HiCAD

UNLIMITED CAD PERFORMANCE **DEVELOPED BY ISD**

HiCAD 2025 - What's new?

Version 2025

News Overview

Date of issue: 13/07/2025

isdgroup.com

I·S·D

THE WORLD OF CAD AND PDM SOLUTIONS

TOC

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. 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.

Basics

Service Pack 2

Freely configurable date format in PostScript file names

When creating PostScript files using the **Drawing** > **Save/Print** > **Print** > **Selected model drawings**  function, the keywords **<Day>**, **<Month>** and **<Year>** are now supported in addition to the placeholder **<Date>** for the file name. This means that the date format can be freely configured at this point. The year is displayed as 4 digits, and the month and day are always displayed as 2 digits.

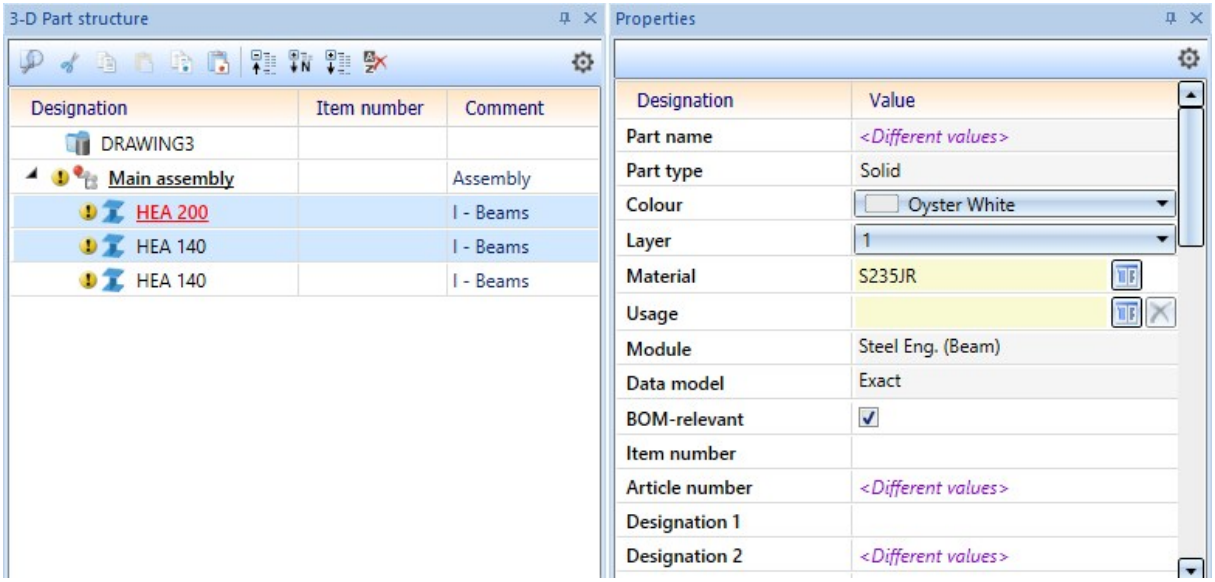
Reorganisation of the ICN

Properties dialogue window

The Properties dialogue has been supplemented with a multiple selection option: If several parts are selected in the part structure or in the drawing, a common mask is displayed in the Properties dialogue for all selected parts. In this dialogue, values for all selected parts can be changed or entered together. If only parts of a certain part type are selected, the corresponding part type dialogue appears; for parts with different part types, a special multiple selection dialogue is displayed.

A row is still displayed for attributes that are not present in the selected part so that they can be set directly via the properties dialogue box.

The HELiOS user display has been removed from the Properties dialogue and is now located in the title bar of the drawing.



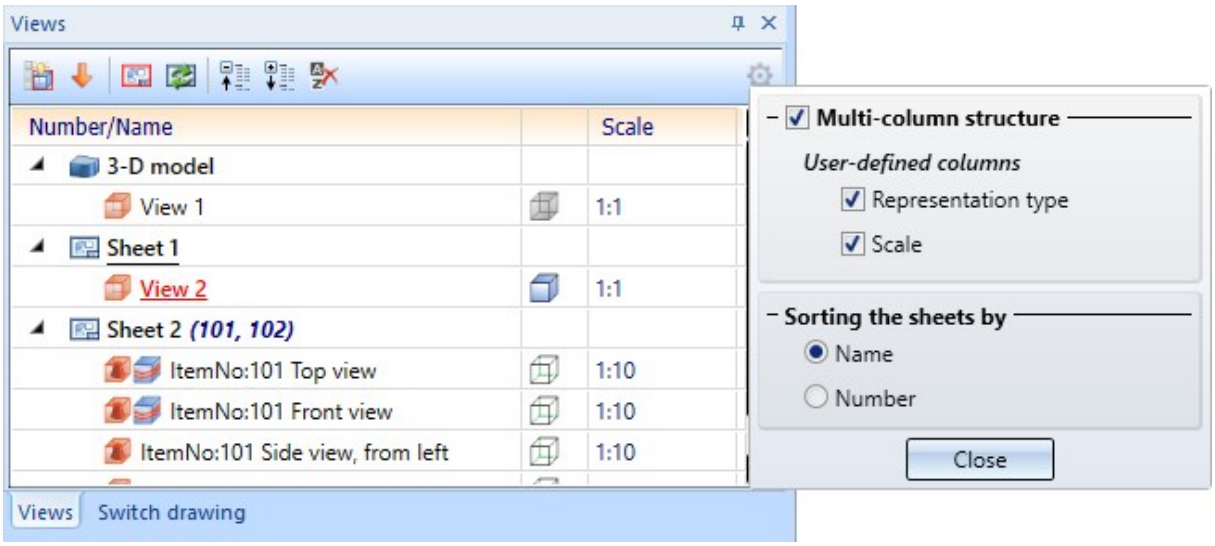
Reworking of the view window in the ICN

The settings menu for the view window in the ICN has been expanded to include the areas **User-defined columns** und **Sorting the sheets**.

You can use the checkboxes **Representation type** and **Scale** to show or hide the corresponding columns. By clicking on the column header, views can be sorted in ascending or descending order according to the **Representation type** or **Scale**.

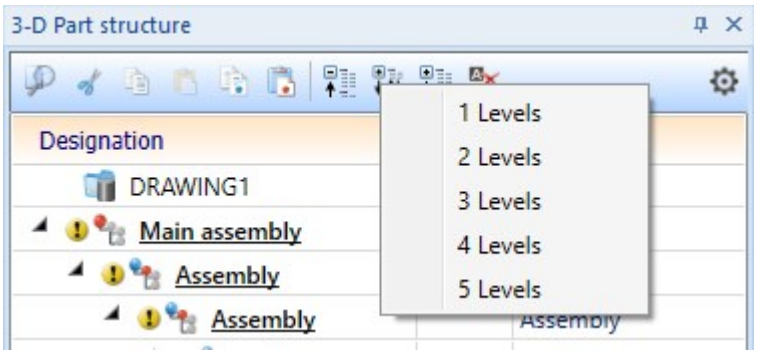
At the bottom of the page, you can decide whether the sheets containing the production drawings should be sorted by **Name** or **Number**. The sheet name, if available, is given in brackets after the sheet number. The sorting criterion is printed in bold. Clicking on the **Number/Name** header allows you to change whether the sorting is ascending or descending.

You can access the settings menu by clicking the ⚙ symbol on the toolbar of the view window.



Extension of the part structure toolbar

The 2 and 3 buttons from the toolbar of the 3-D and 2-D Part structure have been replaced by the **Expand** N button. Clicking the button opens a list of all the levels in the drawing. You can then select how many levels you want to expand.



Improved layout of options in the Novice Configuration

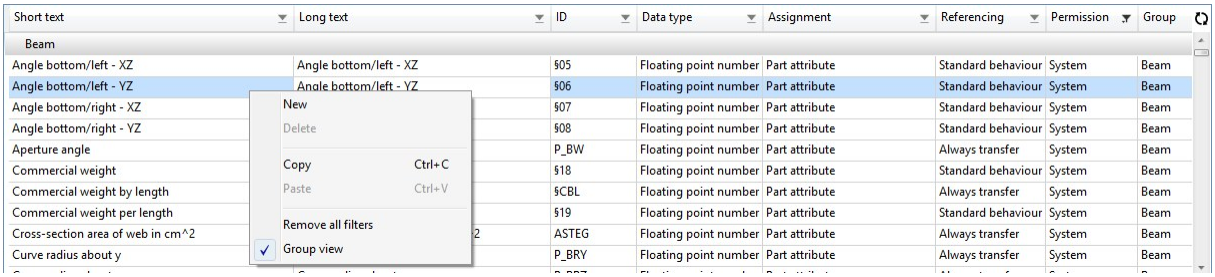
In the **Drawing** > **Others** > **Extras** > **Novice configuration** dialogue window, the **Right-click to use active surface as plane (Element snap mode)** option has been moved from the **Sketch and 3-D sketch** to the **Shape** area, as the option affects not only sketches but also standard processing. The arrangement in the Configuration Editor under **System settings** > **Novice Configuration** has been adjusted accordingly.

Attribute management

The management of attributes in the Configuration Editor under **System settings** > **Attribute management** > **Attributes** has been revised:

Duplicating attributes

The context menu of the **Attribute management** now includes the option to duplicate attributes. To do this, right-click on the attribute to be duplicated and select **Copy**. Then right-click again to open the context menu and select **Paste**. This creates another row in the attribute table with the attribute as a user attribute.



Transfer of referencing settings to attribute management

The attribute management table has been expanded to include the **Referencing** column. This column replaces the table for synchronizing specific attributes, which was previously located in the Configuration Editor under **System settings** > **Referencing**.

Attribute description

The **Description** column has been added to the attribute table. Some attributes are explained in more detail here. Double-click to change the description or add one for attributes without a description. If you select the **Multilingual** checkbox, you can add the description in other languages.

The descriptions are used as tooltips when selecting attributes in the text editor.

Attribute calculation

In the Configuration Editor's **Attribute calculation** table, the texts in the **Description** and **Comment** columns and in the selection lists have been changed to make the settings easier to understand. For example, the setting **Transfer coated surface to** has been renamed to **Assignment of value to attribute Surface area**. In the comments column, the attribute name is now always given next to the attribute ID (e.g *Weight (§01)* instead of *§01*).

Attribute calculation for locked assemblies

In locked assemblies, it is possible to change sub-parts that are referenced externally. A change to sub-parts of an assembly causes previously calculated attributes to become invalid. If the assembly is locked, the invalid attributes are not recalculated.

Previously, in such cases, each time the part was reactivated, the system checked whether the assembly was locked, which led to a performance issue.

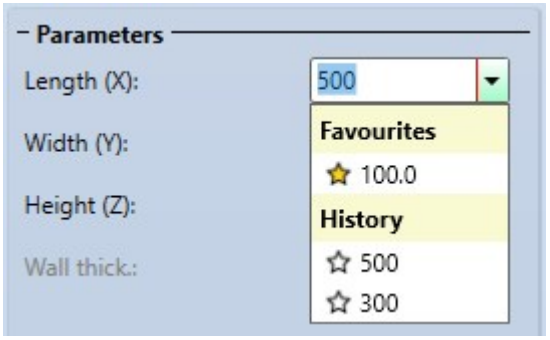
From now on, a part that leads to the cancellation of an attribute modification is marked, and another modification attempt is only made once the part has been modified again.

As part of this, the setting **Always** in the attribute calculation table in the Configuration Editor has been renamed to **With every change**.

The change means that the calculated weight is no longer updated in locked assemblies, which was previously the case. If desired, the old behaviour can be set in the Configuration Editor under **Compatibility** > **Attribute calculation**.

Favourites in input fields

In the input fields of the modernised dialogue windows, it is now possible to save values from the history as favourites. To do this, click on the star next to the value.



Refinement in the combination of manual and automatic itemisation

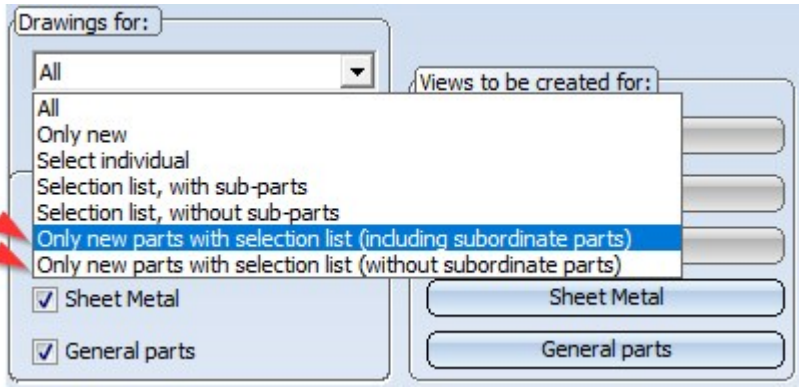
When assigning item numbers, you can choose between automatic **Rule-compliant** itemisation and manual **Rule-compliant** or **Unregulated** itemisation.

Previously, when the settings of some part groups were changed from **Unregulated** to **Rule-compliant** and then automatically itemised, the item numbers of the parts that had previously been manually itemised and were still listed as **Unregulated** both before and after the change were marked as invalid. This behaviour has been fixed so that the item numbers of these parts are retained.

The itemisation settings are set using the dialogue window **Itemisation with options**. The Automatic itemisation is carried out using the functions **Itemisation** 1 ... n and **... with options** 1 ... n. When either of these functions is activated, an item number is automatically assigned to all BOM-relevant parts in the drawing, unless the corresponding part group is set to unregulated in the settings. The Manual itemisation is done using the function **Itemisation, individual parts** 1 ... n.

