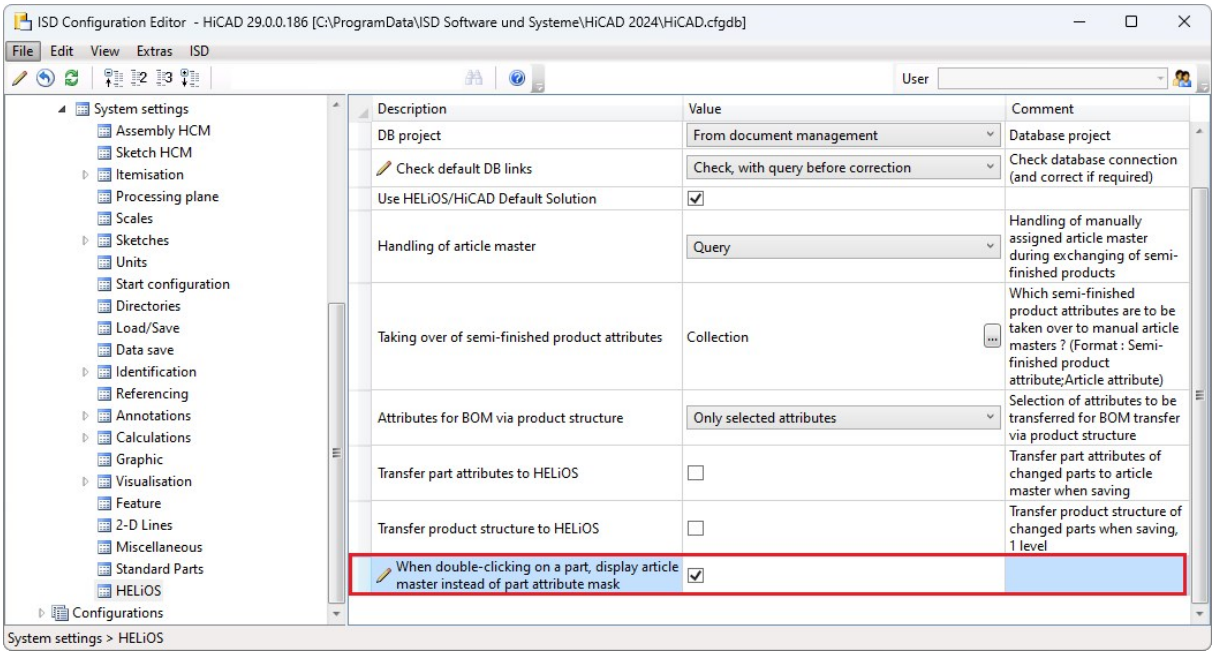
Design Checker - BOM-relevant parts without article master

In the Design Checker, a new test is available under **General** that searches for all parts/assemblies that are BOM-relevant but do not have a HELiOS article master. In contrast to the previous test under **Steel Engineering**, standard parts are also taken into account here.



Article master display via double click

Until now, double-clicking the left mouse button on a part in the drawing or in the ICN called up the **Part attributes** dialogue window. As of HiCAD 2024, the **Article master** can now be displayed as an alternative when using HELiOS. This can be set in Configuration Management under **System settings > HELiOS**.

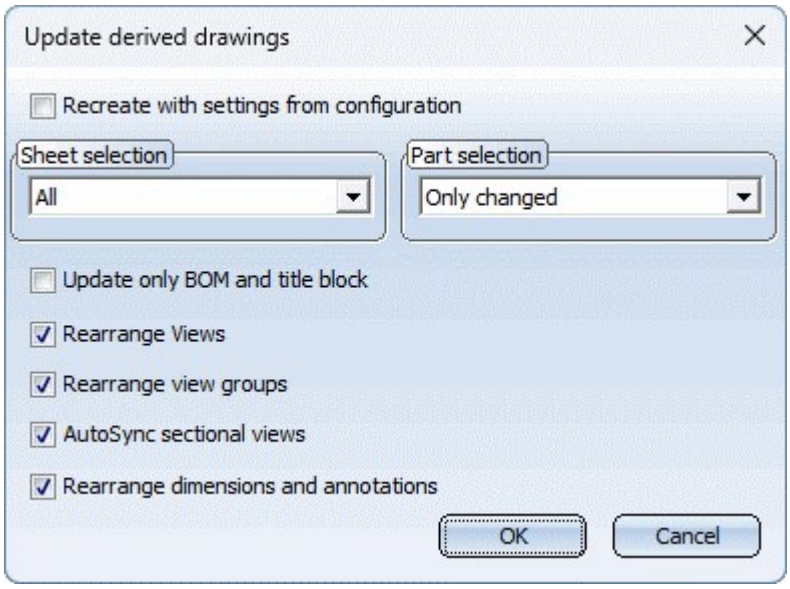


If the checkbox is active and the clicked part does not have an article master, the **Part attributes** dialogue window is automatically displayed.

View groups when updating workshop drawings

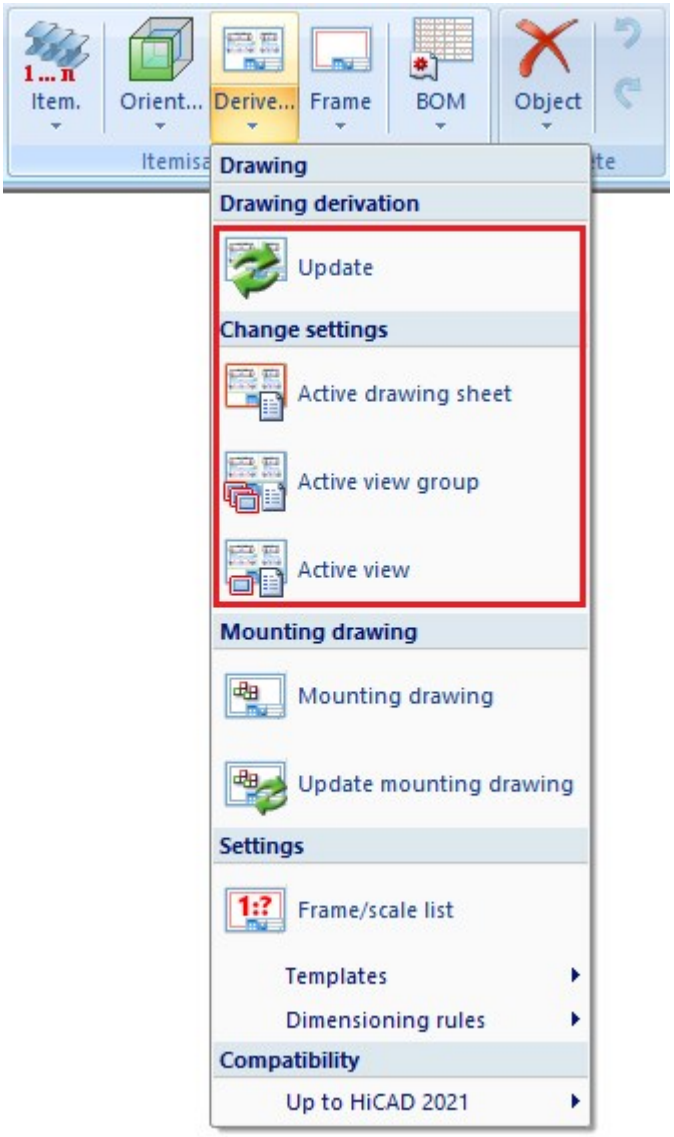
When updating workshop drawings, it is now possible to choose whether only the view groups as a whole and not the contained views should be rearranged. Therefore, when updating, a distinction is now made between the check-boxes

- Rearrange Views and
- Rearrange view groups.



Rearrange Views	This checkbox only controls the arrangement of the views in the view groups. If you do not want to rearrange the views of the updated sheets, for example to keep your own view rearrangements, then deselect the Rearrange Views checkbox.
Rearrange view groups	If you do not want the view groups of the updated sheets to be rearranged, uncheck the Rearrange view groups checkbox. Note that if the checkbox is active, only the view groups as a whole will be rearranged. The views contained in the groups will only be rearranged if the Rearrange Views checkbox is also active.

This change affects the **Update derived drawing** function and the functions under **Drawing > Change settings**.



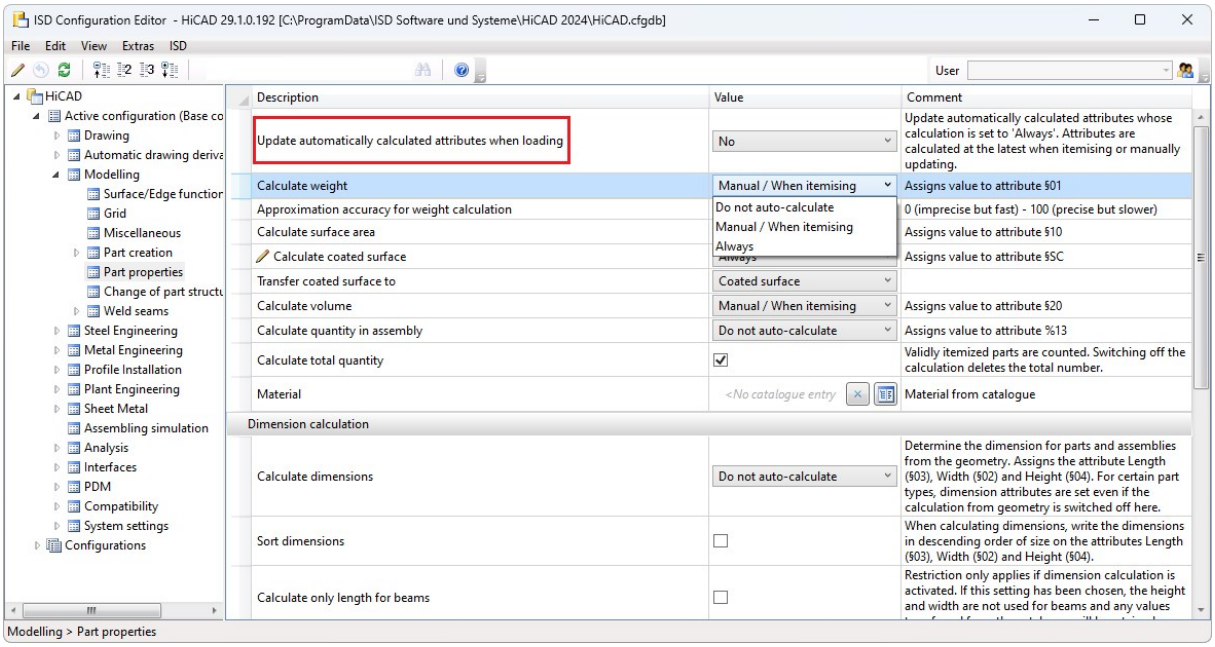
Support tool

The support tool is now also available in Italian and Polish.

Update automatically calculated attributes when loading

In the Configuration Editor, you can specify under **Modelling > Part properties** when certain attributes and dimensions are to be calculated. This applies, for example, to the weight, volume, surface area and much more. For example, this can be done **always**, i.e. after each change of a part, **manually** or **when itemising**.

For the calculations that are set to always, as of HiCAD 2024 it is now also possible to specify whether these calculations should be performed automatically when loading a drawing or not. The setting is also made under **Modelling > Part properties** with the parameter **Update automatically calculated attributes when loading**.



If the parameter is set to **Yes**, then the calculations are always executed directly for all parts when loading a drawing. In large drawings, this can lead to considerable waiting times.

If the parameter is set to **No**, then the calculations are done only when processing assemblies/parts, when itemising or when updating manually with the new function

Update part attributes  .

In this way, you determine yourself at which point in time the calculations should take place and avoid waiting times.

The ISD default setting is **No**.

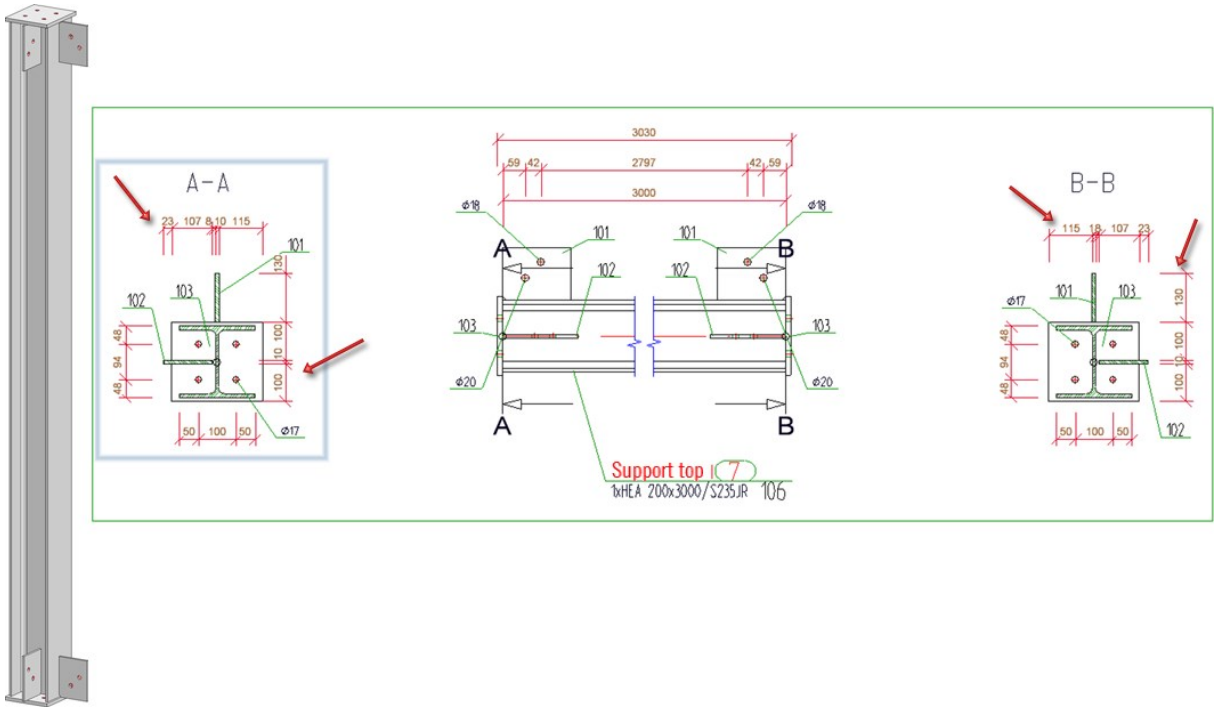
Dimensioning rules

Position of sub-parts in sectional view

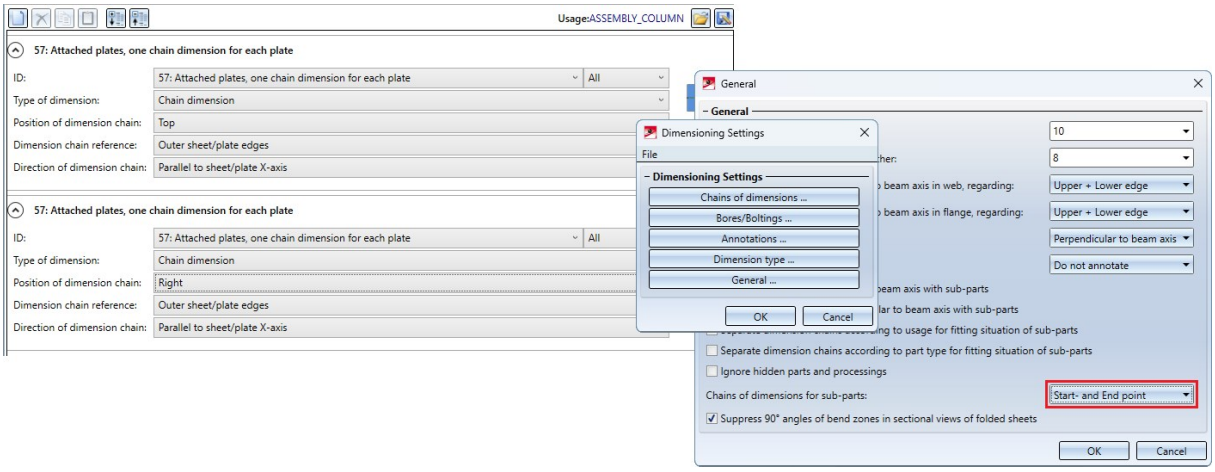
Up to now, only the position of the beam in relation to the sub-part was dimensioned during drawing derivation with automatic dimensioning (dimensioning rules). As of HiCAD 2024, the position of all sub-parts is dimensioned. The dimensioning rules

- 8: ATTACHING_PARTS,
- 10: ATTACHING_SHEETS and
- 57: ATTACHING_SHEETS_SEPARATELY

have been updated accordingly.



In the example, the Dimensioning rules shown have been used for the sectional view and the Dimensioning Settings on the **General** tab.



Usage for railing segments

In the automatic drawing derivation, the usage RAILINGSEGMENT has been used up to now for the dimensioning of railing segments created by the railing configurator. However, there is often the wish to dimension the railing segments differently for different railing types, for example, for segments with glass infill or segments with knee rail infill, etc.

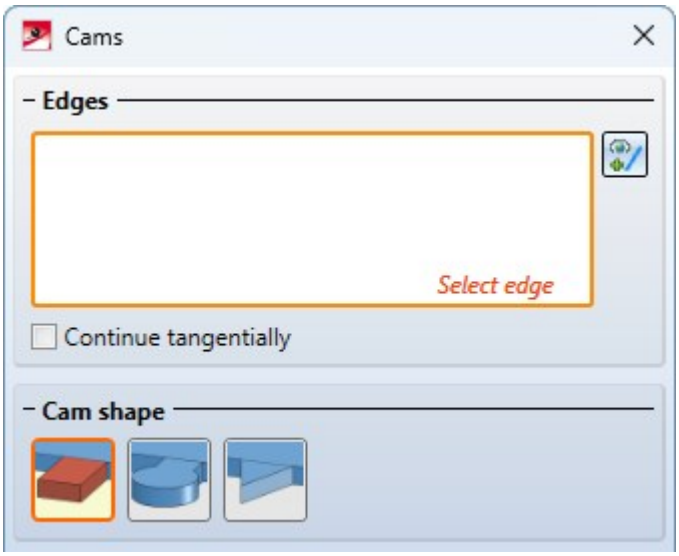
From HiCAD 2024 this is now possible. For this purpose, corresponding usages must be defined and associated configurations created whose name contains the expression RAILINGSEGMENT. For more information, see HiCAD Steel Engineering - What's New?

3-D

Service Pack 1 2024 (V 2901)

Cams and cam processings

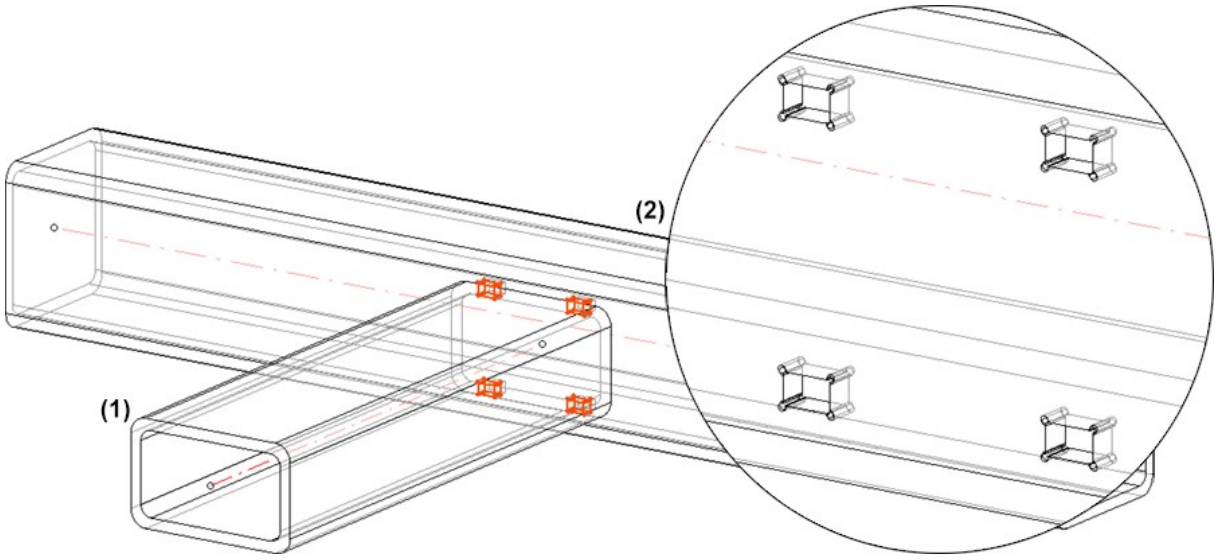
The dialogue window of the **Cams**  function has been expanded to include images of the cam shape for better comprehensibility.



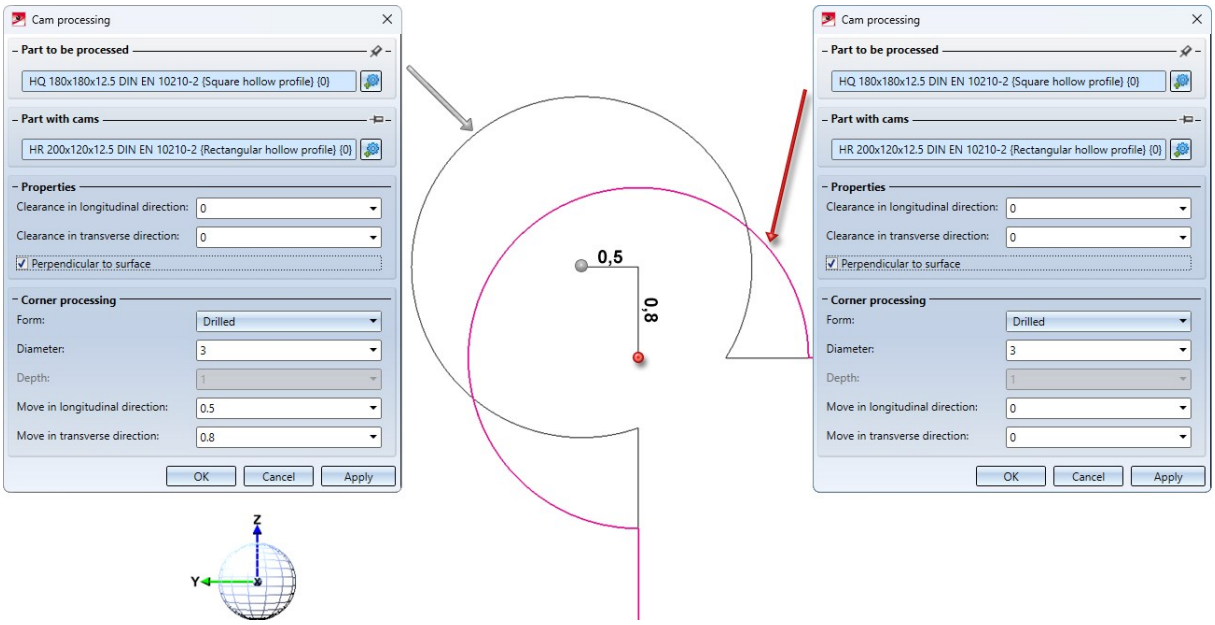
The **Cam processing**  function now allows you to specify a move in the longitudinal and transverse direction. This is possible if the cam shape is straight and **Drilled** is selected as the corner processing.

The bores on the part to be processed can then be moved by a value in the longitudinal and transverse direction. The longitudinal direction is the cam direction, the transverse direction is the left/right side of the cam.

The illustration shows a cam connection. (1) is the part with cams and (2) is the part with the cam processings.

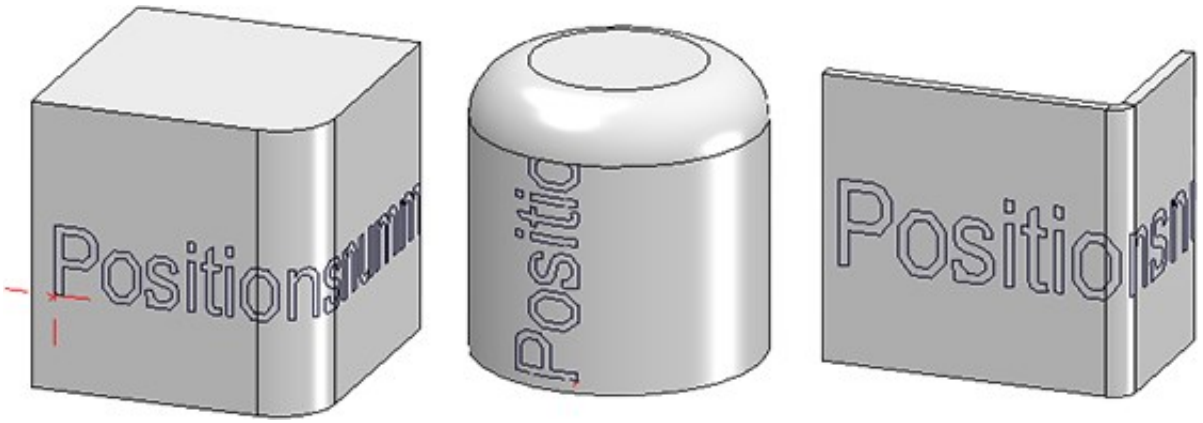


The following figure shows a section of the side view of part (2) - with and without indication of a move.



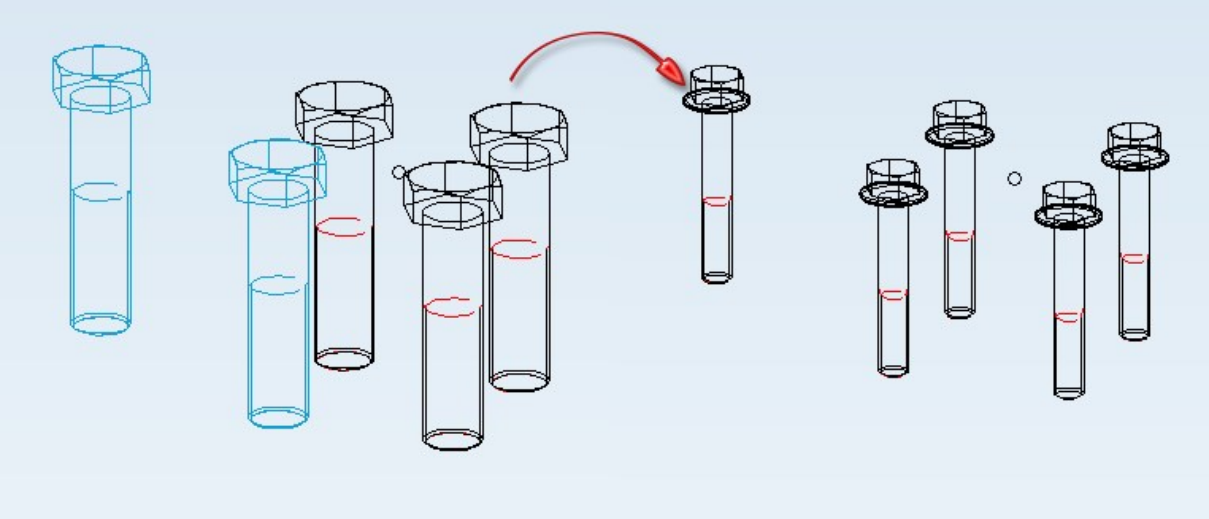
Lettering

You can use the **Lettering**  function to letter almost any surface. You can now use the lettering function to sign almost any surface. Subtracted from this are cones, spheres and toruses. If the surface has a tangential transition, the lettering continues over the fillet to the next edge. Cylinders are excluded from this. On edge sheets, the lettering continues over the flanges. It is now also possible to create a lettering in the bending simulation.



Replace standard parts

With the new **Replace standard parts** , you can replace several selected standard parts of one type (e.g. nuts) by selecting a different variant and/or size in the catalogue. If you have created standard parts in a grid and a standard part from the standard part group is in the multiple selection, the entire standard part group is changed.



The function can only be applied to standard parts and standard part groups that correspond to the same type (e.g. only screws or only nuts). However, the selection may contain different variants (e.g. hexagon nut and cap nut). If the multiple selection contains standard parts of a standard parts group, the entire standard parts group is changed.

The following options are rejected with an error message:

- Multiple selection with standard parts/standard parts groups that do not correspond to the same type,
- Multiple selection with parts that are neither standard part nor standard parts group,
- Standard parts/standard parts groups from design variants and configurators (e.g. in steel engineering),
- Standard parts/standard parts groups that belong to a bolting,
- standard parts/standard parts groups with feature log and
- blocked standard parts/standard parts groups.

Please note:

- Processing operations such as bores are not adapted. The **Replace standard parts** function only allows you to change the standard. Other properties such as the representation of the thread cannot be changed.
- For standard parts with a free value (e.g. the effective length for rivets), this value can be entered. It is then used for all replaced standard parts.

Referencing

Updating identical parts

When externally or internally referenced parts are changed, all identical parts in the current drawing are automatically updated from SP1. The parameters

- Automatically update referenced parts after each change and
- Update identical parts of referenced parts before saving

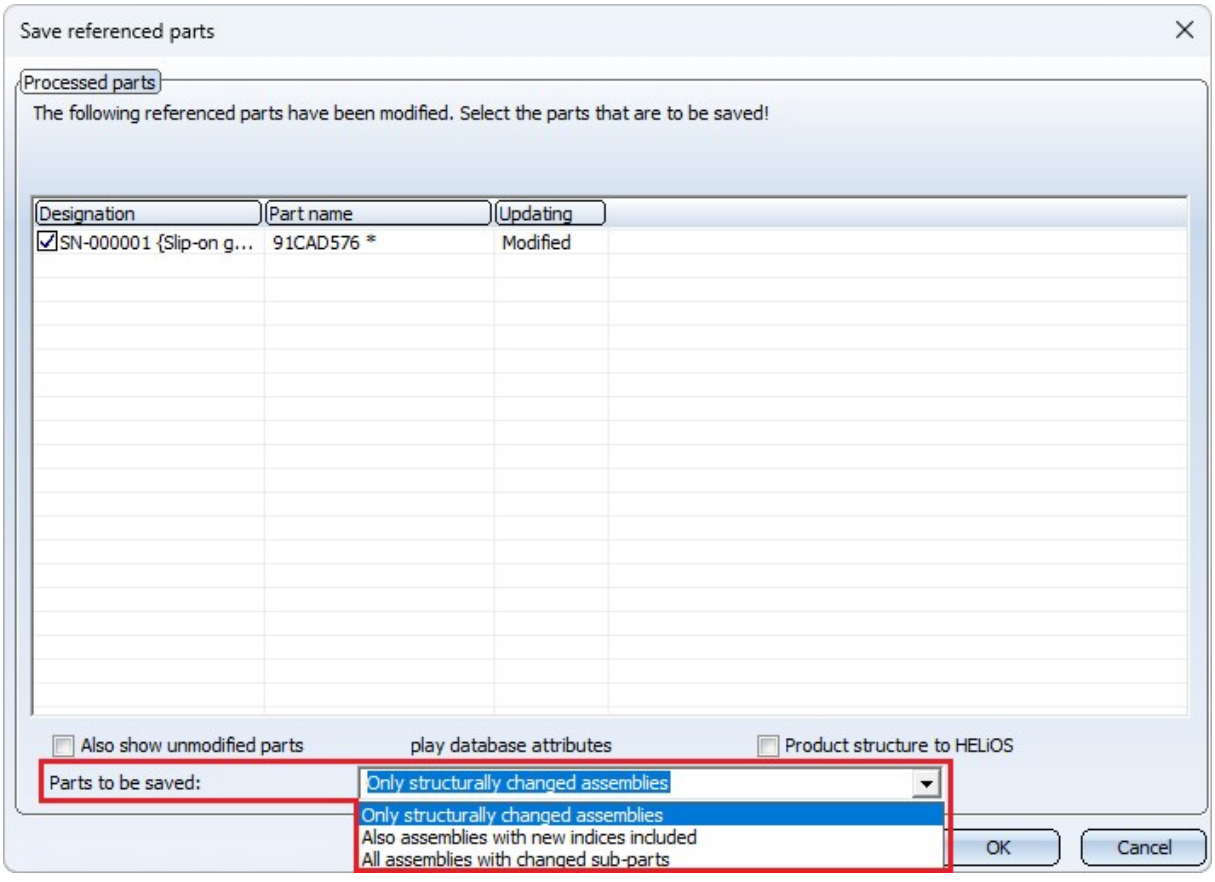
have therefore been removed from the Configuration Editor under System settings > Referencing.

Referenced assemblies with referenced parts

Referenced parts with changed geometry are always offered for saving when the drawing is saved. This may mean that the KRA file of the part is updated, but that of the assembly is not. When reloading the drawing or inserting the KRA file of the higher-level assembly, HiCAD then detects that the part it contains has a newer KRA version, which is then reloaded.

In practice, there is sometimes a desire for the KRA of the higher-level assembly to always contain the current status of its subordinate parts, e.g. so that the HELiOS document structure always matches the KRA file. As of SP1, it is therefore possible to define how externally referenced assemblies are to be handled when saving drawings and when changing drawings if they contain externally referenced parts.

Various options are now available in the Save referenced parts dialogue window under Parts to be saved.



Only structurally changed assemblies

Referenced parts with changed geometry are offered for saving. However, the referenced assemblies to which the parts belong are only taken into account if they have been structurally changed. This is the previous procedure before HiCAD 2024 SP1.

Also assemblies with new indices included

If this option is selected, the assemblies whose directly referenced parts have a new document master index are also offered for saving in addition to the structurally changed assemblies. This option only has an effect if the parts and assemblies are managed in HELiOS.

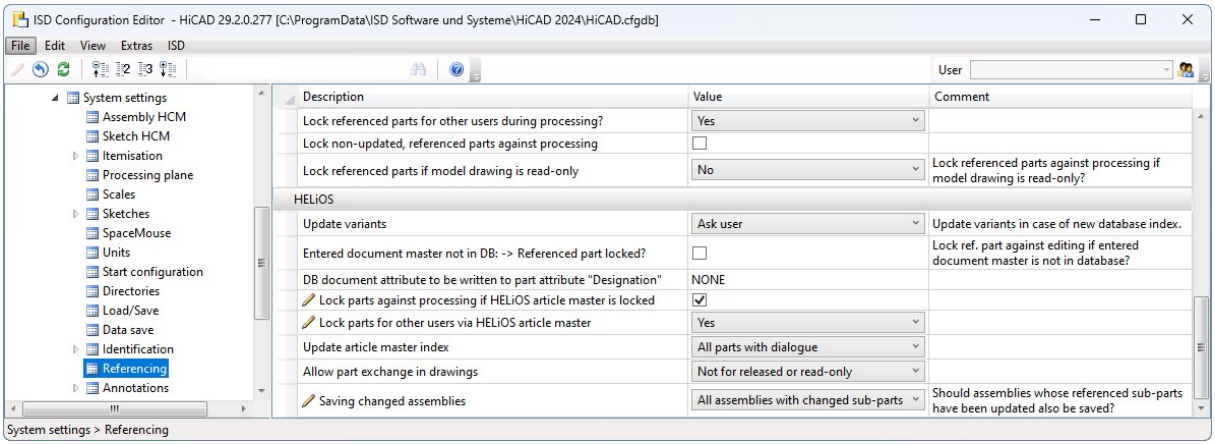
All assemblies with changed sub-parts

If this option is selected, the assemblies that contain changed referenced parts in their entire structure are also saved, regardless of the level. If this option is preset in the Configuration Editor, the corresponding assemblies are marked with the symbol in the ICN.

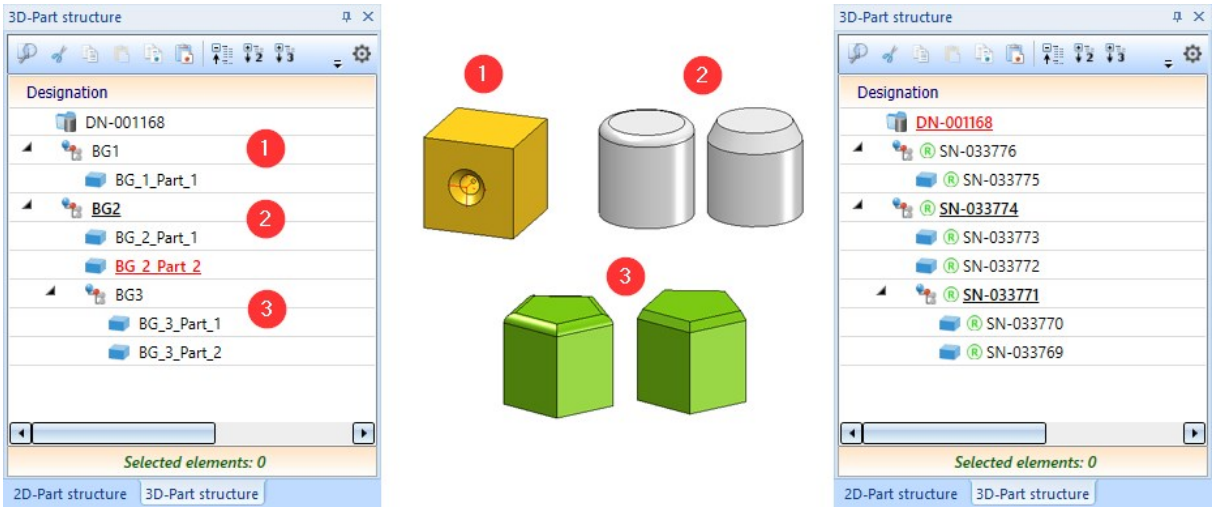
Which setting is active here when saving is determined by the setting in the Configuration Editor under **System settings > Referencing > Saving changed assemblies**. The ISD default setting is **Only structurally changed assemblies**. Even if you select a different setting when saving a drawing during the current HiCAD session, the setting from the Configuration Editor will take effect again the next time it is called up.

Example

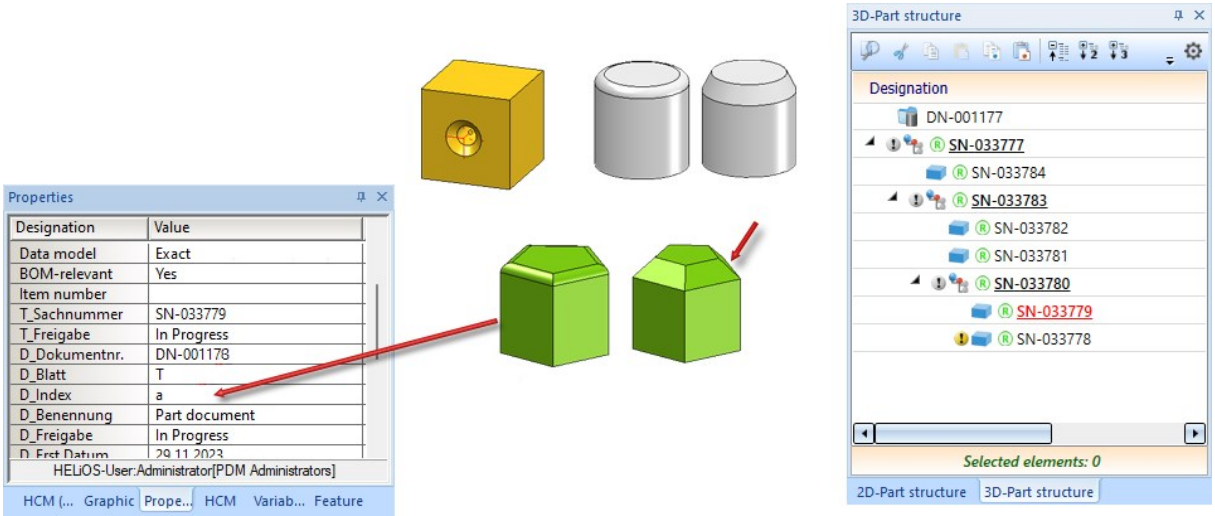
In the Configuration Editor, the parameter Saving changed assemblies has been preset to All assemblies with changed sub-parts under System settings > Referencing.



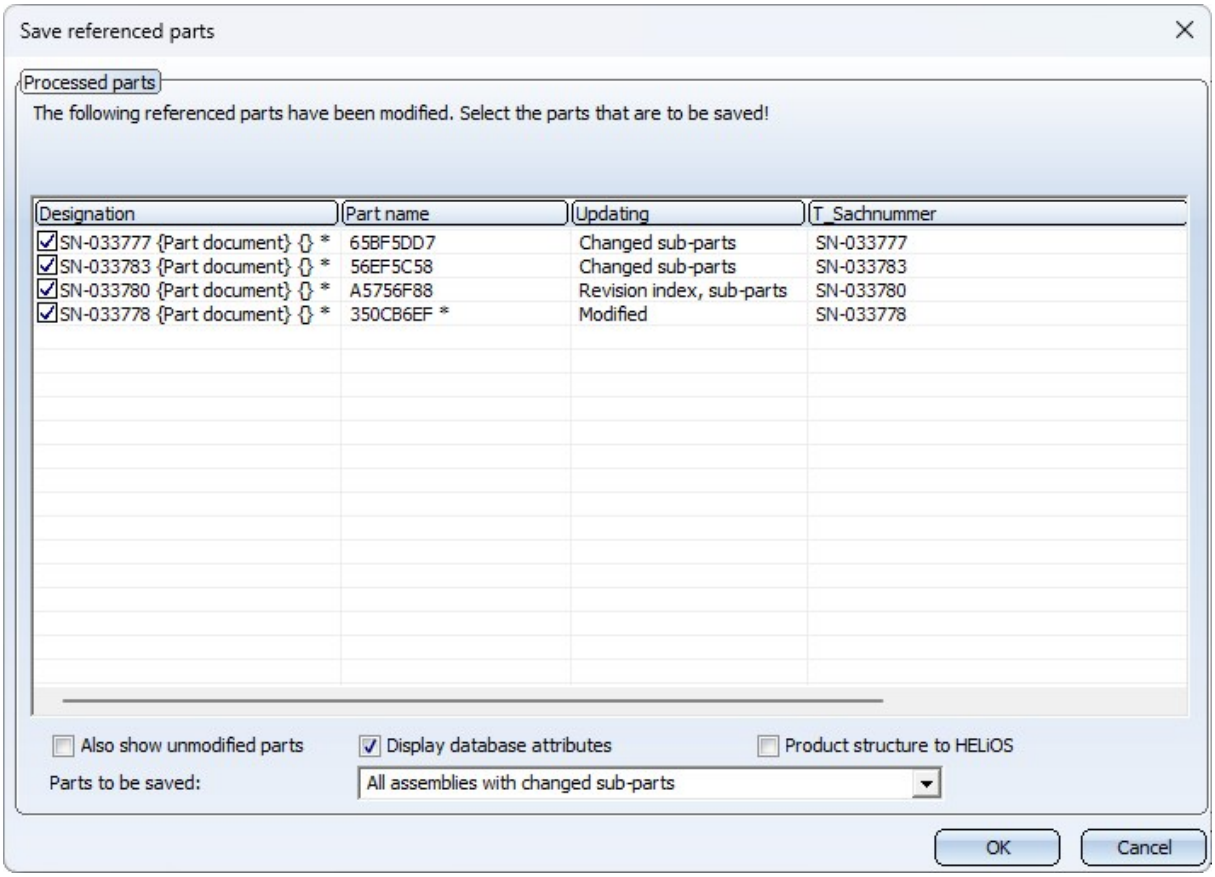
The following figure shows a drawing managed in HELiOS with three assemblies BG1, BG2 and BG3. Assembly BG2 is subordinate to assembly BG1 and contains assembly BG3. All assemblies contain different solid primitives. In the first step, both the parts and the assemblies have been referenced externally with the part and document master.



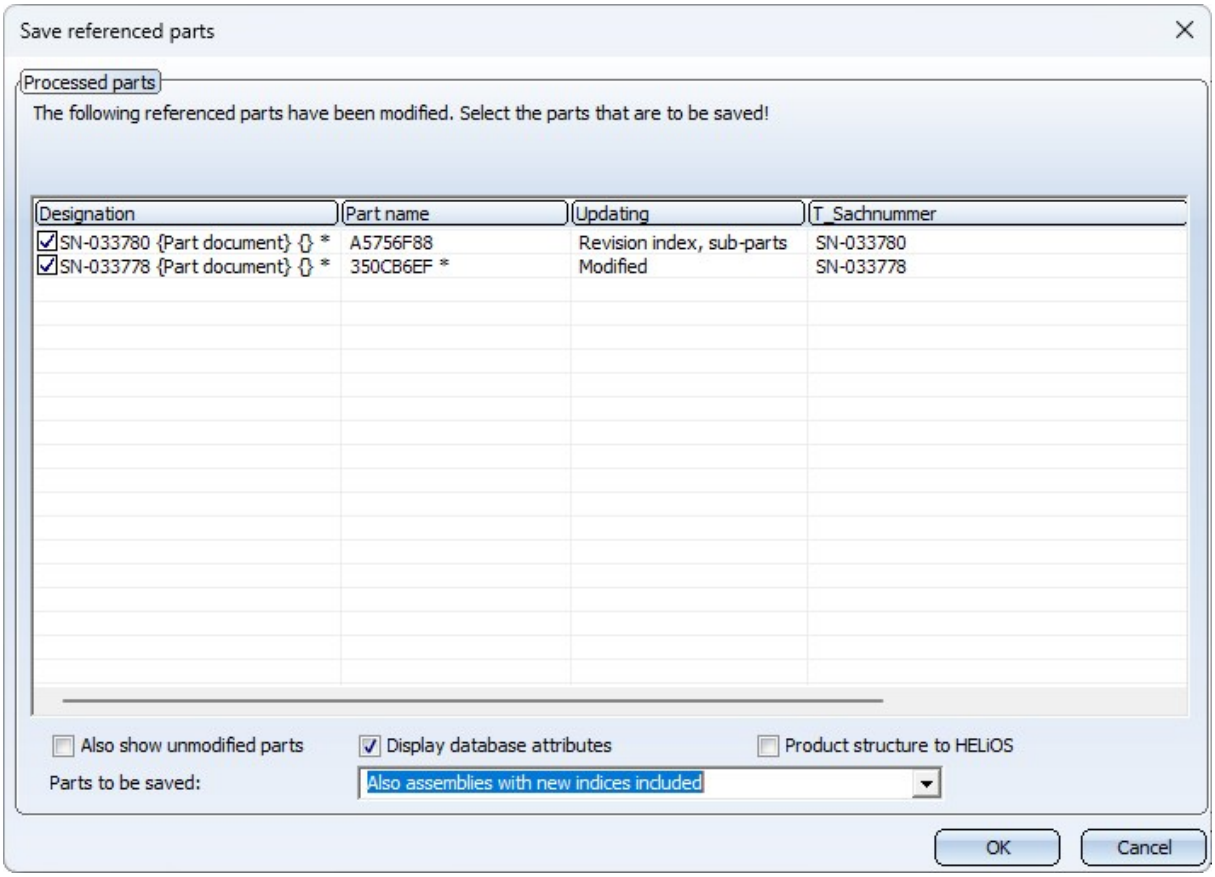
The chamfer length of the right prism has then been changed in assembly BG3 and an index has been created in the document master of the left prism.

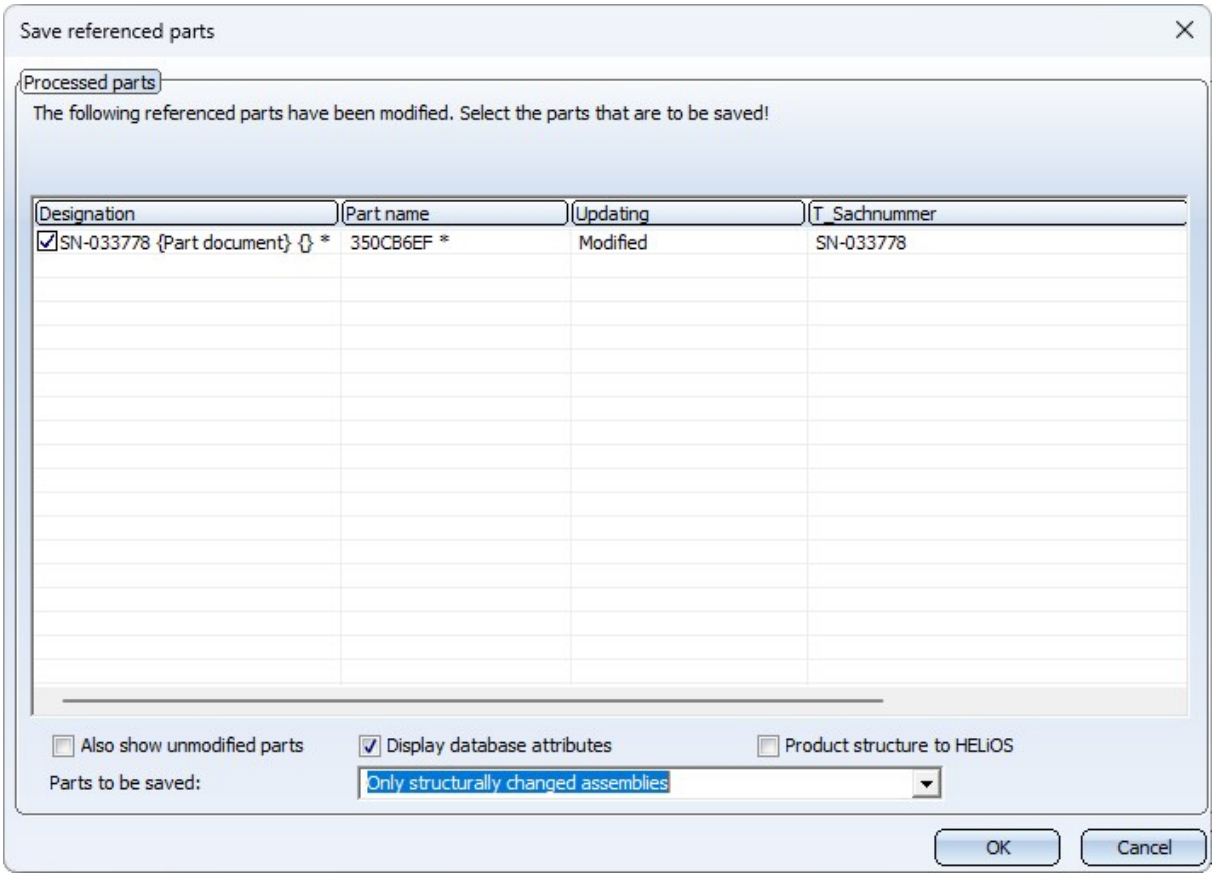


Parts whose geometry has changed - in the example, the right-hand prism - are marked with the  symbol in the ICN. Due to the default setting in configuration management, all assemblies with changed sub-parts are additionally marked with the  symbol. If the drawing is now saved, all marked assemblies and the prism with the changed geometry are listed in the **Save referenced parts** dialogue window.



If you change the selection under **Parts to be saved**,the list of parts changes.




Save referenced parts
Processed parts
The following referenced parts have been modified. Select the parts that are to be saved!
Designation
Part name
Updating
T_Sachnummer
SN-033778 {Part document} {} *
350CB6EF *
Modified
SN-033778
Also show unmodified parts
Display database attributes
Product structure to HELiOS
Parts to be saved:
Only structurally changed assemblies
OK
Cancel

Dimensioning and annotations

Adopt tolerance

The **Adopt tolerance** 

can be used to transfer dimensional tolerances from one dimension to another. To do this, you can click on the reference dimension, i.e. the dimension whose tolerance you want to adopt, to select the tolerance you want to copy. Then click on all the dimensions into which you want to insert the tolerance. To end the function, press the middle mouse button once.

Please note:

Brackets or symbols that were assigned to the reference dimension using other functions such as **Edit dimension figure** or **Set symbol** are not taken into account during the transfer.

3-D annotation with HELiOS data

As of HiCAD 2024 SP1, HELiOS data is saved in annotations with the drawing. This data is then used when working without HELiOS. This also applies if a different sheet is printed via the plot manager (from HiCAD 2024 SP1) than the sheet active when saving the drawing. Previously, the HELiOS data was missing in the annotation tags in this case.

Please note that this change does not affect existing drawings. These must first be saved again.

Form and positional tolerances (3-D) - Preview window

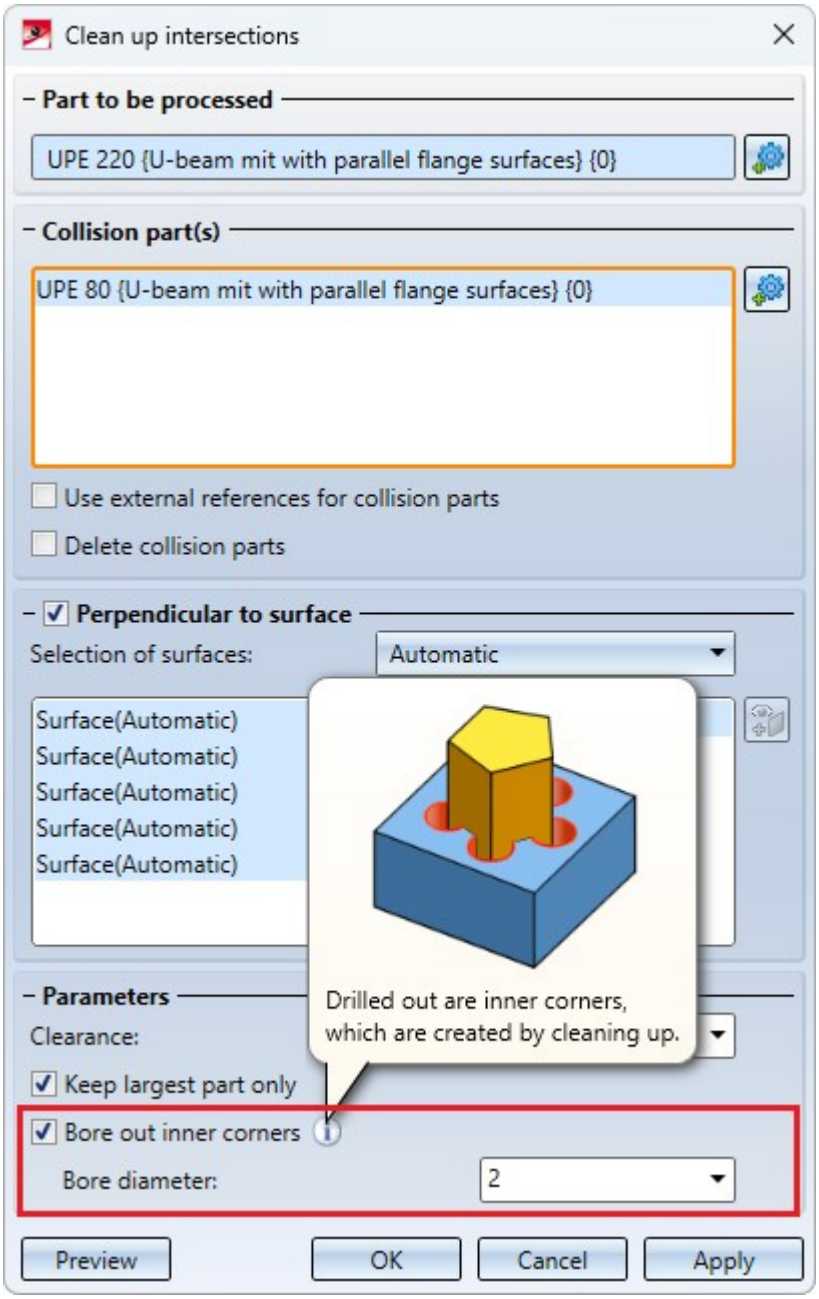
The background of the active sheet/model area is automatically displayed in the preview window of the form and positional tolerances (3-D). This only changes when the background of the sheet/model area is changed.

Edge state - preview window

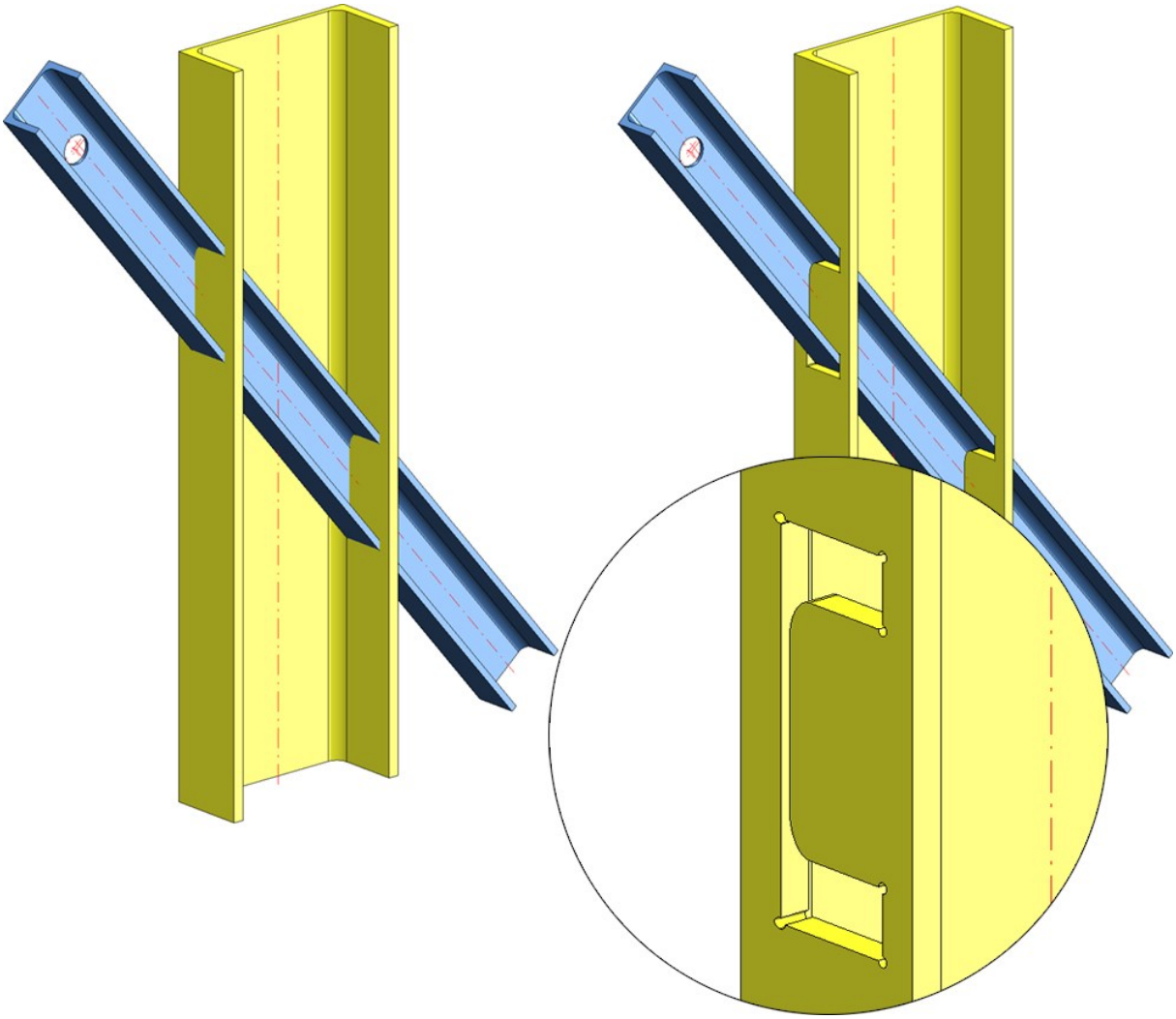
The preview window for the edge state (3-D) automatically displays the background of the active sheet/model area and only changes when the background of the active sheet/model area is changed. The font and line colour is also displayed in the preview window.

Clean up intersections - Bore out inner corners

The **Clean up intersections**  function has been extended.



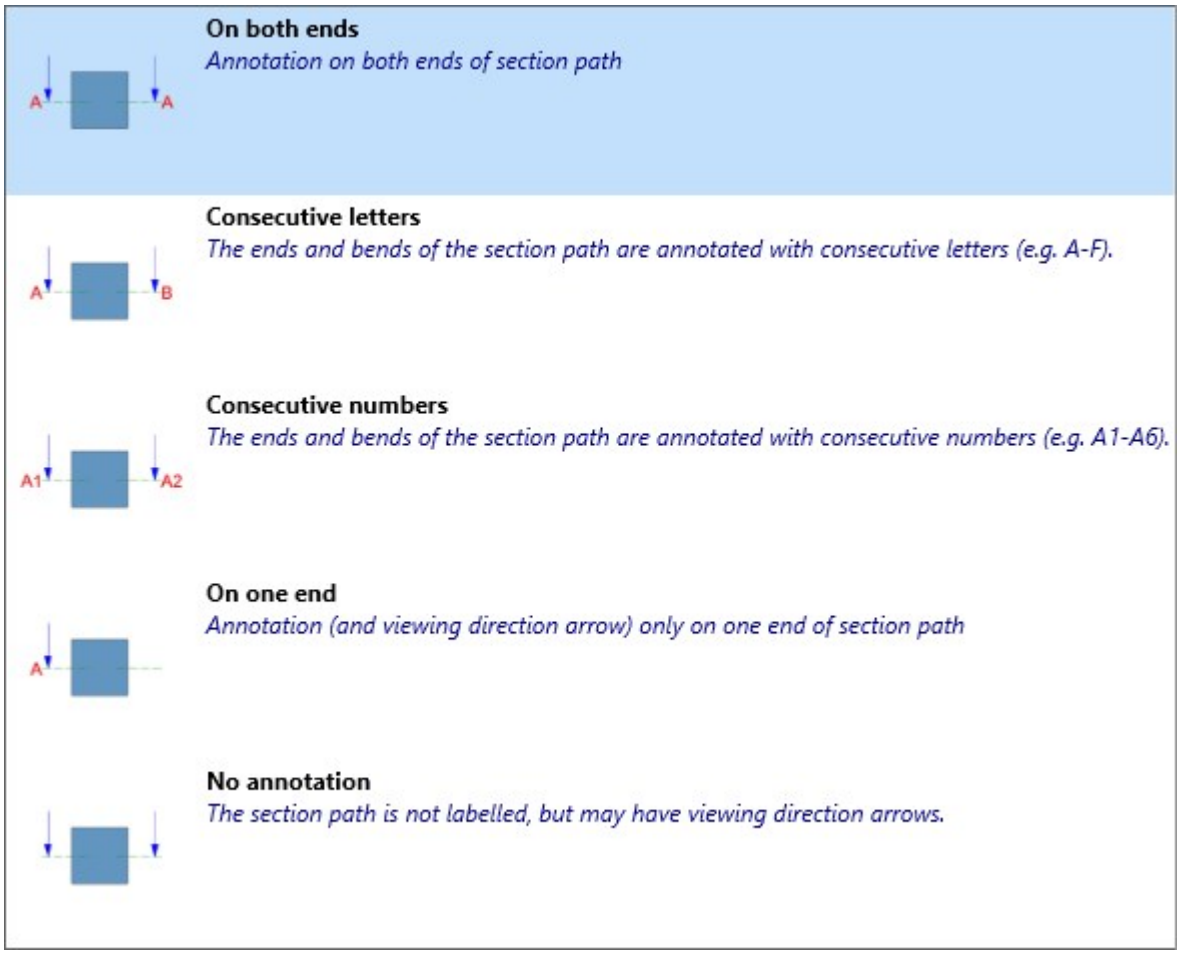
In certain situations, clean-up can result in concave corners. These can now be bored out if required by activating the **Bore out inner corners** checkbox and specifying a bore diameter. For example, this can be useful for parts with cams that were not created with the Cam function. Another use case is "jumbled" parts, such as the profiles in the following illustration.



Views

Create sectional view - Extensions

The dialogue window of the **Sectional view** function has been expanded in the **Ident** area on the **Parameters** tab with the **Annotation** selection box to include images for better comprehensibility.



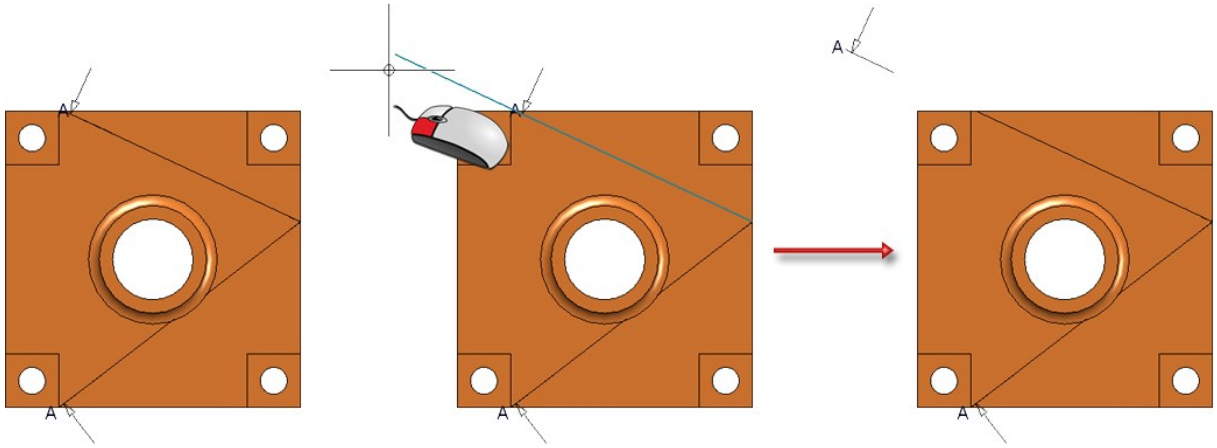
Change section path

New in the menu under **Views > Edit > Section** is the function **Change section path** .

The function can be used to edit the sketch for the section path directly. After calling up the function, the following window is displayed:

Then edit the sketch using the functions of the **Sketch** ribbon and finally click **Apply sketch**. The sectional view is then adjusted directly.

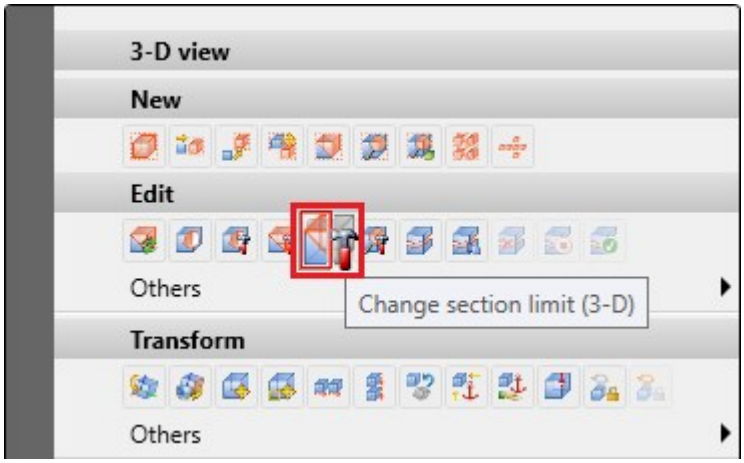
The section path can also be extended later by simply dragging it without having to call up the **Change section path** function.



Change section limit

The new **Change section limit**  allows you to directly change the sketch for the sectional view limit. After calling up the function, you can adjust the sketch as desired. Click on **Apply sketch** to apply the change.

You can find the function in the **Views** ribbon under **Edit > Section** and in the context menu for views.



Fixed view point when changing scale

Previously, if the view scale was changed, a fixed point had to be set again. From SP1, the fixed point is retained.

Align views horizontally/vertically, via points

The following functions are new in the context menu of views:

Align views horizontally, via points

Align views vertically, via points

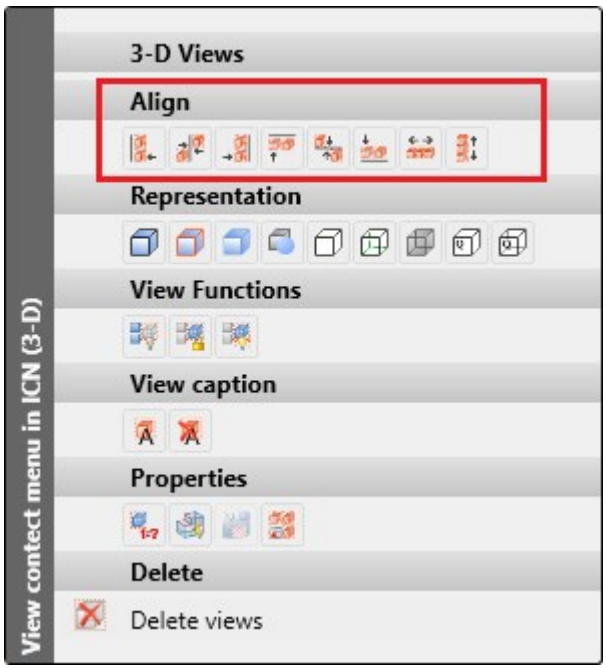
These functions can be used to align individual views horizontally or vertically to a specific point.

1. After calling up the function, first select a point for the alignment. Depending on the function selected, a horizontal or vertical alignment guideline is then shown through this point.
2. Then select a point in a view. This view is then moved orthogonally to the alignment guideline so that the point selected in the view lies on the alignment guideline.
   Please note that the selected point must belong to a view, otherwise a corresponding error message will be displayed.

The function does not end automatically after it has been executed, i.e. you can align further views to the displayed line by selecting further viewpoints. You end the function with the middle mouse button or with ESC.

Align and distribute views flush/centred

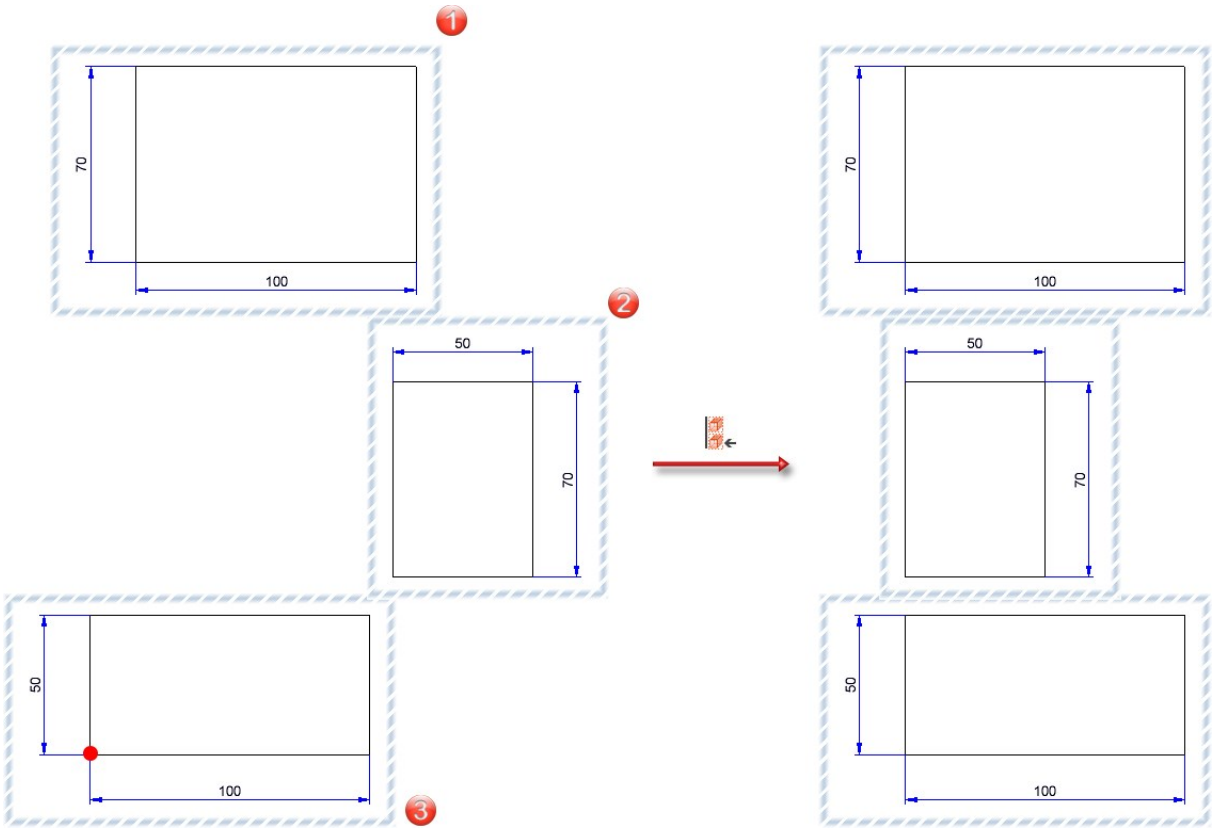
As of SP1, HiCAD 2024 also offers the option of aligning several views flush (right, left, top, bottom) or centred (horizontal, vertical). In addition, views can be evenly distributed horizontally or vertically. The corresponding functions are available in the context menu for views if a view list (multiple selection) is active.



Align		Distribute	
	Align views left		Distribute views horizontally
	Centre views horizontally		Distribute views vertically
	Align views right		
	Align views at top		
	Centre views vertically		
	Align views at bottom		

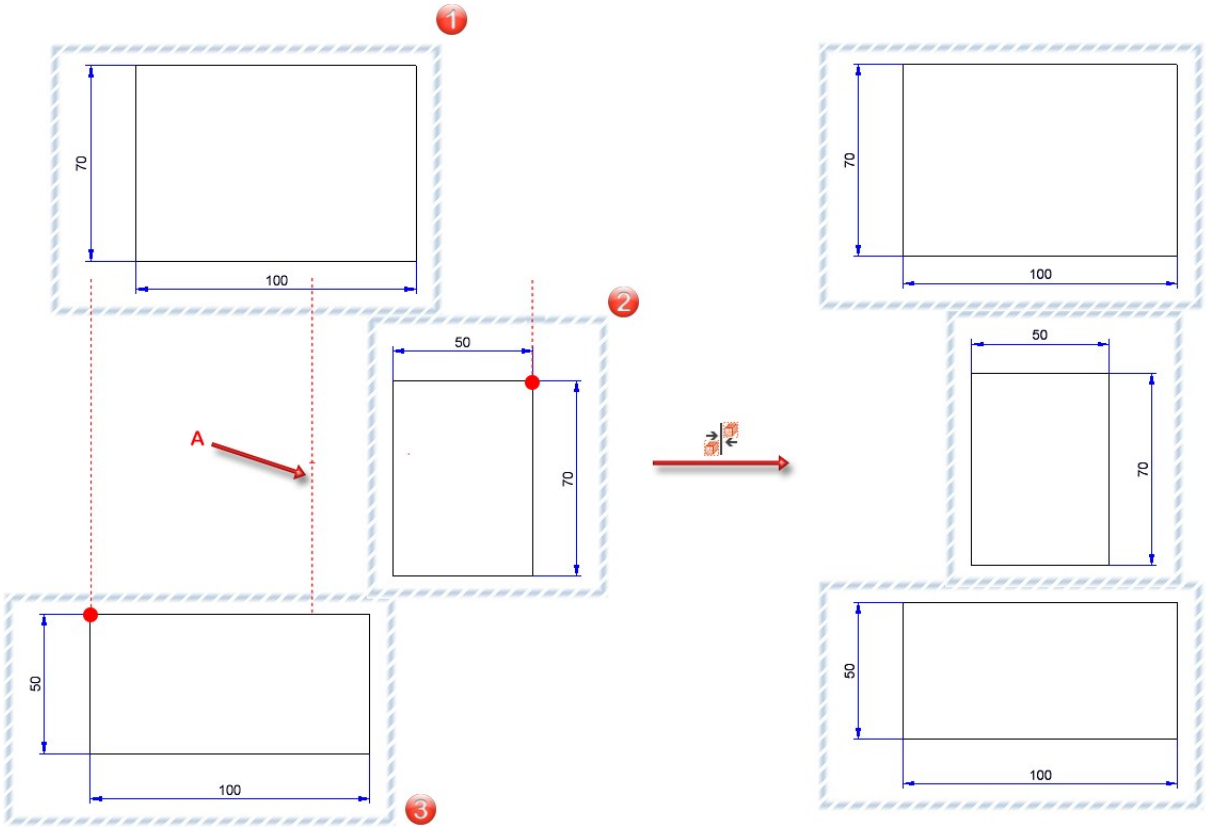
Align

With the flush alignment functions, only the displayed geometry is aligned; dimensioning, annotations etc. are not taken into account. This means that the smallest enveloping rectangle is used, which only completely encloses the geometry. The alignment line is determined by the "extremum" of the views. For example, in the right-aligned alignment, the view whose geometry extends furthest to the right remains in place. With the top alignment, the view with the highest geometry point remains in place.



The illustration shows three views, with view 3 projecting furthest to the left. On the right of the image you can see the result with left-aligned alignment. The position of view 3 is retained.

If you want to centre vertically, the alignment line is halfway between the top and bottom geometry points of the selected views. The views are then moved so that their centre lines lie on the alignment line. For horizontal centring, the alignment line is determined accordingly by the points furthest to the left and right of the selected views.

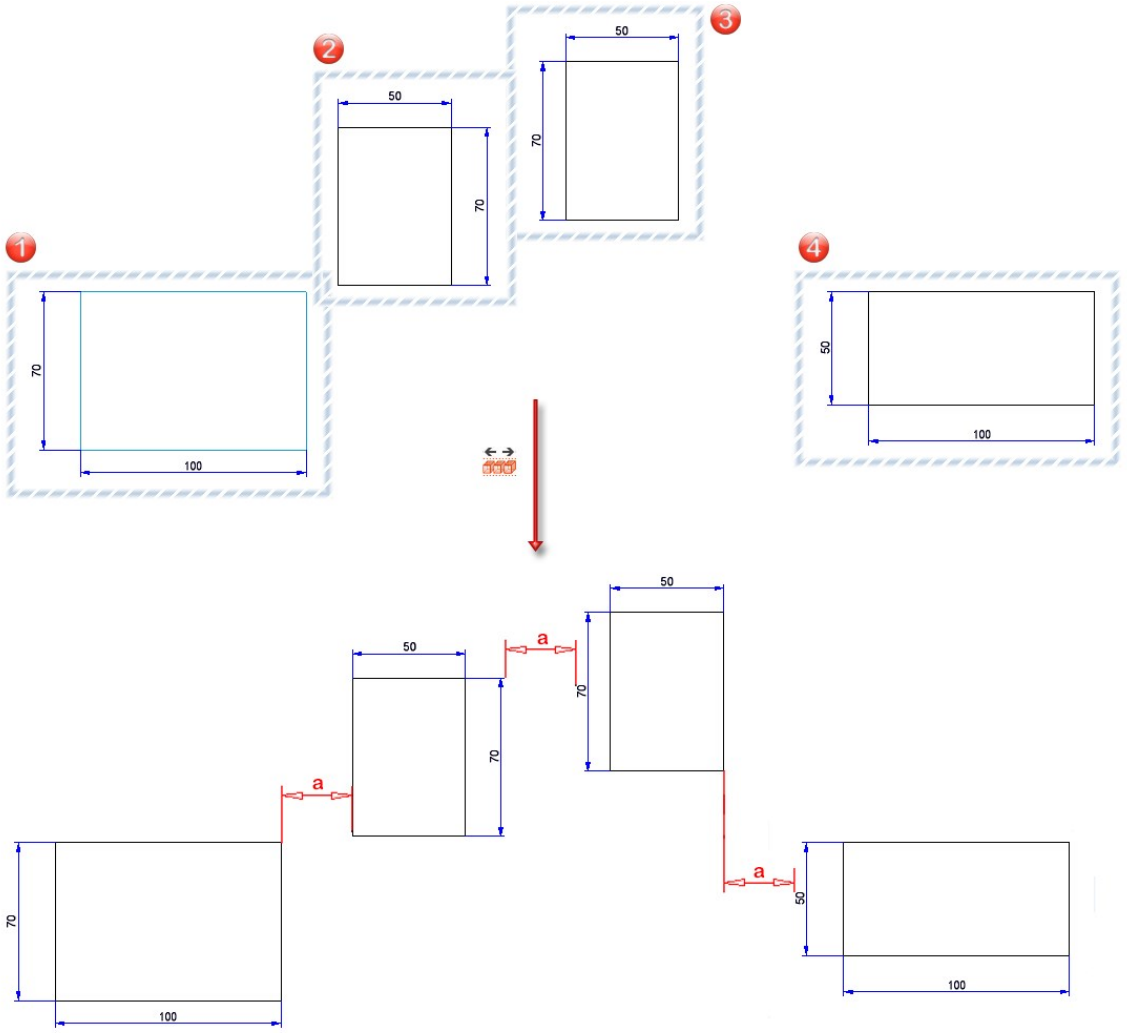


The illustration shows three views, with view 3 projecting furthest to the left and view 2 furthest to the right. A is then the alignment guideline. On the right of the image you can see the result with horizontally centred alignment.

Distribute

In contrast to alignment, dimensioning, annotations etc. are also taken into account when distributing in addition to the geometry. This means that the smallest enveloping rectangle that completely encloses all objects in a view is processed here.

Horizontal distribution moves the views to the right or left so that they are distributed laterally with the same lateral distance to the next view. The two views that protrude furthest to the left and right determine the overall width for the distribution and are not moved themselves.



The illustration shows four views that are to be distributed horizontally. Views (1) and (4) remain in place as they project furthest to the left and right respectively. Views (2) and (3) are moved horizontally so that the distance a between all views is the same.

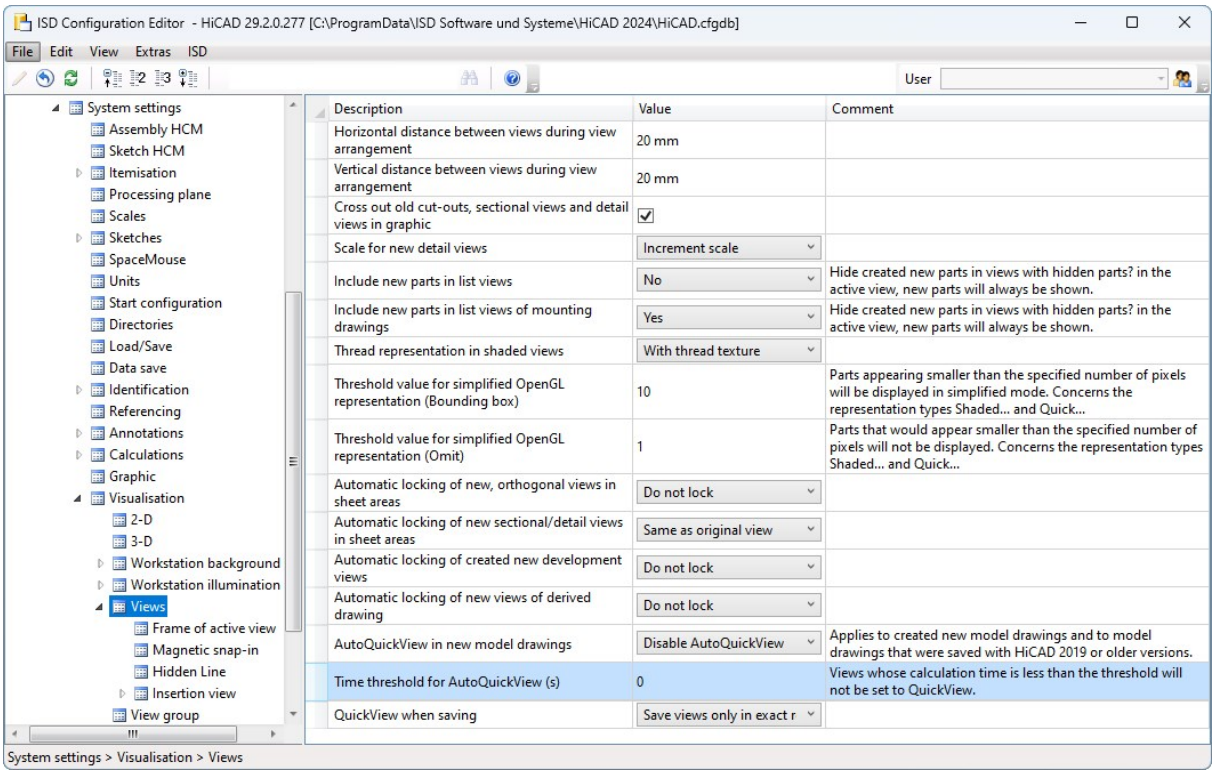
Time threshold for AutoQuickView (s)

A threshold value in seconds can now be specified for the AutoQuickView in the Configuration Editor.

If a threshold value > 0 is specified, the HiddenLine or glass model calculation is aborted if it takes longer than this threshold value. The view is then switched to QuickView. Subsequent calculations of the view no longer start a HiddenLine or glass model calculation, as the view is already in QuickView. The threshold value applies at the respective workstation for all drawings in which the AutoQuickView is switched on.

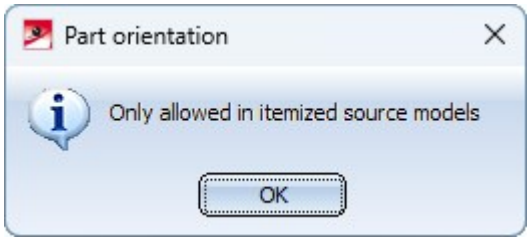
For example, in workshop drawings, a value of 0.1 to 0.5 seconds could mean that the views of the individual items remain in exact representation and only "large" views of the entire drawing are switched to QuickView.

The ISD default setting is 0, i.e. the QuickView takes effect for every HiddenLine and glass model calculation. This corresponds to the previous behaviour before HiCAD 2024 SP1.



Part and dimension orientation

The **Part and dimension orientation** can only be defined in drawings that are itemised source models. If this is not the case, a corresponding message is now displayed, e.g.



Transform and clone - Move+rotate via planes

The previous function

Move+Rotate part, via fitting points has been replaced in SP1

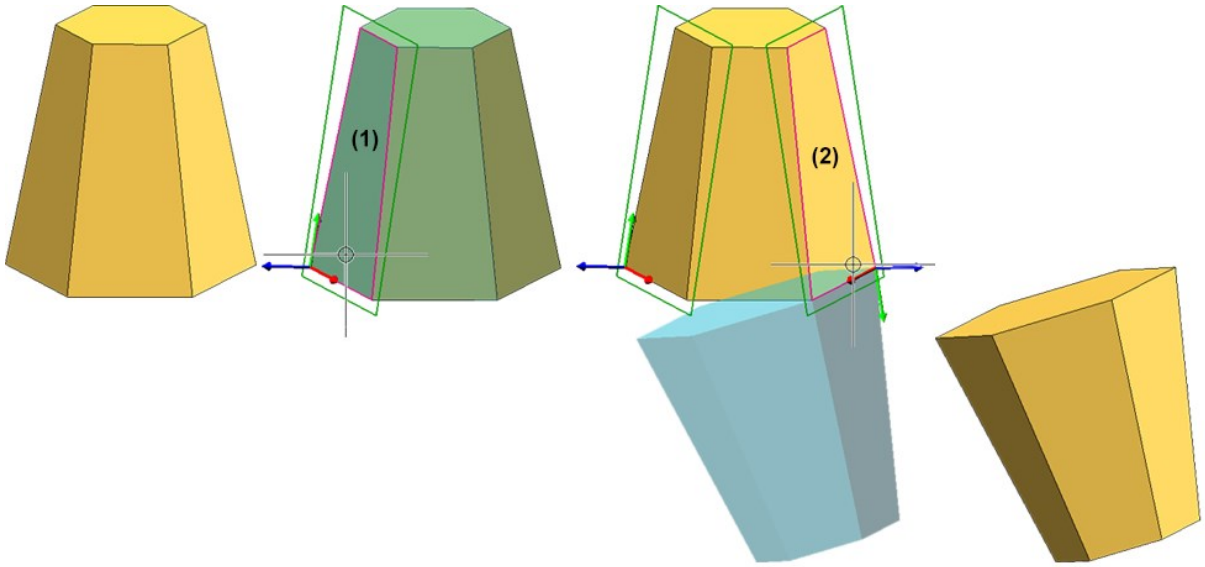
by the function

Move+Rotate part, via 2 planes.

Use the new function to move+rotate the active object, i.e. it is moved and rotated at the same time. The object can now be either a single part or a part list.

After calling the function, first select a plane on the object and then a plane in the drawing. The parts are transformed so that the coordinate systems of the two planes are aligned.

As soon as the plane in the drawing has been completely defined with the next click, a preview of the transformed part or part list is displayed.



To determine the planes, you can also use the functions of the context menu - just like when drawing sketches.

Similarly, the previous function

Clone+Move part

has been replaced by the function

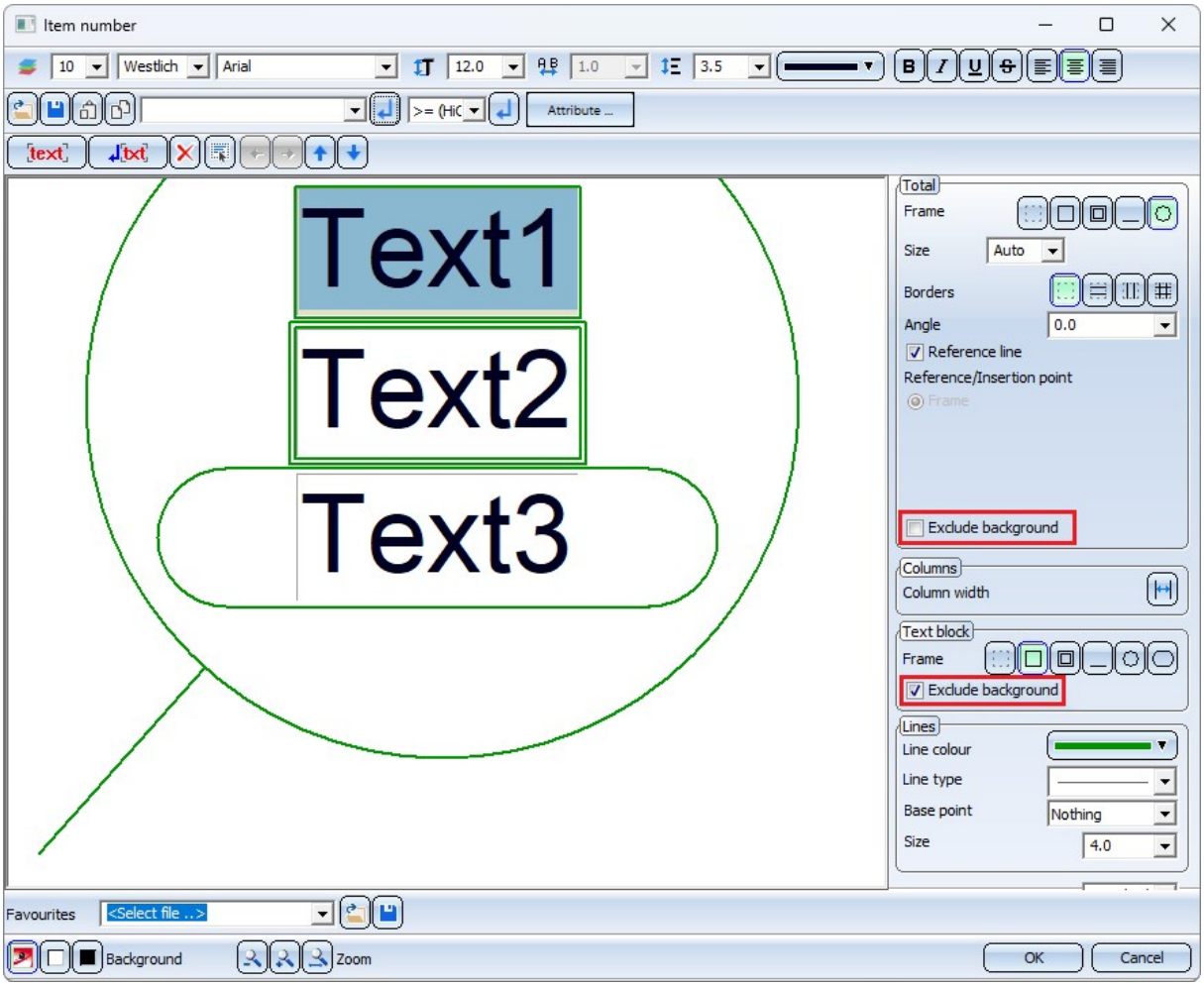
Clone part, via 2 planes.

Major Release 2024 (V 2900)

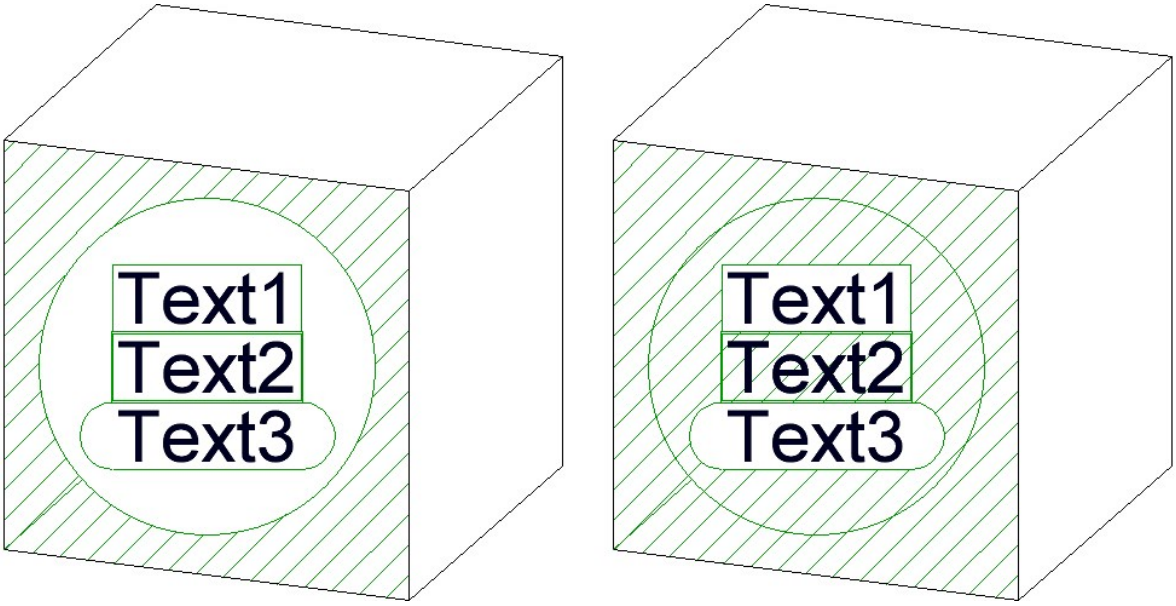
Dimensioning and annotations

3-D annotations - background cut-out

In 3-D annotations, the background cut-out can now be activated either for the entire annotation or for the individual text blocks.



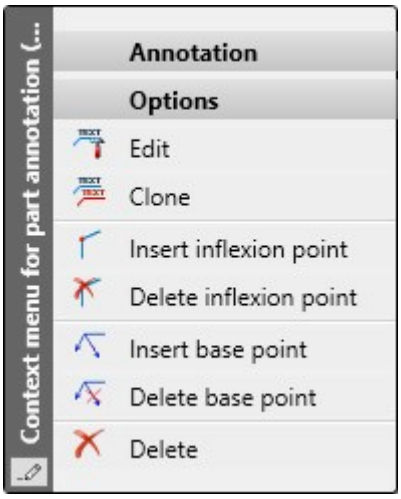
The checkbox for text blocks is only visible if the corresponding checkbox under **Total** is inactive.



Background cut-out - Left: Annotation, Right: Text blocks 1 and 3

New symbols in the context menus

The symbols in the context menus for annotations and annotation tags have been changed, e.g.



Form and positional tolerances

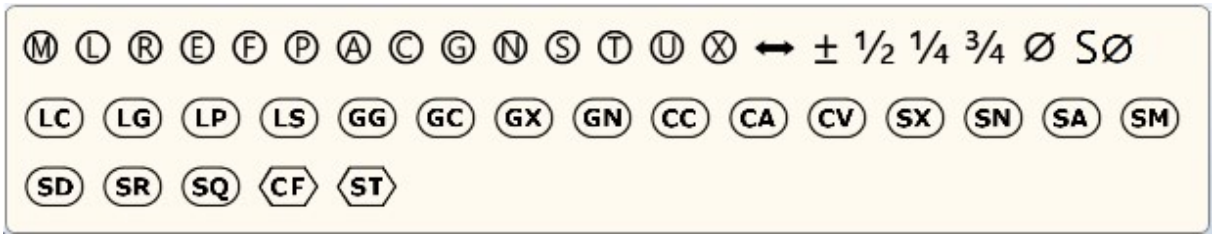
The new function for form/positional tolerances introduced with HiCAD 2023 has now been extended once again.

New inflexion point

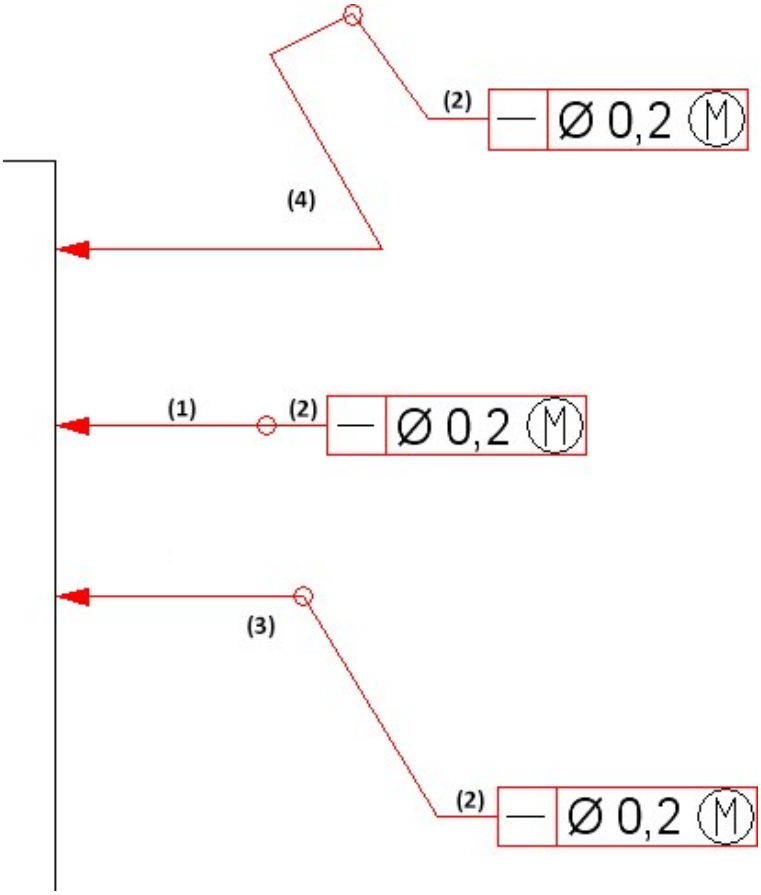
When setting form and positional tolerances, it was previously only possible to insert new inflexion points of the leader line by calling up the context menu (right mouse button) and then selecting the **New inflexion point** function. As of HiCAD 2024, this function can now also be called simply by pressing the **CTRL key**.