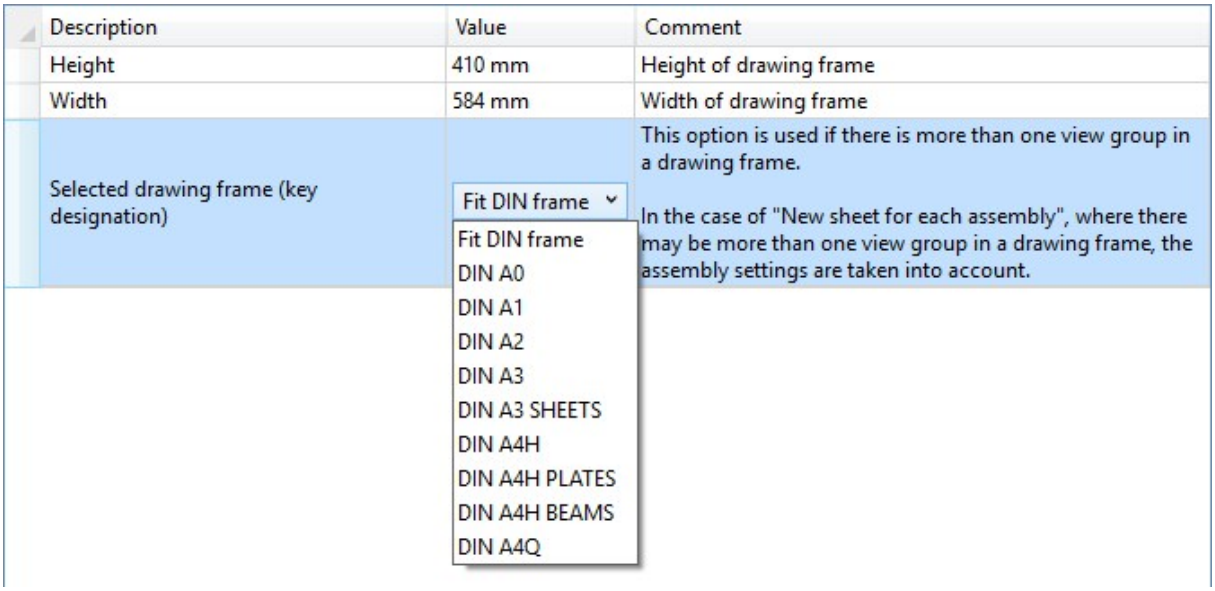
Drawing derivation: New options for selecting parts

In the dialogue of the **Drawing** > **Itemisation/Detailing** > **Drawing derivation** function, the options **Only new parts with selection list (including subordinate parts)** and **Only new parts with selection list (without subordinate parts)** have been added to the selection list in the area for drawings. If further parts are added to the drawing after the production drawings have been created, drawings can be created explicitly for these parts at a later stage if they are included in the selection list in the ICN **Part structure** window. If the selection list contains an assembly among the newly added parts, the derived parts of the assembly are either also derived **(including subordinate parts)** or not **(without subordinate parts)**.



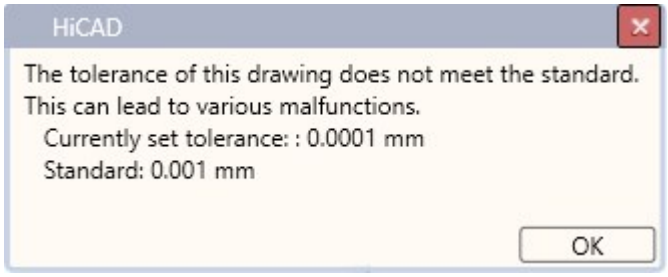
Default setting for the drawing frame in the Configuration Editor

In the Configuration Editor, the **Selected drawing frame** list box has been added to **Automatic drawing derivation** > **Production drawing** > **Drawing** > **Drawing frames** from the **Settings for drawing sheets** dialogue. If a certain frame is preselected here, it can be applied when creating a **Production drawing**. To do this, activate the **From configuration** checkbox in the **Drawing derivation** dialogue under **Drawing parameters**.



Drawing tolerance

Due to rounding errors, numerical values such as point coordinates can be checked for equality up to a certain tolerance level. The default value is 0.001. This means that two points are considered identical if their distance is less than a thousandth of a millimeter. If you change the drawing tolerance using the **Tolerance** function, a warning message now appears when you load the drawing.



In the Configuration Editor, go to **Compatibility** > **Warnings** > **Check design tolerance** to disable the message if the value is between 0.0001 and 0.01.

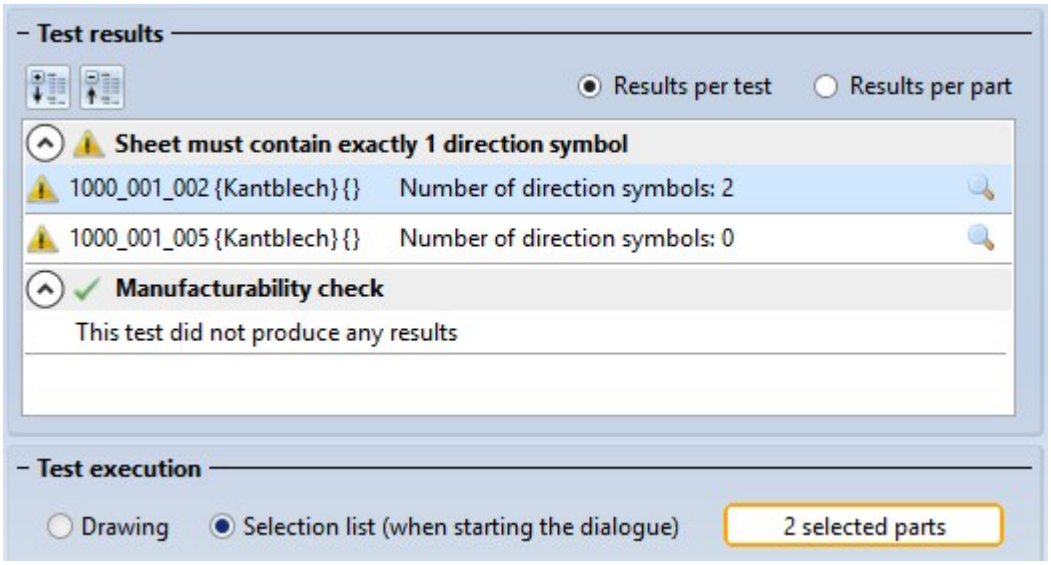
Please note:

The tolerance is a mathematical tolerance, not a tolerance for identical part search during itemisation. A change should only be made for good reason, as otherwise any modelling and part generation functions may fail.

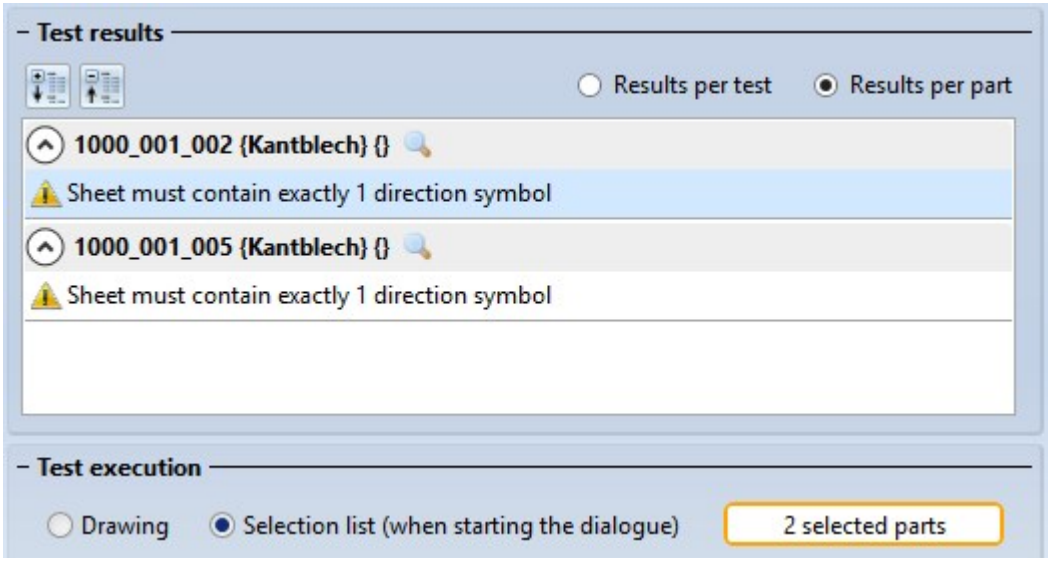
Design Checker

Test results

The display of test results has been expanded in the Design Checker. You can now display the results **per test** or **per part**.



If you have selected **Results per test**, all parts with errors are displayed under the check. Parts without errors are not displayed. If the check is successful, a corresponding message is displayed below the check.



The **Results per part** option lists only those parts that contain errors. The faulty check is listed below the part.

Manufacturability check

The Manufacturability check in the Design Checker has been expanded to include several additional checks. As with the existing checks, you can specify whether these checks are to be performed in the Configuration Editor under **Sheet** > **Manufacturability check**. You can also enter the corresponding reference values there.

Alternatively, the reference values can be loaded from the catalogue. To do this, the new columns must first be filled with the corresponding practice-oriented data. For sheets with semi-finished products, the tables under **Factory standards** > **Sheets** apply, and for sheets to which a material from the catalogue has been assigned in the part attributes, the tables under **Factory standards** > **Manufacturability check** apply.

If no reference values are stored in the Configuration Editor or in the catalogue, the calculation cannot be performed. In this case, the Design Checker displays a corresponding message.

In addition, you can now specify in the Configuration Editor that the manufacturability check is also performed when creating sheet developments.

The following new checks are available:

- **Minimum flange length**
  As with chamfering, the minimum length of flanges is measured directly on the outside or directly on the outer tangent. Only the outside of the cover surface is measured; chamfers may reduce this. A distinction is made between acute and obtuse angles.
- **Minimum bend radius**
  To avoid overloading the material, a certain minimum bend radius must be ensured.
- **Distance between processings**
  To check the distance between processings during the manufacturability check, activate the **Check minimum distance between processings** parameter in the Configuration Editor.
- **Minimum diameter for standard bores**
  This test only applies to standard bores.
- **Minimum Z-fold height**
  The comparison values are loaded from the catalogue during this check. You must therefore first fill in the columns **min. Z-fold height (<90°)** and **min. Z-fold height (>=90°)** of the corresponding tables.

Please note:

You can also use the following tests to check Steel Engineering plates:

- **Distance of processings to edge**,
- **Maximum sheet dimensions**,
- **Collisions in developments**,
- **Distance between processings**,
- **Minimum diameter for standard bores**.

For this purpose, the catalogues under **Semi-finished products** > **Plates** have been expanded to include the corresponding columns.

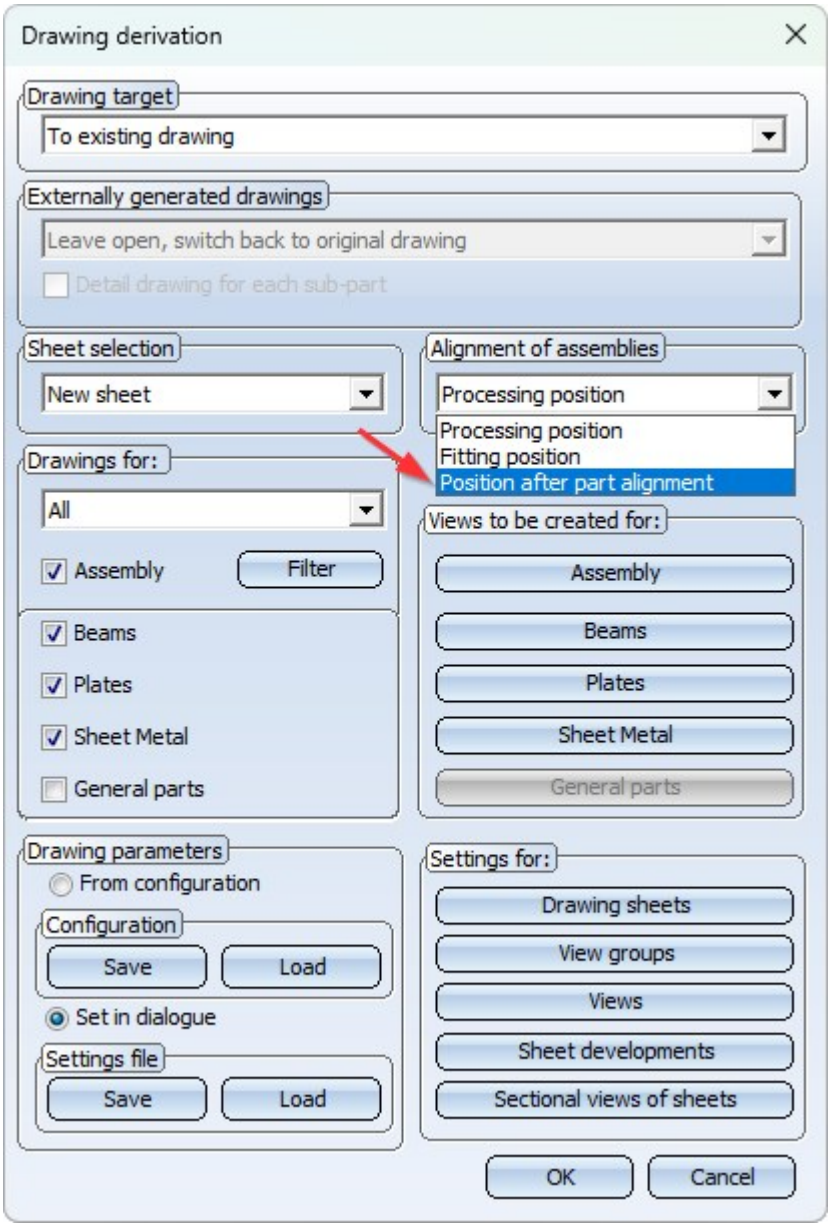
Optimise cursor size

With wider windows, the cursor size was unnecessarily large. The new formula calculates the size of the cursor as a percentage of the graphics window. This can be adjusted in the Configuration Editor under **System settings > Visualisation > Curser size**.

Service Pack 1

Drawing derivation: New option for alignment of assemblies

The new option **Position after part alignment** has been added to the **Drawing** > **Itemisation/Detailing** > **Drawing derivation** function during the **Alignment of assemblies**.



An example of how part alignment influences the creation of the drawing derivation can be found here.

Attribute for the pre-bend length of beams

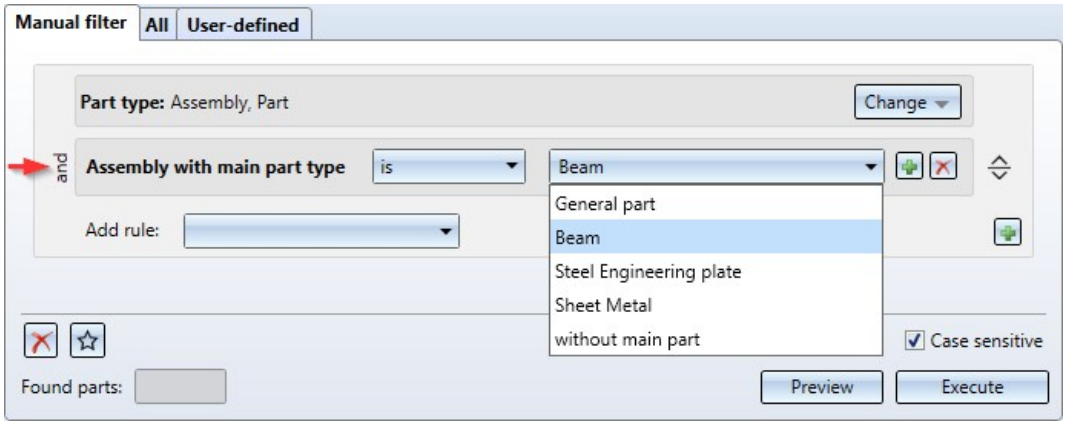
The new HiCAD attribute **§PSL** has been added, which is assigned the **Pre-bend length** of beams. The pre-bend length is the original length of a curved beam. For straight beams, this corresponds to the normal beam length. The part attribute is assigned as soon as a beam is newly created or processed. The value of the attribute is always cal-

culated and cannot be entered manually. The attribute does not appear in dialogue masks such as the **Part attributes** window, but must be configured manually if required.

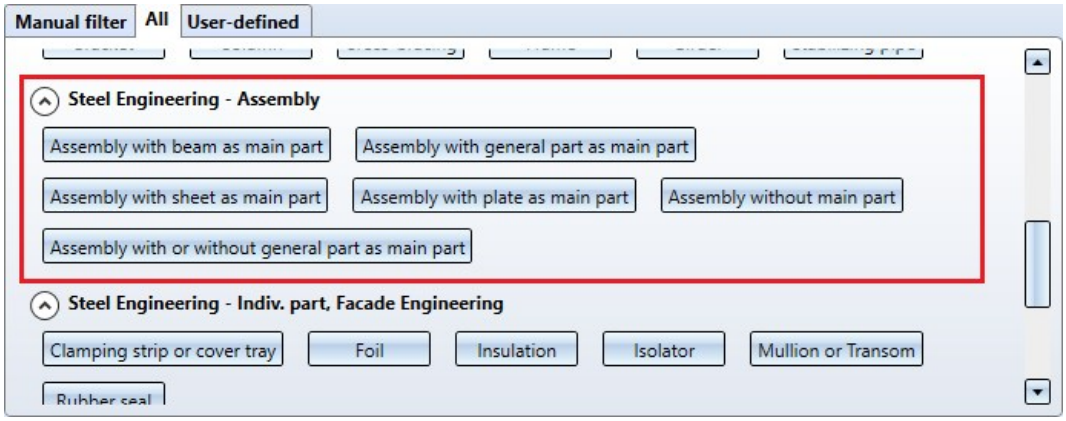
New options for part filters

New options have been added to the **Part filters** dialogue window of the **Find** function in the transparent toolbar:

The rule **Assembly with main part type** has been added to the **Manual filter** tab. This allows you to filter for assemblies that have a part marked as the main part. Use the right-hand list box to select whether you are looking for beams, plates, sheets or general components as the main part, or for assemblies without a main part. A part is marked as the main part of an assembly either automatically, e.g. when a connection from the Civil Engineering functions is inserted, or manually using the part's context menu under **Assembly/Part structure** > **Change part** > **Into assembly main part**.



The **All** tab contains the settings for the rule **Assembly with main part type** summarised under **Steel Engineering - Assembly**.



Filter favourites that contain the rule **Assembly with main part type** cannot be read or executed in versions prior to HiCAD 2025 SP 1.

Configuration of drawing derivation

Further settings are now available in the Configuration Editor under **Automatic drawing derivation** > **Production drawing** for the automatic creation of workshop drawings using the **Drawing derivation** function.

Settings for beams

The settings **Production drawing** and **Processing note for unprocessed beams** were previously under **System settings** > **PDM** > **Drawing Management** > **Production drawing** and were only relevant for the **Drawing** function within Drawing Management.

The option **Production drawing for unprocessed beams** allows you to specify whether production drawings should be created for all or only for processed beams in a drawing.

The option **Processing note for unprocessed beams** can be used to specify 'exceptions', for which type of unprocessed beams a production drawing should nevertheless be created.

You can find more information here about when beams are considered (un)processed and how to add new processing instructions.

Settings for Steel Engineering plates

The option **Top view for steel engineering plates dependant on powder marking lines and lettering** has been added. for Steel Engineering plates. If you select **Yes**, the top side is determined based on powder marking lines and letterings when the **View from above** is created. The following applies:

The side with powder marking lines is selected as the top side. If there are powder marking lines on both sides of the sheet, the side with the higher number of lines is on top. If the sheet has powder marking lines and lettering, the side with the lettering is on top.

With the default setting **No**, one side of the sheet is randomly selected as the top side.

Powder marking lines and **Lettering** are created using the functions of the same name under **3-D Standard** > **Standard Processings** > Pull-down menu: **Cams**.

New function for creating a structure assembly

The new functions **Create structure assembly** and **Create structure assembly as sub-part** have been added to the **3-D Standard** > **New** > **New assembly** menu for creating structure assemblies. In addition, the new functions have been added to the **Context menu for drawings** under **New 3-D Main Part** and to the **Context menu for assemblies** under **New Part** > **Main part** or **New Part** > **Sub-part**. The functions behave like the other assembly functions and create an assembly that is not BOM-relevant and has the **Part typeStructure assembly**.

Until now, there was no direct function for creating structure assemblies. To identify an assembly as such, the **Part type** had to be set to **Structure assembly** either manually or after creation.

The settings for assemblies in the Configuration Editor have been moved from the **Steel Engineering** > **Assembly** area to the **Modelling** > **Part creation** > **Assembly** area. As part of the new functions for structure assemblies, the settings have been extended to include the item **Article number for structure assemblies**.

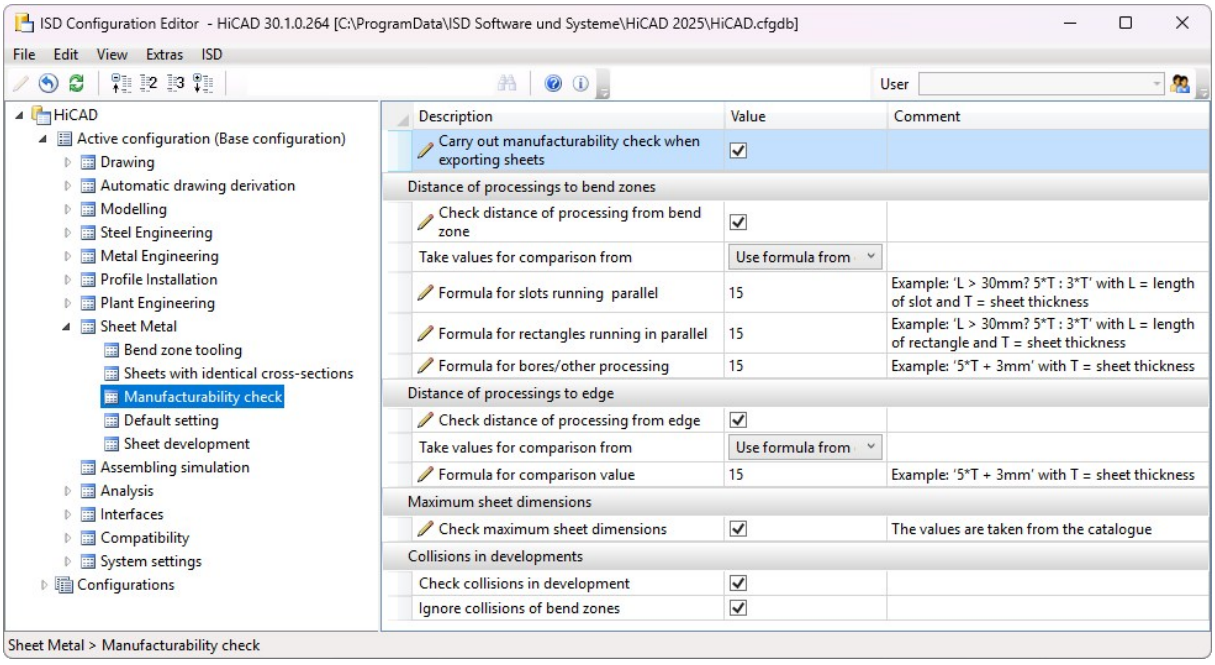
Design Checker

Manufacturability check for sheets

The new **Manufacturability check** is now available for sheets in the Design Checker. The following points can be checked:

- check distance from bend zones to other processings
- check distance from edges to other processings
- check maximum sheet dimensions
- check for collisions in development
- ignore collisions of bend zones during the check

You define the checks to be carried out when checking manufacturability in the Configuration Editor under **Sheet Metal** > **Manufacturability check**.



You also enter the distances from the processing to the bend zone or to the edge here. The **Design Checker** can take the values for the distances either from the catalogue or from the Configuration Editor. When taking the values from the Configuration Editor, simply enter a value or a **Formula** for the corresponding parameters (e.g. formula for parallel slots).
If you want to load the values from the catalogue, you must first fill in the columns

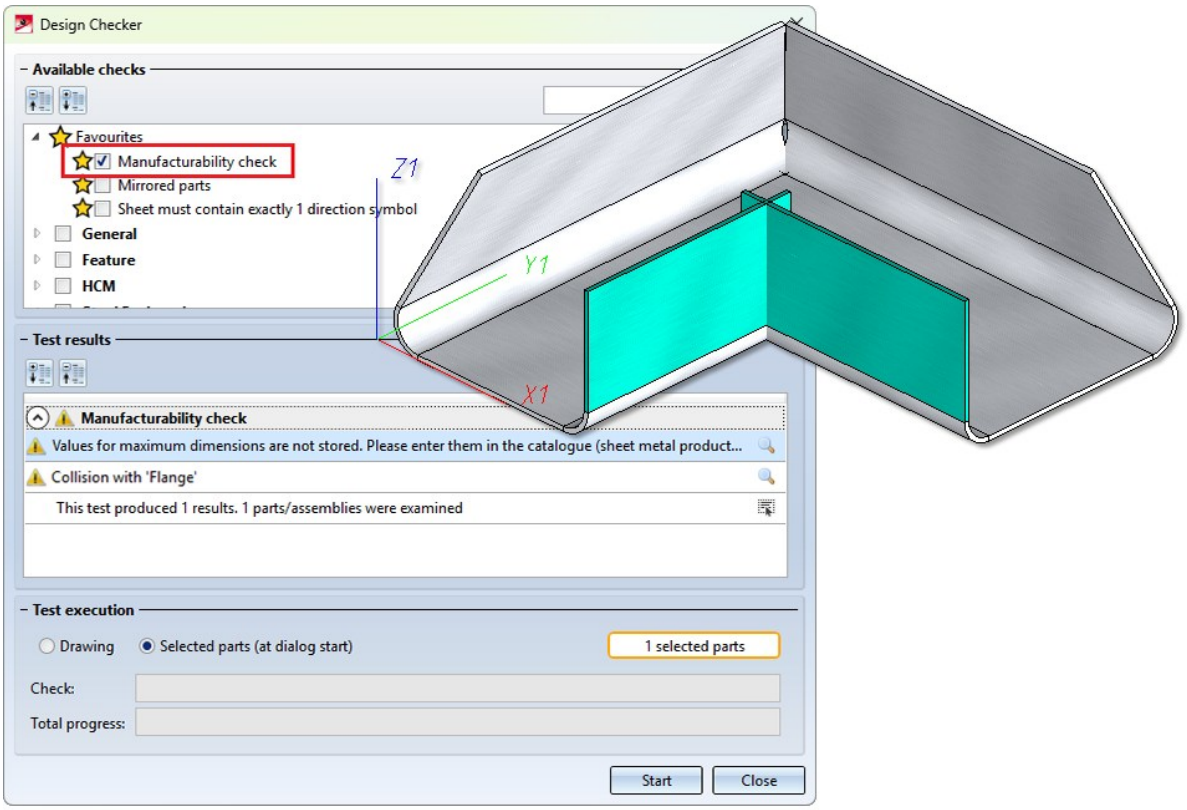
- Distance to edge
- Distance to bend zone
- Distance slot to bend zone
- Distance rectangle to bend zone

of the corresponding tables. For sheets with semi-finished products, these are the tables under **Factory standards** > **Sheets** and for sheets to which a material from the catalogue has been assigned in the part attributes, under **Factory standards** > **Sheet metal production**. If neither a semi-finished product nor a material has been assigned or if the columns are not filled in, the test cannot be carried out.

he check of the maximum sheet metal dimensions is now also part of the manufacturability test. What is new is that there is now only one table in the catalogue under **Factory standards** > **Sheet metal production** from which the

values are taken when material is assigned. The values from the tables under **Factory standards** > **Sheets** are evaluated when a semi-finished product is used. These tables have been supplemented by the columns **Max. length** and **Max. width** and must be entered by the user.

The collision-free development check is also part of the manufacturability test. It checks whether collisions between flanges or bend zones occur during a development or bending simulation. Depending on the selected option, the collision check is carried out with or without bend zones.



Changes in user guidance

- **Favourites and the meaning of the checkbox**
  If you select ☑ the category (e.g. General or HCM) when selecting the tests , all subordinate tests are automatically activated. If you deactivate some tests from the list, the category is marked with a square ■. If the test is marked with a yellow star ★, it is also listed under Favourites and can be selected from there.
- **Checked parts**
  The checked parts are now marked in newer checks (e.g. Check distance of processing from bend zone) by activating this symbol (next to the test result).
- **Custom tests in C#**
  Until now, tests for Design Checker had to be written in Python. This meant that the administrator had to learn an additional programming language to C#. From Service Pack 1 onwards, it is therefore possible to write individual tests in C#. Either scripts or precompiled tests can be used. Precompiled tests usually perform better.

Major Release

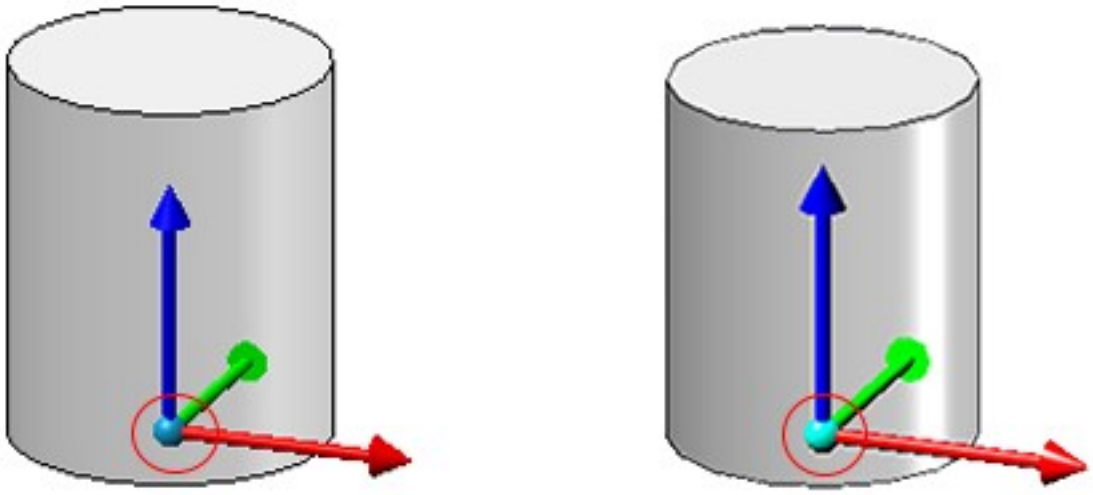
Update of the part attribute Total quantity

From HiDAD 2025, the part attribute Total quantity is no longer updated in drawings that are not itemised source models.