Other new features

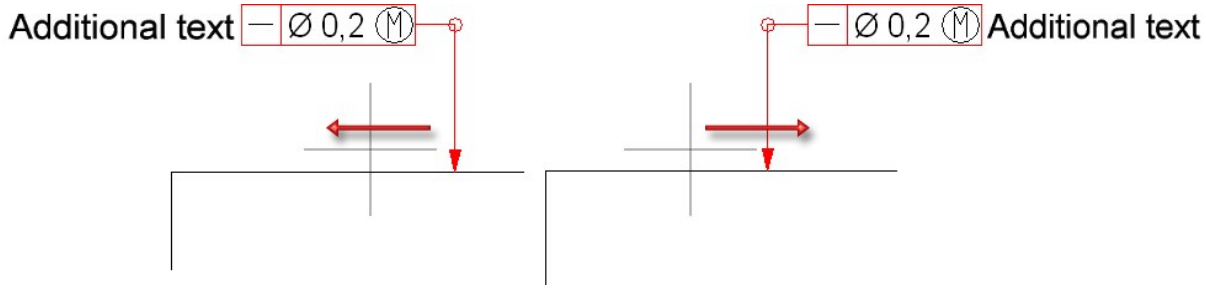
- The list of available symbols has been extended.



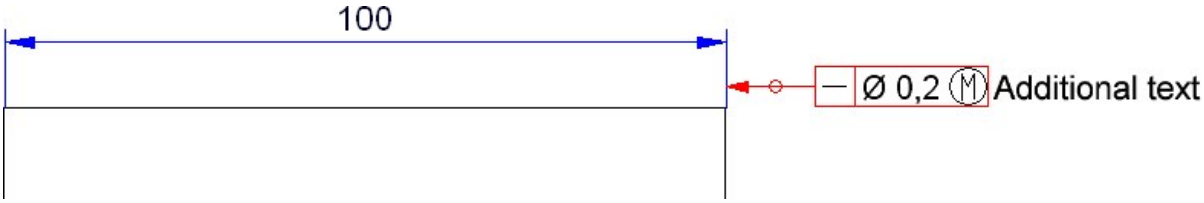
- The wrap-around symbol is now placed at the last inflexion point. If a reference line does not have an inflexion point, the wrap-around symbol is placed between the reference line (1) and the tying line (2).



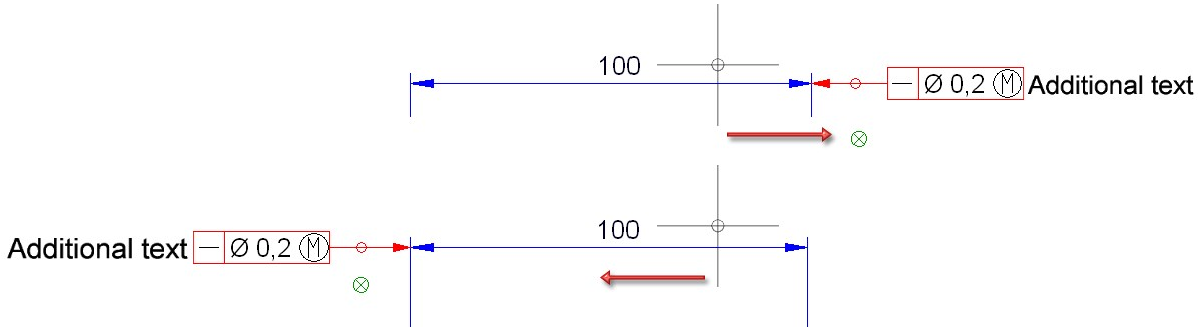
- Flags with a reference symbol can also be set at form/positional tolerances. The reference flag can then only be set at the top and bottom. If the form/positional tolerance is moved, the reference flag is also moved.
- The additional text of a line in form/positional tolerances changes the side when the side (right/left) of the form/-positional tolerance changes.



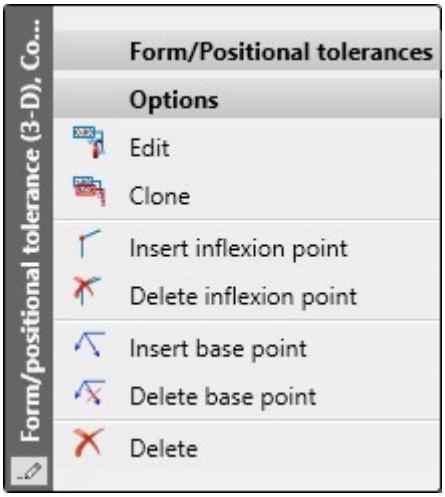
- Form/positional tolerances can be set at projection lines. If the dimension is moved, then the tolerance is also moved.



- If a tolerance is set to the dimension line of a distance or diameter dimension, then flag is no longer rigidly attached to the dimension line. By dragging the arrow, the reference point can also be set to the other side of the dimension.



- The context menu for form/positional tolerances has been adjusted to that of annotations.

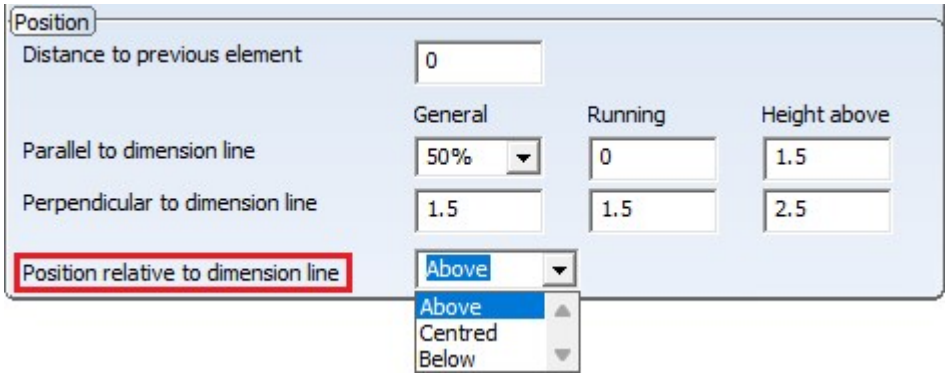


Simplified orientation of the dimension figure

Previously, the dimension number could only be defined by the distance to the dimension line. As of HiCAD 2024, it is now also possible to arrange the dimension number in a simplified way by selecting the desired option

- below or above the dimension line or
- centred on the dimension line.

For this purpose, the **Dimension figure** tab has been extended in the dimension parameter settings for interactive dimensions as well as for HCM and parameter dimensions.



3-D part annotation - Insert base point

With the **Insert base point** function, which you can find in the context menu of a 3-D part annotation, you can now directly add several base points to the selected annotation. You end the base point input with the middle mouse button.

Coating of general parts

Up to now, the coating of general parts (without structure) was not displayed in sectional and detail views. In addition, coating was not possible if these parts were part of a sectional or detail view.

As of HiCAD 2024, the coating of general parts (without structure) is also displayed in sectional and detail views.

Feature in part creation

In the part creation function dialogues, the **Feature** checkbox is no longer available as of HiCAD 2024. This means that a corresponding feature is now always generated during part generation.

This affects the following functions:

 3-D Standard > New > Extruded solids

 3-D Standard > New > Primitive

 3-D Standard > New > From Skt

 3-D Standard > Process > Wall > Envelope

 3-D Standard > Clone > Param.

 3-D Standard > New > Revolved

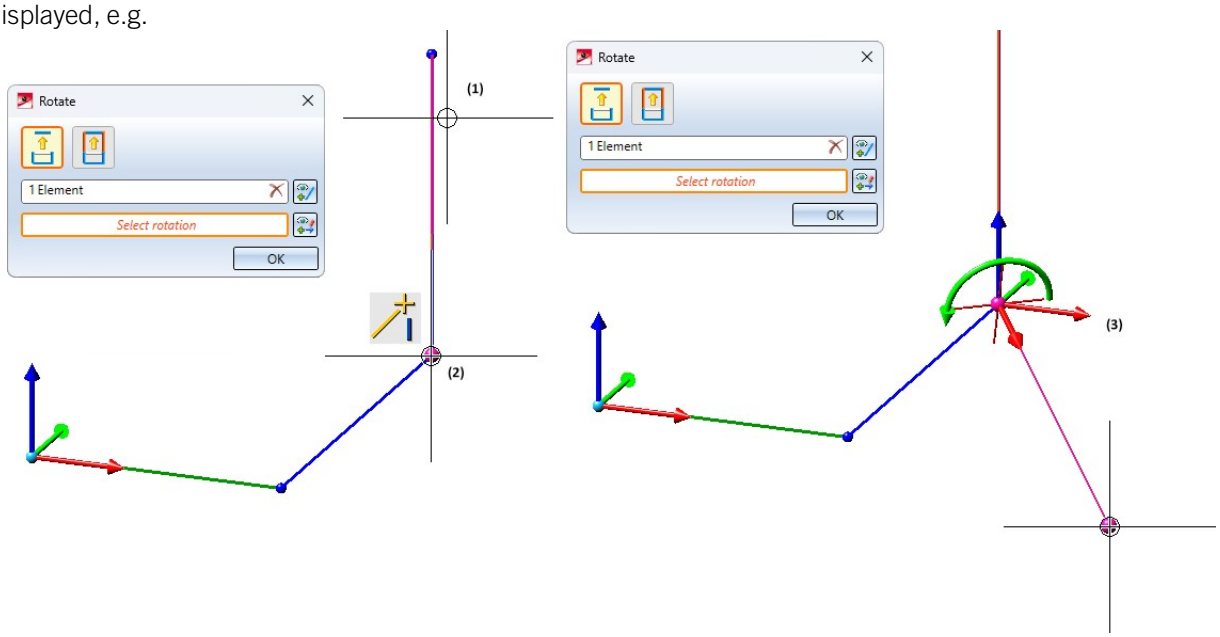
 3-D Standard > New > C-edge sweep

 Simplify > Envelope

Sketches

Simplified rotation of 3-D sketch elements

To simplify the determination of the rotation axis when **rotating elements of a 3-D sketch**, after selecting the sketch elements to be processed and determining the first point for defining the rotation axis, the active coordinate system is displayed, e.g.



(1) Selected sketch element, (2) 1st point for rotation axis, (3) Display of the coordinate system.

This affects the following functions:

 Sketch > Transform > Move > Move+Rotate

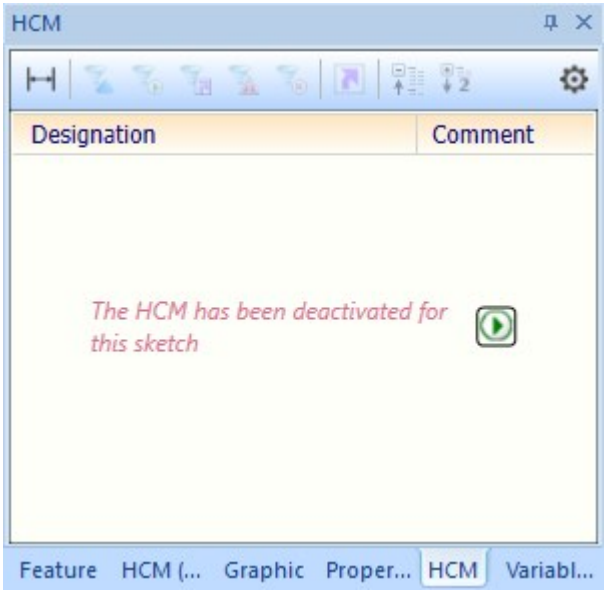
 Sketch > Transform > Rotate

 Sketch > Clone > Move > Move+Rotate

 Sketch > Clone > Rotate

Marking of sketches with deactivated HCM

If a sketch is active for which the automatic assignment of HCM constraints has been deactivated, this is now indicated in the HCM window of the ICN with a corresponding message.

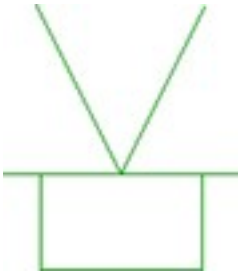


With a click on  the HCM can be activated again, i.e. HCM constraints are automatically assigned to subsequently created elements of the sketch - if possible.

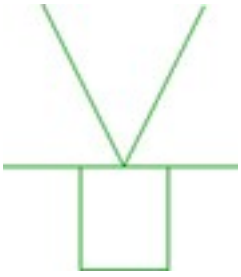
Weld seam and weld symbols

Extended weld symbols

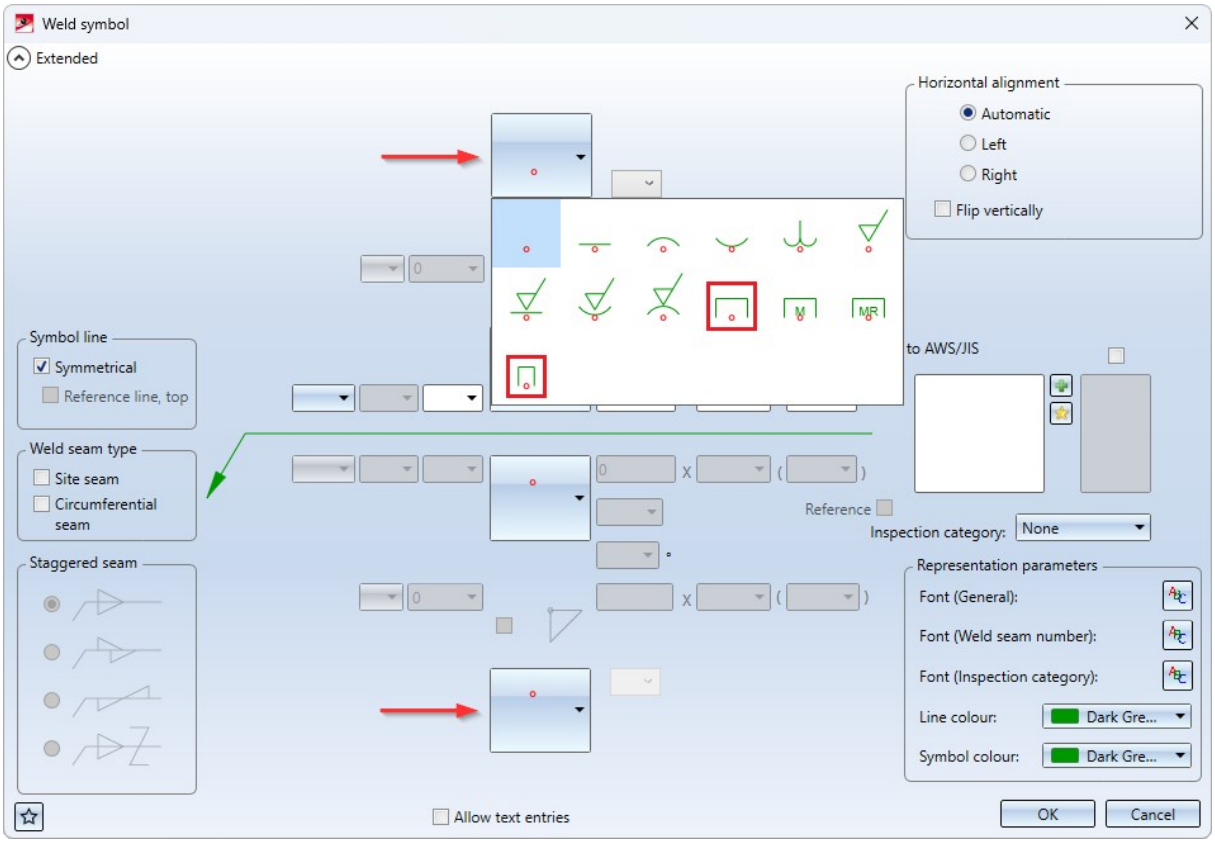
Additional weld symbols are available for symbolic representation on both the reference and opposite sides:



Weld pool fuse (not specified)

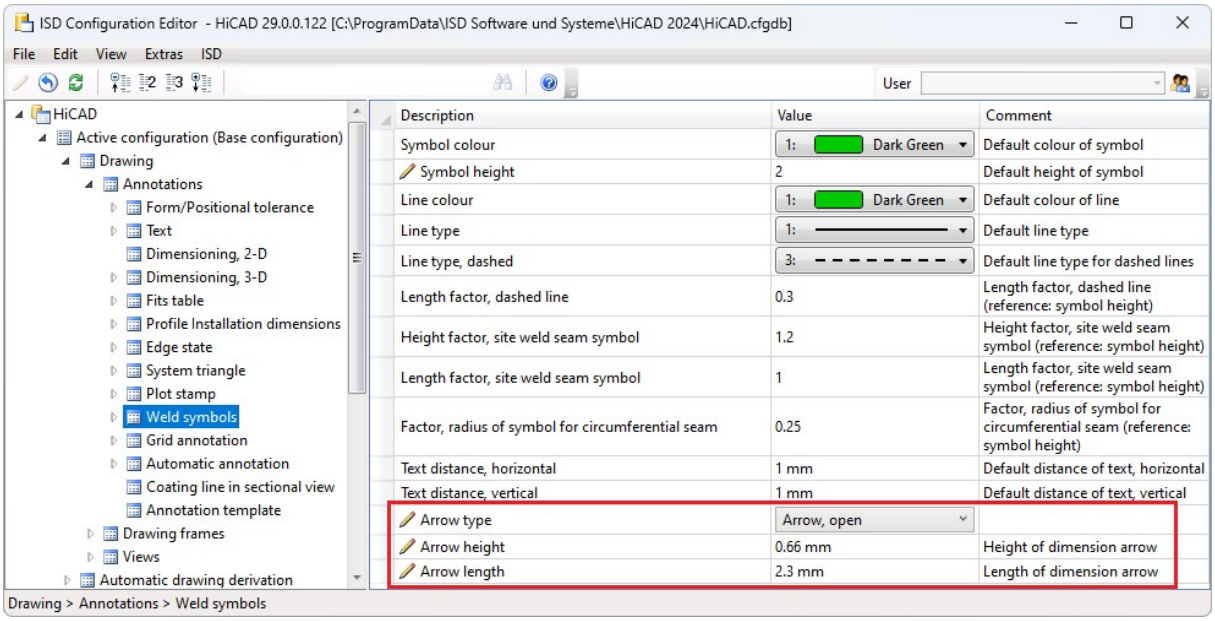


Fusible inlay



Reference line boundary arrow for 3-D weld seam symbols

When processing a 3-D weld seam symbol, the current settings from the Configuration Editor under **Drawing > Annotations > Weld symbols** were previously always used for the boundary of the reference line.



This could result in the display of the boundary changing, for example, when moving or copying. As of HiCAD 2024, the settings for the boundary of the reference line are saved directly to the weld seam symbol. This means that the limitation of existing weld seam symbols is not changed when the corresponding parameters are changed in Configuration Editor.

Show/hide weld seams by view

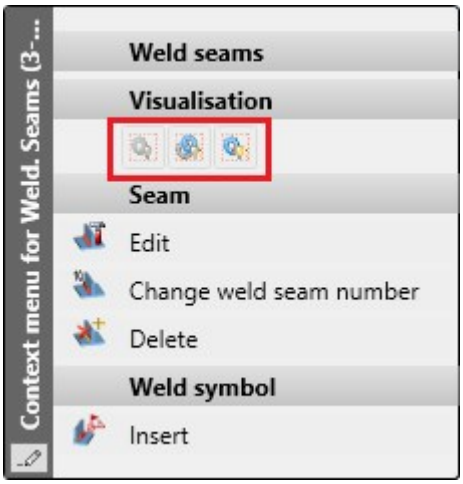
The functions

 Hide part list in view selection

 Show part list in view selection, hide all other parts and

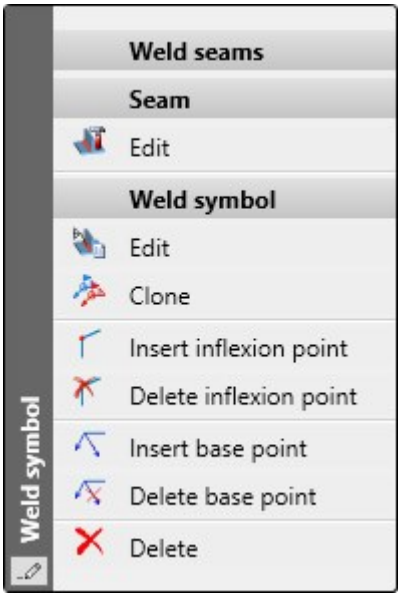
 Show part list in view selection

are now also available in the context menu for weld seams. This allows single or multiple seams to be shown/hidden in certain views.



New symbols in the context menus

The symbols in the context menus of welds have been changed.

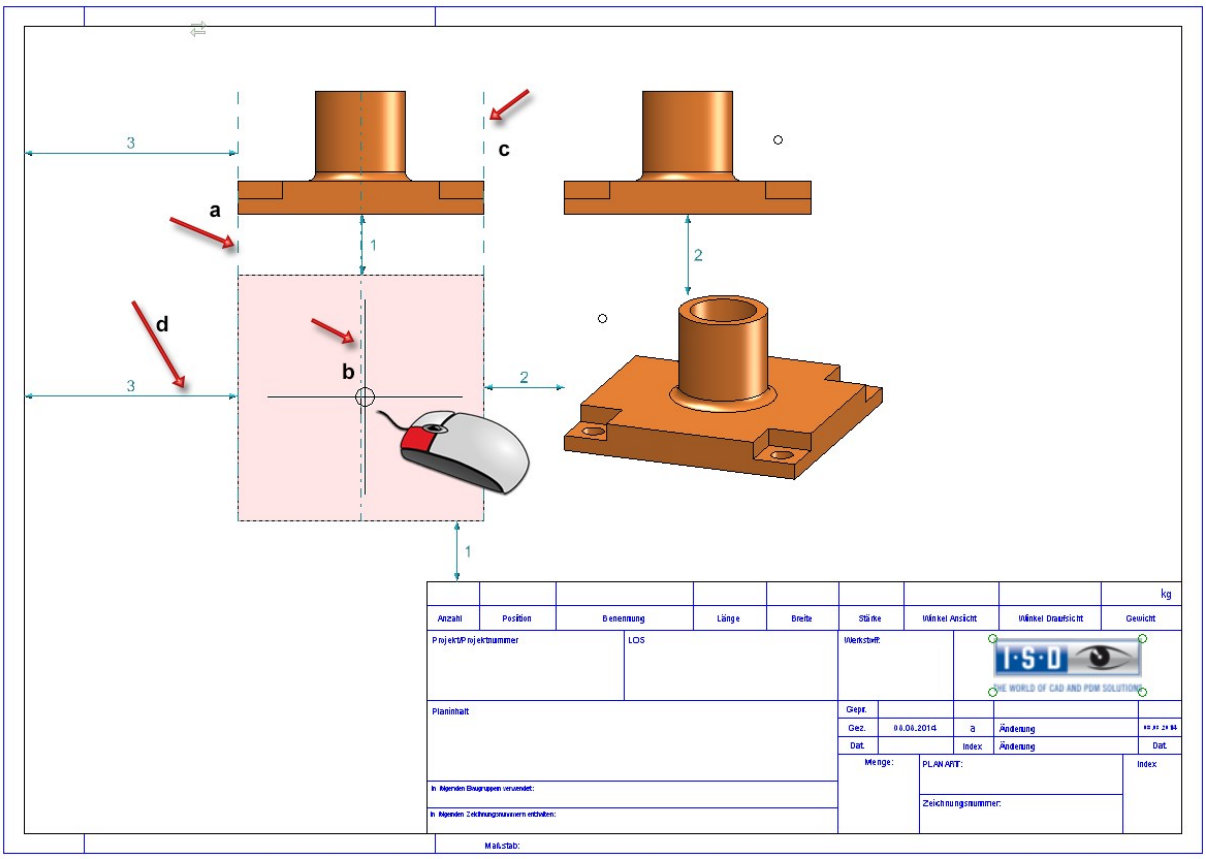


Views

Magnetic snap-in when moving

Magnetic snap-in of views is now supported when moving views. This procedure allows easy alignment with other views or with the optional additional elements of the drawing sheet (drawing frame, title block and BOMs).

1. The view edges can be aligned to be flush with each other or with the additional elements of the drawing sheet. If, for example, during dragging a view moves to the same height as an existing view or one of the additional elements, this is indicated in the graphic, e.g. by fading in the horizontal line. If you put the view down, it is aligned with this line. The same applies to vertical alignment.
2. Equal spacing does not refer to the geometry but to the view rectangle, i.e. the "envelope rectangle" that completely encloses the view including the annotations etc.

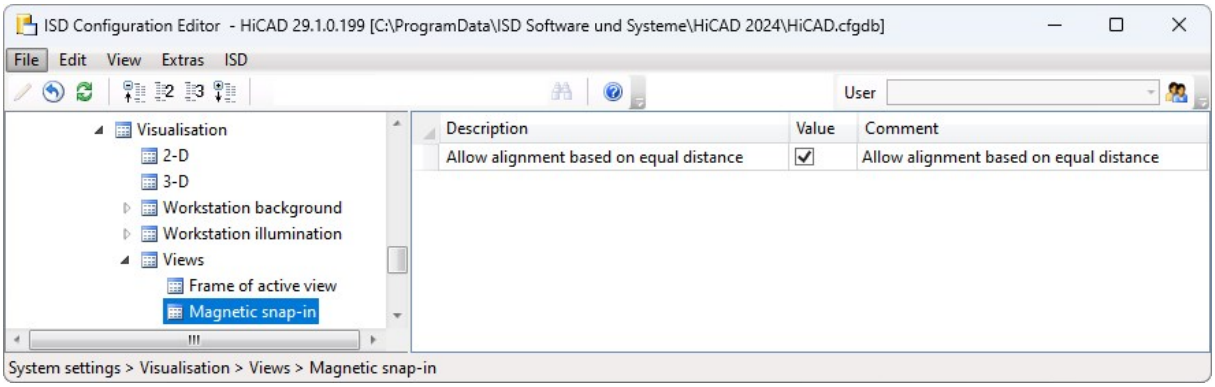


a-c: alignment lines left, centre, right, d: distance line

As soon as a corresponding constellation is created when dynamically moving a view/view list, graphical help elements such as distance arrows or orientation lines are displayed. Equal distances are numbered. By means of these lines the view can be aligned and the desired position can be taken over by lowering the cursor. While moving, the selected view is only displayed as an envelope rectangle.

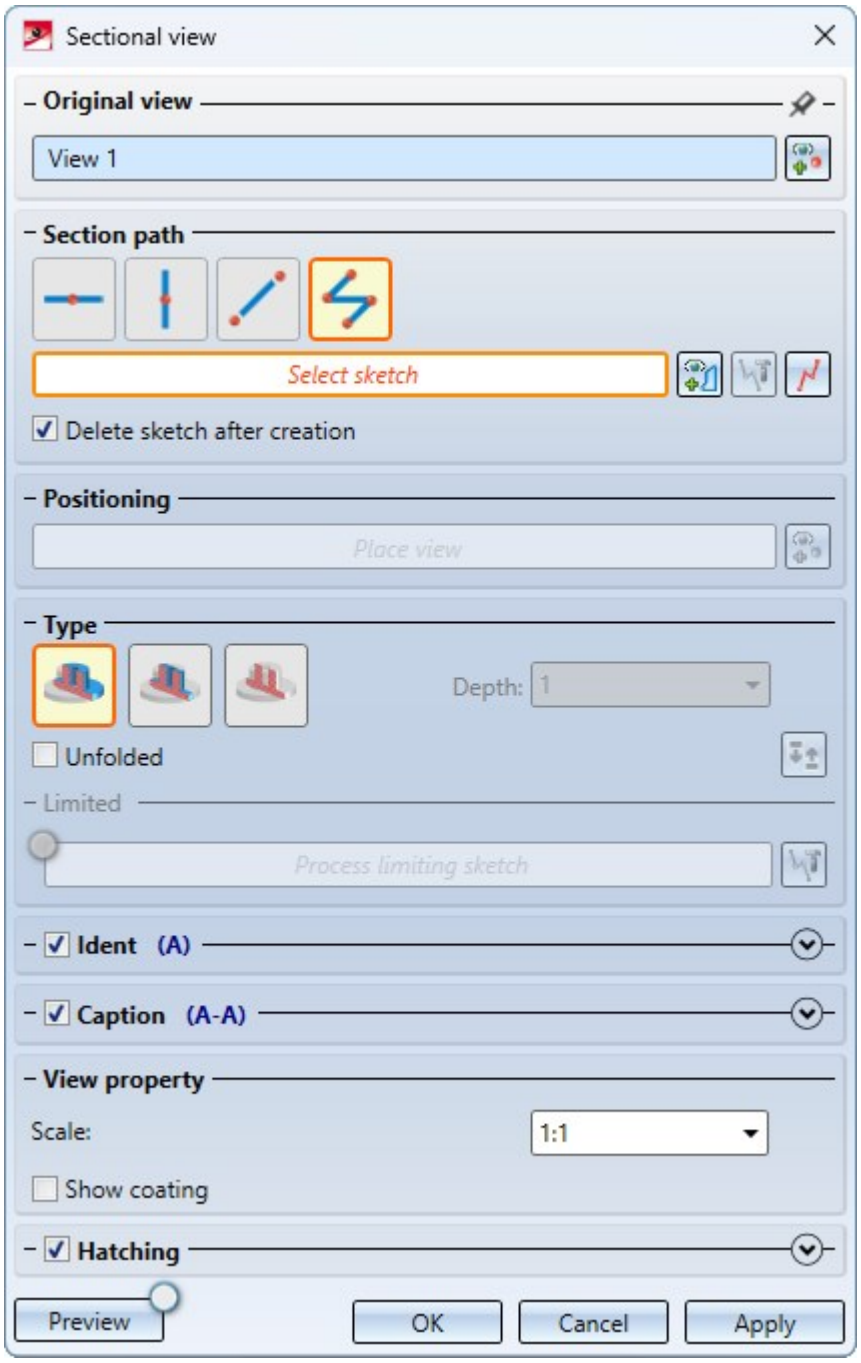
Magnetic snap-in is automatically active when moving with the Move view function and when moving using drag&drop.

Magnetic snap-in can be switched off in the Configuration Editor. To do this, under **System settings > Visualisation > Views > Magnetic snap-in**, deactivate the **Allow alignment based on equal distance** checkbox.



New dialogue for sectional views

The dialogue for creating and processing sectional views has been completely revised.



In addition, the course of a sectional view can now also be changed subsequently without having to call up the sectional view dialogue again. The


Process sketch

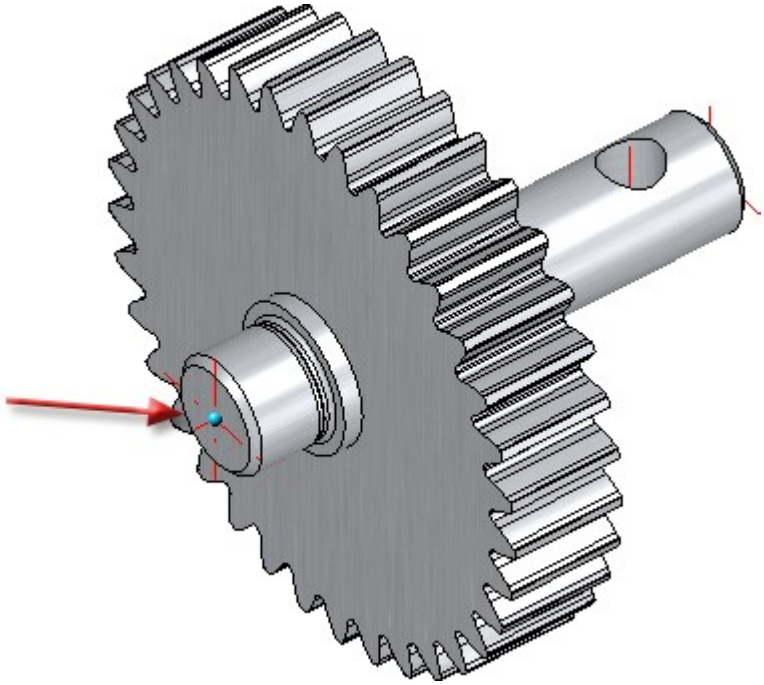
function is available for this purpose. You will find the function in the context menu of the annotation (right mouse button) in the original view.

Multiple selection of views

Until now, it was only possible to select multiple views in the ICN. This can be useful, for example, if you want to change the display of several views in one step. As of HiCAD 2024, this is now also possible in the graphics area of the drawing. To do this, hold down the CTRL key and select the desired views. All selected views are marked by a dashed view frame. If a view that has already been selected is selected again, then it is removed from the view list.

Rotating views

When rotating views with the mouse, the rotation point is displayed as of HiCAD 2024. If the rotation point is determined automatically with the middle mouse button, it is also displayed.



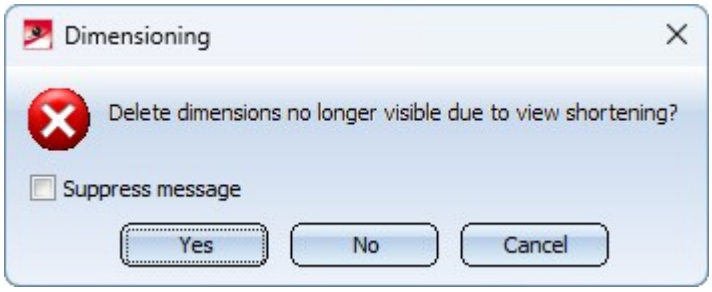
3-D rotation point

Deleting dimensions in shortened views

If dimension base points are present in the shortening area when shortening views, then from HiCAD 2024 you can select whether the corresponding dimensions should only be hidden or deleted.

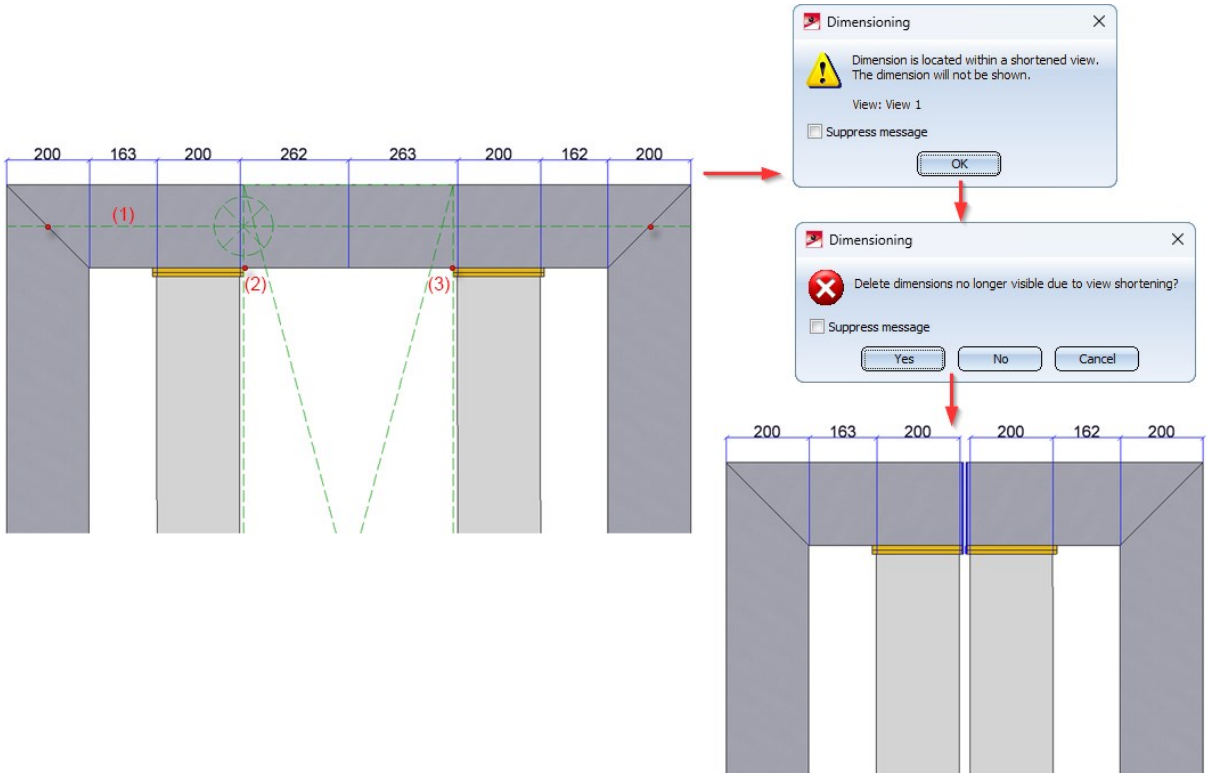
If such dimension base points are present, HiCAD will first display a corresponding message.

This message can be switched off for the current HiCAD session by activating the **Suppress message** checkbox. Click **OK** to continue the function.



Clicking **Yes** deletes the dimensions with base points in the shortening area. If you click **No**, the dimensions are only hidden. If the shortening is deleted, then these dimensions are visible again.

If the **Suppress message** checkbox is active, the selection of **Yes** or **No** applies to the current HiCAD session. The query will then only be displayed again after a restart.



Left: Initial view with shortening axis (1) and division points (2), (3), Right: Result

Temporarily deactivate view shortening

View shortening can now be temporarily deactivated. This can be useful, for example, to view or edit dimensions that are in the shortening area. You can find the functions under **Views > Edit > Shorten** as well as in the context menu for views.

Temporarily deactivate view shortening

The shortening of the active view is temporarily deactivated, i.e. the view is displayed unshortened until the deactivation is removed. The deactivation is saved with the drawing, i.e. if the view shortening is deactivated when saving, it will also be deactivated when the drawing is reopened.

Cancel temporarily activation of view shortening

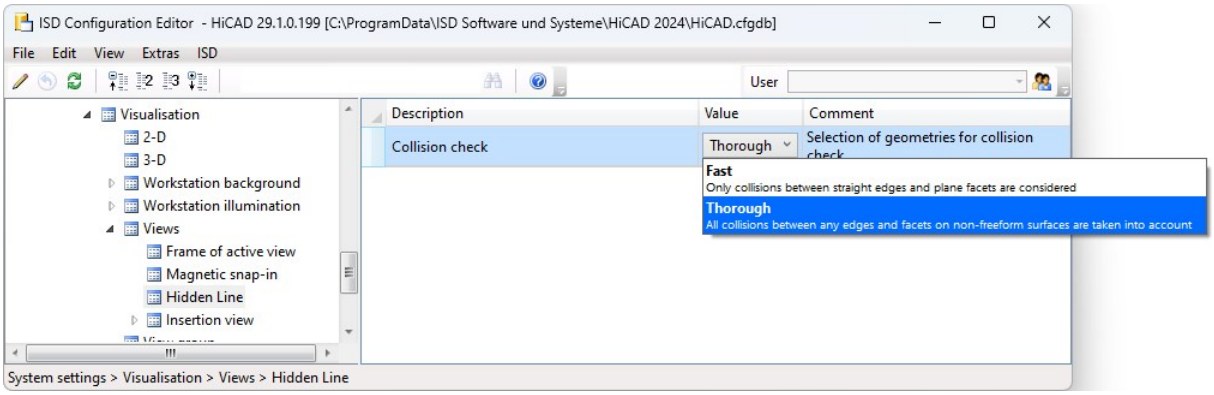
This function cancels the temporary deactivation of the active view. The view is displayed shortened again.

Consideration of intrusions in Hidden Line representations

Up to now, only intrusions/collisions found between straight lines and planes or those involving circles and cylinders were taken into account in Hidden Line calculations.

As of HiCAD 2024, all collisions with analytical curves/surfaces or with NURBS curves/surfaces are now also taken into account.

What exactly is taken into account can be defined in Configuration Management under **System settings > Visualisation > Views > Hidden Line > Collision check**.



Fast

Only collisions between straight edges and plane facets are considered.

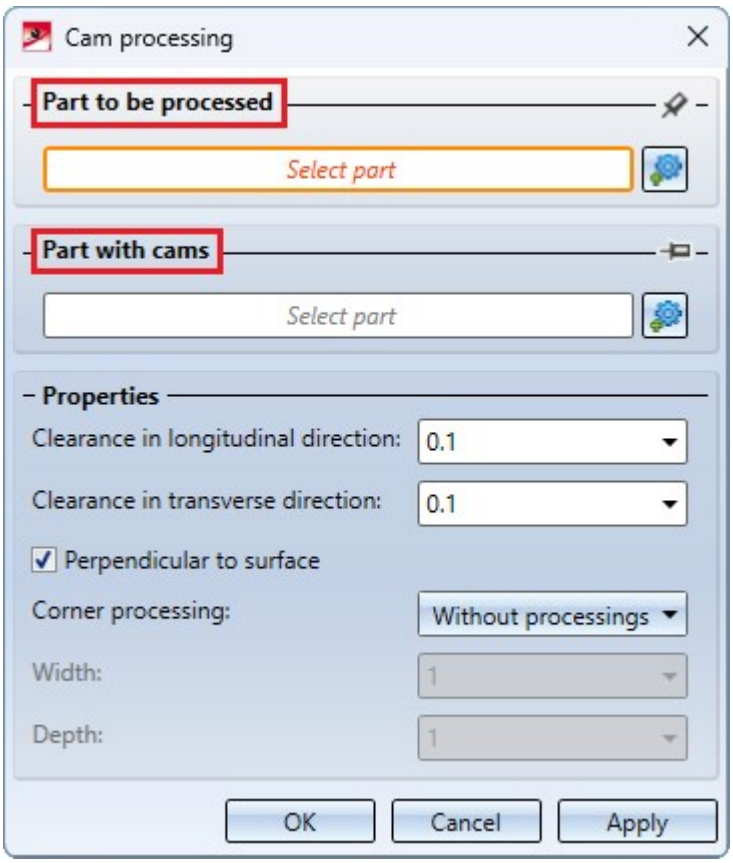
Thorough

With this setting, all collisions between any edges and facets on non-freeform surfaces are taken into account. This is the ISD default setting.

Cams and cam processings

The controls and dialogues for cams and cam edits have been slightly changed.

- When creating cams, multiple edges can now be selected in the selection list of the **Cams** dialogue window so that you can provide them with the same cams. This means you no longer need to make the settings for each individual edge. Alternatively, you can set the parameters for shape and distribution first and then select the edges.
- In the **Cam processing** dialogue window, the dialogue texts have been adapted.



- Processings that are not perpendicular also go all the way through.
- Both dialogue windows can be resized.

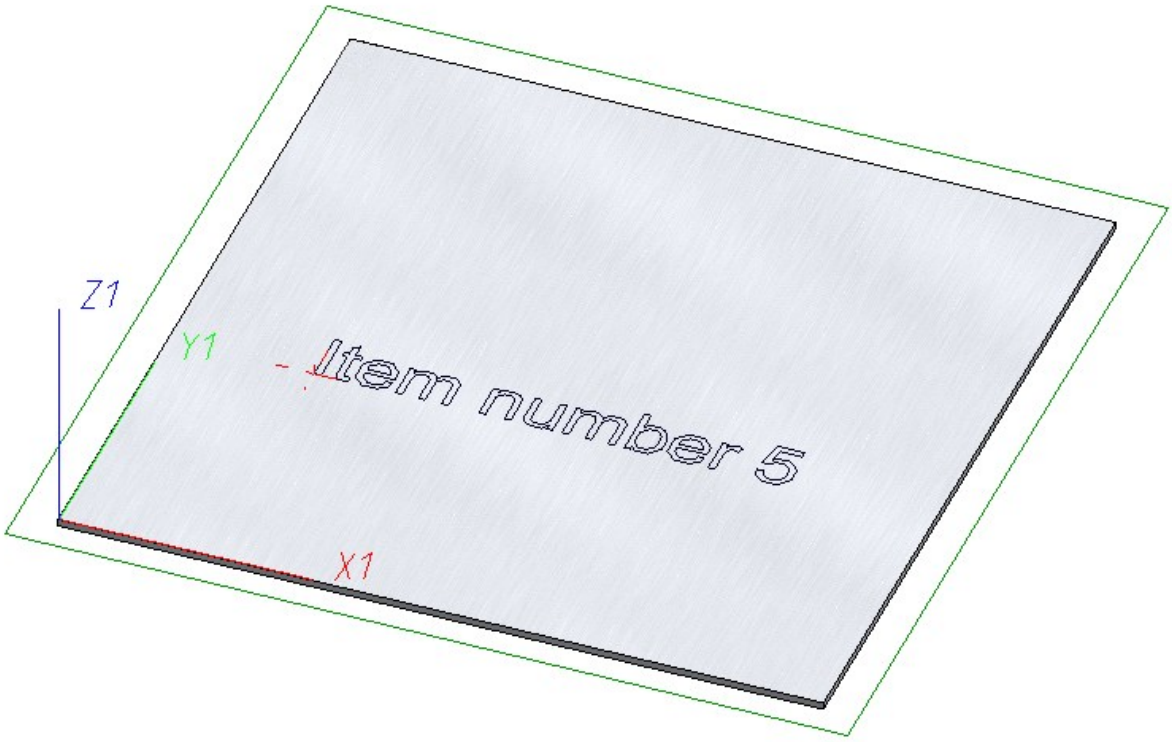
Update automatically calculated attributes when loading

In the Configuration Editor you can now define whether the dimensions/attributes for which the automatic calculation is activated there should be updated automatically or not when loading a drawing. Read the information on this under Basics - What's New?.

Extensions in lettering

The extensions in lettering allow you to insert attributes, save favourites and control the display in the view.

- **Attributes**
  Attributes can be easily copied from the list box. Select the attribute and click **OK**. There are some restrictions. Attributes of superordinate parts may not be used. The lettering text must not depend on the position in the part structure. If the part is referenced, only attributes that are transferred via referencing may be used. This can be set in the Configuration Editor under **System settings** > **Referencing** > Area: **Updating**.
  Examples of non-permitted attributes: The attribute **Benennung 2** is configurable, i.e. non-permitted attributes can be used. The attribute **Positionsindex** is not permitted because parts with the same position number can have different item indices.
- **Multi-line signatures**
  You can now enter multi-line texts in the text editor of the lettering.
- **Save settings as favourites**
  Letterings can be saved as favourites. The file format is *.FTD. In the HiCAD SYS directory you will find various FTD files with predefined lettering.
- **Showing and hiding letterings**
  The lettering can be shown or hidden in the view. To do this, right-click on the view frame and select the function Show/hide elements in view.



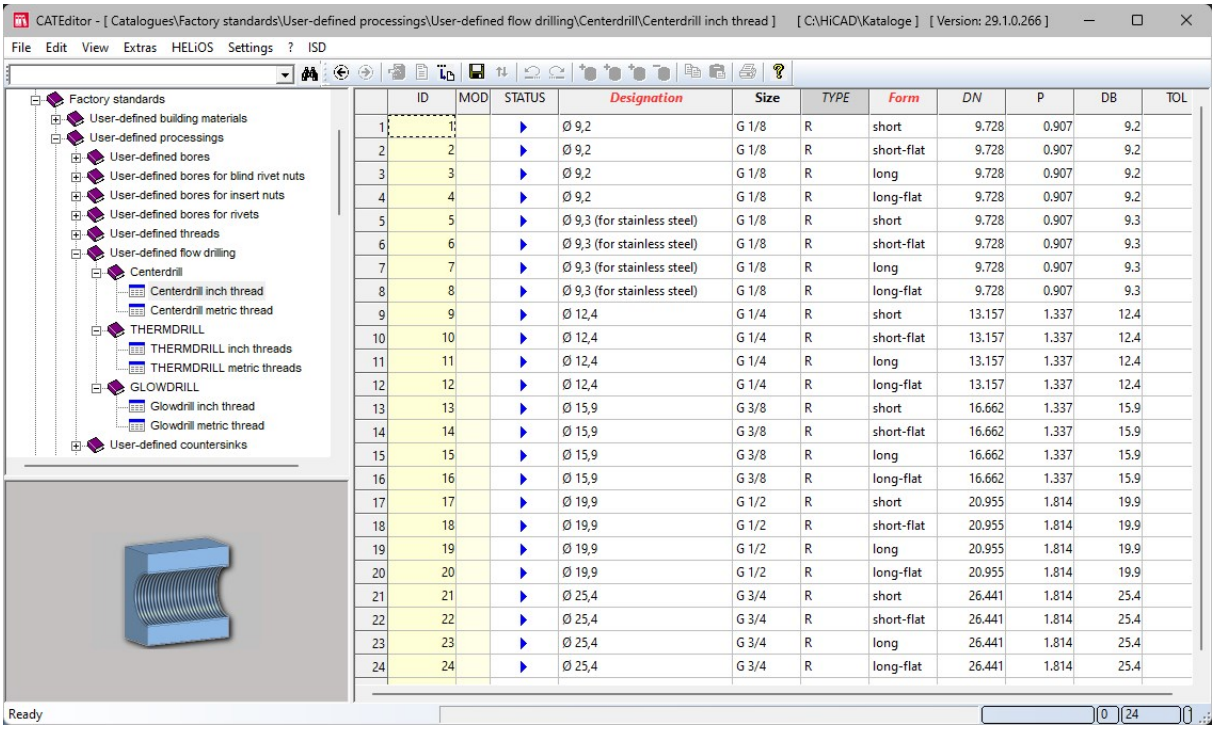
Catalogue Editor

Service Pack 1 2024 (V. 2901)

Flow drillings

New in the **Factory standards > User-defined processings** catalogue are **User-defined flow drillings** with inch thread and metric thread from the suppliers

- Centerdrill GmbH,
- Ontool GmbH (THERMDRILL®) and
- GLOWDRILL GmbH.



	ID	MOD	STATUS	Designation	Size	TYPE	Form	DN	P	DB	TOL
1	1			Ø 9,2	G 1/8	R	short	9.728	0.907	9.2	
2	2			Ø 9,2	G 1/8	R	short-flat	9.728	0.907	9.2	
3	3			Ø 9,2	G 1/8	R	long	9.728	0.907	9.2	
4	4			Ø 9,2	G 1/8	R	long-flat	9.728	0.907	9.2	
5	5			Ø 9,3 (for stainless steel)	G 1/8	R	short	9.728	0.907	9.3	
6	6			Ø 9,3 (for stainless steel)	G 1/8	R	short-flat	9.728	0.907	9.3	
7	7			Ø 9,3 (for stainless steel)	G 1/8	R	long	9.728	0.907	9.3	
8	8			Ø 9,3 (for stainless steel)	G 1/8	R	long-flat	9.728	0.907	9.3	
9	9			Ø 12,4	G 1/4	R	short	13.157	1.337	12.4	
10	10			Ø 12,4	G 1/4	R	short-flat	13.157	1.337	12.4	
11	11			Ø 12,4	G 1/4	R	long	13.157	1.337	12.4	
12	12			Ø 12,4	G 1/4	R	long-flat	13.157	1.337	12.4	
13	13			Ø 15,9	G 3/8	R	short	16.662	1.337	15.9	
14	14			Ø 15,9	G 3/8	R	short-flat	16.662	1.337	15.9	
15	15			Ø 15,9	G 3/8	R	long	16.662	1.337	15.9	
16	16			Ø 15,9	G 3/8	R	long-flat	16.662	1.337	15.9	
17	17			Ø 19,9	G 1/2	R	short	20.955	1.814	19.9	
18	18			Ø 19,9	G 1/2	R	short-flat	20.955	1.814	19.9	
19	19			Ø 19,9	G 1/2	R	long	20.955	1.814	19.9	
20	20			Ø 19,9	G 1/2	R	long-flat	20.955	1.814	19.9	
21	21			Ø 25,4	G 3/4	R	short	26.441	1.814	25.4	
22	22			Ø 25,4	G 3/4	R	short-flat	26.441	1.814	25.4	
23	23			Ø 25,4	G 3/4	R	long	26.441	1.814	25.4	
24	24			Ø 25,4	G 3/4	R	long-flat	26.441	1.814	25.4	

Flow drill holes can be inserted in HiCAD with the function **3-D Standard > Standard Processings > Bore/Thr**  . The bores are displayed in HiCAD like normal drilled holes.

Major Release 2024 (V 2900)

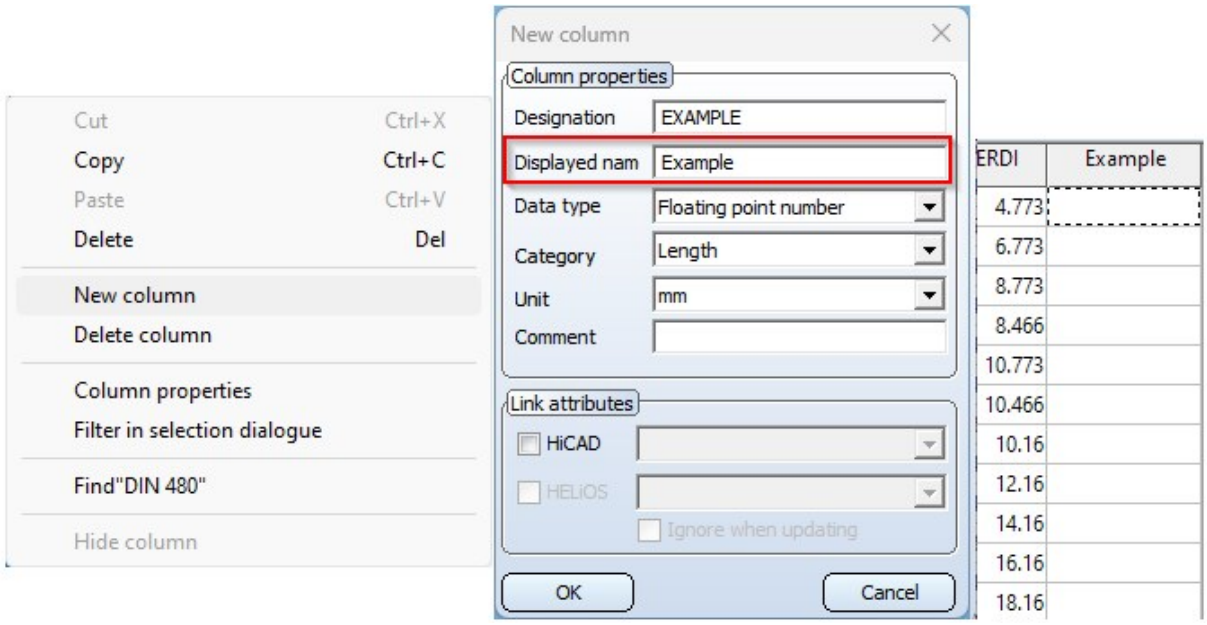
Tool numbers in sheet metal processing

In sheet metal processing, machines are also used which use different tools for top and bottom processing, for example punch- laser combination machines. Until now, different representations (TOPSYMBOL / BOTTOMSYMBOL) could be used in the development, but different tool numbers for these processings could not be assigned. As of HiCAD 2024, this is now possible. For this purpose, the catalogs for molding, embossing and punching tools have been extended by the WZNR_BOTTOM column, which can be individually assigned values.

In the sheet development, the side of the processing is read out and the appropriate symbol is used - TOPSYMBOL or BOTTOMSYMBOL. At the same time, the values of the columns WZNR or WZNR_BOTTOM (depending on the processing page) are read out and written to the development. If WZNR_BOTTOM is empty, then WZNR is used instead. (see also Sheet Metal - What's new?)

User-defined columns in catalogue tables

Until now, the creation of user-defined columns was only possible for tables in the **Factory standards** catalogue. From HiCAD 2024 this is supported for tables in all catalogues. The table function **New column** has been extended for this purpose. It can now be used to create user-defined columns of any type and - if desired - to name them meaningfully. In addition to the name, which was previously also used as the column heading, a **Displayed name** can now be specified, for example for translations. If no displayed name is specified, the designation as column header is still used.



With this extension, the columns CUSTOM1 to CUSTOM9, which were previously predefined for this purpose (before HiCAD 2024), are no longer needed and have been removed from the tables.

During a catalogue update, however, only empty CUSTOM columns are removed from the tables.

Displayed names of table columns adapted

In various ISD catalogues, more informative displayed names have now been assigned to the table columns, so that it is easier to recognise which data the table columns contain.

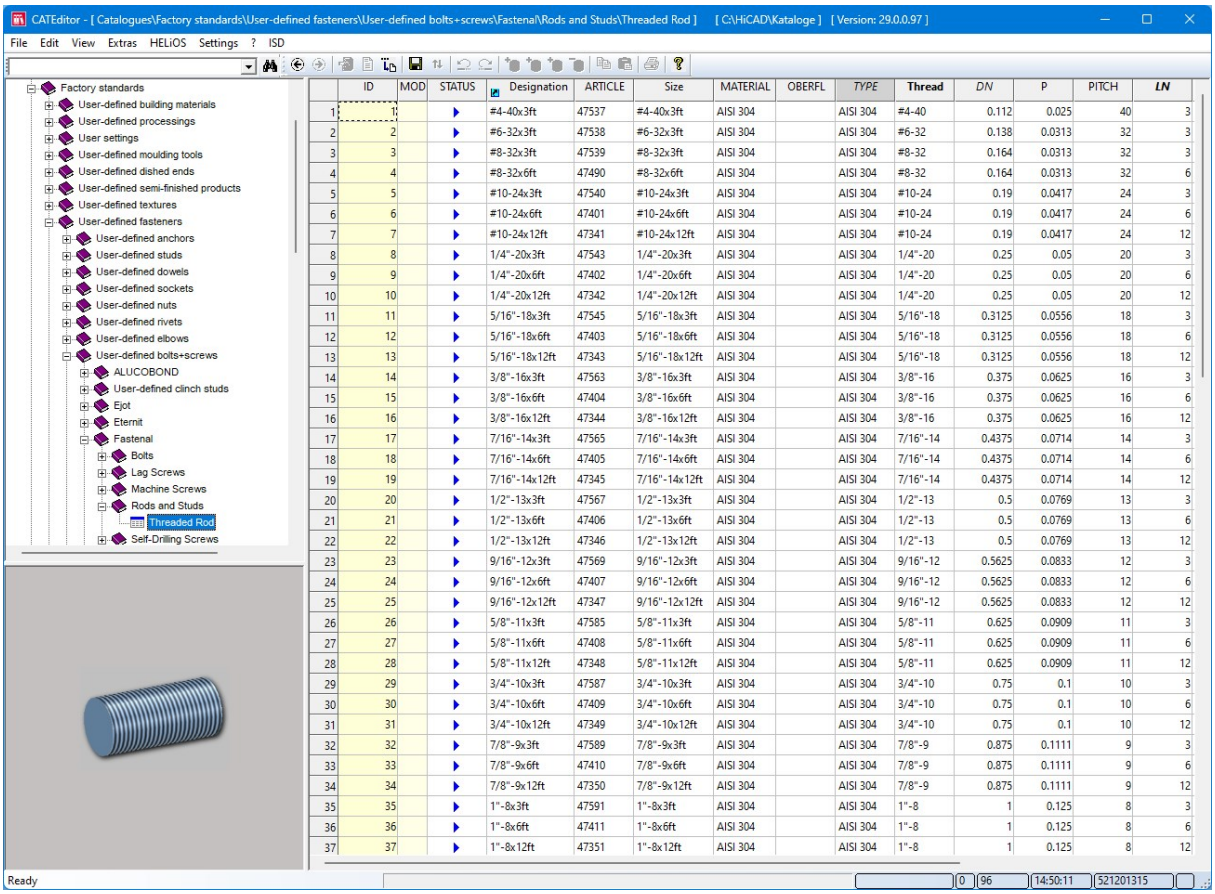
The following displayed names have been assigned:

Column designation	Displayed name
BZ	Designation
GEW	Thread
GEW	Weight
KILO	Weigth
SIZE	Size
HGEW	Commercial weight
BZ_2004	Designation as of 2004
RHO	Density

Furthermore, these displayed names are language-dependent and adapt to an English installation, for example.

Fastenal - Threaded Rods

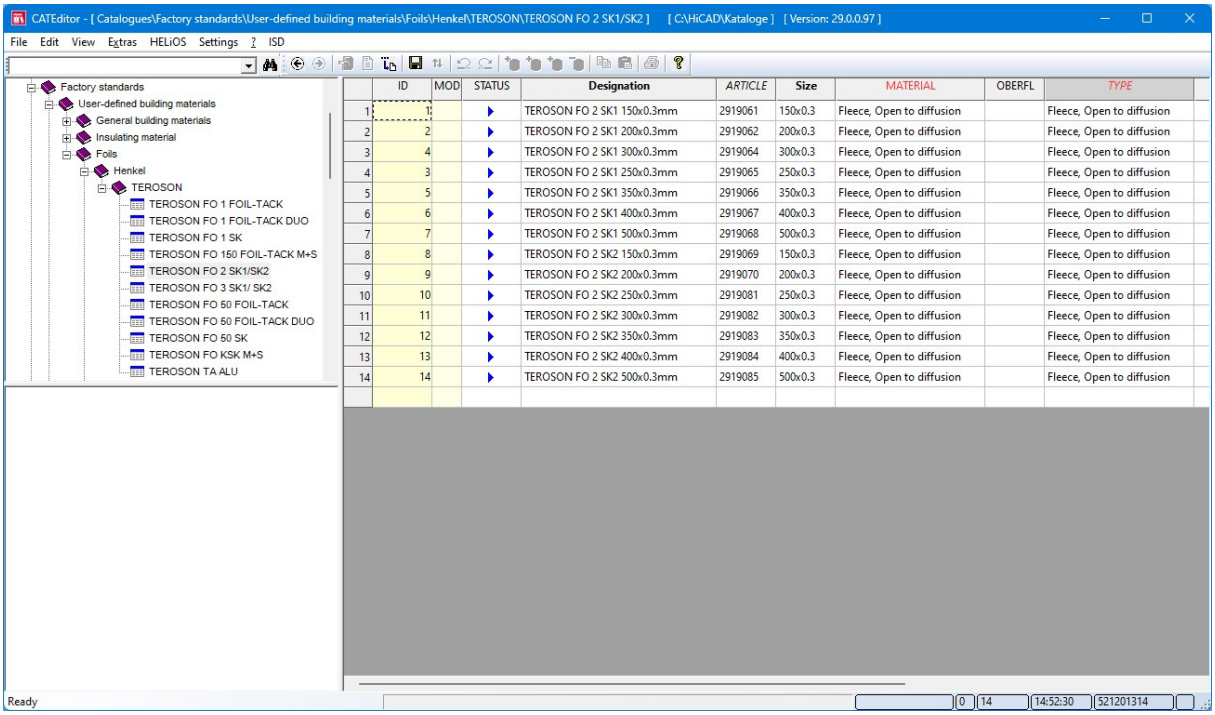
The catalogue **Factory standards > Custom fasteners > Custom bolts + screws > Fastenal** has been extended by Threaded Rods. You can find them at **Rods and Studs/Threaded Rods**.



Henkel Teroson foils

Der Katalog **Werksnormen/Anwender Baustoffe/Folien/Henkel/TEROSON** ist um eine Tabelle mit dampfoffenen Fassadenfolien erweitert worden: **TEROSON FO2 SK1/SK2**.

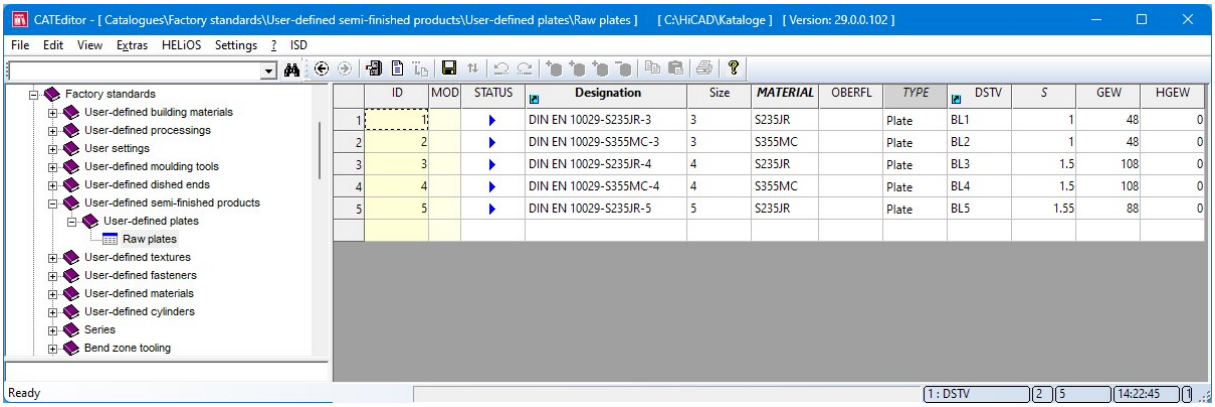
The table **TEROSON FO2 SK1/SK2** with vapour permeable facade foils has been added to the catalogue **Factory standards/User-defined building materials/Foils/Henkel/TEROSON**.

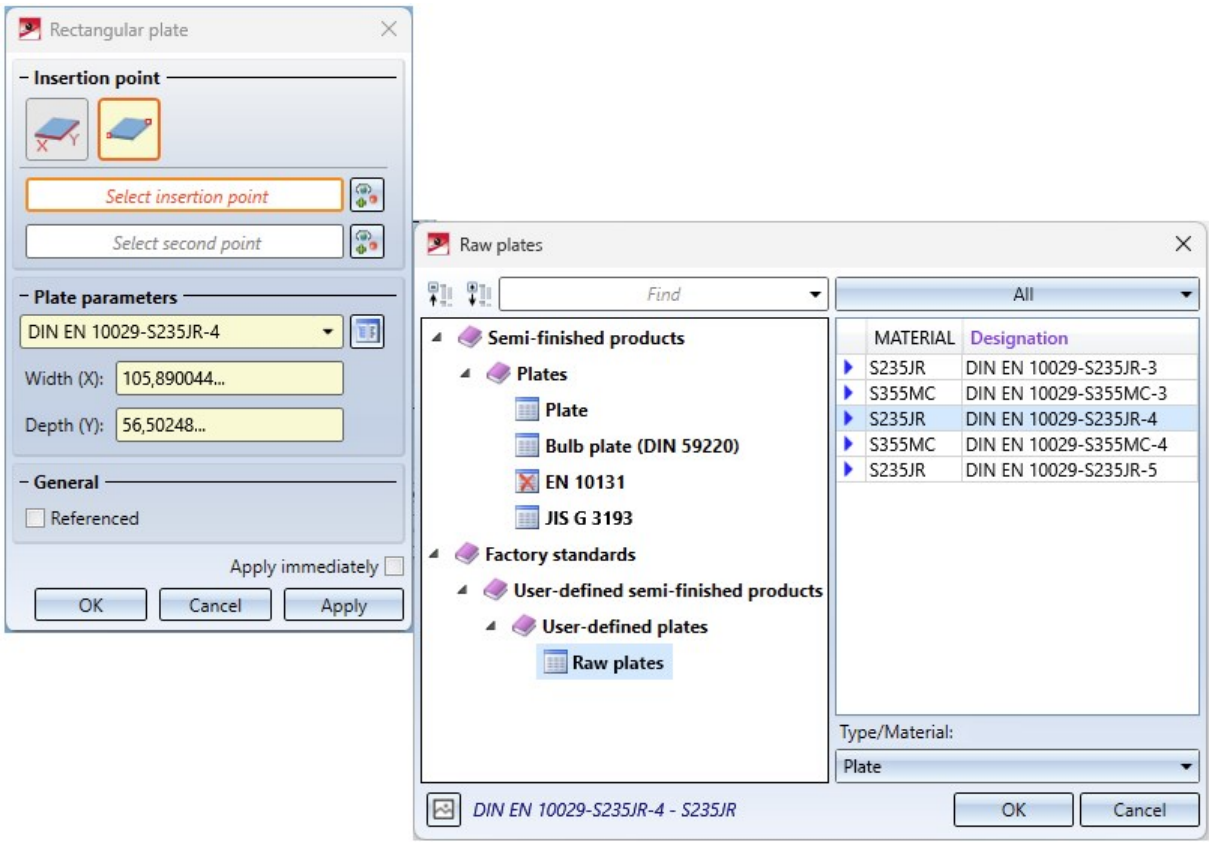


	ID	MOD	STATUS	Designation	ARTICLE	Size	MATERIAL	OBERFL	TYPE
1	1			TEROSON FO 2 SK1 150x0.3mm	2919061	150x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
2	2			TEROSON FO 2 SK1 200x0.3mm	2919062	200x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
3	4			TEROSON FO 2 SK1 300x0.3mm	2919064	300x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
4	3			TEROSON FO 2 SK1 250x0.3mm	2919065	250x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
5	5			TEROSON FO 2 SK1 350x0.3mm	2919066	350x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
6	6			TEROSON FO 2 SK1 400x0.3mm	2919067	400x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
7	7			TEROSON FO 2 SK1 500x0.3mm	2919068	500x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
8	8			TEROSON FO 2 SK2 150x0.3mm	2919069	150x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
9	9			TEROSON FO 2 SK2 200x0.3mm	2919070	200x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
10	10			TEROSON FO 2 SK2 250x0.3mm	2919081	250x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
11	11			TEROSON FO 2 SK2 300x0.3mm	2919082	300x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
12	12			TEROSON FO 2 SK2 350x0.3mm	2919083	350x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
13	13			TEROSON FO 2 SK2 400x0.3mm	2919084	400x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
14	14			TEROSON FO 2 SK2 500x0.3mm	2919085	500x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion

User-defined tables with Steel Engineering plates

At **Factory standards > User-defined semi-finished products > User-defined plates** you now have the possibility to create tables with your own steel plates. These tables are then also offered for selection via the function **Steel Engineering > Plate, new > Rectangular plate**  .





Simply copy a suitable table from the catalogue **Semi-finished products > Plates** into the catalogue **Factory standards > User-defined semi-finished products > User-defined plates** and edit it.

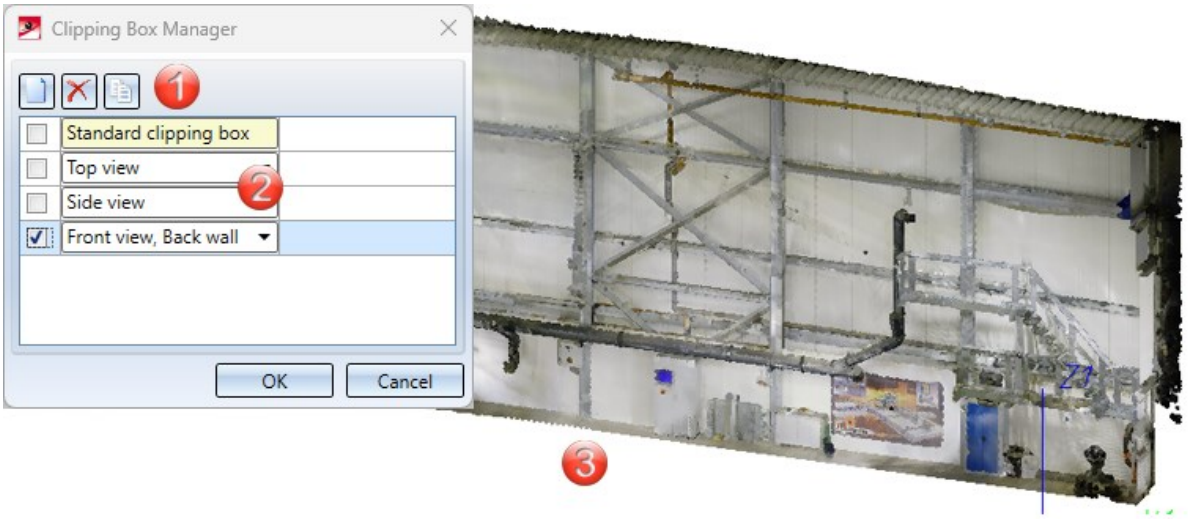
Point clouds

Major Release 2024 (V 2900)

Clipping Box Manager

The new **Clipping Box Manager**  makes it easier to work with different views or clipping boxes of the point cloud.

- You can switch between different clippings in the **3-D model view** to see the appropriate geometry for modelling.
- You can create clippings of the point cloud for the different views of a **Sheet view** to select the appropriate representation for the detail views.

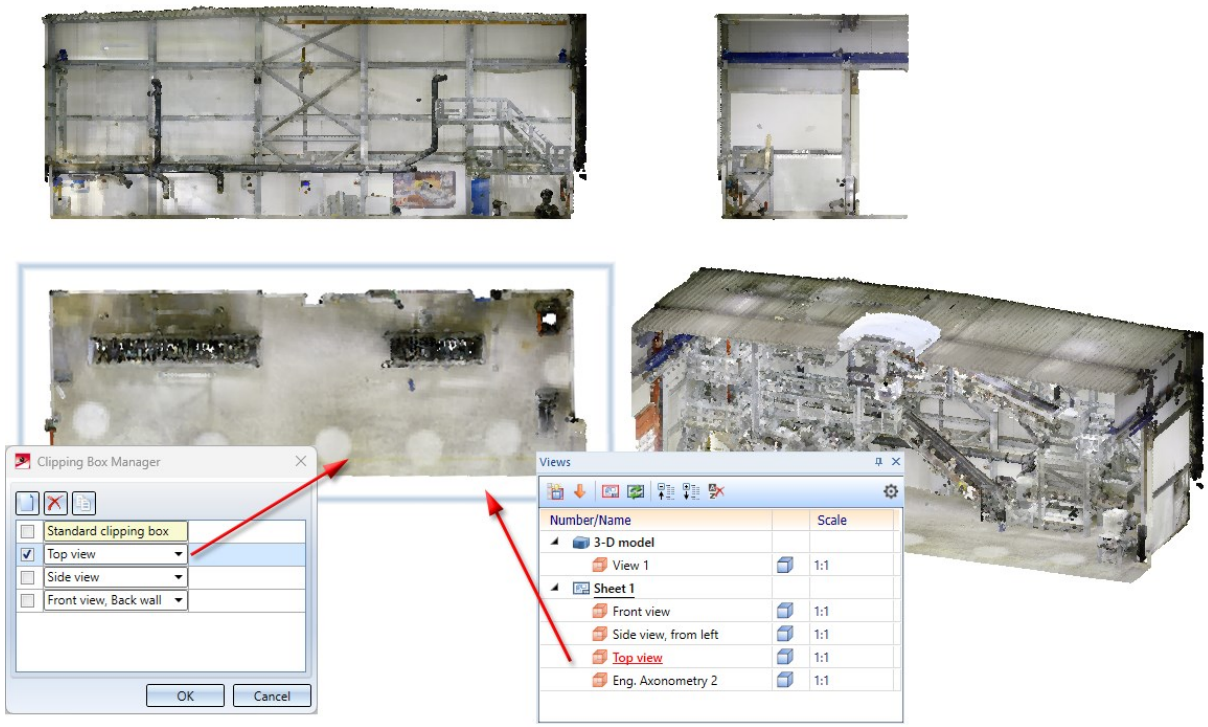


(1) Icons for creating and deleting clippings of the point cloud
(2) List of the different views with differently clipped point cloud
(3) Active clipping **Front view, Back wall** in the 3-D model view

Use the functions in the Clipping Box Manager to create, copy and delete views from the point cloud. The function **New**  derives a new view of the complete point cloud without clipping box. If you select **Copy**  , the active view of the point cloud with clipping box and recess is duplicated. The new view is automatically active. If you then exit the manager with **OK**, all edits, e.g. **Edit clipping box**  or **Subtract**  , refer to the active view. It is indicated by an activated checkbox ☑.

The Clipping Box Manager for Sheet views

If you create a sheet with several views, you can also use the **Clipping Box Manager** to assign the different representations of the point cloud to the views. First activate a sheet view, e.g. **Top view**, and then call up the **Clipping Box Manager**. Now you can create a new view or select an existing view of the clipped point cloud. The sheet views of the point cloud are only displayed in shaded mode.



Sheet with 4 different views in which the point cloud has a different clipping box and subtraction each time.

Feature Technology

Service Pack 1 2024 (V 2901)

Naming of feature functions

The function names of the feature formulas have been revised and standardized. A list of the current functions can be found here.

Functions in existing drawings that were replaced with HiCAD 2024 SP1 will continue to be evaluated.

For a miter cut between adjacent beams/profiles, a separate feature is created for each part. In this case, the two features are linked. This means that if changes are made to one of the features, the other feature is automatically changed or deleted accordingly.

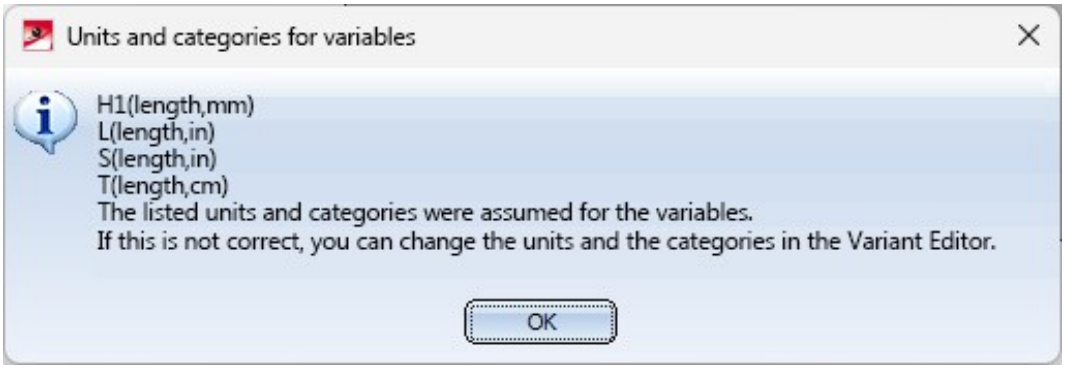
Major Release 2024 (V 2900)

Feature for part and sheet creation

From HiCAD 2024 onwards, the checkbox **Feature** is no longer available in the function dialogues for the creation of parts and sheets. This means that a corresponding feature is now always created during generation.

Create feature variant

When creating a **Feature variant** (as a VAA file), the assigned units are now taken into account and displayed for information when saving the variant with the **Create feature variant**  function. You can then use the Variant Editor to change the units in the VAA file.



HCM

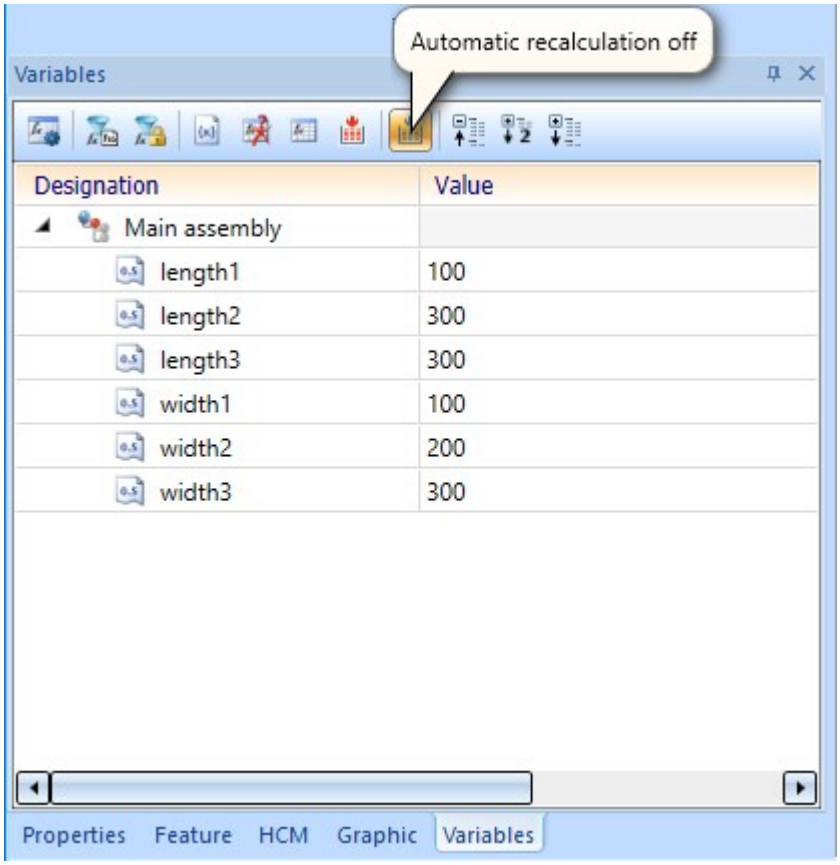
Major Release 2024 (V 2900)

If you deactivate the HCM constraint for a sketch, this is indicated in the HCM window of the ICN. Click on the  symbol to activate the HCM. If possible, HCM constraints are automatically assigned for the following sketches.

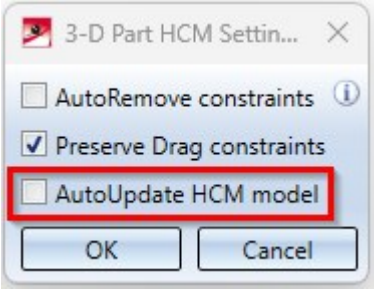
HCM update when variables are changed

If the automatic recalculation (in the **Variables** window of the ICN) is deactivated when variables are changed, the HCM model is not updated either. Previously, the HCM was always calculated. The automatic refers only to actions within the variable window. The rest of HiCAD remains unaffected. For a HiCAD-wide, automatic recalculation of the HCM, the option **AutoUpdate HCM model** (Ribbon: 3-D Standard > Function group: HCM > Tools > Settings) must still be activated.

This allows several variables to be changed, which can then be updated together later.



The variables of an assembly



Settings for the 3-D Part HCM.

Configuration Management

Service Pack 1 2024 (V 2901)

Time threshold for AutoQuickView

In the Configuration Editor, it is now possible to set a time threshold for the **AutoQuickView** display of hidden line or glass model calculations. If the threshold value is exceeded, the calculation is cancelled and the view is displayed in QuickView. Subsequent calculations then no longer start a HiddenLine or Glass model calculation, as the view is already in QuickView.

The setting is made at **System settings > Visualisation > Views > Time threshold for AutoQuickView** . The default setting for the time threshold is 0 seconds. This means that the AutoQuickView behaves in the same way as before and is applied to every Hidden line and Glass model calculation. In workshop drawings, a value of 0.1 to 0.5 seconds can cause the views of the individual items to be displayed "Exact" and only "large" views of the entire drawing to appear in the QuickView.

SpaceMouse®

There is a new setting in Configuration Management under System settings > Miscellaneous: **Exit SpaceMouse mode by moving the mouse**. If this setting is activated, SpaceMouse mode must be ended by a mouse action.

Referencing

Update identical parts

The parameters **Automatically update referenced parts after each change** and **Update identical parts of referenced parts before saving** have been removed in Configuration Editor at **System settings > Referencing**, as from SP1 all identical parts in the current drawing are automatically updated when externally or internally referenced parts are changed.

Referenced assemblies with referenced parts

When saving drawings with changed referenced parts, you can determine the procedure for the referenced assembly that contains the referenced part. In the Configuration Editor at **System settings > Referencing > Saving changed assemblies**, set the default setting for saving referenced assemblies. The default setting is **Only structurally changed assemblies**. If you select a different setting during a current HiCAD session, the setting from the Configuration Editor will be used again in the next session.

Additional column in the Packaging dialogue

When choosing **Profile installation > New/Change > Packaging**  , there is now an additional column for a user-specific attribute for the **Packaging** dialogue window. This attribute must be set in the Configuration Editor at **Profile Installation > Packaging > User-specific attribute**.

Development attributes for Steel Engineering plates

If the settings

- Surface area calculated from development contour (§SOC)
- Rectangular surface area of development (§S2D)

are activated in the Configuration Editor at **Modelling > Part properties > Sheet Metal**, then not only Sheet Metal parts will be calculated, but also Steel Engineering plates from HiCAD 2024 SP1 onwards.

Drawing Management

Documents for general documents

There are two new settings in Configuration Editor at **PDM > Drawing Management > External production documents**:

- Creation of external documents
  This parameter determines whether external documents should only be created for the active sheet or for all sheets. This is possible as of HiCAD SP1. The default setting is Active sheet.
- HELiOS attribute for HiCAD sheet names
  Here you specify which HELiOS attribute the HiCAD sheet name should be assigned to. The HELiOS attribute BENENNUNG (Designation) is preset.

Managing general 3-D parts via part filter

In the Configuration Editor at **PDM > Drawing Management**, you can set whether general 3-D parts should also be taken into account when managing drawings.

As of SP1, the new **Via filter** option is available for the **Manage general 3-D parts** parameter. With this setting, only the general 3-D parts that meet the part filters defined in the Favourites file **Steel Engineering > Drawing Management > General parts** (BIM-3DPartFilter.xml) are taken into account.

Plant Engineering

Pipe length check no longer as macro

In HiCAD 2024 SP1 the pipe length check can be carried out together with the nominal width check and the structure check in a check routine during loading and saving. For this purpose, the setting **Checks the entire drawing** (Plant Engineering > Plant Engineering drawing check) must be activated in configuration management. By default, the check is switched off.

Insert plane flange

In the Configuration Editor at **Plant Engineering > Layout plan** you can define a distance between a straight pipe and the plane flange. Use the parameter **Insert plane flange, with projection**.

Settings for down-grade symbols

The settings for the **Down-grade symbols** have been changed. You can now select the unit for displaying the down-grade symbol in the Configuration Editor. You can also set the number of decimal places and the distance between the down-grade symbol and text.

Interfaces

It is now possible to specify in the configuration editor that when opening foreign formats (e.g. STEP) via Drag & Drop or double-click (in Windows Explorer), a dialogue window is no longer displayed to make changes. To do this, use the parameter **Import files directly during Drag & Drop** at **Interfaces > Import**.

Major Release 2024 (V 2900)

Part properties

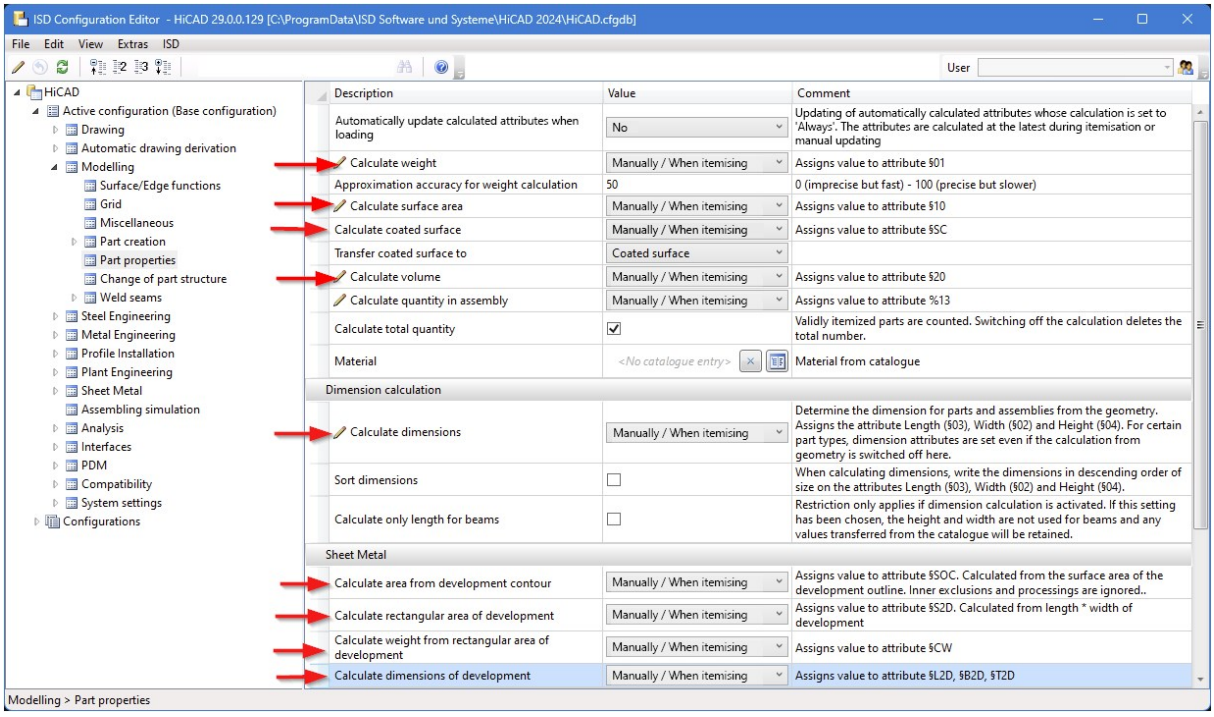
Automatically calculated attributes

In the Configuration Editor, at **Modelling > Part properties**, you can now determine the time of the calculation for attributes that are set to **Always**, with the setting **Automatically update calculated attributes when loading**. With the default setting **No**, the attributes are not updated during loading. The attributes are recalculated in HiCAD if you change the part, itemise it or recalculate it with the function **Update part attributes**.

If you include an attribute that has not yet been calculated, e.g. the weight, in the calculation and have changed the default setting, the calculation is carried out for all parts when the drawing is loaded. This can lead to waiting times.

Parameter configuration

The default settings of the standard template for **Steel / Metal Engineering** have changed with HiCAD 2024. This affects the settings at **Modelling > Part properties** in the Configuration Editor.



The default setting of the parameters indicated in the image below was previously **Always**.

The parameter configuration **Steel Engineering** can be selected either during installation or subsequently with the tool **ParKonfigComp.exe** (or ParKonfigUser.exe).

CFGDBTool.exe

If you want to transfer your individual settings from the HICAD.CFGDB file of an older version into the Configuration Editor of the new version, the CfgDbTool.exe program is available in the HiCAD EXE directory. This tool is now available in all languages supported by HiCAD.

Intersections in the Hidden Line representation

Up to now, only intersections (or collisions) found between straight lines and planes or those involving circles and cylinders were taken into account in the HiddenLine calculation. From HiCAD 2024, all collisions with analytical curves/surfaces or with Nurbs curves/surfaces are now also taken into account.

What exactly is taken into account can be defined in Configuration Management under **System settings > Visualisation > Views > Hidden Line > Collision check**.

Part master display by double click

Until now, double-clicking the left mouse button on a part in the drawing or in the ICN called up the **Part attributes** dialogue window. In the Configuration Editor at **System settings > HELiOS** you can now set whether the part's article master should be displayed alternatively. To do this, activate the setting **When double-clicking on a part, display article master instead of part attribute mask**. If the clicked part does not have an article master, the **Part attributes** dialogue window will be automatically displayed.

Magnetic snap-in when moving views

The alignment of views to each other or to other elements (e.g. drawing frames, title block frames and bills of materials) of a drawing sheet has been facilitated by "magnetic" snap-in. As soon as a constellation is created during dynamic shifting of a view that facilitates alignment, graphic auxiliary elements (e.g. distance arrows, alignment lines) are displayed.

If no auxiliary elements are to be displayed, deactivate the setting **Allow alignment based on equal distance** at **System settings > Visualisation > Views > Magnetic snap-in**.

Negative and positive position in identical part search

The negative and positive position of the profiles is a distinguishing criterion for the identical part search in the dispatch itemisation. For this purpose, the new attribute DWF_NEG_INSTALL has been created as a distinguishing criterion. In configuration management, it is entered as a distinguishing criterion for the dispatch itemisation at **Profile installation > Dispatch itemisation > Integer attributes** and is evaluated if the setting **Carry out dispatch itemisation** is also activated here.

Processing external drawings

As the processing of external drawings can lead to problems with the automatic drawing derivation, a processing lock has been built into HiCAD. If you still want to allow manual changes in external drawings, activate the parameter **Allow processings in external drawings**. You will find the parameter under **Automatic drawing derivation** and then **Production drawing**.

Processing sheet metal developments in production drawings

In drawing management, processing (e.g. applying fillets to edges) of developments in the production drawing was previously prevented. As of HiCAD 2024, the new parameter **Allow processing of sheet metal developments** is available in the Configuration Editor at **PDM > Drawing derivation > Production drawing**. The ISD default setting is **No**. With **Yes**, it is possible to process and save the development in already created drawings without marking the Sheet Metal part as changed.

If you allow processing of developments in the production drawing, no more automatic STEP, DXF data can be created for Sheet Metal parts, because the CAM data is generated directly from the Sheet Metal part.

Generate pipe spool drawing from sheet view

If pipe spool drawings are created in a sheet area of the active design and then a new pipe spool drawing is created from this sheet view, then as of HiCAD 2024 exactly those parts are taken into account in the new/updated pipe spool drawing that were also visible in the original sheet view. This means that in this case you will no longer be prompted to select the parts for the pipe spool drawing.

If, on the other hand, the pipe spool drawing is generated from the model view, then the behavior does not change and you are prompted to select the parts as before. Unless you have deactivated the checkbox **Part selection before displaying pipe spool drawing dialogue** in the configuration management under **Plant Engineering > Isometry and Pipe Spool Drawing**.

Automatic BOMs for itemised source models

Excel BOMs can now be created and managed automatically for model drawings that are itemised source models. For this purpose, the settings in the Configuration Editor at **PDM > Drawing Management > External production documents** have been extended by:

- **Create BOMs**: With this setting you determine when Excel BOMs are to be created for selected drawings.
- **List of model drawings with external BOM**: With this setting you determine for which drawings external BOMs are to be created and managed.

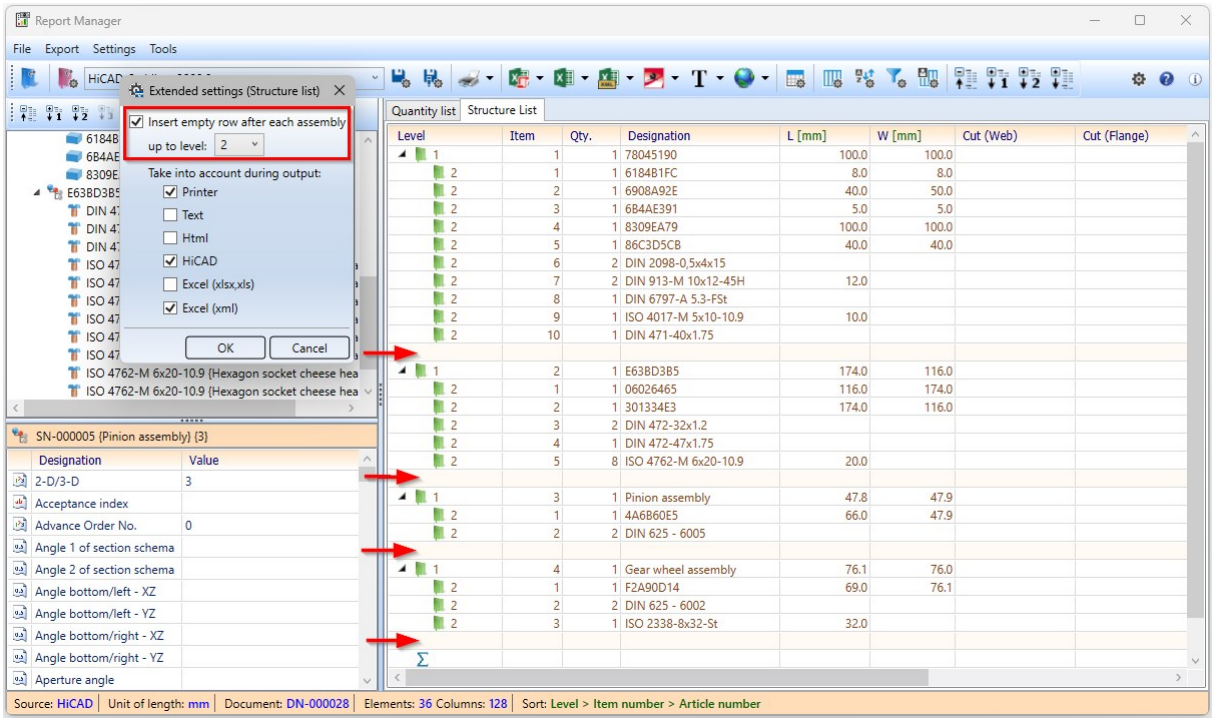
Bill of Materials / Report Manager

Major Release 2024 (V 2900)

General adjustments

Empty rows in the structure list

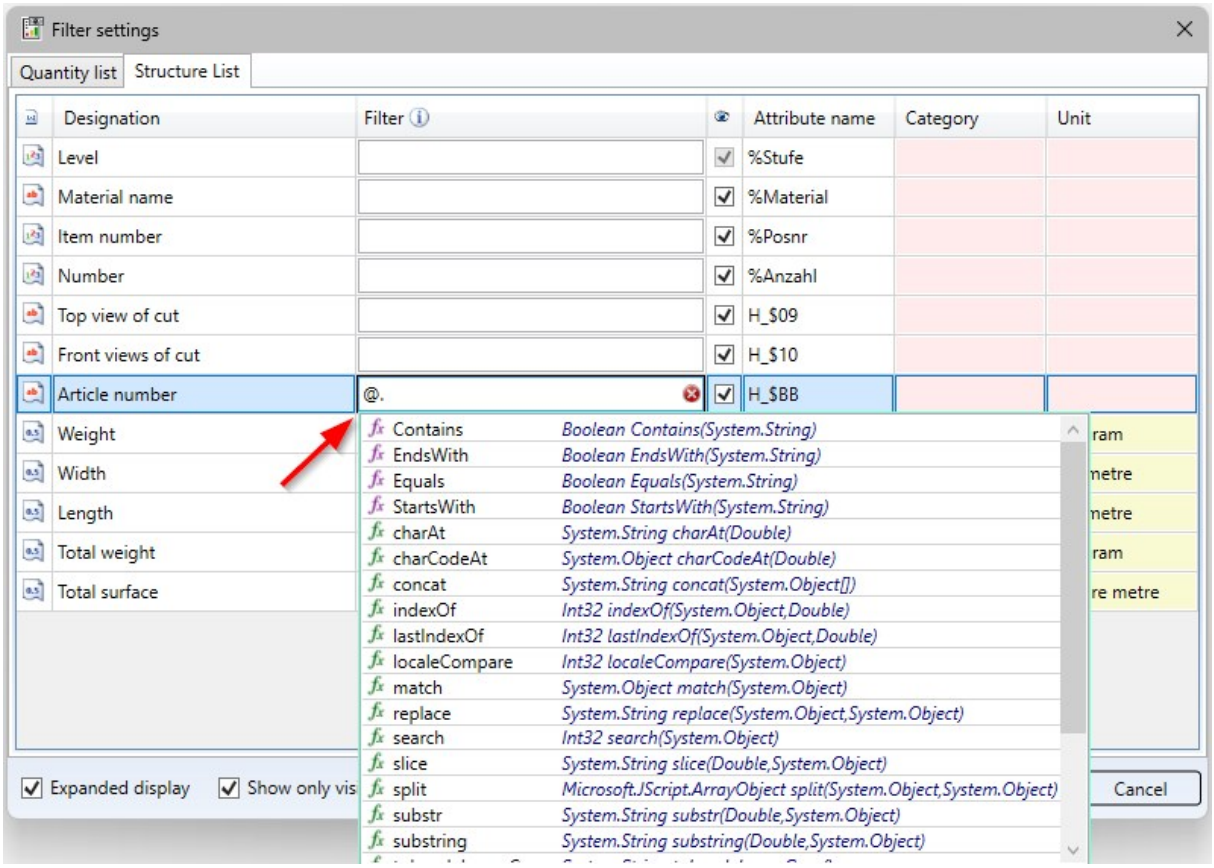
With the new function **Extended settings (Structure list)...** (under Settings) you insert empty rows in the structure list. You can determine the level up to which empty rows are to be inserted. In the output, the empty rows are taken into account in all selected formats.