Drawings that are not saved as itemised source models include, for example, detail drawings that are created using the function **Drawing** > **Save/Reference** > **Reference part, Save, Detail drawing**  . When creating detail drawings, referencing is used to transfer not only the item number but also the part attribute total number from the main drawing to the detail drawing. A component that appears five times in total in the main drawing will also have a total of five in its detail drawing. If the part is changed in the detail drawing, the item number and total number of the part will become invalid. To update the total number, you must switch to the main drawing and itemise it again.

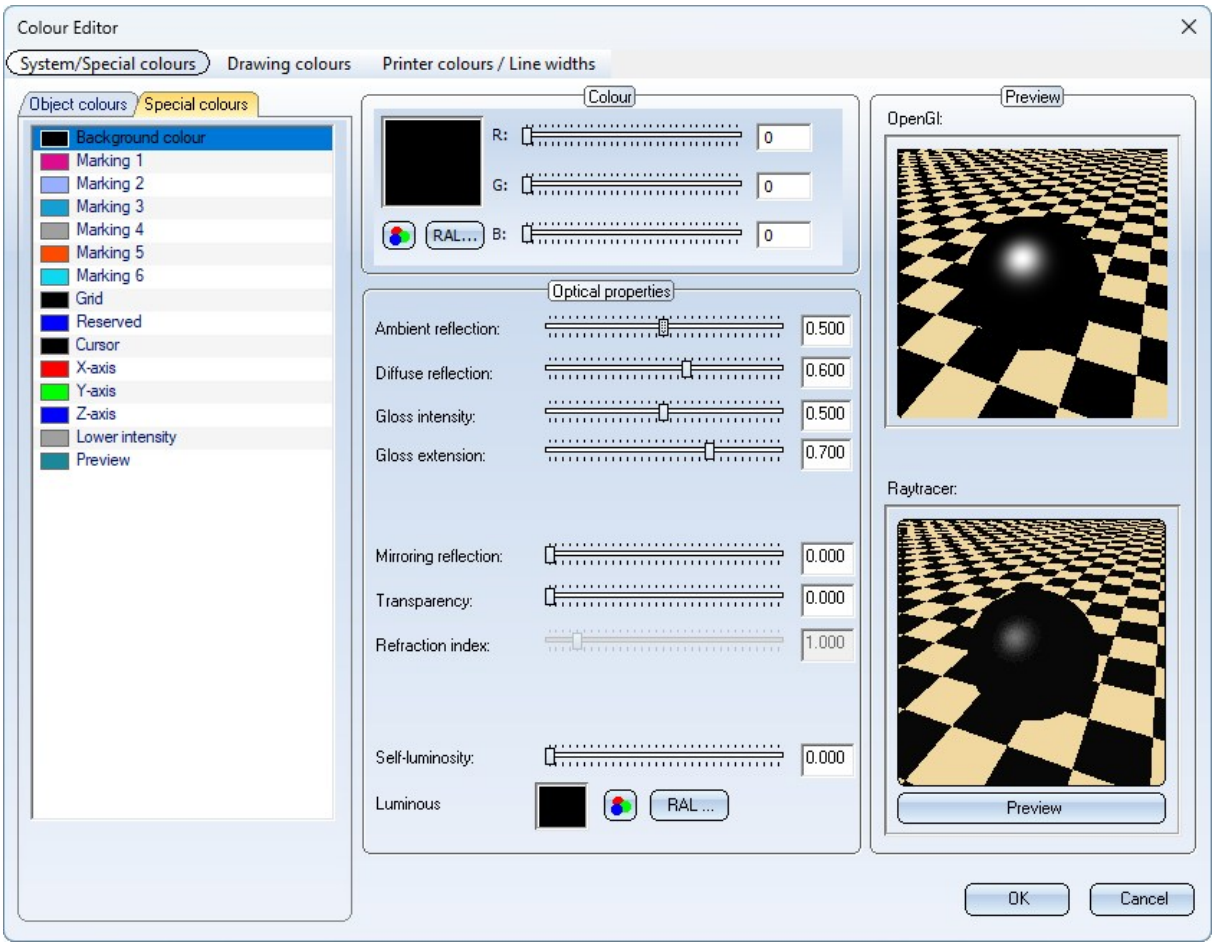
Change of the special colours

The special colours in HiCAD, which, among other things, determine the colouring of the coordinate systems, have been slightly changed. In the course of this, the colour of the sphere of the coordinate system was also changed. Instead of the colour **Marking 2**, the colour **Marking 6** is now used for the sphere.



Left: old coordinate system with sphere in old colour **Marking 2**, right: new coordinate system with sphere in colour **Marking 6**.

You can view and edit the special colours using the **Colour Editor** under **Drawing** > **Others**.

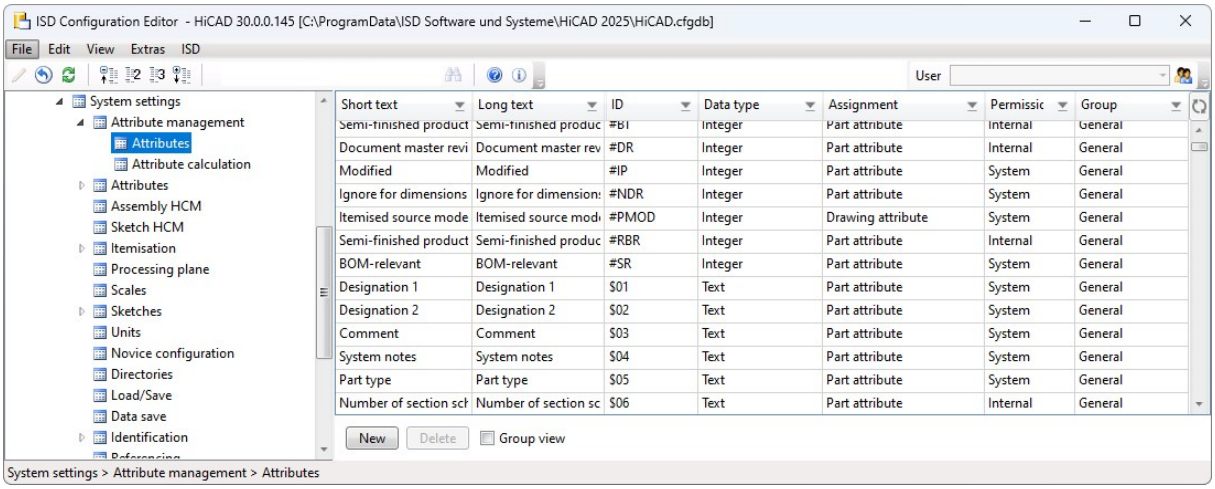


Changes to the management of HiCAD attributes

From HiCAD 2025, the administration of HiCAD attributes can no longer be found in the catalogue, but in the Configuration Editor under **System settings** > **Attribute management**.When an update installation is carried out, the catalogue update transfers the attribute information from the catalogue to the Configuration Editor.

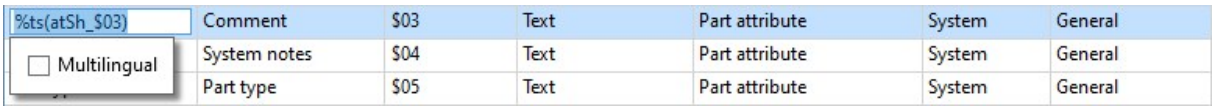
The definition of the drawing and part attributes is done under **System settings** > **Attribute management** > **Attributes**. Via the buttons **New** and **Delete**, new, customer-specific attributes can also be created and edited here.

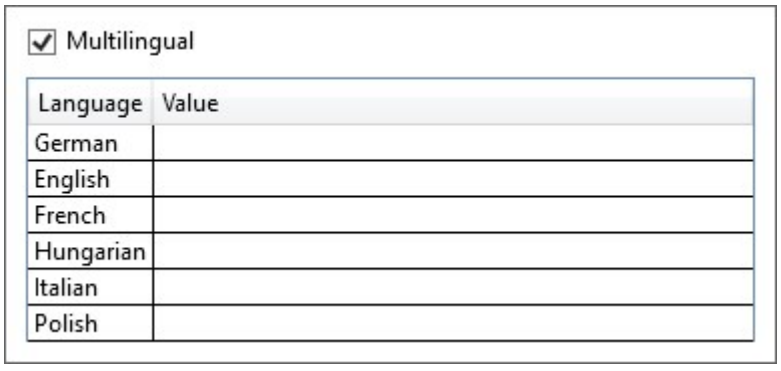
The settings for calculating attributes, which were previously found under **Modelling** > **Part attributes** , can now be found under **System settings** > **Attribute management** > **Attribute calculation**.



Further new developments concerning attributes are:

- The attributes have been divided into groups. The grouping is used in attribute selection dialogues, such as the Text Editor and when creating BOM definitions. In the Configuration Editor, you can also display the attributes sorted by group by activating the checkbox **Group View**.
- Attributes with the **Internal** permission are no longer displayed in selection dialogues.
- The **Short text** and **Long text** attribute descriptions can now be changed manually in the Configuration Editor by double-clicking the corresponding text field. It is also possible to save description texts in multiple languages. To do this, activate the **Multilingual** checkbox in the text field and fill out the table that appears.





A list of all HiCAD attributes can be found here.

Differentiation when selecting parts in a rectangle

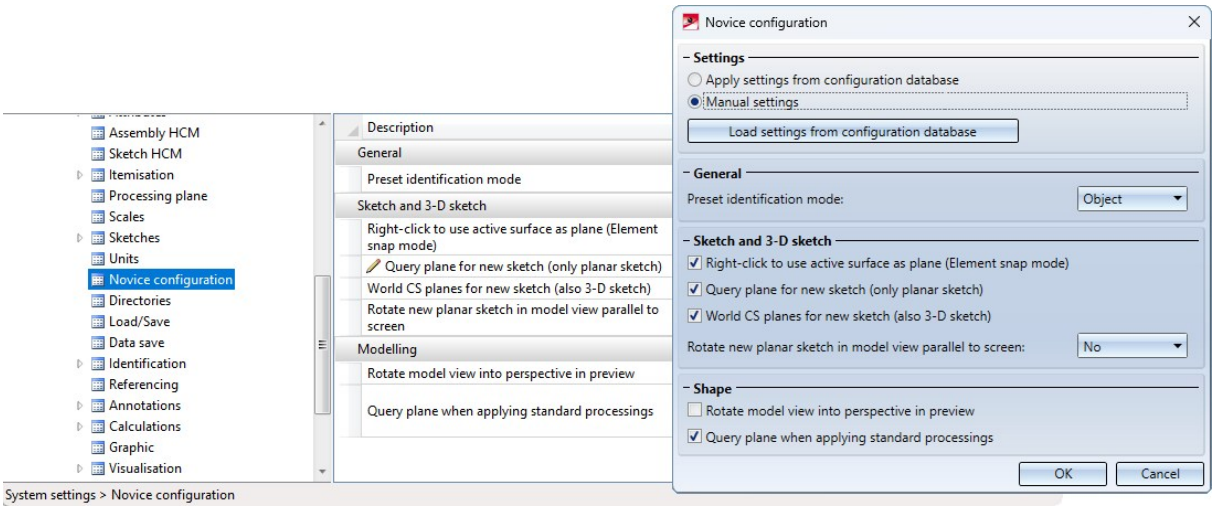
If the CTRL key is held down together with the left mouse button and a rectangle is drawn with the cursor, all 3-D parts within this rectangle are selected. From HiCAD 2025 onwards, a distinction is made when selecting parts in a rectangle depending on whether the rectangle is drawn from left to right or from right to left.

If you drag the rectangle from right to left, all parts within the rectangle are selected, even if they are not completely contained within it, provided that one point of the part lies within the rectangle.

When dragging from left to right, only parts that are completely within the rectangle will be selected.

Novice Configuration - Dialogue change

The dialogue of the **Novice Configuration** has been adapted so that the order and name of the options match those in the Configuration Editor under **System settings** > **Novice configuration**. The option **Default identification mode** has been renamed**Preset identification mode**. The **Modelling** area has been extended by the checkbox **Query plane when applying standard processings**.



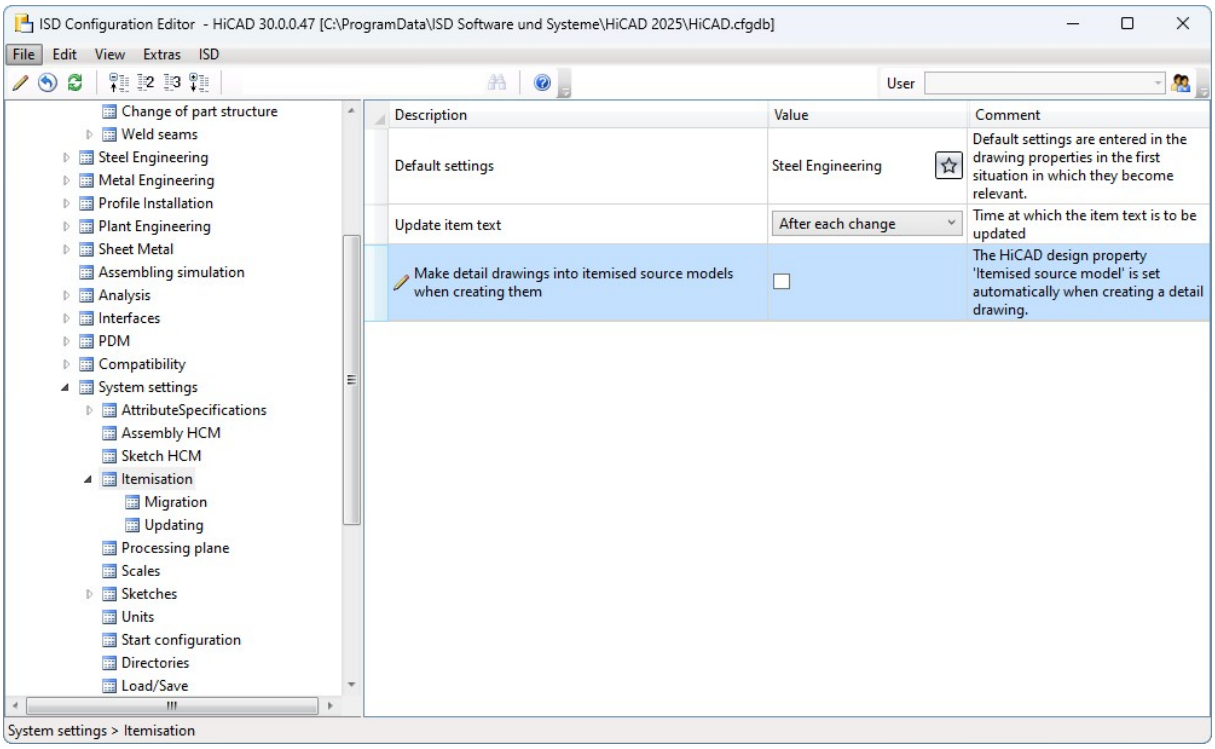
New function in QuickAccess toolbar

The **Start Centre** function has been added to the Die QuickAccess toolbar.

Save referenced detail drawings as itemised source model

As of HiCAD 2025, detail drawings created via the **Drawing** > **Save/Reference** > **Reference part, Save, Detail drawing** function are no longer automatically saved as itemised source models.This means that no itemising is possible when processing the detail drawing. An itemisation restriction 6in the detail drawings can be useful, as the item numbers of referenced parts are transferred to the original drawing. If a drawing is divided into several detail drawings and positioned there, it can happen that the same parts receive different item numbers in the original drawing.

If you still want to save detail drawings as itemised source models, you can set this in the Configuration Editor under **System settings** > **Itemisation** > **Make detail drawings into itemised source models when creating them**.



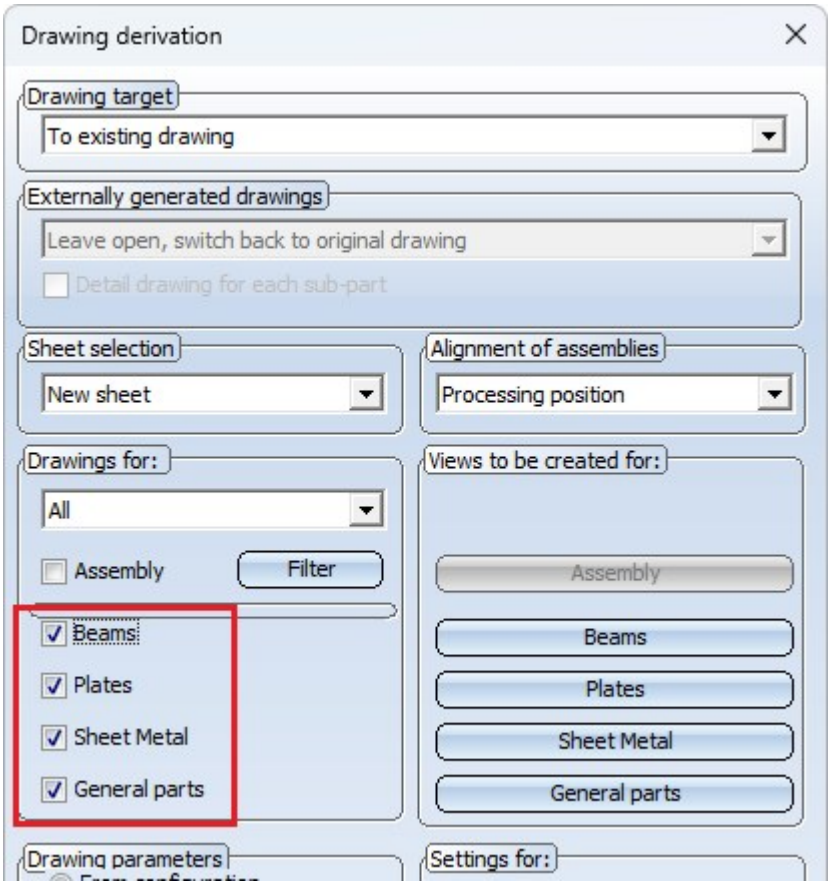
This change does not affect referenced **Detail drawings** that were created as part of drawing management. Here, the design property **Itemised source model** is still set via the checkbox in the **Document input** dialogue window.

SpaceMouse®

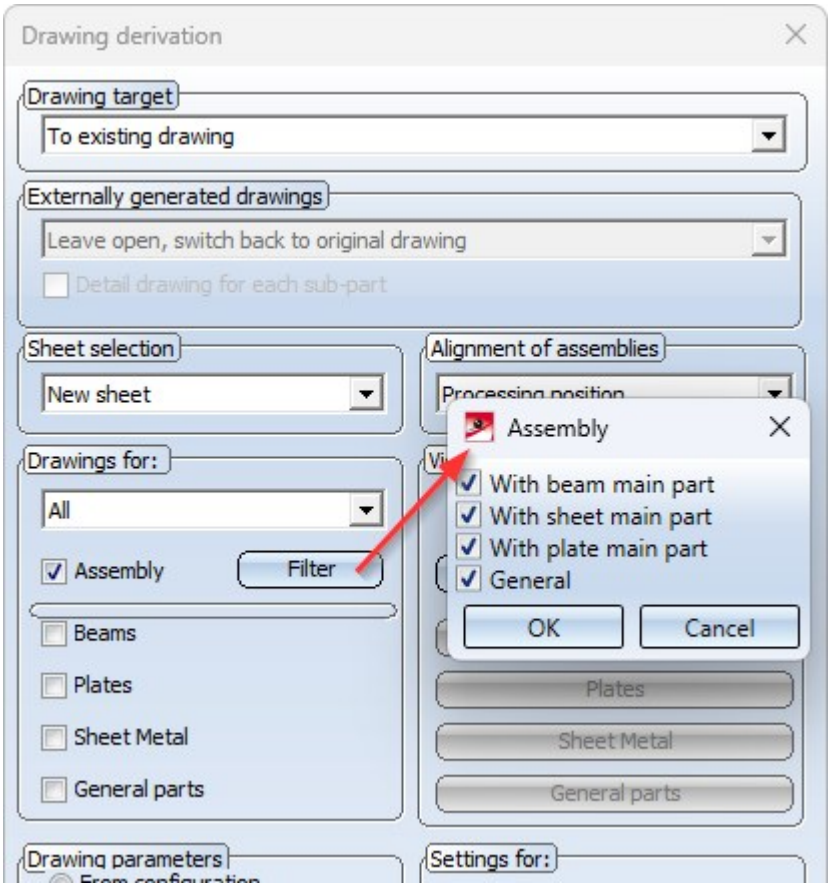
The SpaceMouse connection in HiCAD is now officially certified by 3Dconnexion!

Drawing derivation: Simplified dialogue

The area **View groups to be created for:** has been removed from the Drawing derivation dialogue. Main parts that do not belong to an assembly, i.e. parts at the top plane of the part structure, are now included in the production drawing if the corresponding checkbox is active in the dialogue window under **Drawings for:**.



Filters can now be set for assemblies to limit the number of assemblies to be derived.



2-D

Service Pack 2

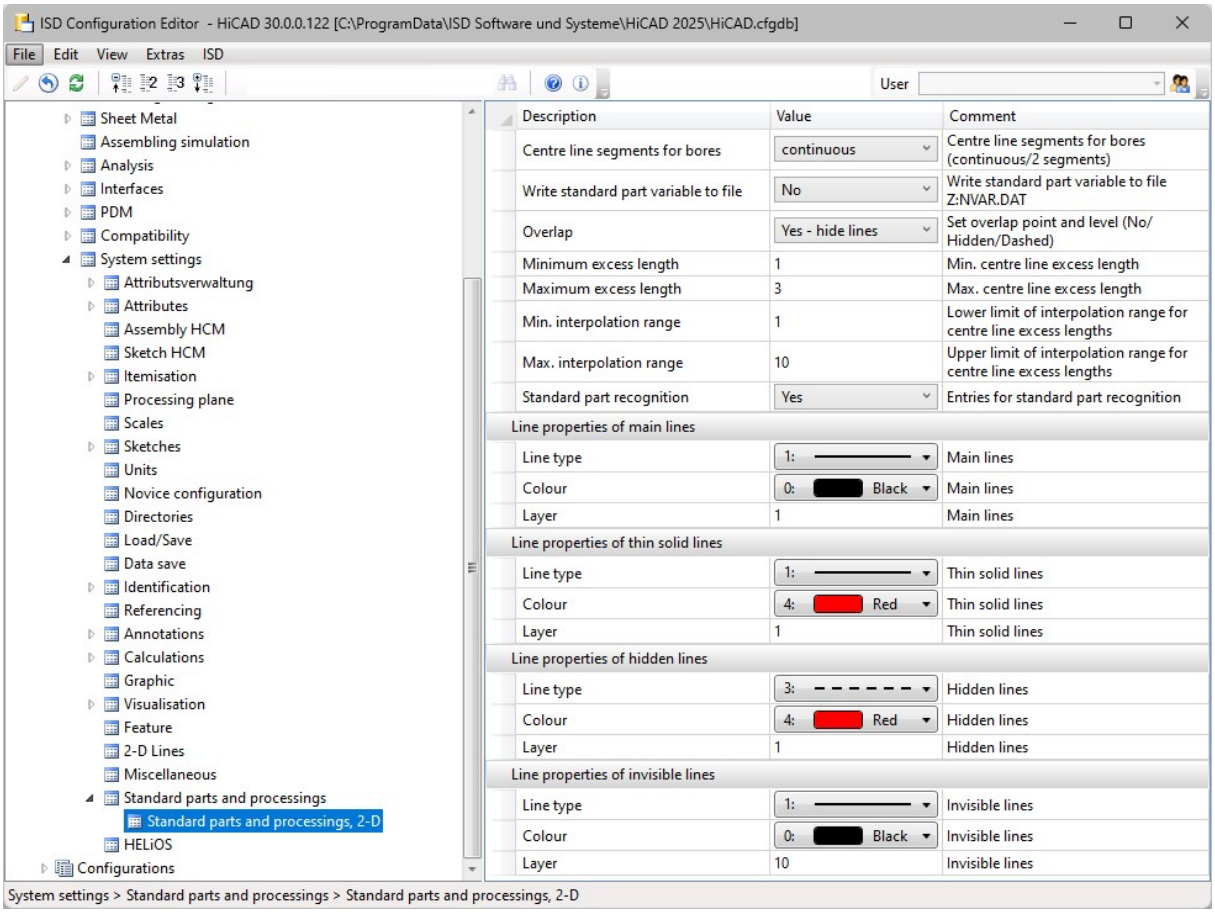
Delete individual graphical elements

The context menu for the 2-D auxiliary lines has been extended to include the **Delete graphical elements, Individual** function.

Major Release

Settings for 2-D standard parts and standard processings

The settings for 2-D standard parts and standard processing have been moved in the Configuration Editor to the path **System settings > Standard parts and processings > Standard parts and processings, 2-D**.



3-D

Service Pack 2

Grid annotation

The functions for creating and editing grid annotation have been revised:

New features in the **Create grid annotation** function:

When you activate the function and fill in the dialogue window, a preview of the annotation is automatically displayed. If you cancel the function, the changes disappear. The grid annotation lines are only created along parts. If parts are outside the grid, the annotation lines are lengthened so that all parts are within the annotation.

Two new options have been added:

- **Do not create larger than grid**: If this checkbox is selected, the annotation lines are not lengthened to reach parts that lie outside the grid.
- **Annotate entire grid**: Grid lines that are not on parts are also annotated.

Magnetic snap-in has been introduced when moving the unannotated ends of lines.

New features in the **Edit grid annotation** dialogue:

The favourite settings have been revised: While favourites can be saved and loaded using the **Create grid annotation** function, they can only be loaded using **Edit grid annotation** dialogue. When editing, only the settings for the **Annotation text**, the **Representation parameters** and the **Interrupt annotation line** option are applied.

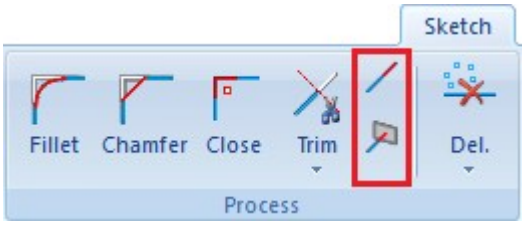
The **Representation parameters**, **Adopt display parameters from reference annotation** and **Favourites** symbols are only active if at least one line is selected.

New features in the Grid annotation context menu:

The context menu has been expanded to include the **Delete, Multiple selection** function. The **Delete, Individual** function is automatically terminated after the first line has been deleted.