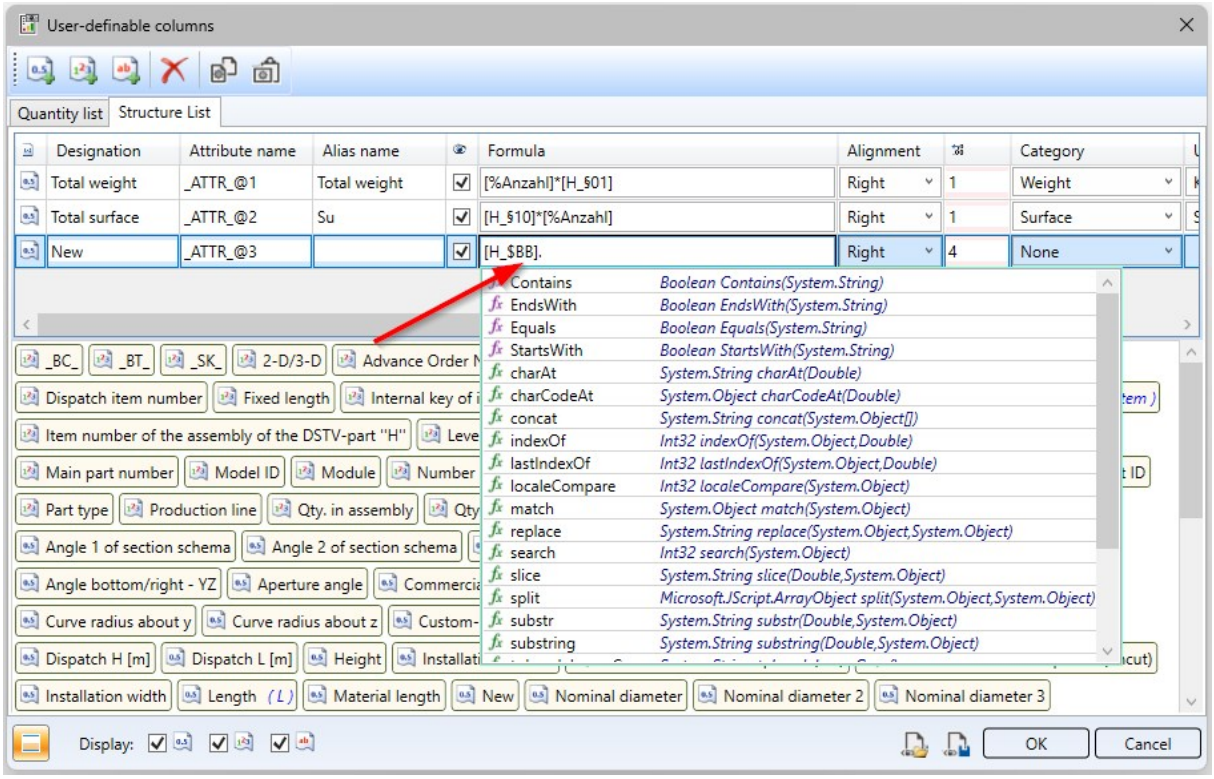
Empty row after each 1st and 2nd level of structure list

Auto-completion

The auto-completion of formulas in the **Filter settings** and in the **User-definable columns** is displayed after entering a point. The complete list appears when pressing the key combination **ALT + space bar**.



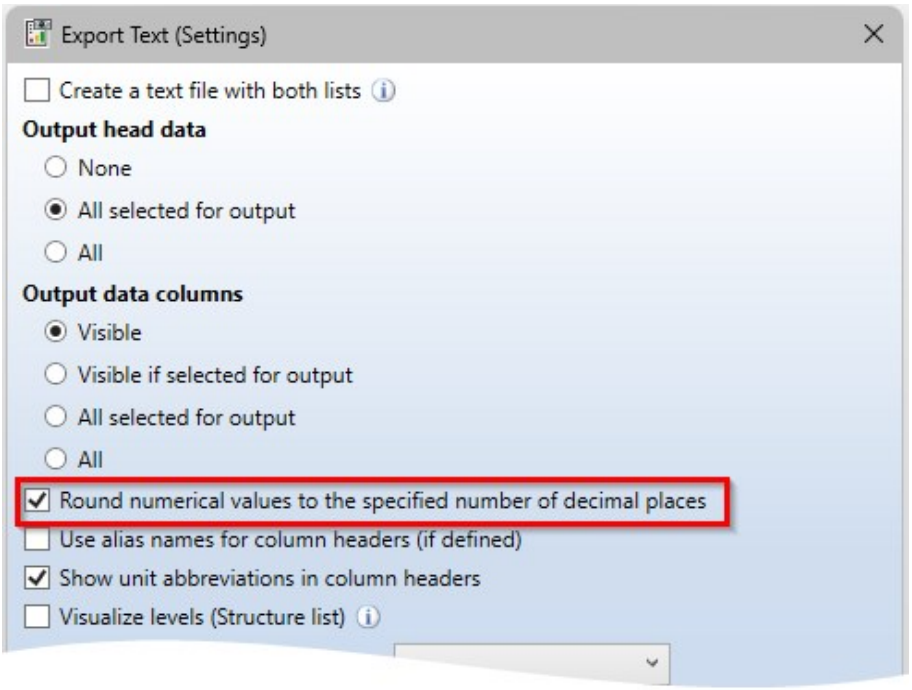
Placeholder and dot for displaying conditions



Dot for selection for formula creation

Round decimal places

In the settings for the output of XML, text and HTML BOMs, the option **Round numerical values to the specified number of decimal places** is now available. This means that if you have specified a number of decimal places for the **Column** in the Column settings, the value is rounded to this number when creating the BOM.



Settings for exporting the text BOM

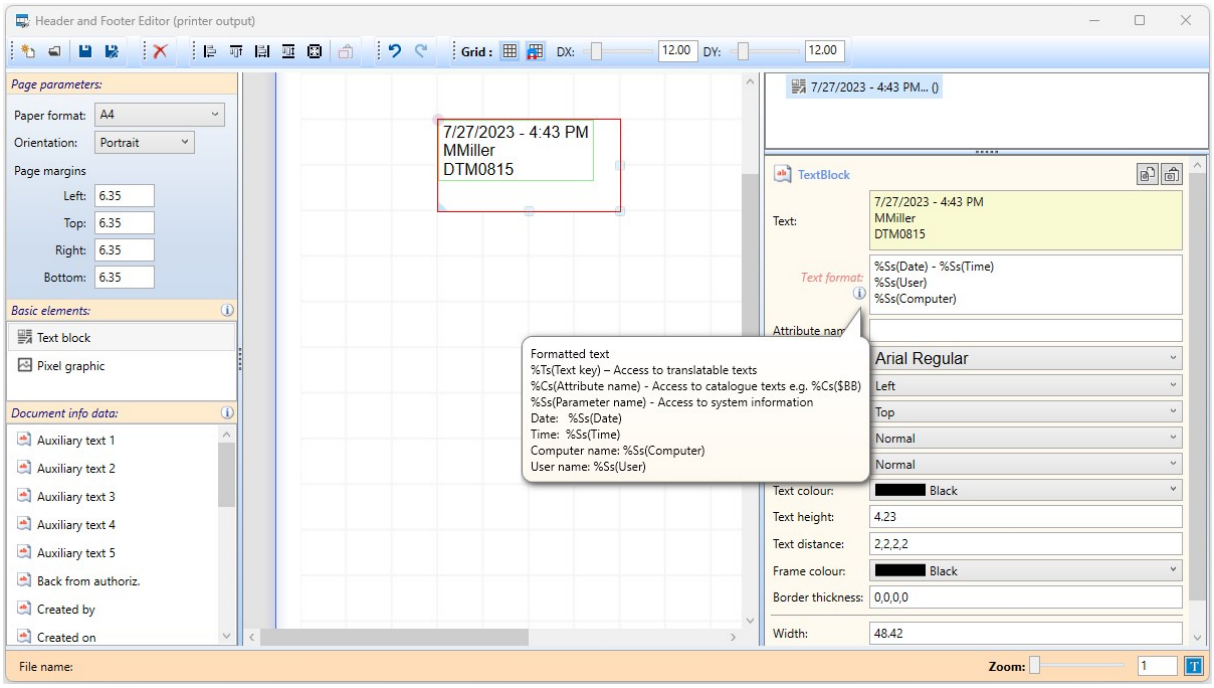
Header and Footer Editor

In the **Header and Footer Editor**, placeholders for date, time, user name and computer name are now supported for printer output.

Input:

- Date: %Ss(Date)
- Time: %Ss(Time)
- User name: %Ss(User)
- Computer name: %Ss(Computer)

Simply drag the **Text block** (in the area: Basic elements) into the **Workspace** and enter the placeholder for accessing the system information in the **Text formatting** field on the right.



New in the Excel BOM templates for Steel Engineering (HiCAD_Stahlbau.DE2900.0.XLSX) is the **Profile structure list**. With this list, parts/profiles of the same type are grouped together and output in a structure list. The profile structure list is a mixture of structure list and total profiles list and is suitable for the combination of Steel Engineering and Profile Installation, e.g. in hall construction.

With the Excel BOM template for Steel Engineering (HiCAD_Stahlbau.DE2900.0.XLSX) you can output images for steel plates on the **Steel plates and metal sheets with image** worksheet from HiCAD 2024.

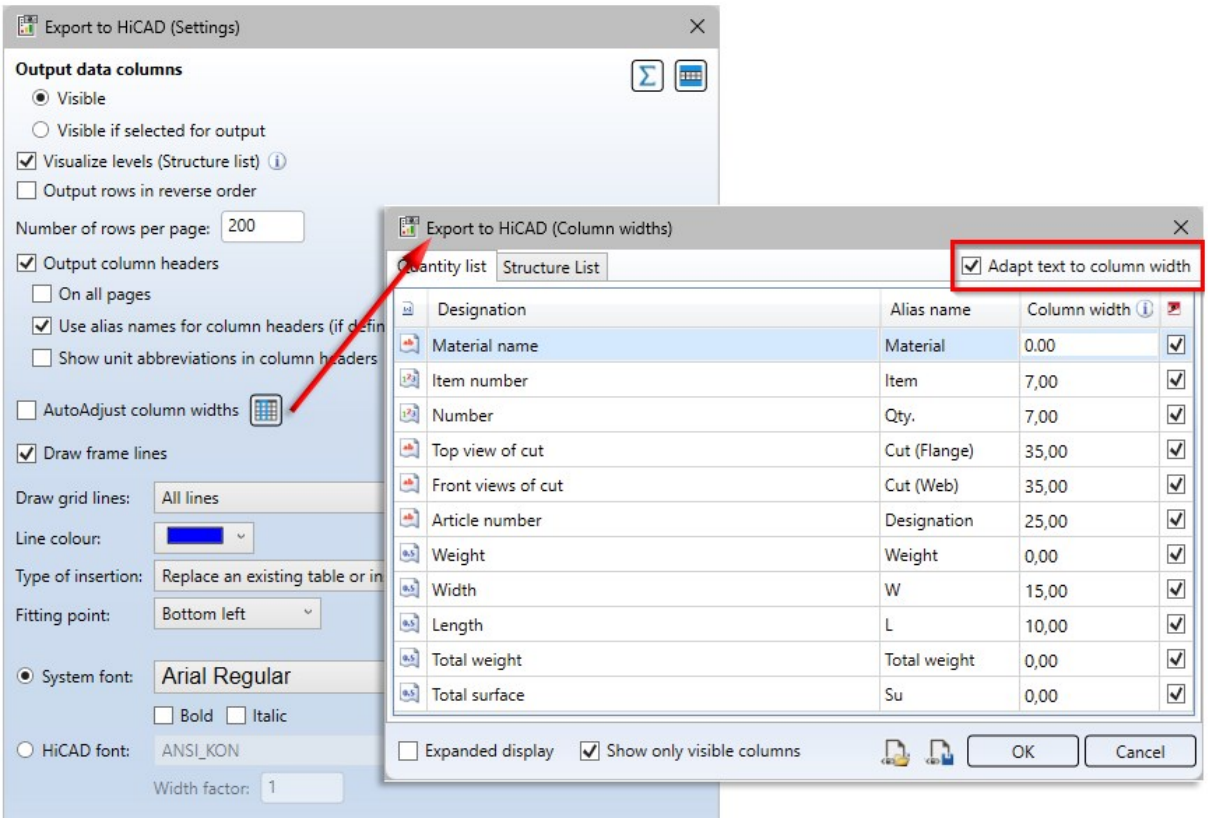
BOMs for HiCAD

Cancelling the insertion of the BOM

In the export settings of BOMs for HiCAD, you can specify the number of lines to be output per table page. The table pages must then be placed individually in the drawing. In HiCAD 2024 you can cancel this process with a right click.

Adjust text lengths

In the settings for exporting BOMs to HiCAD, you now have the option of adjusting texts to the column width.

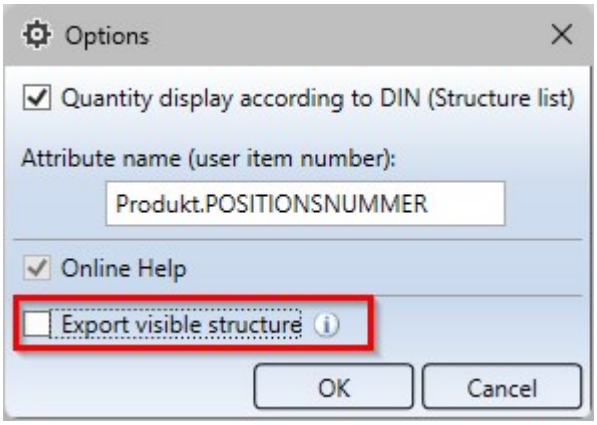


Activate the checkbox ☑ if texts that are too long for the specified column width are to be shortened.

0	1	U 300	15...	100,0	I--------\ 16.4°	I--------I
0	1	U 300	41...	100,0	16.4° /--------\ 28.6°	I--------I
0	1	U 300	24...	100,0	28.6° /--------I	I--------I
0	1	U 300	10...	100,0	I--------I	I--------I
0	1	U 300	15...	100,0	16.4° /--------I	I--------I
0	1	U 300	41...	100,0	28.6° /--------\ 16.4°	I--------I
0	1	U 300	24...	100,0	I--------\ 28.6°	I--------I
0	1	Rohr 48.3x2.6	41...		I--------I	16.4° /--------I
0	1	Rohr 48.3x2.6	96...		I--------X ?	32.9° /--------X ?
0	1	Rohr 48.3x2.6	95...		I--------X ?	I--------X ?
0	1	Rohr 48.3x2.6	41...		I--------I	I--------\ 16.4°
0	1	Rohr 48.3x2.6	96...		I--------X ?	32.9° /--------X ?

Transfer visible structure list

In the **Options** , the new setting **Export visible structure** is now available. By activating this option, the structure list will be handled as with the export in the old Report Manager. This means that the collapsed rows are not taken into account for the export or for the column totals. The option is deactivated in the ISD default setting.



Use in the HDE reports

Within the HDE reports you can specify which RMS file (old Report Manager) should be used. From HELiOS 2024 you can also use the new RM_SETTINGS files and thereby start the new Report Manager (from 2023).

Variant Editor

Major Release 2024 (V 2900)

Files managed by HELiOS

HiCAD cannot determine where the data originates when accessing files for which a document master exists. This can lead to problems when dealing with variants.

For example, a variant could be loaded from the hard disk in the Variant Editor, modified and transferred to the corresponding items in HELiOS with the part data synchronization. In fact, this variant file can be managed by HELiOS e.g. in the Vault Server. So one has not changed the file to which the document master actually points.

This means that when the variant is installed, potentially a different geometry is calculated than expected, because the selected item no longer matches the expression in the VAA file to which the document master points.

Therefore, when a file is opened via the file system, it is checked whether this file is managed by HELiOS. If this is the case, the following message appears:

If you click on **Yes**, the file will be loaded via the document master. This ensures that the file matches the one in the document master.

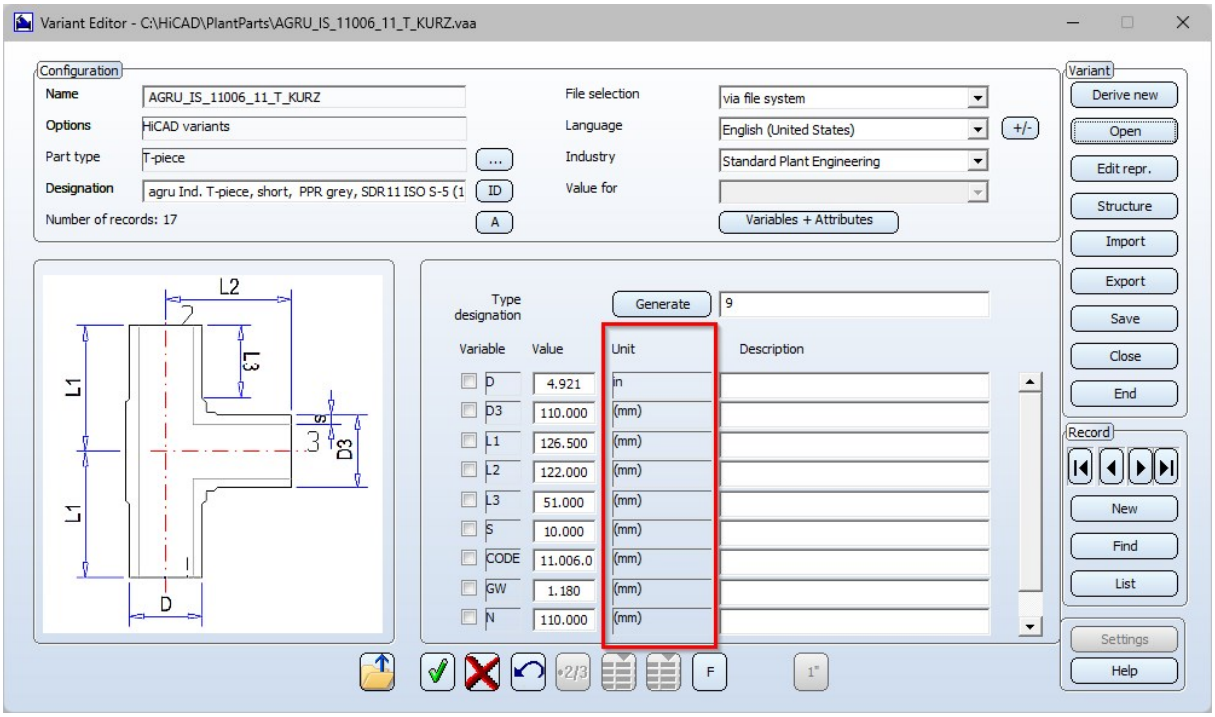
This query also appears when opening takes place via the file system

- in the PAA Editor (AnPaaEdit.exe),
- during part data synchronization (PartDataAutoSync.exe), and
- during the configuration of the HELiOS database (DBPlantDataImport.exe).

Category and Unit

The Variant Editor (VariantenEditor.exe) now also supports working with imperial units.

When opening a variant, the **Unit** column displays whether a variable is assigned and - if so - which unit. The display (mm) means that no unit is assigned.



To assign a unit to a variable or attribute or to change the assignment, click on A for constant attributes or on Variables + Attributes for variables and attributes.

A Constant attributes

Clicking this button lists the constant attributes specified for the current part type.

As of HiCAD 2024, it is possible to define or change the category and unit here for corresponding attributes. The attributes that expect a unitless number are all nominal width attributes and the Pressure attribute. The category **Unitless number** is already assigned to these attributes, a change is not possible here.

To change the assignment of category and unit of an attribute, select the desired entry in the selection box of the corresponding line. Since the category specifies the available units, the category should therefore be selected first.

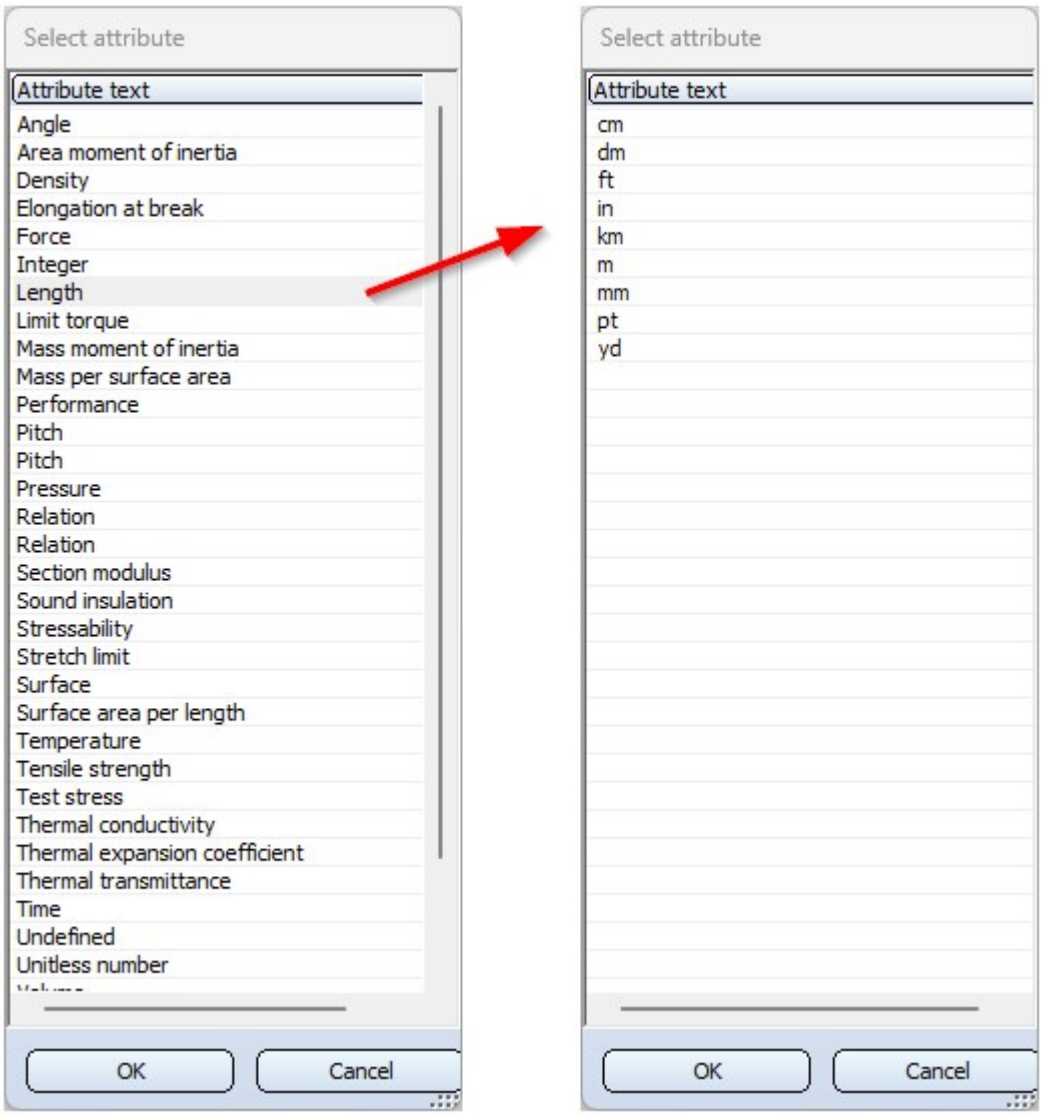
Some attributes are automatically preassigned when the attribute assignment is called up, e.g.:

This means that when you exit the window with OK, the variant has been changed, even if you do not make any changes.

Variables + Attribute

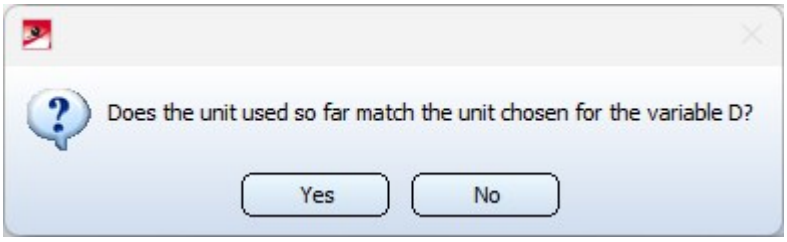
Clicking the Variables + Attributes button lists all variables defined in the variant with their associated data type (Integer, Floating point or String) and HELiOS attribute. .As of HiCAD 2024, the assigned category and unit are also displayed here.

To assign a category and unit to a variable or to change the assignment, click in the **Category** and **Unit** column in the corresponding row. Since the category specifies the available units, the category should be selected first and then the unit.



There are some attributes that are nominal values without a unit and are defined as a Unitless number. Here the assignment is done automatically.

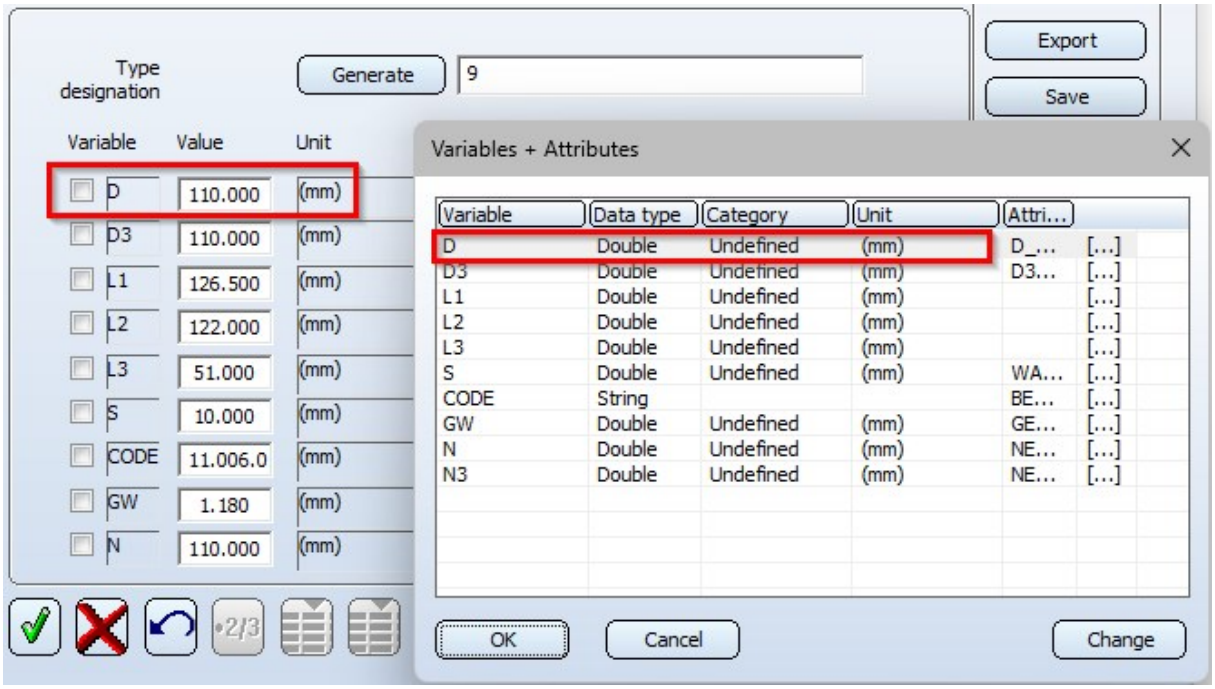
If you are editing variants and assign a unit to a previously unitless variable, then when you exit the **Variables+Attributes** window you will be asked whether the previously used unit matches the newly assigned one.



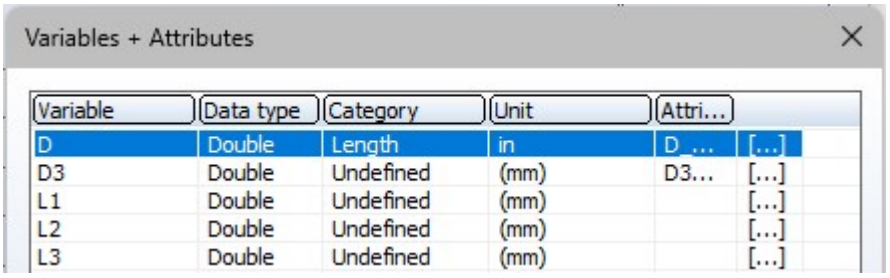
If you choose **Yes**, the category and unit will be assigned to the variable, but without conversion. If you choose **No**, you must select the original unit. Only then will the variable be assigned the category and unit and and the value of the variable be converted to the new unit.

Example:

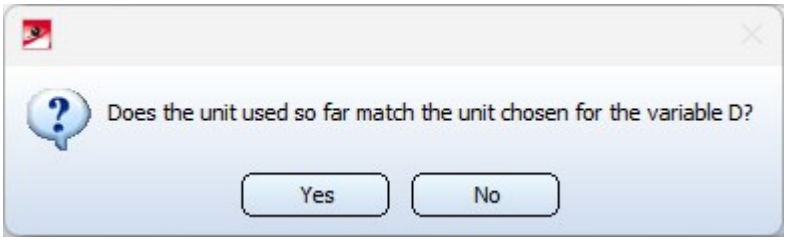
The variable D in the figure has neither a category nor a unit assigned to it, therefore **Undefined** and **(mm)**.



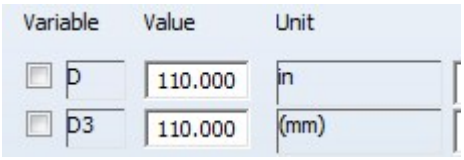
Now change the assignment as shown below:



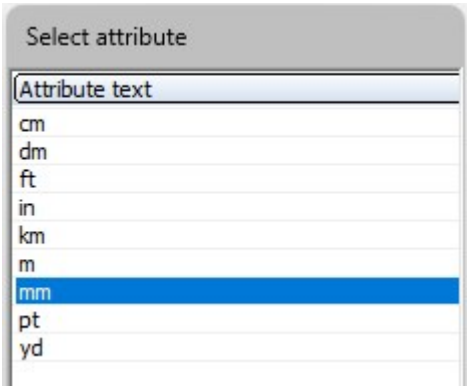
When you exit the dialogue window with **OK**, the following query appears:



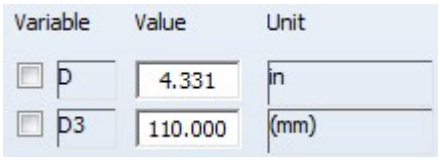
If you choose **Yes**, the category and unit are assigned to variable D, but no conversion takes place.



If you choose **No**, you will be prompted to select the original unit, e.g. mm.



If you confirm the unit with **OK**, the Category and Unit are assigned to variable D here as well, but the previous values are now converted from mm to inch.



Automation

Discontinuation

Discontinuation of the ISD.PDM.API

Before carrying out a HELiOS update for an older HiCAD version, please note that from HELiOS 2022 omwards, the previous ISD.PDM.API will be discontinued and replaced by the new API from Helios.Interface. If you use customisations that use functionalities from the previous ISD.PDM.API, you must update the customisations to the new API before carrying out the HELiOS update. If you use customisations that use functionalities from the HiCAD API, you should ensure that the HiCAD version used is at least version 2502.5 or 2601.1 or newer. If you are unsure whether you are using corresponding adaptations, please talk to your administrator or contact the ISD in case of doubt.

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Cam joints

The HiCAD API now also supports the Cam joint (3-D Standard > Standard Processings > Bore > Cams ). For this there is the new class:

- ISD.CAD.Modifiers.Cam/CamProcessing)

Annotation tags

The HiCAD API provides the following command for the optimised arrangement of Annotation tags  (3-D Dimensioning + Text > Text > LLine > Optimise arrangement):

- LabelCreation.OptimizeArrangement

Executing UI tasks

The HiCAD API provides the ExecuteTask command for executing user interface tasks.

- Example: Context.ExecuteTask("Core.SolidPrimitivesDialog")

Event after drawing derivation

With the HiCAD API you can trigger an event (a function to which you log on) when a drawing sheet is created during drawing derivation. The following command is available for this purpose:

- ISD.CAD.Steel.Drawings.WorkshopDrawings.SheetCreated

Form and positional tolerances

To create the new form and positional tolerances via the API, there is a Create function with which you can load favourites.

- ISD.CAD.Dimensioning.FormPosTolerance.Create()

Accessing fixed view points

In the API, there is now a property (true or false) set or not set for the fixed point of a view. If the fixed point is set, you can access it. This allows you to move views back to their origin that do not have a fixed point.

- View.FixPoint

Interfaces

Service Pack 1 2024 (V 2901)

Update to CADfix 13

With the update to CADfix 13, the following format versions are now available:

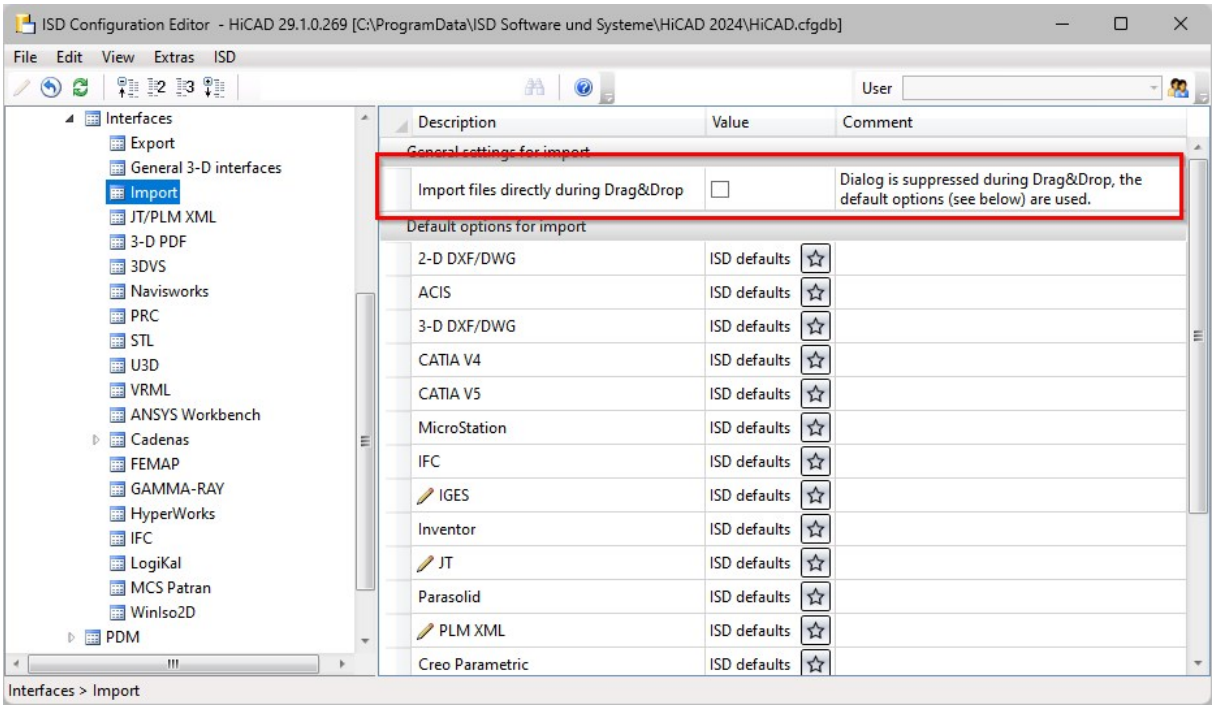
- ACIS R1 - 2023 1.0.0 (R33)
- CATIA - V5-6 R2023
- JT - JTOpen 6.4 - 11.3
- PLM XML - JTOpen 6.4 - 11.3
- Parasolid - 9, 13 – 35
- NX - 1 - 2212 Series
- SOLIDWORKS- 98 – 2023
- SolidEdge - V18 (2006) - 2023

Please note:

In the topic Interfaces Overview you will find extensive information on the import and export of the data formats supported by HiCAD.

Opening foreign formats without dialogue

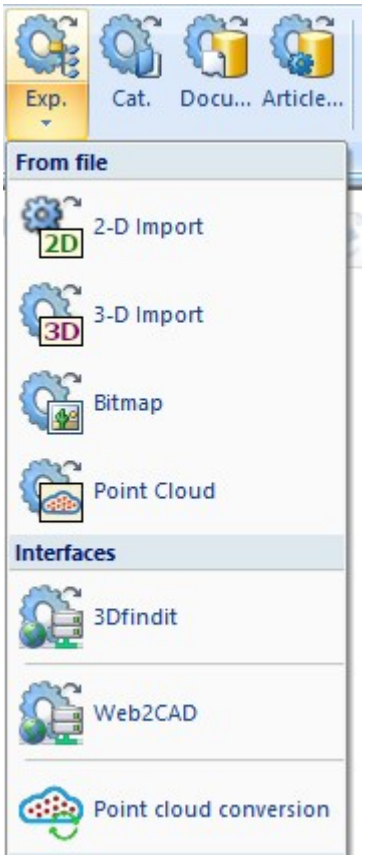
Foreign formats (e.g. STEP) can now also be opened directly via Drag & Drop or double-click in the Windows Explorer without any further dialogue. To do this, the new checkbox **Import files directly during Drag & Drop** must be activated in the Configuration Editor at **Interfaces > Import**.



The default option will then be used.

Combining the function for 3DFindit

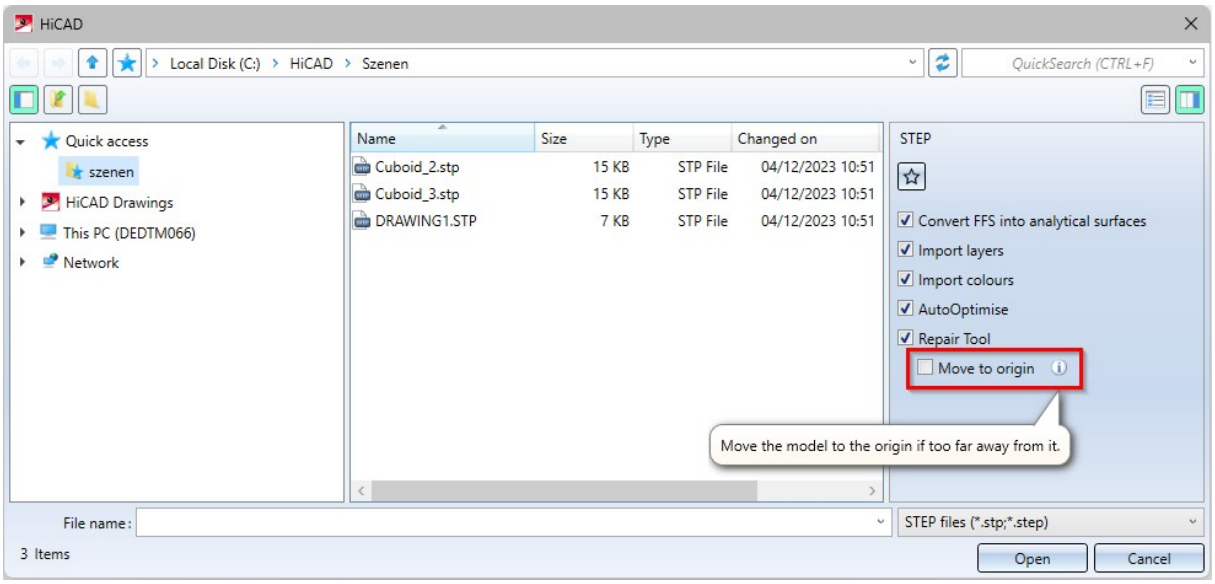
Previously, two functions were available for accessing 3DFindit. In SP1, these functions have been combined. The title bar of the dialogue window now shows whether it is the free basic version or the paid full version.



3-D Import: Move to origin

When importing external data, the problem occasionally arose in the past that data was defined in coordinate systems in which the geometry was far away from the origin. In HiCAD, this led to calculation inaccuracies and inaccurately shaded models.

As of HiCAD 2024 SP1, a new Move to origin checkbox is now available for many file formats when importing with the **Open > 3-D Import** function. This applies, for example, to STEP (only if the Repair Tool is activated), VDAFS, IGES, 3D DXF/DWG, JT/PLM XML and other CAD formats.



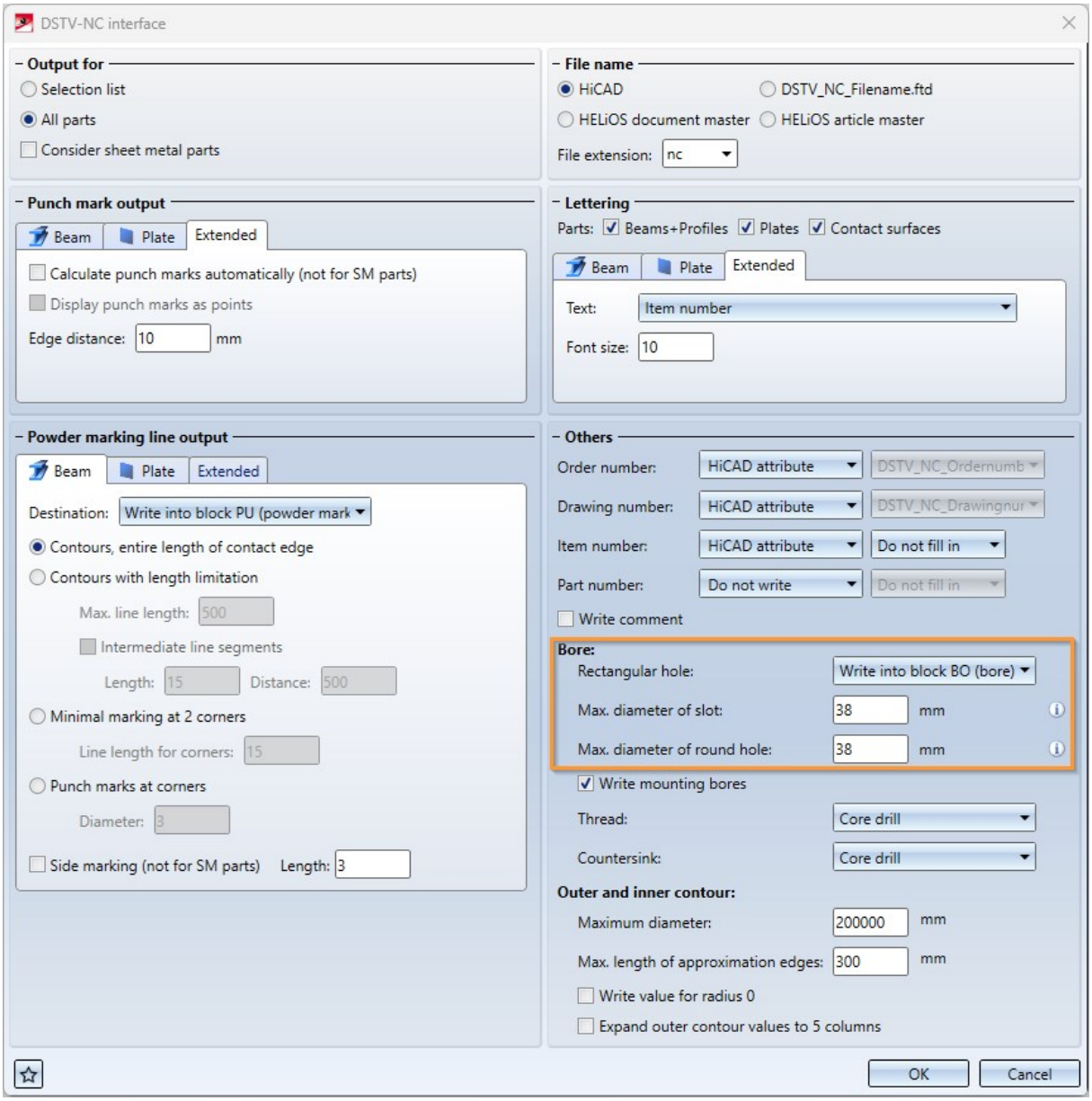
If you activate the checkbox when inserting, the model is automatically positioned near the origin of the coordinates. This also increases the accuracy of the model.

DSTV-NC

The display of bores for the **DSTV-NC export** has been extended as follows:

You can specify a maximum diameter for both slots and round holes.

Slots or round holes smaller than or equal to the diameter defined at this point are then written to the BO block (bore), larger holes are automatically written to the IK block (inner contour).



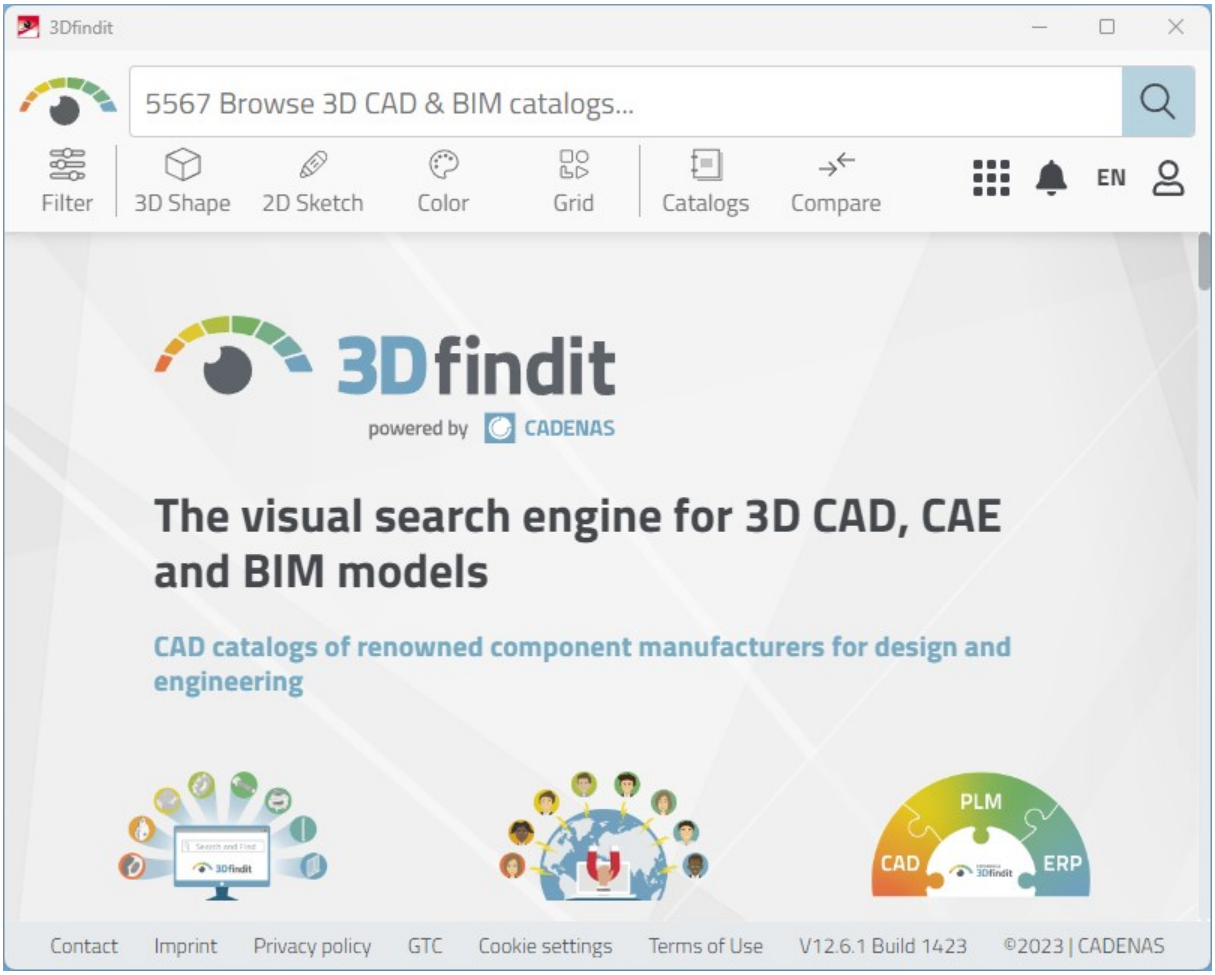
Major Release 2024 (V 2900)

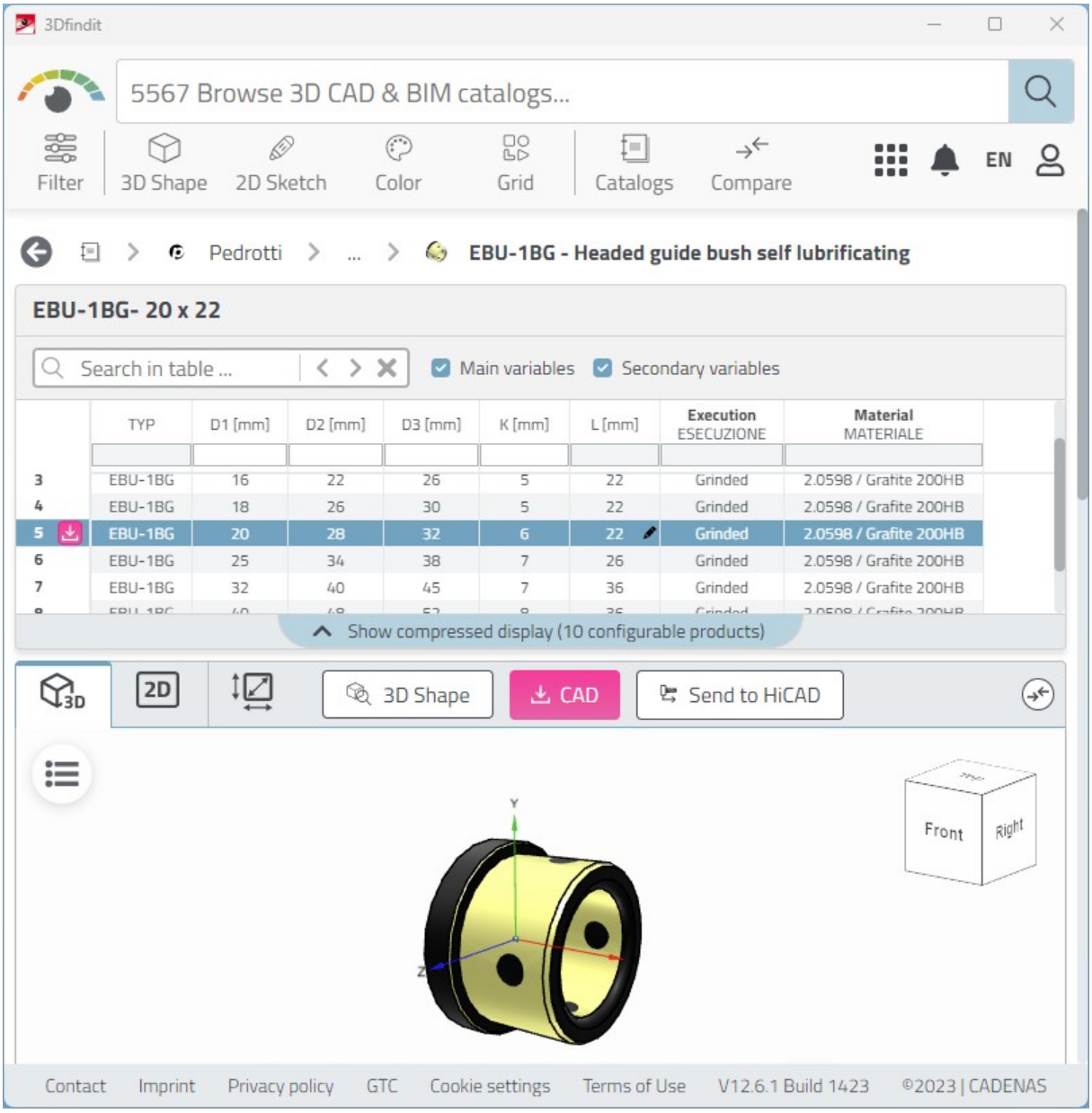
Access to 3Dfindit

3Dfindit by CADENAS is the visual search engine for 3-D CAD, CAE and BIM models. It enables access to CAD catalogues of well-known component manufacturers for design and engineering.

For direct access to the search engine, two new functions are available in HiCAD at Drawing > Insert Part > Exp >...:

- 3Dfindit (Basis)
  The free version offers you access to the available 3D catalogues that have a download contract with CADENAS (as of 06/2023, there are approx. 900 manufacturers).
- 3Dfindit
  The paid version gives you access to the DIN/ISO/EN standards and to the available 3-D catalogues that have a download contract with CADENAS (as of 06/2023 there are approx. 900 manufacturers).





Registration is required for access.

The previous functions

- parts4cad (Basis),
- parts4cad und
- bimcatalogs

are covered by the new functions and are therefore no longer available from HiCAD 2024.

New import formats: MicroStation and Solid Edge

In HiCAD 2024, the following file formats are newly available for import:

- MicroStation:
  Parts and assemblies in the (3-D) geometry format .dgn.
  DGN Versions 7 to 8 are supported.
- Solid Edge:
  Parts and assemblies in the (3-D) geometry formats .par, asm and .psm
  Solid Edge versions V18 (2006) to 2022 are supported.

IFC interface: Basic settings

The IFC export option **Output identical parts as referenced parts** is activated by default from HiCAD Version 2900..

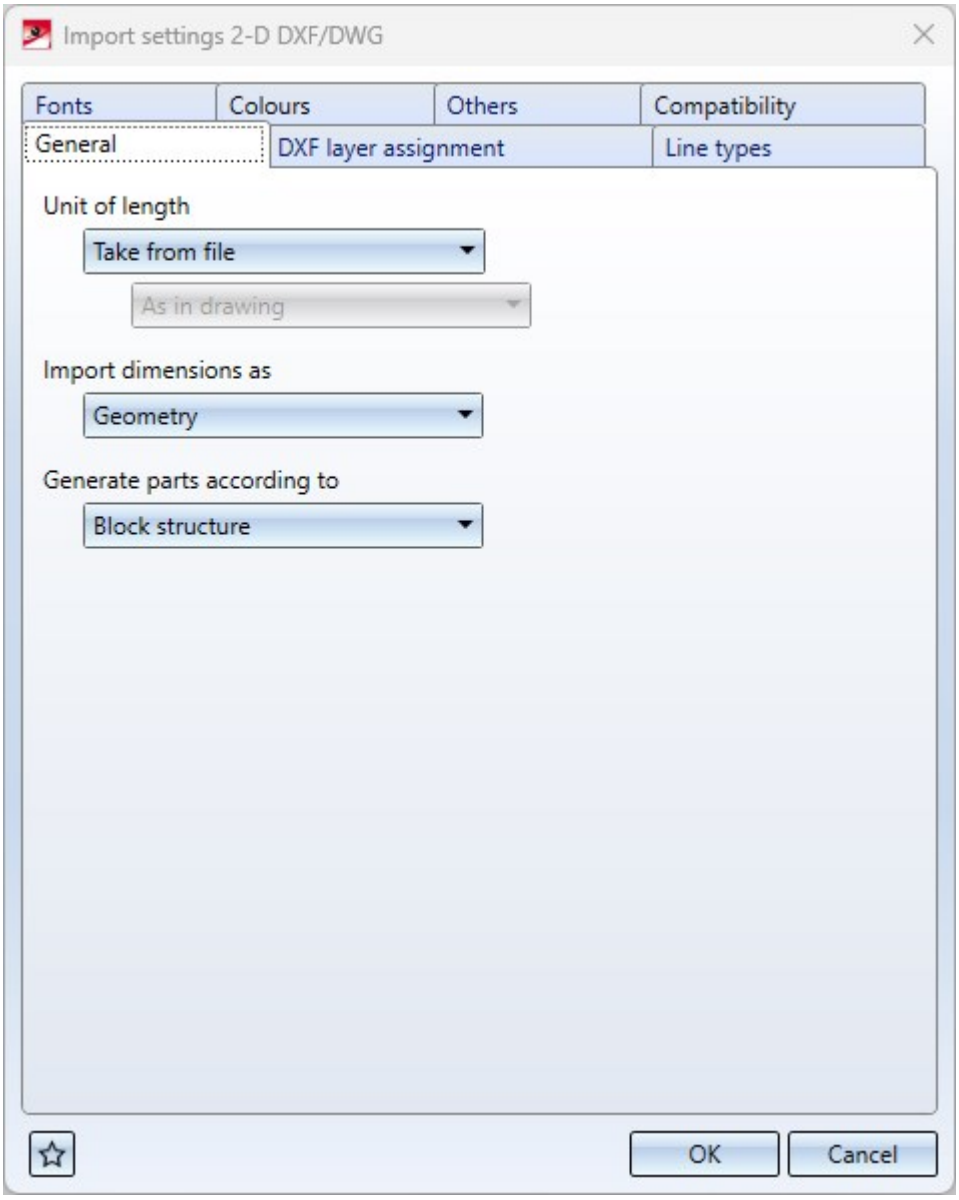
Please note:

Please note that this change applies to new installations of HiCAD. Update installations do not overwrite existing configurations.

2D-DXF/DWG: Improved Import and Export options

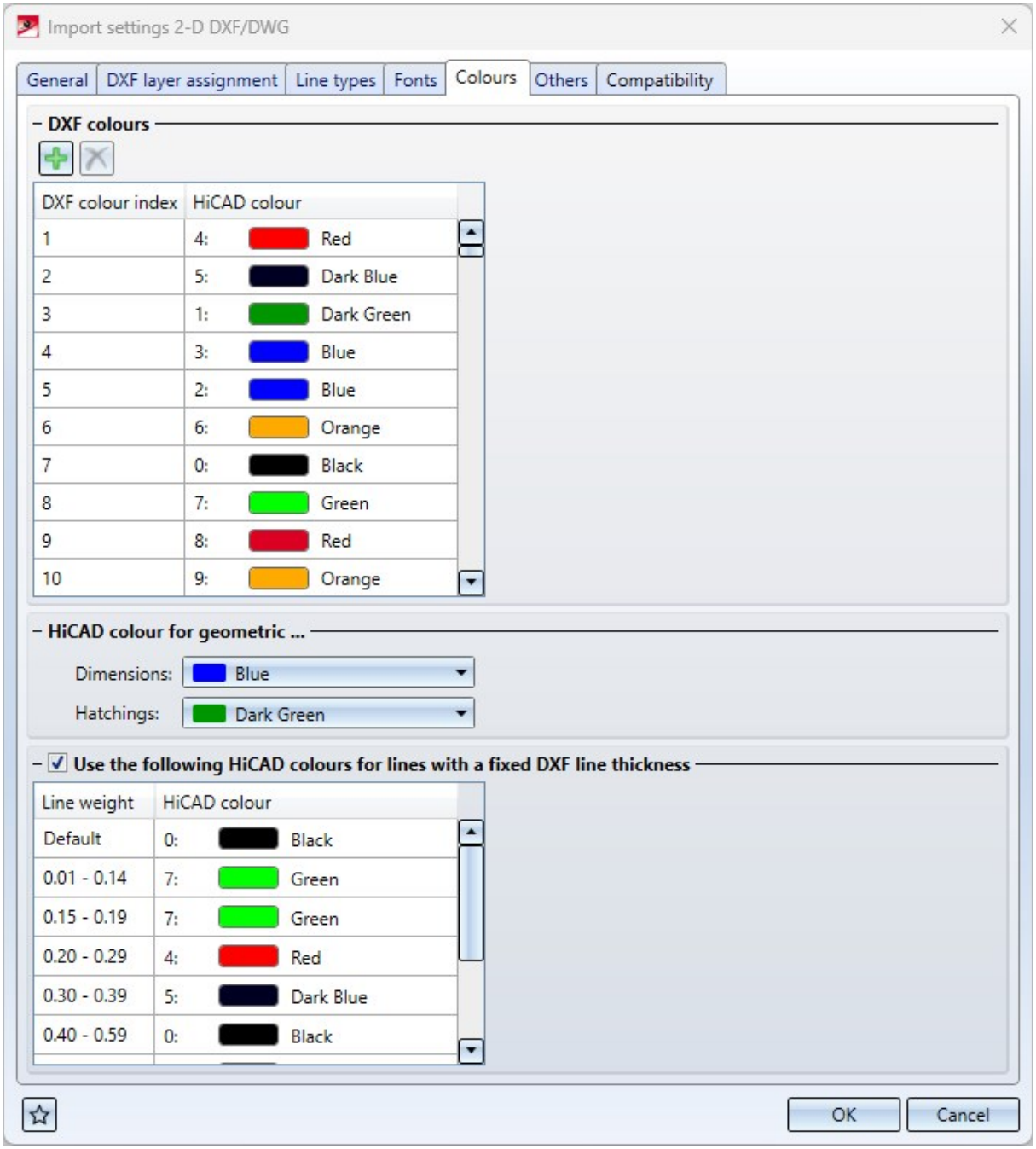
The settings dialogues for both import and export of **2-D DXF/DWG** files have been further improved in HiCAD 2024.

In the menu window of the **Import** settings you will find the new tab subsections **Line types**, **Fonts** and **DXF layer assignment**. The latter replaces the former "Layers" area with an extended range of options.

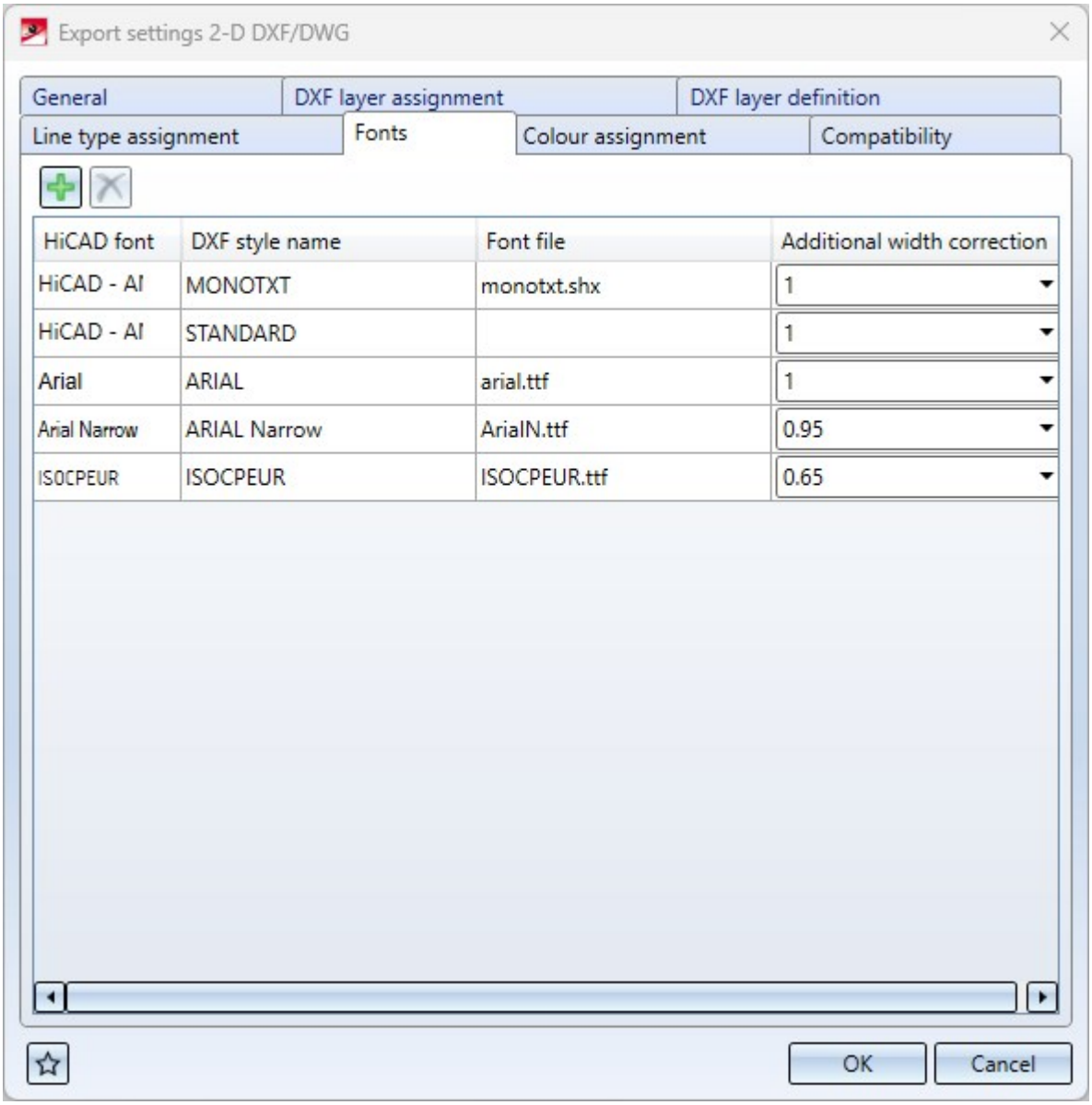


On the **Others** tab you will find a new option for importing **Splines**, which can be imported from the 2-D DXF file to HiCAD as **3-D polyline**, **B-spline** or **B-spline with approximation**.

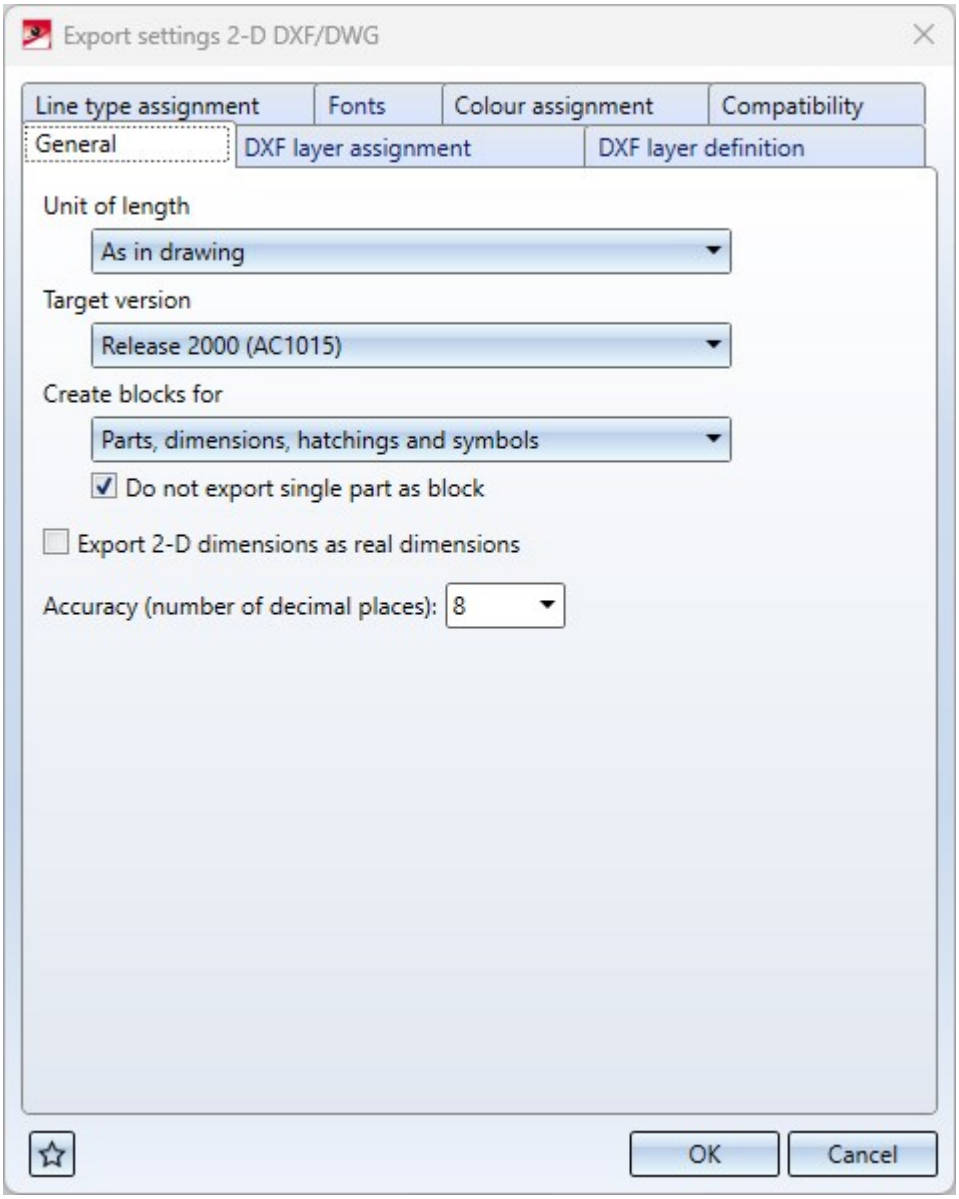
On the **Colours** tab an optional assignment table for lines with fixed DXF line thickness has been added.



In the dialogue window of the **Export** settings the tab **Fonts** has been added, in which HiCAD fonts and DXF styles can be assigned to each other.



On the **General** tab you will find the new checkbox option **Do not export single part as block**. If only one part is exported during an export, this checkbox controls that no "BLOCK" is created in the DXF file for a single part.



Manual editing of keywords in corresponding configuration files, as was the case in older HiCAD versions, is thus no longer necessary.

The configuration file keywords that are thus omitted include:

- For import: LAYER, LTYPE, COLAR, STYLE, SPLIN, COLWE
- For export: FIGB1, STYLE

Please note:

Options available in earlier HiCAD versions via the configuration files **hcadacad** or **acadhcad**, which were not taken over into the settings dialogues in the context of the conversion to the new menu structure, can still be set for compatibility reasons on the **Compatibility** tab in the respective import or export dialogue, in order to continue to guarantee possible behaviour beyond the settings dialogues.

For reasons of clarity, however, these options are only displayed if they differ from the default settings.

Sheet Metal

Service Pack 1 2024 (V 2901)

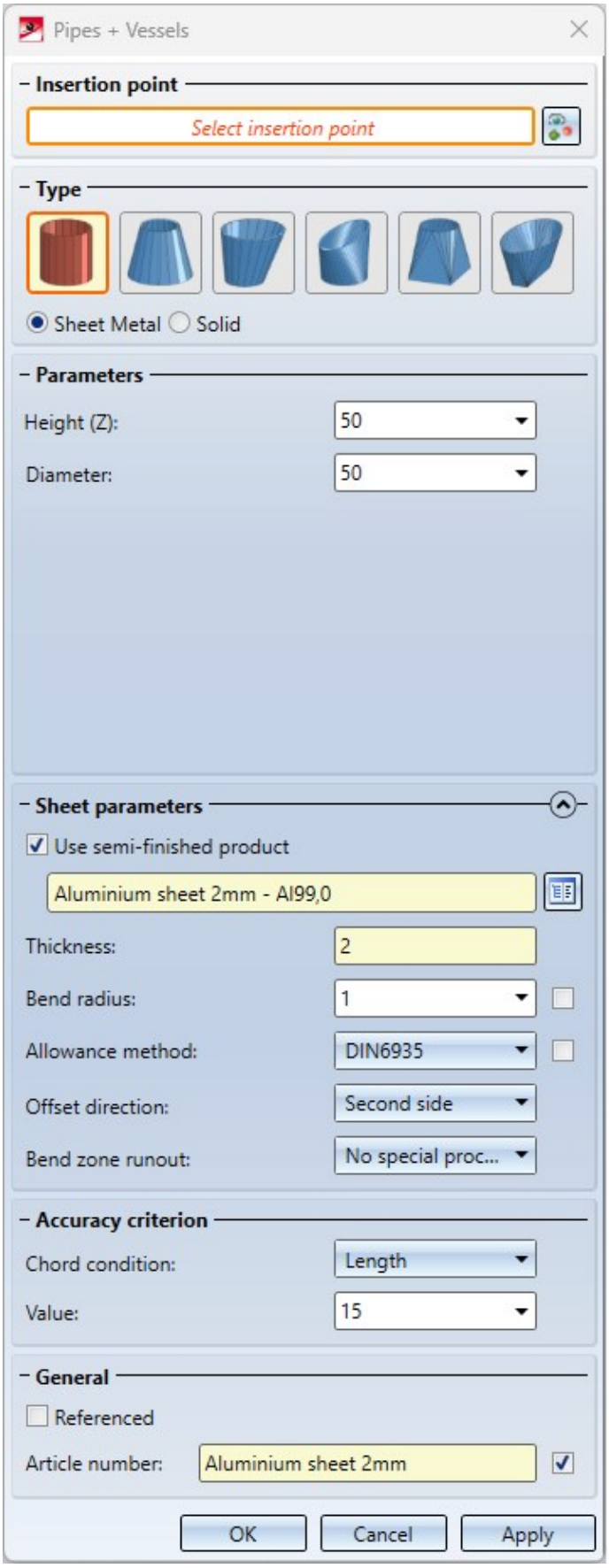
Revised Pipes + Vessels function

The **Pipes + Vessels**  function has been revised and made more user-friendly. As a result, you can now create parts as hollow and solid bodies in different variants and insert them as main or sub-parts.

There are six different types to choose from. You then decide whether the part should be created as a **Sheet Metal** part or as a **Solid**. The **Parameters** can be used to customize the parts as required.

The **Sheet parameters** are only active if you have selected **Sheet Metal** in the dialogue. You can either select a material and a thickness from the catalogue, or you can determine the **Thickness** in the input field without selecting a material. You also have the option of selecting a **Bend radius** and one of ten different **Allowance method**s. The **Offset direction** defines the direction in which the sheet is to be created. In the **Bend zone runout** field, you can choose from three options for how the bend zones on the edges should look. Under **Accuracy criterion**, you can define the **Chord condition** via the **Angle**, the **Distance** or the **Length**. You also need to enter a value that determines the accuracy of the part.

Finally, you can save the part **Referenced** as usual and assign an **Article number**. If you have selected a semi-finished product, an article number will be suggested.

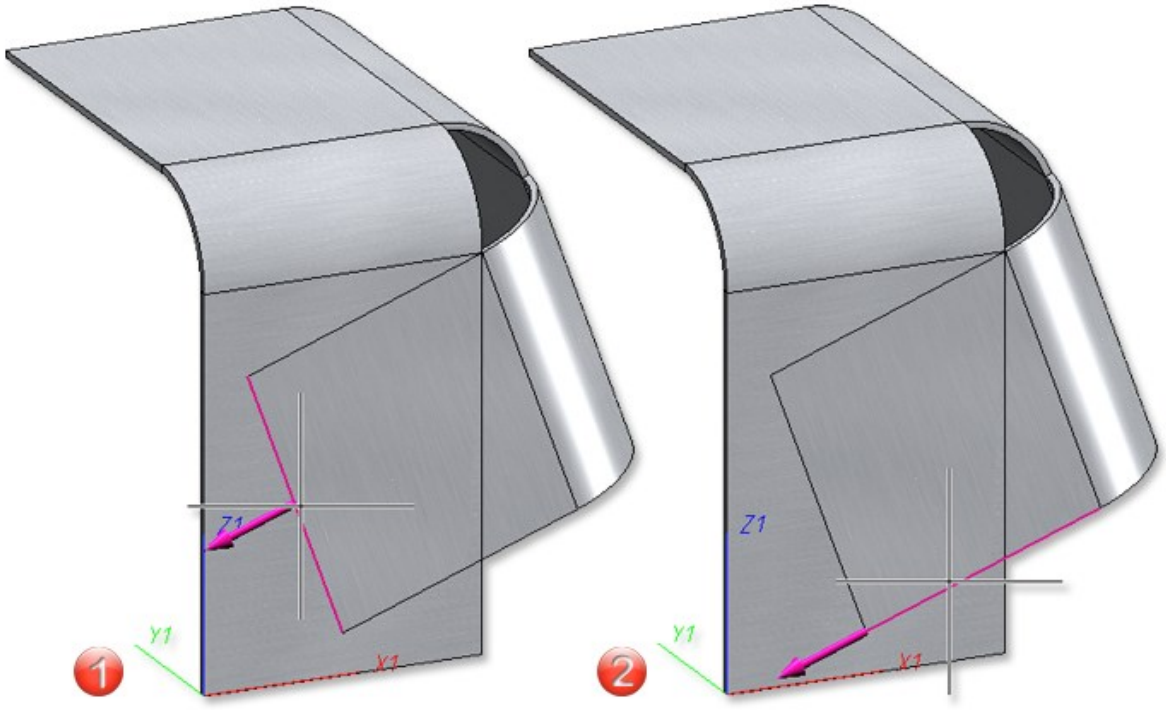

Pipes + Vessels
Insertion point
Select insertion point
Type
Sheet Metal
Solid
Parameters
Height (Z):
50
Diameter:
50
Sheet parameters
Use semi-finished product
Aluminium sheet 2mm - Al99,0
Thickness:
2
Bend radius:
1
Allowance method:
DIN6935
Offset direction:
Second side
Bend zone runout:
No special proc...
Accuracy criterion
Chord condition:
Length
Value:
15
General
Referenced
Article number:
Aluminium sheet 2mm
OK
Cancel
Apply

Length change by selecting front edge

By default, as of HiCAD 2024 SP1, the **Front edge** is requested for identification in the functions for length changes of sheets. This affects the functions:

- **Corner/Mitre**  ,
- **Trim**  ,
- **Change length** 

If the selection of the front edge is not suitable for the desired purpose, you can switch to the longitudinal edge with the right mouse button. In the Configuration Editor, go to **Sheet Metal > Default setting** and use the parameter **Start mode when identifying front surfaces** to change the ISD default setting from Front edge to Longitudinal edge.



(1) Front edge, (2) Longitudinal edge

Major Release 2024 (V 2900)

Feature for Sheet Metal part creation

In the function dialogues of the sheet creation, the checkbox **Feature** is no longer available from HiCAD 2024. This means that a corresponding feature is now always created when sheets are created.

This affects the following functions in Sheet Metal:

Sheet Metal > New > Create base sheet 

Sheet Metal > New > New sheet from sketch 

Sheet Metal > New > New sheet from 2-D development 

Sheet Metal > New > New sheet along sketch 

Sheet Metal > New > Create connecting sheet 

Sheet Metal > New > New sheet from solid 

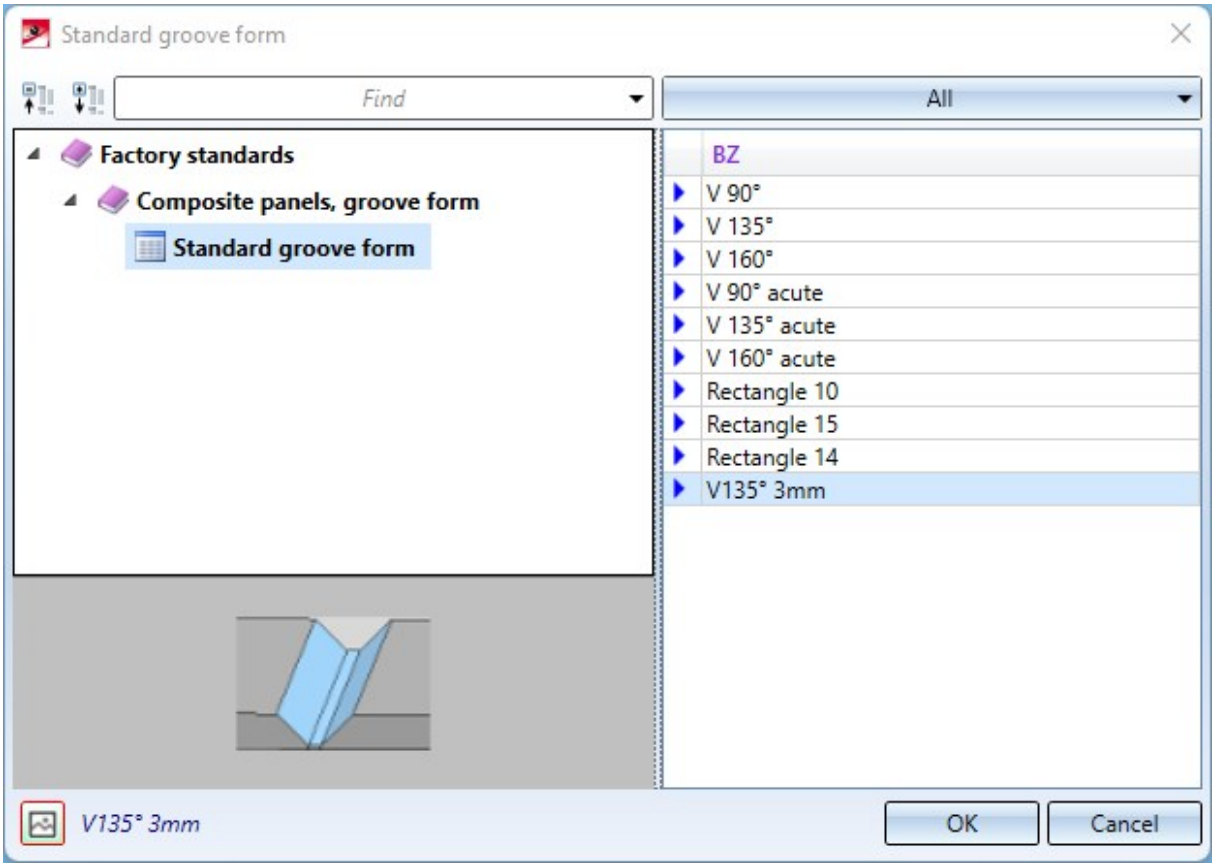
Sheet Metal > New > New sheet from surface 

Free milling

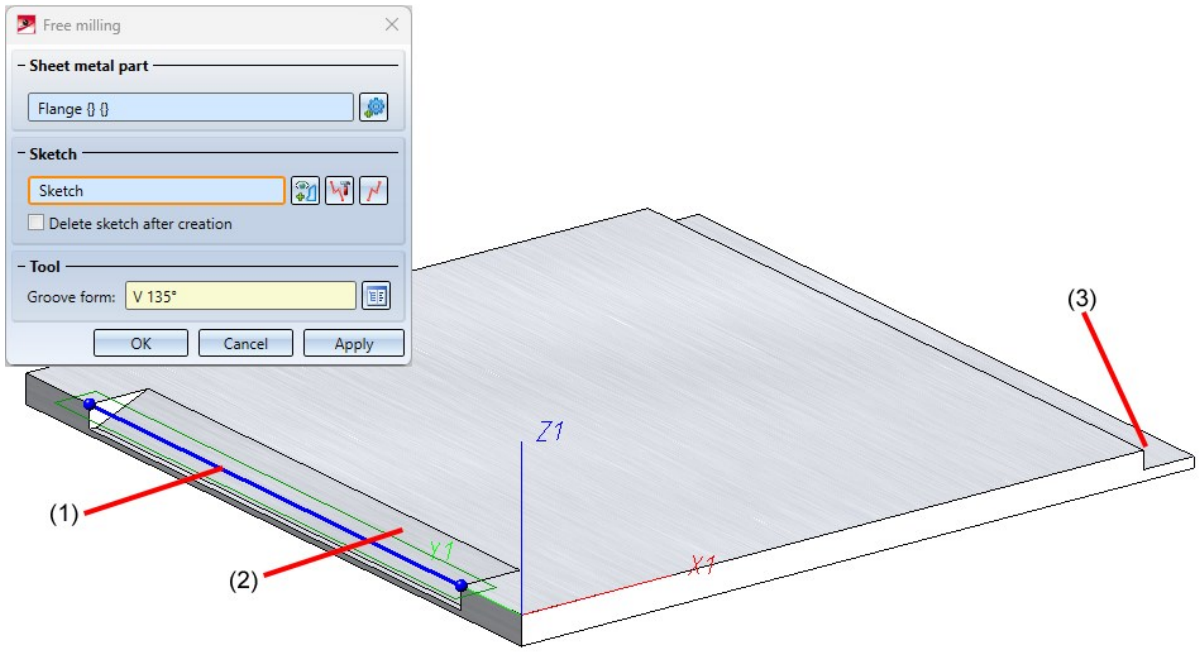
With the new **Free milling** function, you can provide the edges of composite panels with a milling tool path. The tool path is defined by a sketch and the milling tool is loaded from the catalogue.

After selecting the function, first identify the sheet metal part. For the sketch, you can either select an existing sketch and process it if necessary or draw a new sketch in the plane.

You determine the shape of the tool path by selecting the tool in the catalogue **Factory Standards > Composite panels, groove form > Standard groove forms**.



Tools for the tool path

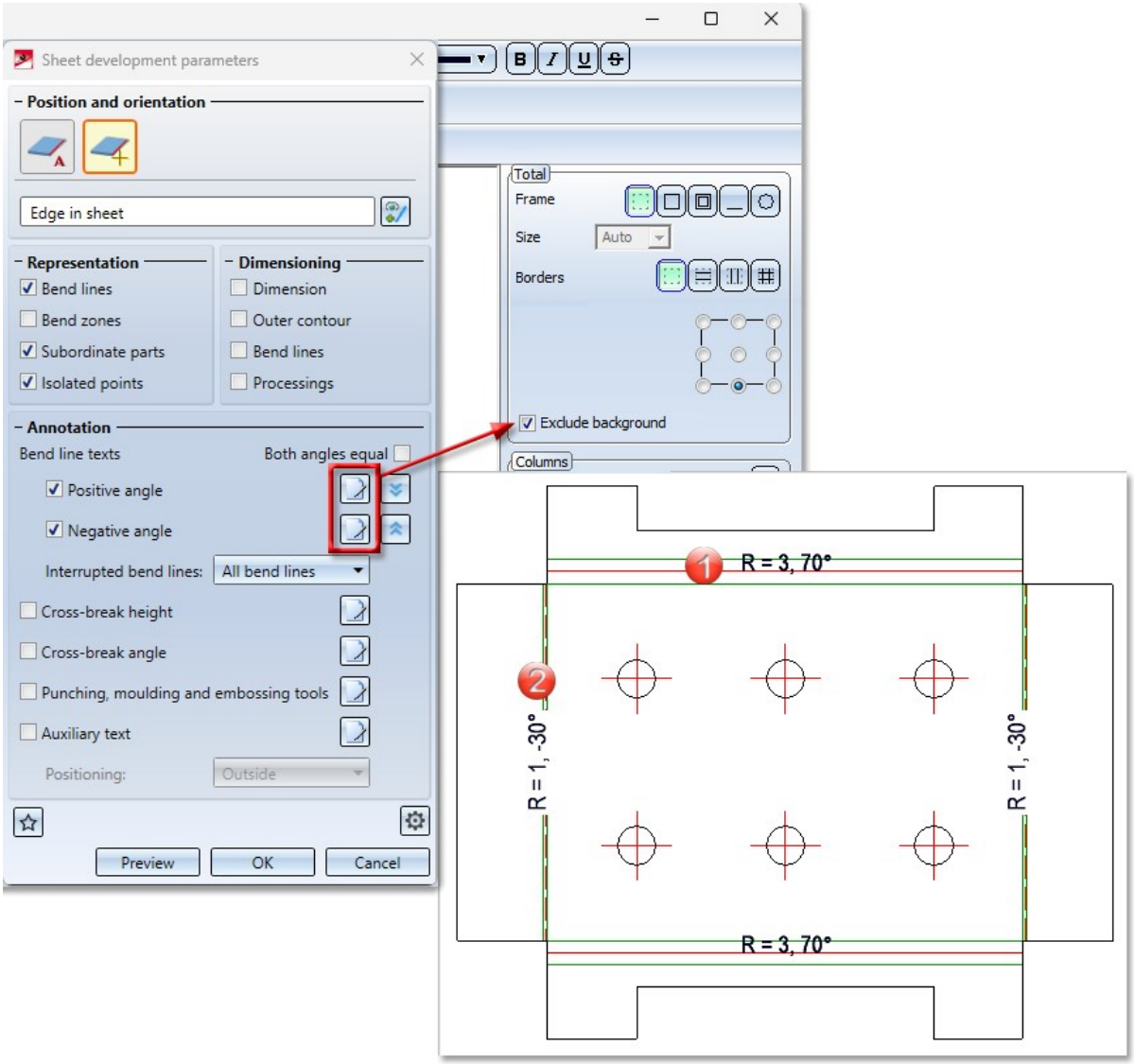


(1) Sketch, (2) Groove form: V 135°, (3) Groove form: Rectangle15

Sheet development

Exclude annotations in sheet developments

For bend line texts in sheet developments, you now have the choice of whether the text background should be left out. For this purpose, there is now the option **Exclude background** in the Annotation Editor.

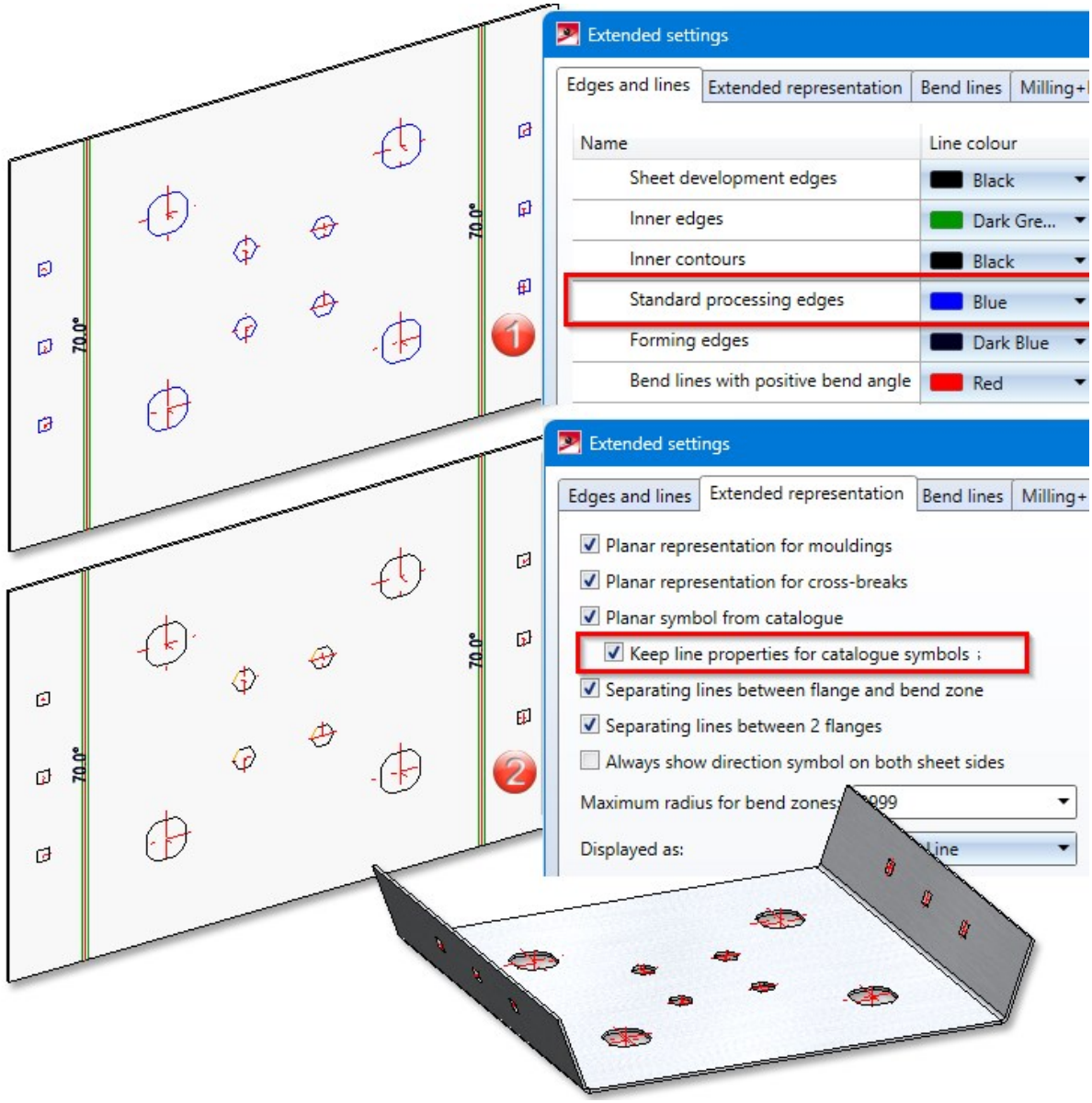


(1) Positive angle without background exclusion, (2) Negative angle with background exclusion

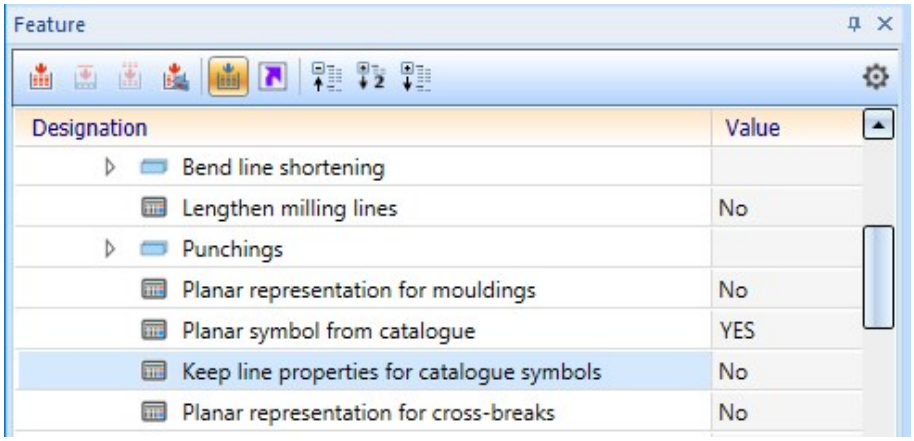
Display of the catalogue symbols

The development of a sheet is always a 3-D part. To exclude catalogue symbols (bore patterns, moulding tools and punching tools) from the 3-D display and to display them as symbols, activate the respective checkbox for the sheet development in the **Extended settings** > Tab: **Extended setting**.

The parameters (line colour, line type, ...) of the catalogue symbols are taken from the **Edges and lines** tab by default. To keep the parameters from the catalogue you can now activate the new option **Keep line properties for catalogue symbols**.



(1) Parameters of the standard processing edges changed in the **Extended settings** of the development.
(2) Line properties taken from the catalogue.



You can also change the display of the catalogue symbols in the feature.