Optimisation of some icons in the sketch area

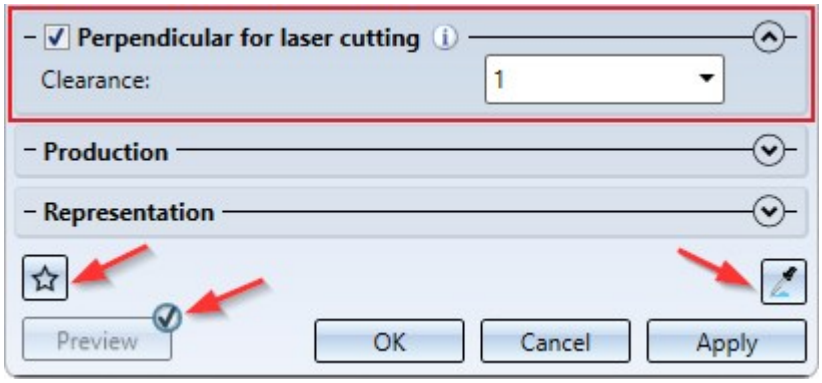
The small icons from the **Sketch** > **Process** ribbon have been slightly changed:



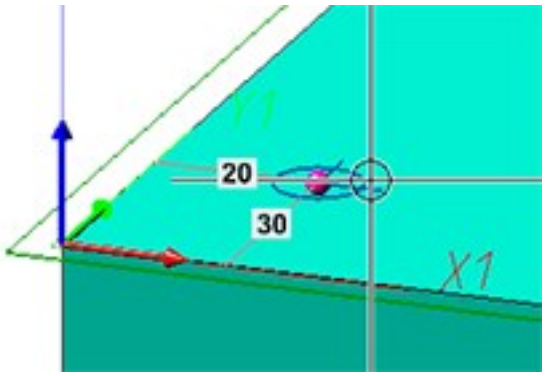
Further development of the standard processing function

The function **3-D Standard** > **Standard processings** > **Bores, Countersinks, Threads**  has been reworked:

New features in the dialogue:



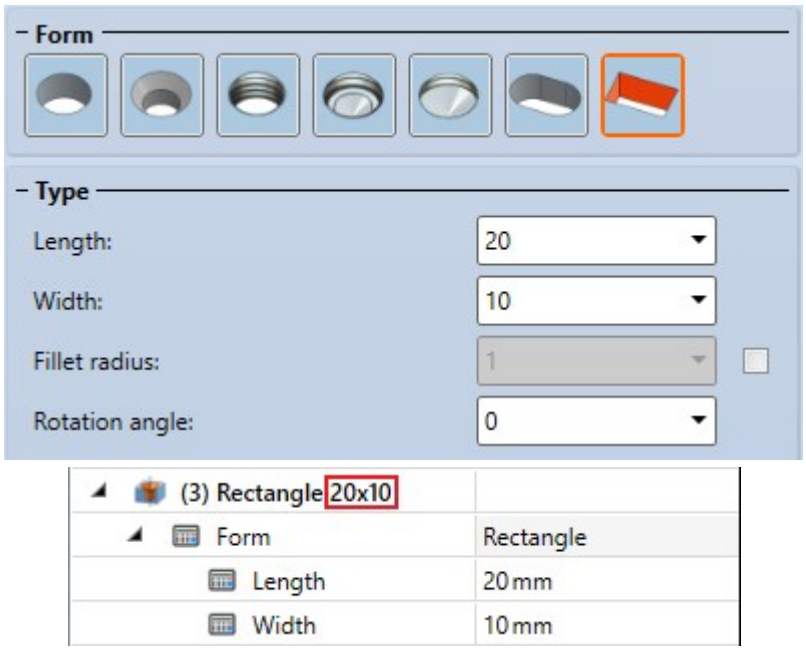
- The menu in the **Perpendicular for laser cutting** now has the same structure for all available forms (**Bore**, **Rectangle** and **Slot**).
- At the end of the dialogue, the star symbol for saving settings as favourites and the pipette symbol for transferring standard processing already created from the drawing have been added. Wherever possible, the settings of old standard processing can also be picked up. In the input fields, it is now possible to save values from the history as favourites.
- When selecting an installation point, the coordinates are now displayed in a grid relative to the processing plane.



- The new checkbox above the preview button can be used to disable automatic preview generation.

Changes to the forms:

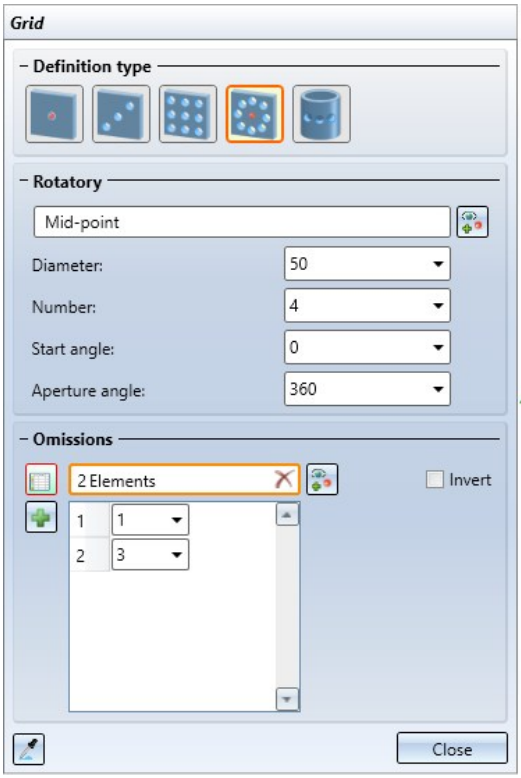
- A checkbox has been added next to the **Fillet** field in the **Rectangle form**. If the checkbox is deactivated, the rectangle is inserted without filleted corners, which was previously only possible by specifying the **Fillet radius0**. In the feature log, the size of the rectangle form is now displayed as *Length x Width* instead of *Width x Length*. This setting has also been applied to rectangle bores created using the old **Rectangle** function.

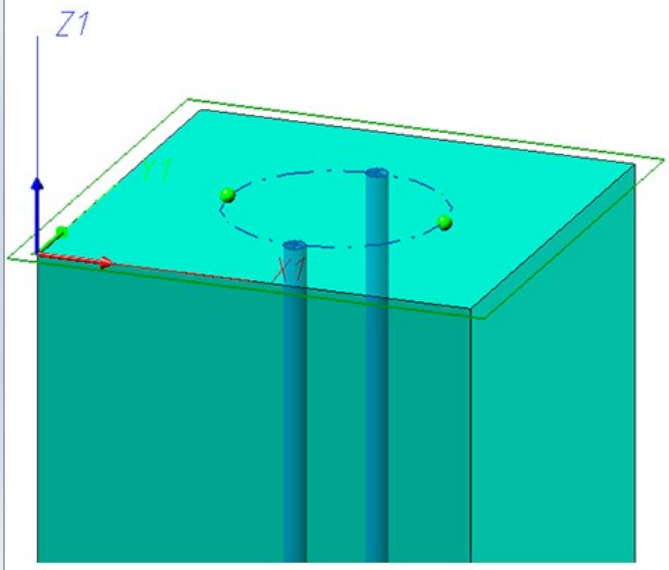


- For **Counter bores**, you can now select from the catalogue from the tables **Cone countersink** and **DIN 74 AF**. Please note that the **Head distance** for counter bores of this type is specified in the tables and cannot be set manually. In addition, a value for the **Bore depth** and an **End form** can now be specified for counter bores, making it possible to create a blind hole with countersink. When selecting the bore depths **Through** and **Next exit surface**, you can now also insert a **Countersink at outlet**.
- The checkbox **Use hole width as total length** has been added to **Slots**.
- Countersinks can now also be inserted at the **Entry and Stair end points** for the **Bore**. To do this, activate the corresponding checkbox and specify the **Countersink angle** and **Countersink depth**. For the **With drilling depth** option, only one countersink is possible at the entry point.

New features of the grid:

- The default setting for the number of standard processing operations for grids of the definition type **Rotatory** and **Radial** has been increased from **3** to **4**.
- A pipette icon has been added to the grid dialogue for transferring grid settings from existing standard processing.
- For the grids, you can now create a list of exceptions that should not be processed. The areas to be excluded can be selected directly in the preview graphic.





Transfer of reference settings to attribute management

The referencing settings for synchronising individual attributes have been moved from **System settings** > **Referencing** to **System settings** > **Attribute management** > **Attributes** in the Configuration Editor. This has the advantage that when adding new attributes, you no longer need to configure them in two different places. The table that was previously found under the **Synchronization of attributes** parameter has been removed and the **Referencing** column has been added to the attribute management table. Double-click to change the referencing setting for each individual attribute. The following options are available:

- **Always transfer** (default setting): Changes are transferred to all referenced main and sub-parts.
- **Transfer for sub-parts**: Changes are only transferred to sub-parts.
- **Standard behaviour**: The settings from the Configuration Editor under **System settings** > **Referencing** > **Default behaviour for transferring attributes for referenced parts** are used. This parameter was previously called **Synchronize item numbers / part attributes when updating file** and has been renamed for clarity.
- **Do not transfer**: Changes will never be transferred.

Short text	Long text	ID	Data type	Assignment	Referencing	Permission	Group	Description
Angle bottom/right - YZ	Angle bottom/right - YZ	§08	Floating point number	Part attribute	Standard behaviour	System	Beam	
Aperture angle	Aperture angle	P_BW	Floating point number	Part attribute	Always transfer	System	Beam	
Commercial weight	Commercial weight	§18	Floating point number	Part attribute	Standard behaviour	System	Beam	
Commercial weight by length	Commercial weight by length	§CBL	Floating point number	Part attribute	Always transfer	System	Beam	
Commercial weight per length	Commercial weight per length	§19	Floating point number	Part attribute	Standard behaviour	System	Beam	
Cross-section area of web in cm^2	Cross-section area of web in cm^2	ASTEG	Floating point number	Part attribute	Always transfer	System	Beam	
Curve radius about y	Curve radius about y	P_BRY	Floating point number	Part attribute	Always transfer	System	Beam	
Curve radius about z	Curve radius about z	P_BRZ	Floating point number	Part attribute	Always transfer	System	Beam	
Flat length	Flat length	§PSL	Floating point number	Part attribute	Always transfer	System	Beam	
IY	Moment of inertia IY	IY	Floating point number	Part attribute	Always transfer	System	Beam	
IZ	Moment of inertia IZ	IZ	Floating point number	Part attribute	Always transfer	System	Beam	
Shipped length	Shipped length	§25	Floating point number	Part attribute	Always transfer	System	Beam	
Surface area by length	Surface area by length	§SBL	Floating point number	Part attribute	Always transfer	System	Beam	
Surface per length	Surface per length	§17	Floating point number	Part attribute	Standard behaviour	System	Beam	
Volume by length	Volume by length	§VBL	Floating point number	Part attribute	Always transfer	System	Beam	
Weight by length	Weight by length	§WBL	Floating point number	Part attribute	Always transfer	System	Beam	
Weight per length	Weight per length	§16	Floating point number	Part attribute	Standard behaviour	System	Beam	
WY	Section modulus WY	WY	Floating point number	Part attribute	Always transfer	System	Beam	

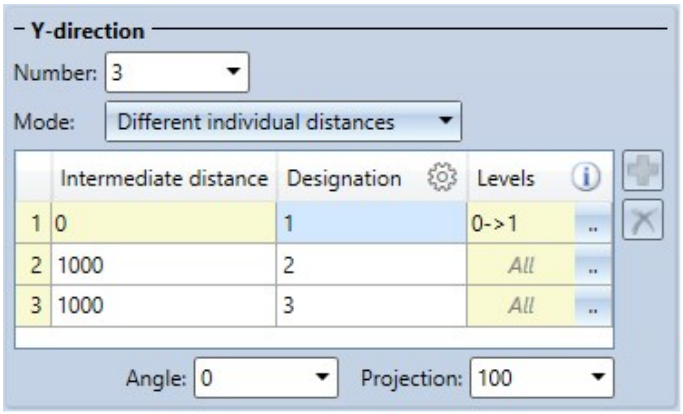
Excerpt from the table under **Attribute management** > **Attributes** with the ISD-side default settings for referencing

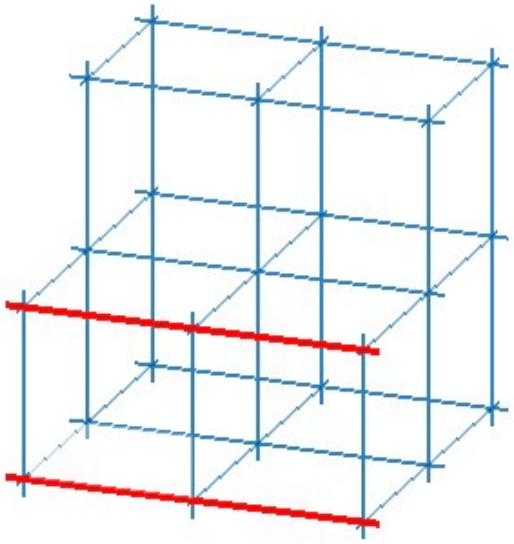
The change removes the **Active** and **Comment** columns from the table. Attributes that were set to inactive in the old table are transferred to attribute management with the **Always transfer** setting. The same applies to attributes that do not appear in the list. When creating a new attribute, the default setting is **Always transfer**. No changes to the referencing setting are possible for attributes with **Internal** permission.

Optimisations to the planning grid

The **Planning grid**  has been optimised in some areas:

- To enable faster editing, the **Edit grid** function has been added to the context menu of the planning grid. This opens the **Planning grid** function dialogue.
- Axes and planes that are clicked on in the table in the function dialogue for editing are marked in red in the graphic.
- If an axis is not displayed on all planes, all intersecting axes are shortened accordingly so that they do not run into empty space.





In the upper image, for example, the Y-axes labelled **1** are selected in the table and the lines in the graphic are marked red accordingly. Since the Y-axis **1** is hidden on the top plane, the X-axes in this plane are also shortened.

Views

Change scale of view with multiple selection

When you change the scale of multiple views simultaneously using the **Change scale of view** you will no longer be asked for a fixed point after confirming the scale. The scale is changed immediately. This new feature ensures that alignments, flushes and equal distances are maintained.

However, if a view has a view fixed point, it will continue to be used as before.

Undo for changing the scale

The **Change view scale** unction is now reversed by the **Undo** function.

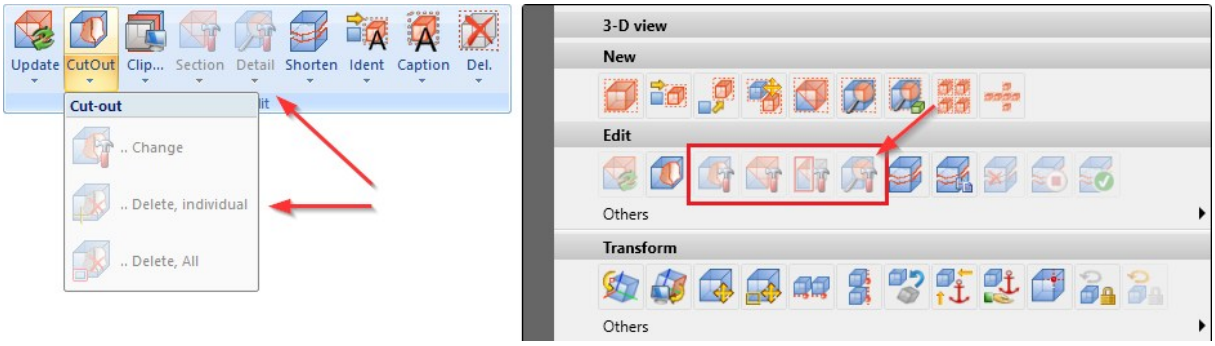
In addition, the **Undo last view transformation** has been renamed to **Last view rotation back**.

There are two scenarios in which the **Last view rotation back** function does not reset the view to its original position.

- Scenario 1: Rotate view > **Change view scale** > **Last view rotation back** > **Change view scale** (return to original scale).
- Scenario 2: Rotate view > **Change view scale** > **Change view scale** (return to original scale) > **Last view rotation back**

Greying out of view functions

If a view is active that is not a sectional view or a detail or cut-out, the functions **Change sectional view**, **Change detail view** and **Change cut-out** are deactivated and shown in grey. This occurs in the ribbon and in the context menu for views.



Optimisation of fillet preview

After you started the **Fillet** function and selected an edge, an arrow was displayed so far. This arrow has affected the overview of the preview. The arrow is only displayed for variable rounds and at the start or end point.