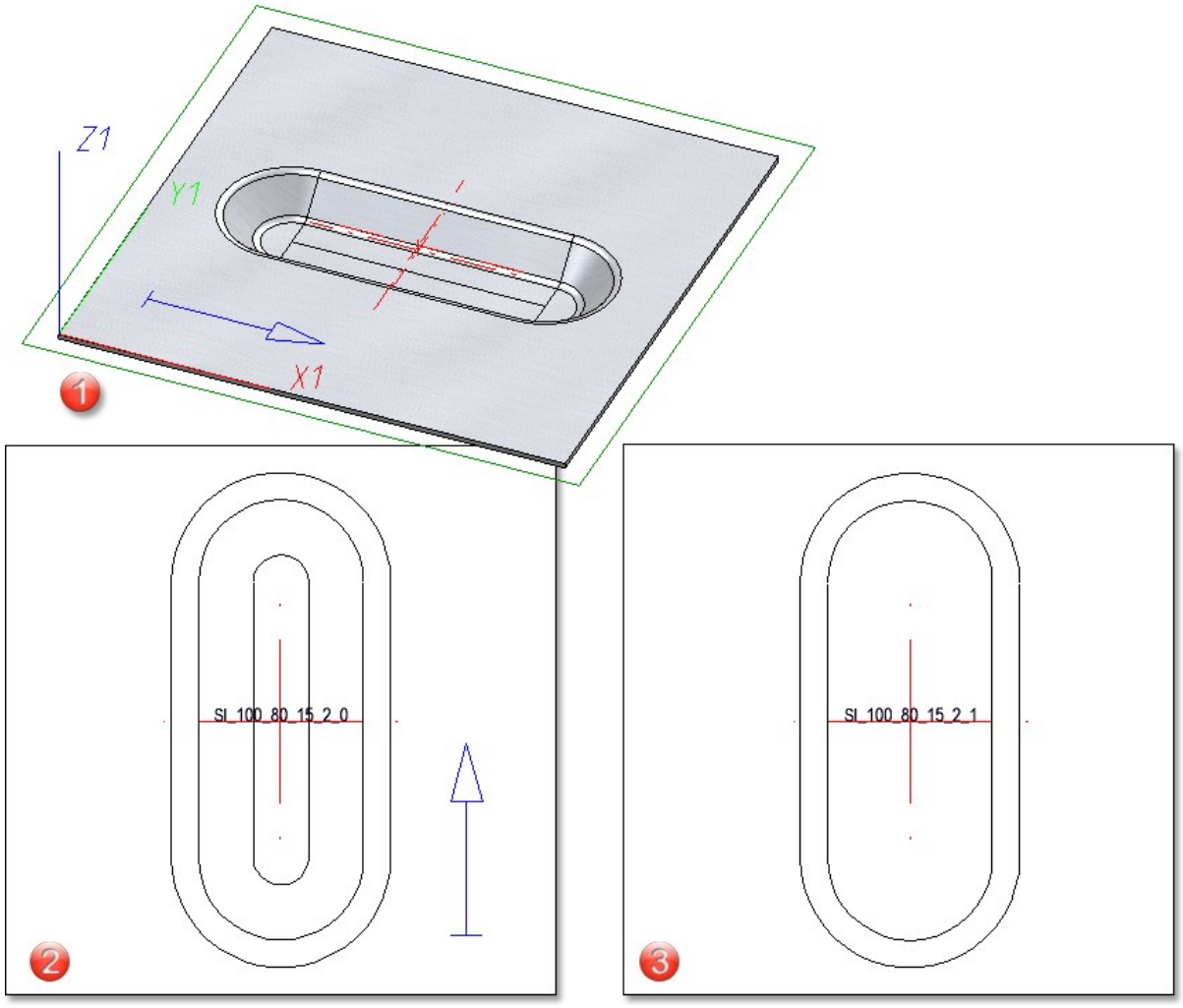
Tool numbers in sheet metal processing

In sheet metal processing, there are machines that use different tools for processing the front and back of sheets, e.g. punch-laser combination machines. Until now you could only load different representations from the catalogue columns TOPSYMBOL and BOTTOMSYMBOL for processing the front and back side in the sheet metal development. Now it is also possible to read out different tool numbers (WZNR and WZNR_BOTTOM) from the catalogue. The catalogues for moulding, embossing and punching tools have been extended by the column WZNR_BOTTOM. If the column **WZNR_BOTTOM** is empty, WZNR is used instead.

	ID	MOD	STATUS	Designation	WZNR 1	NAME	SKETCH	TYP	PREVIEW	TOPSYMBOL 1	BOTTOMSYMBOL 2	WZNR_BOTTOM 2
1	3			Beading 100/80/15/2	SI_100_80_15_2_0	Formwerkzeuge\BEADING.KRA	Formwerkzeuge\BEADING_Sketch.KRA	1	Formwerkzeuge\image\BEADING.bmp	Praegewerkzeuge\BEADING_TOP.FGA	Praegewerkzeuge\BEADING_BOTTOM.FGA	SI_100_80_15_2_1
2	1			Beading 20/20/5/2	SI_20_20_5_2_0	Formwerkzeuge\BEADING.KRA	Formwerkzeuge\BEADING_Sketch.KRA	1	Formwerkzeuge\image\BEADING.bmp	Praegewerkzeuge\BEADING_TOP.FGA	Praegewerkzeuge\BEADING_BOTTOM.FGA	SI_20_20_5_2_1
3	2			Beading 50/50/10/2	SI_50_50_10_2_0	Formwerkzeuge\BEADING.KRA	Formwerkzeuge\BEADING_Sketch.KRA	1	Formwerkzeuge\image\BEADING.bmp	Praegewerkzeuge\BEADING_TOP.FGA	Praegewerkzeuge\BEADING_BOTTOM.FGA	SI_50_50_10_2_1

(1) Table in the catalogue with WZNR and TOPSYMBOL for the front side of the development

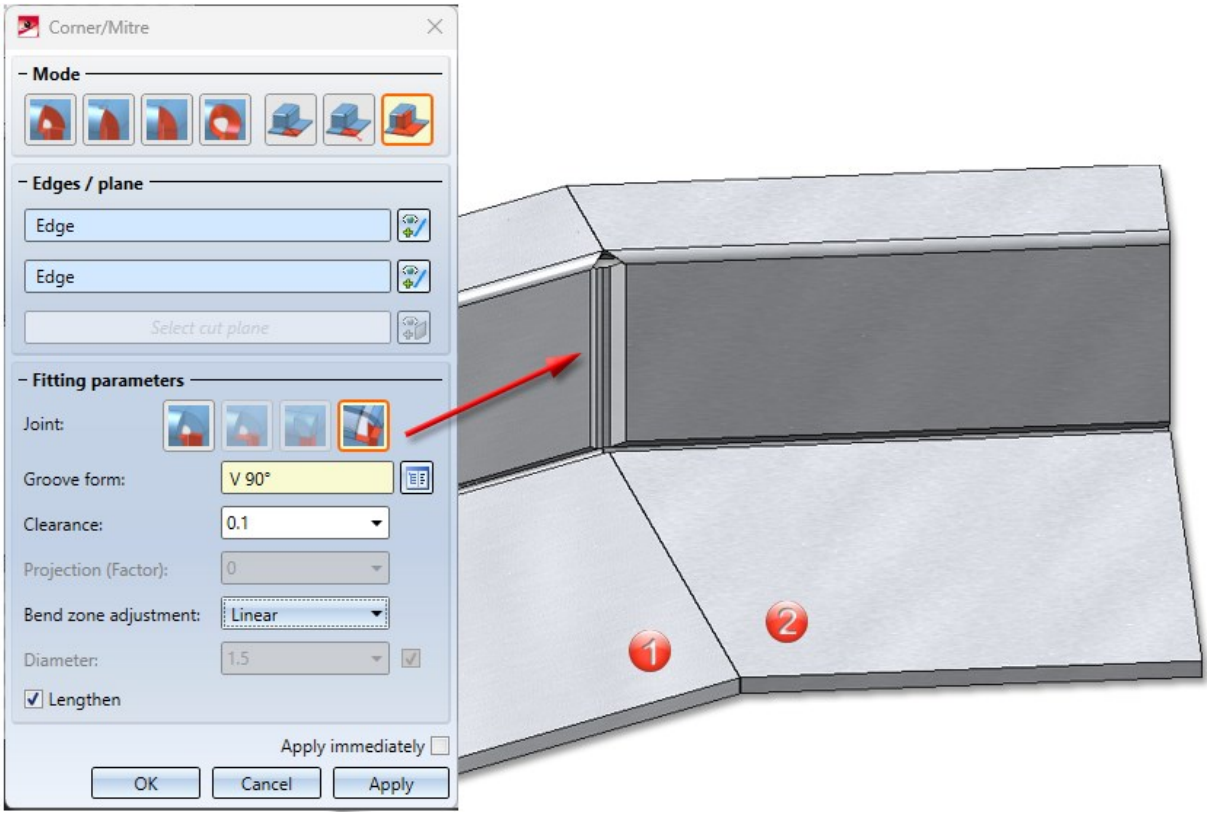
(2) (2) WZNR_BOTTOM and BOTTOMSYMBOL for the rear side of the development



(1) Sheet with beading
(2) Development of the front side with plane representation of the moulding tool (TOPSYMBOL) and tool number (WZNR)
(3) Development of the back side with plane representation of the moulding tool (BOTTOMSYMBOL) and tool number (WZNR_BOTTOM)

Mitre with neighbouring sheets

With the **Corner/Mitre** function you can now add a milling edge in the **Mitre, with neighbours** mode. To do this, you must activate the **Sheet edge as milling edge** option in the dialogue.



(1) First sheet, (2) Second sheet connected with milling edge zone

Apply length value

To change the length of various sheets by the same value, the function **Change length** has been lextended. When lengthening sheets and bend zones, you can now fix the value for the **Length specification** options **By value** and **Total length** .

With this switch position, the value (active option **By value**) for the extension is set to **0** again after the first length change is accepted and the next edge is selected. This means that the last entry is not saved. If the **Total length** option is active, the total length is displayed here after the new selection.

If this setting is active, the value or the total length is also available for the next edge after choosing **Apply**.

Extension of the coating function

HiCAD now also allows you to coat bulb plates and checker plates. You can assign different parameters (colour and description) for the front and back side.

Until now, the coating of general parts (without structure) was not displayed in section and detail views. From HiCAD 2024, the coating of general parts (without structure) is also displayed in sectional and detail views.

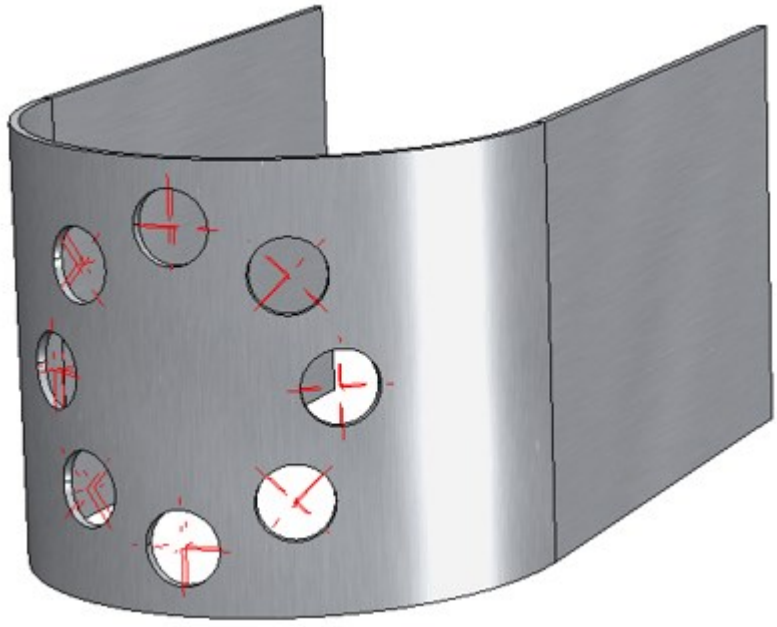
Settings for 2-D DXF export

The settings for exporting developments as 2-D DXF files have been revised. Important settings are now moved from the **Compatibility** tab to the front tabs in a user-friendly, revised version. The remaining settings are now only displayed on the **Compatibility** tab if they do not correspond to the default settings (values of the former acad-hcad/hcadacad.dat).

The settings for DXF can be customised when exporting developments using the **Edit settings** function and then saved as **Favourites**.

Improved handling of crosshairs during bending simulations

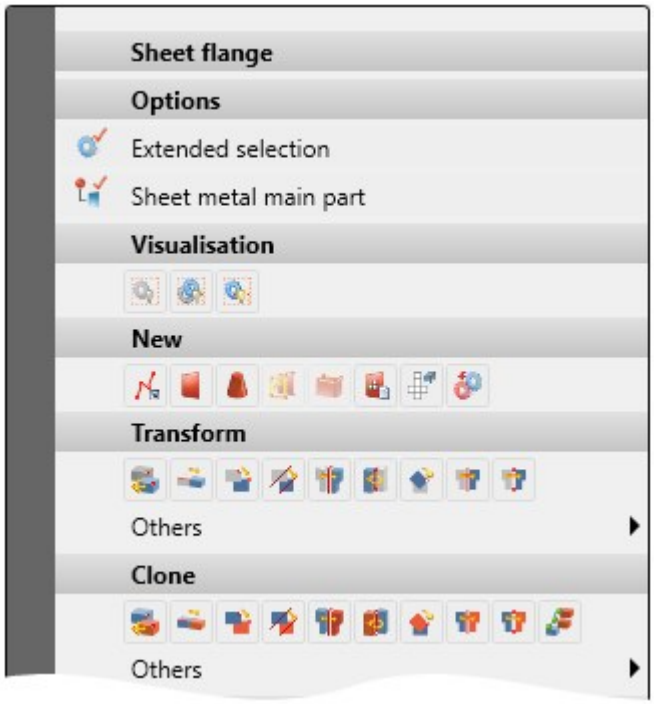
Crosshairs in bend zones of sheets are now moved to the appropriate position during a bending simulation, provided you have activated the bend zone and not the entire sheet during the process.



Bores forming a circular pattern in a bend zone

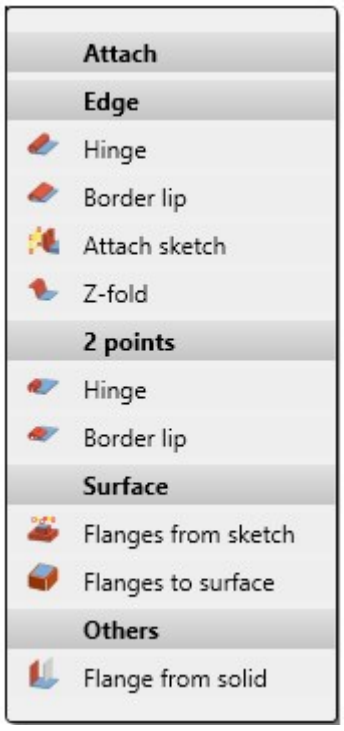
Transform + Clone in context menu

The context menu of flanges and bend zones has been extended by the functions for transforming and cloning parts. If the feature protocol is active, the entire Sheet Metal part is always transformed or cloned.



Attach flange function in context menu

In the context menu, accessible by right-clicking on a flange and then under **Attach**, the functions **Flange from sketch** and **Flanges to surface** can now be found:



Design variant SZ-20

User-defined Steel Engineering plates in catalogue

For steel plates you can create your own tables in the Catalogue Editor at **Factory standards > User-defined semi-finished products > User-defined plates**. These tables are then also offered for selection in the function **Steel Engineering > Plate, new > Rectangular plate**.

Steel Engineering

Service Pack 2024 SP1 (V 2901)

Development attributes for Steel Engineering plates

As with Sheet Metal parts, the attributes **Rectangular surface area of development (§S2D)** and **Surface area from development contour (§SOC)** are also now calculated for Steel Engineering plates if the settings in the Configuration Editor at **Modelling > Part properties** in the **Sheet Metal** area are set accordingly. .

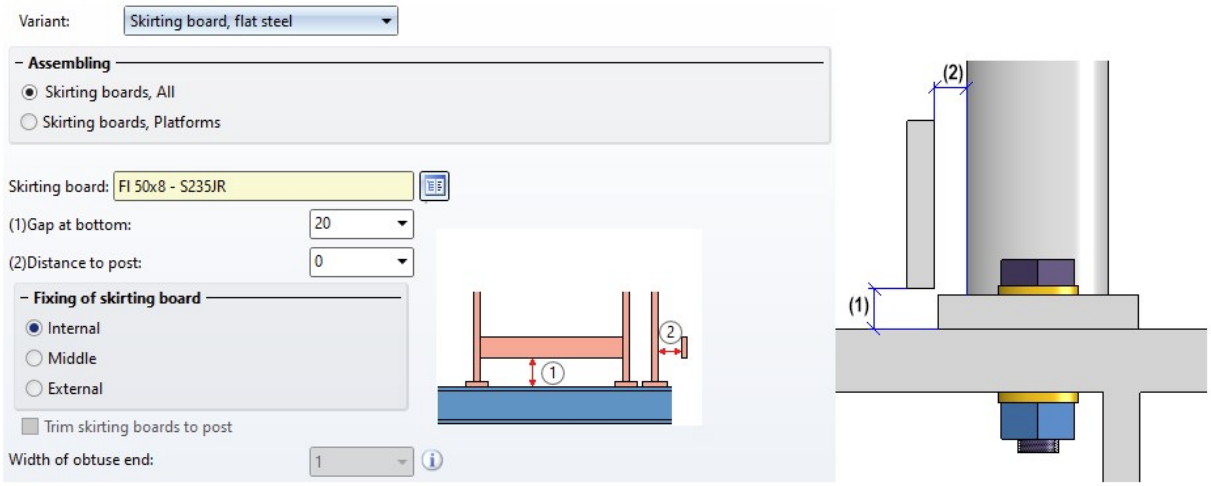
Railing Configurator

Faster start

The start of the Railing Configurator has been significantly accelerated with Service Pack 1.

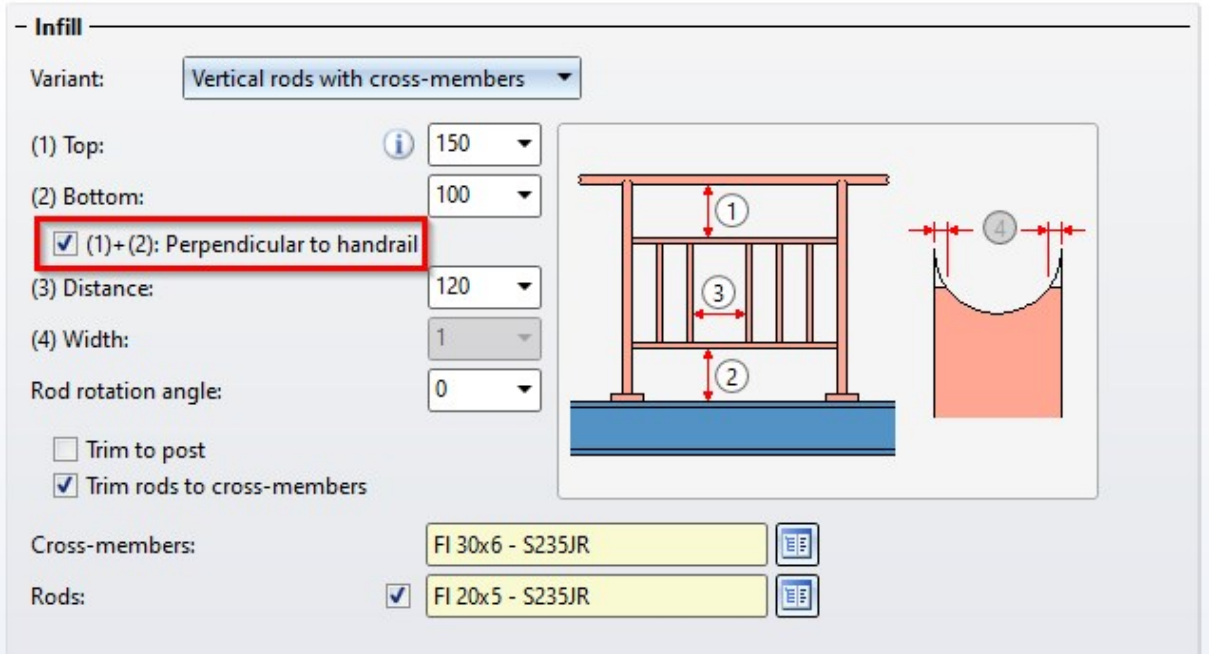
Distance between skirting board and post

The **Skirting board** tab has been expanded. The distance between the skirting board and the post can now be specified here.



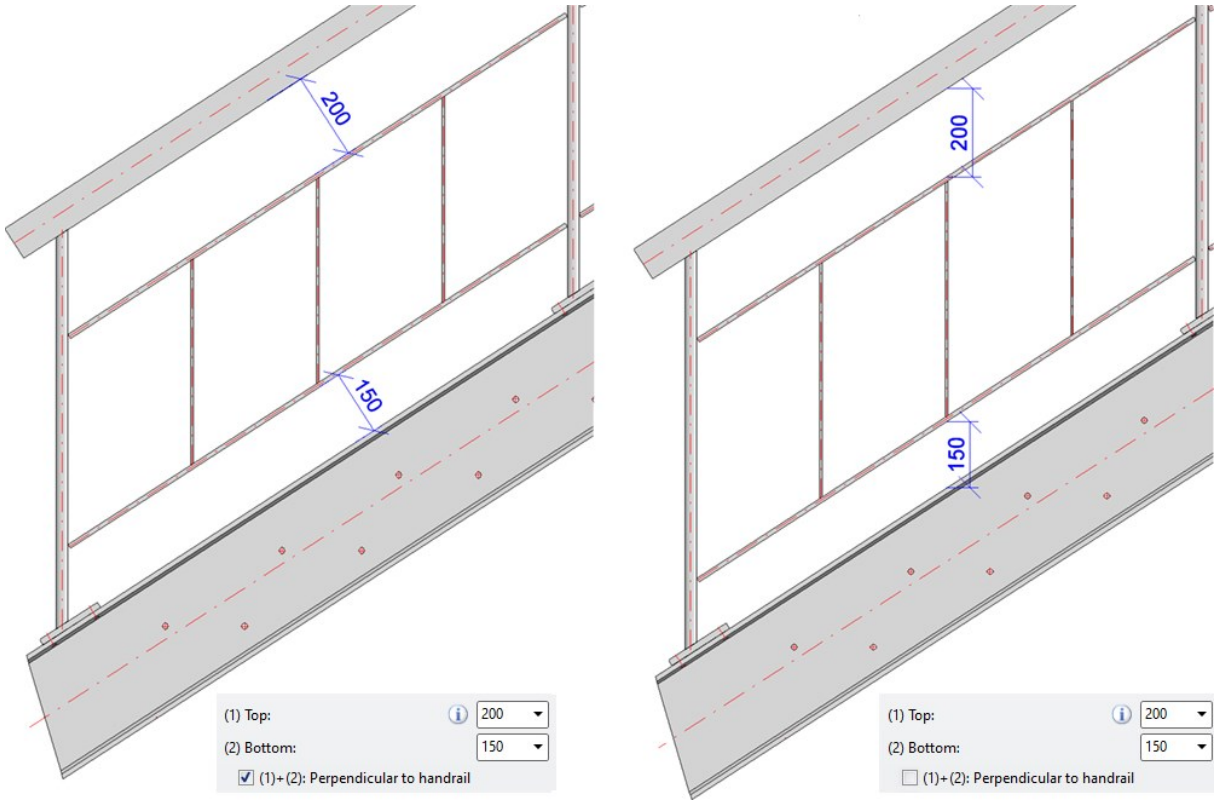
Distance between handrail and cross-member

On the **Infill** tab, the additional checkbox **Perpendicular to handrail** is available for infills with cross-members and infills with bottom cross-members, e.g.



If the checkbox is activated, the distance at the top/bottom is interpreted as a vertical distance between the handrail and cross-member, or cross-member and beam. This only has an effect on sloping railings.

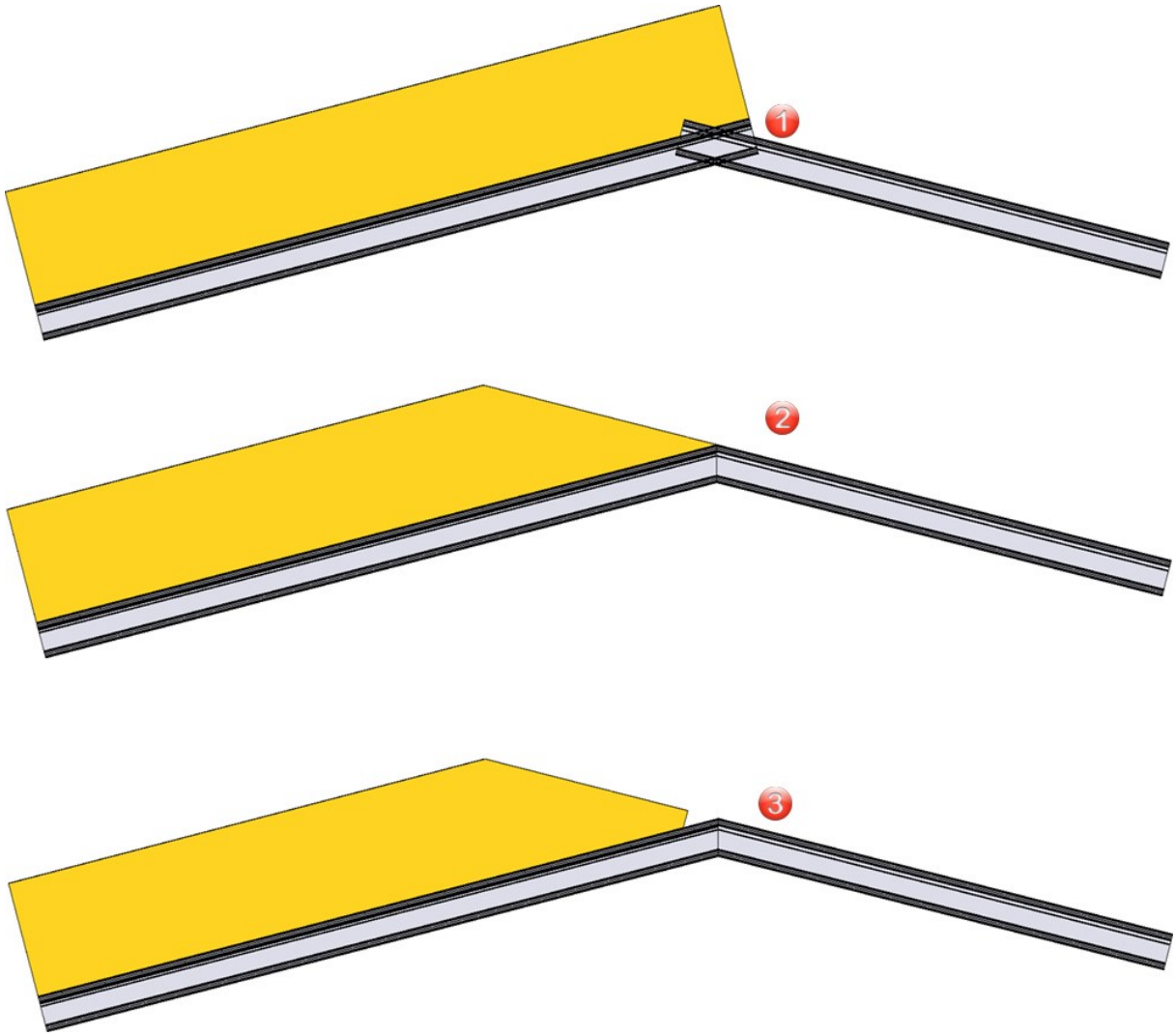
The following illustration shows the difference.



Feature for mitre cuts

For mitre cuts, a feature with the name **Mitre cut** is created for both processed beams. From SP1, the two features are linked. This means that if changes are made to one of the features, the other feature is automatically changed accordingly. If, for example, the mitre cut feature is deleted from one of the two beams, then - unlike previous HiCAD versions - the feature of the other beam is also deleted.

Linking the two features improves the use of the function for variable-controlled assemblies, e.g. for user-defined infills in the Railing Configurator. Previously, the feature recalculation, which is carried out in particular when variables are changed, could lead to undesirable results. The image below shows such a case:



(1) shows the initial state, (2) the result as of HiCAD 2024 SP1, (3) an undesired result that could occur before HiCAD 2024 SP1.

This new behaviour does not apply if the mitre is created via the HiCAD API.

Civil Engineering - Part type catalogue, 3-D

The **Civil Engineering - Part type catalogue > 3-D** function in the **Civil Engineering functions > Civil Engineering, general** docking window contained functions that have since been replaced by new developments and are therefore no longer required. The function is therefore no longer available as of SP1.

Galvanization holes on beams

In the settings for the galvanisation holes on the beam, the Y-distance of the bores now refers to the fillet on the beam and not to the flange as before. This applies to the following connections:

- Front Plate Connection to Web/Flange (2320) ,
- Front Plate Connection to Web, Double-sided (2322),
- Front Plate Connection to Flange (2330),
- Column Connection, Frame Corner (2203) and
- Column Connection, Frame Corner (2204).

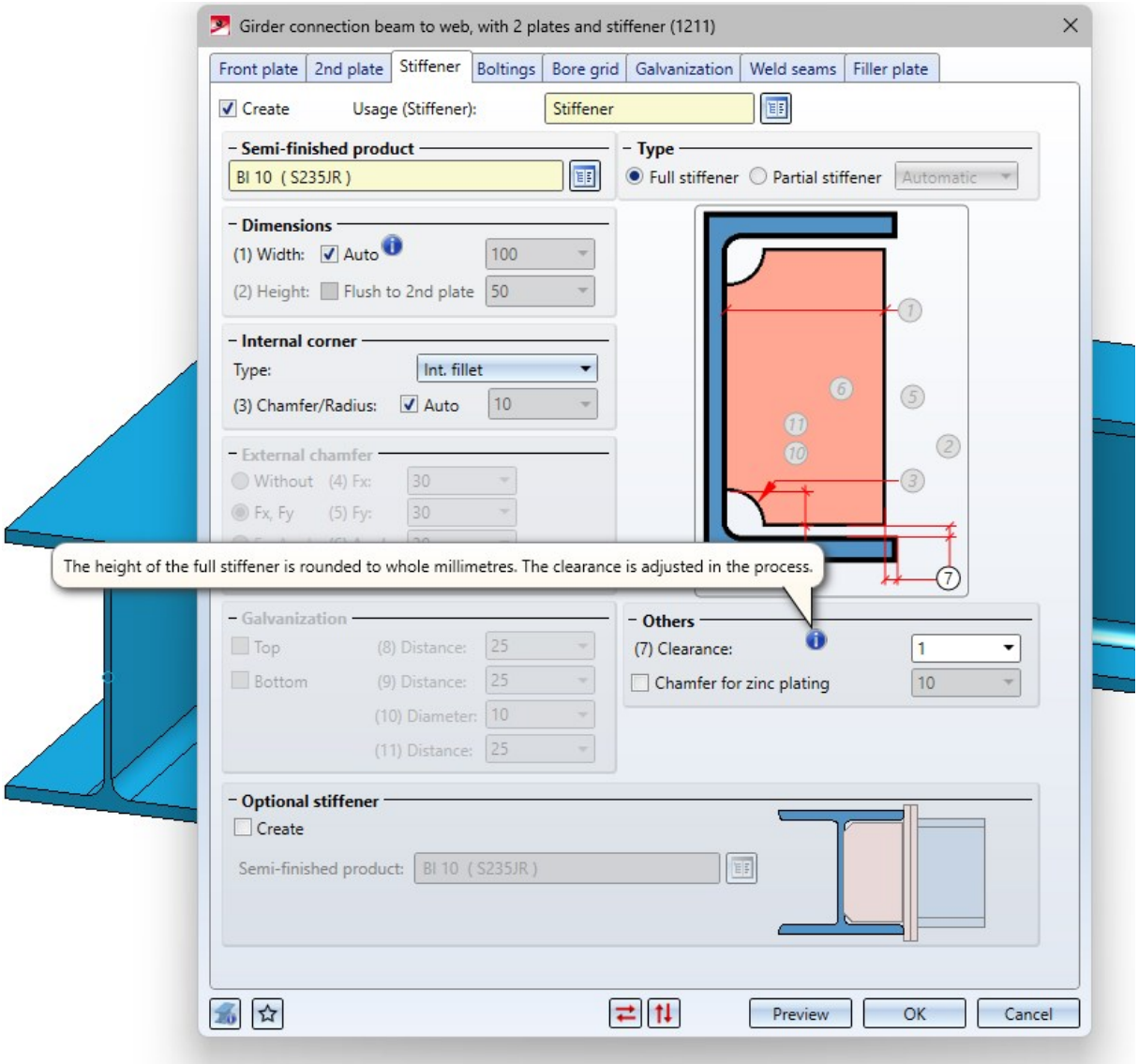
Clearance for stiffeners

When inserting stiffeners, the height of the full stiffener is rounded down to whole millimetres. The clearance is automatically adjusted.

This applies to the following design variants:

- Stiffener (2401)
- Purlin Joint - 1 Girder/1 Purlin (3206),
- Front Plate Connection to Flange (2330),
- Strap Joint (2310),
- Beam to Web with 2 Plates and Stiffener (1211),
- Column Connection, Frame Corner (2203) and
- Column Connection, Frame Corner (2204).

This is indicated by an info symbol in the dialogues for these variants.



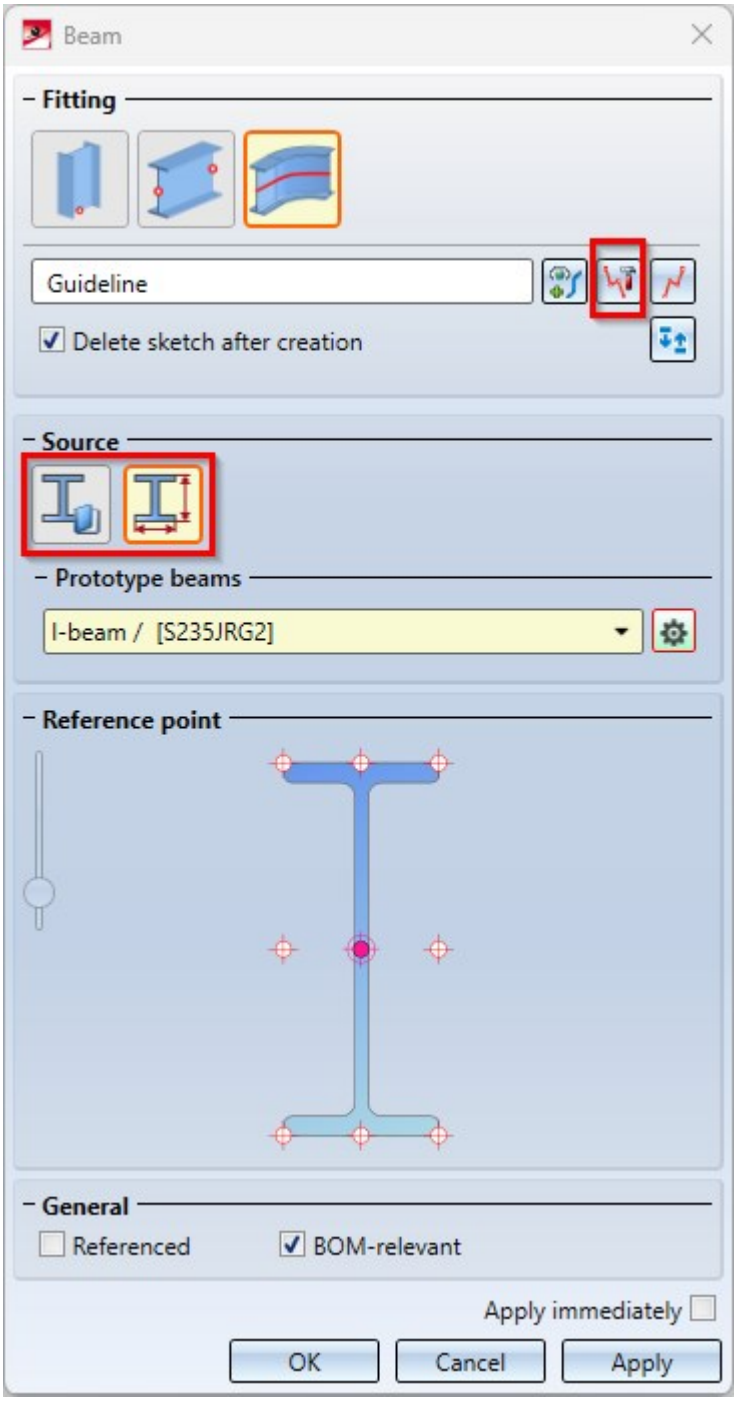
Major Release 2024 (V 2900)

Insert new beam

The function **Insert new beam**  has been extended:

- When inserting beams along a guideline, it is now also possible to process the sketch in the dialogue window.
- Two sources for inserting beams or profiles are distinguished :

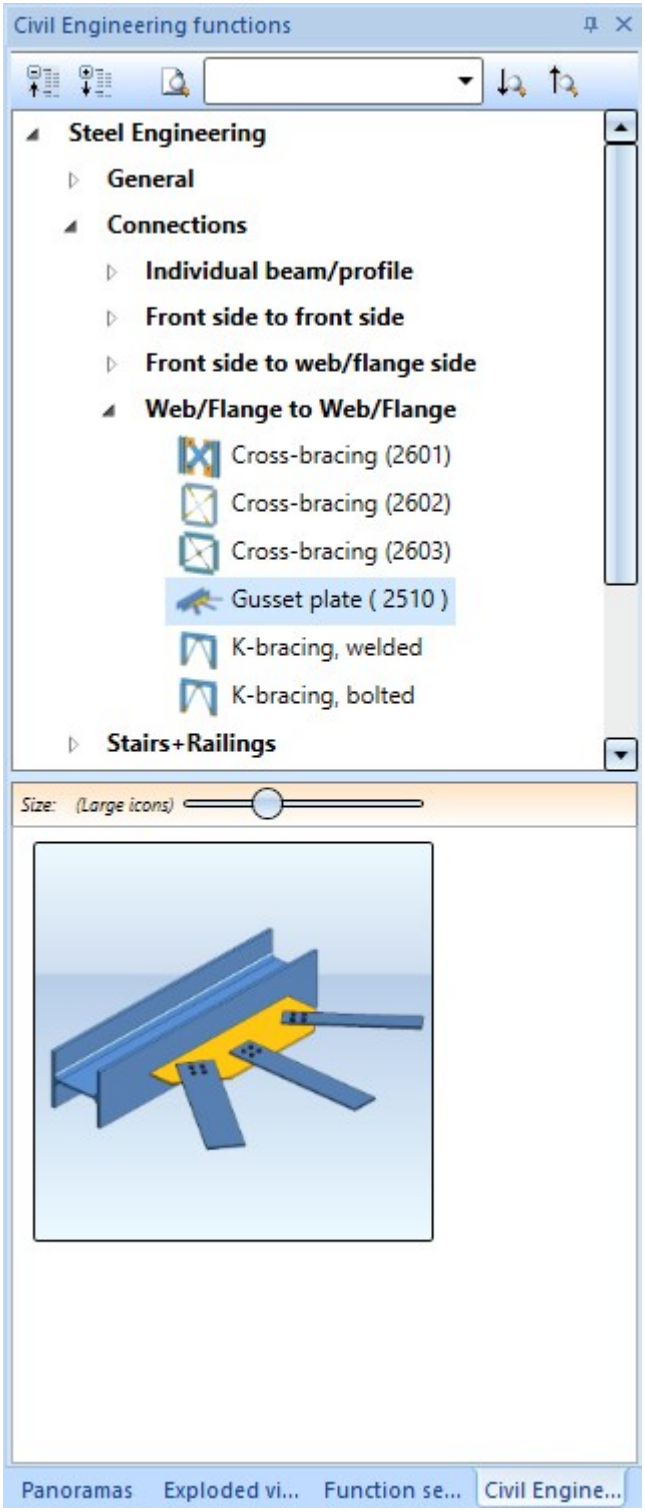
  Via catalogue...
  With this option you insert beams from the allowed catalogues mentioned above at **Semi-finished products** or **Factory standards**.
  Prototype beam...
  With this option you can insert configurable prototype beams.
- Beam and profiles from the catalogues **Factory standards > Series** and **Factory standards > Factory beams** can now also be inserted.


Beam
Fitting
Guideline
Delete sketch after creation
Source
Prototype beams
I-beam / [S235JRG2]
Reference point
General
Referenced
BOM-relevant
Apply immediately
OK
Cancel
Apply

Connections

New connection - Gusset plate (2510)

In the **Civil Engineering functions** docking window the new design variant **Gusset plate (2510)** is available at **Steel Engineering > Connections > Web/flange to web/flange**.



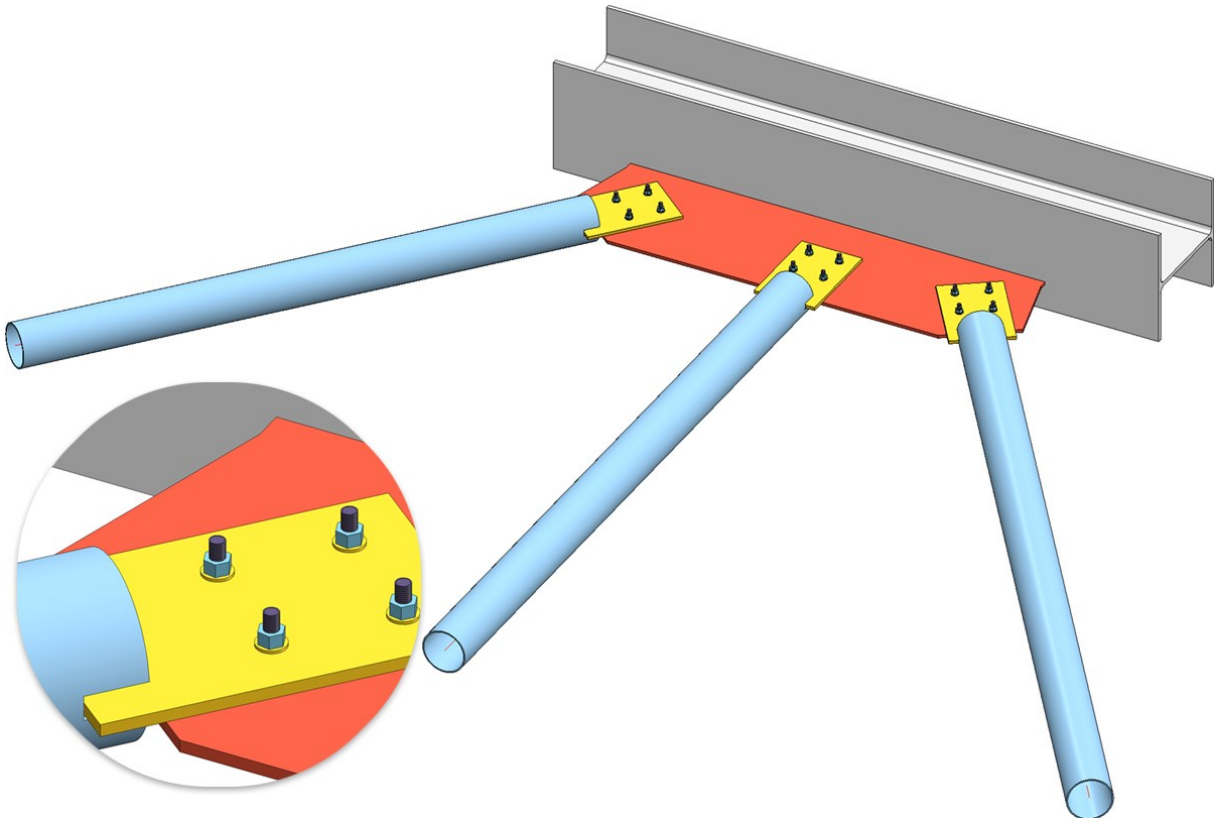
With this new design variant, any beam/profile can be connected with one to three beams/profiles by means of a gusset plate connection - as it often occurs in hall construction.

The beams/profiles to be connected must fulfil the following criteria:

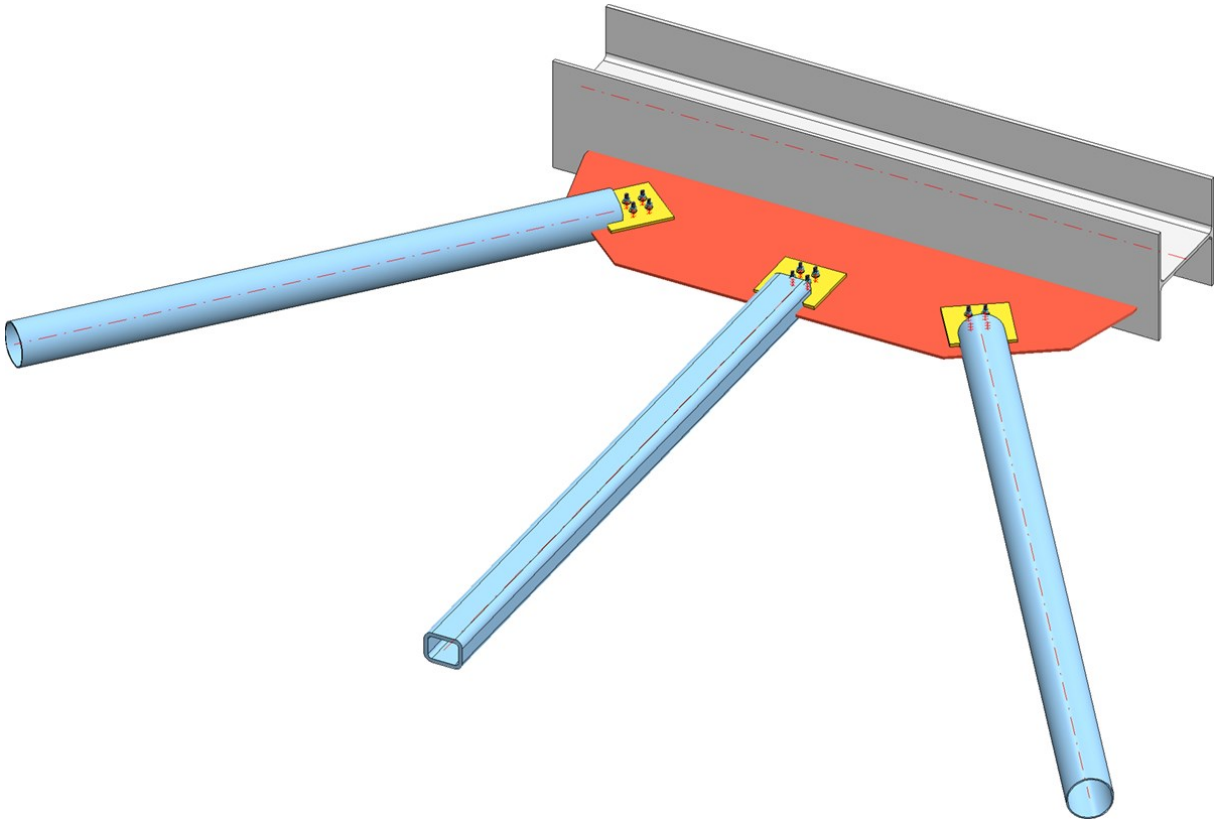
- The beams/profiles must be of the type steel pipe, hollow profile, L-beam, round or flat steel.
- In the case of hollow sections, steel pipes and round steels, the X-axes of the beams/profiles must lie in one plane. This plane must intersect the connecting beam/profile at a planar facet and be parallel to the X-axis of the connecting beam/profile.
- Flat steels and L-beams must have at least one common plane in which the gusset plate can lie.
- If more than one beam/profile is connected, all beams/profiles to be connected must be of the same type. This means that the beams/profiles must be
  - all flat steels or
  - all L-beams or
  - a mix of steel pipes, hollow profiles and round steels.

  The connection of an L-beam and a steel pipe, for example, is not possible.

The connection consists of a gusset plate, the connection plates between gusset plate and the beam/profile to be connected and - optionally - the bolting between gusset plate and connection plate. Gusset plates are steel plates, the connecting plates can be of the flat steel or steel plate type.

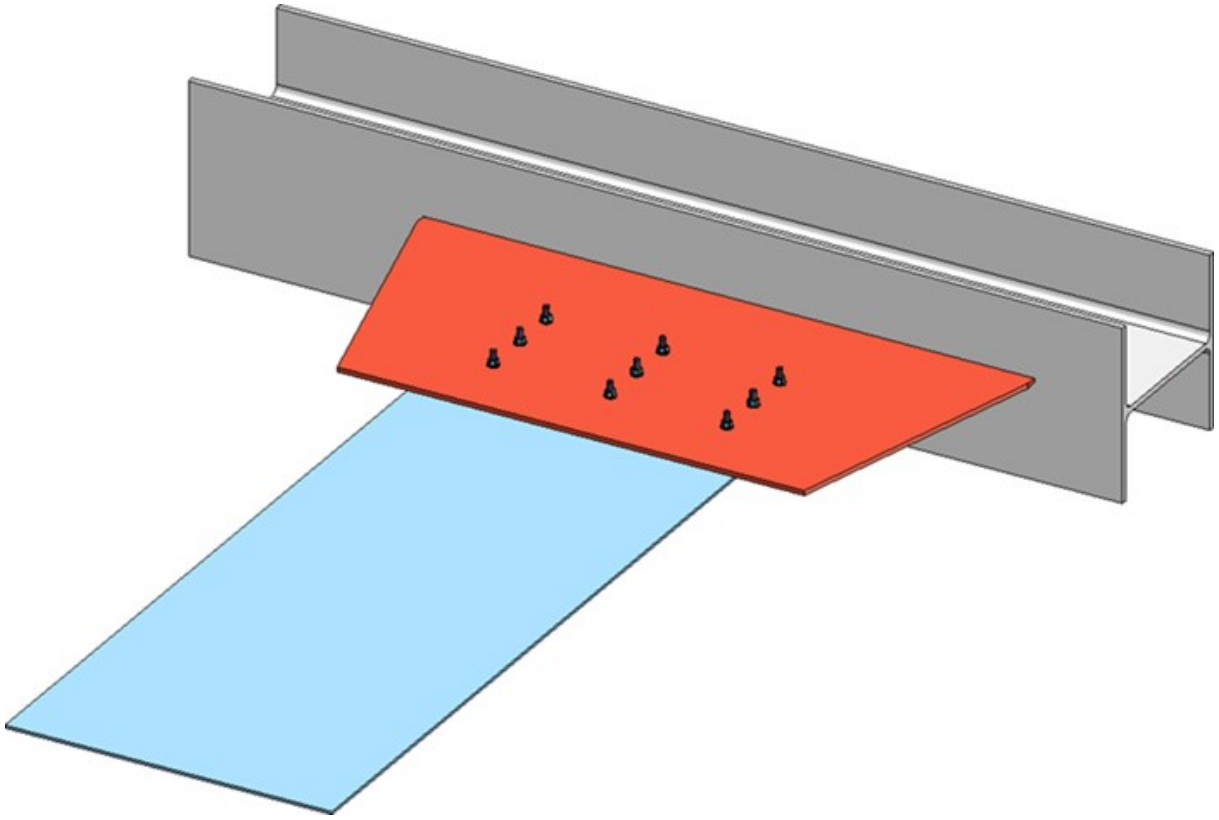


Gusset plate connection with three steel pipes



Gusset plate connection with two steel pipes and one hollow profile

If the beam/profile to be connected is of the flat steel or L-beam type, then no connecting plates will be created.



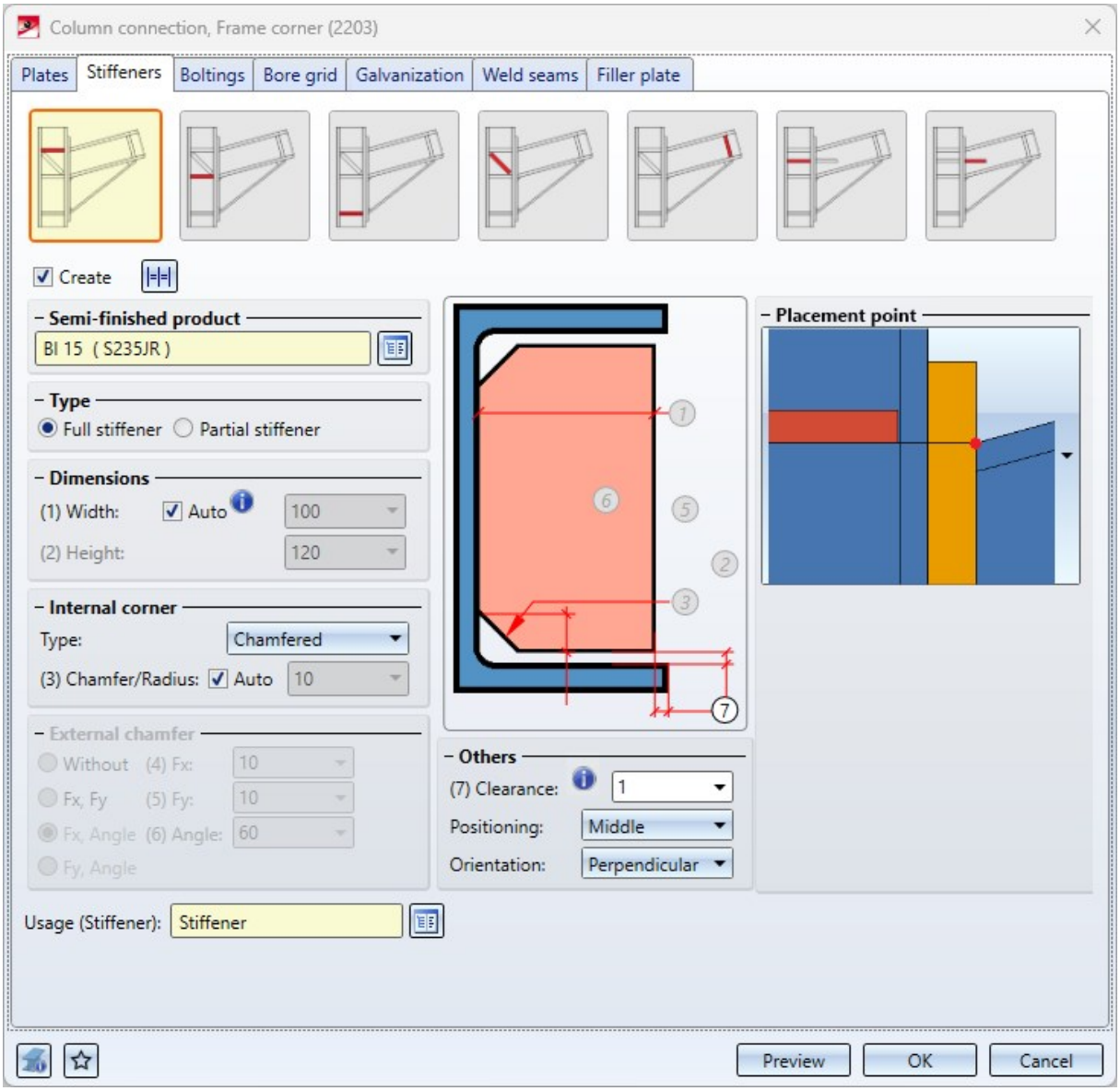
Connection of a flat steel profile

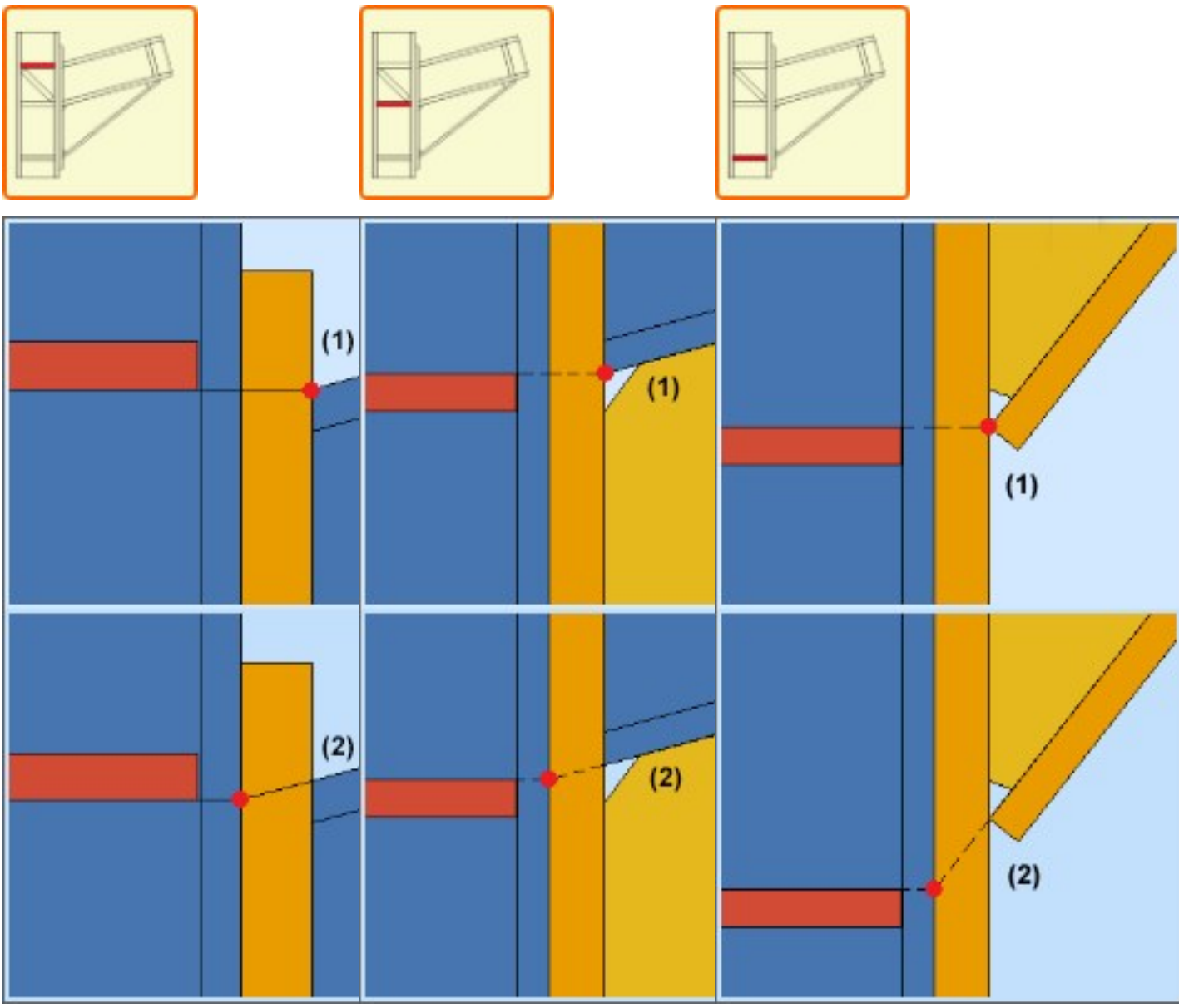
Frame corners - placement points for stiffeners

The **Frame corner 2203** connection has been extended with placement points for the stiffeners on the 2nd beam. The stiffeners are installed relative to the selected placement point. Possible placement points are:

- Web of the connecting beam (1)
- Flange of the connecting beam (2).

The **Stiffeners** tab has been extended accordingly.

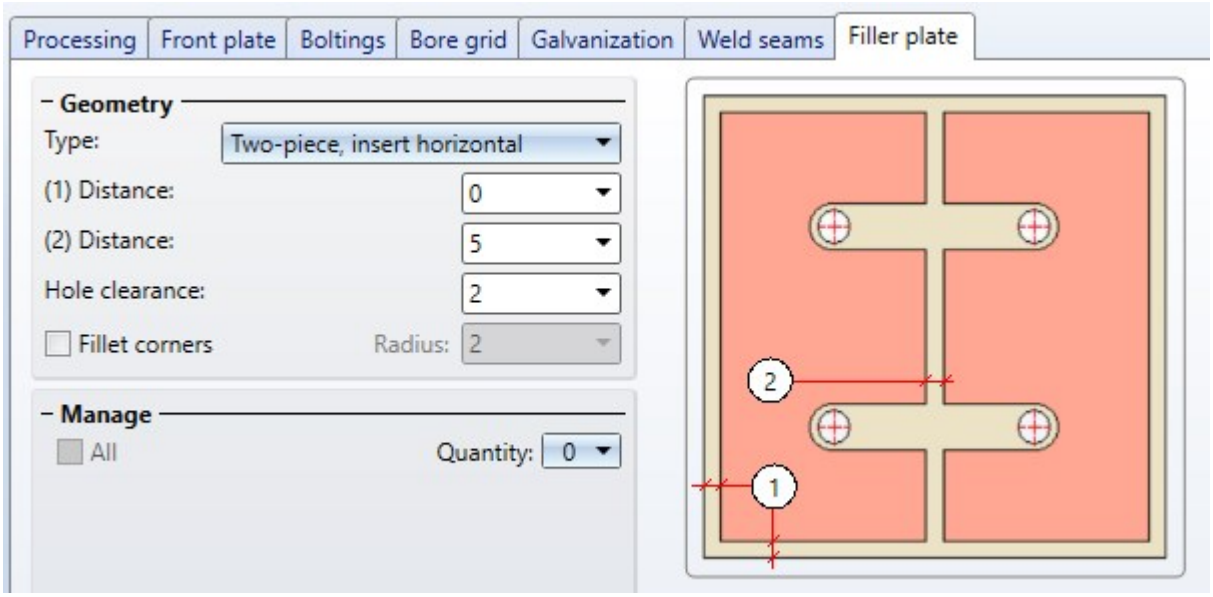


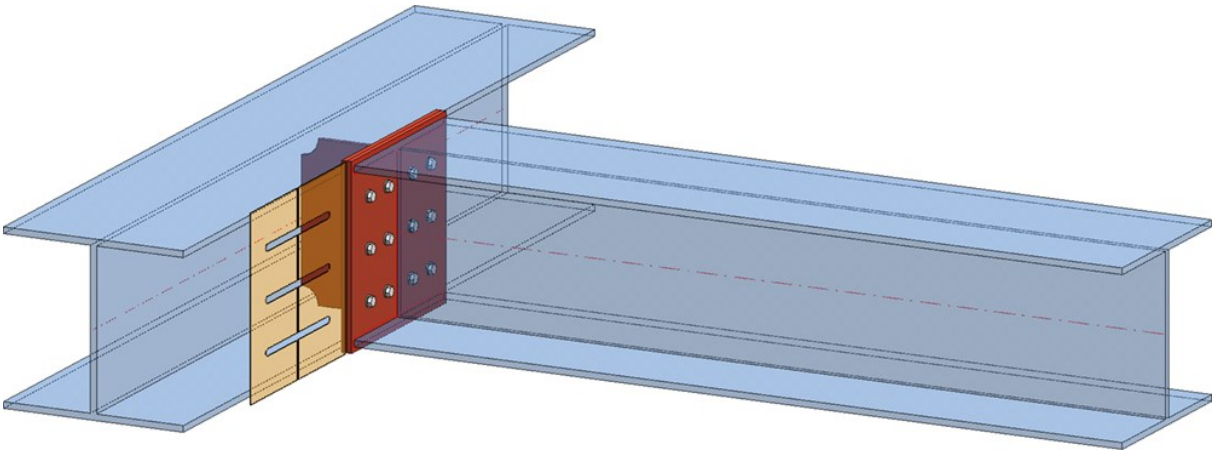


This extension also applies to **Frame corner 2204**.

Beam to web, with 2 plates + stiffener - Filler plates

The Design variant **Girder connction beam to web, with 2 plates + stiffener (1211)** now supports the installation of filler plates between the face plate and the 2nd plate, i.e. the plate on the beam to be connected. For this purpose, the dialogue window has been extended by the **Filler plates** tab.

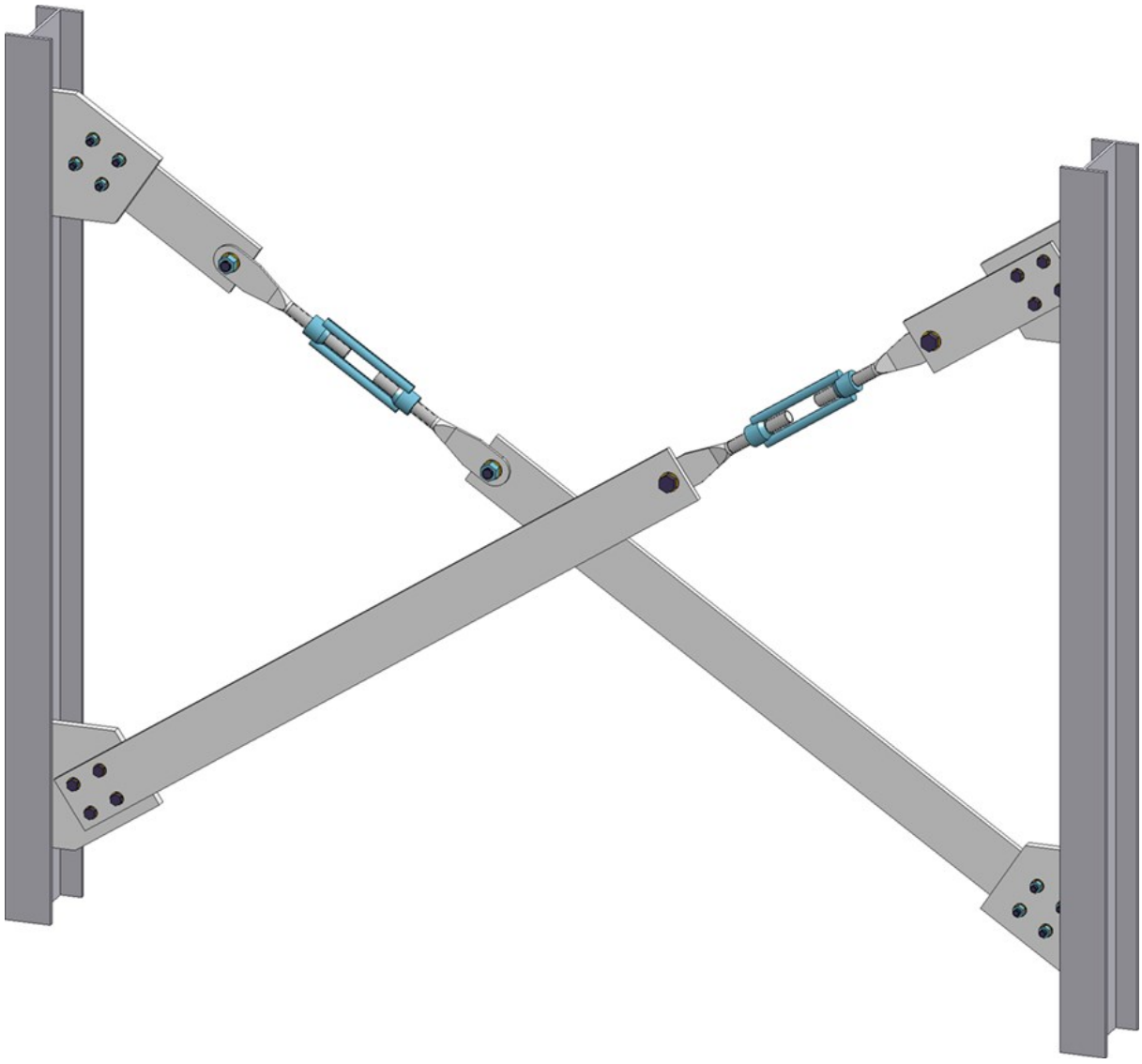




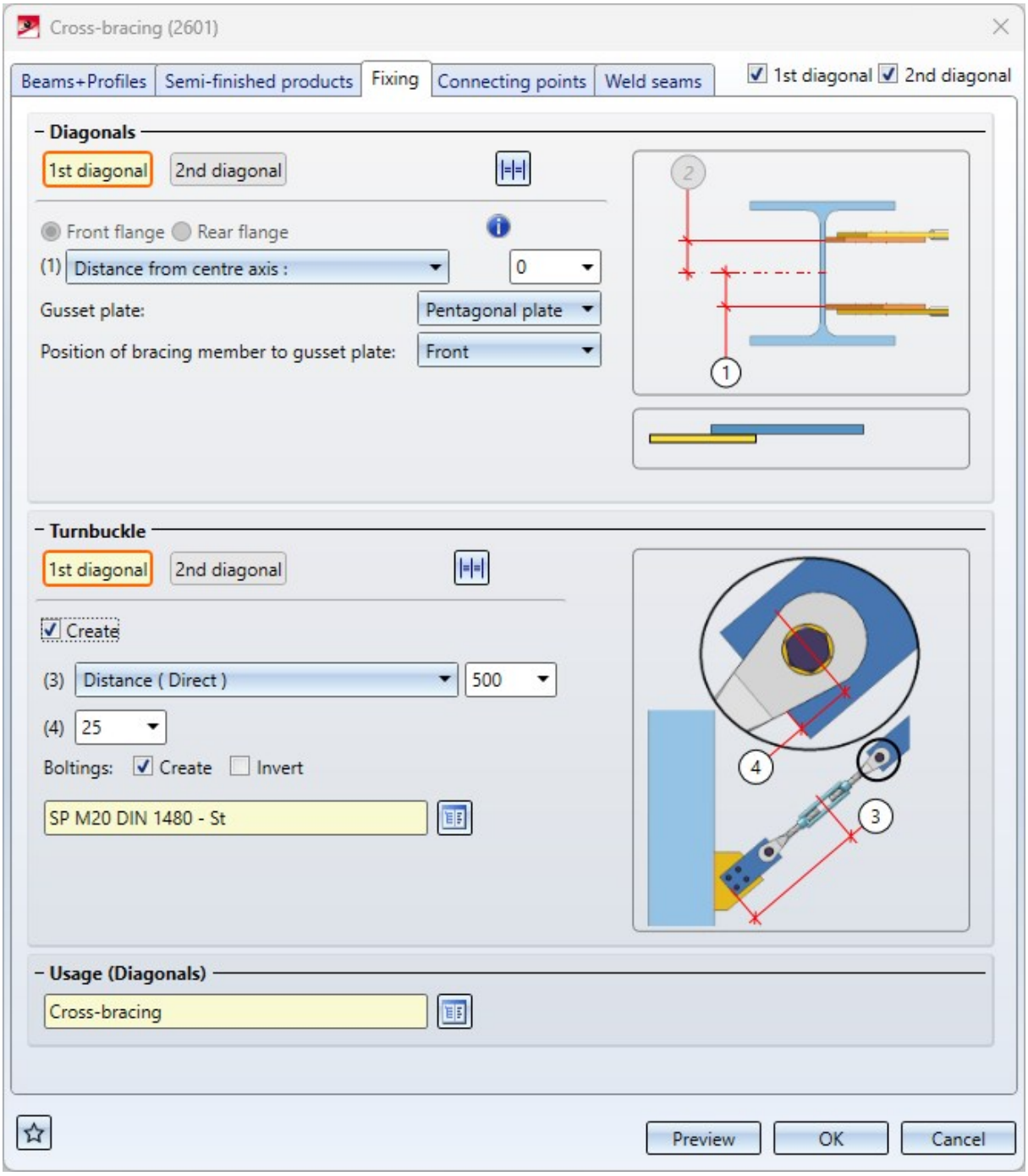
Girder connection, filler plate not pre-planned

Cross-bracing (2601) with turnbuckle and blade screws

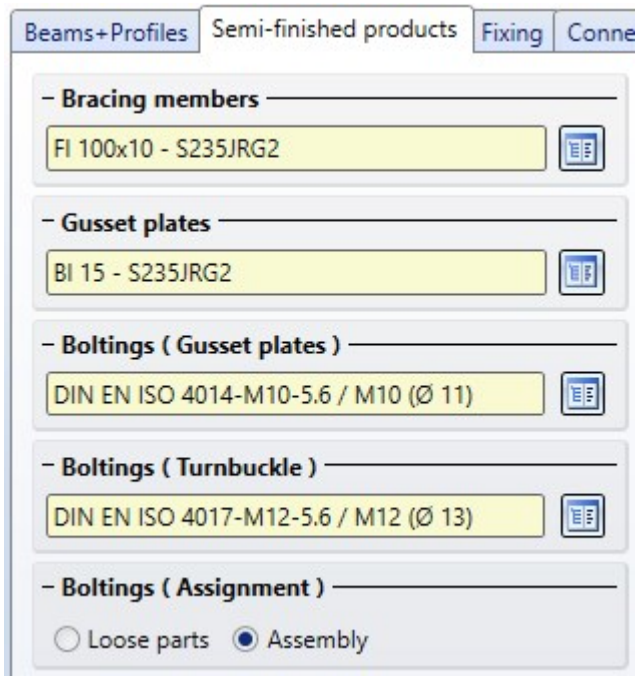
With this Design variant, the cross-bracing can now also be installed with a turnbuckle according to DIN 1480 SP.



For this purpose, the **Fixing** tab has been extended accordingly.



You define the mounting of the turnbuckle to the bracing members on the **Semi-finished products** tab.

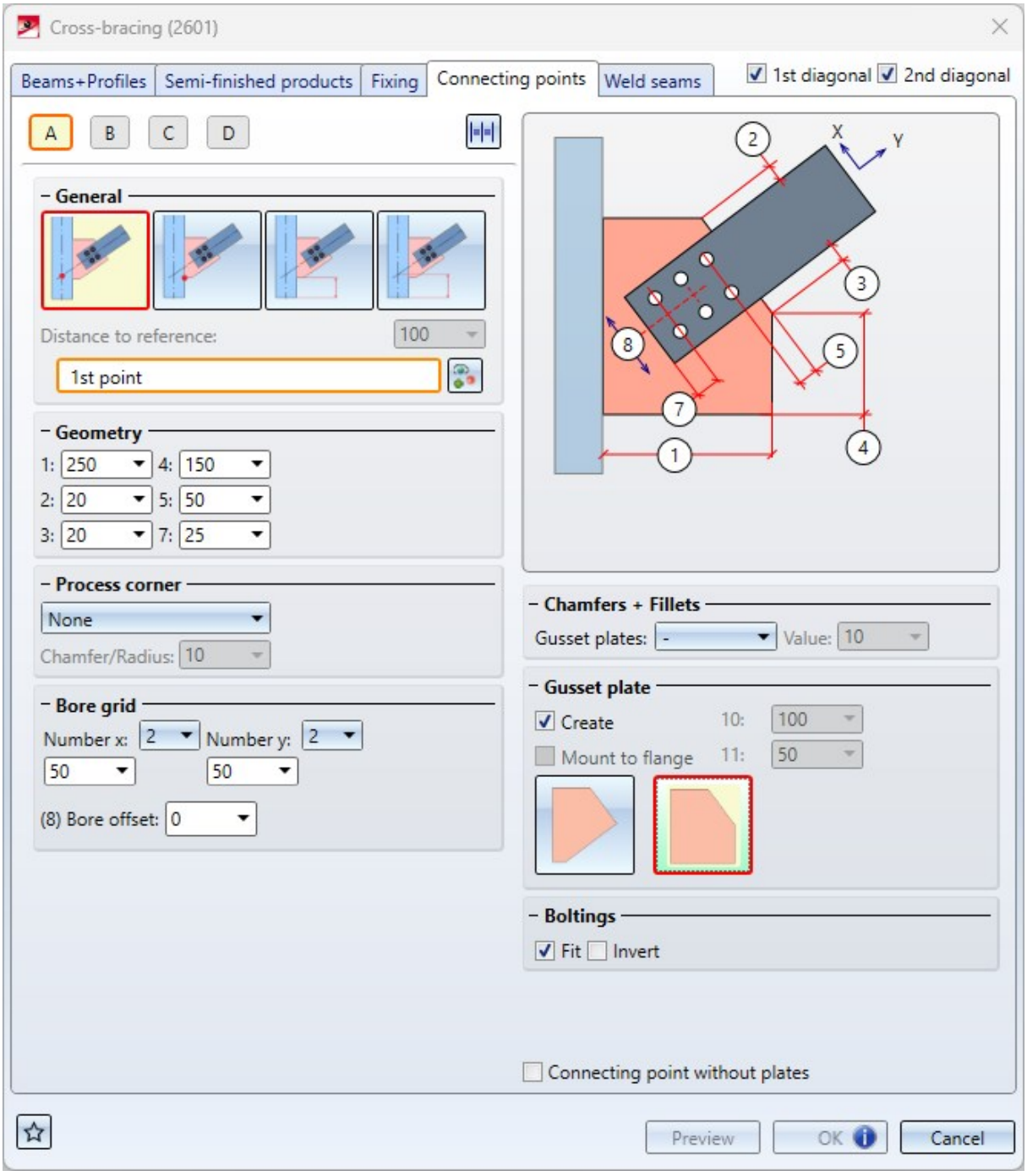


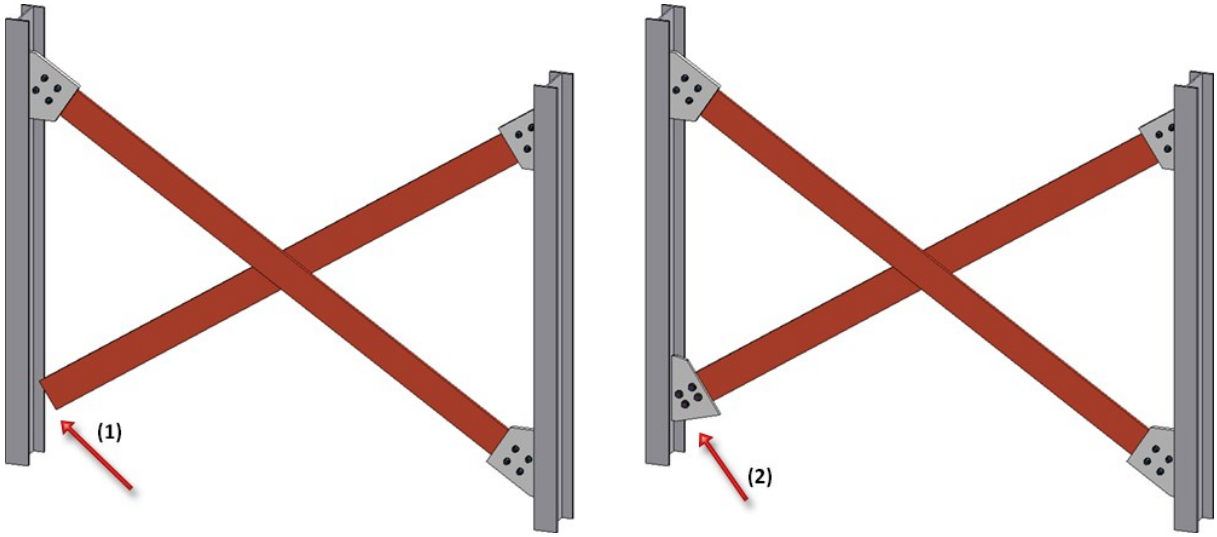
Cross-bracing without gusset plates and connecting plates

If for the connections **Cross-bracing (2601)** and **Cross-bracing (2602)** on the tab **Connecting points** for the connection point the option

Point (as intersection point of axes)

is selected for the connecting point on the tab Connecting points, the insertion at the corresponding end can now also be carried out without gusset plates and connecting plates. To do this, activate the checkbox **Connecting point without plates**. For example, you can subsequently generate a **Gusset plate (2510)** connection at these free ends or install a connection manually.

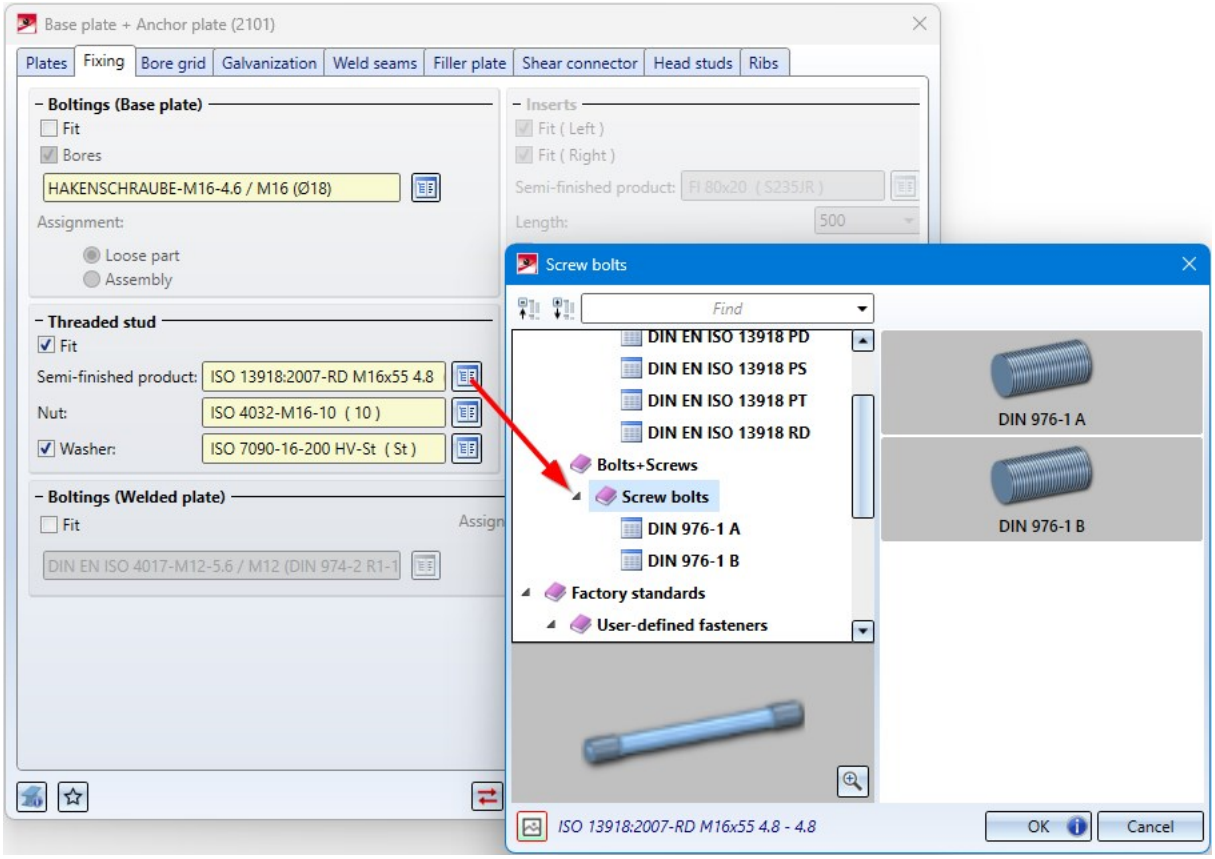




(1) Cross-bracing without plates at connection A, (2) subsequently inserted gusset plate

Base plate + Anchor plate (2101)

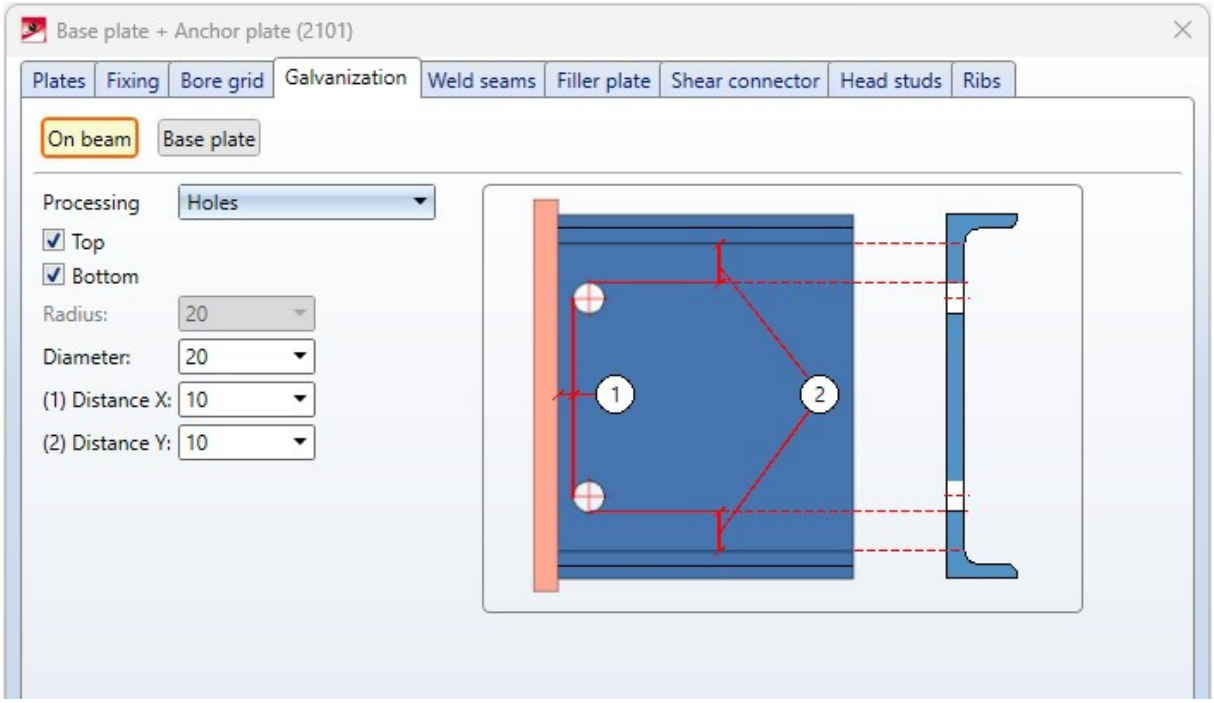
For fastening the anchor plate with screw bolts, it is now also possible to select screw bolts according to DIN 976-1A and 1B.

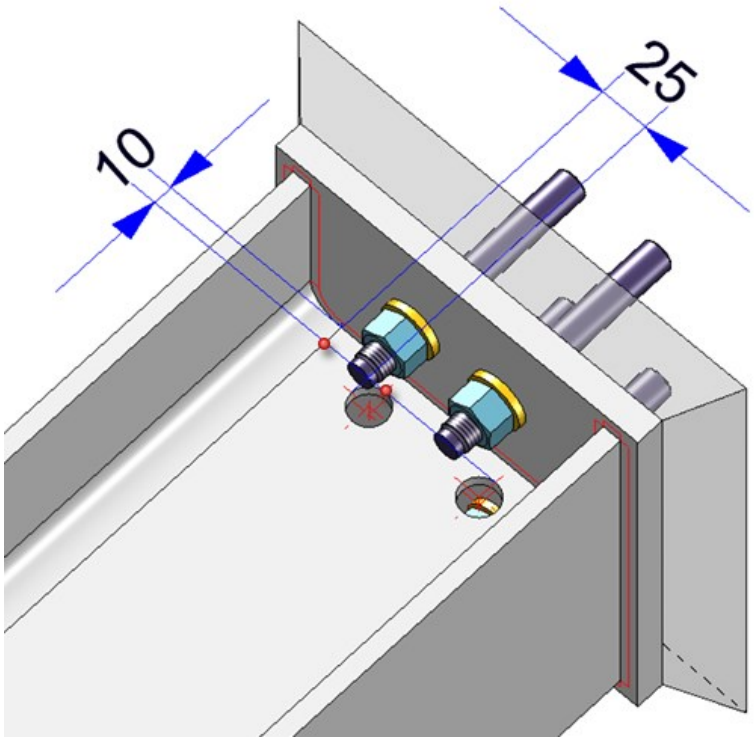


Connections- Galvanization holes on beam

In the settings for the galvanisation holes on the beam, the Y-distance of the holes now refers to the fillet on the beam and not to the flange as before. This affects the following connections:

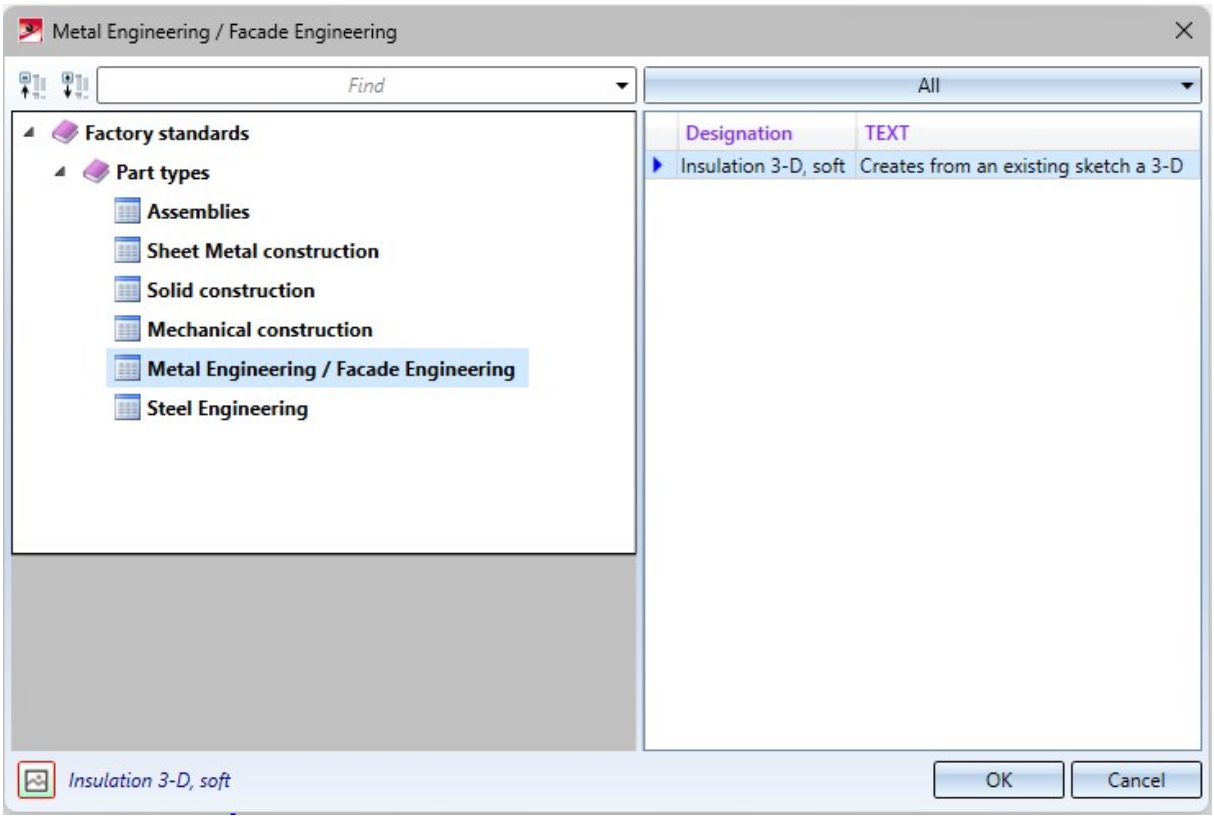
- Base plate + Anchor plate (2101),
- End plate (2102),
- Purlin joint, 2 plates with m itre cut acc. to DAST IH (2201) and
- Beam to web, with 2 plates + stiffener (1211).





Civil Engineering - Part type catalogue 3-D

The function **Civil Engineering - Part type catalogue > 3-D** in the **Civil Engineering functions** docking window at **Steel Engineering > Civil Engineering, general** contains functions that have been replaced by new developments in the meantime and are therefore no longer needed.



In a first step, the entries under

- Sheet Metal Processing,
- Mechanical constructions,
- Metal Engineering / Facade Engineering and
- Steel Engineering

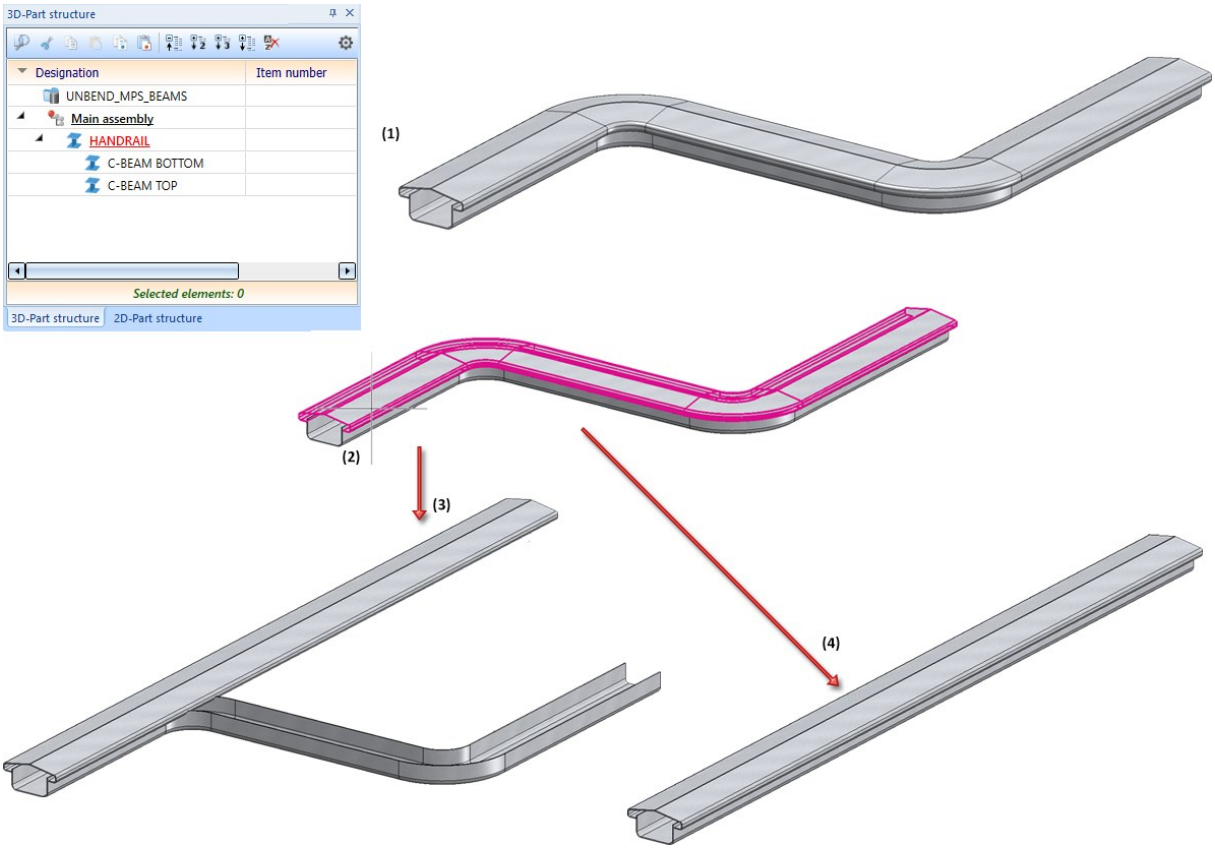
have been removed. However, the structure of the part type catalogue will be retained for the time being so that customer-specific catalogue entries in the above-mentioned areas can continue to be used after a HiCAD update.

Unbending multi-part standard beams

The function **Unbend beam**  has been changed for multi-part beams. Previously, when a sub-beam was selected, only this beam was processed. As of HiCAD 2024, the entire multi-part beam is processed.

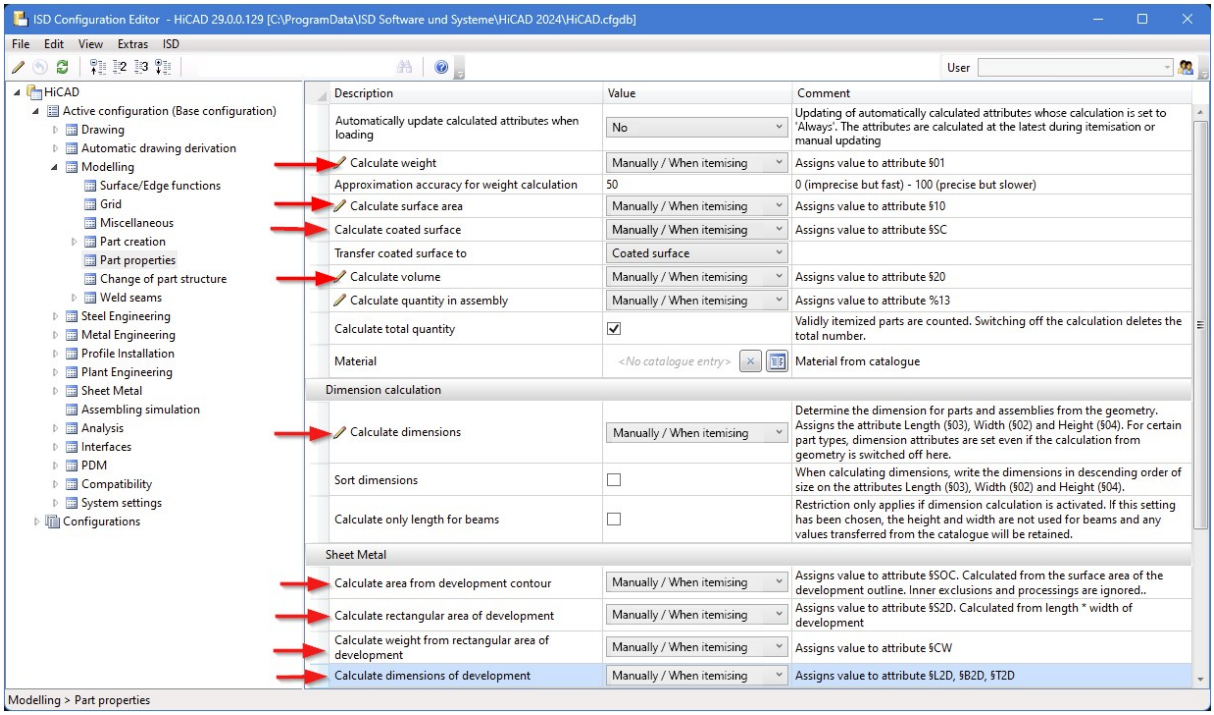
An example:

In the image below, the series beam **ISD Handrail** has been moved along a sketch (1). This has created a multi-part beam. Subsequently, the **Unbend beam** function was called and the upper sub-beam (2) was selected. (3) shows the result before HiCAD 2024, (4) the current result.



Parameter configuration

The default settings of the standard template for **Steel / Metal Engineering** have changed with HiCAD 2024. This affects the settings at **Modelling > Part properties** in the Configuration Editor.



The default setting of the parameters indicated in the image below was previously **Always**.

The default template can be selected either during installation or subsequently with the tool **ParKonfigComp.exe** (or **ParKonfigUser.exe**).

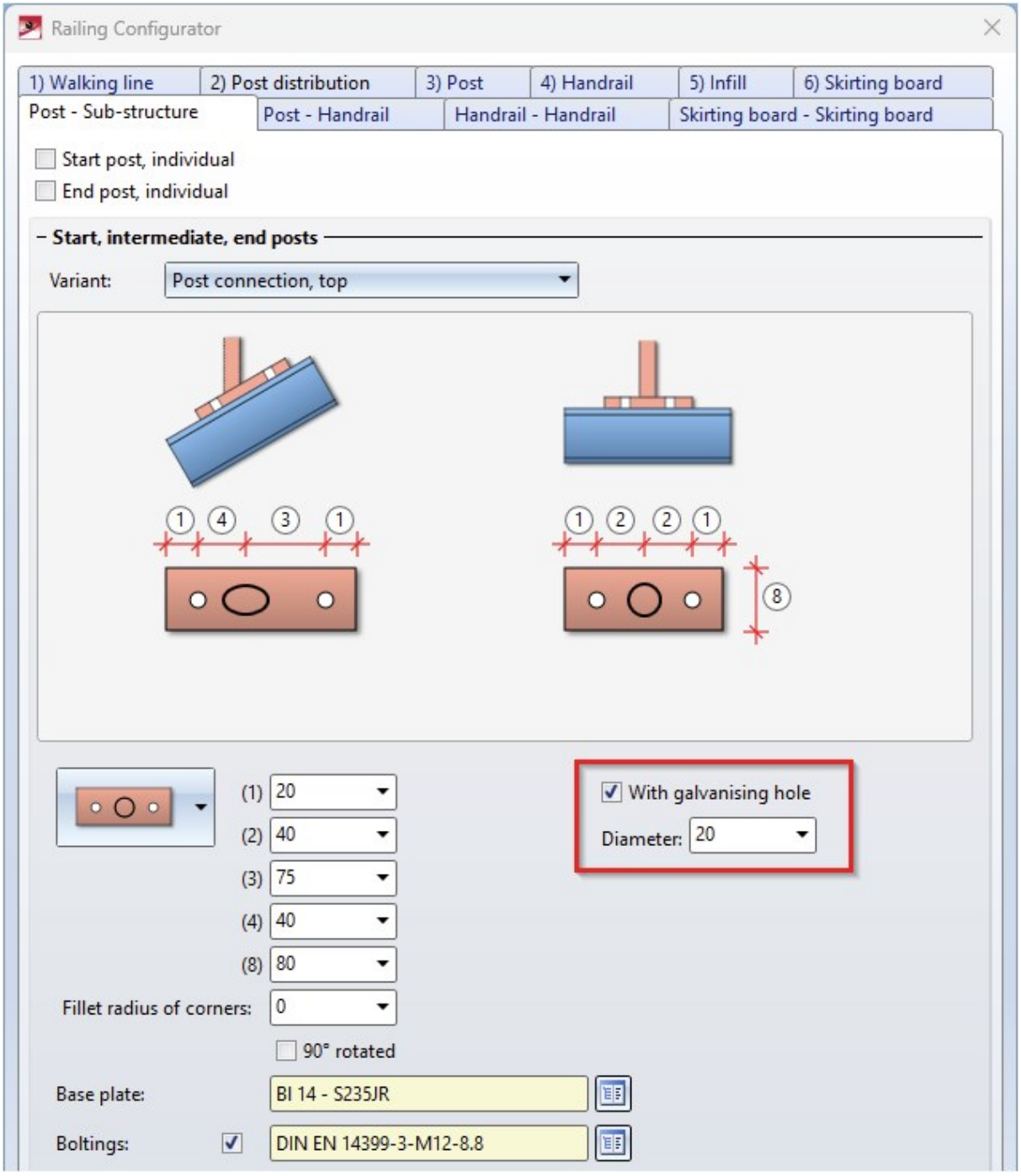
Feature when inserting Rectangular plates

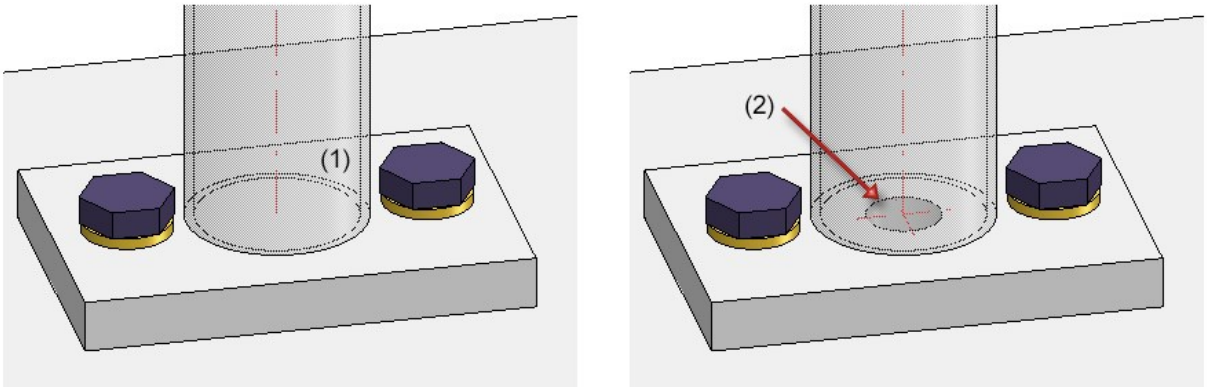
In the dialogue window of the **Rectangular plate**  function, the **Feature** checkbox is no longer available as of HiCAD 2024. This means that a corresponding feature is now always generated when plates are created.

Railing Configurator

Post connection top, with galvanising hole

If the **Post connection, top** variant is selected for fastening the railing posts to the beam, then as of HiCAD 2024 the base plate can be provided with a galvanising hole. The **Post - Sub-structure** tab has been extended for this purpose.



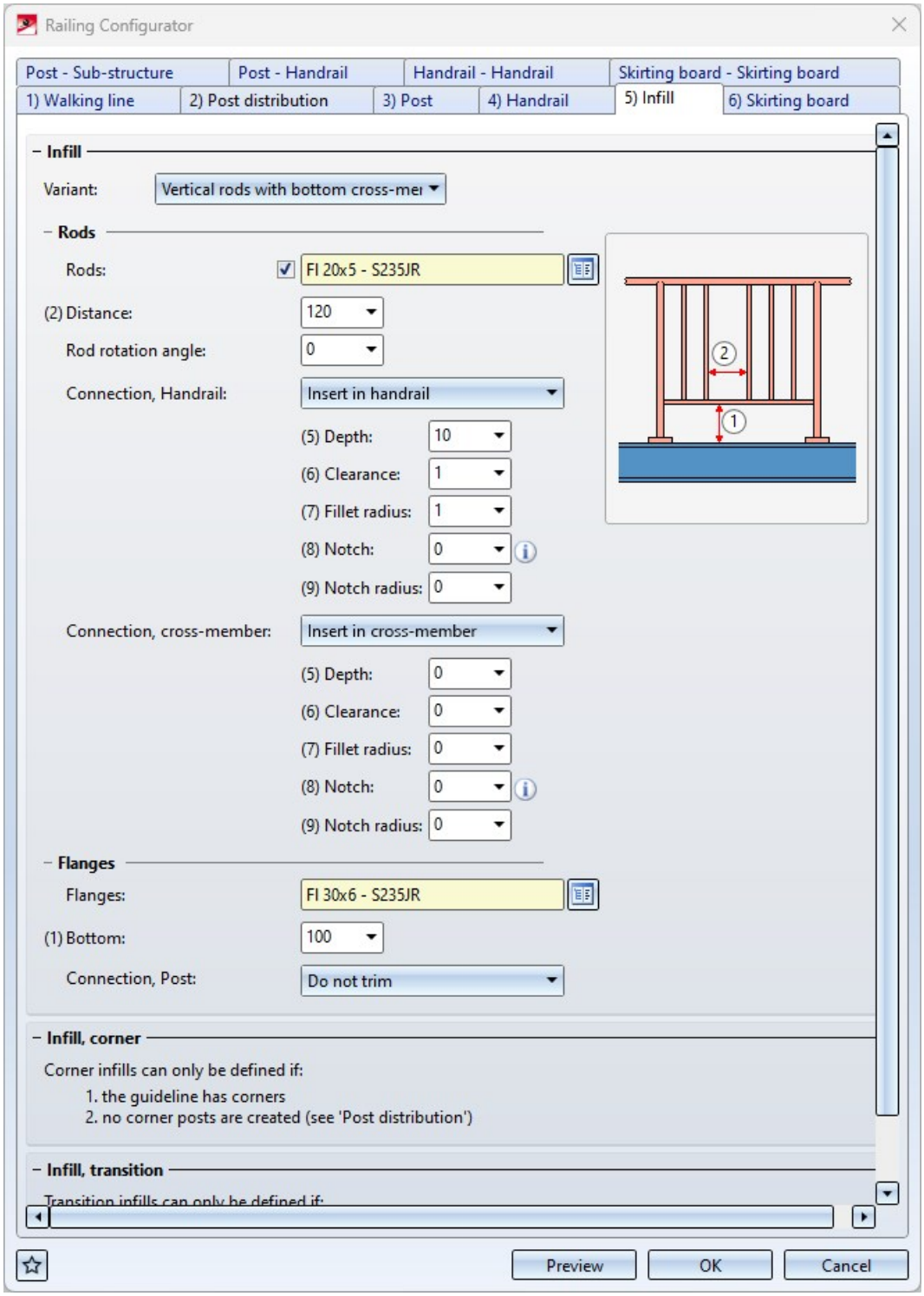


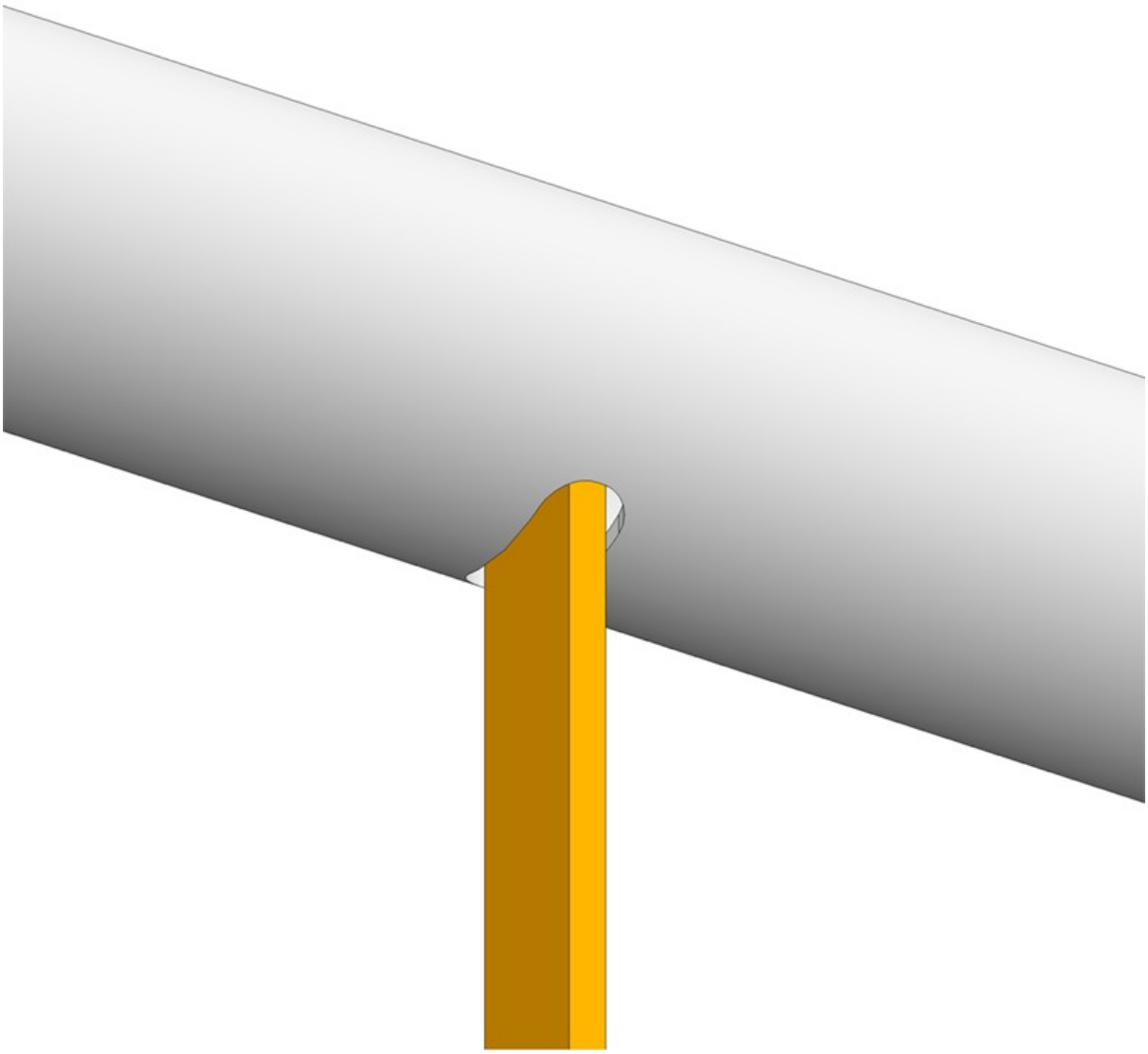
(1) without, (2) with galvanising hole

Infill with bottom cross-member - Insert filling rods

For infills with bottom cross-members it is now also possible to insert the filling rods in handrail or cross-member. Two new variants are available for this purpose:

- Insert in handrail and
- Insert in cross-member.





Filling rod, inserted in handrail

Post-Handrail - Wall console (prefabricated part)

The variant **Wall console (prefabricated part)** has been replaced by a new variant in which the screws and threads for mounting to the handrail can now also be installed.

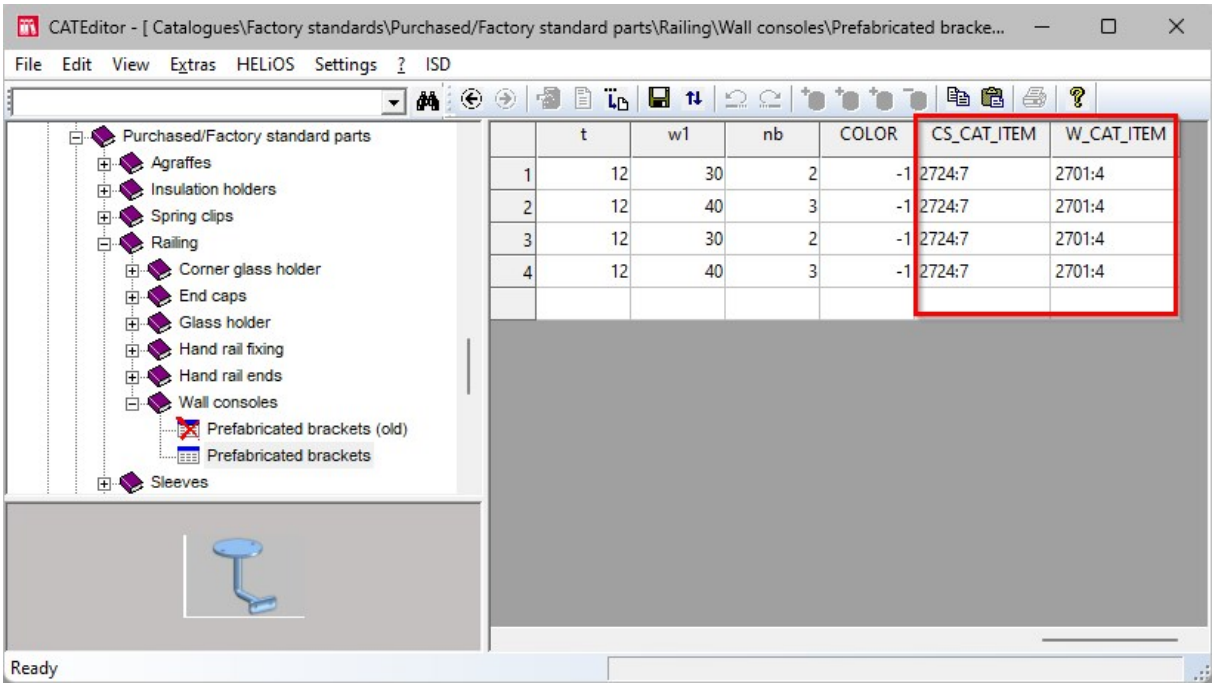
The following has been preset for the bores on the prefabricated element:

- Bores for handrail mounting: Countersink DIN 66, size 5
- Bores for wall mounting: Countersink DIN 74-1 F, size 6

If you want to preset other drillings, you can do this by changing the table **Factory standards > Purchased/Factory standard parts > Railing > Wall consoles > Prefabricated brackets** (RAILING_BRACKET_29_ISD.IPT). There you have to change the columns

- **CS_CAT_ITEM** (Bore for handrail mounting) and
- **W_CAT_ITEM** (Bore for wall mounting)

accordingly.



The first value in each case is the table ID, the second value is the ID of the corresponding data record. The two values are separated by a colon. For example, **2724:7** stands for the table **DIN 66** at **Processings, general > Processing> Countersink** and the data record with the **ID 7**. If you double-click with the cursor in one of the columns, you can directly select the desired table and data record.

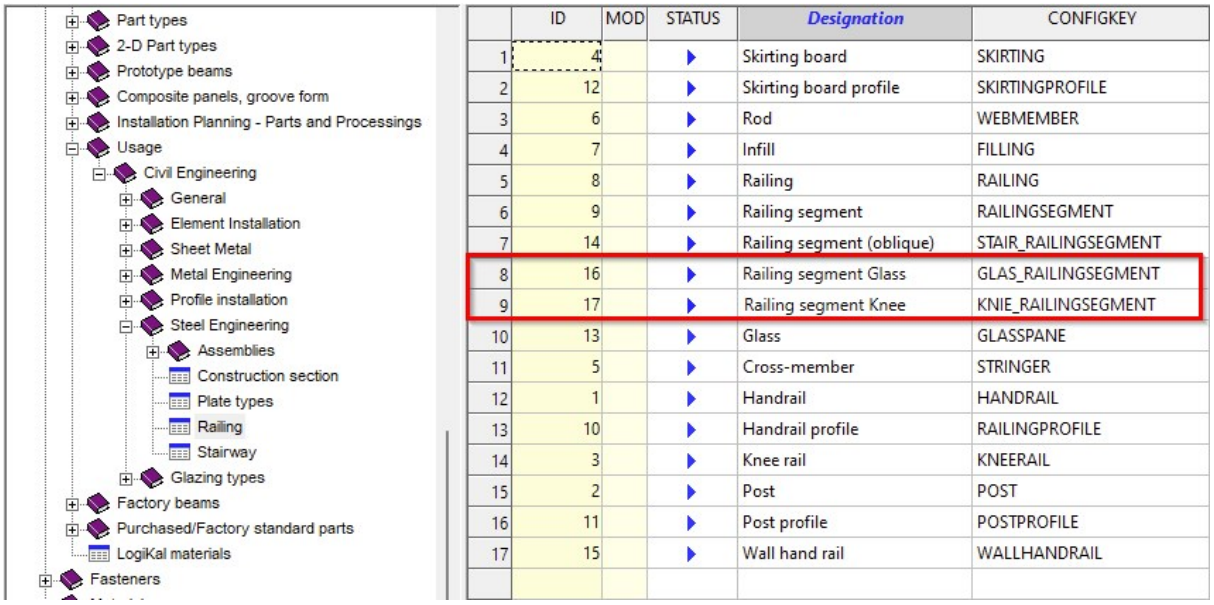
Usage for railing segments

In the automatic drawing derivation, the usage RAILINGSEGMENT has been used up to now for the dimensioning of railing segments created by the railing configurator. However, there is often the wish to dimension the railing segments differently for different railing types, for example, for segments with glass infill or segments with knee rail infill, etc.

From HiCAD 2024 this is now possible. For this purpose, corresponding usages must be defined and associated configurations created whose name contains the expression RAILINGSEGMENT, e.g. GLASS_RAILINGSEGMENT or KNEEL_RAILINGSEGMENT.

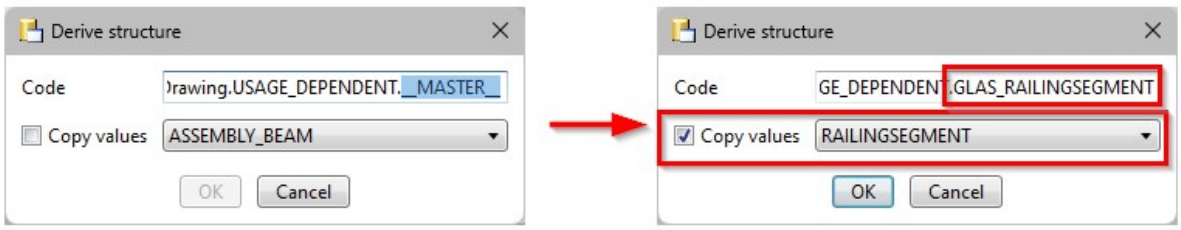
Proceed as follows:

1. You define the desired usages with the Catalogue Editor at **Factory standards > Usage > Civil Engineering > Steel Engineering > Railing**, e.g.

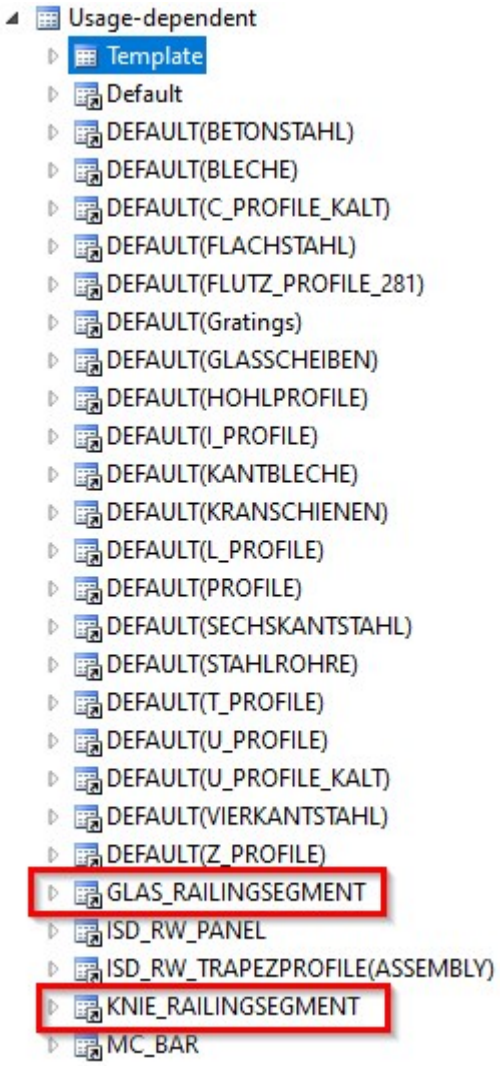


	ID	MOD	STATUS	Designation	CONFIGKEY
1	4			Skirting board	SKIRTING
2	12			Skirting board profile	SKIRTINGPROFILE
3	6			Rod	WEBMEMBER
4	7			Infill	FILLING
5	8			Railing	RAILING
6	9			Railing segment	RAILINGSEGMENT
7	14			Railing segment (oblique)	STAIR_RAILINGSEGMENT
8	16			Railing segment Glass	GLAS_RAILINGSEGMENT
9	17			Railing segment Knee	KNIE_RAILINGSEGMENT
10	13			Glass	GLASSPANE
11	5			Cross-member	STRINGER
12	1			Handrail	HANDRAIL
13	10			Handrail profile	RAILINGPROFILE
14	3			Knee rail	KNEERAIL
15	2			Post	POST
16	11			Post profile	POSTPROFILE
17	15			Wall hand rail	WALLHANDRAIL

2. In the Configuration Manager, you derive the new usages GLAS_RAILINGSEGMENT and KNIE_RAILINGSEGMENT from the usage RAILINGSEGMENT, for example, at **Automatic drawing derivation > Production drawing > Usage-dependent**. (The names must correspond to the entry in the CONFIGKEY column of the above table in the Catalogue Editor). To derive, right-click on **Template** and select **Derive structure**. Instead of _MASTER_, enter the name of the new usage, e.g. GLAS_RAILINGSEGMENT, activate the checkbox **Copy values** and select **RAILINGSEGMENT** as the template.

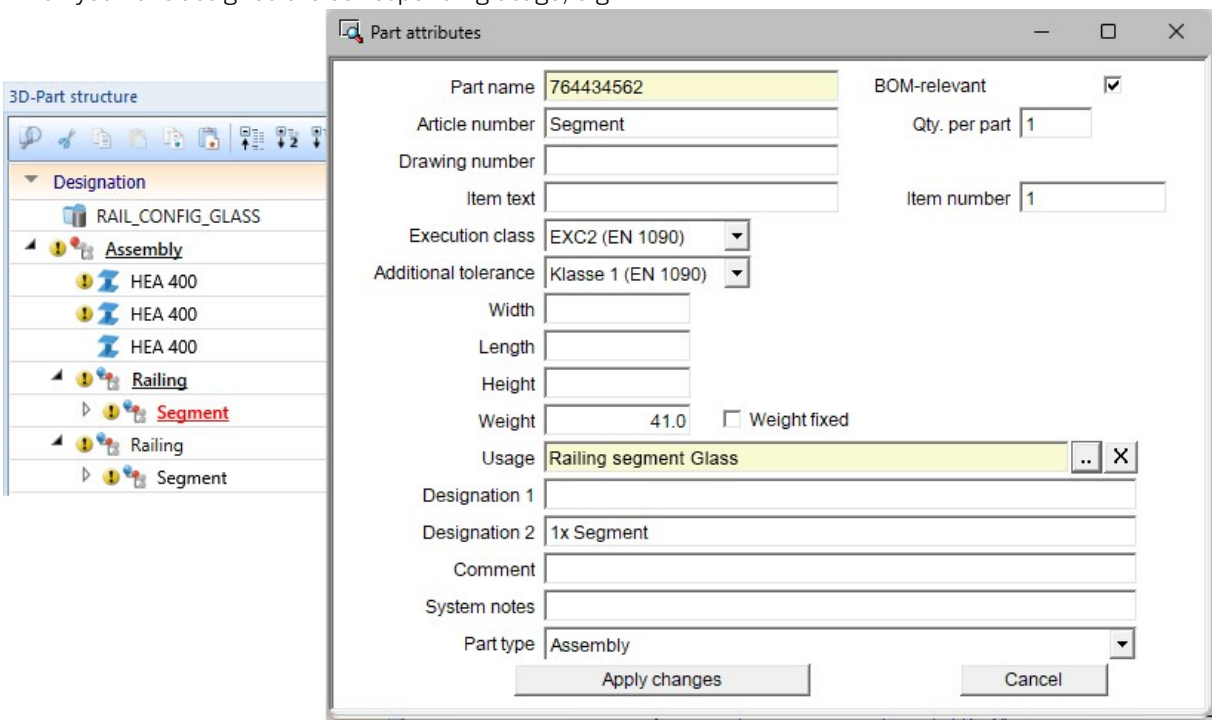


3. With **OK**, the new usage is created.



4. The new usages must now be assigned. To do this, open the entry **Usage assignment**, activate a row in the **Railing** area and click on **New**. A new rowis created. In the column, select the name of the usage in the selection list, e.g. **Railing segment Glass**, in the second column the part type and in the third column the name of the template, e.g. GLAS_RAILINGSEGMENT.Now you can define the dimensioning rules for the new types of use in HiCAD with the dimensioning rule editor. To do this, open the corresponding templates, e.g. **GLAS_RAILINGSEGMENT**, adjust the dimensioning rules accordingly and save the template..

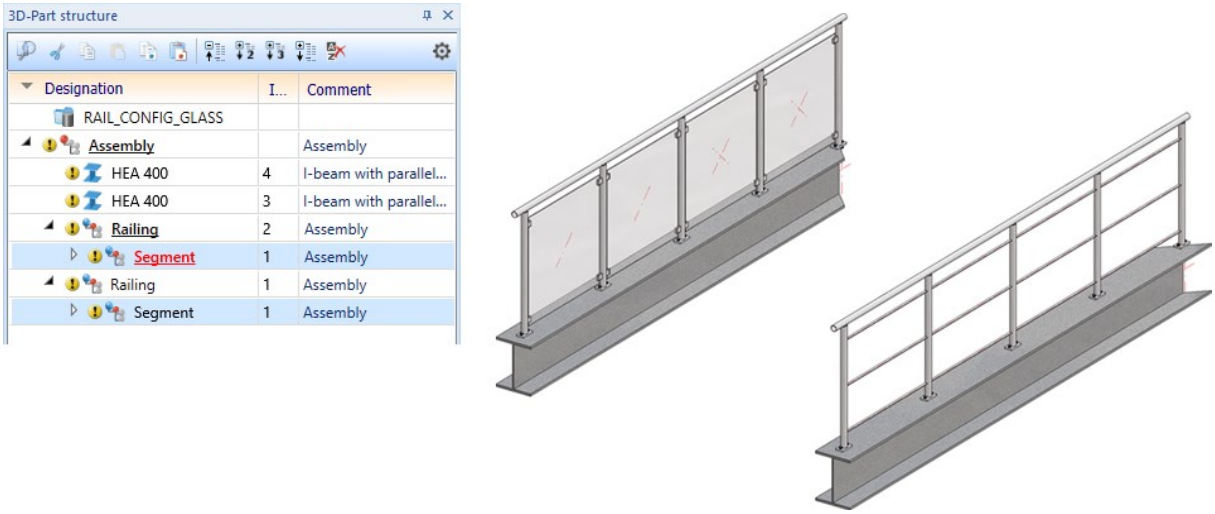
During automatic drawing derivation, these templates are then taken into account for the railing segments to which you have assigned the corresponding usage, e.g.



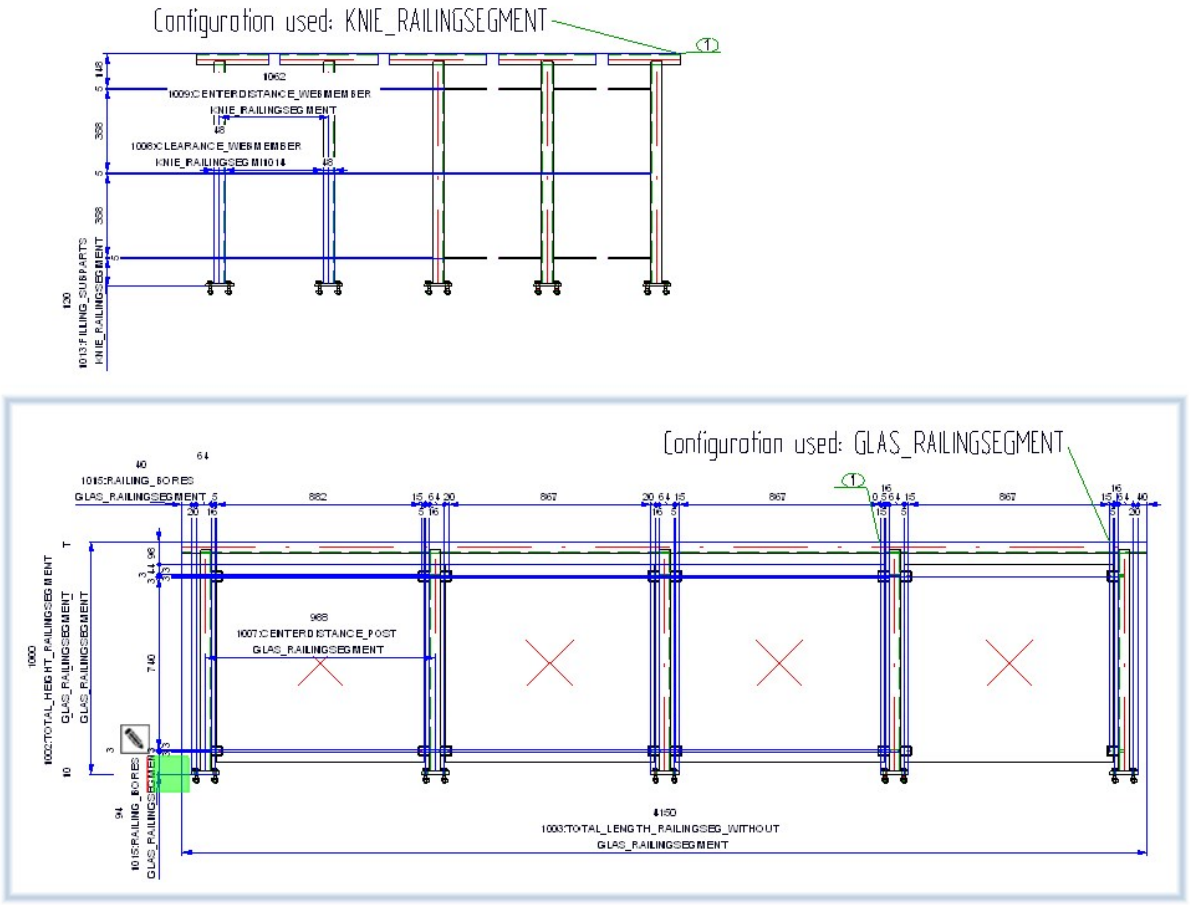
A simple example:

Based on the above procedure, two new usages **Railing segment Glass** and **Railing segment Knee** as well as the corresponding templates **GLAS_RAILINGSEGMENT** and **KNIE_RAILINGSSEGMENT** have been defined. The dimensioning rule sets of these templates were changed and differed.

The example construction contains two railing segments - one with knee rails and one with a glass infill. The corresponding new usages were assigned to these segments.



Example of an automatic drawing derivation



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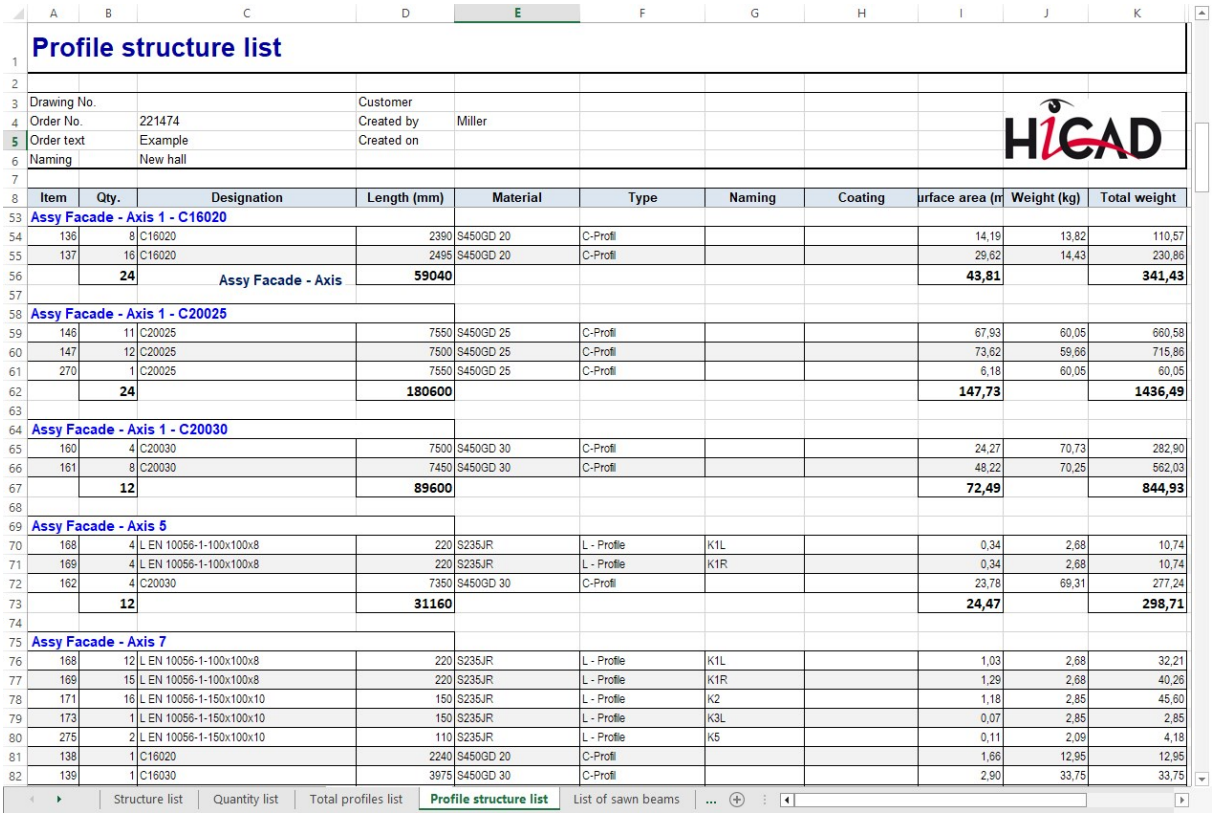
Extension of the BOM template for Steel Engineering

The Excel BOM template for Steel Engineering, HiCAD_Stahlbau.DE.2900.0.xlsx, has been extended. An additional sheet **Profile structure list** is now available. On this sheet, the parts/beams/profiles of the same type are combined and output in a structure list. The profile structure list is a mixture of structure and total profiles list and is ideal for combining steel engineering and profile installation, e.g. for hall construction and industrial facades.

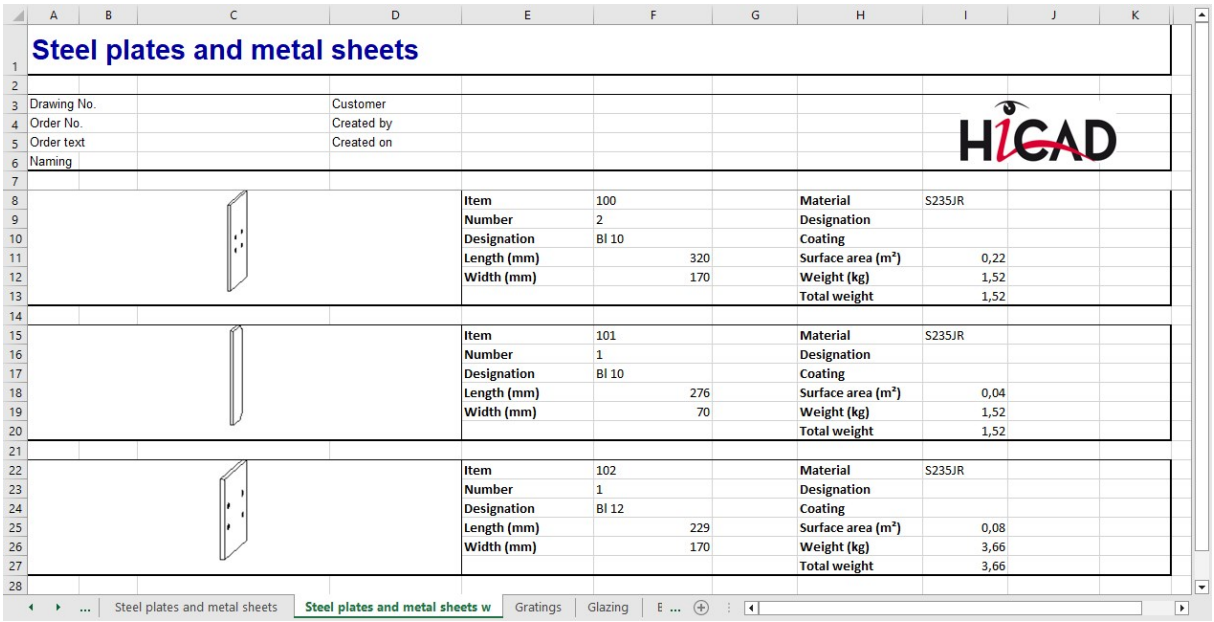


Profile structure list

Drawing No.			Customer							
Order No.		221474	Created by	Miller						
Order text		Example	Created on							
Naming		New hall								
Item	**Qty.**	**Designation**	**Length (mm)**	**Material**	**Type**	**Naming**	**Coating**	**urface area (m**	**Weight (kg)**	**Total weight**
Assy Facade - Axis 1 - C16020										
136	8	C16020	2390	S450GD 20	C-Profil			14,19	13,82	110,57
137	16	C16020	2495	S450GD 20	C-Profil			29,62	14,43	230,86
	24	**Assy Facade - Axis**	**59040**					**43,81**		**341,43**
Assy Facade - Axis 1 - C20025										
146	11	C20025	7550	S450GD 25	C-Profil			67,93	60,05	660,58
147	12	C20025	7500	S450GD 25	C-Profil			73,62	59,66	715,86
270	1	C20025	7550	S450GD 25	C-Profil			6,18	60,05	60,05
	24		**180600**					**147,73**		**1436,49**
Assy Facade - Axis 1 - C20030										
160	4	C20030	7500	S450GD 30	C-Profil			24,27	70,73	282,90
161	8	C20030	7450	S450GD 30	C-Profil			48,22	70,25	562,03
	12		**89600**					**72,49**		**844,93**
Assy Facade - Axis 5										
168	4	L EN 10056-1-100x100x8	220	S235JR	L - Profile	K1L		0,34	2,68	10,74
169	4	L EN 10056-1-100x100x8	220	S235JR	L - Profile	K1R		0,34	2,68	10,74
162	4	C20030	7350	S450GD 30	C-Profil			23,78	69,31	277,24
	12		**31160**					**24,47**		**298,71**
Assy Facade - Axis 7										
168	12	L EN 10056-1-100x100x8	220	S235JR	L - Profile	K1L		1,03	2,68	32,21
169	15	L EN 10056-1-100x100x8	220	S235JR	L - Profile	K1R		1,29	2,68	40,26
171	16	L EN 10056-1-150x100x10	150	S235JR	L - Profile	K2		1,18	2,85	45,60
173	1	L EN 10056-1-150x100x10	150	S235JR	L - Profile	K3L		0,07	2,85	2,85
275	2	L EN 10056-1-150x100x10	110	S235JR	L - Profile	K5		0,11	2,09	4,18
138	1	C16020	2240	S450GD 20	C-Profil			1,66	12,95	12,95
139	1	C16030	3975	S450GD 30	C-Profil			2,90	33,75	33,75

Structure list | Quantity list | Total profiles list | **Profile structure list** | List of sawn beams

Also new is the Excel sheet **Steel plates and metal sheets with image**:



Steel plates and metal sheets

Drawing No.		Customer			
Order No.		Created by			
Order text		Created on			
Naming					

Item	100		**Material**	S235JR	
Number	2		**Designation**		
Designation	Bl 10		**Coating**		
Length (mm)	320		**Surface area (m²)**	0,22	
Width (mm)	170		**Weight (kg)**	1,52	
			Total weight	1,52	

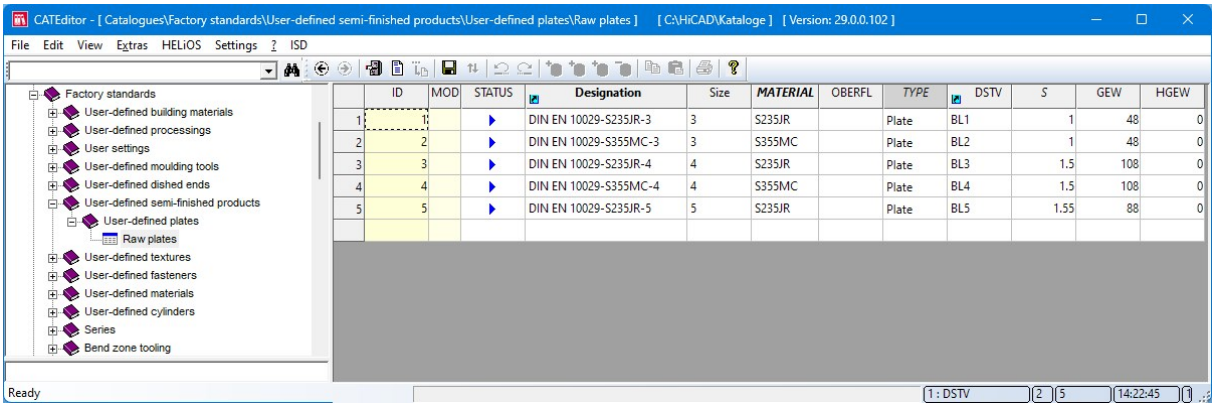
Item	101		**Material**	S235JR	
Number	1		**Designation**		
Designation	Bl 10		**Coating**		
Length (mm)	276		**Surface area (m²)**	0,04	
Width (mm)	70		**Weight (kg)**	1,52	
			Total weight	1,52	

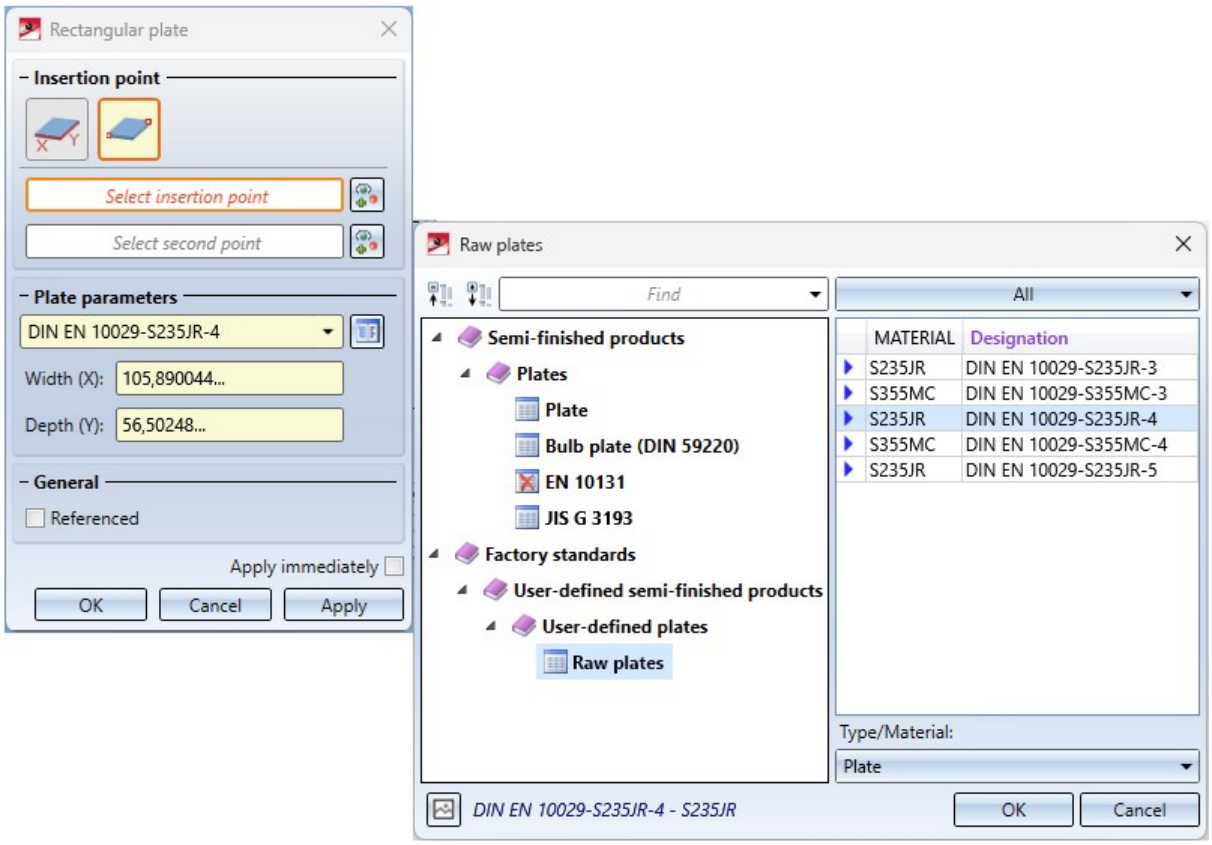
Item	102		**Material**	S235JR	
Number	1		**Designation**		
Designation	Bl 12		**Coating**		
Length (mm)	229		**Surface area (m²)**	0,08	
Width (mm)	170		**Weight (kg)**	3,66	
			Total weight	3,66	

Steel plates and metal sheets | **Steel plates and metal sheets w** | Gratings | Glazing

User-defined tables with Steel Engineering plates

At **Factory standards > User-defined semi-finished products > User-defined plates** you now have the possibility to create tables with your own steel plates. These tables are then also offered for selection via the function **Steel Engineering > Plate, new > Rectangular plate**  .





Simply copy a suitable table from the catalogue **Semi-finished products > Plates** into the catalogue **Factory standards > User-defined semi-finished products > User-defined plates** and edit it.

Drawing Management

Service Pack 1 2024 (V 2901)

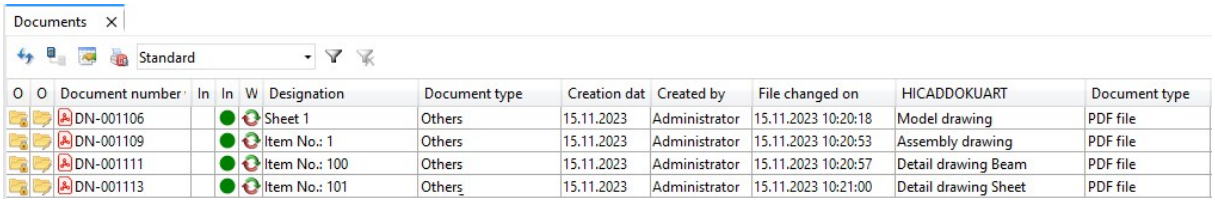
Documents for general documents

Previously, only the active sheet was taken into account when creating external documents, such as PDF files. From SP1, this can also be done for all sheets. For this purpose, the Configuration Editor at **PDM > Drawing Management > External production documents** has been expanded to include the following settings:

- **Creation of external documents**
  This parameter determines whether external documents are to be created for the active sheet only or for all sheets. The default setting is **Active sheet**.
- **HELiOS attribute for HiCAD sheet names**
  Here you specify which HELiOS attribute the HiCAD sheet name should be assigned to. The HELiOS attribute **BENENNUNG** (Designation) is preset.

Two new document attributes are also available in this context:

- **HICADDOKUART**
  When creating external documents, e.g. PDF files, the identification of the document, e.g. as a PDF file, is assigned to the HELiOS attribute **DOKUART**. The **DOKUART** attribute of the drawing from which the external documents are derived is assigned to the **HICADDOKUART** attribute of the external documents, e.g.:



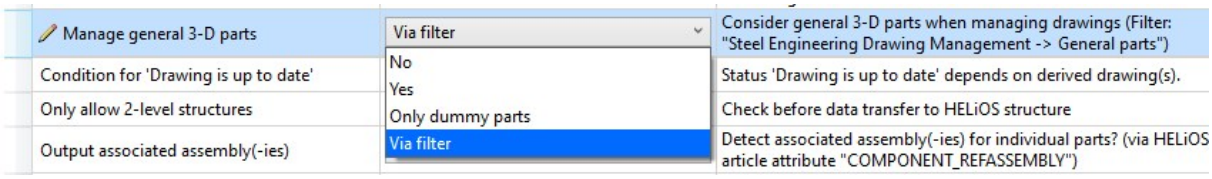
O	O	Document number	In	In	W	Designation	Document type	Creation dat	Created by	File changed on	HICADDOKUART	Document type
		DN-001106				Sheet 1	Others	15.11.2023	Administrator	15.11.2023 10:20:18	Model drawing	PDF file
		DN-001109				Item No.: 1	Others	15.11.2023	Administrator	15.11.2023 10:20:53	Assembly drawing	PDF file
		DN-001111				Item No.: 100	Others	15.11.2023	Administrator	15.11.2023 10:20:57	Detail drawing Beam	PDF file
		DN-001113				Item No.: 101	Others	15.11.2023	Administrator	15.11.2023 10:21:00	Detail drawing Sheet	PDF file

- **HICADPROJNUMMER**
  This attribute is assigned the project of the HiCAD document..

If the new attributes are to be displayed in title blocks or in the HELiOS results lists, you must adjust them manually.

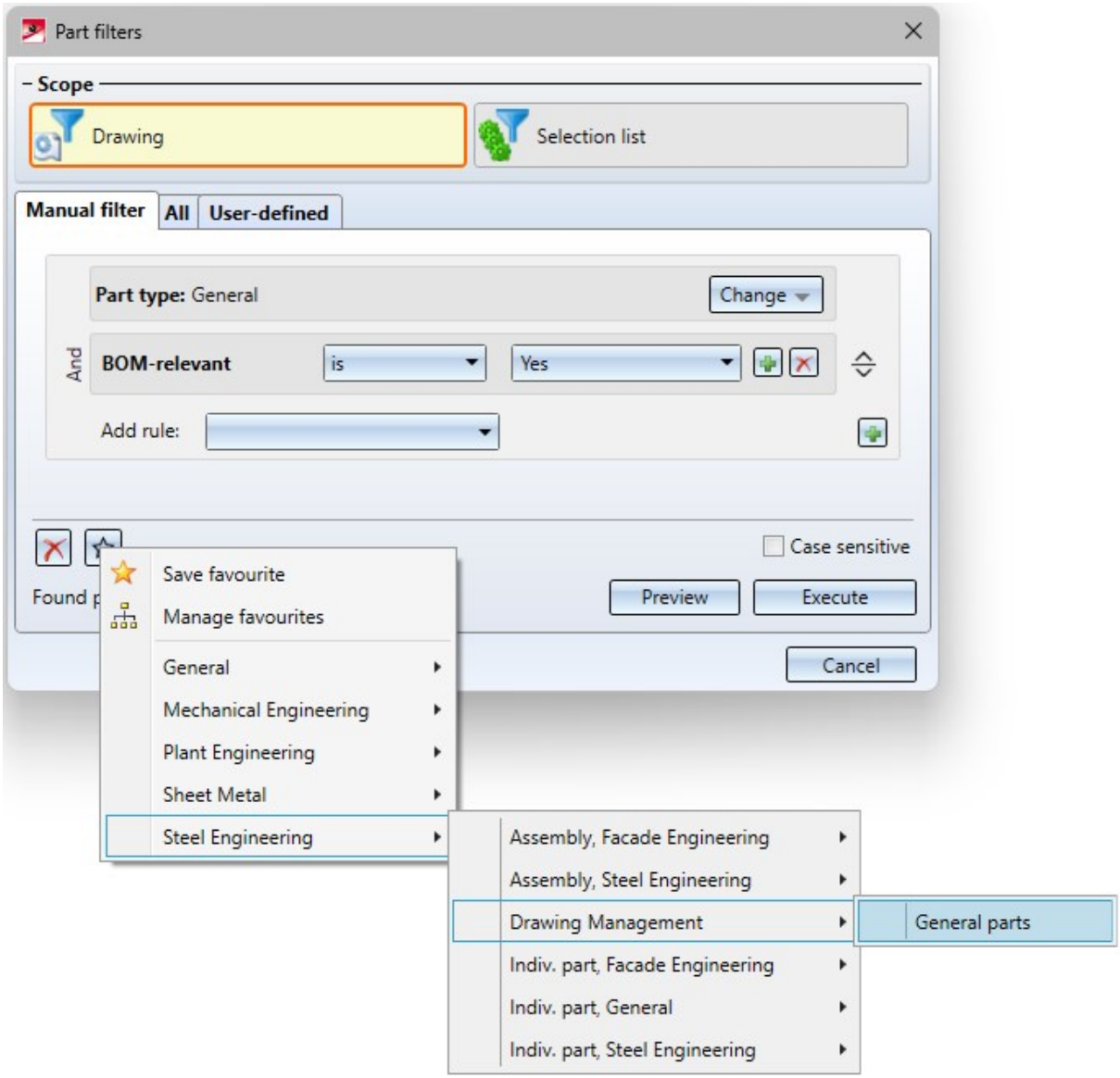
Managing general 3-D parts via part filter

In the Configuration Editor at **PDM > Drawing Management**, you can set whether general 3-D parts should also be taken into account when managing drawings. With SP1, the new option **Via filter** is available here.



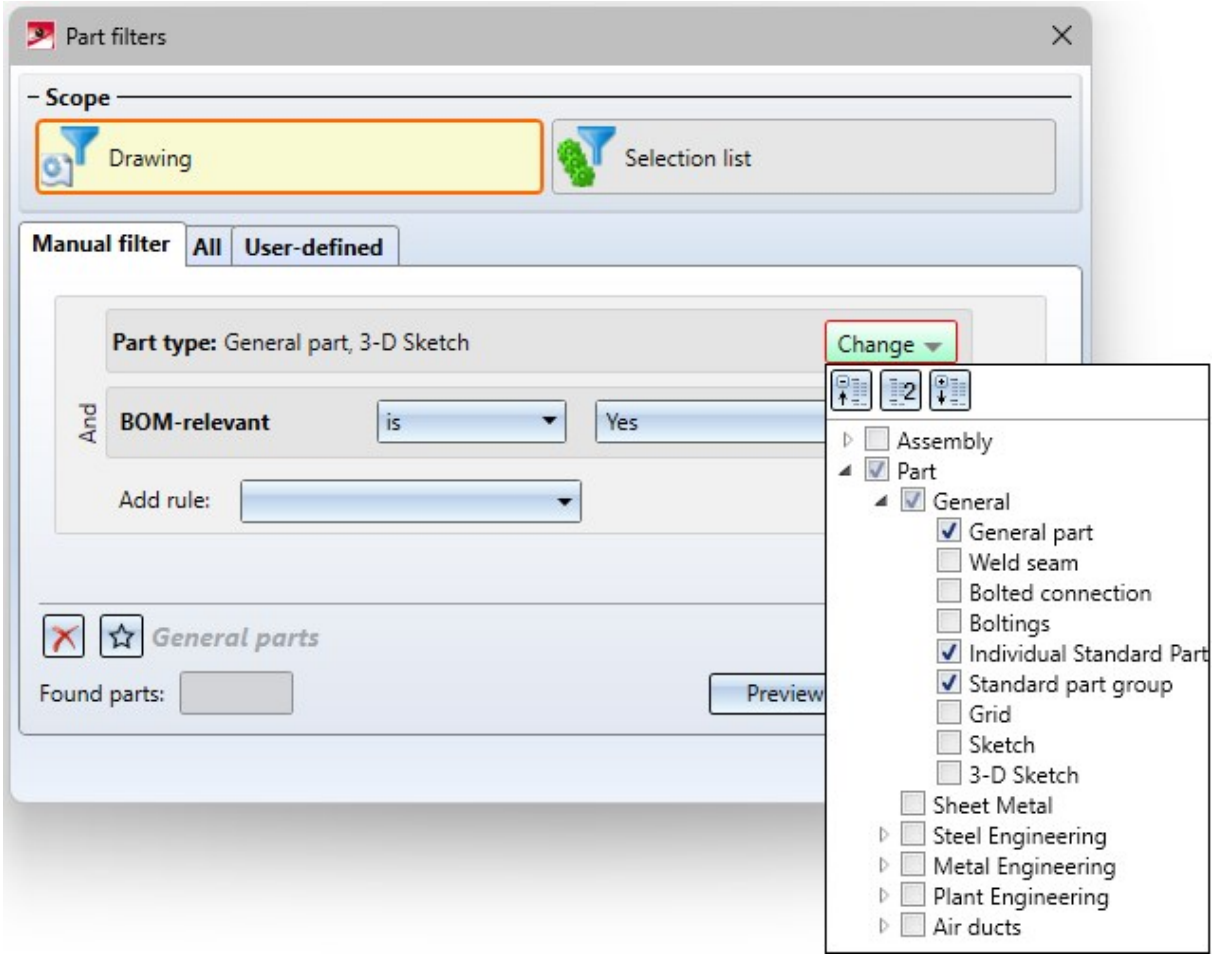
With this setting, only the general 3-D parts that meet the part filters defined in the Favourites file **Steel Engineering > Drawing Management > General parts** (BIM-3DPartFilter.xml) are taken into account.

The preset part filters can be customized using the **Find** function (in the transparent toolbar).



ISD default setting for general parts in the Drawing Management

If, for example, you also want individual standard parts and standard part groups to be taken into account, open the Favourites file **Steel Engineering > Drawing Management > General parts** as shown above and change the file accordingly.

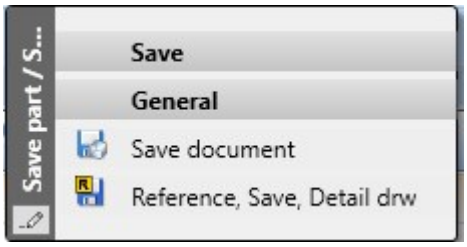


Then save the modified Favourites file under the same name.

Simplified handling when referencing in project structures

When working with referenced detail drawings, project structures are often used, i.e. main projects and sub-projects. This has been simplified with SP1. The project of the article master is now automatically entered as the project of the documents.

This affects the functions under **Reference part, Save, Detail drawing**  in the context menu of assemblies.



Major Release 2024 (V 2900)

Remove parts from production drawings

Instead of detailed drawings, overview drawings are often created in practice, which contain, for example, all the beams or plates used in the model. If the initial model is changed, then existing overview drawings can be updated, but the subsequent creation of a detail drawing for a beam or plate shown in the overview is not directly possible. Sometimes, however, it is desired to remove certain parts from the overview drawing and create a detail drawing afterwards. In this case the function **Remove active part list from production drawing** provides a remedy.

Proceed as follows:

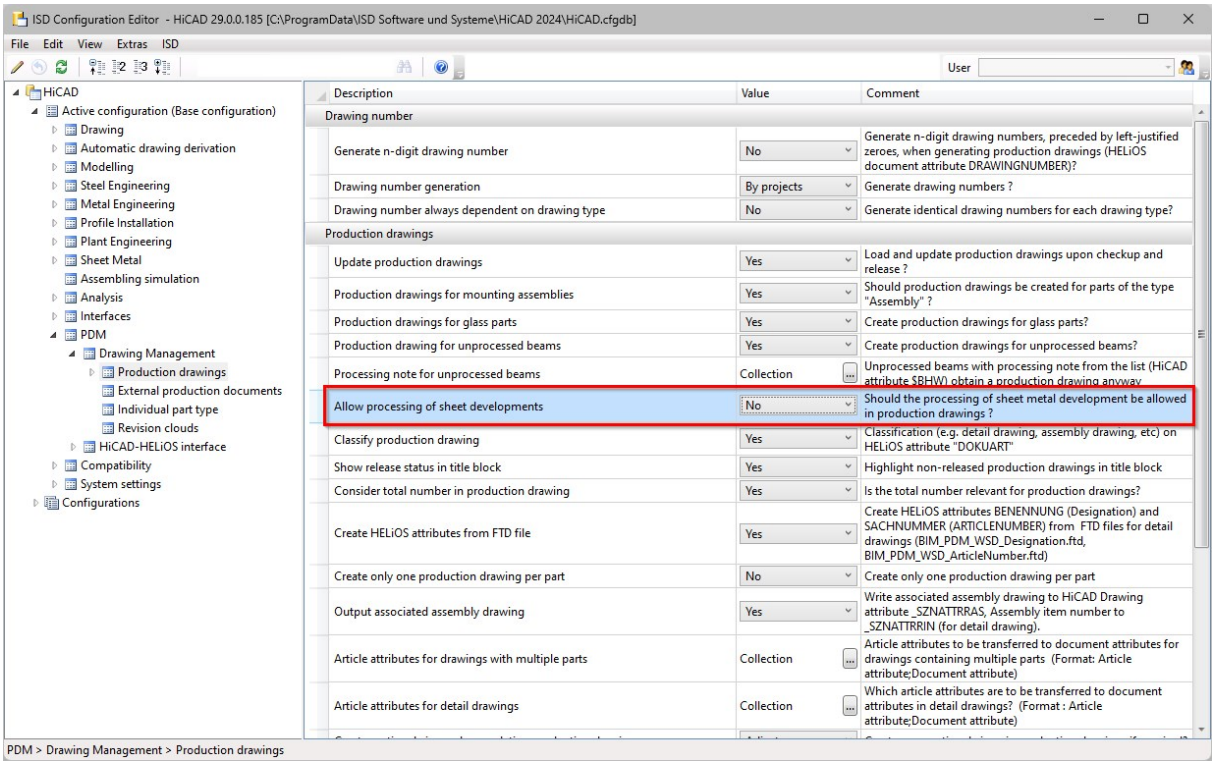
- Load the overview drawing and select the parts to be removed from the overview in the model view of this drawing.
- Call up the function **Remove active part list from production drawing**. The parts are removed from the overview.
- Save and close the drawing.
- If you then call the Drawing  or Manual settings  function in the original drawing, the parts deleted from the overview have the link class **Without drawing**.

The function **Remove - Active part list**  can only be called up if a corresponding overview drawing is open.

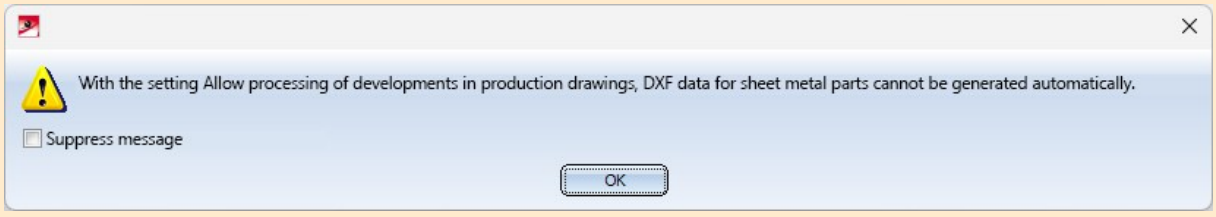
Processing sheet metal developments in production drawings

When working with sheets without using the Drawing Management, the sheet development can be processed without changing the original sheet. This was previously not possible in the Drawing Management, i.e. developments in the production drawing were blocked for editing. From HiCAD 2024 this is now supported. For this purpose, the new parameter **Allow processing of sheet developments** is available in the Configuration Editor at **PDM > Drawing derivation > Production drawings**. The default setting is **No**.

If the setting is set to **Yes**, then you have the option of subsequently processing the development in the production drawing, for example, filleting or chamfering sheet edges etc. or inserting holes. The original model in the model drawing remains unchanged.



If the parameter is set to **Yes**, external part data in DXF format for sheet metal parts cannot be created automatically. This means: If the configuration parameter **Create DXF data** under **External part data** (PDM > Drawing Management > External production documents...) is set to **Yes**, it is not evaluated. In this case, the message shown appears when HiCAD is started.



Releasing assemblies with already released parts

If an assembly contains sub-parts whose drawings have already been released, then as of HiCAD 2024 the drawings of these parts are no longer loaded in the background when the assembly is released. This leads to a performance increase.

Automatic BOMs for itemised source models

Excel BOMs can now be created and managed automatically for model drawings that are itemised source models. For this purpose, the settings in the Configuration Editor at **PDM > Drawing Management > External production documents** have been extended.

Create BOMs

This setting determines whether or not Excel BOMs should be automatically generated for certain model drawings.

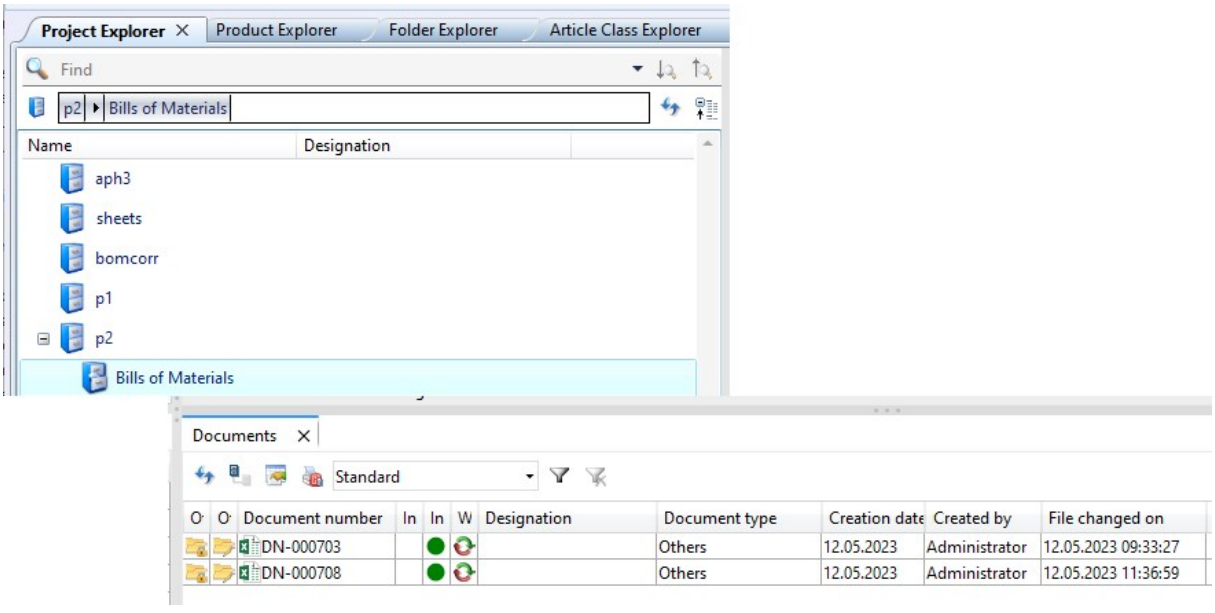
The following settings are possible:

- **None**
  Excel BOMs are not generated automatically. This is the default setting.
- **Upon Checkup and Release**
  The Excel BOMs are created automatically when checking and releasing drawings.
- **Upon creation and update**
  The Excel BOMs are created automatically when creating and updating drawings.
- **Automatically upon Release**
  The Excel BOMs are created automatically upon release of drawings.

List of model drawings with external BOM

Here you define for which drawings external BOMs are to be created and managed. At present, this is only possible for itemised source models, i.e. for the model drawings in which the parts are itemised (see document attribute ITEMISATIONMODEL).

In HELiOS, the generated BOMs (Document type **Others**) are stored in the **Bills of Materials** subfolder of the respective project, e.g.



When saving the itemised source model, these BOMs are automatically updated.

Please note:

- The templates HiCAD-DB_Stahlbau_BIM.DE.2900.0.xlsx and HiCAD-DB_Stahlbau_BIM.rm_settings are used for BOM creation.
- For parts managed with the Drawing Management, the HiCAD attribute **Item text $INTXT** is assigned the HELiOS attribute **COMPONENT_ITEMNR_TEXT**. For external parts / standard parts, the HiCAD attribute **$INXT** is assigned the item number of the part..

Updating manually created production drawings

Production drawings that are created manually with the function **Drawing Management > Workshop > Drawing > Manual settings** can now also be updated with this function when changes are made in the model drawing.

Previously, the **Drawing Management > Workshop > Drawing** function had to be used for this.

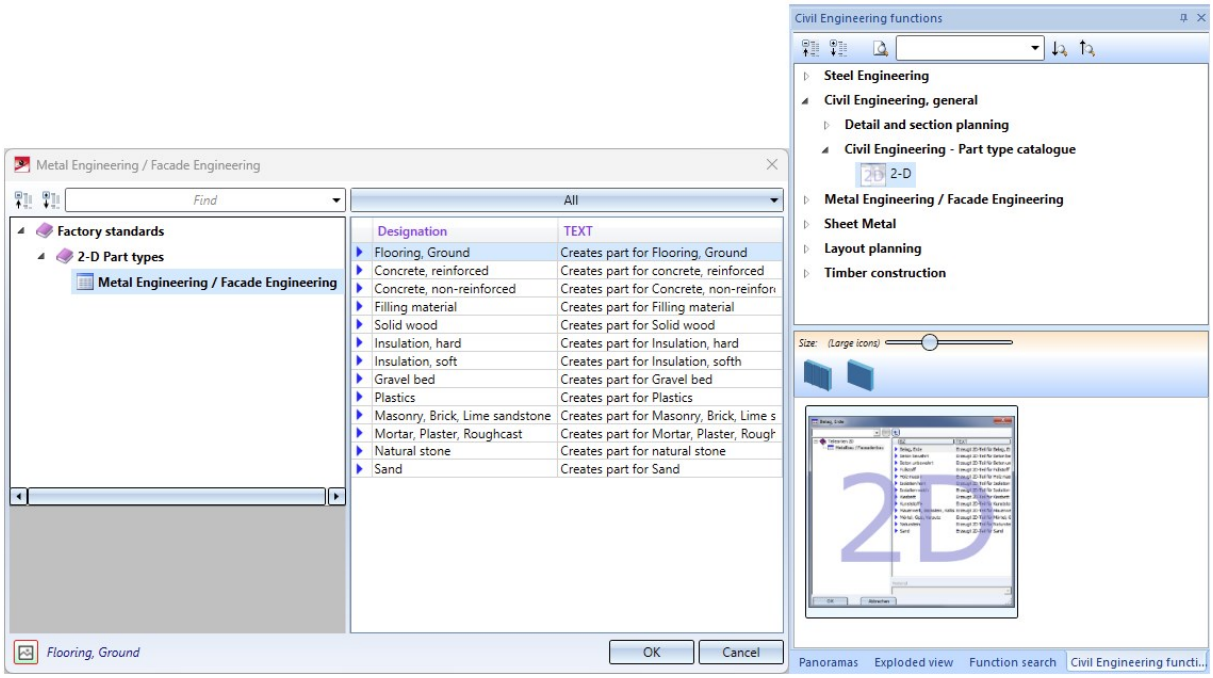
Metal Engineering

Service Pack 1 2024 (V 2901)

Civil Engineering - Part type catalogue, 3-D

The **Civil Engineering functions** docking window previously contained functions at **Civil Engineering, general > Civil Engineering - Part type catalogue** that have been replaced by new developments and are no longer needed.

The 3-D group was therefore removed from this sub-menu with HiCAD 2024 Service Pack 1.

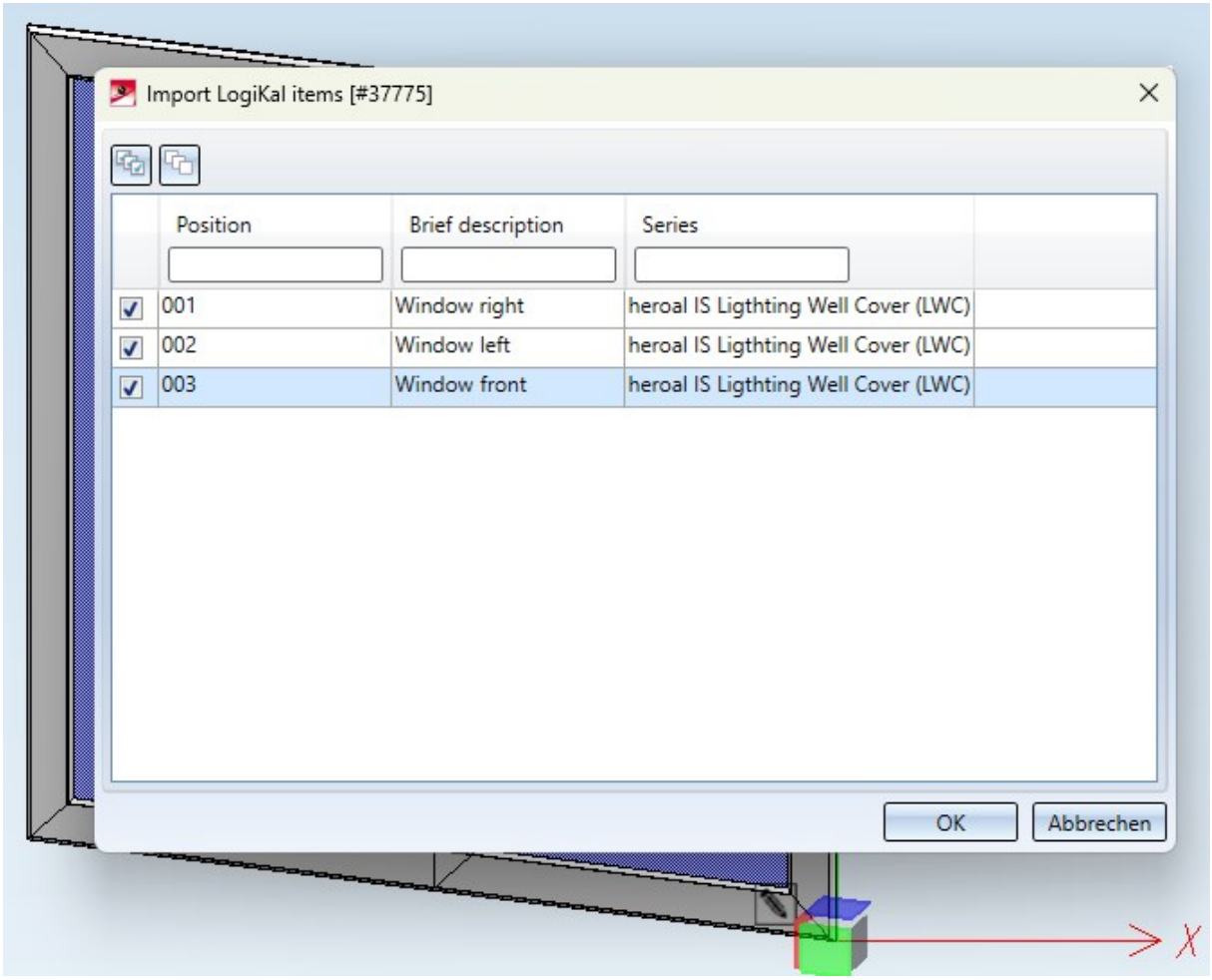


This also eliminates the menu item **Part type (Create part via part type catalogue, as main part)** in the **New** function group of the **Metal Engineering** Ribbon tab and in the context menu of the right mouse button.

Major Release 2024 (V 2900)

Multiple selection of items when importing from LogiKal to HiCAD

When importing several LogiKal items to HiCAD via **Import > Several facades+inserts** , a new selection dialogue appears.



Please note:

In earlier HiCAD versions the function call was called **All facades+inserts**. This imported all positions of a selected LogiKal project without de-/selection options.

Replacing individual profiles

Previously the function **Process > Exchange via LogiKal, with facade profile** only applied to multi-part facade engineering profiles that were installed via the function **New > Insert facade profile** .

With HiCAD 2024 this has been extended to single profiles whose installation was done via **New > Individual beam/-profile** or **New > Window/door profile**.

Extended transfer of LogiKal attributes

The attribute mapping between HiCAD and LogiKal, which you can specify via the Configuration Editor, has been extended by the Text attributes **[Profile] Colour** and **[Glass] Name** and by the Integer attributes **[Position] Quantity** and **[Insertion] escape door function**.

Note on Integer attributes:

As with Double number attributes, integer number attributes are not transmitted if the value coming from LogiKal cannot be converted into a number. I.e.: If the value of the piece count item attribute should not be transferred as an integer from LogiKal for some reason (e.g. "0.5"), then no attribute value will be transferred to HiCAD in this case.

Note on "[Insertion] Escape door function":

Three LogiKal escape door attributes are checked for determination. If one of the three LogiKal attributes says "Escape door = yes", the configured HiCAD (Integer) attribute is set to "1". If all three LogiKal attributes state "Escape door=No", then the HiCAD attribute is set to "0".

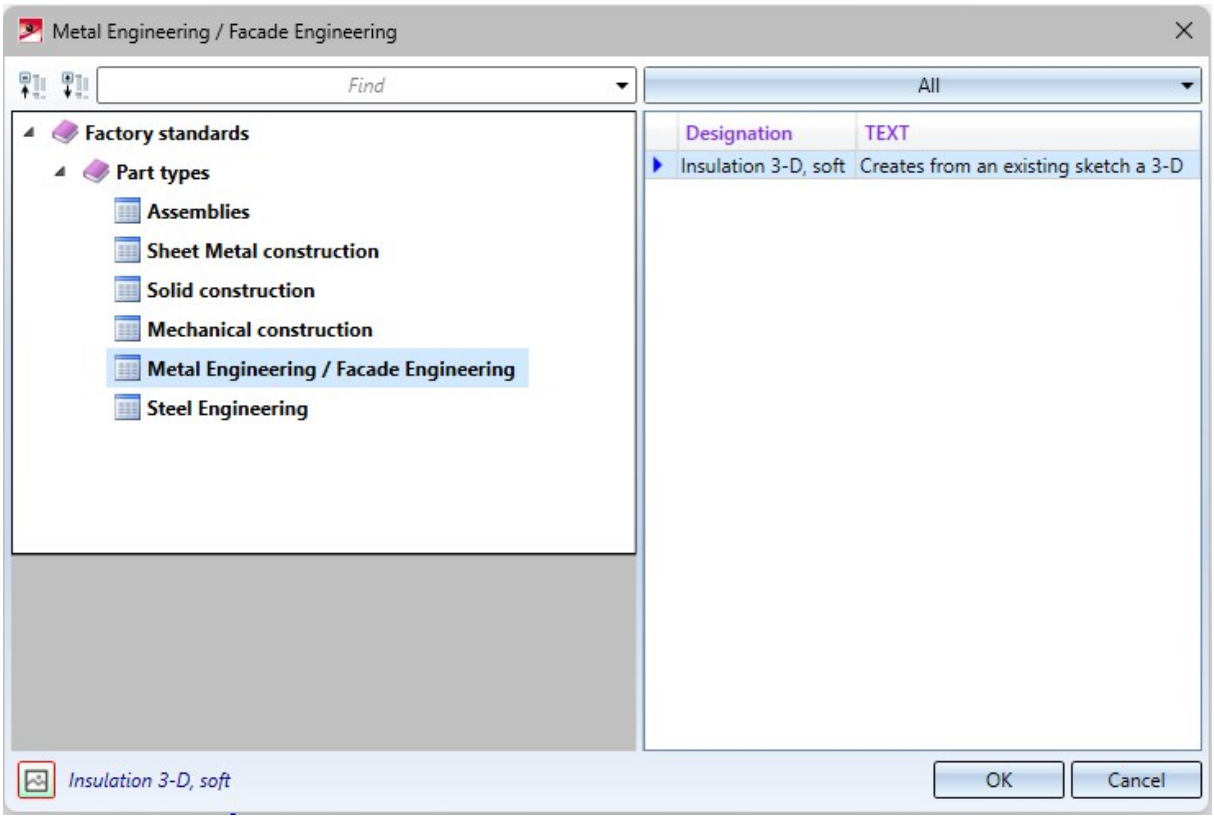
Facade/insert with sketch: Sketch on upper glass edge

A behavioural optimisation of the HiCAD/ LogiKal interaction concerns the function **Metal Engineering > New > Sketch (Facade/insert with sketch)**:

When using with sketch, the sketch is transformed to the position of the upper glass edge.

Civil Engineering - Part type catalogue, 3-D

The function **Civil Engineering - Part Type Catalogue > 3-D** in the **Civil Engineering functions**docking window under **Civil Engineering, general** contains functions that have been replaced by new developments in the meantime and are therefore no longer needed.



In the first step, therefore, the entries under

- Sheet Metal construction,
- Mechanical construction,
- Metal Engineering/Facade Engineering and
- Steel Engineering

have been removed. However, the structure of the part type catalogue is retained for the time being so that customer-specific catalogue entries in the above-mentioned areas can continue to be used after a HiCAD update.

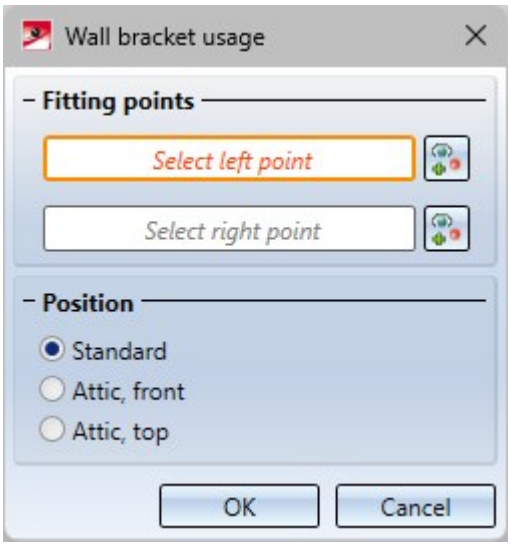
Layout Planning

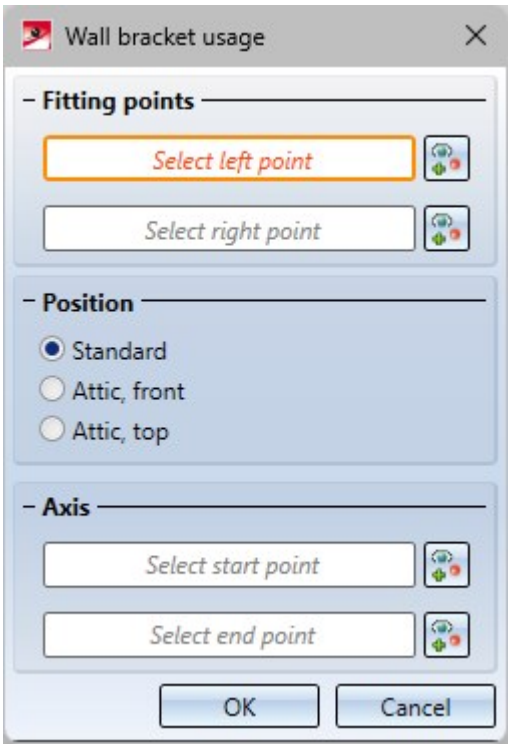
Service Pack 1 2024 (V 2901)

Wall bracket usage for parts without a steel engineering axis

The **Wall bracket usage** function is now also available for parts that do not have a steel engineering axis. For these parts, the axis must be explicitly defined in the dialogue window using two points.

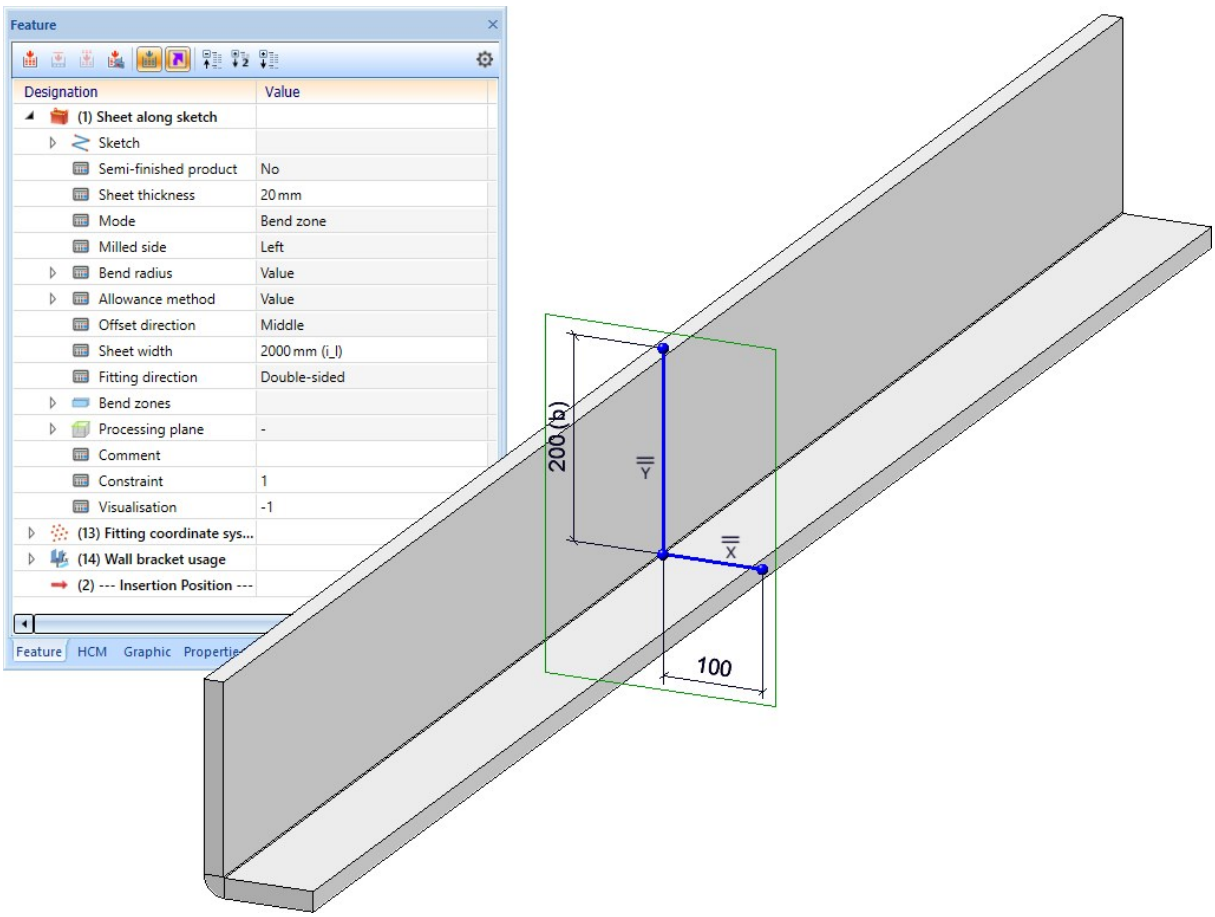
The dialogue windows for beams and sheets or other parts without a steel engineering axis are therefore different.



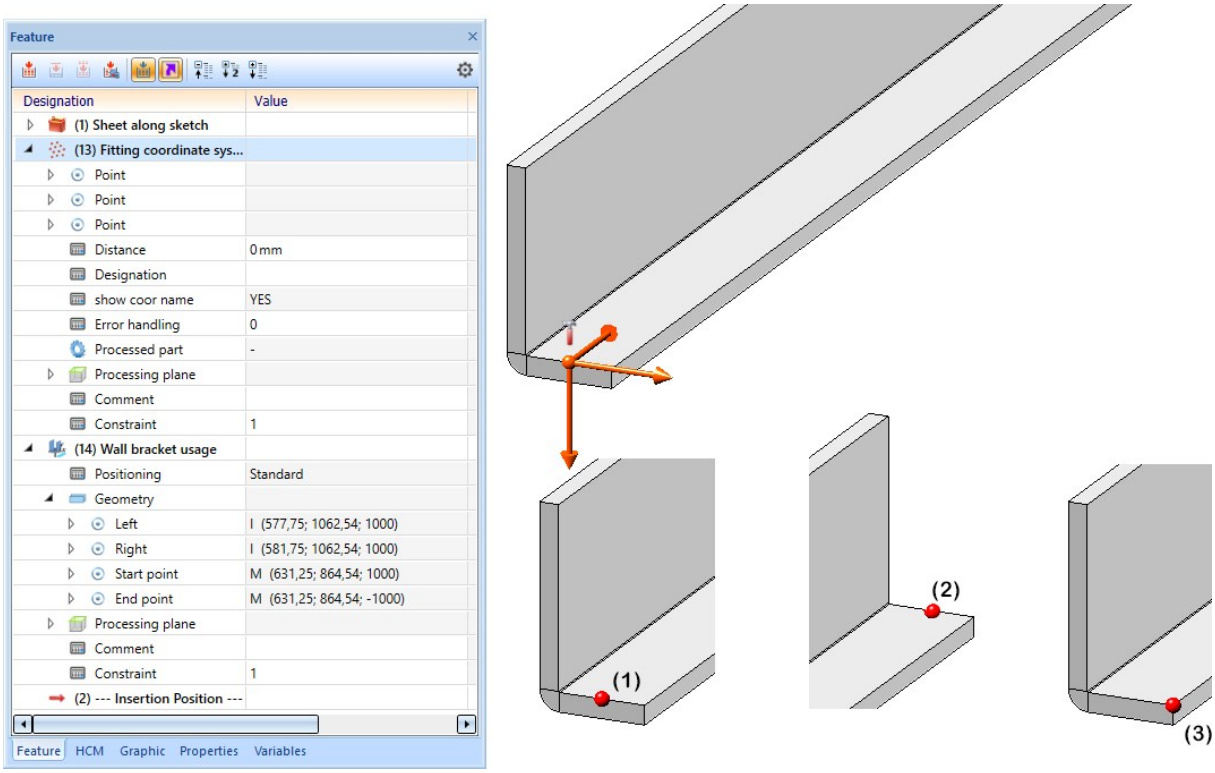


Creating your own sheets is essentially the same as creating your own beams and profiles. However, there is a difference when using wall brackets. Here, for parts that do not have a steel engineering axis, an axis must be defined via two points, which is then used in the **Wall bracket** function in the same way as the steel engineering axis.

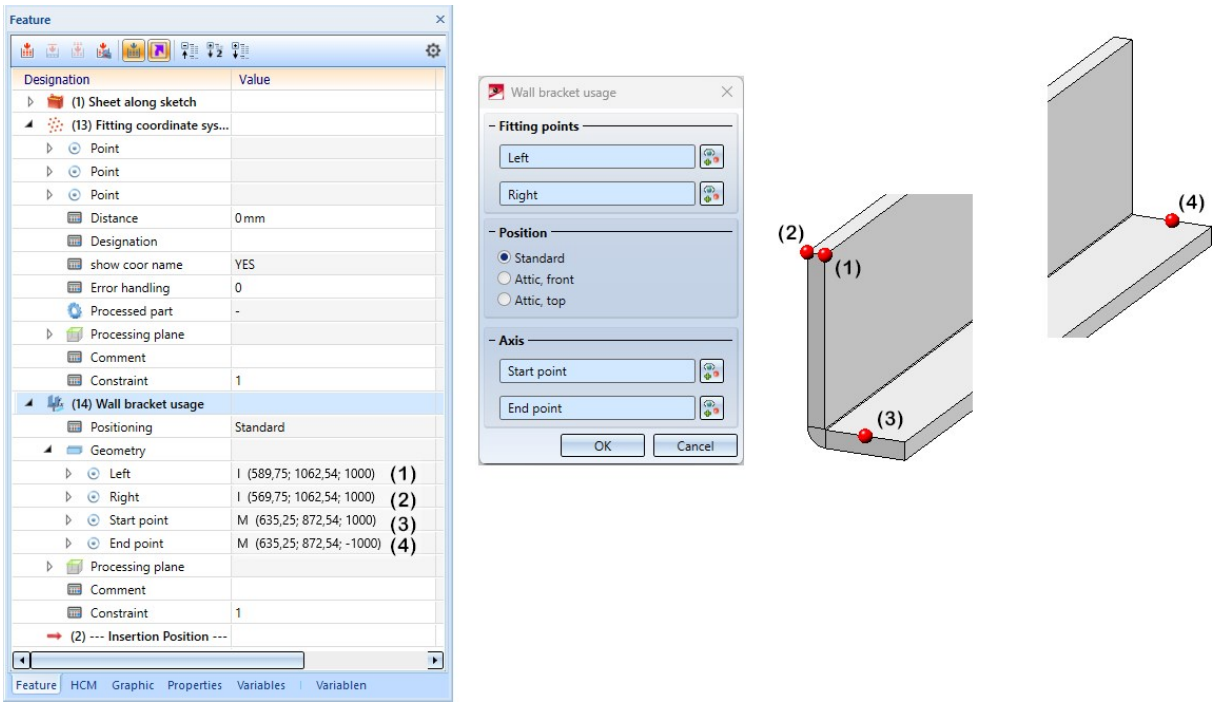
In the image below, a sheet has been created along a parameterised sketch. The variable i_l has been assigned to the sheet.



The Fitting coordinate system has then been defined using points (1), (2) and (3).

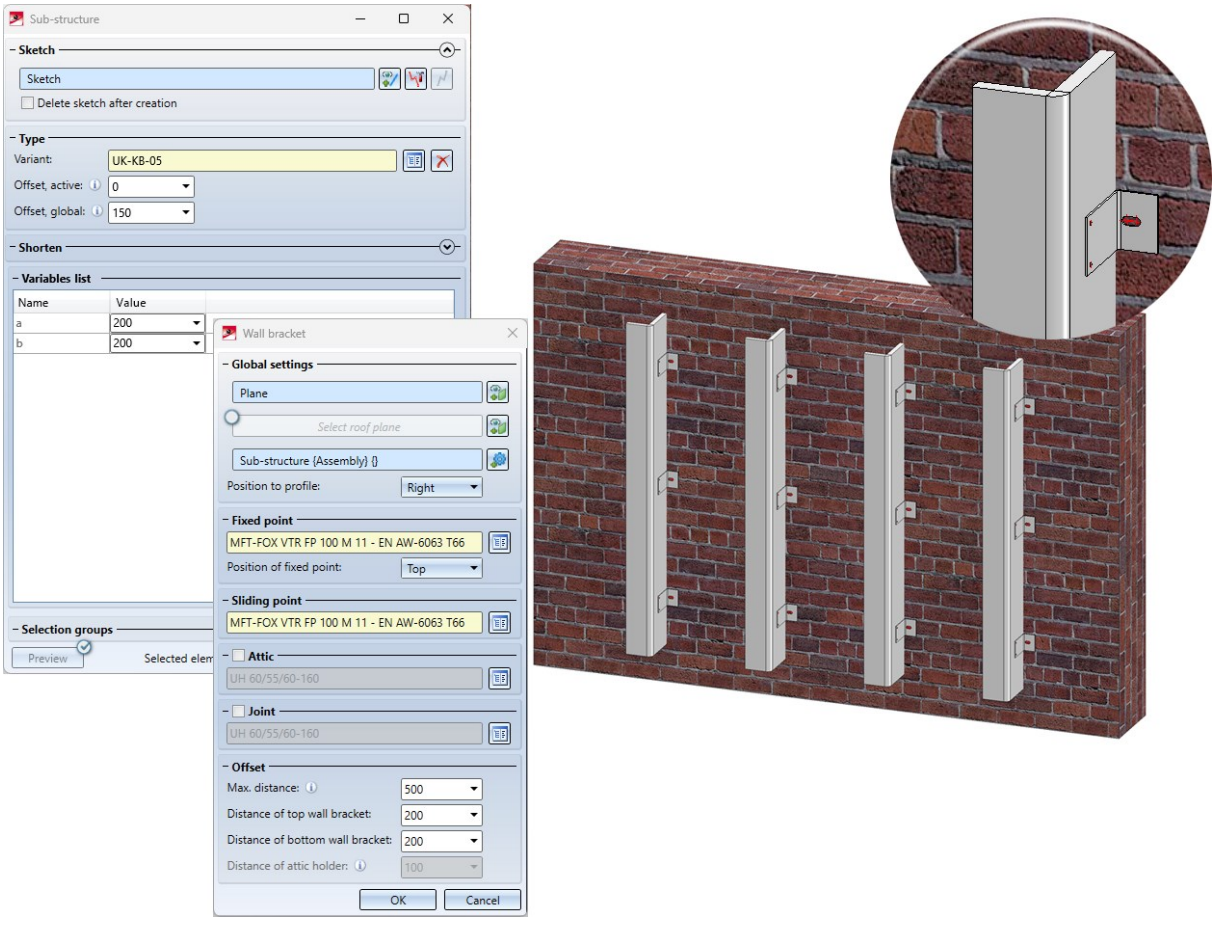


The wall bracket utilisation was then defined as shown using points (1) to (4).