Preview of KRA files

When a large number of referenced KRA files are stored, a large part of the waiting time is taken up by the generation of thumbnails. To speed up the display of thumbnails for KRA files, only one view is generated and reused. The view is only generated when the result is needed. This means that no effort is required if no KRA file is stored.

Drawing in a new view for the first time has also been accelerated. This applies not only to the generation of preview images, but in general.

Activate BOM-relevance

When you start HiCAD for the first time, the **BOM relevant** ☑ option is now activated for the part generation functions in the **3-D Standard**

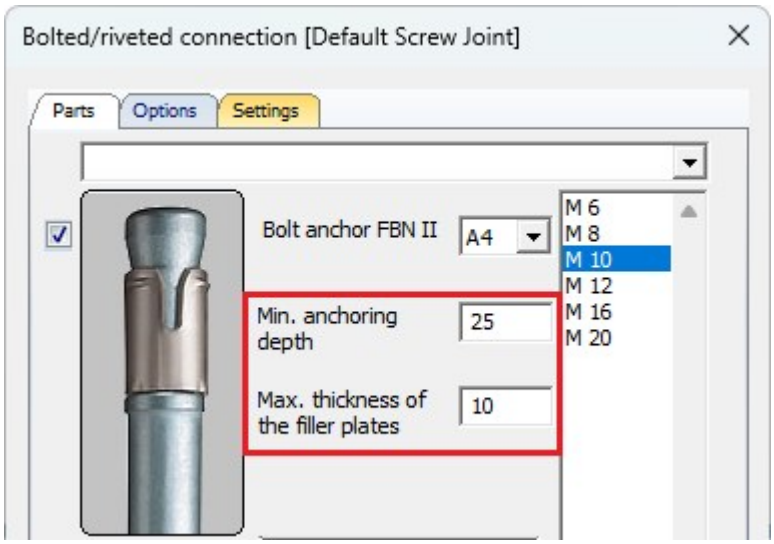
- **Assembly**
- **Solid primitive**
- **Extruded solid**
- **Revolved solid**
- **C-edge sweep**
- **Part from 3-D sketch**

menu bar. If you change this setting in the individual functions, the changed setting is retained even after you restart HiCAD.

The default state can be restored using HicadGUIReset.exe.

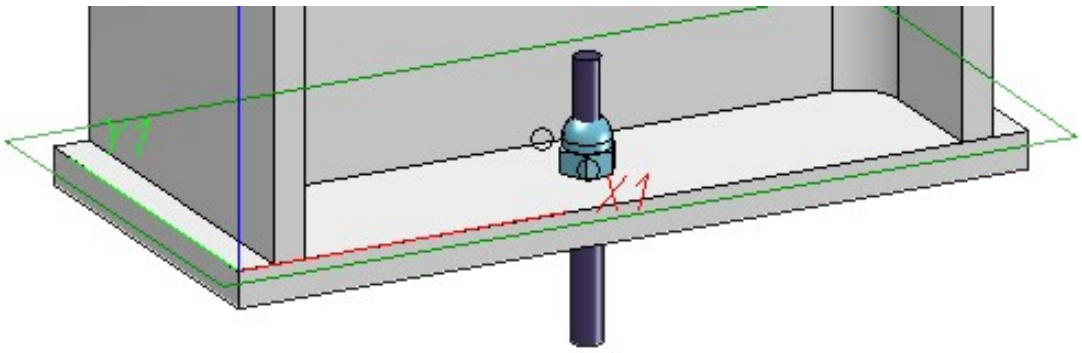
Installation depth can be selected for anchors in boltings

In the **New bolting/riveting** function, the minimum anchoring depth and the maximum thickness of the filler plates must now be specified for the anchors. This is because various clamping lengths of the filler plates and various anchoring depths are available for some anchors.



The new method for inserting anchors also fixes the following error:

As shown in the image below, the Fischer Anchor FAZ II was previously inserted into the drawing in combination with a cap nut so that the anchor protruded from the cap nut. By entering a **Min. anchoring depth** of 0 or greater, the anchor remains below the cap nut.



Boltings: Parts from the catalogue Purchased / Factory standard parts

From now on, you can access fasteners that were previously saved as parts in the **Factory standards**> **Purchased / Factory standard parts** catalogue in the **New bolting/riveting**  function dialogue window.

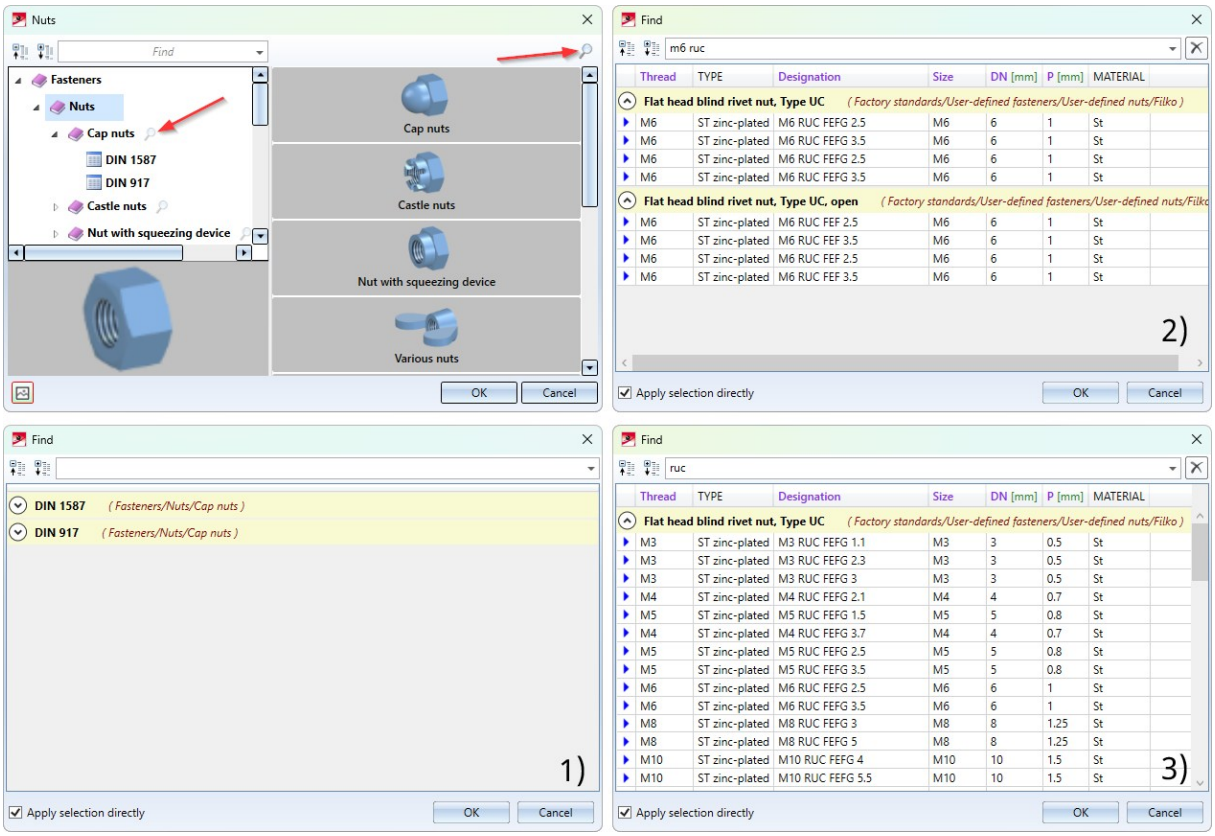
Filleting and chamfering: Value input

In the **Fillet** and **Chamfer** functions, you can now specify the radius or length before selecting an edge.

Search in the catalogue selection

The catalogues have been expanded to include the **Find** function  . After clicking on the icon, you can use the search bar to search for exact names or names that contain the specified search term. The standard parts are sorted into groups according to their name in the **Find** dialogue window.

You can now also find a search icon behind some of the folders in the catalogues. Clicking on the icon limits the search to the selected folder.

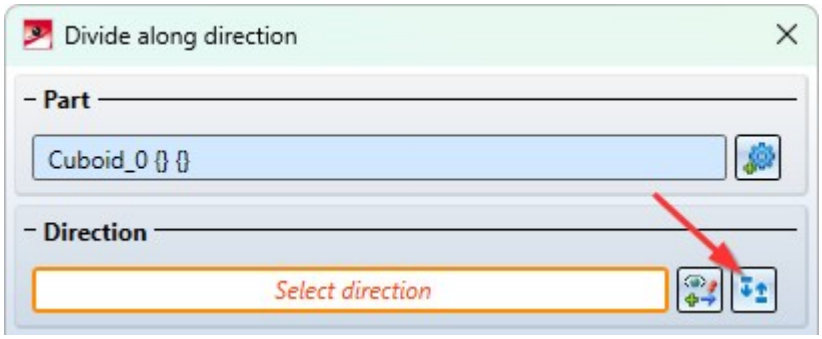


1) Search in selected folder 2) Exact search term 3) Included search term

Alternatively, the search can also be accessed using the key combination **CTRL + F**, provided that a catalogue is open.

Divide along direction

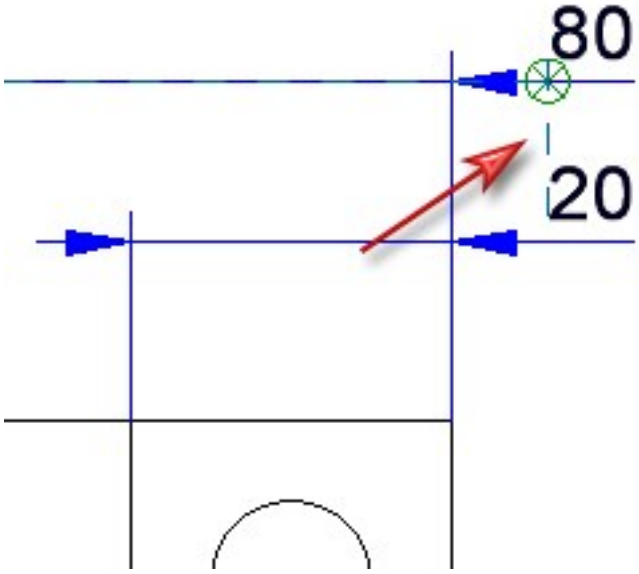
In the **Divide along direction** function, **Reverse processing direction** has been changed from a checkbox to a button.



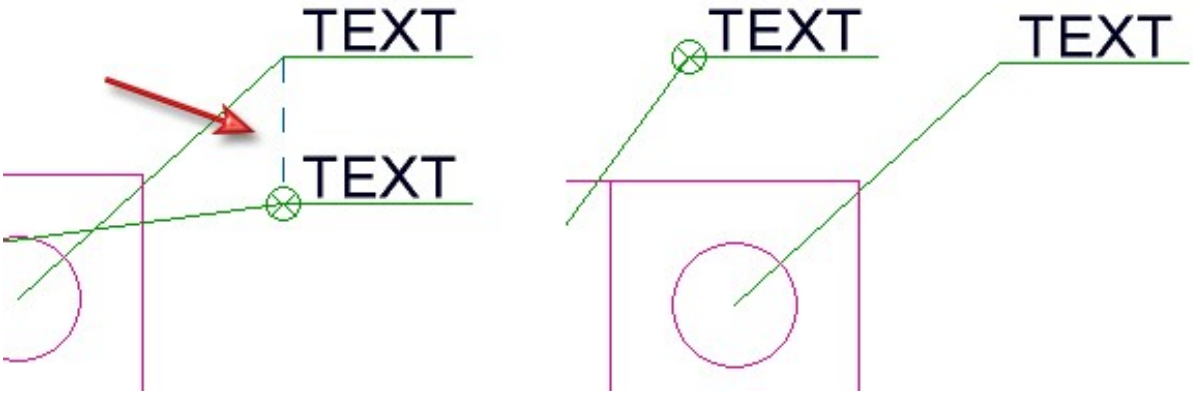
Dimensioning

Magnetic snap-in dimensioning and annotation

When **Dimensioning** the dimension figure can be pulled out to the side. If another dimension figure is pulled out at a different height, it can be aligned with the first dimension figure by magnetic snap-in using an auxiliary line. After the dimension figure has been itemised, the auxiliary line disappears.



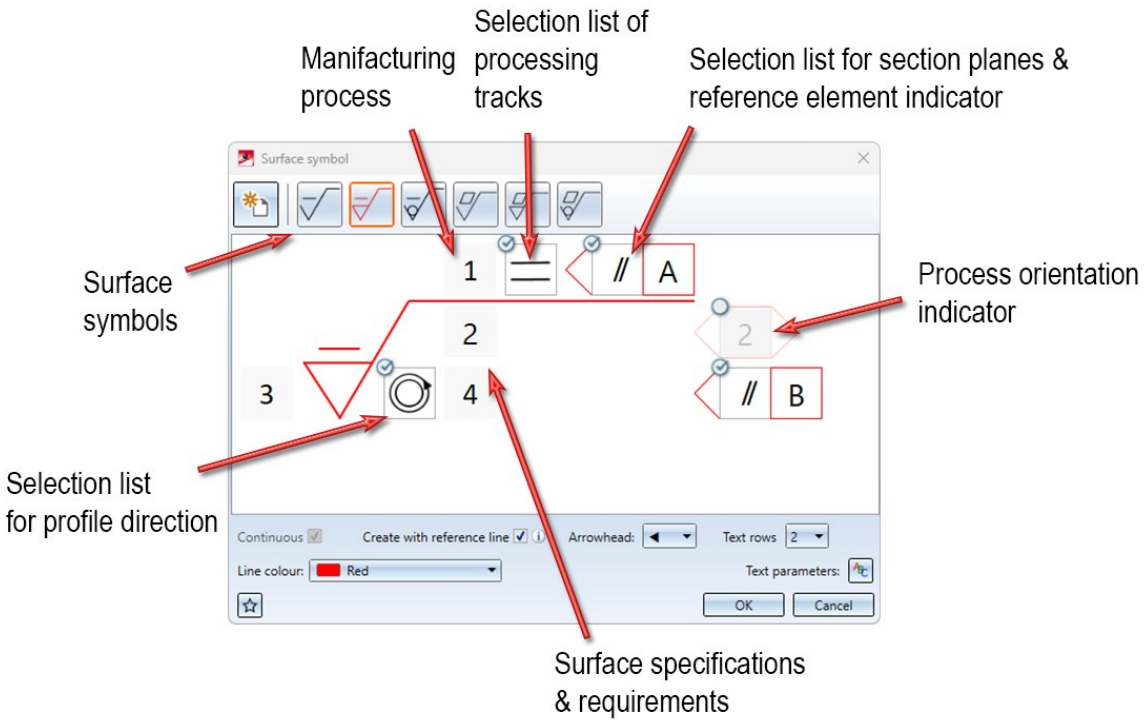
When dragging **Annotations** , you can select a line for the magnetic direction by clicking with the left mouse button. If a horizontal line is selected, the annotation will only be aligned horizontally with the other annotations. However, if a vertical line is selected, the annotation can only be aligned vertically with the other annotations.