The sheet was then saved in the **Factory standards > Installation planning - Parts and Processings > Sub-structure > Installation elements > ISD Example** catalogue.

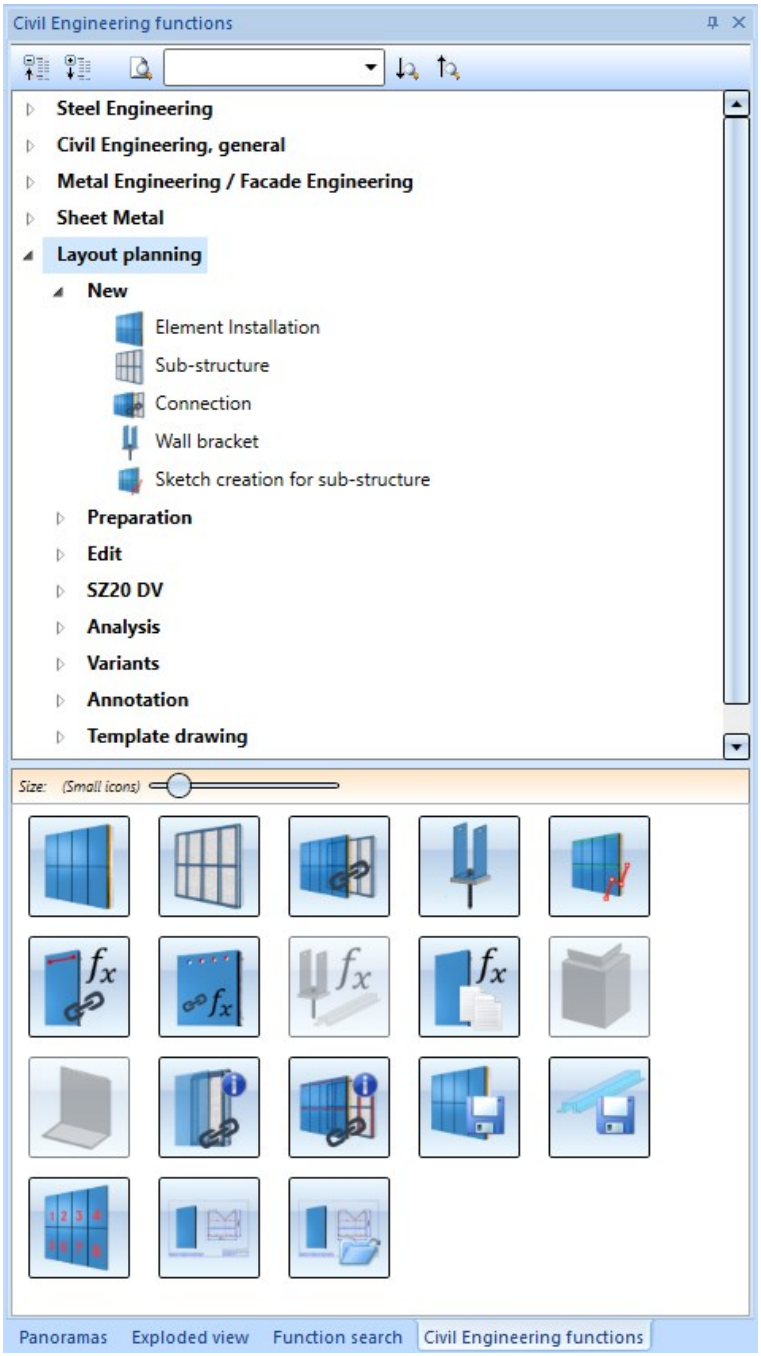
The image below shows a sub-structure with the sheet created above and wall brackets.



Major Release 2024 (V 2900)

Change of the structure in the docking window

In the **Civil Engineering functions** docking window, the functions for the Element installation, the Sub-structure and the Wall brackets have been combined and restructured under the new entry **Layout planning**.



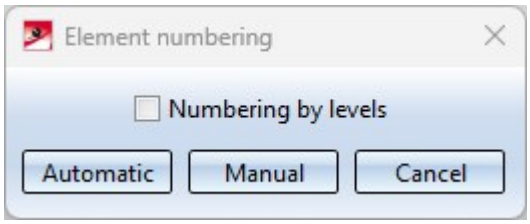
The Design Variants **Flange for SZ20** and **SZ20 Base point with projection** can now also be found here.

Annotating installation elements

New in the **Civil Engineering functions** docking window at **Layout planning > Annotation** is the function


Element numbering.

With this function, the installation elements of an element installation can be annotated automatically, e.g. for overview drawings.



Numbering by levels

If this checkbox is activated, the numbering will be done per "level", always from the first element at the bottom left to the last element. The Y-axis determines the level direction. If the checkbox is deactivated, numbering will be done consecutively.

Automatic

With automatic numbering, the XY-plane of the active coordinate system determines the numbering sequence (first in X-direction, then in Y-direction).

Manual

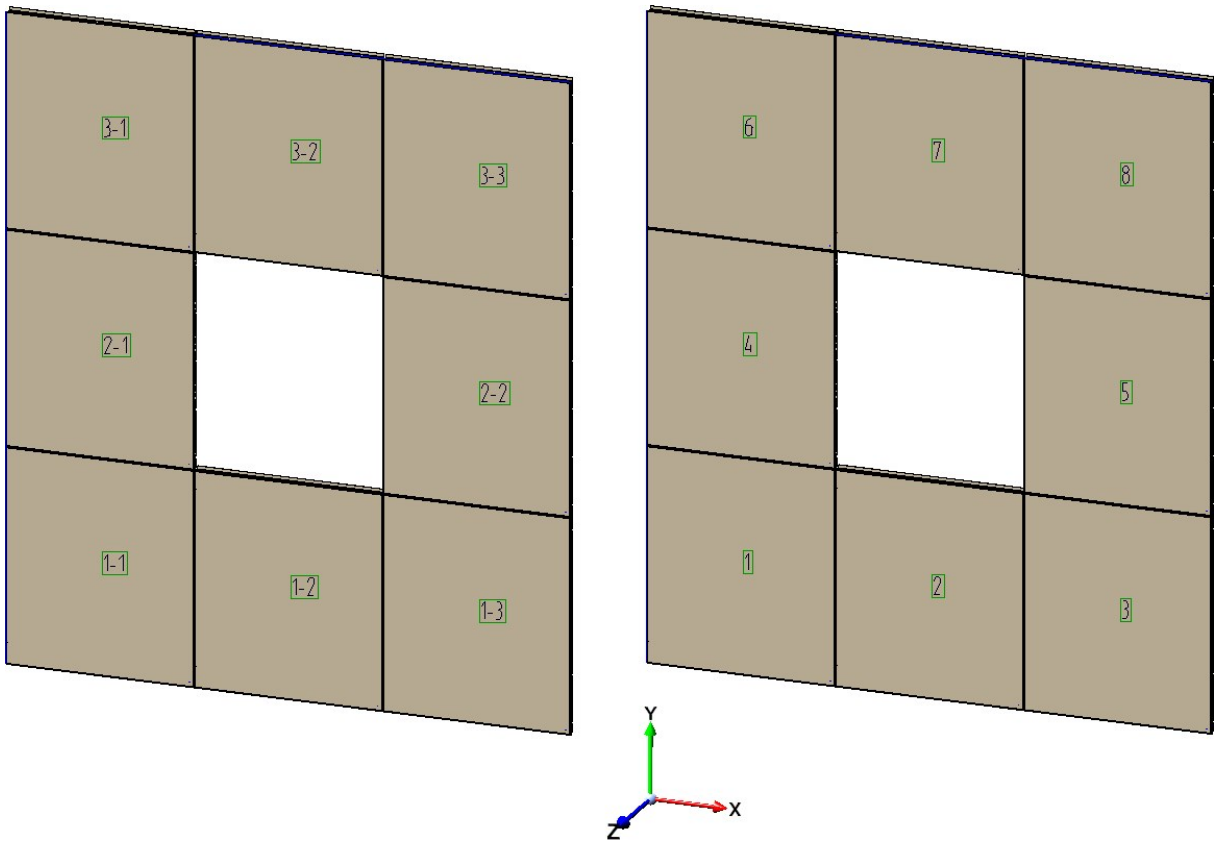
Here you simply select the elements to be numbered with a mouse click.

The content of the numbering is controlled by the following annotation templates:

- EINumbering_PerLevel.ftd (for numbering by levels)
  {Level of element installation numbering (part attribute)}-{Running number of element installation numbering (part attribute)}
- EINumbering_Consecutive.ftd (for consecutive numbering)
  {Consecutive number of element installation numbering (part attribute)}

As with the other HiCAD annotation templates, the files can be individually configured if required, e.g. with the HiCAD Annotation Editor. The files are located in the HiCAD sys directory.

Example:



Left: Numbering by levels, Right: Consecutive numbering.

It will not be checked whether the numbered elements are identical.

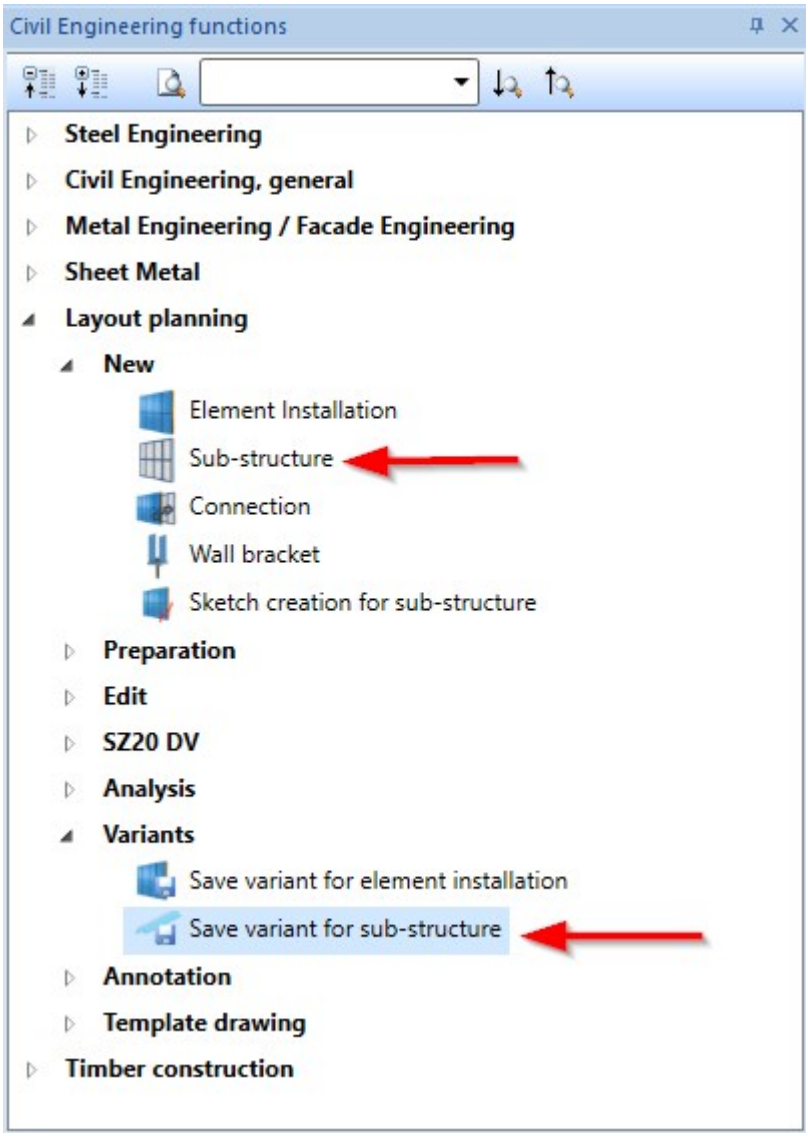
Profile Installation

Service Pack 1 2024 (V 2901)

Licensing

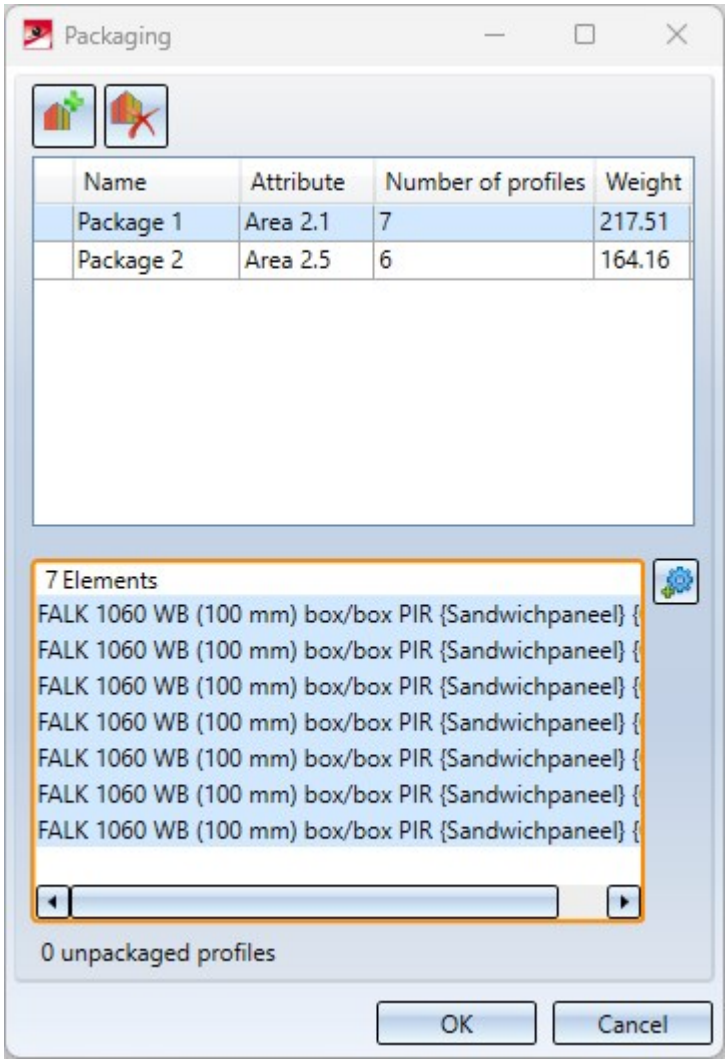
As of HiCAD 2024 SP1, the HiCAD Extension Module **Profile installation** contains the functions

- **Save sub-structure** and
- **Variant for sub-structure**.



Additional column in Packaging dialogue

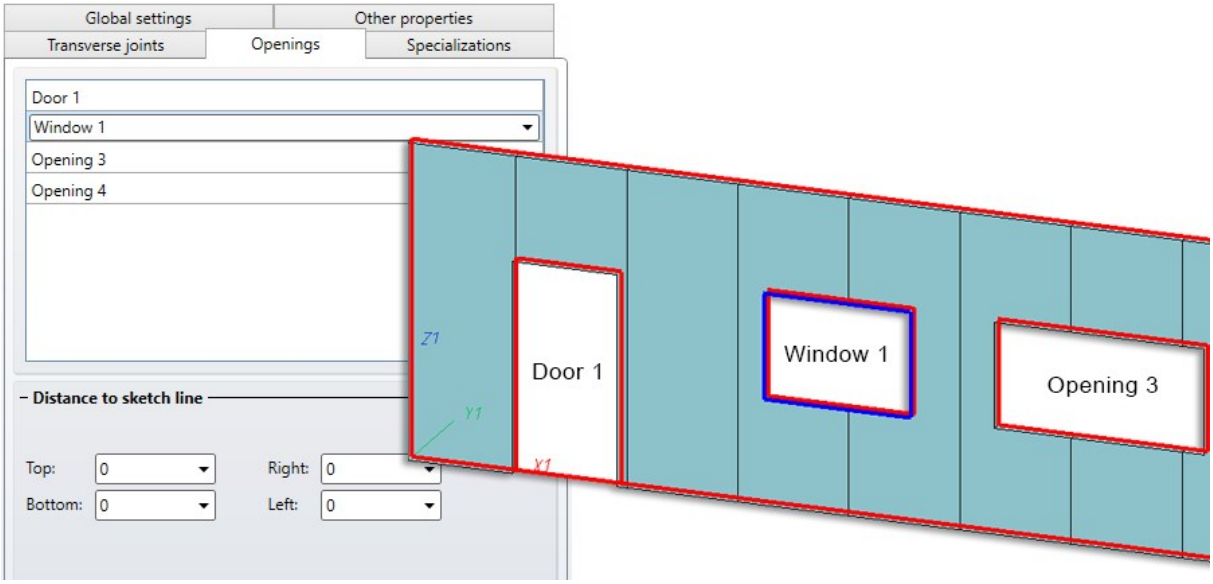
There is now an additional column for a user-specific attribute in the **Packaging** dialogue. This attribute must be set in the Configuration Editor at **Profile installation > Packaging > User-specific attribute**.



Attribute **Comment ($03)** set in the Configuration Editor.If you activate the **Attribute** field for a package by double-clicking on it and then enter a comment, this will be transferred to the part attributes of all parts that belong to the package.

Rename openings

When creating or changing in the **Profile installation**  dialogue, you can now edit the naming on the **Openings** tab. By double-clicking in the field of the Opening 1, 2, ... the name can be overwritten. This name is displayed in the preview.



Major Release 2024 (V 2900)

Preset default values for joint width towards sketch line

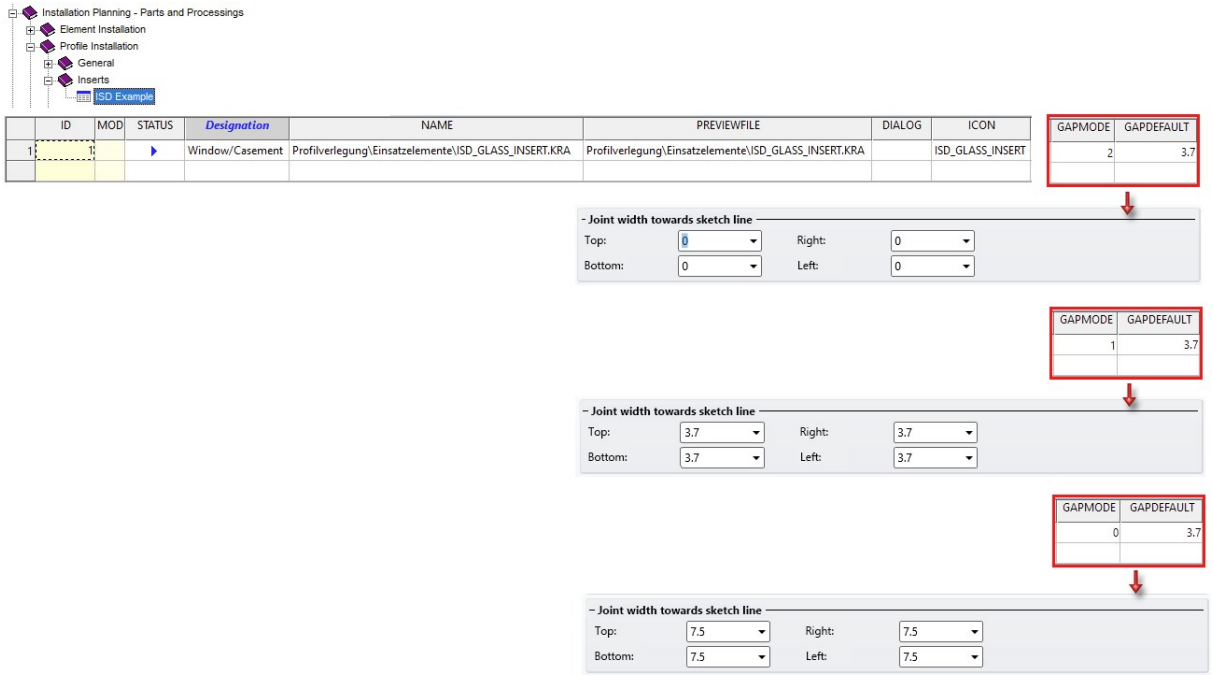
The joint width towards the sketch line can also be preset in the tables at **Factory standards > Installation Planning - Parts and Processings > Profile Installation > Inserts**.

To do this, add the table columns **GAPMODE** and **GAPDEFAULT** to the tables.

Column	Data type	
GAPDEFAULT*	Floating point value	The desired value for the joint width is entered here. If this value is to be activated as a default, the column GAPMODE must be set to 1. If the corresponding installation element is then selected in HiCAD as insert element of the profile installation, the value for the joint width is automatically set to the value entered in the column GAPDEFAULT.
GAPMODE*	Integer	The value entered here can be 0, 1 or 2. 0 The default value from the GAPDEFAULT column is deactivated. The internal HiCAD default value is used instead. 1 The value specified in the GAPDEFAULT column is used as the default value. 2 No default value is used, i.e. 0 is preset for the joint width.

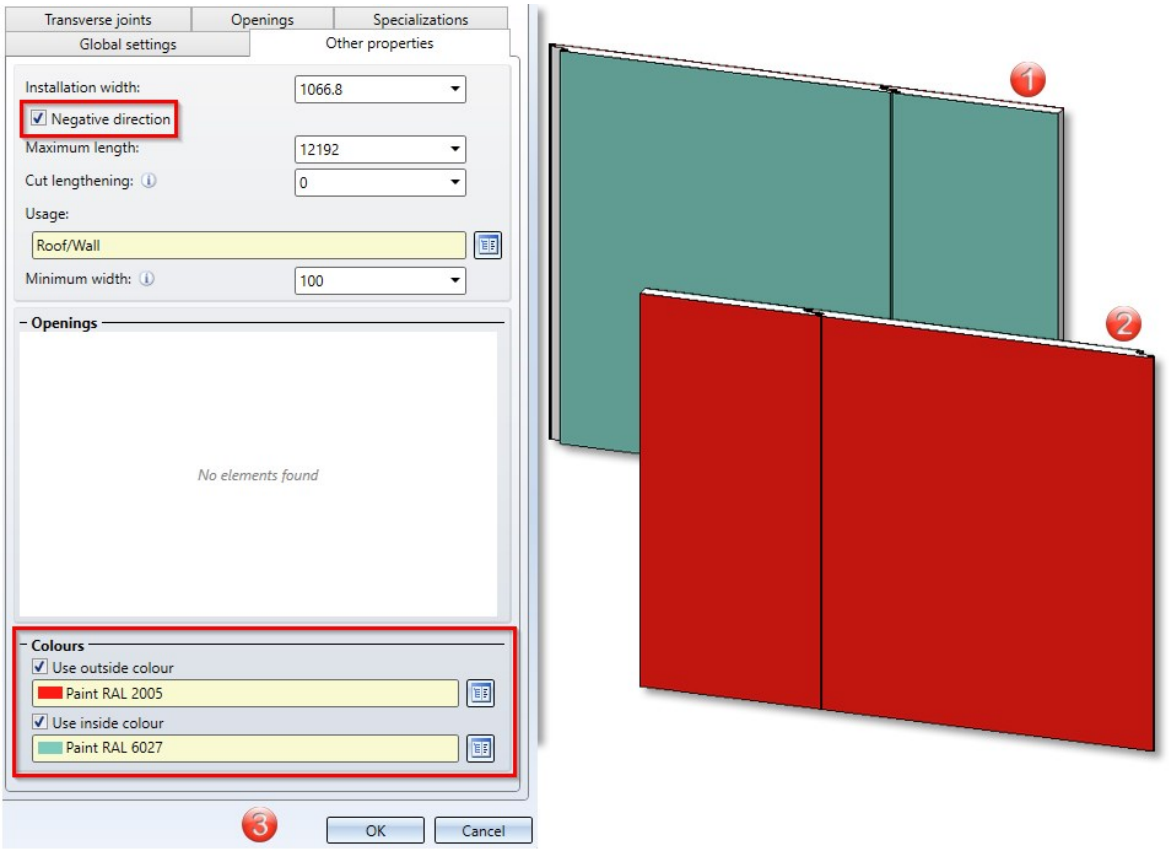
* only for own installation elements and installation elements predefined by the ISD

Example:



Enhancements for Coating

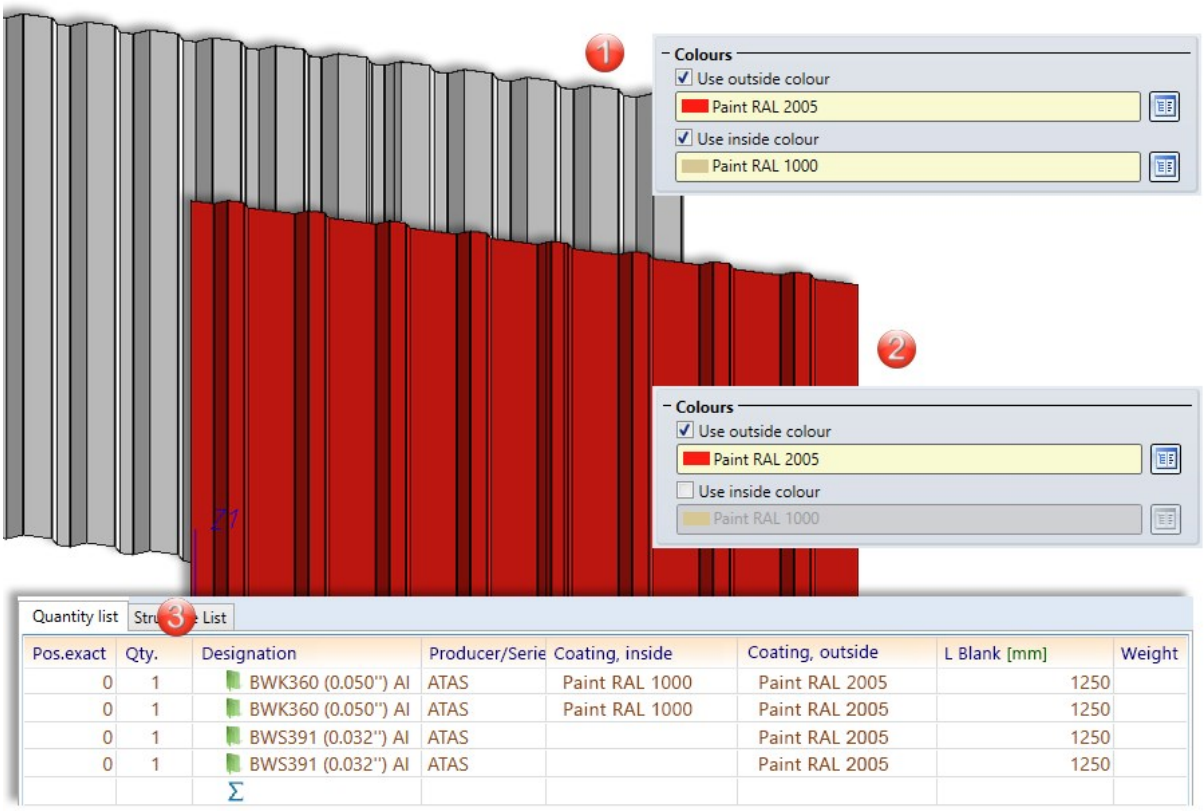
When coating profiles (sandwich profiles), you can now use different colours for the outside and inside. The negative layer no longer affects the coating, i.e. the outer sheet always receives the outer colour. Attributes are set for both coating colours and transferred to the bill of materials.



(1) Inside, (2) Outside, (3) Settings in the **Other properties** tab of the **Profile Installation** function

Coating of one-piece profiles

One-piece profiles (corrugated and trapezoidal profiles) can now be coated on both sides. If you select only one colour, the profile is also coloured in the graphic. If it is coated on both sides, only the attributes are set and transferred to the BOM.



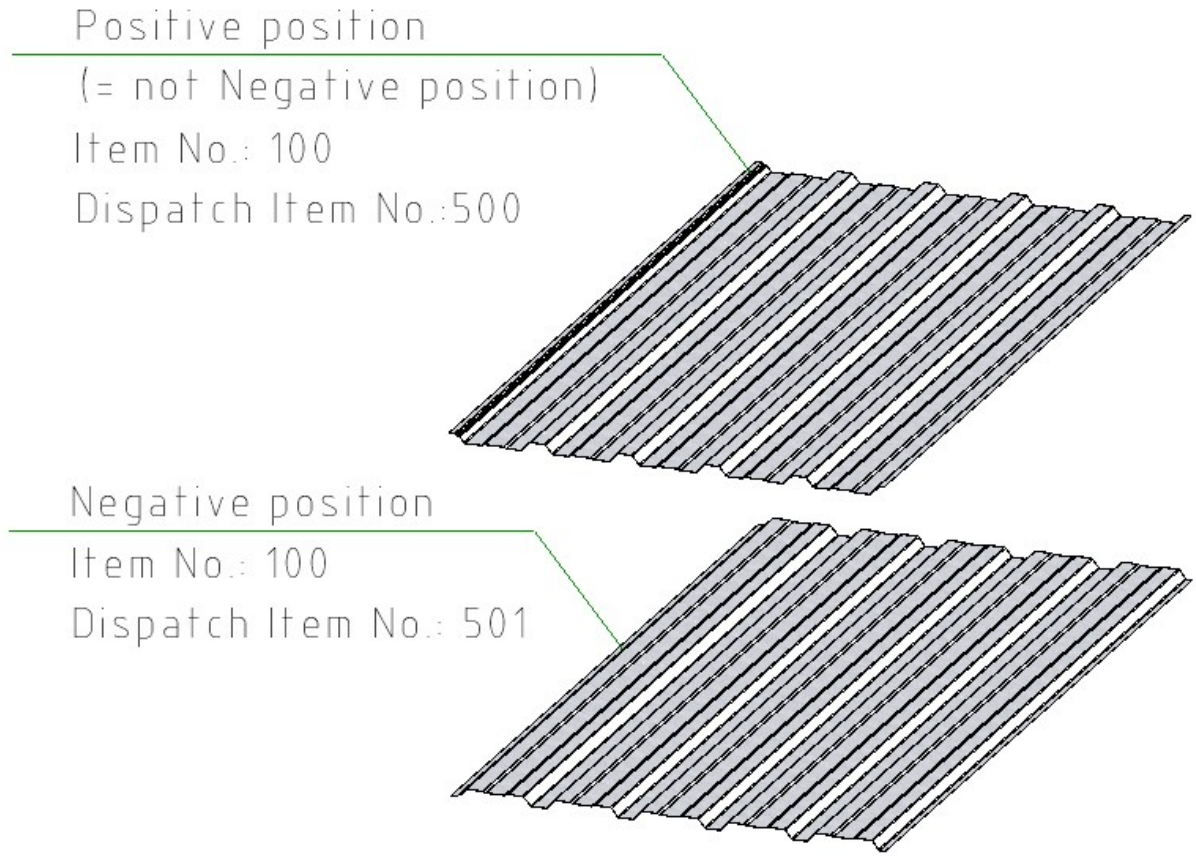
Pos.exact	Qty.	Designation	Producer/Serie	Coating, inside	Coating, outside	L Blank [mm]	Weight
0	1	BWK360 (0.050") Al	ATAS	Paint RAL 1000	Paint RAL 2005	1250	
0	1	BWK360 (0.050") Al	ATAS	Paint RAL 1000	Paint RAL 2005	1250	
0	1	BWS391 (0.032") Al	ATAS		Paint RAL 2005	1250	
0	1	BWS391 (0.032") Al	ATAS		Paint RAL 2005	1250	
		Σ					

(1) One-piece profile coated on both sides,
(2) One-piece profile coated on one side,
(3) Coating in the BOM

Negative and positive position and identical part search

The negative and positive position of the profiles is a differentiation criterion for the same-part detection during dispatch itemisation. If the value of the attribute DWF_NEG_INSTALL is set to 0, the profile is in a positive position and if it is set to 1, it is in a negative position. In configuration management, it is entered as a distinguishing criterion for dispatch itemisation under **Profile Installation > Dispatch itemisation > Integer attributes** and is evaluated if the setting **Carry out dispatch itemisation** is also activated here.

The attribute is not applied to profile installations prior to HiCAD 2024, not even when recalculating the feature.

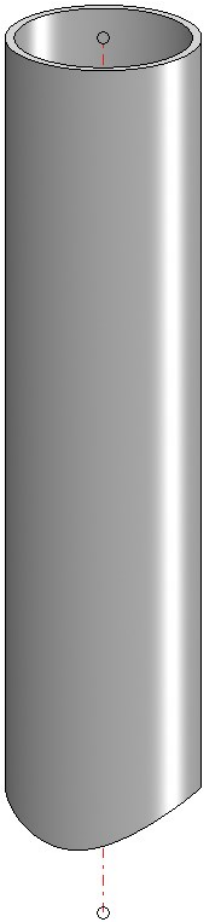


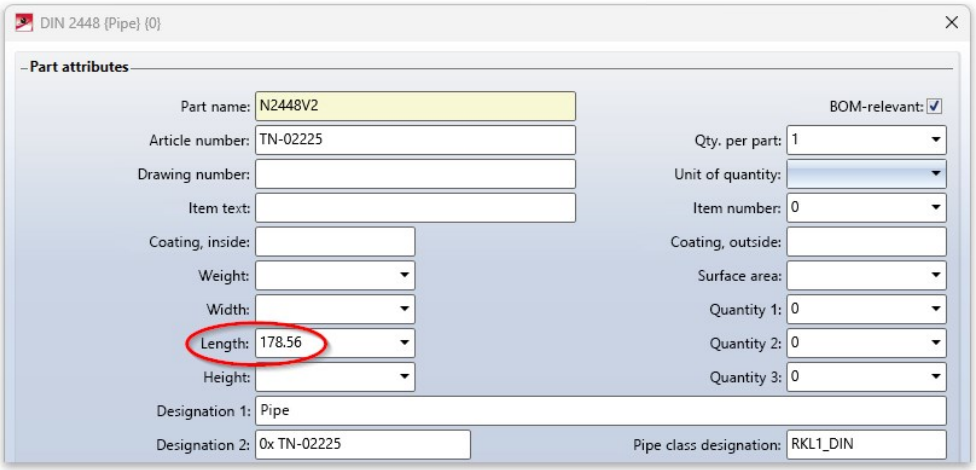
Plant Engineering

Service Pack 1 2024 (V 2901)

Length of inserted pipes

Previously, the length of inserted pipes was always calculated up to the center of the pipe into which it was inserted. As of SP1, the length is now calculated correctly. This also applies to the isometry and the pipe spool drawing.





Knee - Horizontal bending direction (VEERING_RESTRICTION)

The new HELiOS attribute **VEERING_RESTRICTION** complements the already existing attributes BENDING_RESTRICTION (bending direction) and PLANE_RESTRICTION (bending plane) in that also with VEERING_RESTRICTION the route of a guideline can influence the part search.

To make this attribute available in your HELiOS database as well, you may have to update HELiOS for Plant Engineering before. To do this, use the DbPlantDataImport.exe tool.

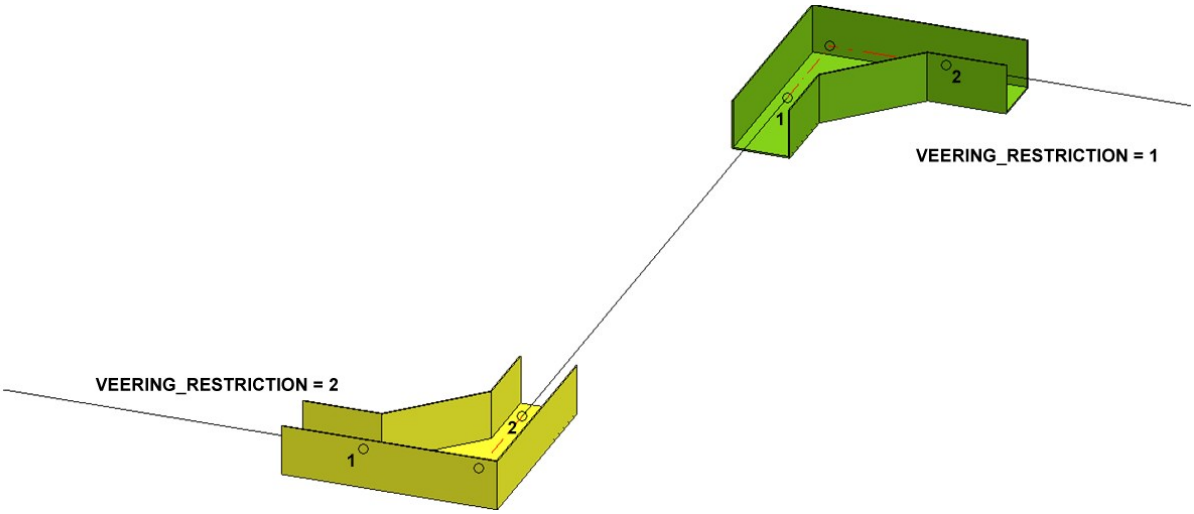
The VEERING_RESTRICTION attribute knows the following values:

0	no restriction, treated like regular knee; in this case, however, the attribute should simply be left unassigned, which has the same meaning. Not used in search conditions.
1	only allowed to the right
2	only allowed to the left
3	symmetrical; allowed to the left or right

The only part type whose insertion is affected by the attribute is the **Knee**.

Example - VEERING_RESTRICTION

In the following example, a simple composite edge has been automatically assigned two knees. The flow direction is from left to right.



When searching for a suitable knee for **bends to the left**, HiCAD adds the following condition to the HELiOS search:

VEERING_RESTRICTION = ''\'3*'\'**2***''

This means that only parts that do not have VEERING_RESTRICTION, are symmetrical or bend to the **left** will be found.

In the above example, the yellow knee was found and fitted.

When searching for a suitable knee for **bends to the right**, the following condition was added to the HELiOS search:

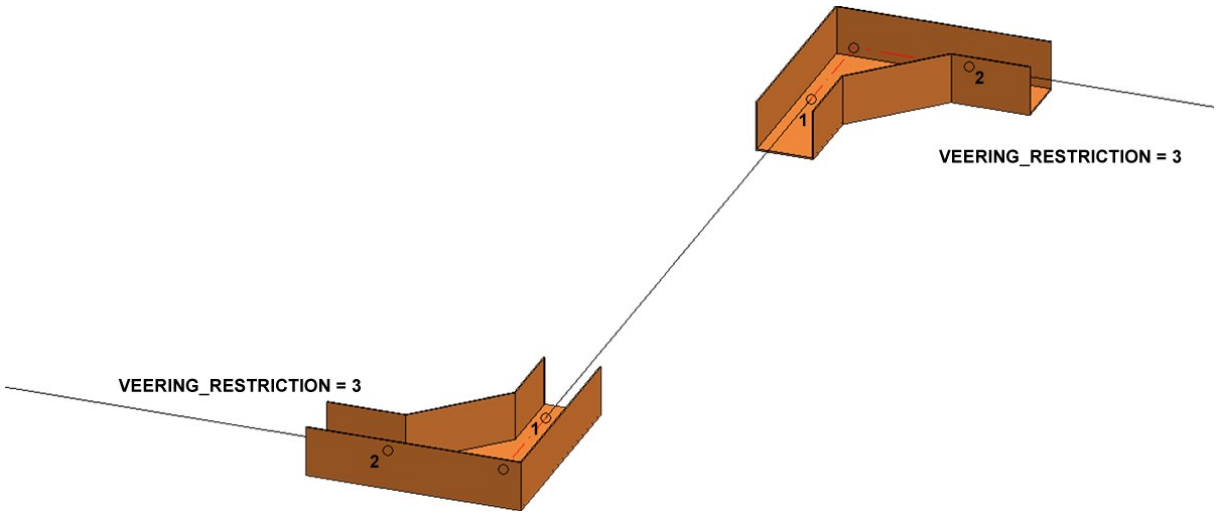
VEERING_RESTRICTION = ''\'3*'\'**1***''

This means that only parts that do not have VEERING_RESTRICTION, are symmetric or bend to the **right** will be found.

In the above example, the green knee was found and installed.

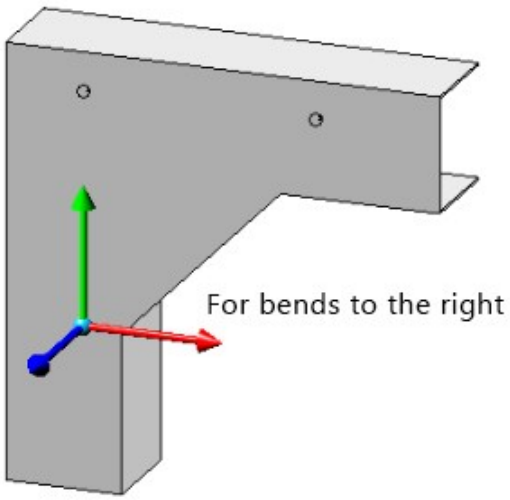
For **symmetrical** parts, one would like to be able to additionally express that these can be used for both left and right bends, which at the same time implies that these parts may also be installed against the flow direction (namely for left bends).

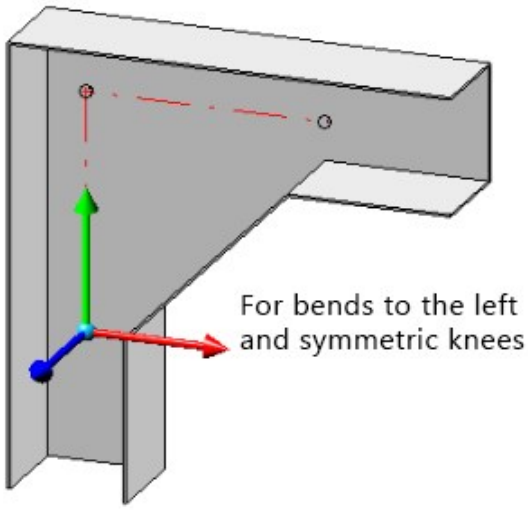
In the following example, the same composite edge has been fitted with a part whose VEERING_RESTRICTION has the value 3:



Note that for the bend to the left the order of the connection points is switched , so the part was installed against the flow direction running from left to right.

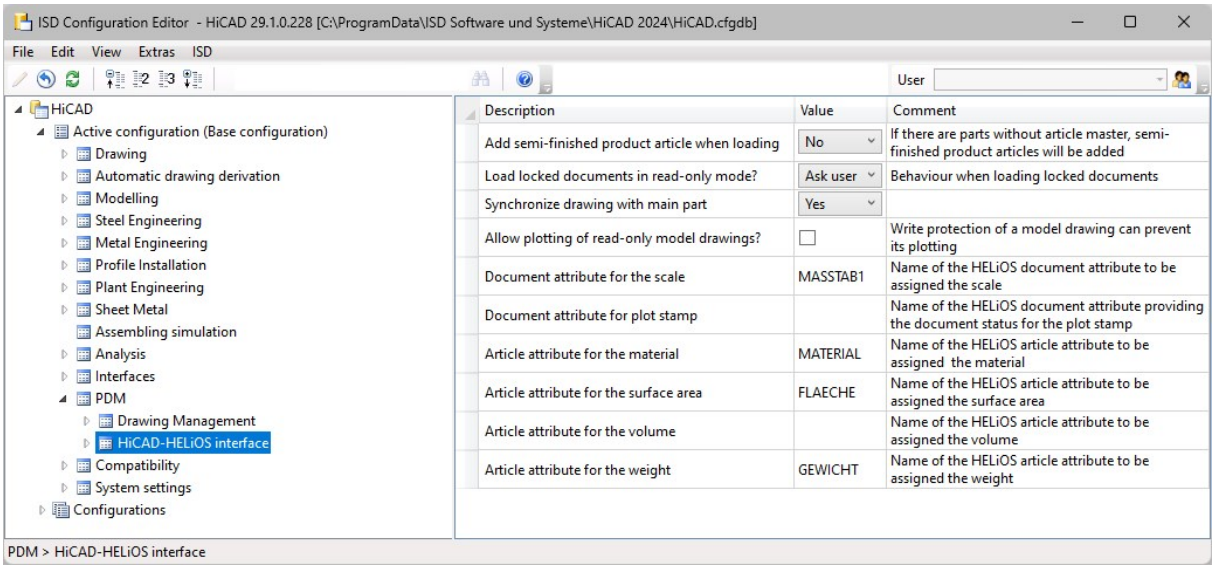
To use a custom variant with VEERING_RESTRICTION, this attribute must be assigned in the VAA file. The knees follow the usual design guidelines for knees. For the above examples, the variants look as follows:





Presettings for article attributes in the Configuration Editor

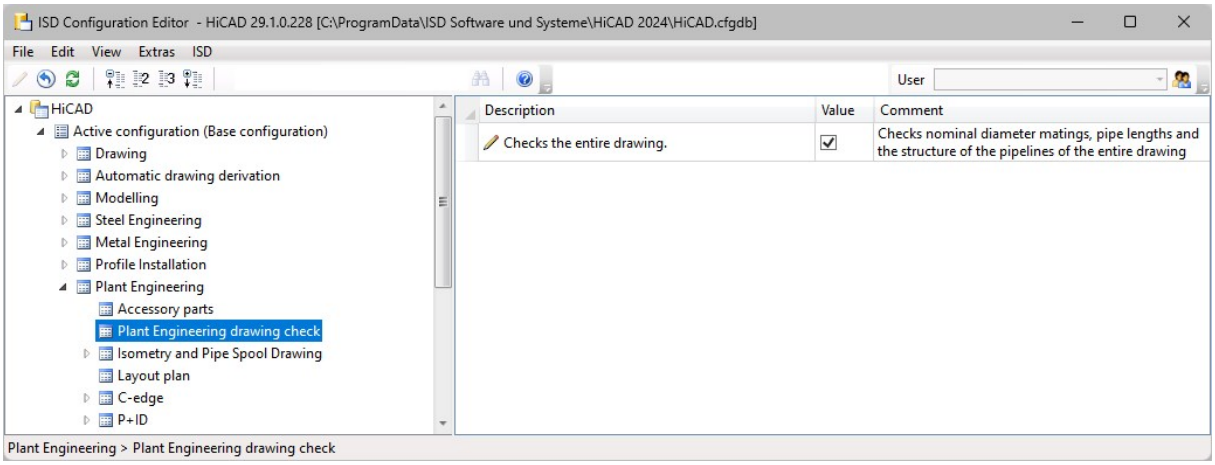
The MATERIAL, SURFACE AREA, WEIGHT and VOLUME attributes were previously considered fixed by pipeline planning, although they have been separately definable in the Configuration Editor for some time. As of Service Pack 1, these specifications are now taken into account.



However, a special feature applies to variants, which must of course work for all settings in the Configuration Editor. Therefore, material, surface area, weight and volume will still be assigned to the MATERIAL, SURFACE AREA, WEIGHT and VOLUME attributes in variants. During part data synchronization, however, these four attributes are then mapped to the attributes defined in the Configuration Editor.

Automatic checking of nominal diameters, pipe lengths and part structure

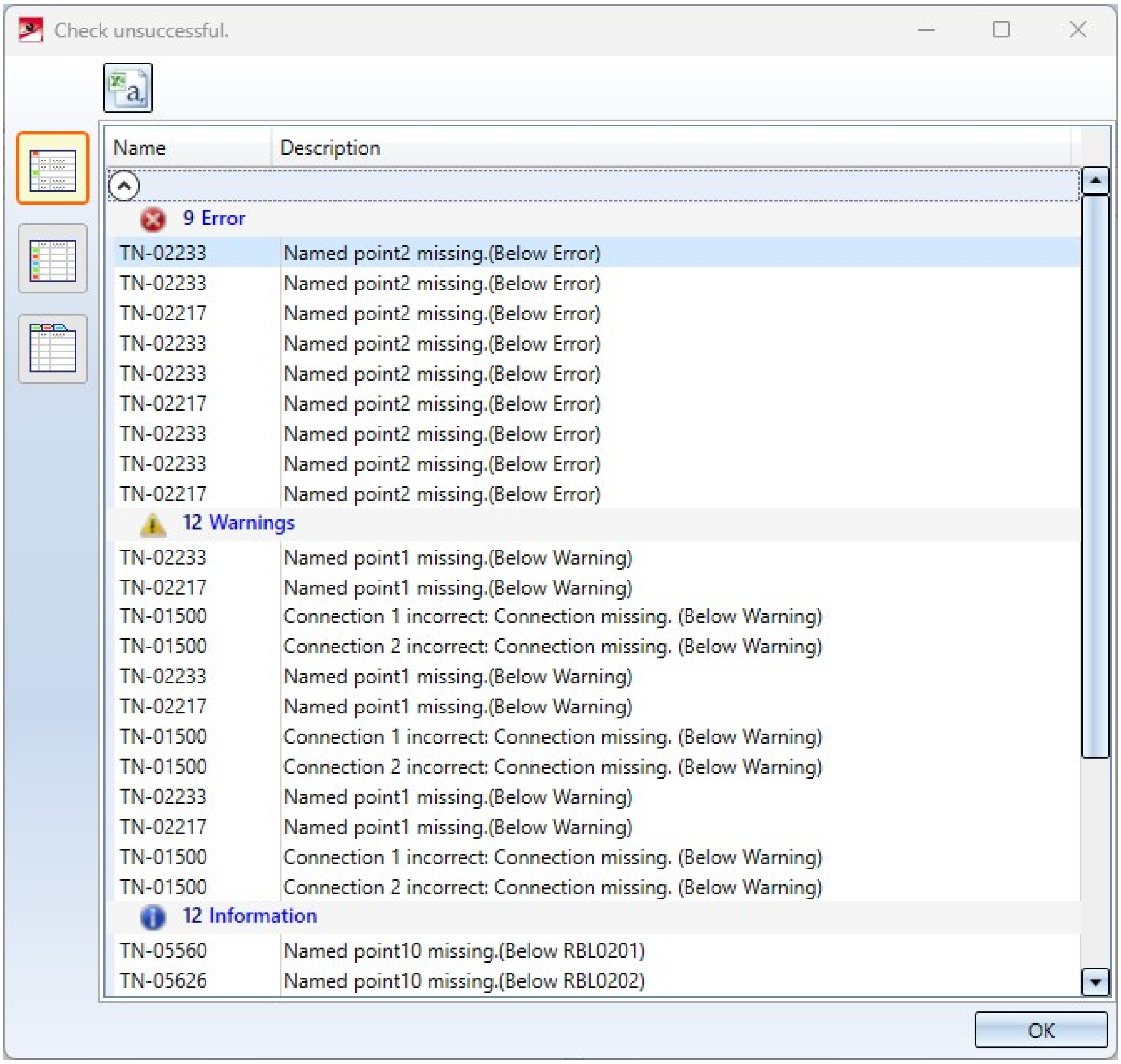
In addition to checking nominal diameters, pipe lengths and pipeline structure with the functions at Plant Engineering > Pipeline Tools > Coll... HiCAD also offers the possibility to perform these checks automatically when loading and saving layout plans or when switching from a P+ID to the layout plan. Previously, this could be defined in the Plant Engineering settings under **Actions during Load/Save**. As of SP1, this tab is no longer available. Instead, the parameter **Checks the entire drawing** is available in the Configuration Editor at **Plant Engineering > Plant Engineering drawing check**.



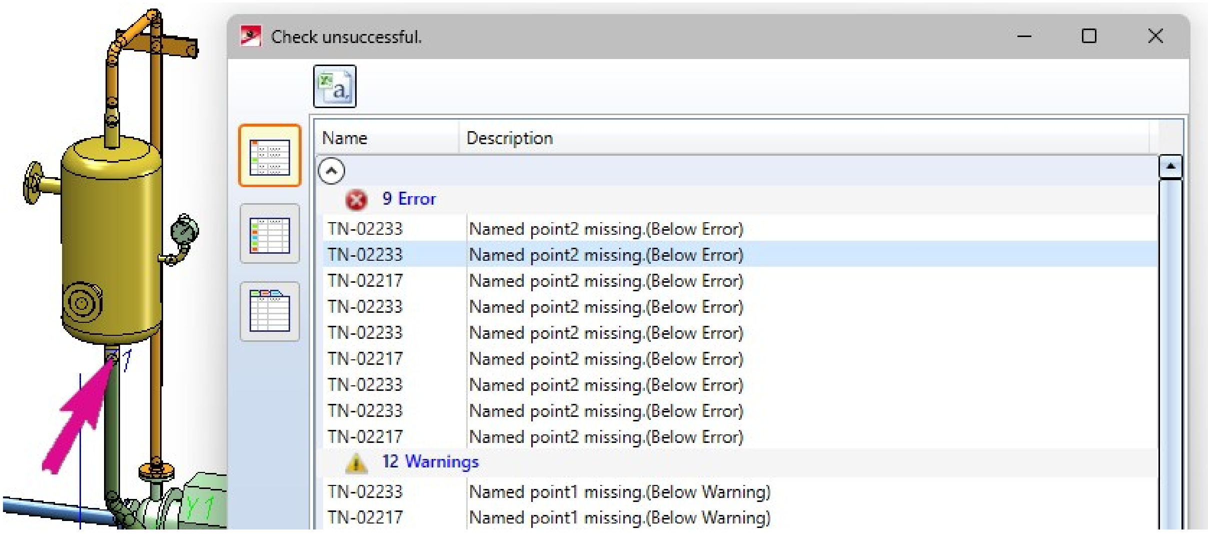
If this checkbox is active, HiCAD automatically performs the following checks during load/save:

- **Nominal diameter matings**
  Here it is checked whether only parts with matching nominal diameters are connected with each other in the layout plan.
- **Pipe lengths**
  It is checked whether inadmissible pipe length changes have been made in the layout plan.
- **Structure of the pipelines**
  The part structure of all pipelines is checked here.

If errors occur during the check, they are displayed in a results list, e.g.



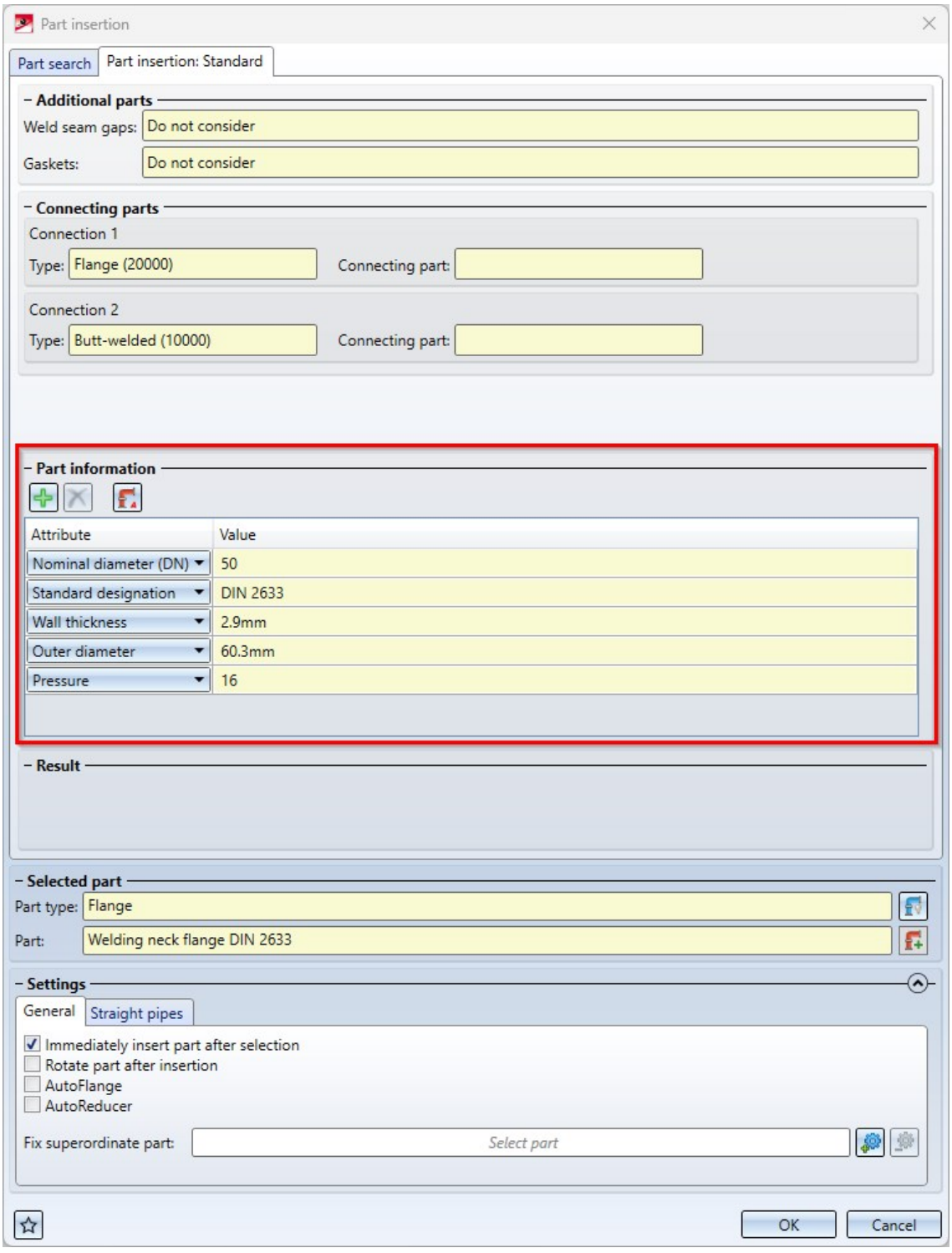
If you click on one of the errors in the list, the corresponding location in the layout plan is marked accordingly.



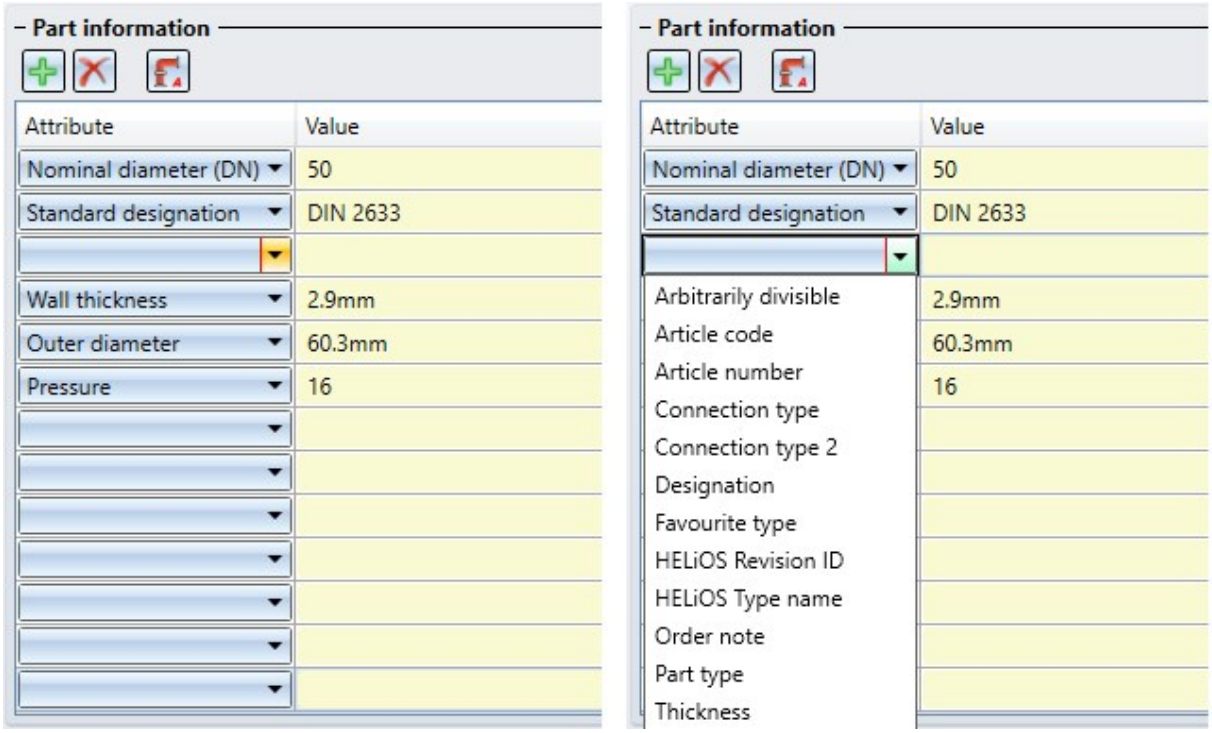
Changes/enhancements for pipe part insertion

Additional part information

The tab that is displayed with the **Pipe parts**  function after selecting a part has been extended. Additional information about the selected part is now displayed here, e.g.



Clicking this button inserts a new row in the attribute list after the current line. Using the selection box of the row, you can then select the attribute to be displayed.



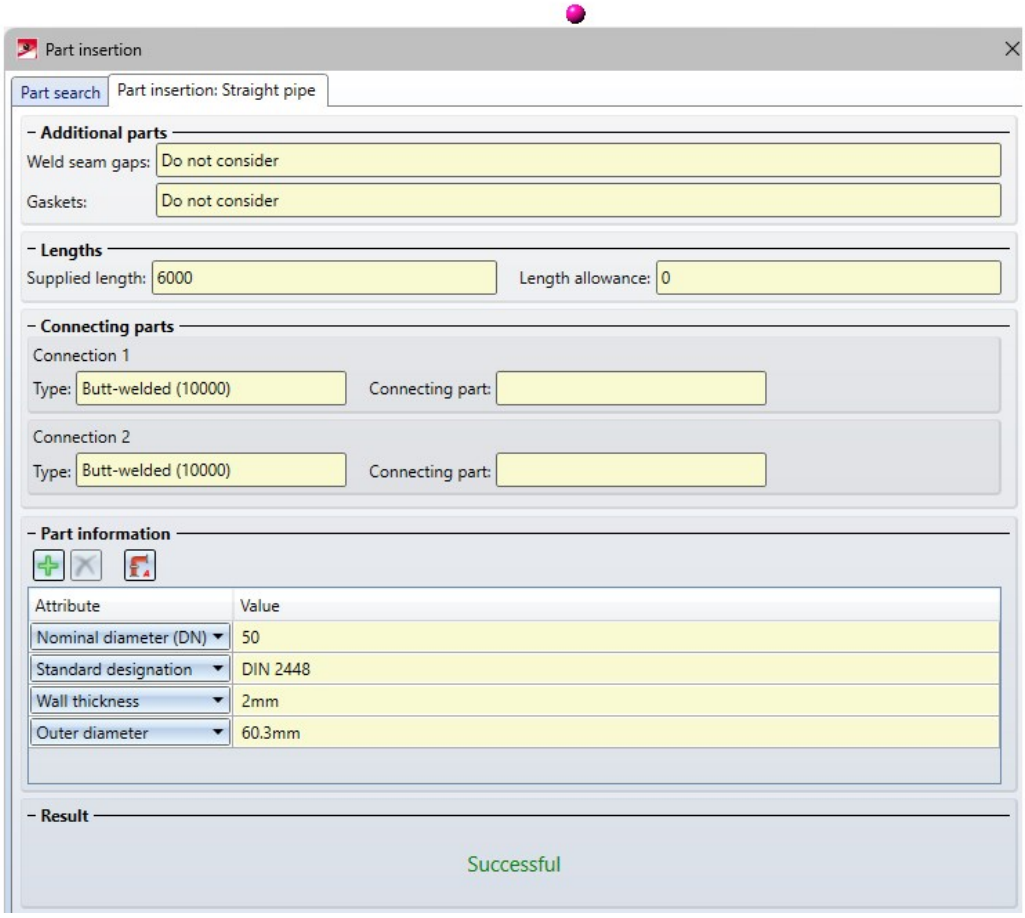
To change the attribute display of a row, simply select the desired attribute in the selection box.

A click on this button deletes the current row of the attribute list..

Clicking this button restores the default state of the attribute list..

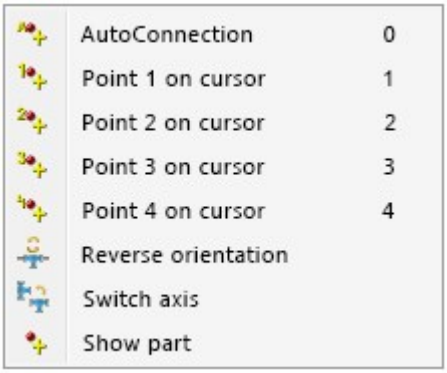
Undo/Redo

As of Service Pack 1, the Undo and Redo functions of the transparent toolbar or the quick access toolbar can be used within the **Pipe parts**  function after a part has been inserted.

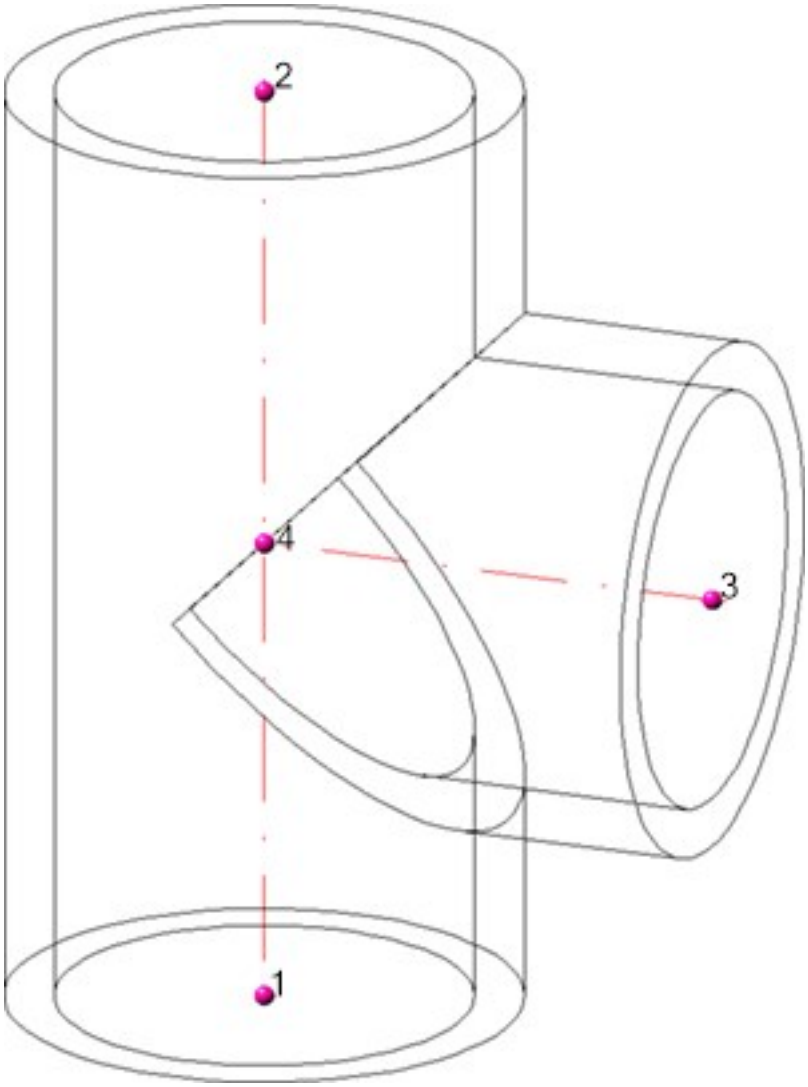


Free point selection

The context menu for defining the insertion position has been extended.

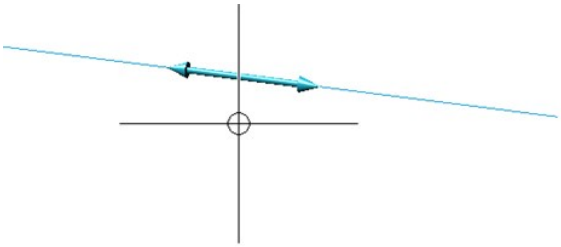


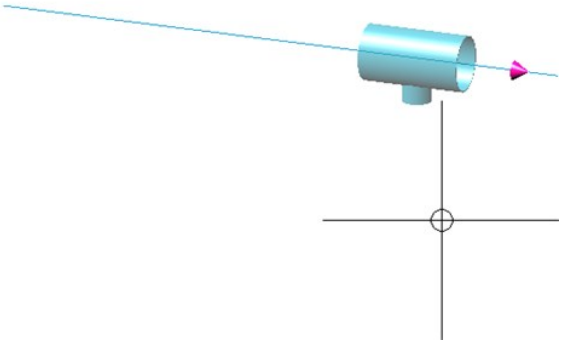
The **Show part** option for freely positioning the part at an individual point has been added. For this purpose, a preview of the part is displayed where you can select any point.



After selecting the point, the part is attached to the cursor at the corresponding position and can be freely positioned in space.

Another option is to select any point and then select an edge for positioning. Then the cursor is attached to the selected point of the part. This is projected onto the edge, so you can align the part to the selected point on the edge.



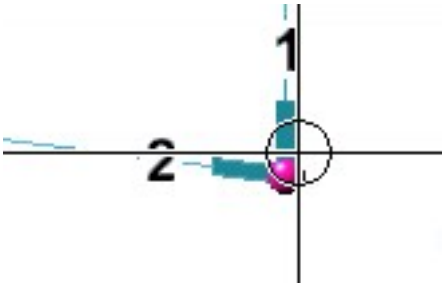


Revised "Set all" option

In the **Part insertion**  dialogue window, the checkbox **In all similar places** has been removed. Instead, two new options are now available in the **Selected part** area.

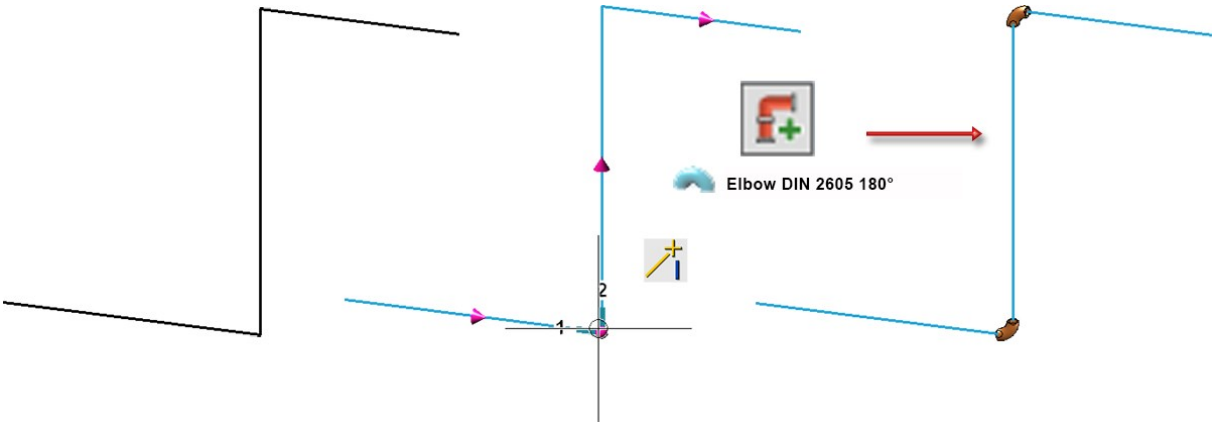
Insert everywhere

This option allows you to automatically insert a part in all similar places. If you have not selected a part so far, first define the insertion position of the selected part type, e.g.



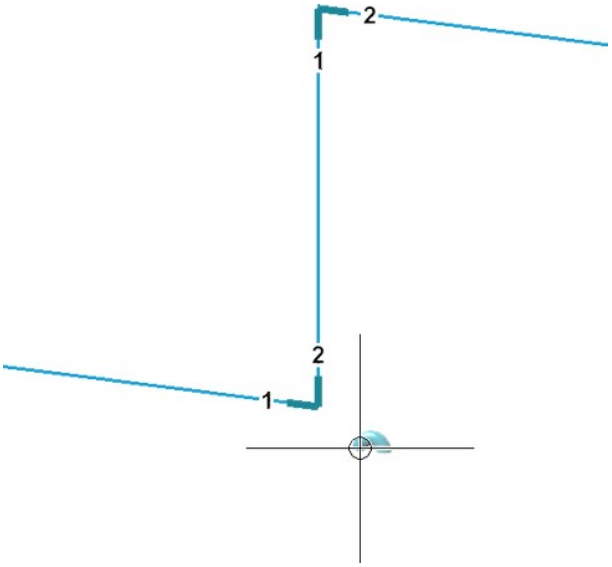
Then select the desired part after clicking on  . If you have already selected and inserted a part, clicking on  will cause the part to be inserted automatically in the other corresponding places as well.

Example



Highlight similar places

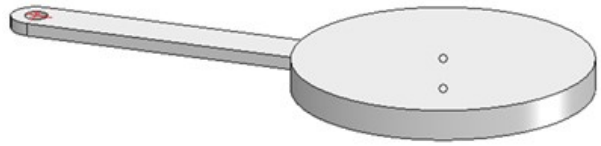
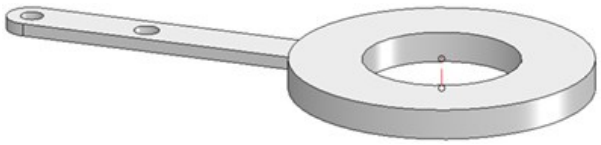
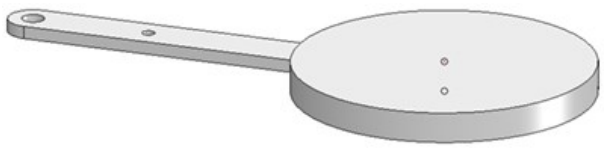
This symbol is only active if you have already selected a part. All possible insertion positions are then displayed for this part, so that you can explicitly trigger the insertion at the desired positions with one click.

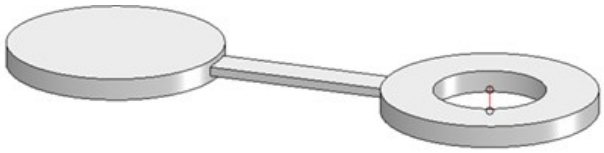
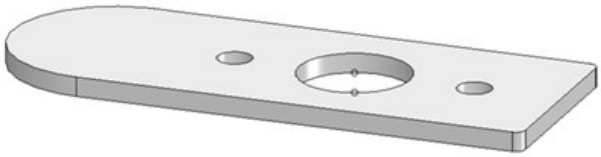


A special feature is the insertion of flanges in all places. If no position is individually pre-marked, counterflanges are matched only with identical flanges. However, flanging can also be performed at free ends by preselecting a free end.

Line blanks acc. to DIN2626

The parts inventory has been expanded to include line blanks acc. to DIN 2626. The new parts are divided into six groups:

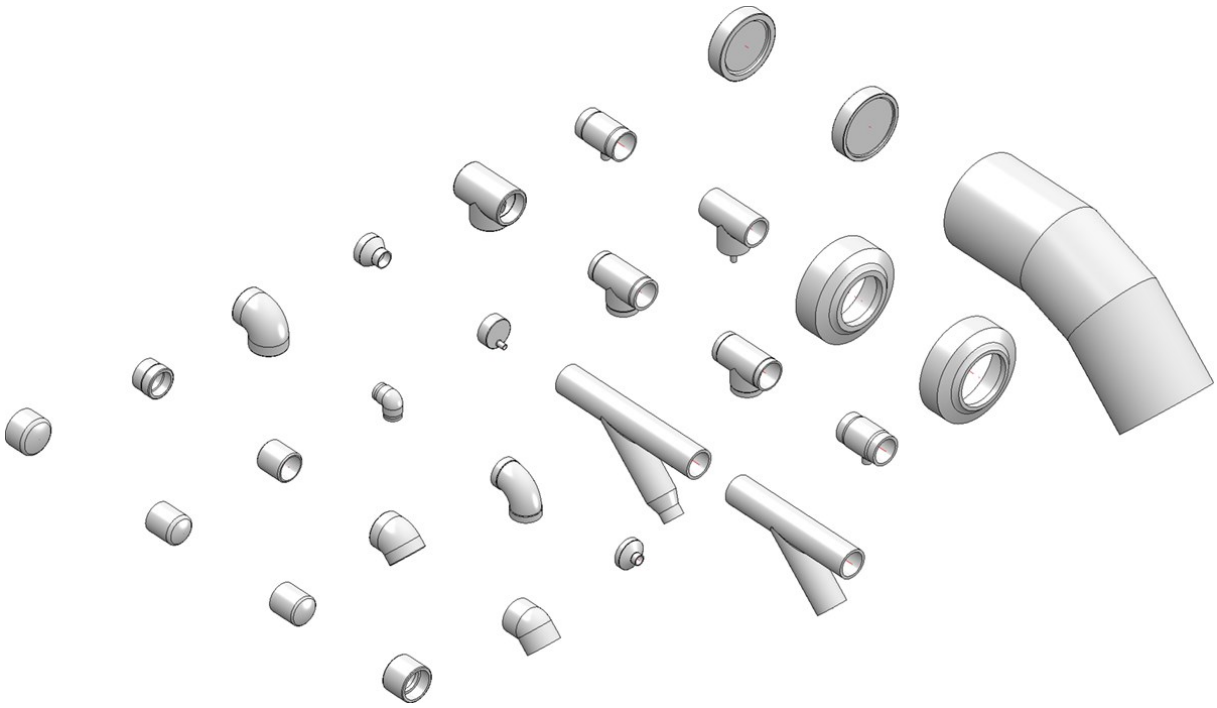
DIN2626 TYP A 	**B**lind **D**isk Variant files: ■ DIN2626-TYP_A-BD-PN10.VAA ■ DIN2626-TYP_A-BD-PN100.VAA ■ DIN2626-TYP_A-BD-PN16.VAA ■ DIN2626-TYP_A-BD-PN160.VAA ■ DIN2626-TYP_A-BD-PN25.VAA ■ DIN2626-TYP_A-BD-PN40.VAA ■ DIN2626-TYP_A-BD-PN6.VAA ■ DIN2626-TYP_A-BD-PN63.VAA You can find these variants summarized in the list file **DIN2626-TYP_A-BD.lst**.
DIN2626 TYP A 	**P**erforated **D**isk Variant files: ■ DIN2626-TYP_B-PD-PN10.VAA ■ DIN2626-TYP_B-PD-PN100.VAA ■ DIN2626-TYP_B-PD-PN16.VAA ■ DIN2626-TYP_B-PD-PN160.VAA ■ DIN2626-TYP_B-PD-PN25.VAA ■ DIN2626-TYP_B-PD-PN40.VAA ■ DIN2626-TYP_B-PD-PN6.VAA ■ DIN2626-TYP_B-PD-PN63.VAA You can find these variants summarized in the list file **DIN2626-TYP_B-PD.lst**.
DIN2626 TYP C 	**O**rifice **P**late ■ DIN2626-TYP_C-OP-PN10.VAA ■ DIN2626-TYP_C-OP-PN100.VAA ■ DIN2626-TYP_C-OP-PN16.VAA ■ DIN2626-TYP_C-OP-PN160.VAA ■ DIN2626-TYP_C-OP-PN25.VAA ■ DIN2626-TYP_C-OP-PN40.VAA ■ DIN2626-TYP_C-OP-PN6.VAA ■ DIN2626-TYP_C-OP-PN63.VAA You can find these variants summarized in the list file **DIN2626-TYP_C-OP.lst**.

DIN2626 TYP D 	Spectacle Blind As A Figure-8 Blank Variant files: DIN2626-TYP_D-SB8B-PN10.VAA DIN2626-TYP_D-SB8B-PN100.VAA DIN2626-TYP_D-SB8B-PN16.VAA DIN2626-TYP_D-SB8B-PN160.VAA DIN2626-TYP_D-SB8B-PN25.VAA DIN2626-TYP_D-SB8B-PN40.VAA DIN2626-TYP_D-SB8B-PN6.VAA DIN2626-TYP_D-SB8B-PN63.VAA You can find these variants summarized in the list file DIN2626-TYP_D-SB8B.lst.
DIN2626 TYP E1 	Spectacle Blind As A Rotating Plate The E1 subtype covers nominal sizes from DN 15 to DN 50. Variant files: ■ DIN2626-TYP_E1-SBRP-PN10.VAA ■ DIN2626-TYP_E1-SBRP-PN16.VAA ■ DIN2626-TYP_E1-SBRP-PN25.VAA ■ DIN2626-TYP_E1-SBRP-PN40.VAA You can find these variants summarized in the list file DIN2626-TYP_E1-SBRP.lst.
DIN2626 TYP E2 	Spectacle Blind As A Rotating Plate The E2 subtype covers nominal sizes from DN 65 to DN 200. Variant files: ■ DIN2626-TYP_E2-SBRP-PN10.VAA ■ DIN2626-TYP_E2-SBRP-PN16.VAA ■ DIN2626-TYP_E2-SBRP-PN25.VAA ■ DIN2626-TYP_E2-SBRP-PN40.VAA You can find these variants summarized in the list file DIN2626-TYP_E2-SBRP.lst.

GF Piping Systems

PROGEF

The parts inventory has been extended by parts from the manufacturer standard PROGEF of the supplier Georg Fischer.

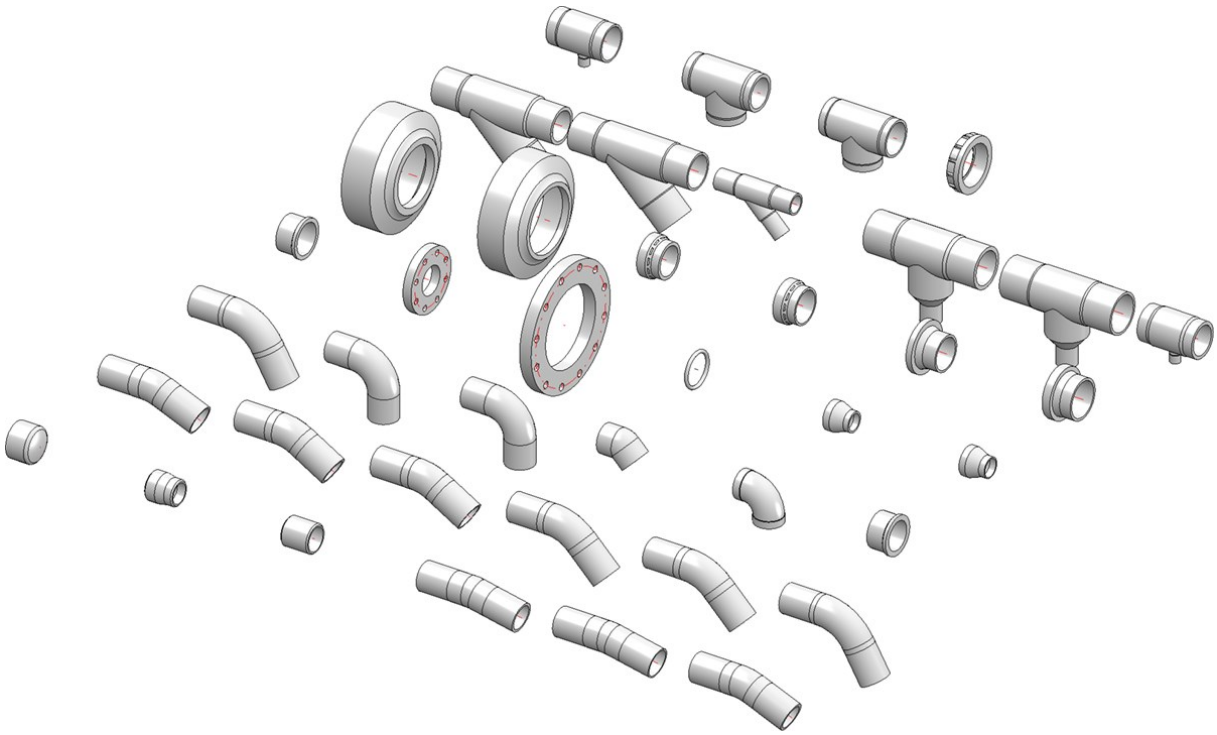


The parts are summarized in the list file **PROGEF.lst**. It contains the following variants:

File	Designation	Type
PROGEF_BEND_45_SDR11.VAA	Segment bend 45° SDR 11	Knee
PROGEF_BEND_45_SDR17_6.VAA	Segment bend 45° SDR 17.6	Knee
PROGEF_CAP_COUPLER_SDR11.VAA	Socket welded end cap SDR 11	Cap
PROGEF_CAP_L_SDR11.VAA	Butt welded end cap L SDR 11	Cap
PROGEF_CAP_L_SDR17_6.VAA	Butt welded end cap L SDR 17.6	Cap
PROGEF_CAP_SDR11.VAA	Butt welded end cap SDR 11	Cap
PROGEF_CAP_SDR17_6.VAA	Butt welded end cap SDR 17.6	Cap
PROGEF_COUPLER_FUSION_SDR11.VAA	Double socket SDR 11	Other pipe part
PROGEF_COUPLER_REDUCER_FUSION_ SDR11.VAA	Socket welded reducer SDR11	Reducer, con-centric
PROGEF_DOUBLENIPPLE_FUSION_ SDR11.VAA	Socket welded double nipple SDR11	Other pipe part
PROGEF_ELBOW_45_FUSION_SDR11.VAA	Elbow 45° L SDR 11 socket welded	Knee
PROGEF_ELBOW_45_L_SDR11.VAA	Elbow 45° L SDR 11	Knee
PROGEF_ELBOW_90_FUSION_SDR11.VAA	Elbow 90° socket welded SDR11	Knee
PROGEF_ELBOW_90_SHORT_SDR11.VAA	Elbow 90° short SDR11	Knee
PROGEF_ELBOW_SWEEP_90_SDR11.VAA	Elbow 90° SDR 11	Knee
PROGEF_PIPE_SDR7_4.VAA	PE100-Pressure pipe SDR7.4	Straight pipe
PROGEF_PIPE_SDR11.VAA	PE100-Pressure pipe SDR11	Straight pipe
PROGEF_PIPE_SDR17_6.VAA	PE100-Pressure pipe SDR17.6	Straight pipe
PROGEF_REDUCER_BW_SDR11.VAA	Butt welded reducer SDR 11	Reducer, con-centric
PROGEF_REDUCER_BW_SDR17_6.VAA	Butt-welded reducer SDR 17.6	Reducer, con-centric
PROGEF_REDUCER_ECCENTRIC_SDR11.VAA	Butt- welded reducer, eccentric SDR 11	Reducer, eccent-ric
PROGEF_REDUCER_SHORT_BW_SDR11.VAA	Butt welded reducer, short SDR 11	Reducer, con-centric
PROGEF_REDUCER_SHORT_BW_SDR17_ 6.VAA	Butt welded reducer, short SDR 17.6	Reducer, con-centric
PROGEF_TEE_45_RED_SDR11.VAA	T 45° reduced SDR 11	Branch
PROGEF_TEE_45_SDR11.VAA	T 45° equal SDR 11	Branch
PROGEF_TEE_FUSION_SDR11.VAA	T 90° SDR socket welded SDR 11	T-piece
PROGEF_TEE_L_SDR11.VAA	T 90° L SDR 11	T-piece
PROGEF_TEE_L_SDR17_6.VAA	T 90° L SDR 17.6	T-piece
PROGEF_TEE_RED_MOLDED_SDR11.VAA	T 90° reduced, molded SDR 11	T-piece
PROGEF_TEE_RED_MOLDED_SDR17_6.VAA	T 90° reduced, molded SDR 17.6	T-piece
PROGEF_TEE_RED_REDUCER_SDR11.VAA	T 90° butt welded reducer SDR11	T-piece

ECOFIT

The parts inventory has been extended by parts from the manufacturer standard ECOFIT of the supplier Georg Fischer.



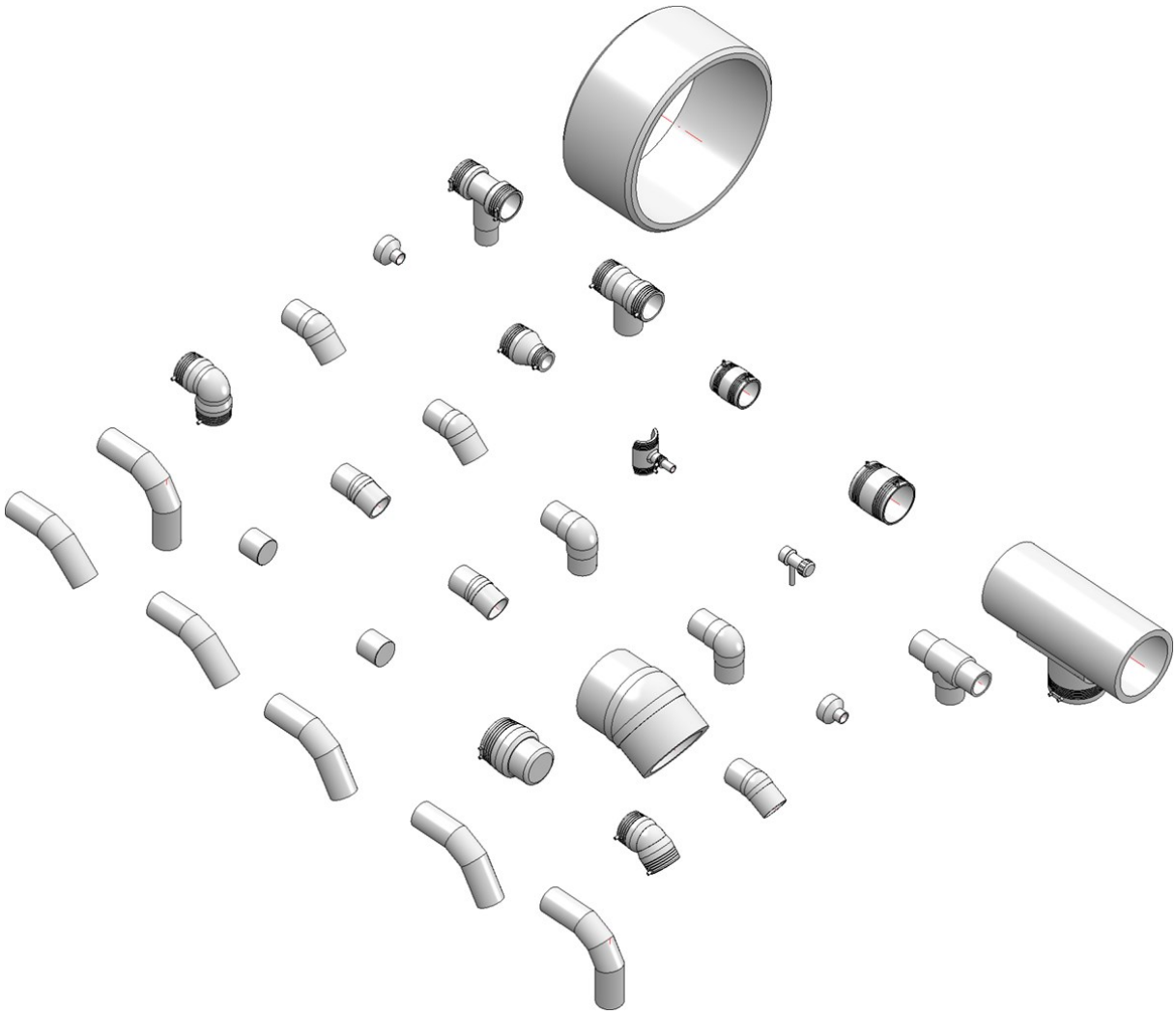
The parts are summarized in the list file **ECOFIT.lst** It contains the following variants:

File	Designation	Type
ECOFIT_CAP_SDR11.VAA	End cap SDR 11	Cap
ECOFIT_COUPLER_REDUCER_SDR11.VAA	Socket welded SDR 11	Other pipe part
ECOFIT_DOUBLENIPPLE_SDR17.VAA	Double nipple SDR 17	Other pipe part
ECOFIT_ELBOW_11_SDR11.VAA	Elbow 11° SDR 11	Knee
ECOFIT_ELBOW_11_SDR17.VAA	Elbow 11° SDR 17	Knee
ECOFIT_ELBOW_22_SDR11.VAA	Elbow 22° SDR 11	Knee
ECOFIT_ELBOW_22_SDR17.VAA	Elbow 22° SDR 17	Knee
ECOFIT_ELBOW_30_SDR11.VAA	Elbow 30° SDR 11	Knee
ECOFIT_ELBOW_30_SDR17.VAA	Elbow 30° SDR 17	Knee
ECOFIT_ELBOW_45_SDR11.VAA	Elbow 45° SDR 11	Knee
ECOFIT_ELBOW_45_SDR17.VAA	Elbow 45° SDR 17	Knee
ECOFIT_ELBOW_60_SDR11.VAA	Elbow 60° SDR 11	Knee
ECOFIT_ELBOW_60_SDR17.VAA	Elbow 60° SDR 17	Knee
ECOFIT_ELBOW_90_SDR11.VAA	Elbow 90° SDR 11	Knee
ECOFIT_ELBOW_90_SDR17.VAA	Elbow 90° SDR 17	Knee
ECOFIT_ELBOW_SHORT_45_SDR11.VAA	Elbow 45° SDR 11, kurz	Knee
ECOFIT_ELBOW_SHORT_90_SDR11.VAA	Elbow 90° SDR 11, kurz	Knee
ECOFIT_INSERT_COMPONENT_ SDR11.VAA	Bolting insert SDR 11	Flange
ECOFIT_INSERT_COMPONENT_ SDR17.VAA	Bolting insert SDR 11	Flange
ECOFIT_LOOSEFLANGE_SDR11.VAA	Loose flange, butt welded SDR 11	Flange
ECOFIT_LOOSEFLANGE_SDR17.VAA	Loose flange, butt welded SDR 17	Flange
ECOFIT_ORING.VAA	O-ring gasket	Seal
ECOFIT_PIPE_SDR11.VAA	PE100 pressure pipe SDR 11	Straight pipe
ECOFIT_PIPE_SDR17.VAA	PE100 pressure pipe SDR 17	Straight pipe
ECOFIT_PIPE_SDR17-FM.VAA	PE100 pressure pipe SDR 17, FM	Straight pipe
ECOFIT_PIPE_SDR41.VAA	PE100 pressure pipe SDR 41	Straight pipe
ECOFIT_PIPE_SDR7_4.VAA	PE100 pressure pipe SDR 7.4	Straight pipe
ECOFIT_REDUCER_BW_SDR11.VAA	Butt welded reducer SDR 11	Reducer, concentric
ECOFIT_REDUCER_BW_SDR17.VAA	Butt welded reducer SDR 17	Reducer, concentric
ECOFIT_REDUCER_SHORT_BW_ SDR11.VAA	Butt welded reducer SDR 11, short	Reducer, concentric
ECOFIT_REDUCER_SHORT_BW_ SDR17.VAA	Butt welded reducer SDR 17, short	Reducer, concentric
ECOFIT_SCREW_FITTING_SDR11.VAA	Bolting screw-in part SDR 11	Flange
ECOFIT_SCREW_FITTING_SDR17.VAA	Bolting screw-in part SDR 17	Flange
ECOFIT_STUB_FLANGE_A_SDR17.VAA	Butt welded collar Type A SDR 17	Flange

File	Designation	Type
ECOFIT_STUB_FLANGE_B_SDR17.VAA	Butt welded collar Type B SDR 17	Flange
ECOFIT_TEE_45_SDR11.VAA	T 45° equal SDR 11	Branch
ECOFIT_TEE_45_SDR17.VAA	T 45° equal SDR 17	Branch
ECOFIT_TEE_RED_45_SDR11.VAA	T 45° reduced SDR 11	Branch
ECOFIT_TEE_RED_REDUCER_SDR11.VAA	T 90° butt welded reducer SDR 11	T-piece
ECOFIT_TEE_RED_REDUCER_SDR17.VAA	T 90° butt welded reducer SDR 17	T-piece
ECOFIT_TEE_RED_SHORT_SDR11.VAA	T 90° reduced SDR 11, short	T-piece
ECOFIT_TEE_RED_SHORT_SDR17.VAA	T 90° reduced SDR 17, short	T-piece
ECOFIT_TEE_SHORT_SDR11.VAA	T 90° SDR 11, short	T-piece
ECOFIT_TEE_SHORT_SDR17.VAA	T 90° SDR 17, short	T-piece
ECOFIT_UNION_NUT.VAA	Bolting - union nut	Fastener, unsym-metric

ELGEF

The parts inventory has been extended by parts from the manufacturer standard ECOFIT of the supplier Georg Fischer.



The parts are summarized in the list file **ELGEF.lst** It contains the following variants:

File	Designation	Type
ELGEF_BEND_45_SDR11.VAA	Segment bend 45° SDR 11	Knee
ELGEF_BEND_45_SDR17.VAA	Segment bend45° SDR 17	Knee
ELGEF_BEND_60_SDR11.VAA	Segment bend60° SDR 11	Knee
ELGEF_BEND_60_SDR17.VAA	Segment bend60° SDR 17	Knee
ELGEF_BEND_90_SDR11.VAA	Segment bend90° SDR 11	Knee
ELGEF_BEND_90_SDR17.VAA	Segment bend90° SDR 17	Knee
ELGEF_CAP_L_SDR11.VAA	End cap Type L SDR 11	Cap
ELGEF_CAP_L_SDR17.VAA	End cap Type L SDR 17	Cap
ELGEF_CAP_SDR11.VAA	End cap SDR 11	Cap
ELGEF_COUPLER_SDR11.VAA	Socket SDR 11	Other pipe part
ELGEF_COUPLER_SDR17.VAA	Socket SDR 17	Other pipe part
ELGEF_COUPLER_SDR26.VAA	Socket SDR 26	Other pipe part
ELGEF_ELBOW_45_SDR11.VAA	Elbow 45° SDR 11	Knee
ELGEF_ELBOW_90_SDR11.VAA	Elbow 90° SDR 11	Knee
ELGEF_ELBOW_L_15_SDR11.VAA	Elbow 15° Typ L SDR 11	Knee
ELGEF_ELBOW_L_15_SDR17.VAA	Elbow 15° Typ L SDR 17	Knee
ELGEF_ELBOW_L_30_SDR11.VAA	Elbow 30° Typ L SDR 11	Knee
ELGEF_ELBOW_L_30_SDR17.VAA	Elbow 30° Typ L SDR 17	Knee
ELGEF_ELBOW_L_45_SDR11.VAA	Elbow 45° Typ L SDR 11	Knee
ELGEF_ELBOW_L_45_SDR17.VAA	Elbow 45° Typ L SDR 17	Knee
ELGEF_ELBOW_L_90_SDR11.VAA	Elbow 90° Typ L SDR 11	Knee
ELGEF_ELBOW_L_90_SDR17.VAA	Elbow 90° Typ L SDR 17	Knee
ELGEF_REDUCER_BW_SDR11.VAA	Butt welded reducer SDR 11	Reducer, concentric
ELGEF_REDUCER_BW_SDR17.VAA	Butt welded reducer SDR 17	Reducer, concentric
ELGEF_REDUCER_SDR11.VAA	Reducer SDR 11	Reducer, concentric
ELGEF_SPIGOT_CLAMP_SDR11.VAA	Spigot clamp SDR 11	Saddle connection
ELGEF_TEE_DRILL_SDR11.VAA	T 90° tapping-T SDR 11	Corner valve
ELGEF_TEE_L_SDR11.VAA	T 90° Type L SDR 11	T-piece
ELGEF_TEE_RED_SADDLE.VAA	T 90° saddle connection reduced SDR 11	T-piece
ELGEF_TEE_RED_SDR11.VAA	T 90° reduced SDR 11	T-piece
ELGEF_TEE_SDR11.VAA	T 90° SDR 11	T-piece

Polyethylene pipes acc. to DIN 8074

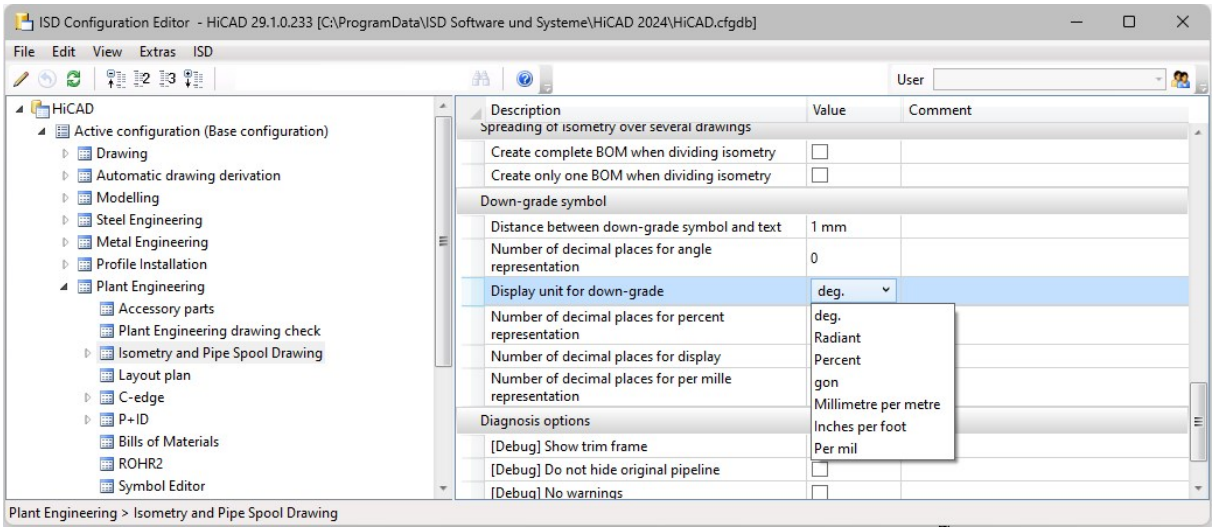
The parts inventory was extended by polyethylene pipes according to DIN 8074. The parts are summarized in the list file **N8074.lst**. It contains the following variants:

File	Designation	Type
N8074_SDR11.VAA	PE 100 pressure pipe	Straight pipe
N8074_SDR17.VAA	PE 100 pressure pipe	Straight pipe
N8074_SDR17_6.VAA	PE 100 pressure pipe	Straight pipe
N8074_SDR26.VAA	PE 100 pressure pipe	Straight pipe
N8074_SDR33.VAA	PE 100 pressure pipe	Straight pipe
N8074_SDR7_4.VAA	PE 100 pressure pipe	Straight pipe

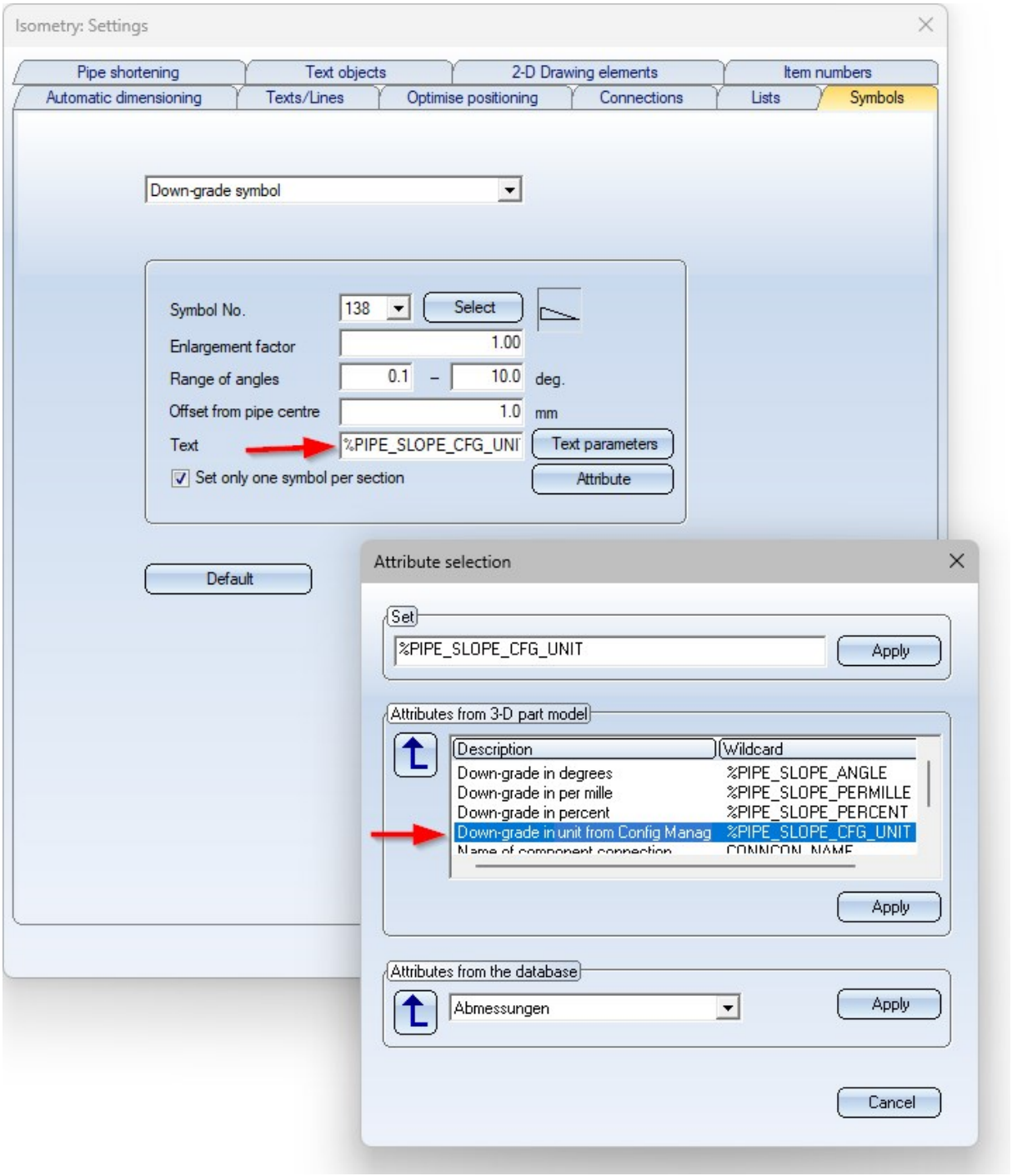
Isometry and pipe spool drawing

Down-grade symbol

In the Configuration Editor, the unit for the display of the down-grade symbol can now be selected. In addition, only one setting option for the number of decimal places is now available there.

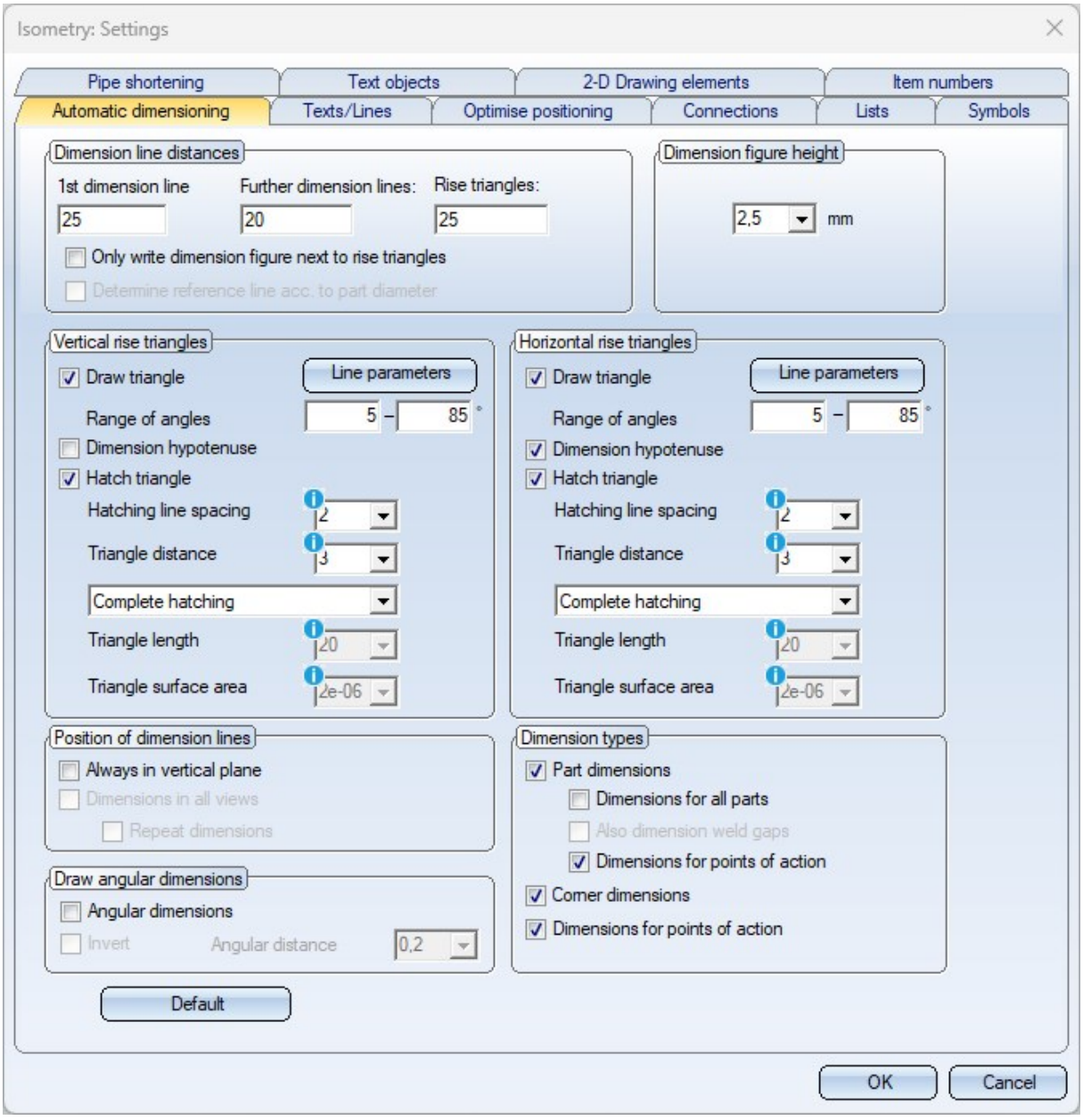


In the isometry/pipe spool drawing settings for down-grade symbols (open the **Symbols** tab), the new text key **%PIPE_SLOPE_CFG_UNIT** can be used to display the unit of the down-grade symbol in the isometry according to the unit specified in the Configuration Editor.

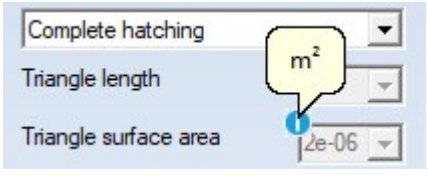


Units in the isometry and pipe spool drawing settings

Different units are now also supported in the settings for isometry and pipe spool drawing. This means that in many cases the mm display is no longer shown in the dialogues, e.g.



The values can be entered in any desired unit and are then automatically converted into the unit of measurement preset in the Configuration Editor. If you point the cursor at the symbol, the preset unit is displayed.



The font height of the dimension figure must be specified in mm.

Generate pipe book

As of HiCAD 2024 SP1, so-called pipe books can be generated for isometries or pipe spool drawings. To do this, go to **Isometry+Pipe Spool Drawing > Edit lists > EditPL >...** and choose the new the new **Pipe book** function.

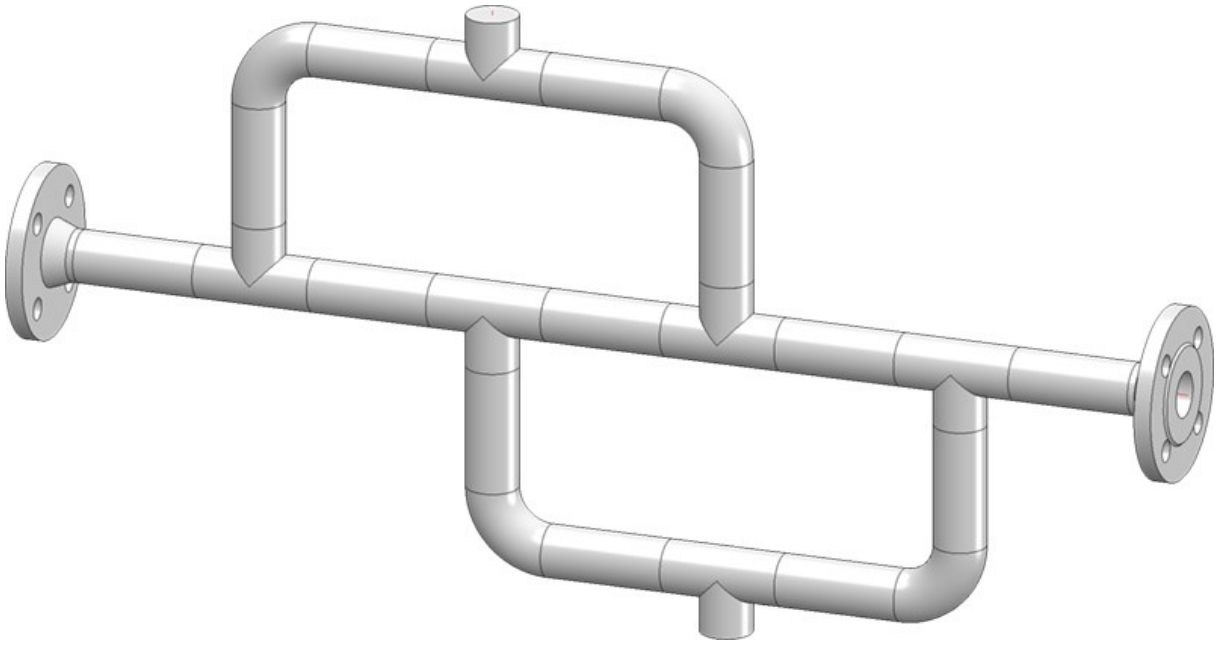
A pipe book - also known as a weld book or weld seam book - is a special bill of materials that contains all relevant information about weld seams and parts of a pipeline. This includes part information such as standards and dimensions, detailed information on the material, the weld seam and the welding process.

Microsoft® Excel is required for creating a pipe book.

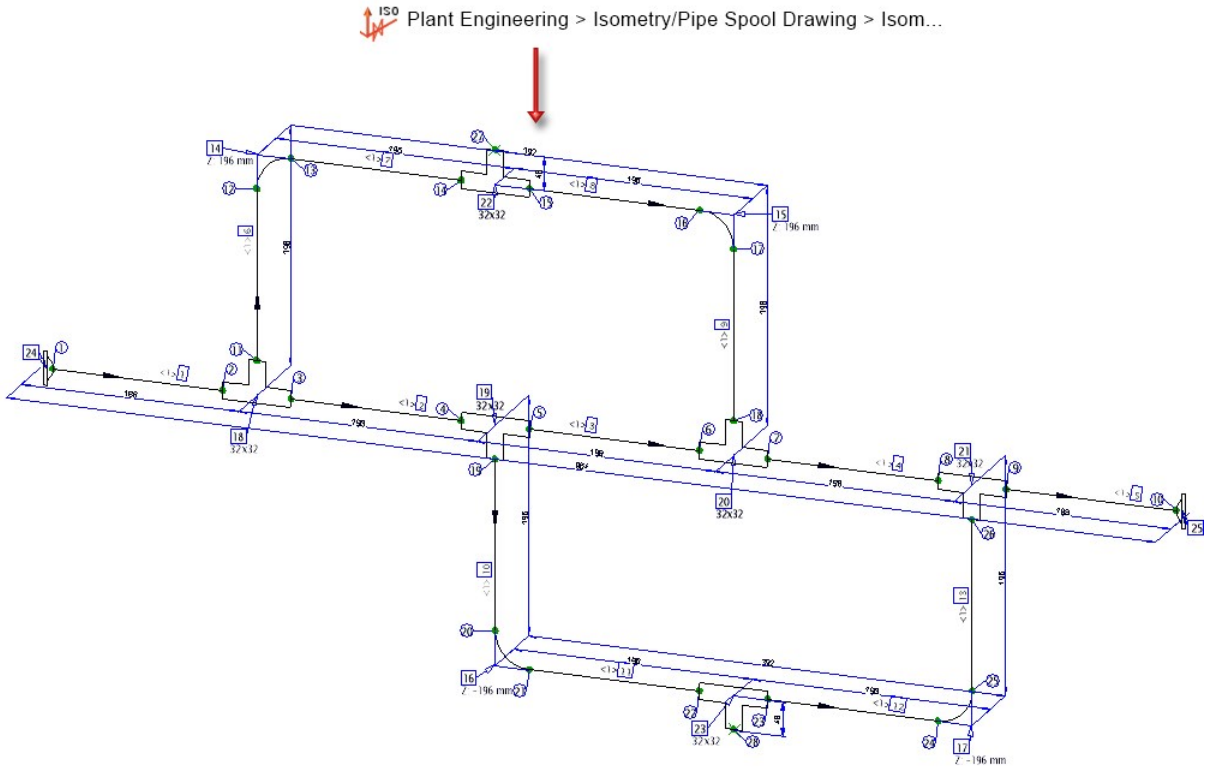
An isometry or pipe spool drawing must be available to generate a pipe book. This is necessary because the item numbers of the welded joints are required for the pipe book and these are only assigned in HiCAD when the isometry or pipe spool drawing is generated.

After calling up the **Pipe book** function, the Report Manager is automatically started with a corresponding quantity list. This quantity list is the basis for the pipe book. You then generate the pipe book itself in the Report Manager using the **Create Excel document (with template or script)** function.

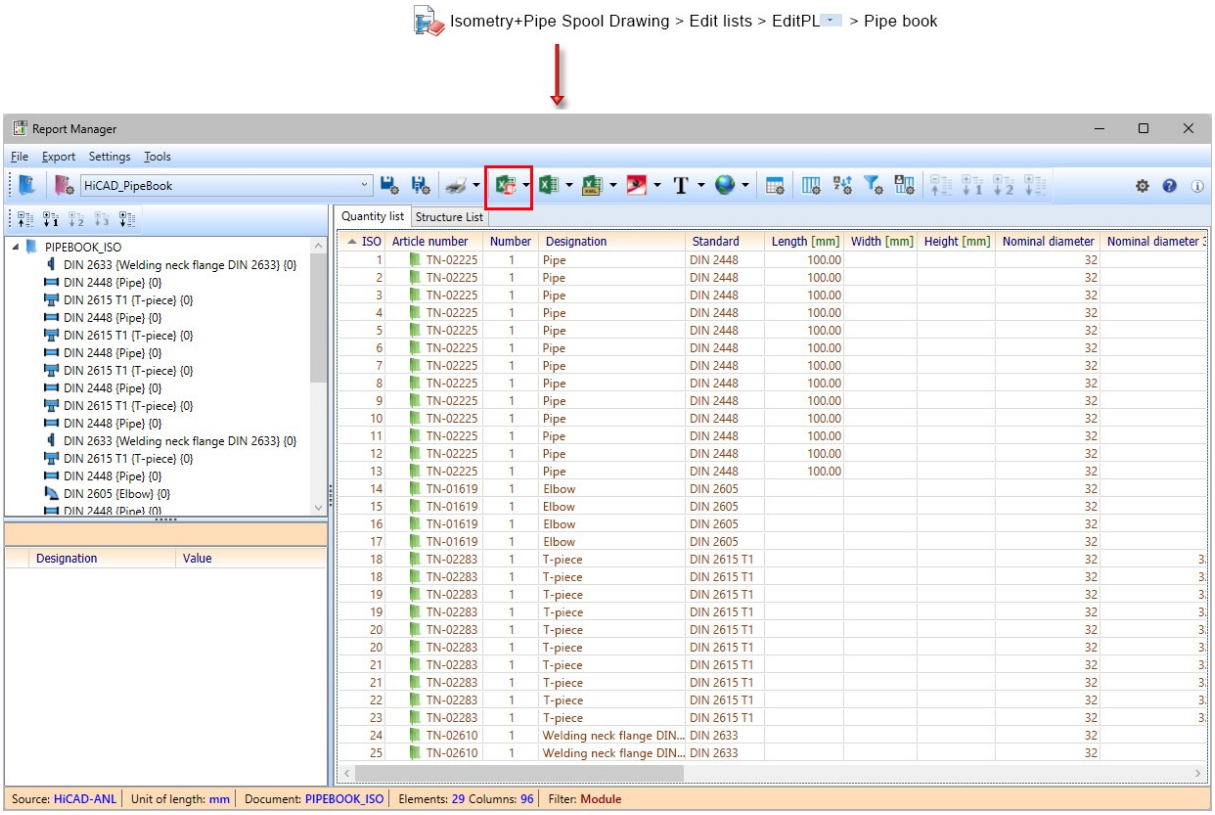
The process is described below using the example shown for an isometry.



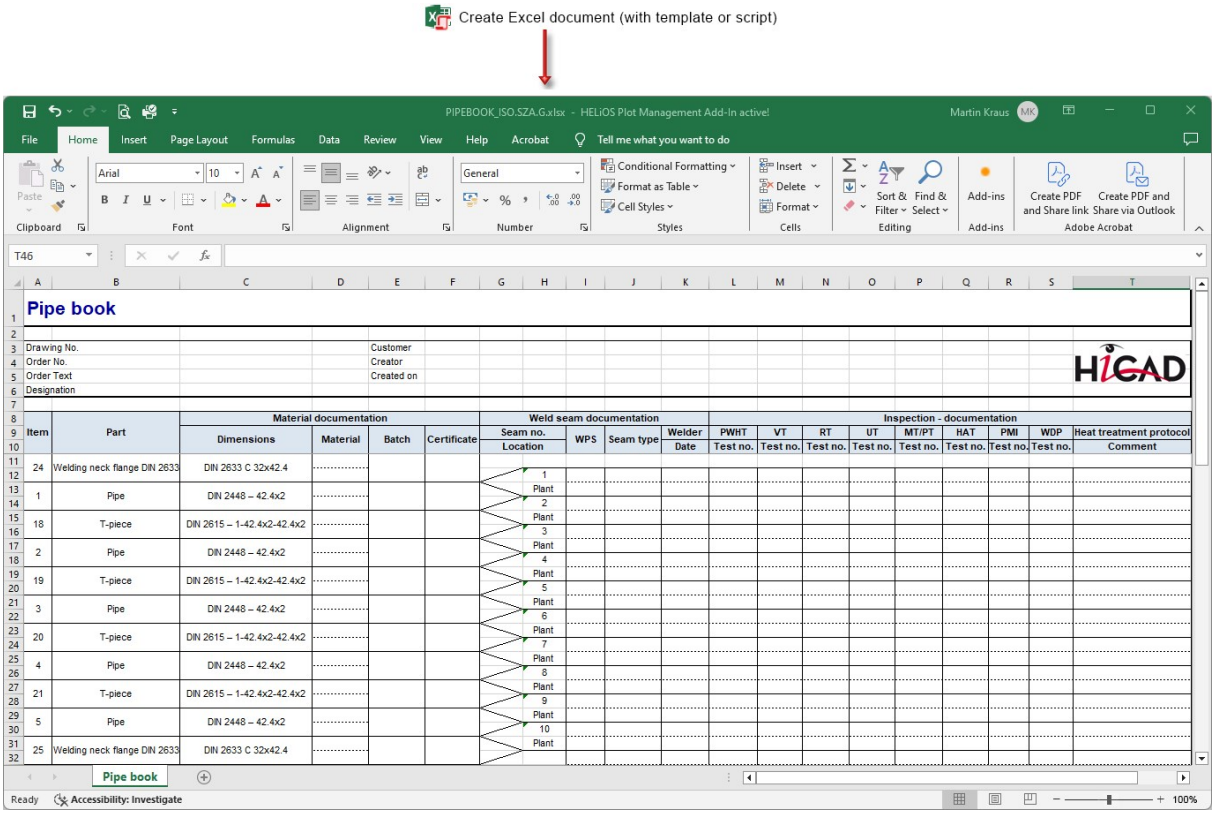
Step 1: Create the isometry



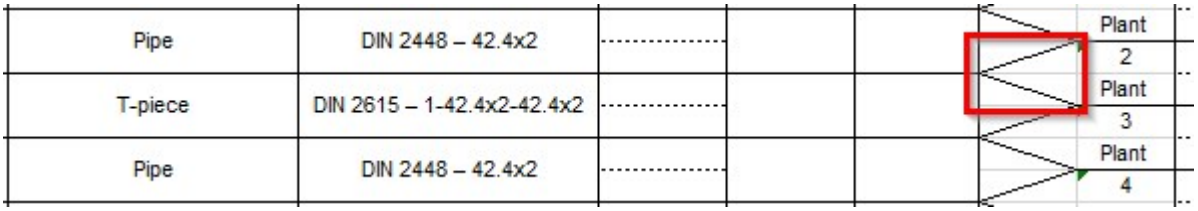
Step 2: Generate a quantity list for the pipe book



Step 3: Create the Excel document



The pipe book itself can be understood as a two-part list. The pipe parts are on the left and the weld seams on the right. Each weld seam is located between two pipe parts, which is indicated in the table by the opening wedges between the two areas.



The table is built up section by section, with the pipe parts on the sections following the direction of flow. Accordingly, parts can occur several times. To make this clearer in the example above, the identical part search has been switched off so that each pipe part has its own item number. This means that you can find the T-pieces with item numbers 18, 19, 20 and 21 twice. The example isometry consists of three sections; the gaps in the column with the weld seam items indicate the start of the next section.

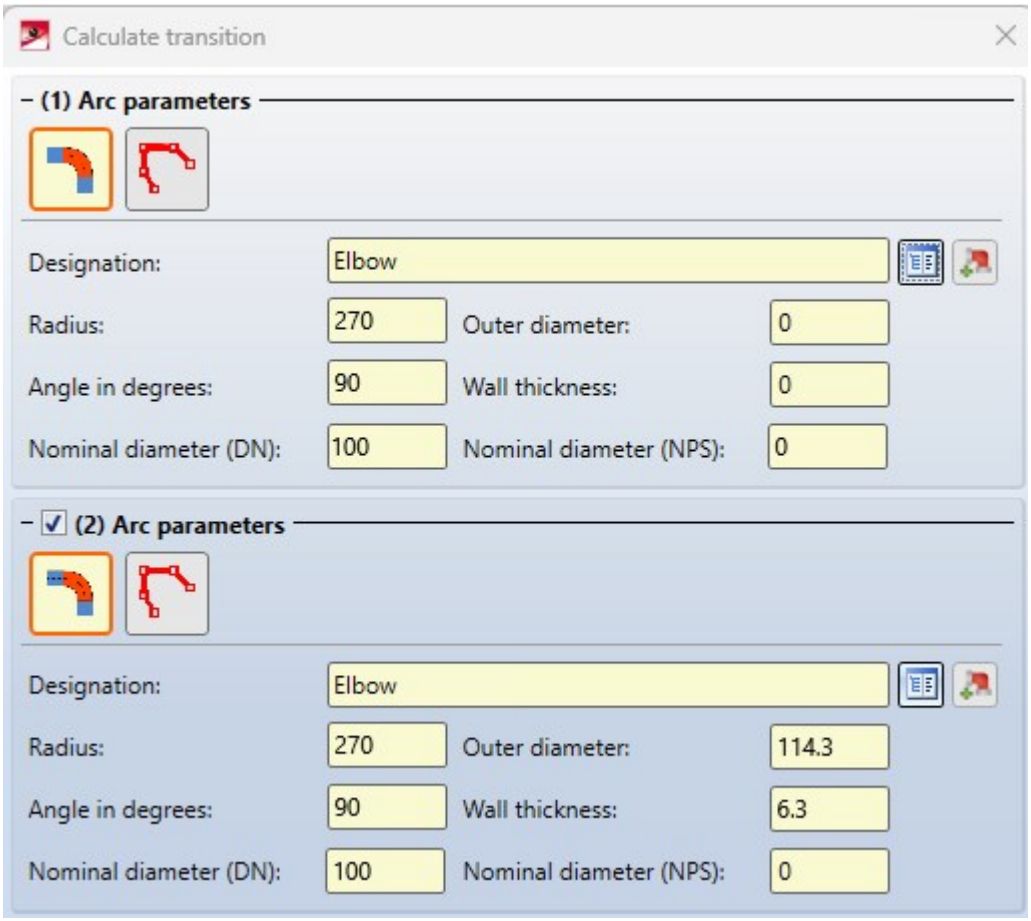
The files listed below, which are all located in the HiCAD sys directory, are relevant for creating the pipe book.

HiCAD_ PipeBook.DE.2901.0.xlsx	The document template for creating the pipe book in Excel.
HiCAD_PipeBook.2901.0.cs	A C# script that fills the Excel template with the data from the Report Manager.
HiCAD_ PipeBook.rm_ settings	The settings for the Report Manager. Here it is specified that the above Excel template and the above C# script should be used.
rm_ anl_ exportpart_ pipebook.hdb	Defines the data that HiCAD transfers to the Report Manager.

Pipeline Tools

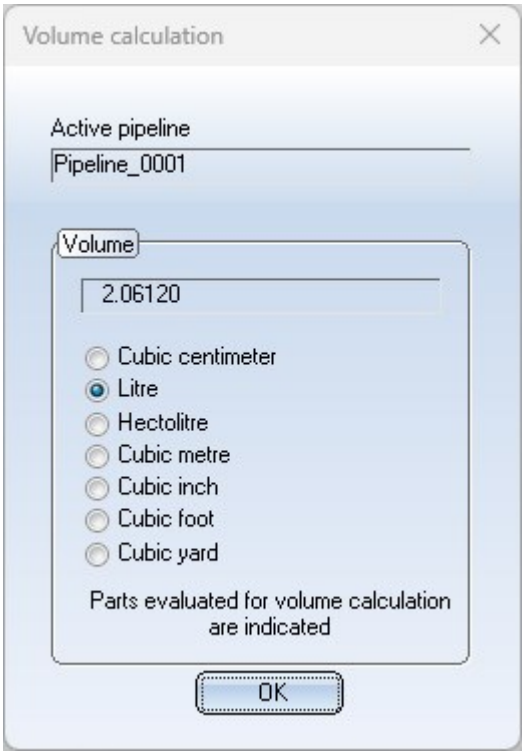
Calculate transition

The NPS nominal diameter is now also displayed in the dialogue for the **Calculate transition** function.



Determine volume

Imperial units are now also available for selection in the **Determine volume** function.



Guideline Editor

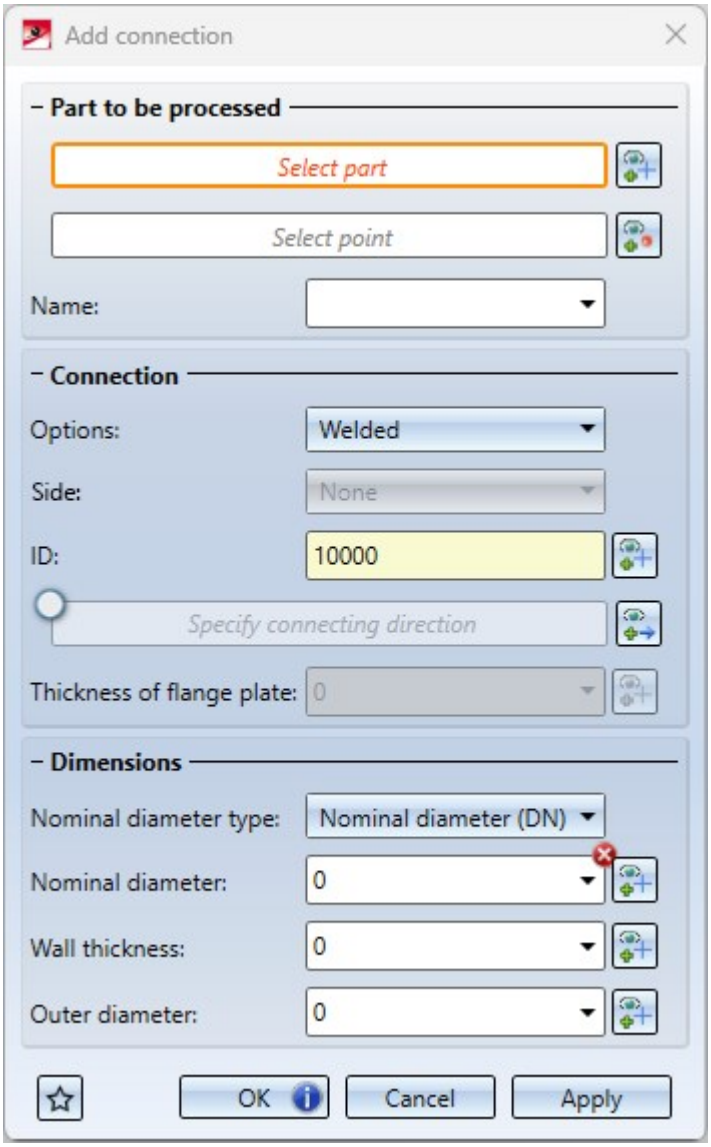
As of SP1, the former Guideline Editor displays the coordinates in the unit selected in the Configuration Editor. In addition, the grid size can be specified here in any unit of length. It is then automatically converted into the preset unit.

Major Release 2024 (V 2900)

Component connections with flange parameters

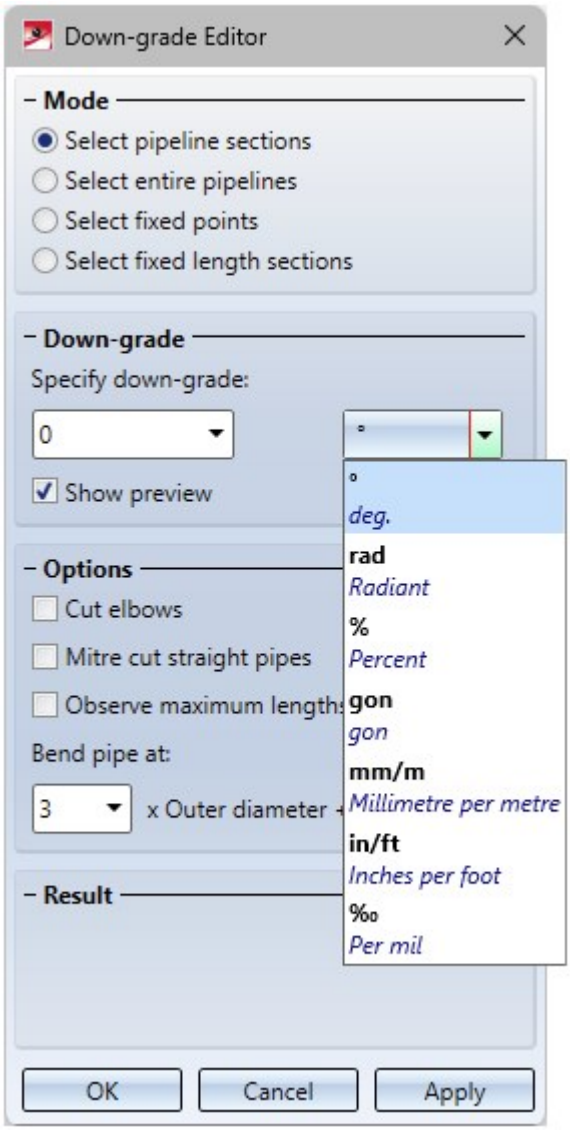
Component connections now also support flange parameters. This makes flanging possible on parts imported via the STEP interface.

The dialogue for creating and editing component connections has been expanded accordingly.



Down-grade Editor - more units

The specification of down-grades in the **Down-grade Editor** can now also be done in Radiant, Gon, mm/m, in/ft and per mil.



Isometry and pipe spool drawing

Generate pipe spool drawing from Sheet view

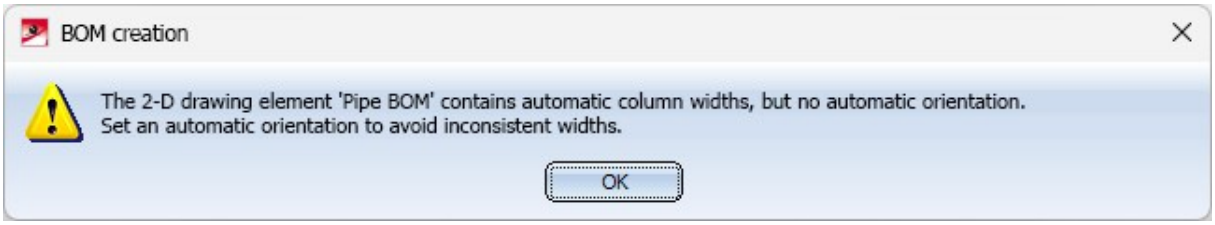
If pipe spool drawings are created in a Sheet area of the active drawing and then a pipe spool drawing is created again from this Sheet view, then exactly those parts are taken into account in the new/updated pipe spool drawing that were also visible in the original Sheet view. This means that in this case you will not be asked to select the parts for the pipe spool drawing.

If the pipe spool drawing is generated from the Model view, then you must select the parts. Unless you have deactivated the checkbox **Part selection before displaying pipe spool drawing dialogue** in the Configuration Editor at **Plant Engineering > Isometry and Pipe Spool Drawing**.

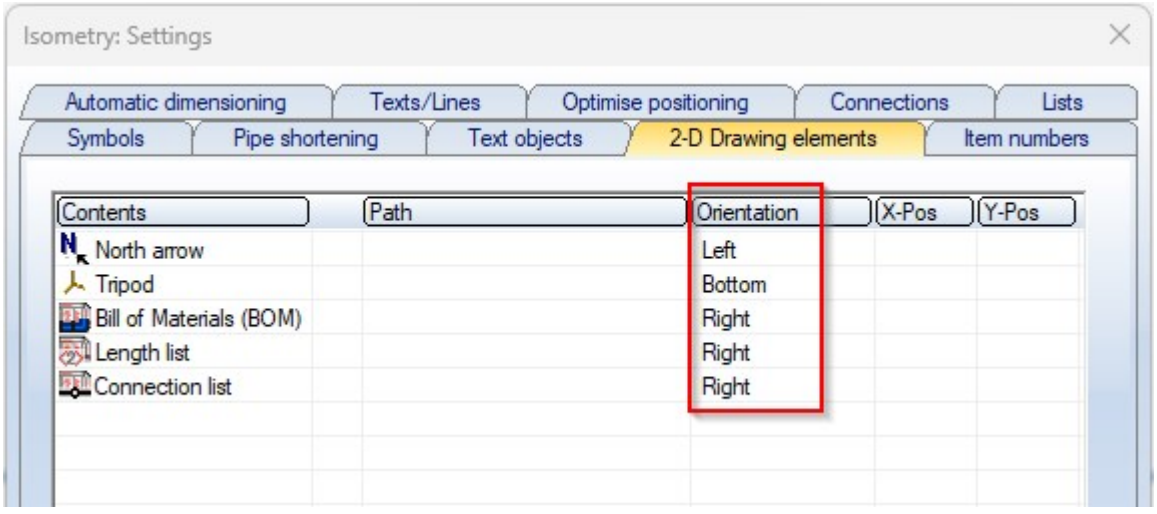
Changed default settings during isometry/pipe spool drawing creation

Orientation of 2-D drawing elements

The previous default settings for the orientation of the 2-D drawing elements of an isometry/pipe spool drawing led to the following message being displayed during a new installation:

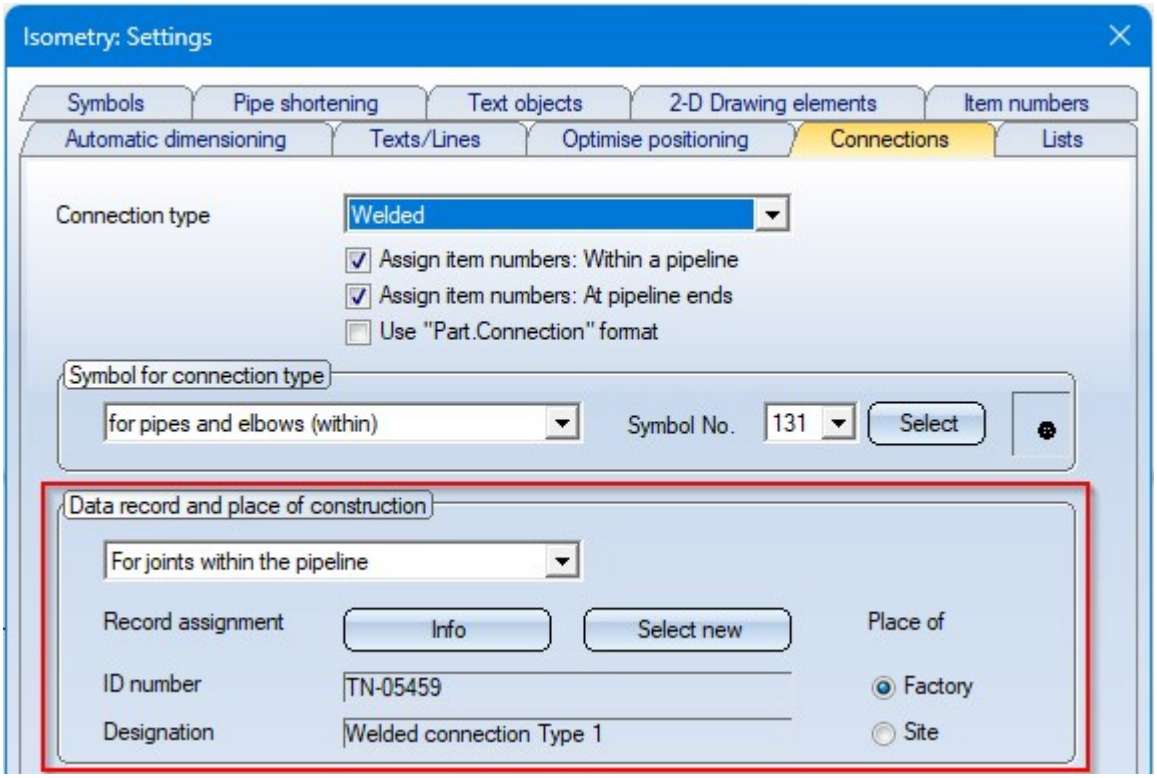


As of HiCAD 2024, these default settings are such that the message no longer appears.

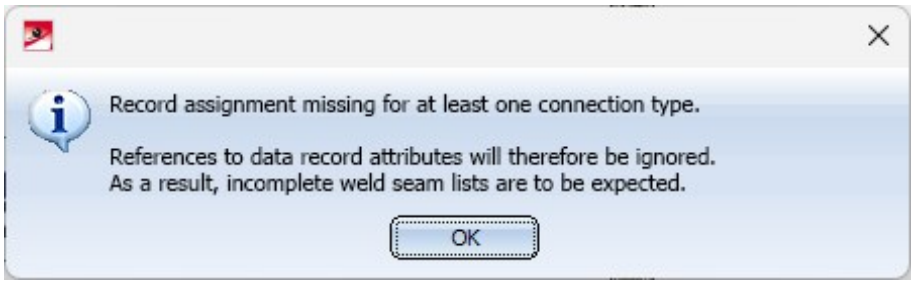


Data record assignment for the connection type

From HiCAD 2024 onwards, a data set assignment is automatically made for all connection types, provided that the default articles are available in the HELiOS database.



This prevents the following message from appearing:



The message will only appear if an automatic assignment is not possible due to missing default articles.

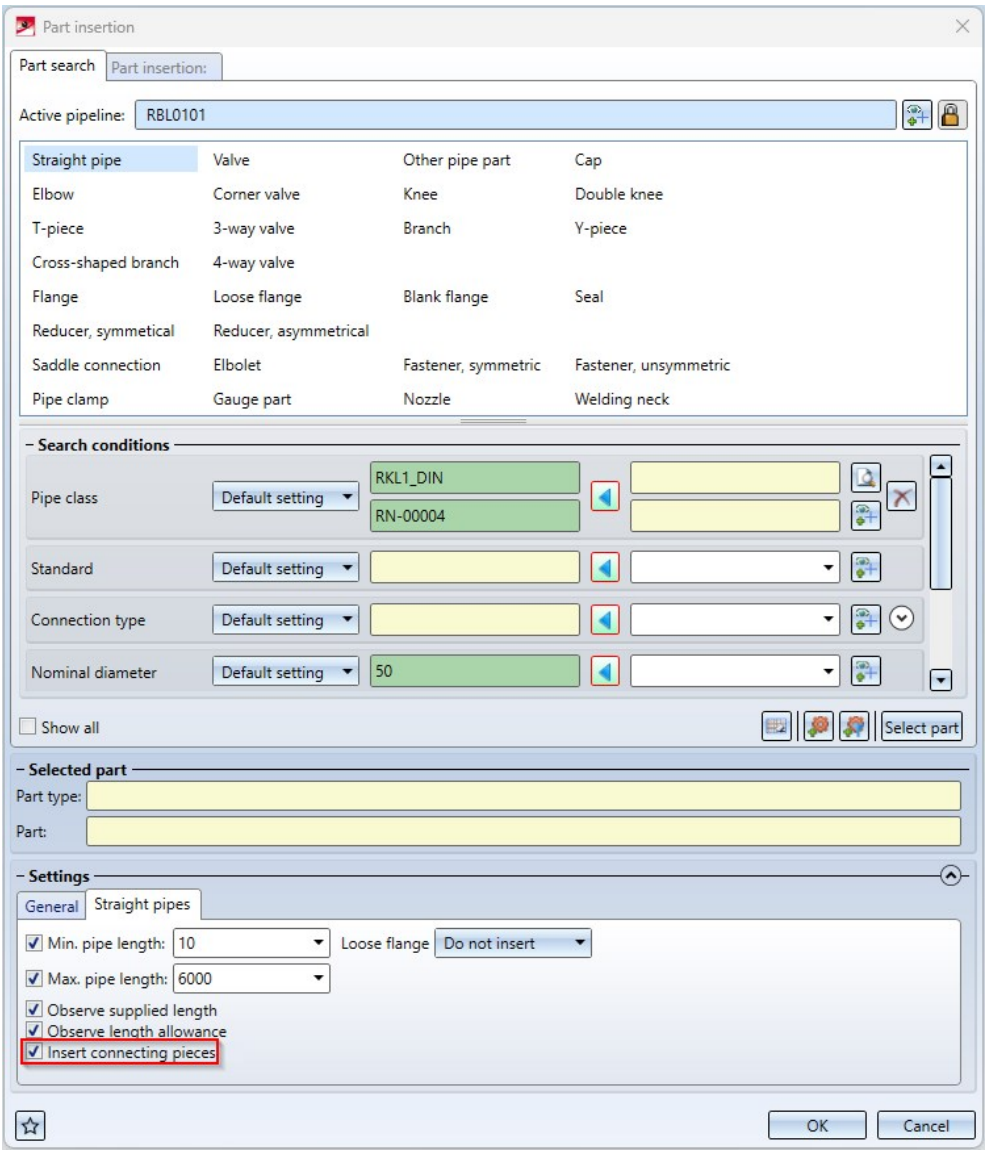
Part insertion

Seach with unit

The function **Pipe parts**  now also supports the specification of units when searching for attributes, e.g.

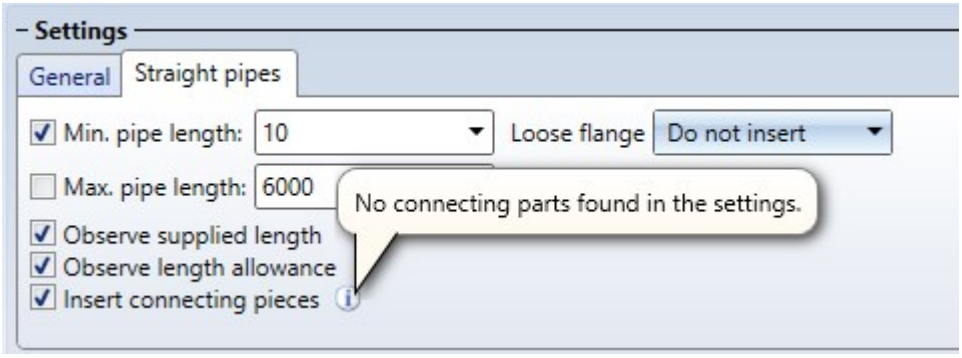
Insert connection pieces

When using the part insertion with the function **Pipe parts**  , the placing of connecting pieces between straight pipes can now be switched on or off. For this purpose, the **Settings** area of the **Straight pipes** tab has been extended accordingly.



If the checkbox **Insert connecting pieces** is active here, the connecting pieces specified in the Plant Engineering Settings for Straight pipes are automatically inserted when straight pipes are inserted.

If no connecting pieces are preselected in the Plant Engineering Settings, this is indicated by the symbol in the **Part insertion** dialogue window.



Please note:
Connecting pieces are also inserted if the checkbox in the Plant Engineering settings for straight pipes is deactivated, but activated for part insertion.

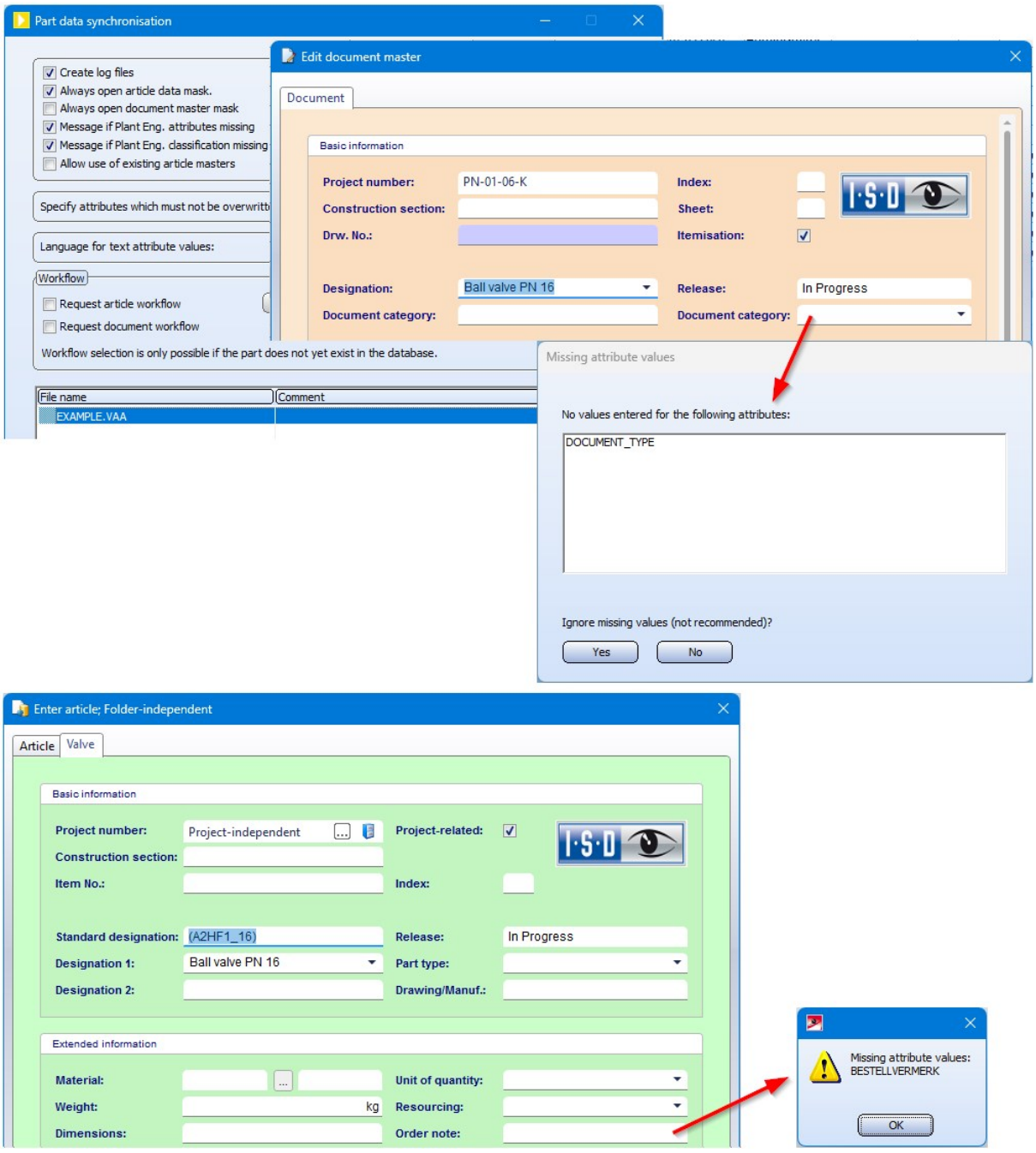
Changed buttons

In the **Part insertion** dialogue window, the button for part selection has been replaced:

Part data synchronisation

Indication of missing attributes

If mandatory attributes in the document or article master are missing during part data synchronisation, for example the document type (DOCUMENT_TYPE) or the order note (BESTELLVERMERK), HiCAD will now indicate this. You then have the possibility to enter the missing attributes. In the case of the document master, you can create it - as before - despite missing attributes, but this is not possible with missing article attributes.

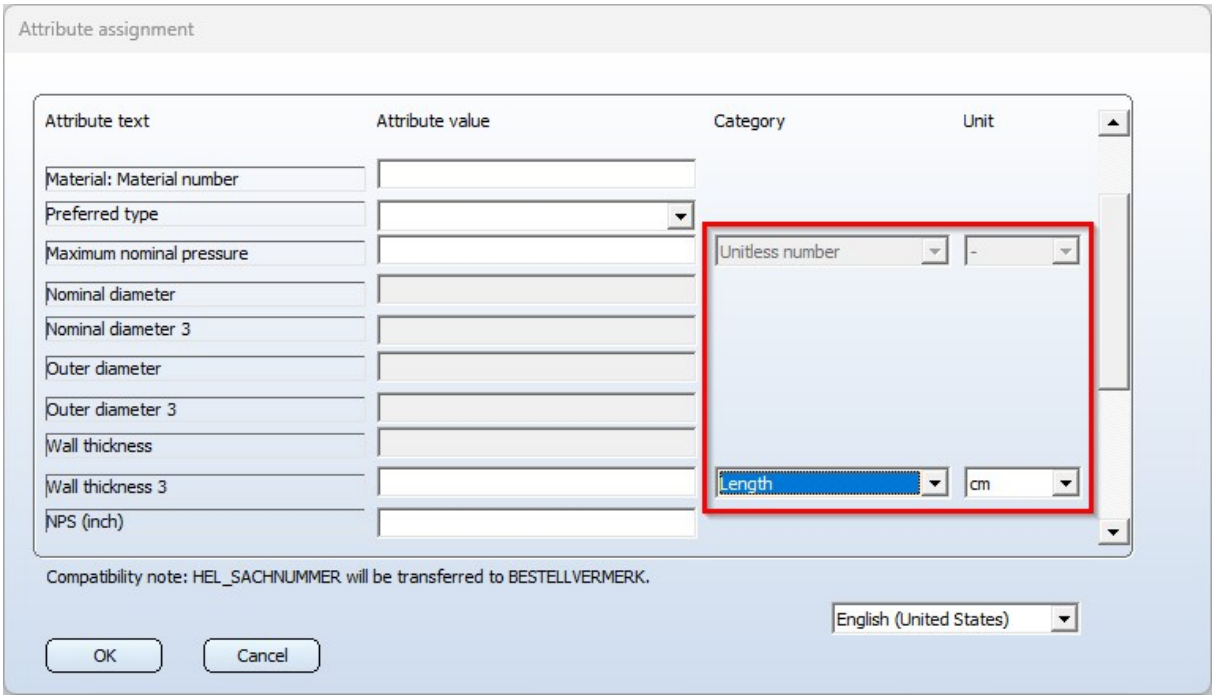


Derived variants with different attributes

If variants are derived on the HELiOS side with the Variant Editor (file selection: with database via document/article master) and attributes of the general type - for example, the material - are changed in the process, then these attributes are not overwritten with the value from the original variant during a data synchronisation.

Synchronisation with catalogue - Units and categories

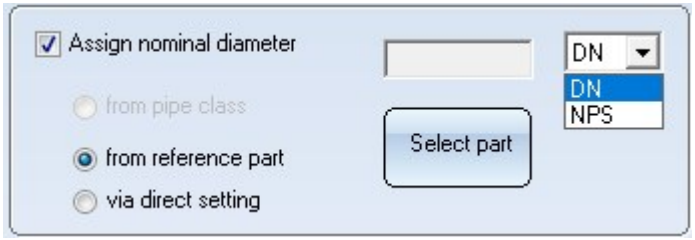
When synchronising part data with the HiCAD catalogue, from HiCAD 2024 it is also possible to select and transfer units and categories when assigning attributes, e.g.:



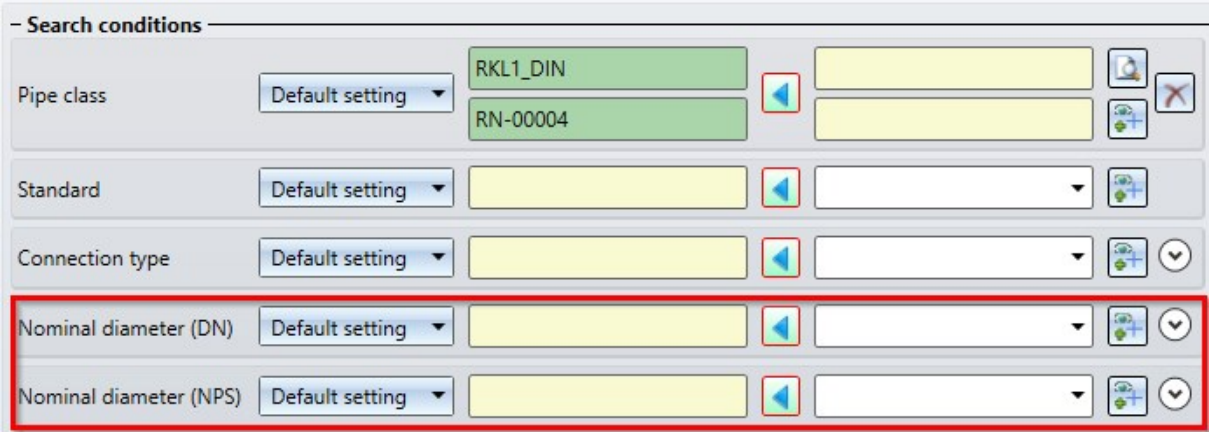
The attributes that expect a **Unitless number** are all Nominal diameter (NENNWEITE) attributes and the attribute Pressure (DRUCK).

Create pipeline - Assigning of nominal diameters

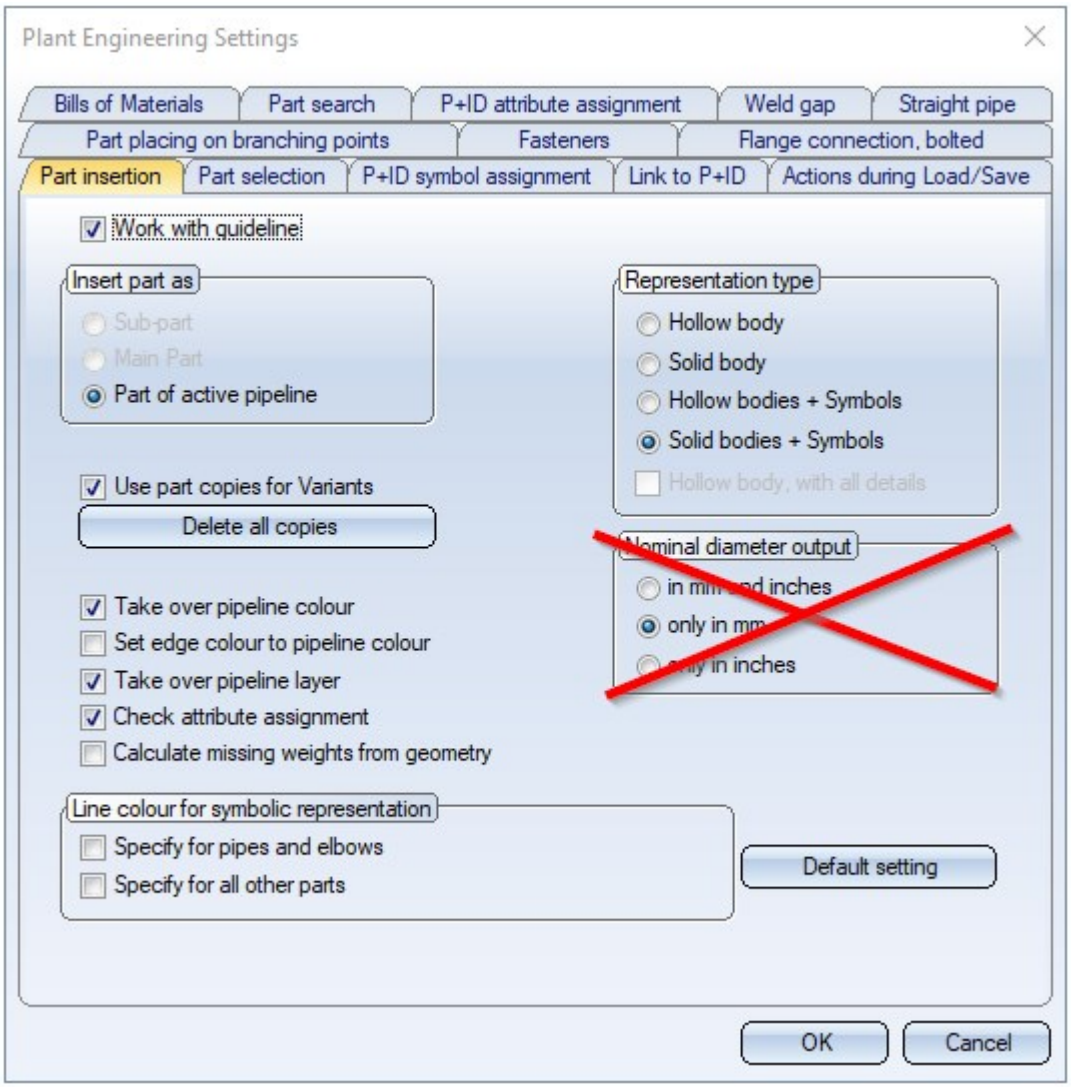
When creating a new pipeline, you must now choose between the nominal diameters **DN** (for nominal diameters that correspond approximately to mm) and **NPS** (for nominal diameters that are applied to inches). This also applies if the option **Assign nominal diameter** is not selected. This selection DN or NPS determines the nominal diameter that will later be included in the search for automatically found search conditions.



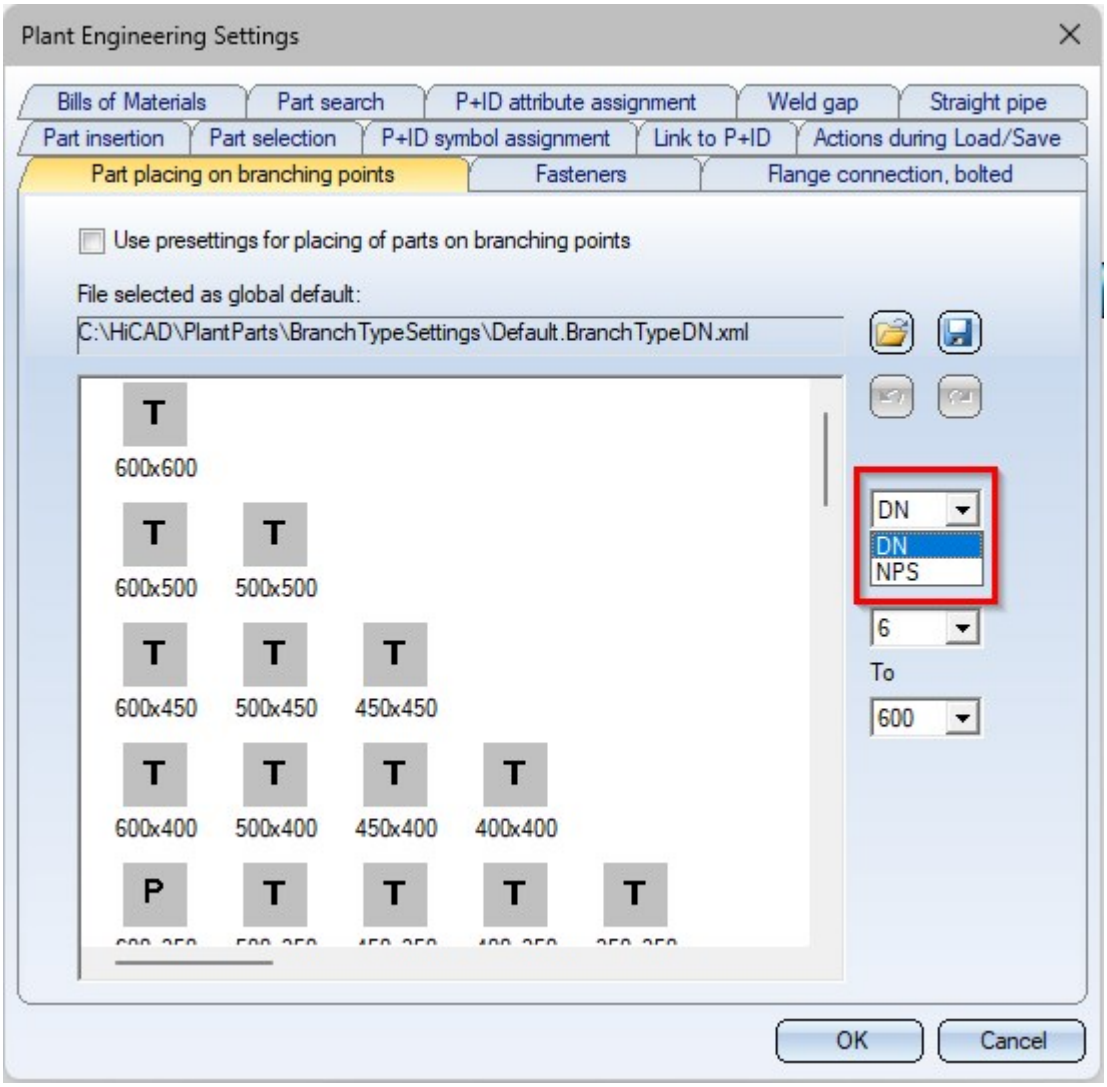
When using the new **Part insertion** function, the nominal diameters can be found in the search as shown:



Since the nominal diameter type must now be selected when creating a pipeline, the selection of the nominal diameter output is omitted on the **Part insertion** tab in the **Plant Engineering Settings** dialogue window.

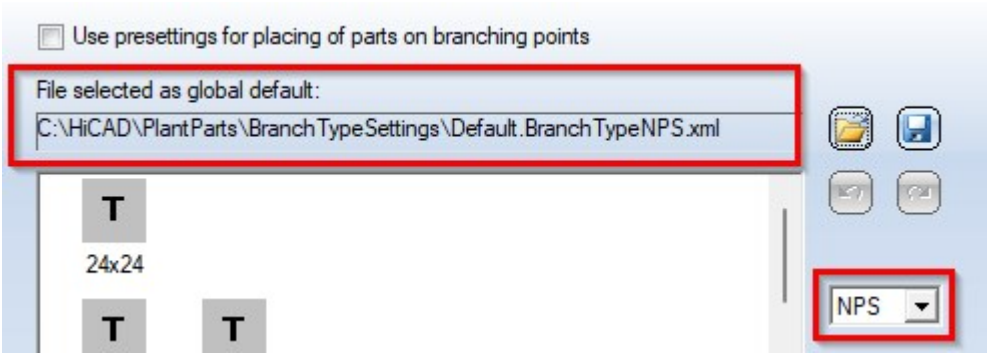


Instead, a selection box for the nominal diameter has been added to the tab **Part placing on branching points**.



Important:

If you select **NPS**, the file selected for global presetting must also be adapted to NPS, i.e. you must select a corresponding settings file. In the **PlantParts\BranchTypeSettings** directory of your HiCAD installation, for example, the **Default.BranchTypeNPS.xml** file is available for NPS nominal diameters.



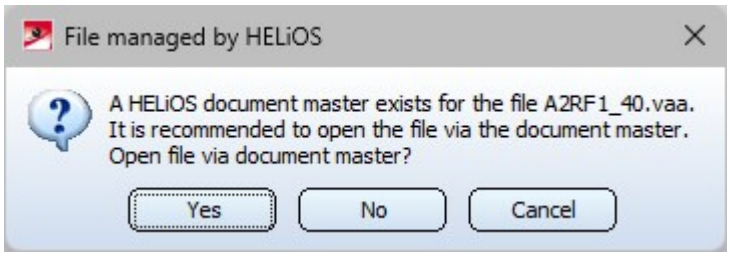
Files managed by HELiOS

HiCAD cannot determine where the data originates when accessing files for which a document master exists. This can lead to problems when dealing with variants.

For example, a variant could be loaded from the hard disk in the Variant Editor, modified and transferred to the corresponding items in HELiOS with the part data synchronization. In fact, this variant file can be managed by HELiOS e.g. in the Vault Server. So one has not changed the file to which the document master actually points.

This means that when the variant is installed, potentially a different geometry is calculated than expected, because the selected item no longer matches the expression in the VAA file to which the document master points.

Therefore, when a file is opened via the file system, it is checked whether this file is managed by HELiOS. If this is the case, the following message appears:



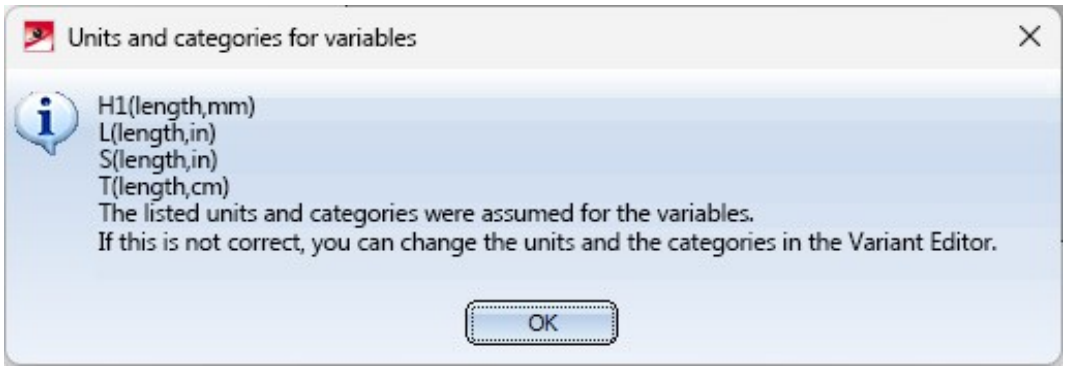
If you click on **Yes**, the file will be loaded via the document master. This ensures that the file matches the one in the document master.

This query also appears when opening takes place via the file system

- in the PAA Editor (AnPaaEdit.exe),
- during part data synchronization (PartDataAutoSync.exe), and
- during the configuration of the HELiOS database (DBPlantDataImport.exe).
- in the Variant Editor (Varienteneditor.exe)

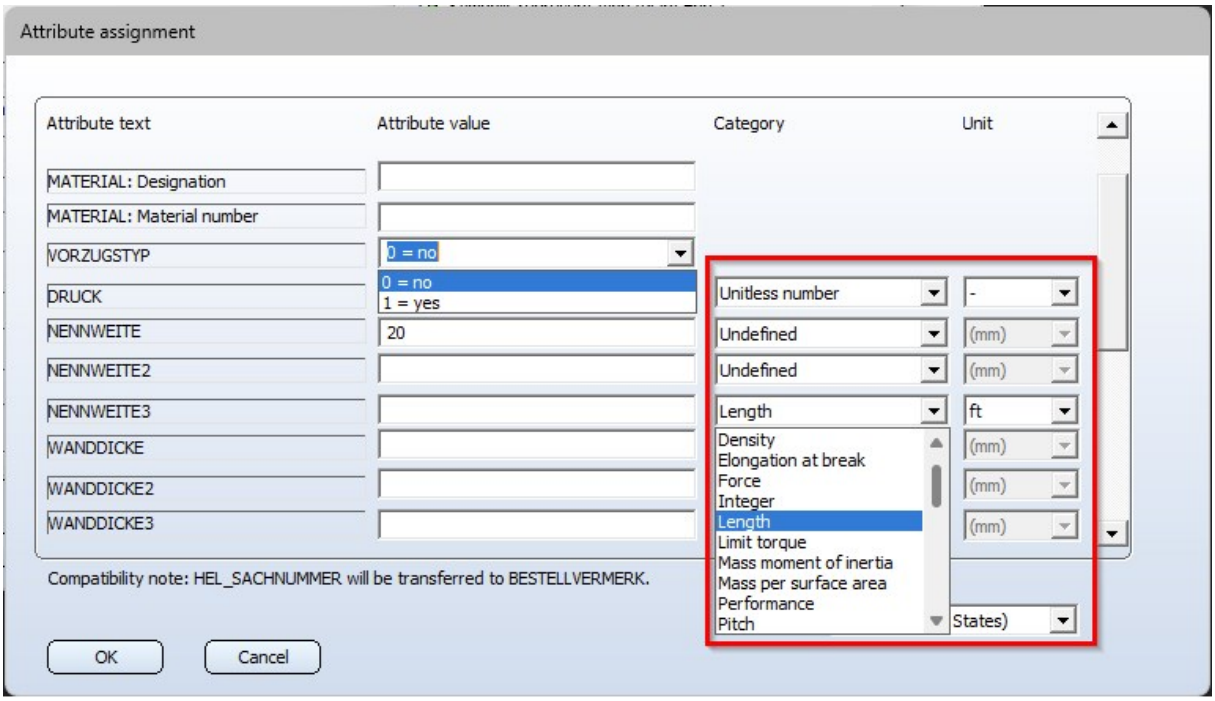
Create feature variant - Units

When creating a feature variant, the assigned units are now taken into account and you are informed about the accepted units.



PAA Editor - Units and Categories

When creating and editing PAA archives, from HiCAD 2024 units and categories can also be selected and transferred during attribute assignment, e.g.

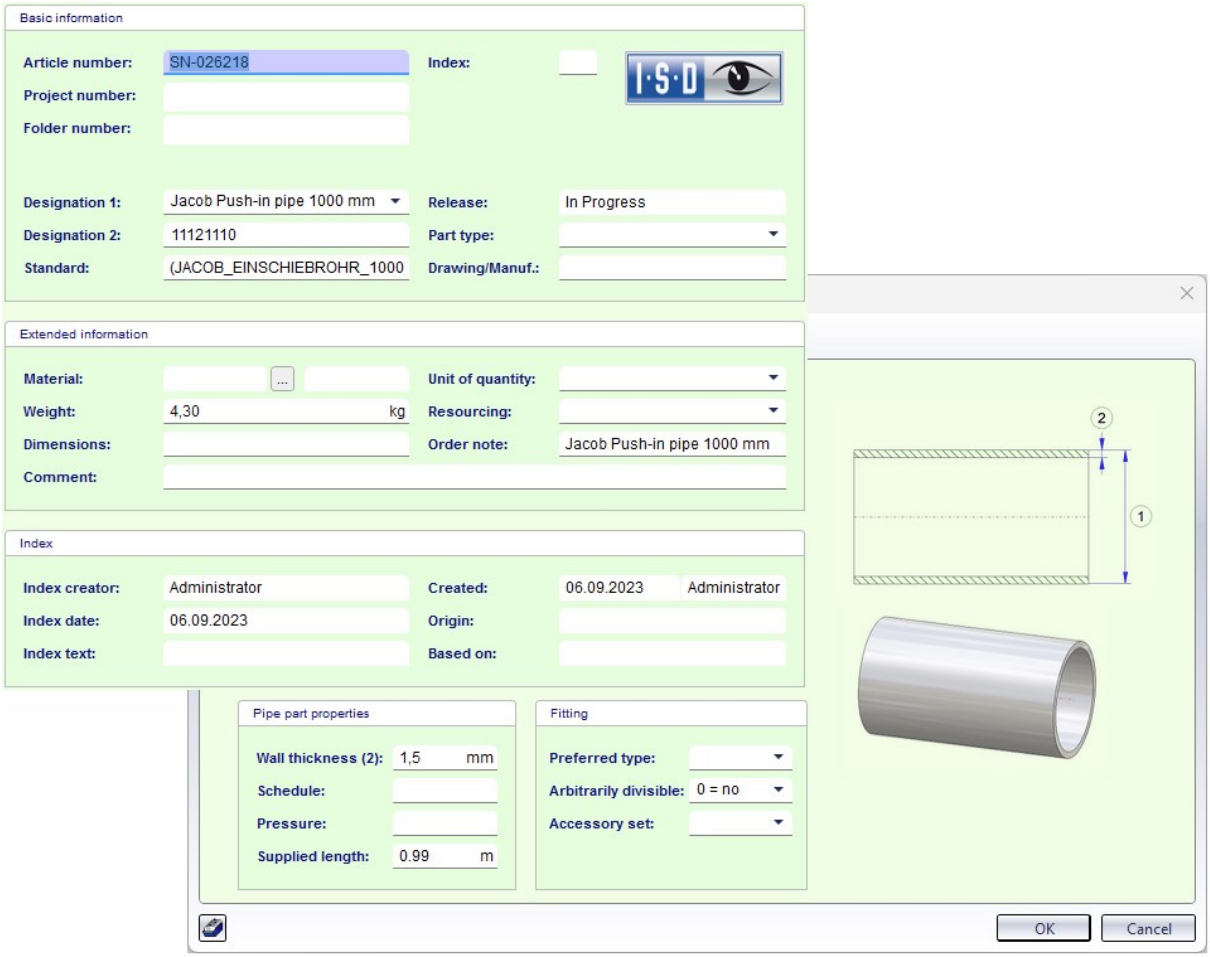


The attributes that expect a unitless number are all Nominal diameter attributes and the Pressure attribute.

Jacob Push-in pipes - Length and Weight

From HiCAD 2024 onwards, Jacob Push-in pipes receive their delivery length as a length attribute. The delivery length is now also no longer adjusted by the dynamic route change.

In the variant files of these push-in pipes, the attribute **Arbitrarily divisible** (BELIEBIG_TEILBAR) is set to **0=No**.



In addition, the attribute **Weight** for Jacob push-in pipes is also taken directly from the database and is no longer interpreted as weight per metre as with other straight pipes.

Check for invalid nominal diameter matings

While the check for nominal diameter matings is running, the message *Check nominal diameter mating (PE)* is now displayed in the HiCAD status bar.

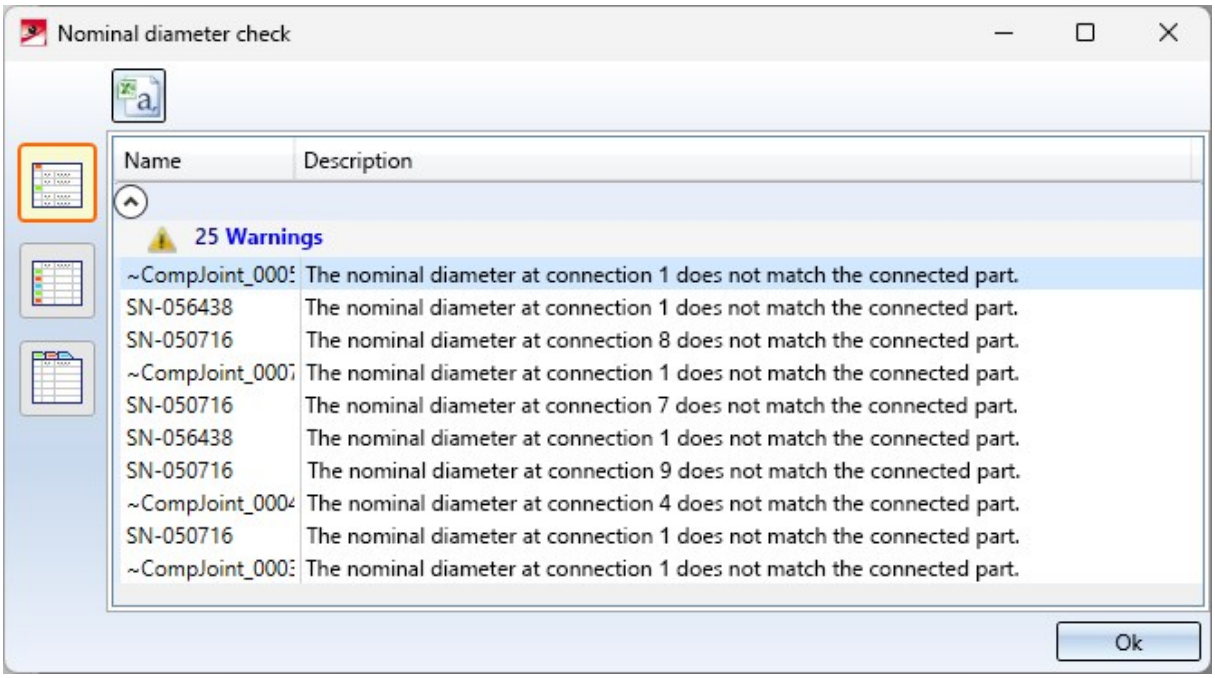
This applies to the functions:

 Check for invalid nominal diameter matings (active pipeline) and

 Check for invalid nominal diameter matings (entire drawing)

as well as the automatic check when opening and saving Plant Engineering drawings.

If errors are found, a list of parts whose connections have different nominal diameters is also displayed, e.g.:



New versions of the EN 1092-1 flanges

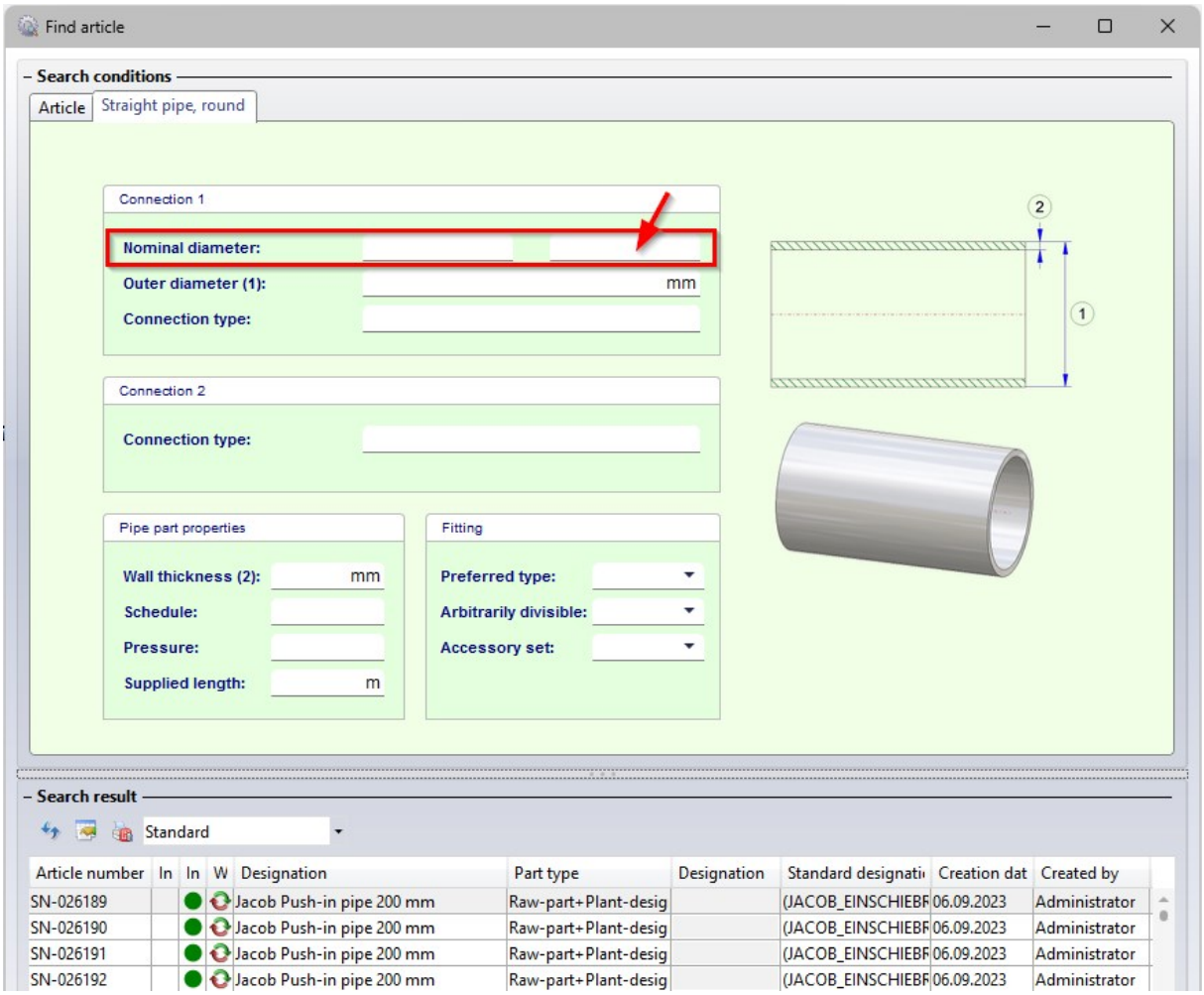
The loose flanges and plane flanges of the EN 1092-1 standard were not clearly recognisable as such until now. This has been improved in HiCAD 2024 by a revision and renaming.

Plane flanges	
previously	from HiCAD 2024
EN1092-1-01-PN10.vaa	EN1092-1-01-PN100_SF.vaa
EN1092-1-01-PN100.vaa	EN1092-1-01-PN10_SF.vaa
EN1092-1-01-PN16.vaa	EN1092-1-01-PN16_SF.vaa
EN1092-1-01-PN25.vaa	EN1092-1-01-PN25_SF.vaa
EN1092-1-01-PN2_5.vaa	EN1092-1-01-PN2_5_SF.vaa
EN1092-1-01-PN40.vaa	EN1092-1-01-PN40_SF.vaa
EN1092-1-01-PN6.vaa	EN1092-1-01-PN63_SF.vaa
EN1092-1-01-PN63.vaa	EN1092-1-01-PN6_SF.vaa

Looose flanges	
previously	from HiCAD 2024
EN1092-1-02-32-PN10.vaa	EN1092-1-02-32-PN10_LF.vaa
EN1092-1-02-32-PN16.vaa	EN1092-1-02-32-PN16_LF.vaa
EN1092-1-02-32-PN25.vaa	EN1092-1-02-32-PN25_LF.vaa
EN1092-1-02-32-PN2_5.vaa	EN1092-1-02-32-PN2_5_LF.vaa
EN1092-1-02-32-PN40.vaa	EN1092-1-02-32-PN40_LF.vaa
EN1092-1-02-32-PN6.vaa	EN1092-1-02-32-PN6_LF.vaa
EN1092-1-02-35-PN10.vaa	EN1092-1-02-35-PN10_LF.vaa
EN1092-1-02-35-PN16.vaa	EN1092-1-02-35-PN16_LF.vaa
EN1092-1-02-35-PN25.vaa	EN1092-1-02-35-PN25_LF.vaa
EN1092-1-02-35-PN2_5.vaa	EN1092-1-02-35-PN2_5_LF.vaa
EN1092-1-02-35-PN40.vaa	EN1092-1-02-35-PN40_LF.vaa
EN1092-1-02-35-PN6.vaa	EN1092-1-02-35-PN6_LF.vaa
EN1092-1-02-36-PN10.vaa	EN1092-1-02-36-PN10_LF.vaa
EN1092-1-02-36-PN16.vaa	EN1092-1-02-36-PN16_LF.vaa
EN1092-1-02-36-PN2_5.vaa	EN1092-1-02-36-PN2_5_LF.vaa
EN1092-1-02-36-PN6.vaa	EN1092-1-02-36-PN6_LF.vaa
EN1092-1-02-37-PN10.vaa	EN1092-1-02-37-PN10_LF.vaa
EN1092-1-02-37-PN16.vaa	EN1092-1-02-37-PN16_LF.vaa
EN1092-1-02-37-PN2_5.vaa	EN1092-1-02-37-PN2_5_LF.vaa
EN1092-1-02-37-PN6.vaa	EN1092-1-02-37-PN6_LF.vaa
EN1092-1-04-34-PN10.vaa	EN1092-1-04-34-PN10_LF.vaa
EN1092-1-04-34-PN16.vaa	EN1092-1-04-34-PN16_LF.vaa
EN1092-1-04-34-PN25.vaa	EN1092-1-04-34-PN25_LF.vaa
EN1092-1-04-34-PN40.vaa	EN1092-1-04-34-PN40_LF.vaa

New masks for part search in Plant Engineering

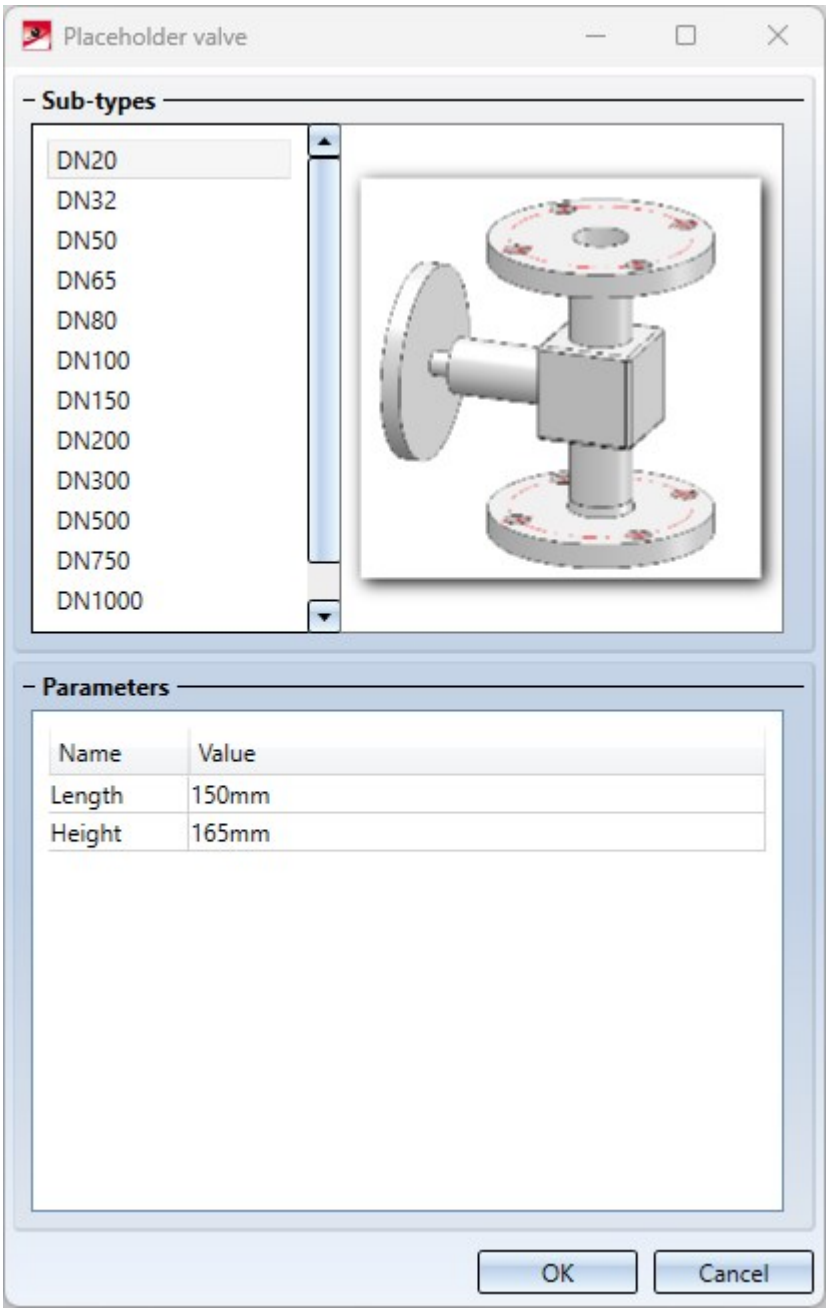
Due to the differentiation between the nominal diameters **DN** (for nominal diameters that correspond approximately to mm) and **NPS** (for nominal diameters that are applied to inches), the HELiOS search masks for the part search in Plant Engineering have also been adapted accordingly, e.g.



In the right field, which is used to search for the nominal diameter in inches (NPS), an entry in floating point numbers is now expected, e.g. "2 instead of 2". This affects all search masks for parts in which the attribute **Nominal diameter** is available in this form..

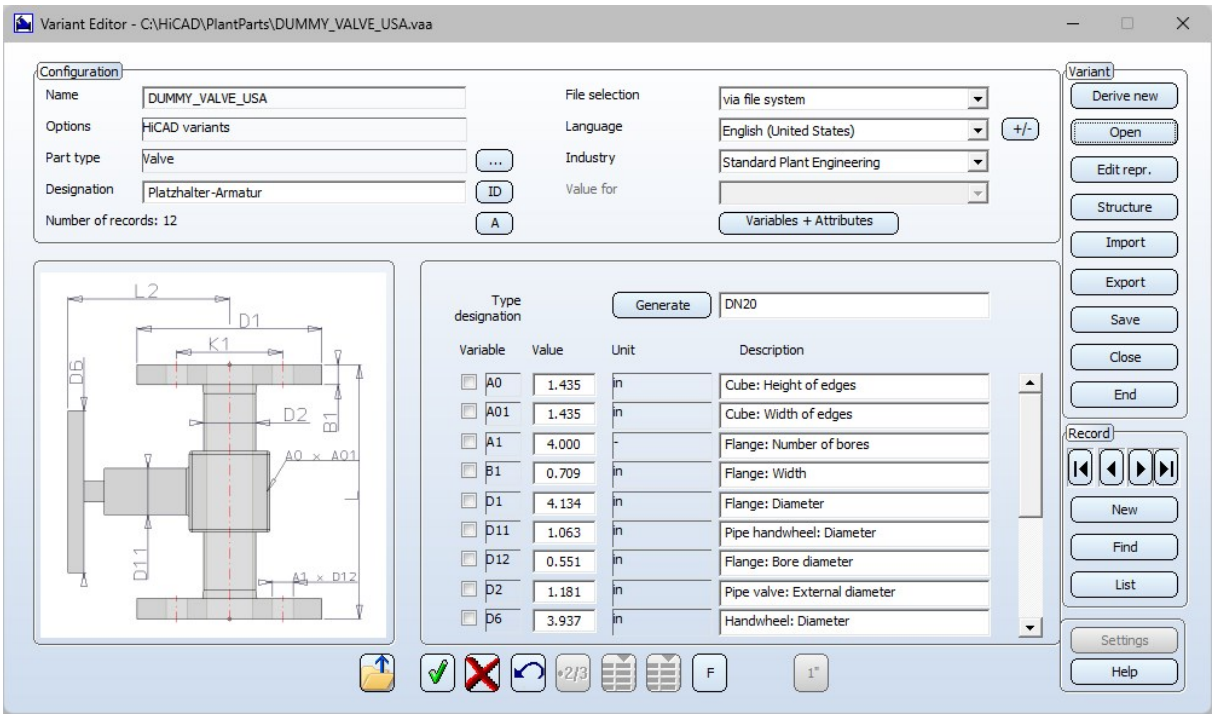
Placeholders in imperial units

When inserting placeholders, units can now be used.



The following variants are available for this purpose in the **PlantParts** directory of the HiCAD installation:

- DUMMY_VALVE.VAA (metric units, stored in mm) and
- DUMMY_VALVE_USA.VAA (imperial units, stored in inches).



Notes on HELiOS Updates

For an update to HELiOS 2021 (Version 2600) from a version older than 2500 a central update of the supplied HELiOS database is required.

Since conflict may occur during the update process in case of inconsistent data stocks, you should do the following:

- Data backup before updating

  Make sure that a data backup was made before carrying out the update of your HELiOS database.
  For the backup, either use the HELiOS Database Creator (further information can be found in the Installation Notes) or your SQL Server Application.
  In case of any questions, or if you need any help with regard to your customized system architecture , contact the ISD Hotline.

- Log file for update

  If any conflicts occur during the update, these will be recorded in the log file **HeliosDbUpdate.txt** (in the system path **%appdata%\ISD Software und Systeme\HeliosDbUpdate**).
  Have this file ready when contacting the ISD Hotline in case of an unsuccessful update, so that they can help you solve the problem and make a successful update.

- New mask format

  Please read all notes on the new mask format introduced with HELiOS 2020 !

- MultiCAD interfaces

  If you are working with a multi-CAD interface (e.g. the HELiOS-Inventor interface), please note that before installing an update of an older version to HELiOS 2020 (Version 2500) or higher, some adjustments may have to be made before the update.
  In this case, please contact the Consulting department of the ISD Group.

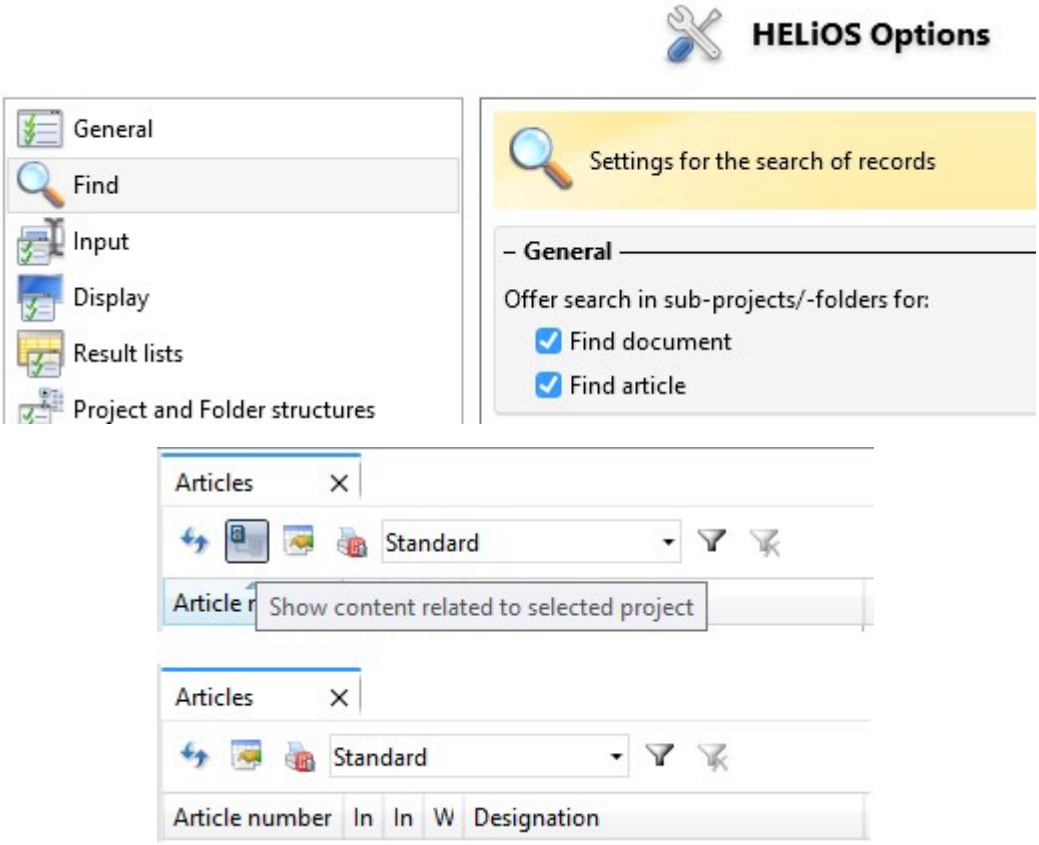
HELiOS Desktop

Service Pack 1 2024 (V 2901)

Search in sub-projects/-folders

The setting options introduced in HELiOS 2024 for the search behaviour of HELiOS data records have been extended with Service Pack 1 and have a further impact on search templates and result lists:

If the **Find article** and/or **Find document** checkbox has been deactivated in the **HELiOS Options** at **Find > General > Offer search for sub-projects/-folders for:...**, the corresponding buttons for searching in sub-projects/folders are removed from the header menus of result lists after a restart of the HELiOS Desktop.

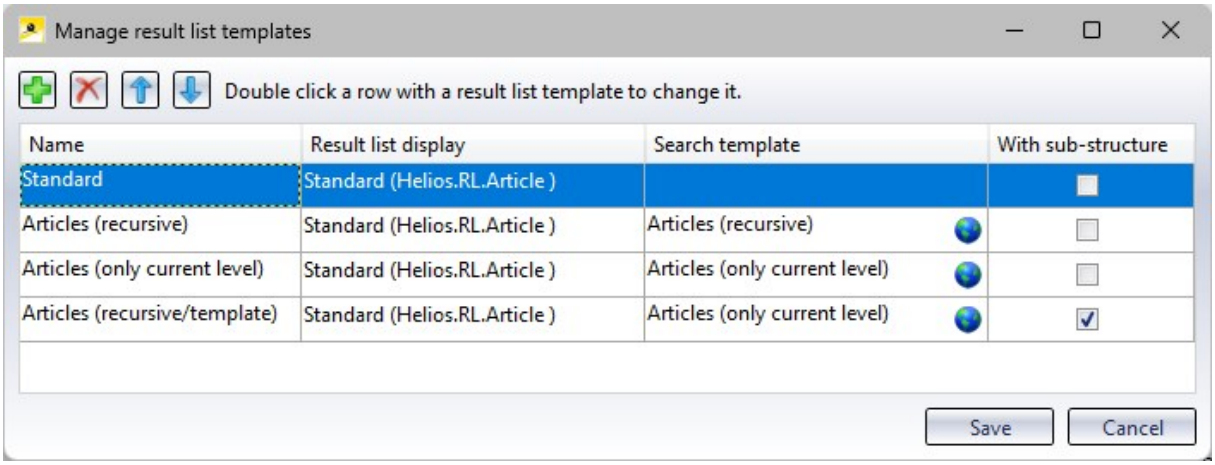


Please note:

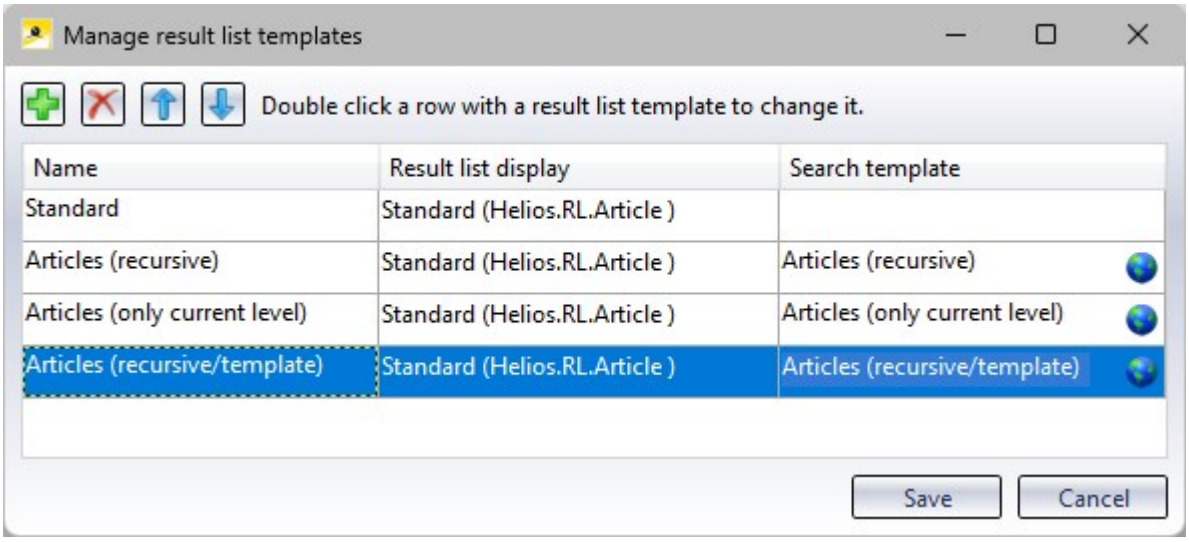
In "mixed" results lists (for articles and documents), the button is removed as soon as it has been deactivated for one of the two object types.

If the options for searching in sub-projects/folders have been deactivated, this also affects the search templates. If the user selects a search template in which the search was originally used in sub-projects / sub-folders, the search will no longer take place in the sub-projects / sub-folders despite the selection of the search template.

In the case of results list templates, the search in sub-projects / sub-folders is only controlled via the **With sub-structure** checkbox. The corresponding setting from the search template is irrelevant here.



If the HELiOS option **Offer search in sub-projects/-folders for:...** is disabled by deactivating the corresponding checkboxes, the **With sub-structure column** is removed from the **Manage result list templates** window.



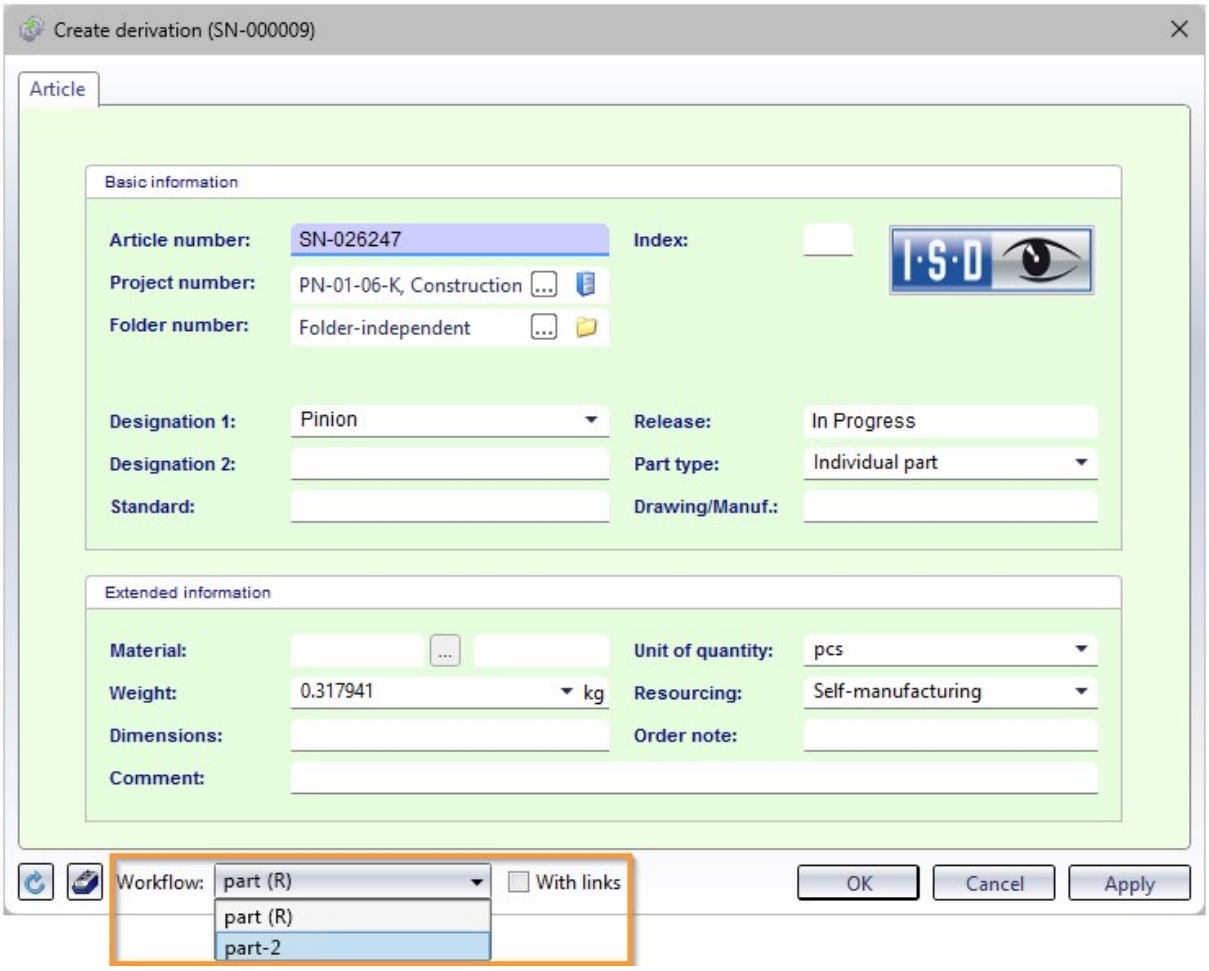
Deactivating the setting has the same effect on results list templates as it does on search templates. New result list templates are then saved without a sub-structure search. However, if an existing results list template (which was created with the sub-structure option active) is edited, the option to search **With sub-structure** saved in the background is retained.

Automatically apply workflow of the original object when creating index or derivation

When using the functions:

- **Create derivation** (for document master or article master)
- **Create derivation, with link** (for document master or article master)
- **Create index** (for document master or article master)
- **Create index, with link** (article master)
- **Create index (with file selection)** (document master)

HELiOS offers a workflow selection in the dialogues for creating the new object.

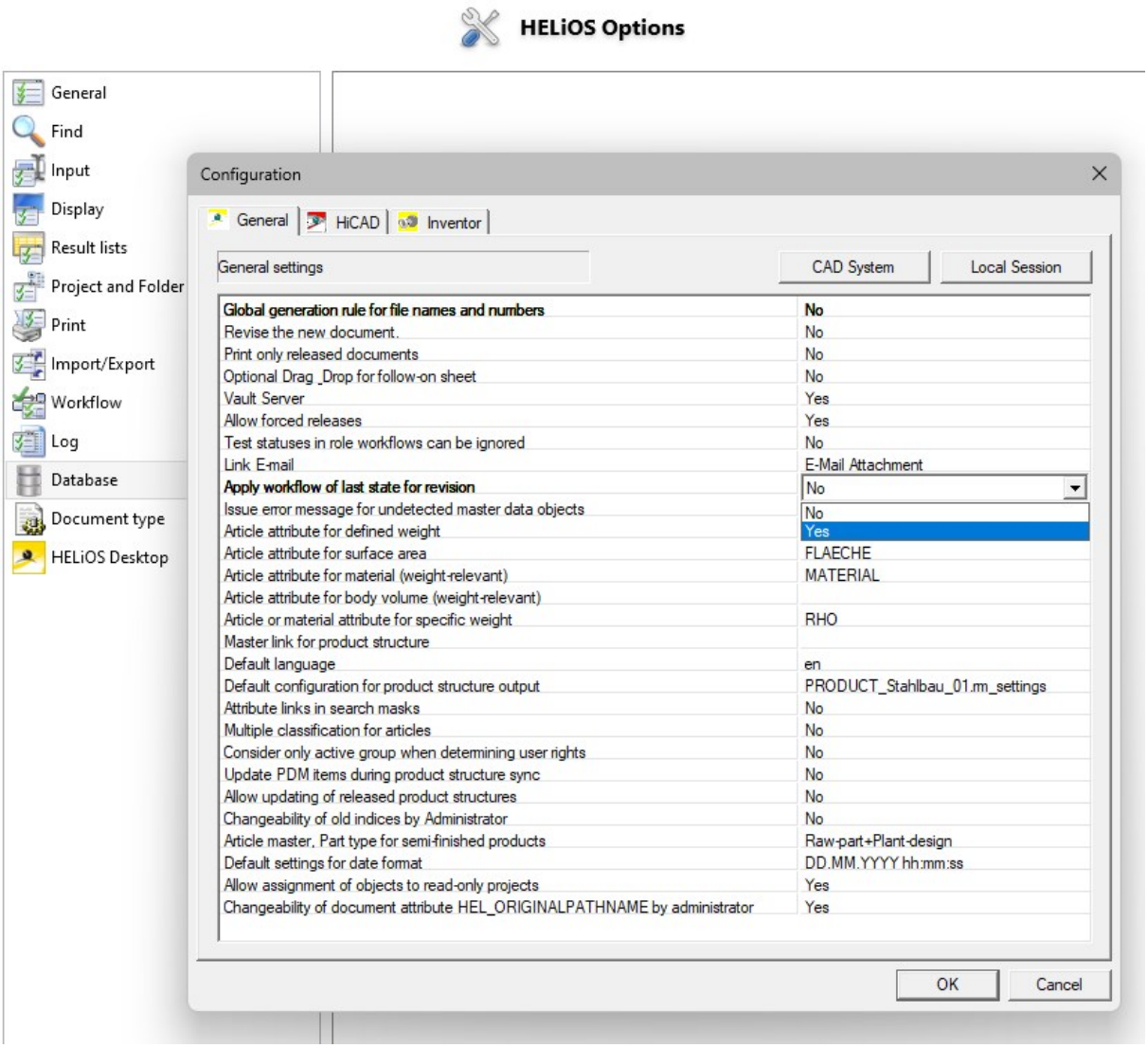


The workflow of the original object is preset.

If there are several workflows in the system for the HELiOS object type, you can also select another one from the pull-down menu.

If it is desired that the workflow of the original object is always adopted without giving the user a choice, administrators can set the Database option **Apply workflow of last state for revision** to **No**.

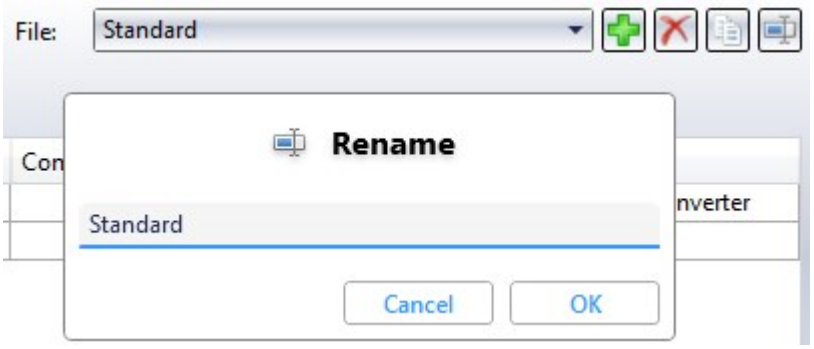
The workflow selection will then restricted for the above functions in such a way that only the workflow of the original object will be applied.



Deleting and renaming attribute assignments

Attribute assignments, such as those that can be set for the transfer of data to HELiOS or the export of files from HELiOS, can also be renamed from Service Pack 1 of HELiOS 2024.

To do this, click on the new **Rename configuration file** button in the overview window of the corresponding attribute assignments for the active **File**. A corresponding input dialogue for changing the name opens:



In previous versions, the preset attribute assignment under the name **Standard** could not be deleted. As of the current version, this can not only be renamed, but also deleted, provided that at least one other attribute assignment has been entered in the system. If there is only one attribute mapping in the system, this cannot be deleted.

Please note:

- The mapping of the Multi-CAD configuration file **Synchronization** may still not be deleted.
- In the event that users manually empty the list of attribute assignments (e.g. by editing the corresponding XML files), HELiOS automatically creates the default mapping from the installation state when it is started. This ensures that all dialogues work.

Different interface, attribute mapping and import/export configurations for different HELiOS users

To ensure that different HELiOS users can use different settings regardless of the Windows user logged in, the corresponding mechanism in HELiOS has been adapted.

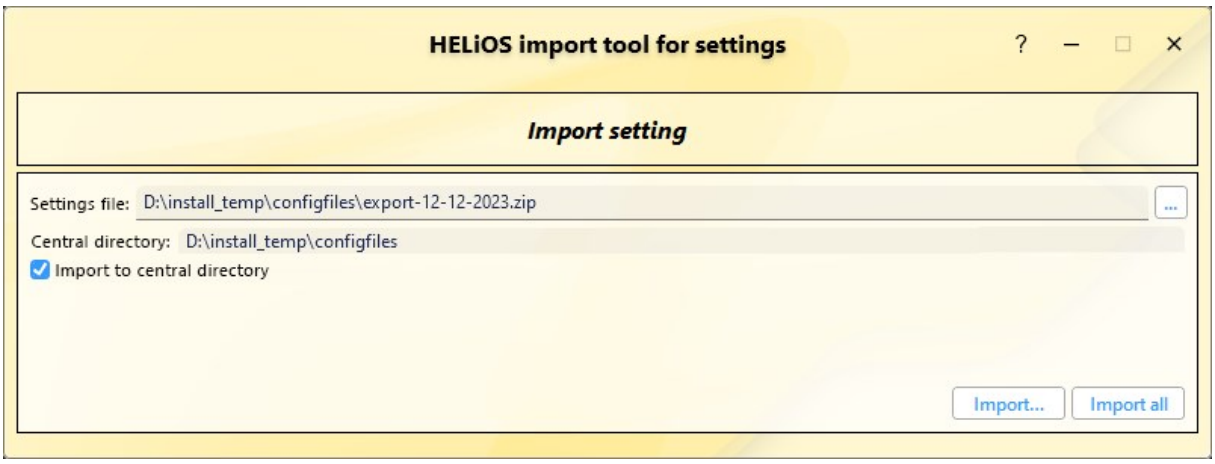
The changes affect the distribution options for specific customisations such as HELiOS masks, attribute mappings etc. to different workstations, as well as the export and import of HELiOS Options.

Please note that the corresponding directory structures or file storage locations may have changed as a result.

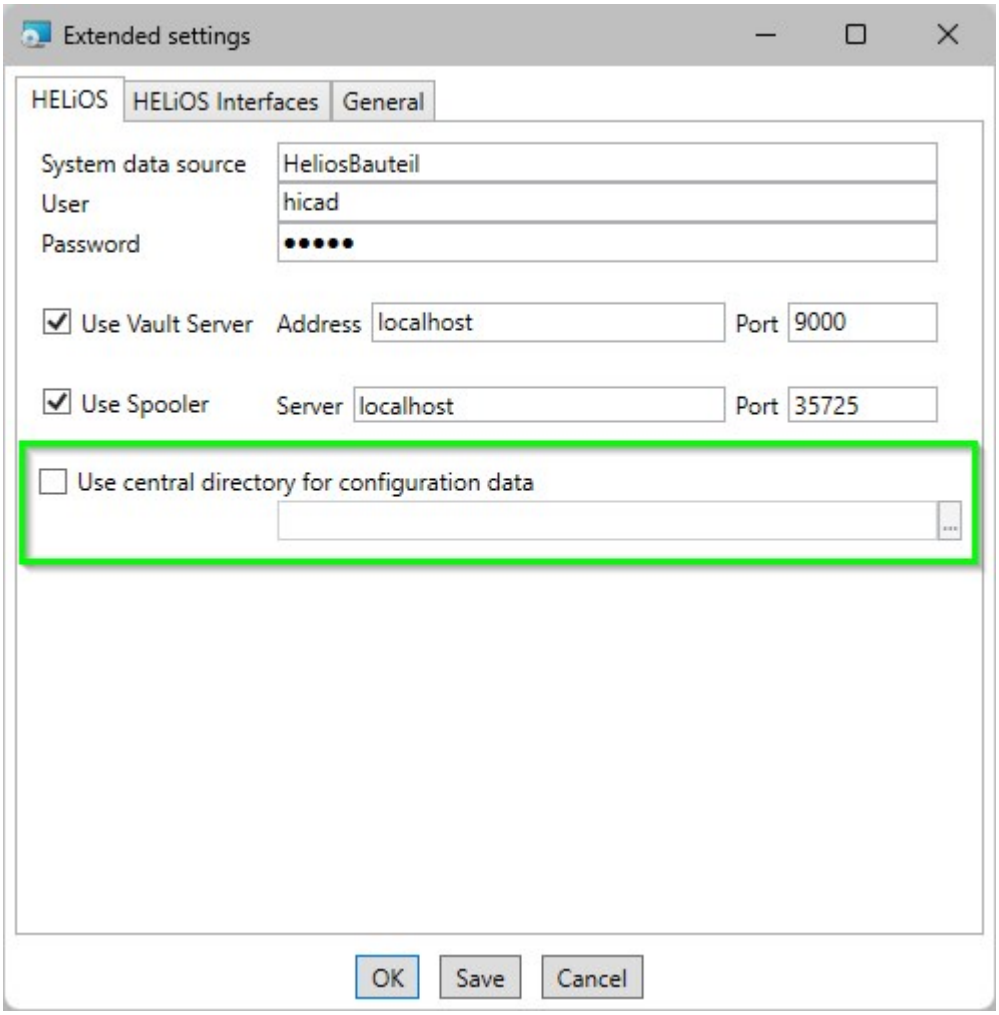
When updating to HELiOS 2024 SP1 (or higher) from an older version, existing settings files are backed up in a directory (%appdata%\Before2901Backup) and then migrated to the new structure.

In order for existing settings to be transferred, the workstations must be updated accordingly, or an export of the settings with HELiOS 2024 SP1 (or newer) must be re-imported.

HELiOS cross-user data can be stored centrally using the **HELiOS.ConfigImport.exe** tool. Manual creation of the directory structure or manual storage of the settings files is therefore not necessary.



Another new feature in this context is that you can set a central directory for the configuration data used across the board during the (update) installation of HELiOS.



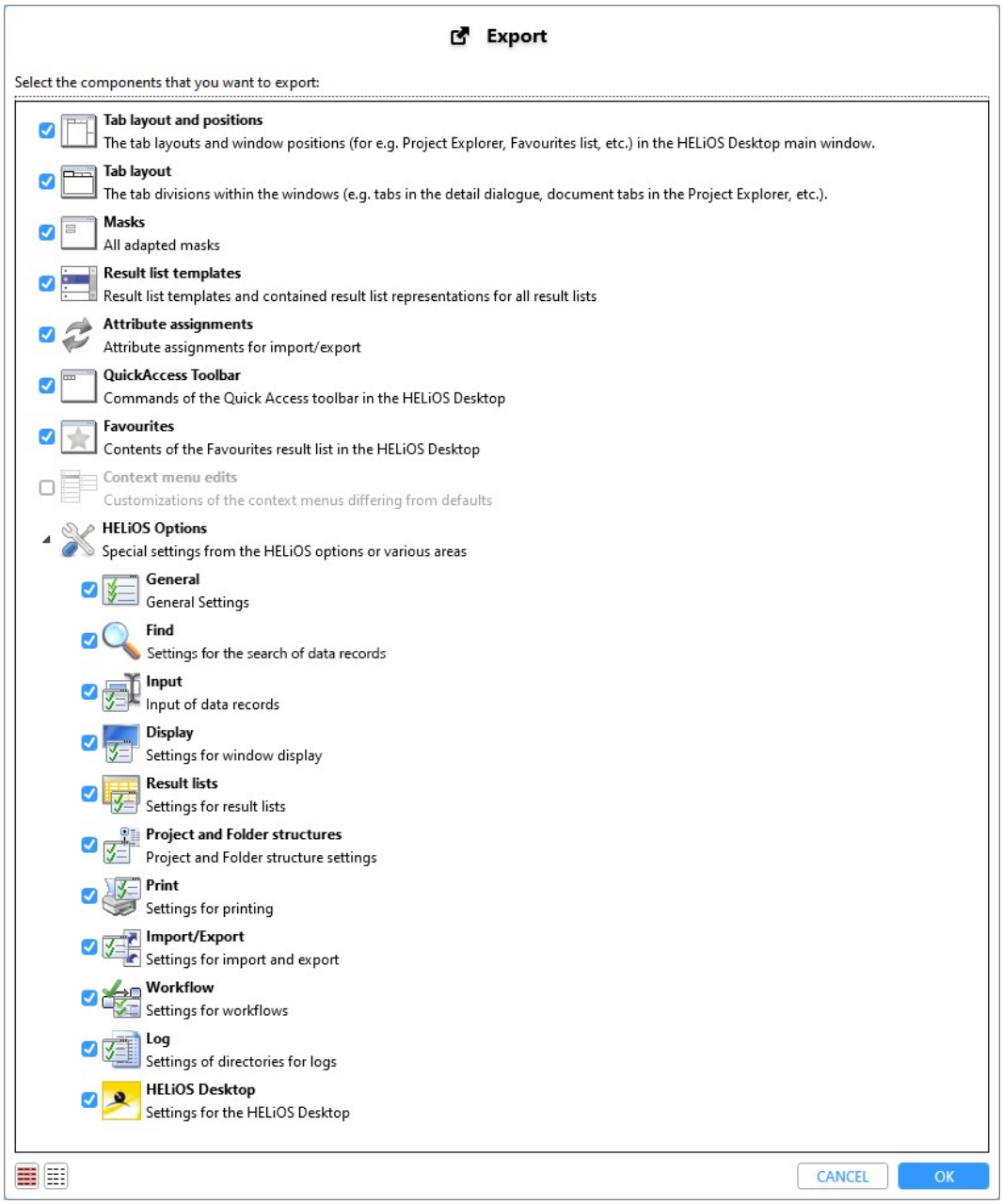
Major Release 2024 (V 2900)

Improved user interface

In HELiOS 2024, the user interface has been revised and improved in many areas.

This also applies to graphical control elements that correspond to the current state of the art, adjusted positions of menus or preselected buttons in masks.

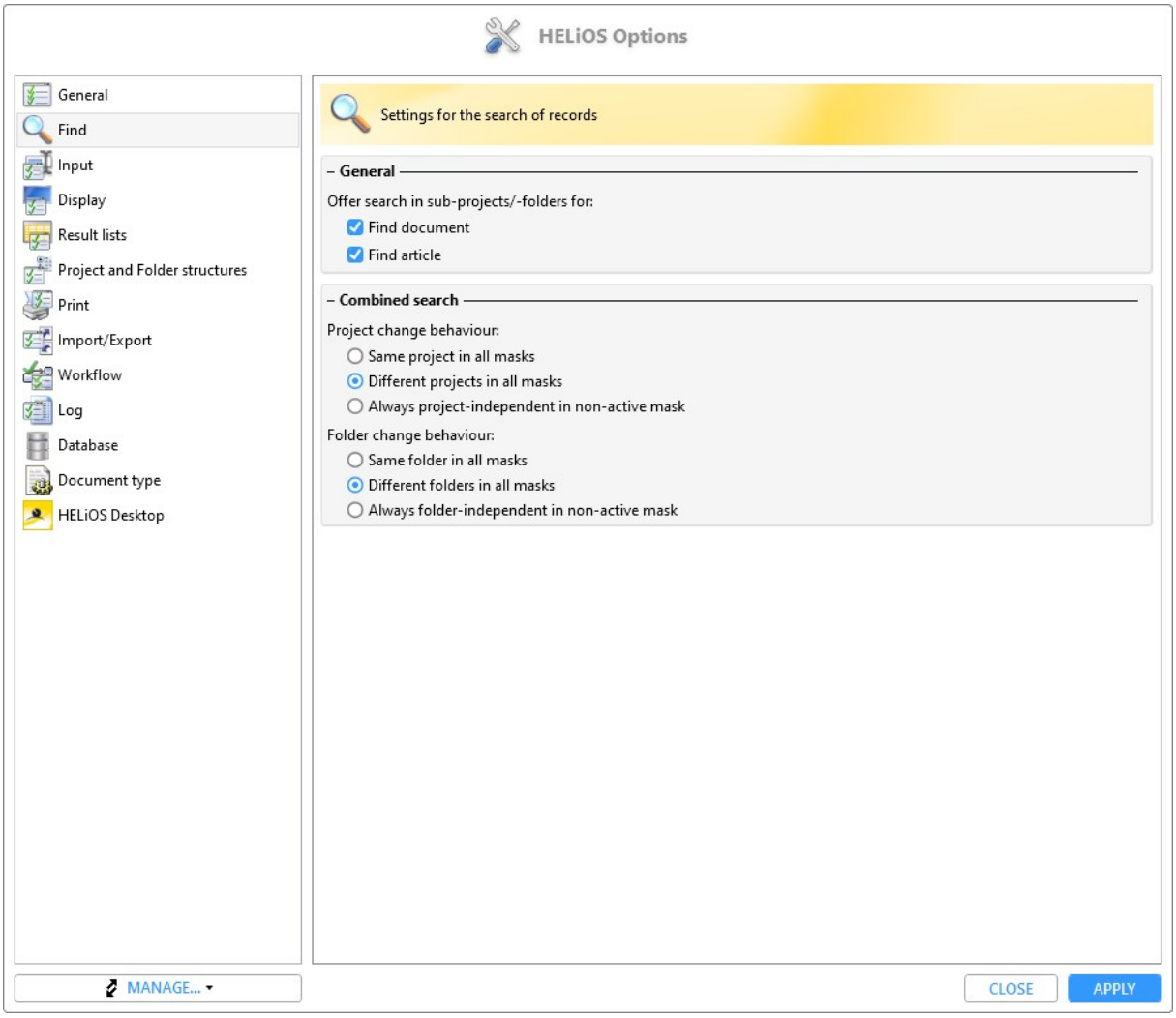
When exporting and importing interface settings, areas in which no changes have been made are greyed out and displayed with an inactive checkbox. In the example, this can be seen in the context menu edits:



HELiOS Options: Search

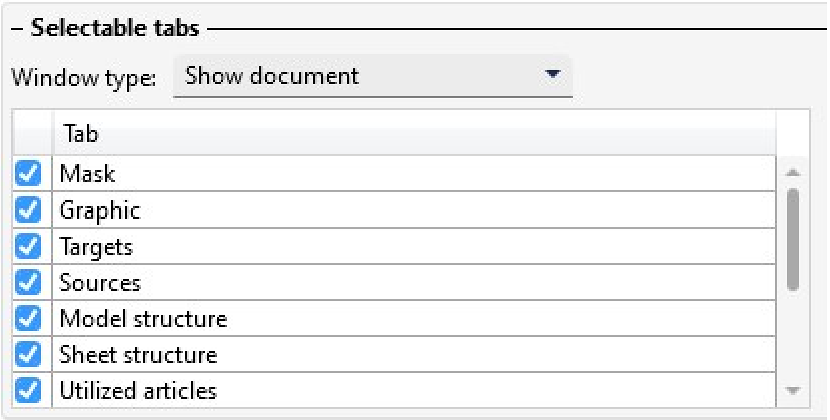
Starting with HELiOS 2024, you will find settings for the search of HELiOS data records in a separate sub-menu item.

New is the option to restrict the search in project and folder sub-structures for documents and articles.

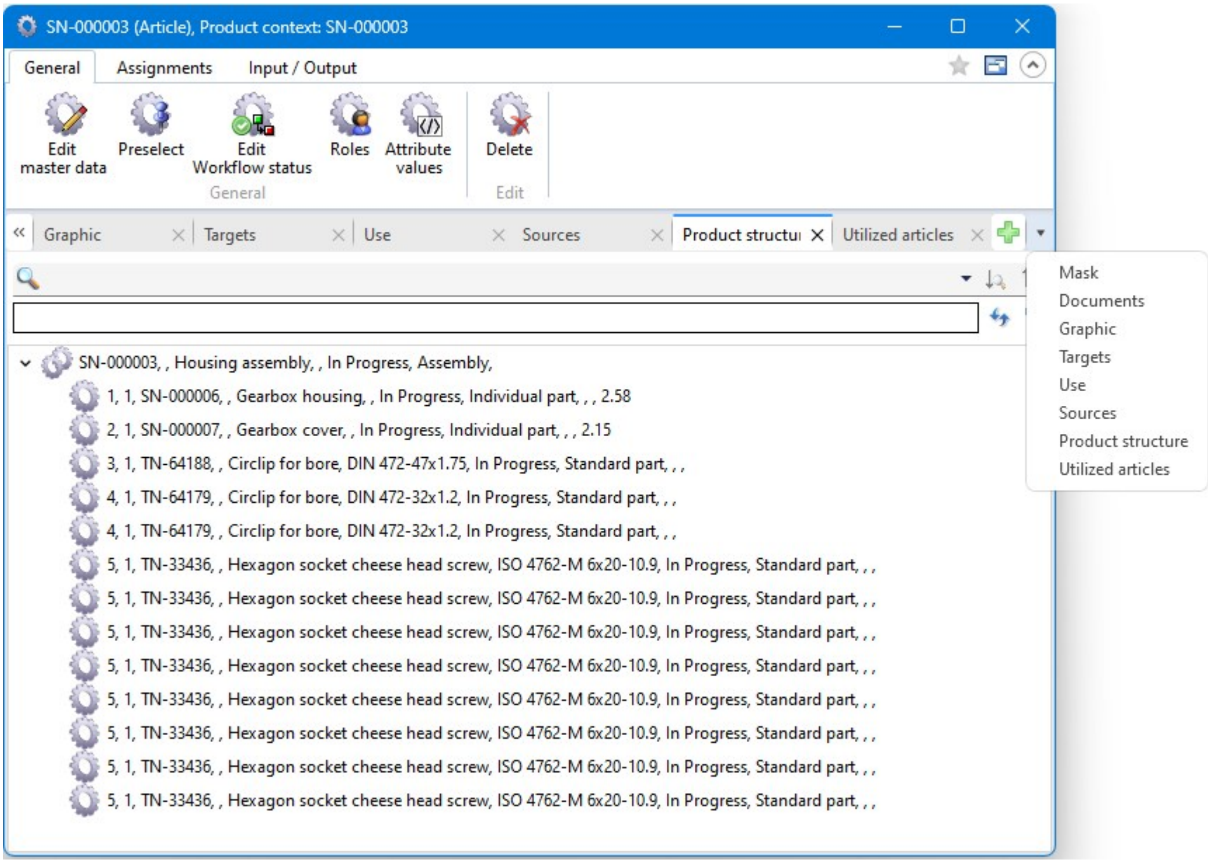


HELiOS Options: Selectable tabs

In the **Display** section of the **HELiOS Options** you will find the new Settings heading **Selectable tabs**.



Under this heading you can determinefor the five object detail window types **Show document**, **Show article**, **Show project**, **Show folder** and **Show pipe class**, as well as for the five result list contexts of the explorer window types **Project Explorer**, **Folder Explorer**, **Product Explorer**, **Article classification** and **Document classification**, whether they should be displayed in the HELiOS user interface or not.



The settings available here affect not only the HELiOS Desktop (and any linked applications), but also the user interface of the **HELiOS Internet Server**.

HELiOS Options: Classification

In the **HELiOS Options** you will find the new sub-menu **Classification** in the **General** section.

Here you can control for documents and articles whether the classification of the corresponding object type is available to the users. If it is deactivated, the call for the Class Explorer, the respective button for class selection when creating, editing or deriving a document or article, the possibility of class selection in search windows and the context menu function for classification in the HELiOS user interface are hidden for the respective object.

The former option **Show Classification tab in Find document/article dialogue** has been removed and replaced by object-specific sub-menu items.

Improvements and extension of the HELiOS URLs

The functionality of the **HELiOS URL**s has been revised and improved.

In the process, the procedure in HELiOS Desktop was aligned with that of the HELiOS Internet Server.

In the course of the improvements, the syntax of URLs specifying search criteria was extended:

For example, project and folder assignments can now also be specified. In the case of multiple assignments of an object, a desired unique context can be specified in a URL, e.g. also by specifying HELiOS attributes such as the name of an object.

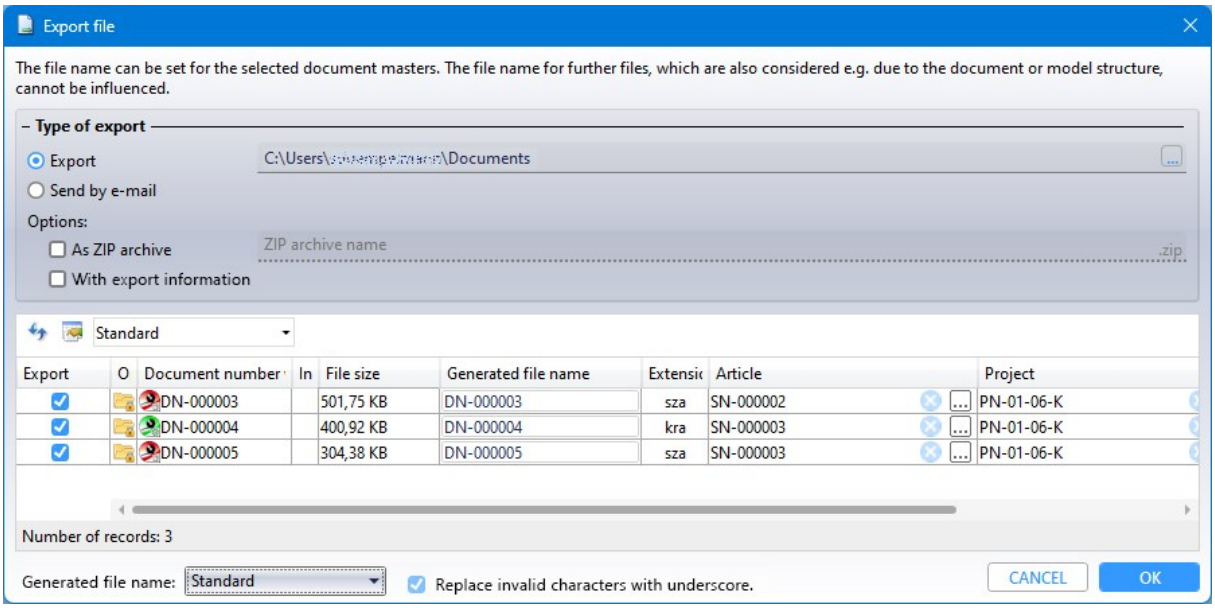
A "nested" specification of substructures can also be taken into account, if this is necessary for the unambiguous selection of a project or a folder.

Note that the URL syntax of earlier HELiOS versions is no longer supported by this revision. Any URLs saved from earlier versions (in documents, bookmarks, etc.) can therefore not be reused and must be manually replaced with URLs in the currently valid syntax in order for the call to work again.

Export file

When exporting files, in addition to saving them in an export directory, you have the option to send files directly from the HELiOS export dialogue by e-mail. If this option is selected, the locally configured e-mail client is started after the export and the files exported from HELiOS are automatically attached to a new e-mail.

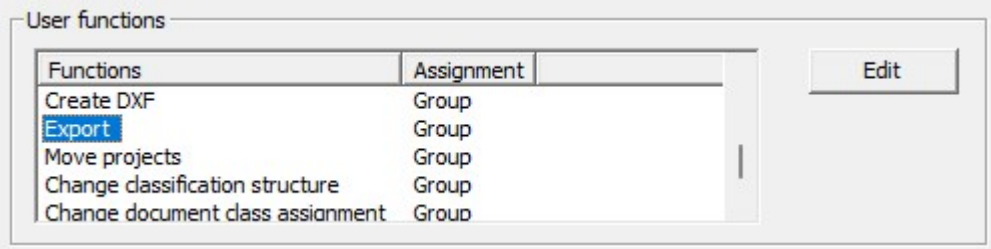
Another new checkbox in the export dialogue also makes it possible to save files directly as a zip archive, which can be very useful when exporting a large number of files at once.



Please note:

The improvement of the export functionalities is accompanied by an adjustment of the HELiOS user rights, which you can set in the user management of the **EDBSETUP**:

The **Export** function allows users to perform the actions **Export file** and **Send file by E-Mail**, as well as to export a document via drag & drop into the file system.



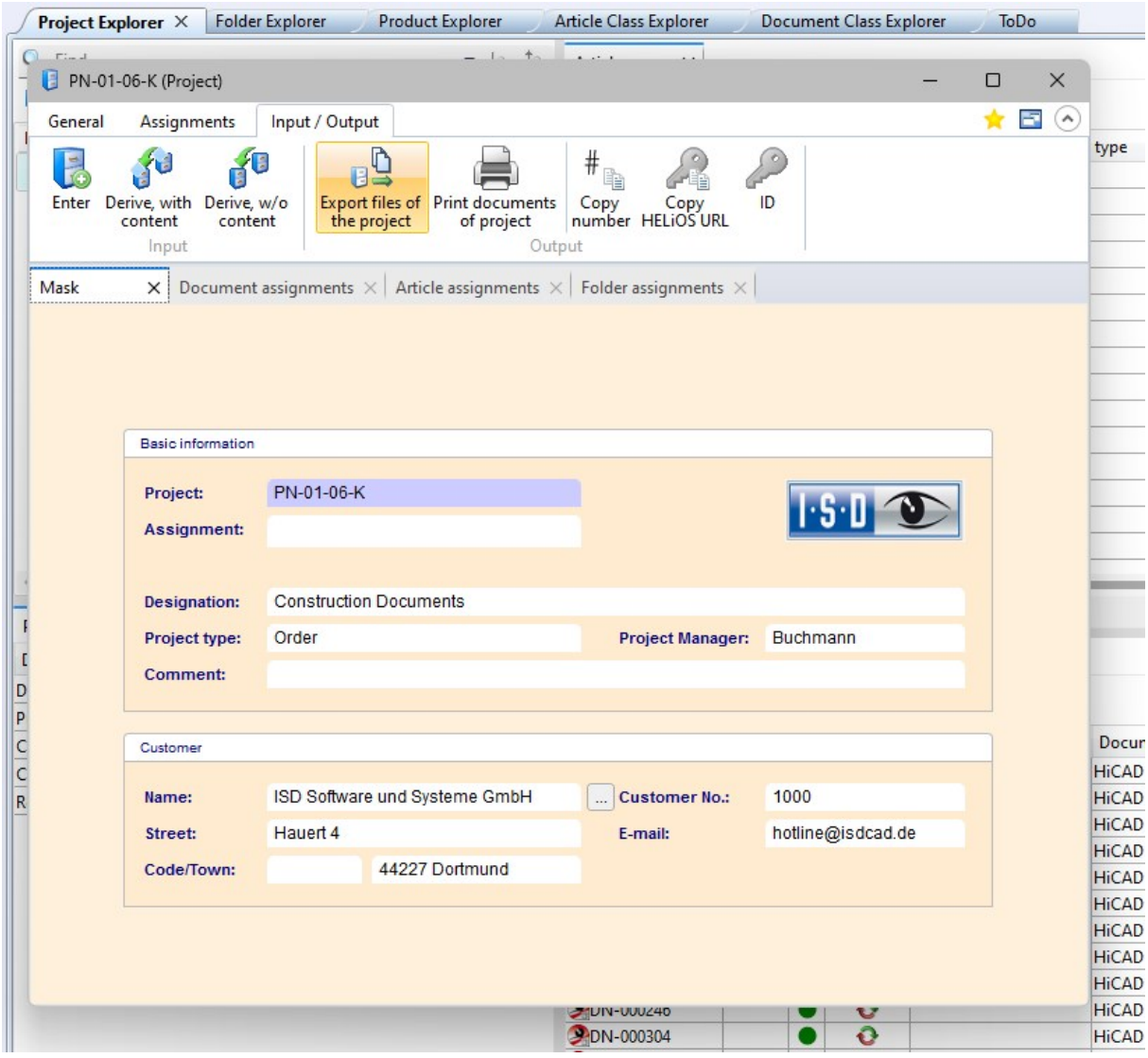
There is no separate user function for import.

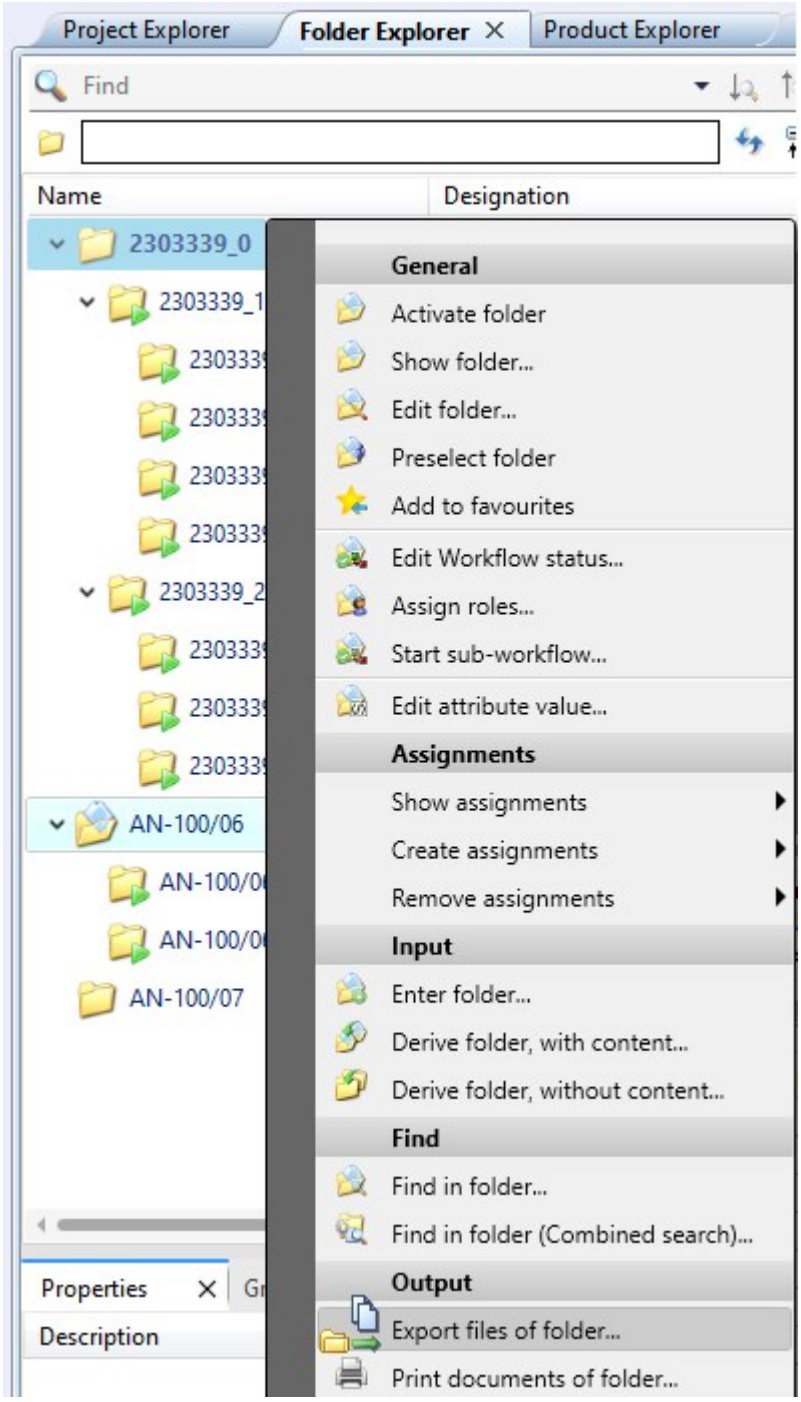
Export files of projects / Export files of folders

In the context menus of projects and folders you will find the new functions **Export files of project** or **Export files of folder** under **Output**.

As with the direct export of documents, this way you can export all files assigned to a certain project or folder from the HELiOS Desktop.

Additional options let you automatically create a parent folder or export the entire structure of a project or folder.

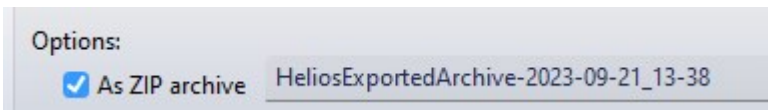




Send file by e-mail: As Zip archive

As with exporting files, you can also use the **Send file by e-mail**  function to pack the file(s) to be sent directly into a zip archive.

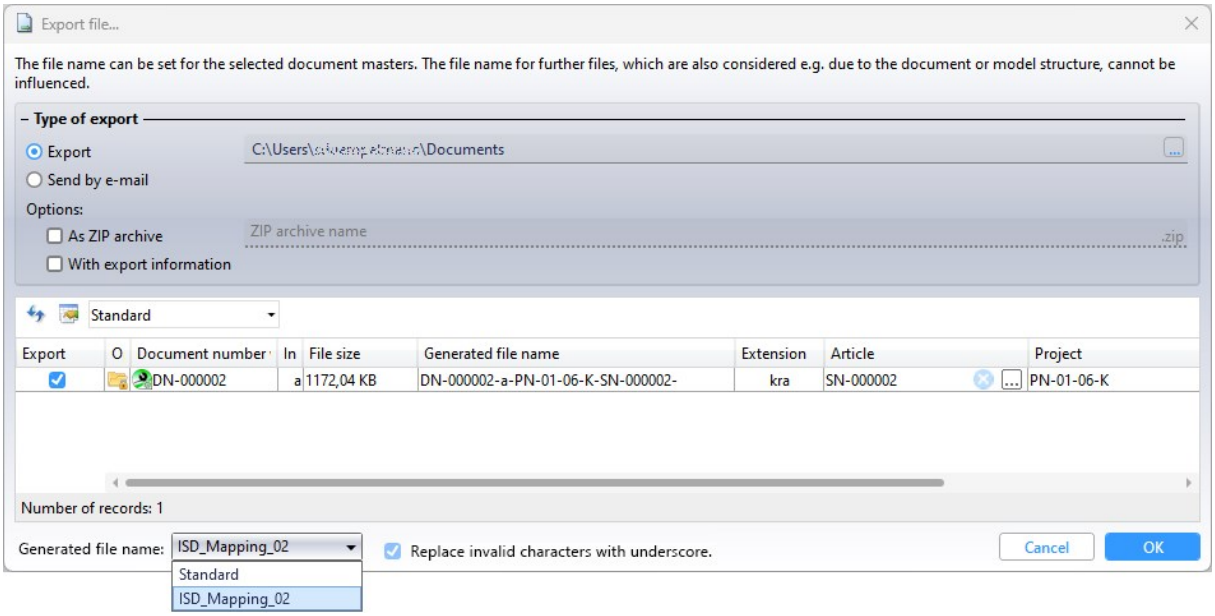
If necessary, you can change the automatically suggested file name by entering it manually.



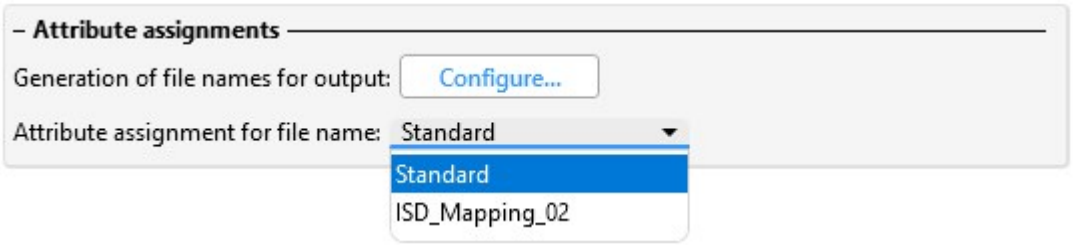
HELiOS Options: Attribute assignment and Export settings

In the **HELiOS Options** at **General > Attribute assignments** you have the possibility to set attribute assignments for the file name to be generated for document exports.

These can then be selected in the export dialogue.



Which attribute assignment is used by default for file name generation can be set under **Attribute assignment for file name**:



The assignment is then used when exporting document masters from HELiOS to the file explorer via drag & drop, when exporting files via the API and when downloading files via the HELiOS Internet Server.

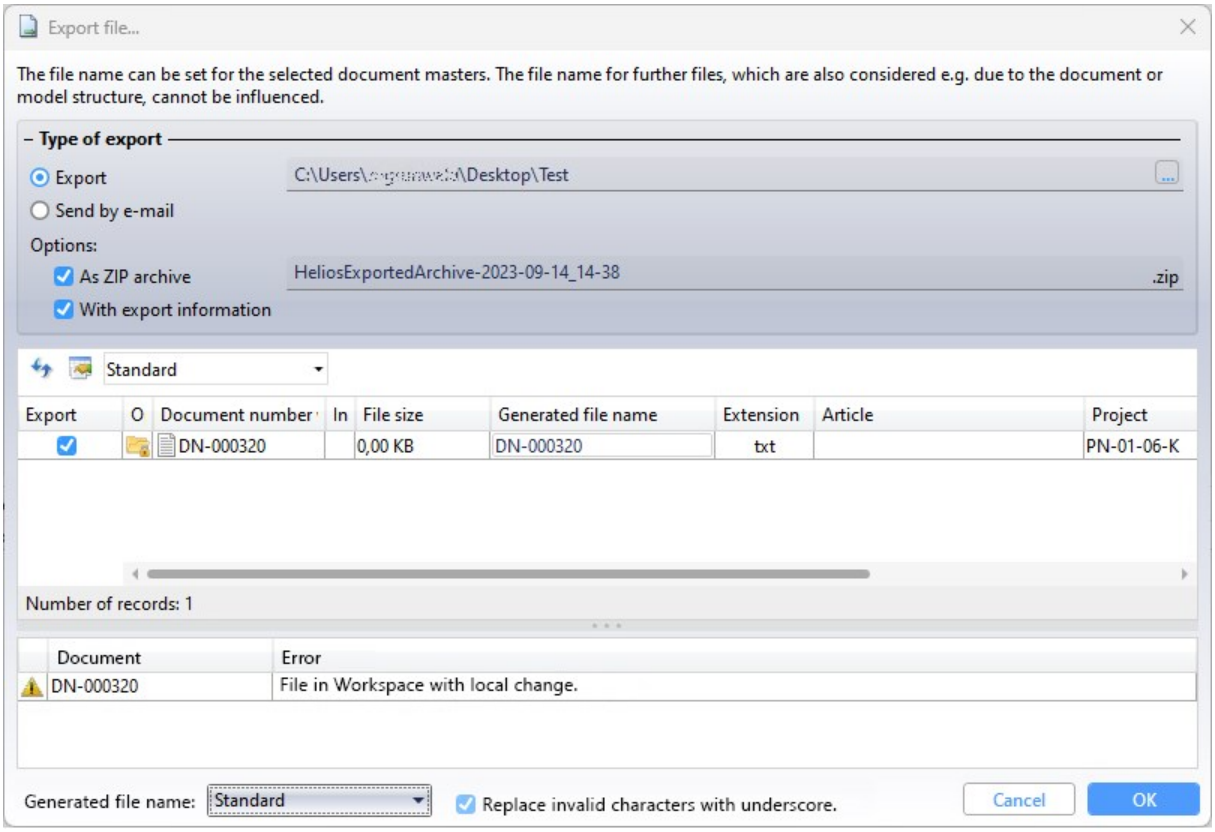
At **HELiOS Options > Import/Export** you can also set for **Export file**, as well as for the new functions **Export files of project** and **Export files of folder**. In each case, or the export settings dialogue should be executed or skipped for each export process.

Please note:

Due to the changeover, the previously existing concrete attribute assignments of the files Helios.ShortFileNameConfig.Export.xml and Helios.ShortFileNameConfig.Print.xml have been omitted and replaced by the new mechanism. For the new standard assignment, the settings have been taken from the mapping used for printing. Other attribute assignments may have to be reconfigured by the user after the update.

Local changes during export, printing and conversion

Before exporting a file, the system checks whether the documents to be exported have local changes. If this is the case, a warning is issued for the respective document, because in case of export not the locally changed status would be exported, but the status known to HELiOS.



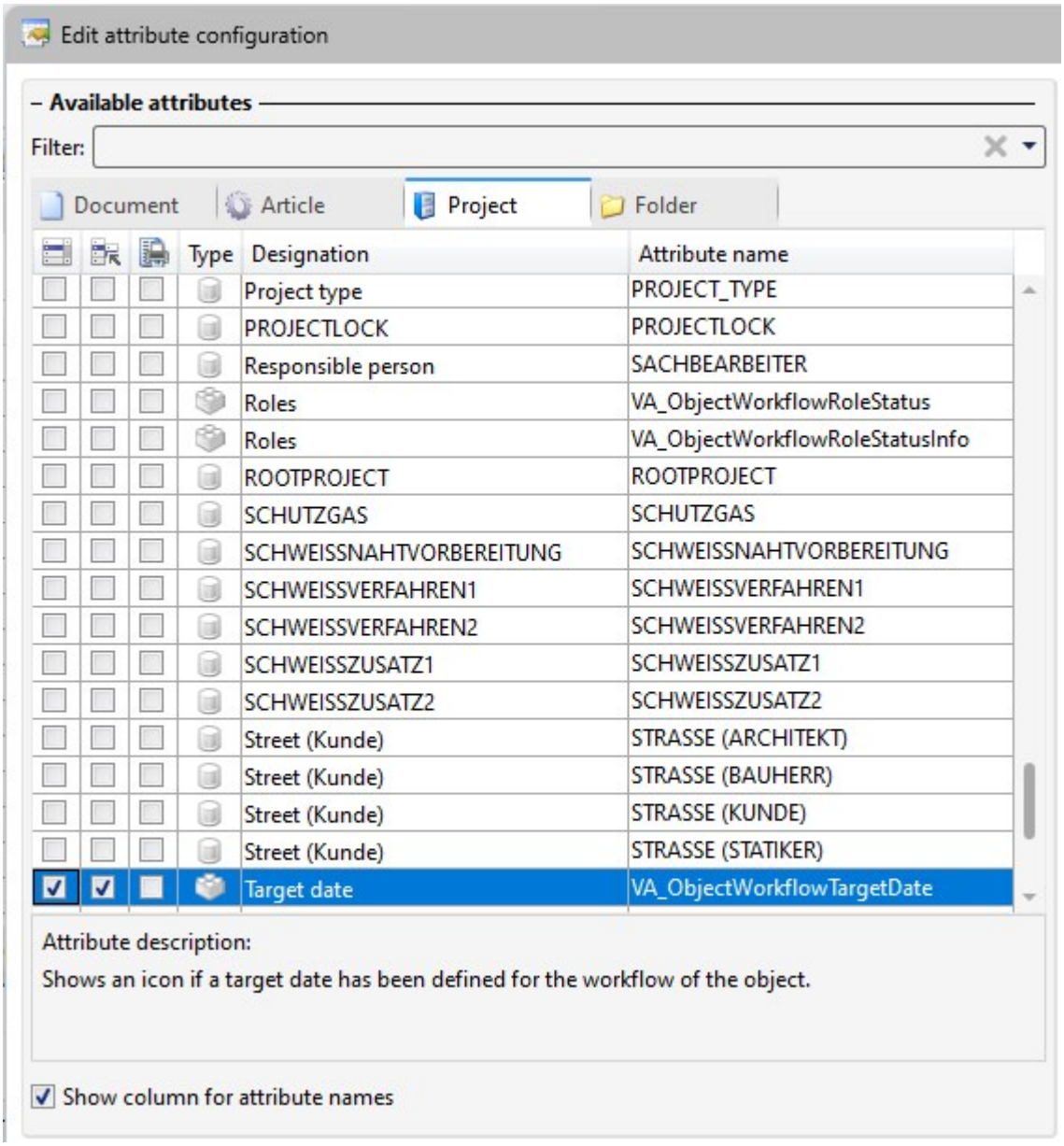
You can then decide whether to **Cancel** or to export the document with **OK** regardless of the local changes.

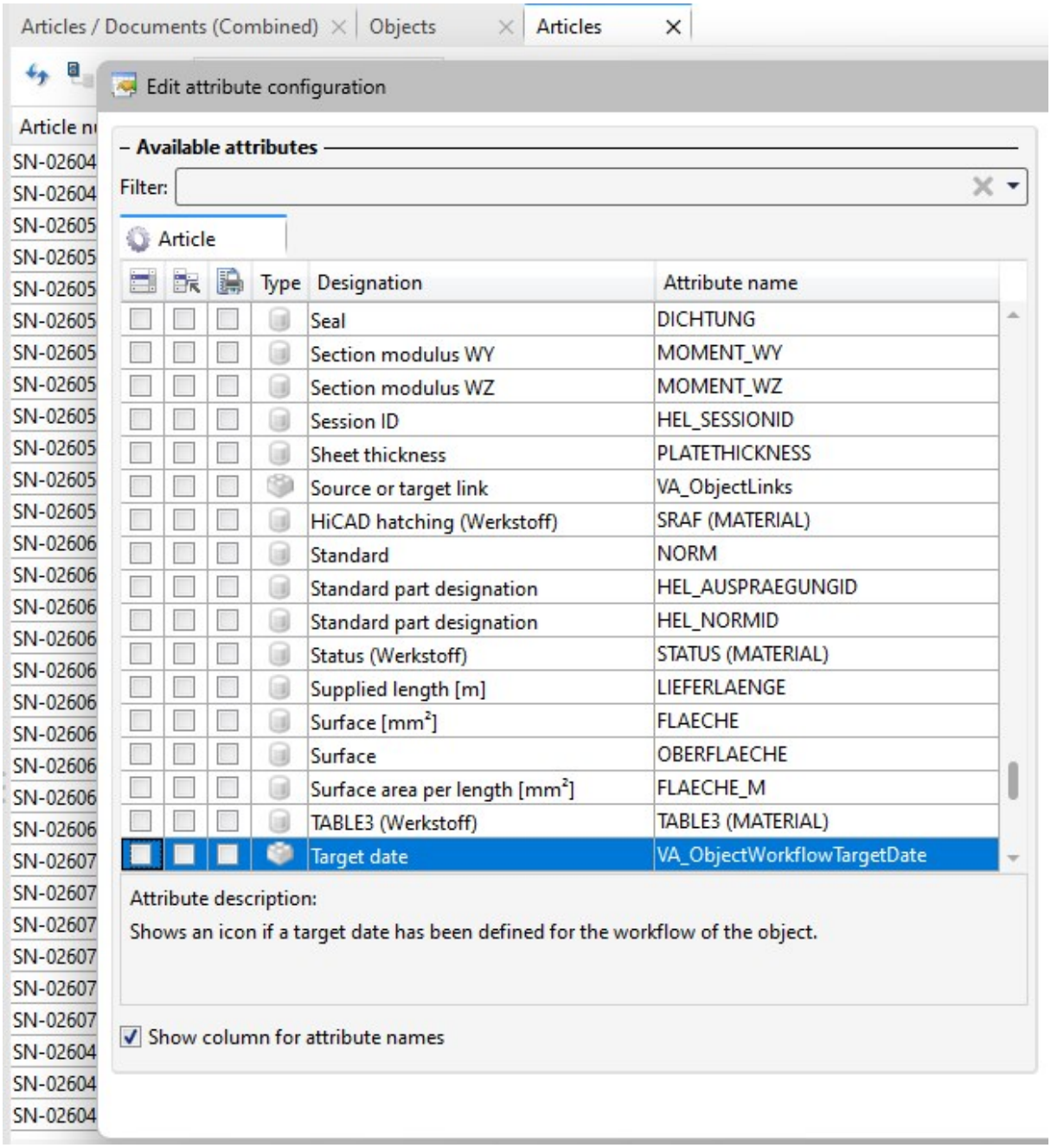
The same applies to **Print (Spooler)**, **Send file by e-mail** and **Convert**.

Result list - Target date display for projects and articles

Basically, target dates can be assigned for folders, projects, articles and documents.

The virtual attribute VA_ObjectWorkflowTargetDate, through which target dates of folders and documents were already displayed in result lists before, can now also be used for projects and articles.




Articles / Documents (Combined)
Objects
Articles
Edit attribute configuration
Article nu
SN-02604
SN-02604
SN-02605
SN-02605
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SN-02605
SN-02605
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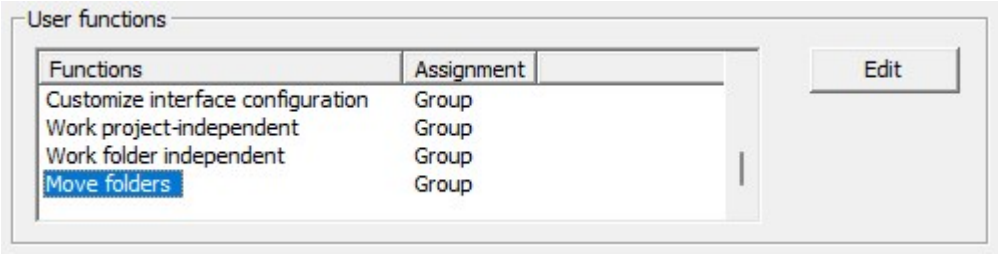
– Available attributes
Filter:
Article
Type
Designation
Attribute name
Seal
DICHTUNG
Section modulus WY
MOMENT_WY
Section modulus WZ
MOMENT_WZ
Session ID
HEL_SESSIONID
Sheet thickness
PLATETHICKNESS
Source or target link
VA_ObjectLinks
HiCAD hatching (Werkstoff)
SRAF (MATERIAL)
Standard
NORM
Standard part designation
HEL_AUSPRAEGUNGID
Standard part designation
HEL_NORMID
Status (Werkstoff)
STATUS (MATERIAL)
Supplied length [m]
LIEFERLAENGE
Surface [mm²]
FLAECHE
Surface
OBERFLAECHE
Surface area per length [mm²]
FLAECHE_M
TABLE3 (Werkstoff)
TABLE3 (MATERIAL)
Target date
VA_ObjectWorkflowTargetDate
Attribute description:
Shows an icon if a target date has been defined for the workflow of the object.
Show column for attribute names


Articles and Documents
Objects
Articles
Standard
Article number
Index
Index up-to-date
Workflow status
Part type
Target date

User rights: Move folders

As with the possibility of allowing or preventing the moving of projects for certain users or groups via the user management of the EDBSETUP, a user right for moving folders can now also be assigned or removed from the authorisation functions.



Existing HELiOS users automatically receive this right through the database version update. So nothing changes at this point, except the possibility that this right can be withdrawn from users.

User-defined types: Change of read permission and reference attributes

The read permission for HELiOS objects of user-defined types has been changed to allow all users to read such objects.

Modification, deletion and creation of these objects is still regulated by the (classical) workflow.

In addition, HEliOS user-defined object types also support reference attributes. For example, you can use reference attributes (such as organisational affiliations of customers in document master data) to narrow down the search for customer data.

Initialised attributes

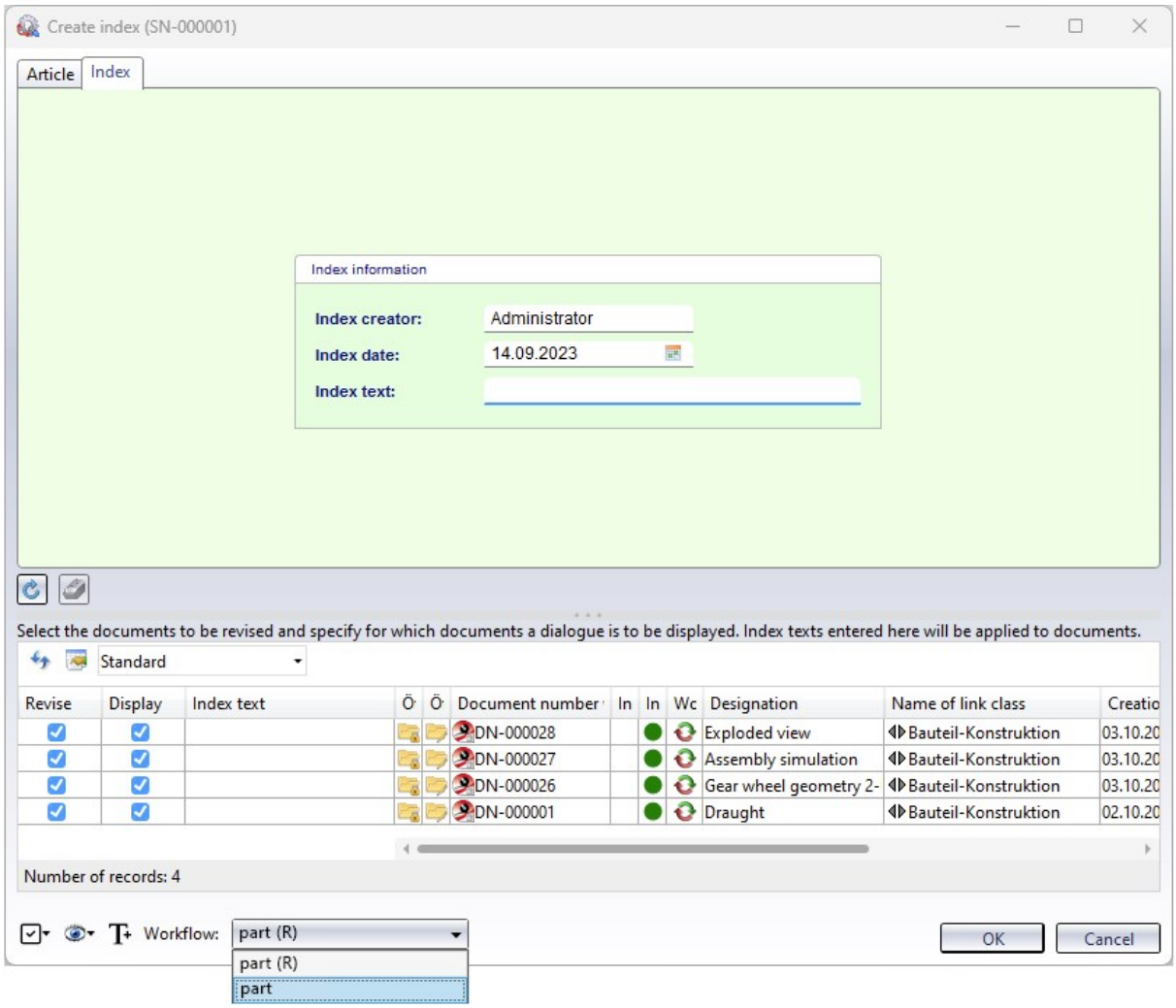
Note the instructions for initialising attributes:

Improving adjustments have been made in the hierarchical order in which initialised attributes are overwritten by further steps if necessary.

Initialisations from attribute assignments or workflow steps can be overwritten by subsequent steps. Only the attributes that are automatically changed by attribute assignments or the workflow will be updated. Further initialisations made by by the user remain unchanged.

Workflow selection for article index

As with indexing of individual articles or documents, it is also possible to make a workflow selection for articles when choosing **Create index, with link**  .

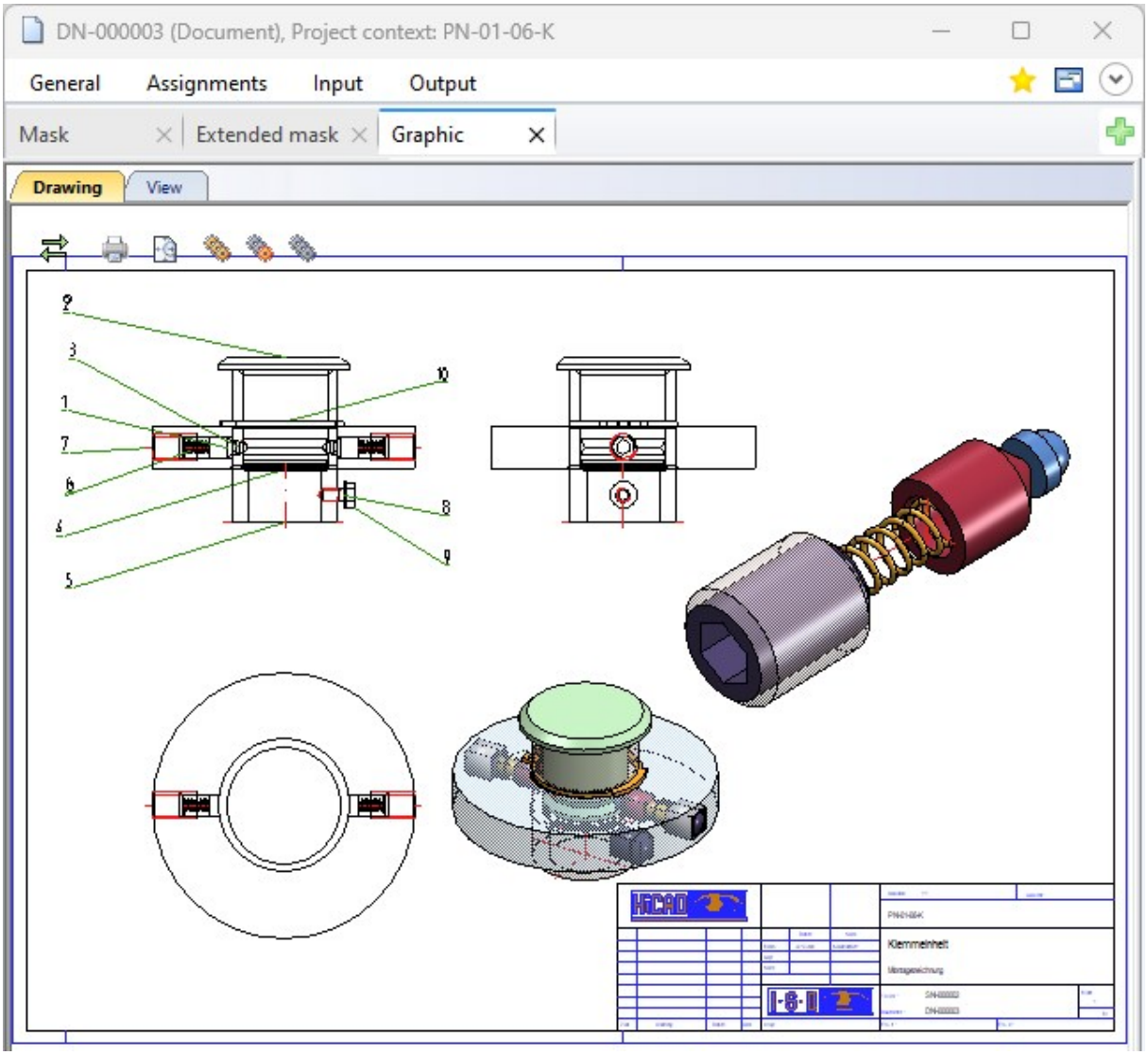


The prerequisite for a **Workflow** selection at the bottom of the window is that the logged-in user has the appropriate permissions.

In addition, the setting **Apply workflow of last state for revision** at **HELiOS Options > Database** must be set to **No**.

New HiCAD Viewer format

The integrated HiCAD Viewer is a tool that can be used to quickly and clearly evaluate HiCAD files also in the HELiOS Desktop interface.



Please note when using the HiCAD Viewer in HELiOS from Version 2024 (2900.1):

- When installing HiCAD with HELiOS, the Active X component of the HiCAD Viewer is installed automatically. This enables the preview of HiCAD files in HELiOS.
- On workstations on which only the HELiOS Desktop is installed in standalone mode, the HiCAD Viewer must be installed separately from HELiOS 2024 and updated if necessary. Otherwise, the preview of HiCAD files cannot be displayed in HELiOS.
- HELiOS 2024 has also changed the HEL_PREVIEW.INI settings file for the Viewer, which enables the configuration of the graphics windows in the HELiOS Desktop.
- For update installations of older HELiOS versions, the original INI file is still used and may need to be adjusted manually based on the new syntax, for example when using additional viewers.
- The above information also applies if only HELiOS is updated when using an older HiCAD version with HELiOS.

Italian and Polish remote maintenance

The support tool **TeamViewer**, which can be accessed from the HELiOS Desktop via  >  **Remote Maintenance** , is also available in Italian and Polish on HELiOS 2024.

Using the new Report Manager with HDE reports

Within the HDE reports you can specify which RMS file (old Report Manager) should be used. From HELiOS 2024 you can also use the new RM_SETTINGS files and thereby start the new Report Manager (from 2023).

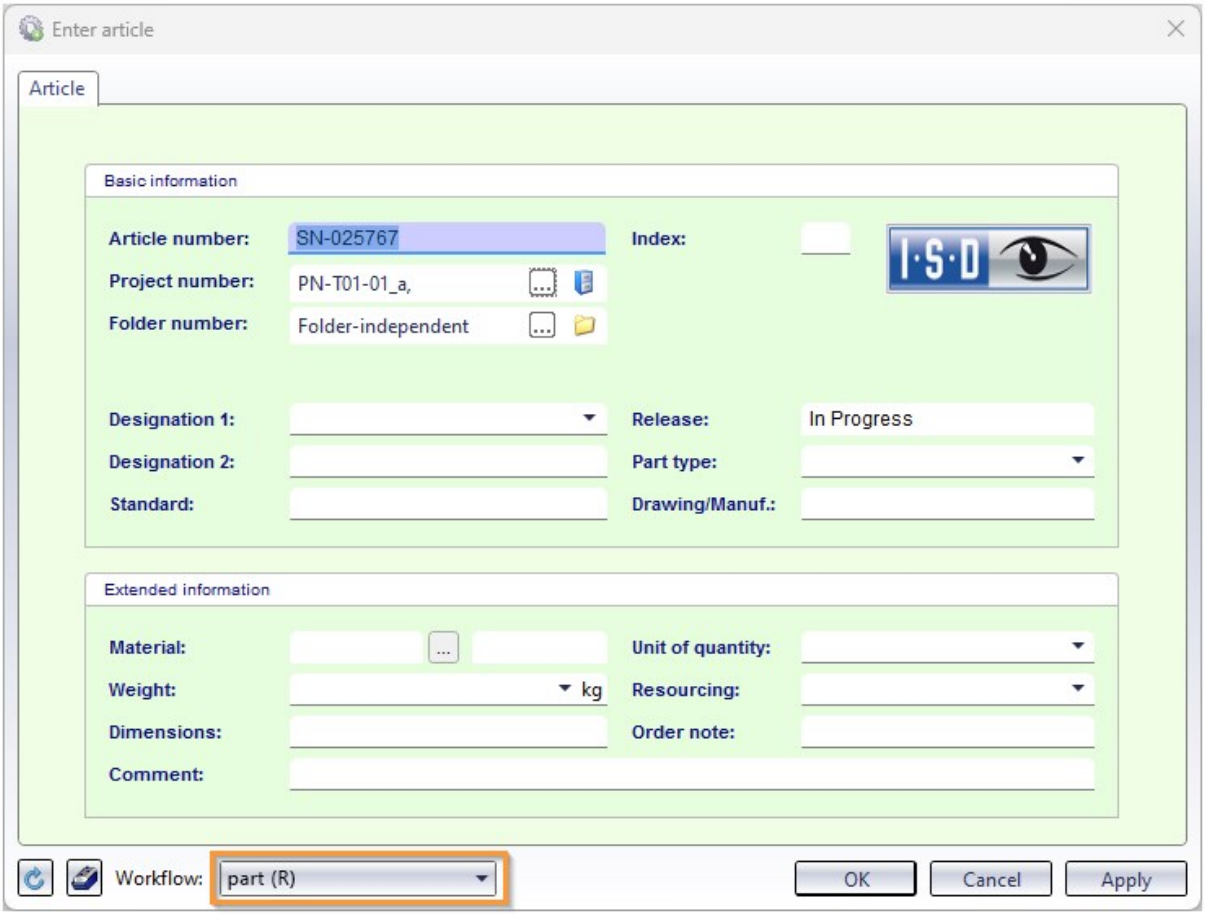
HELiOS in HiCAD

Service Pack 1 2024 (V 2901)

Workflow selection

In earlier HiCAD versions, a dialogue for workflow selection appeared first when calling up new Drawing functions, e.g. **New drawing with database, with new article master**  zunächst ein Dialog zur Workflowauswahl.

As you can now find a pull-down workflow selection at the bottom of an input window, this unnecessary intermediate step has been removed.



The prerequisite for a selection option is, of course, that more than one workflow exists in the system for the corresponding object type.

HiCAD 3-D annotation with HELiOS data

As of HiCAD 2024 SP1, HELiOS data is saved in annotations with the drawing. This data is then used when working without HELiOS. This also applies if a different sheet is printed via the Plot Manager (from HiCAD 2024 SP1) than the sheet that was active when the drawing was saved. Previously, the HELiOS data was missing in the annotation tags in this case.

Please note that this change does not affect existing drawings. These must first be saved again.

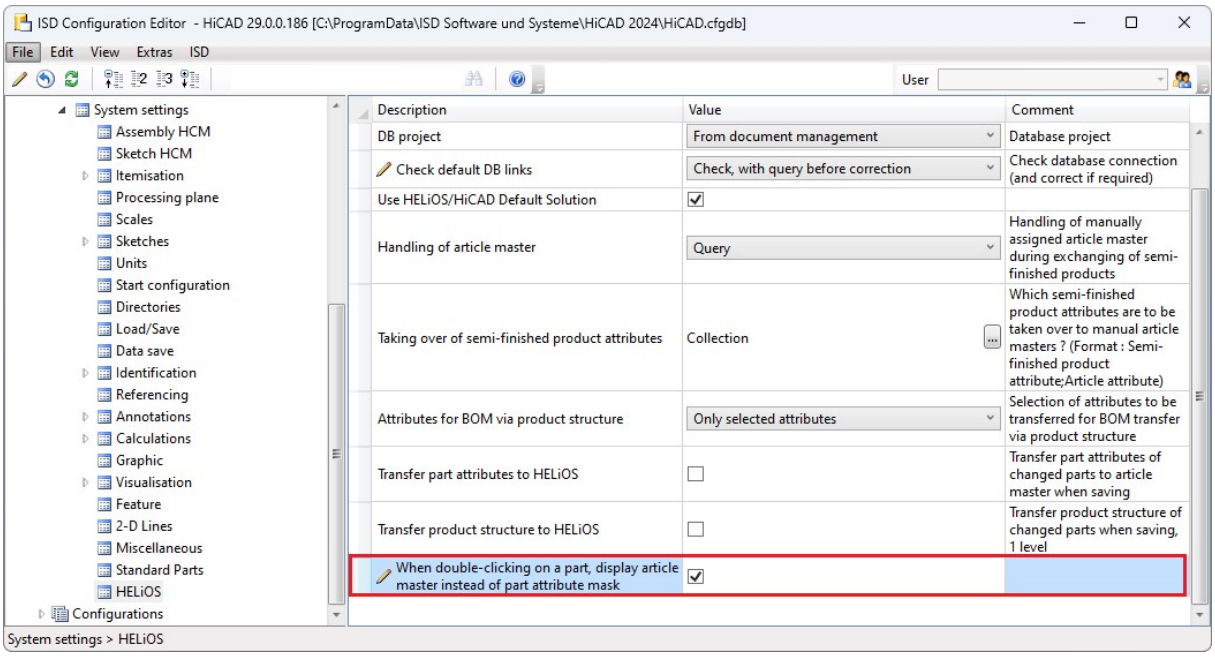
Major Release 2024 (V 2900)

Performance

By reducing HELiOS queries when loading drawings, a significant performance increase in the HiCAD/HELiOS interaction could be achieved.

Article master display by double-click

Up to now, double-clicking on a part in the drawing or in the ICN called up the Part attributes dialogue window. As of HiCAD 2024, the article master can now be displayed alternatively when using HELiOS. This can be set in the Configuration Editor at **System settings > HELiOS** .



If the checkbox is active and the clicked part does not have an article master, the **Part attributes** dialogue window is automatically displayed.

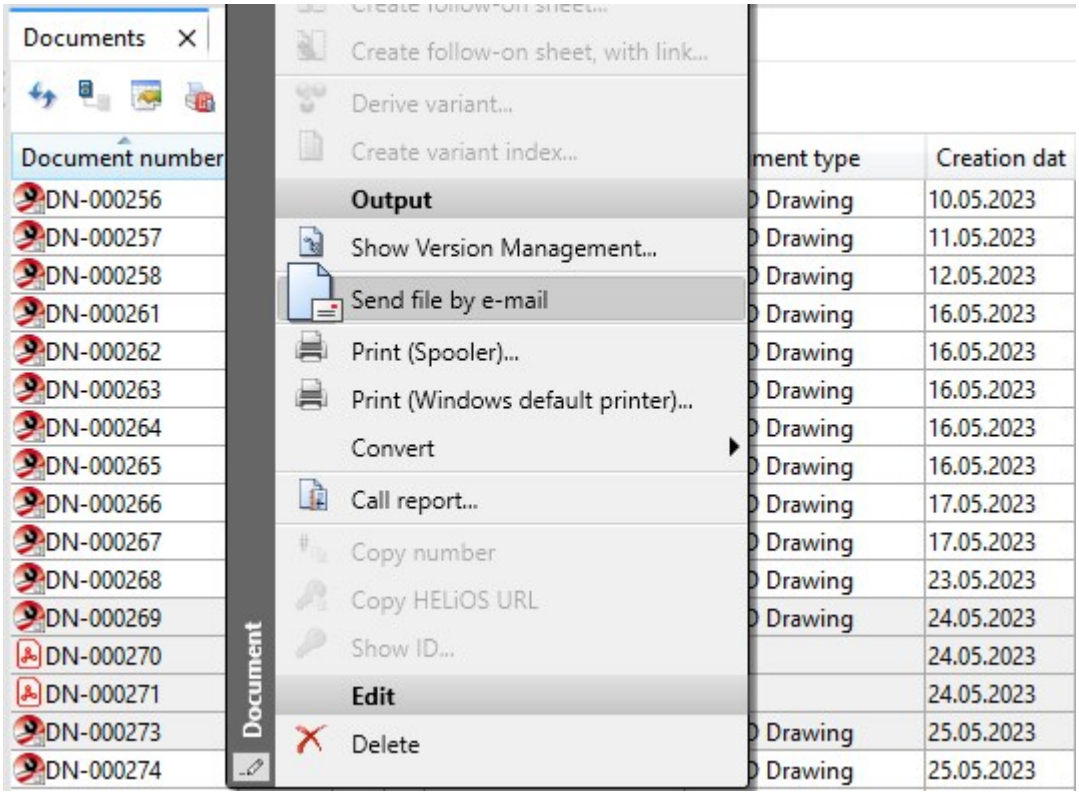
HELiOS MS Office Interface

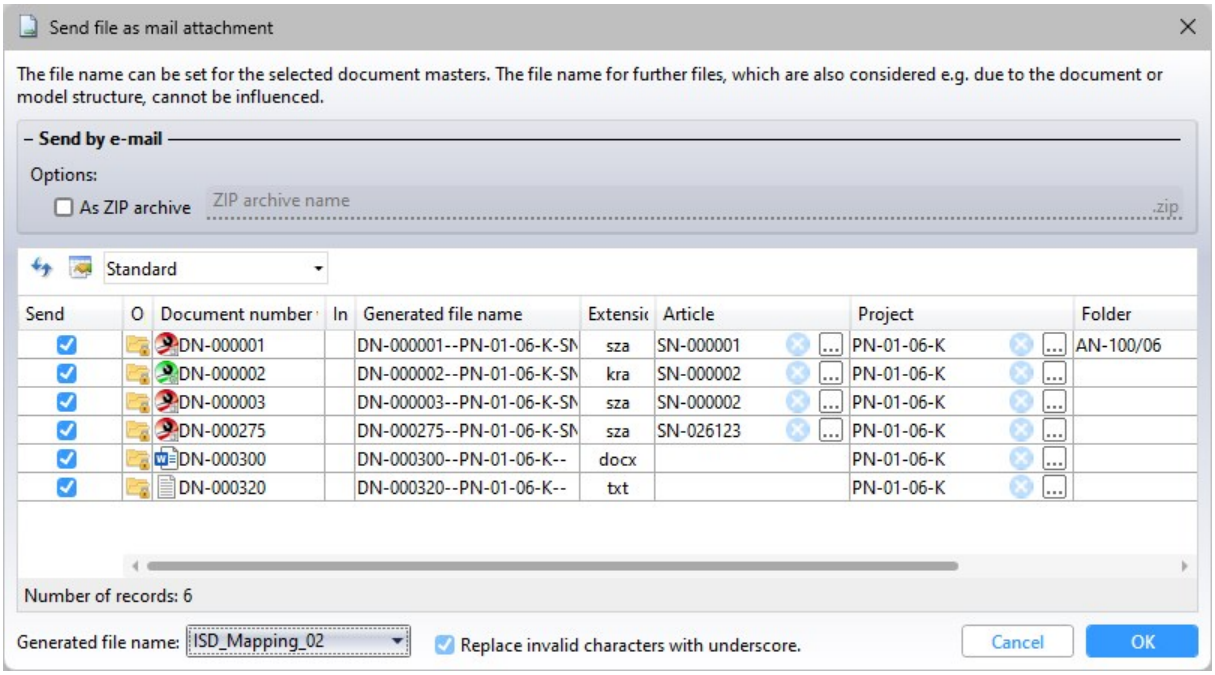
Major Release 2024 (V 2900)

Send file by e-mail

Both via the context menu of document result lists in HELiOS and from document detail windows, you have the option of sending corresponding files via the function  **Send file by e-mail ...** to the standard e-mail programme in your local system.

The range of settings and functions has been improved and simplified in HELiOS 2024.




Send file as mail attachment
The file name can be set for the selected document masters. The file name for further files, which are also considered e.g. due to the document or model structure, cannot be influenced.
– Send by e-mail
Options:
As ZIP archive
ZIP archive name
.zip
Standard
Send
O
Document number
In
Generated file name
Extensi
Article
Project
Folder
DN-000001 DN-000001--PN-01-06-K-SN sza SN-000001 PN-01-06-K AN-100/06
DN-000002 DN-000002--PN-01-06-K-SN kra SN-000002 PN-01-06-K
DN-000003 DN-000003--PN-01-06-K-SN sza SN-000002 PN-01-06-K
DN-000275 DN-000275--PN-01-06-K-SN sza SN-026123 PN-01-06-K
DN-000300 DN-000300--PN-01-06-K-- docx PN-01-06-K
DN-000320 DN-000320--PN-01-06-K-- txt PN-01-06-K
Number of records: 6
Generated file name:
ISD_Mapping_02
Replace invalid characters with underscore.
Cancel
OK

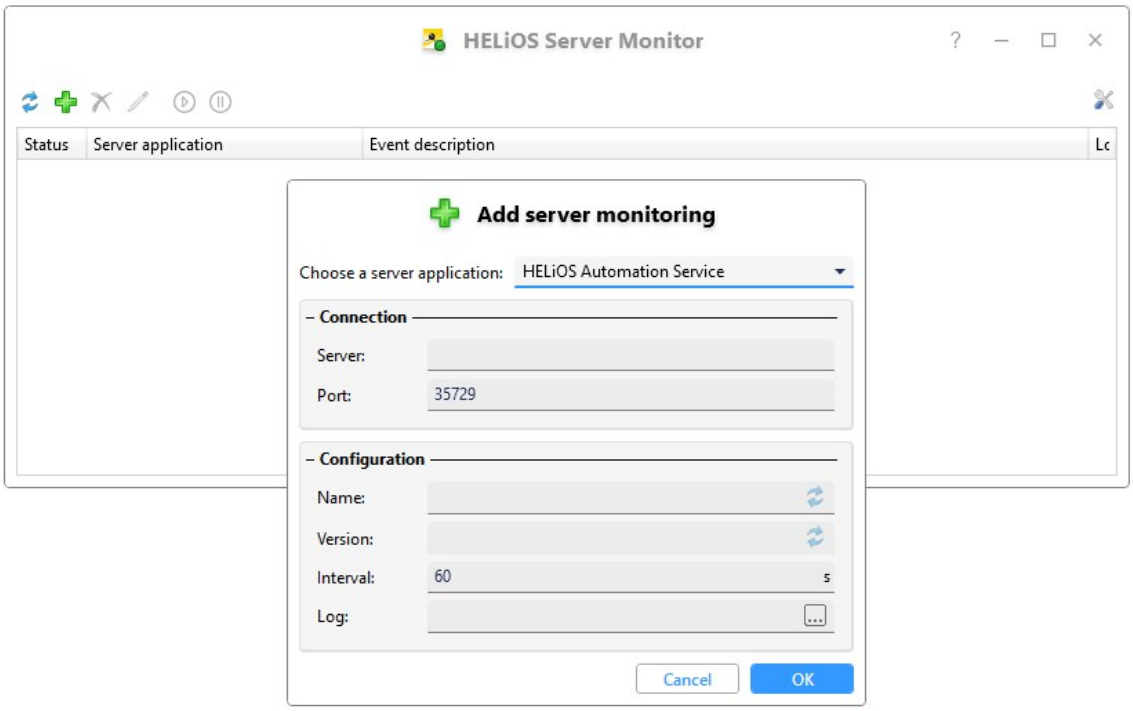
HELiOS Vault Server

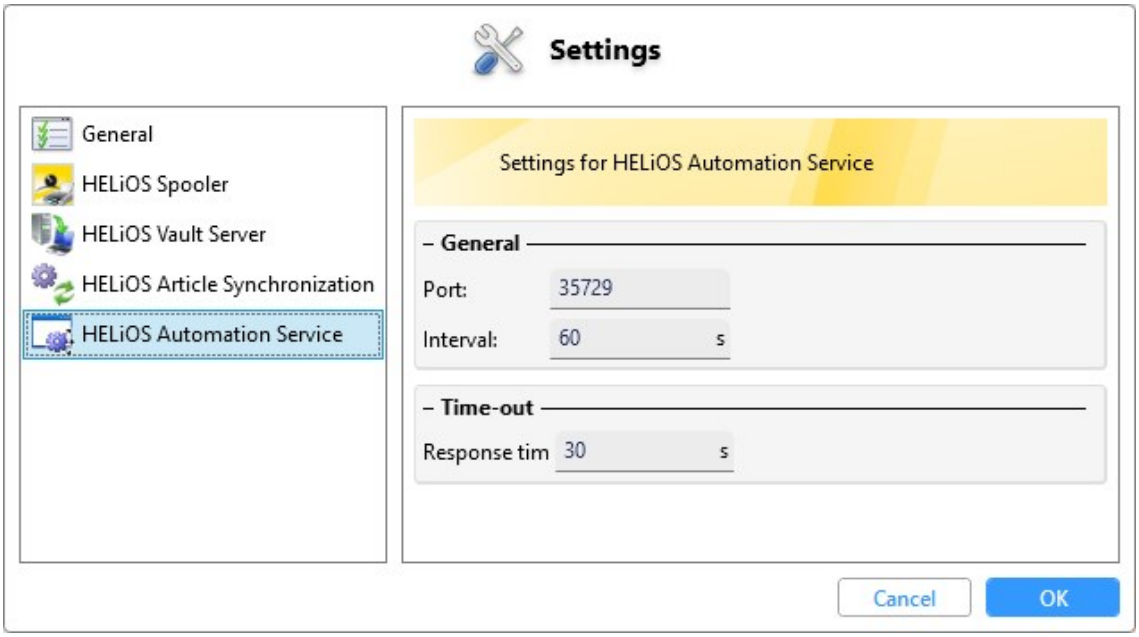
Major Release 2024 (V 2900)

Server Monitor: HELiOS Automation Service

The **HELiOS Server Monitor** has been extended with the menu item **HELiOS Automation Service** by the possibility to monitor the running of the server service Helios.ErpService.exe.

This controls ERP interfaces and other tasks.





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HiCAD_2901

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