Please note:
This functionality is only intended to simplify alignment and is not associative. This means that if, for example, the dimensioning is changed, the aligned dimensions are not updated.

New dialogue for surface finish

The dialogue for the **Surface finish** function has been completely revised for DIN EN ISO 21920.

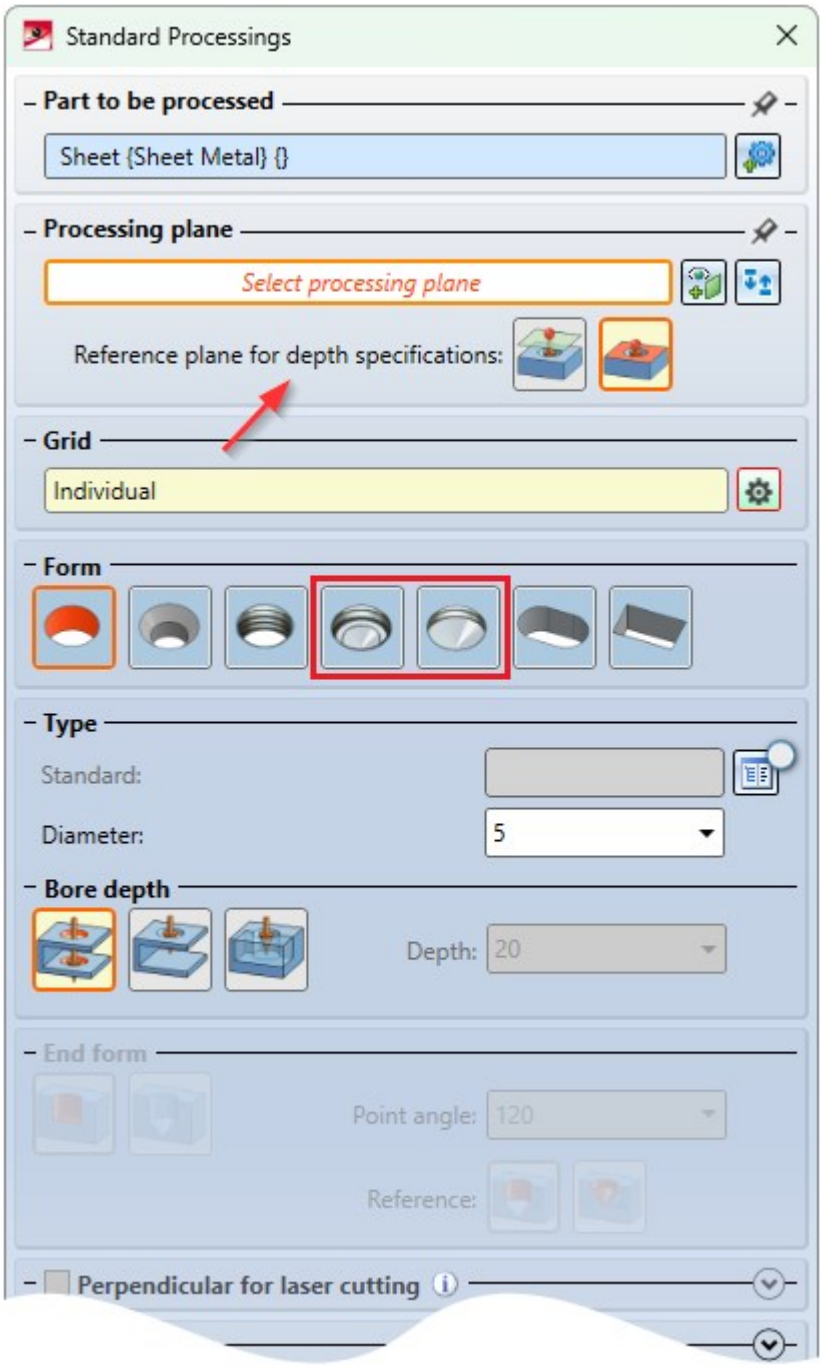


The previous **Surface finish** is still available in the menu under **3-D Dimensioning + Text > Symbols > Surf...**. If a drawing contains surface specifications that were created with this function, the 'old' dialogue window is displayed for editing these surface specifications.

Service Pack 1

Further development of the new standard processing function

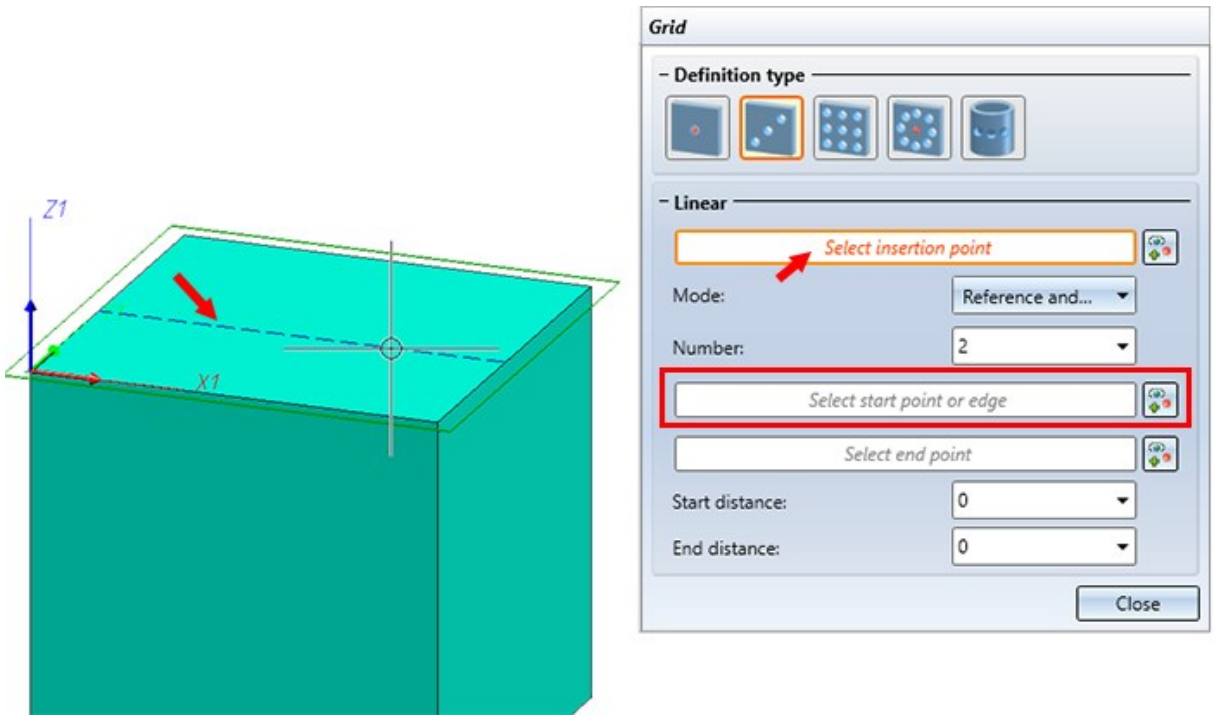
The dialogue window for the function **3-D Standard** > **Standard Processings** > **Bores, Countersinks, Threads**  has been extended.



The two new forms, **Thread undercut** and **Thread runout**, have been added. The **Reference plane for depth specifications** parameter has been moved up to the **Processing plane** area in the dialogue window. The setting made here affects the creation of countersinks, thread undercuts and thread runouts.

The functions for creating linear and rectangular processing grids using the **Reference and Quantity** and **Reference and max. individual distance** modes have also been revised:

When selecting the insertion point, the line along which the processings will be inserted is displayed in the preview. It is now also possible to select an edge instead of a starting point for the insertion.



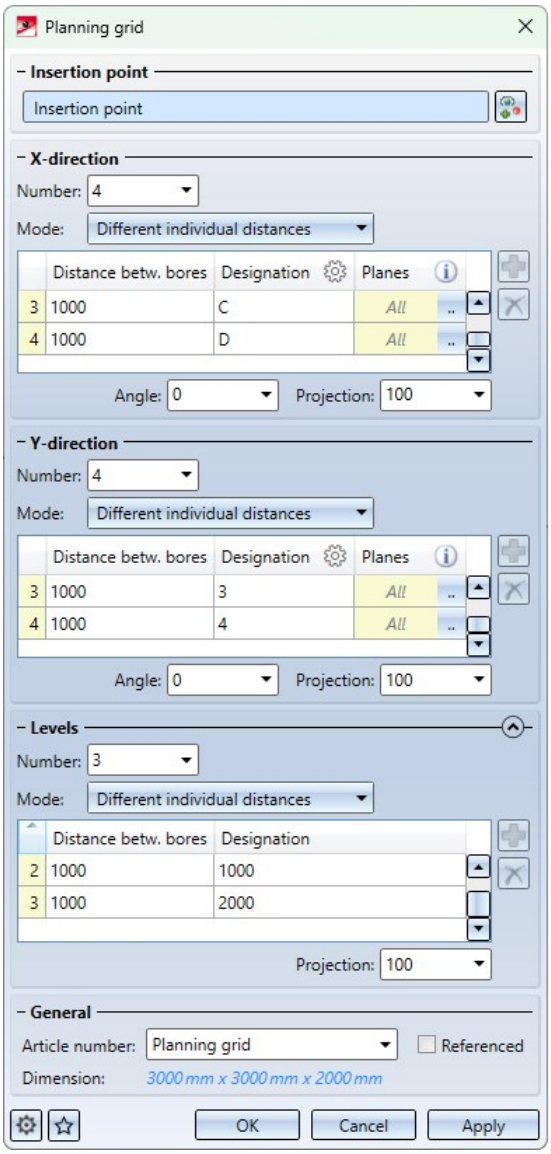
Preview line when selecting a point for a linear grid in the **Reference and Quantity** mode.

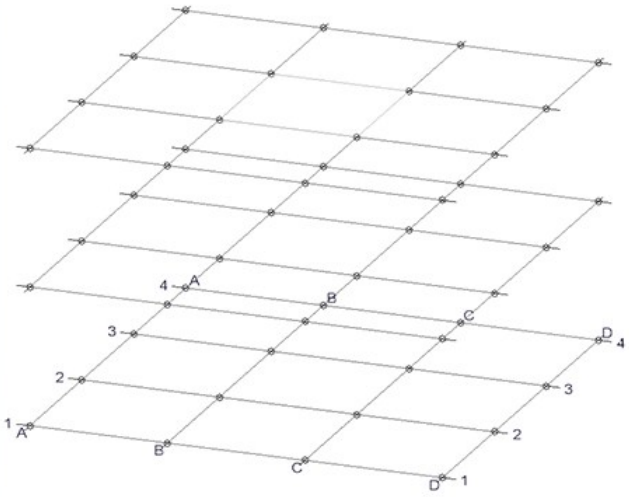
Since most of the functions for creating standard processings have now been integrated into the dialogue of the **Bores, Countersinks, Threads** function, the **Standard Processings** ribbon has been adapted so that the function icon  is in first place here. In the pull-down menu below, you will find the functions **Copy standard processing**  , **Bore pattern**  , **Punch mark**  and **Hole pattern**  . The tab **Compatibility** > **Up to HiCAD 2024** has been added, where you will find the old standard processing functions.

The third icon in the ribbon, the **Variable through hole**  function, has been replaced by the **Cams**  function. The pull-down menu below this now contains the functions **Lettering**  , **Processing direction**  and **Powder marking lines**  .

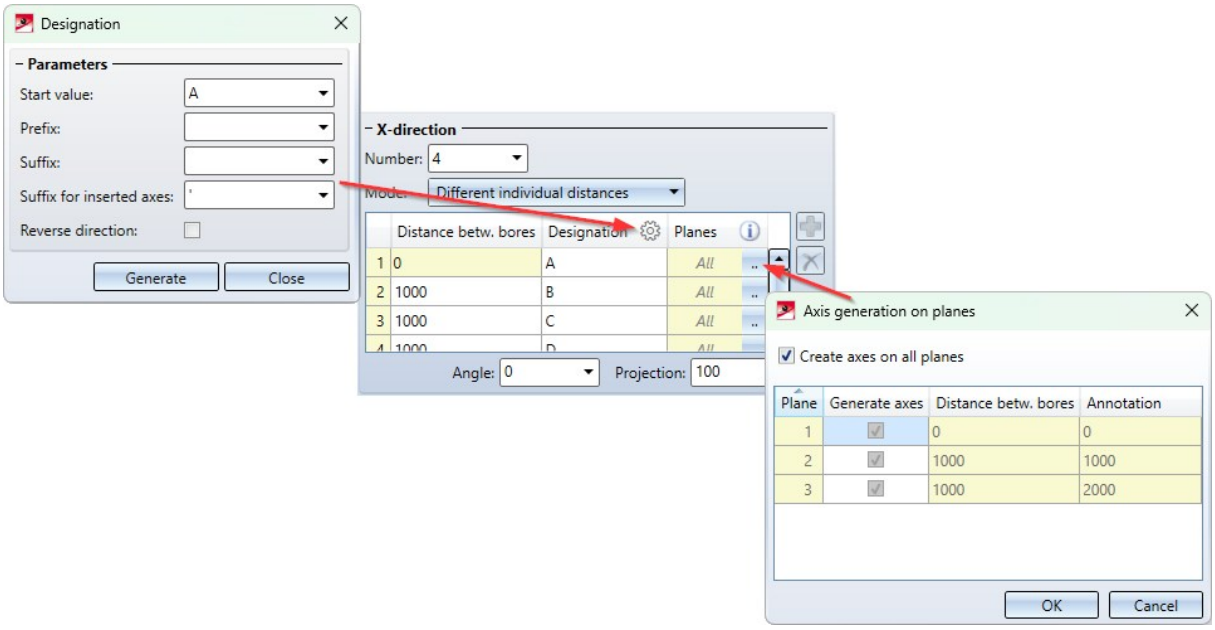
3-D Grid

A new function for creating 3-D grids is now available. You can find the function under **3-D Standard** > **New** > **Others** > **Planning grid**  .





As with the previous grid function, you can create axes in the X and Y direction and in multiple planes. When creating the axes, you can choose from four different modes to create axes at varying or equal distances. The table rows are then described accordingly for further editing of the axes:



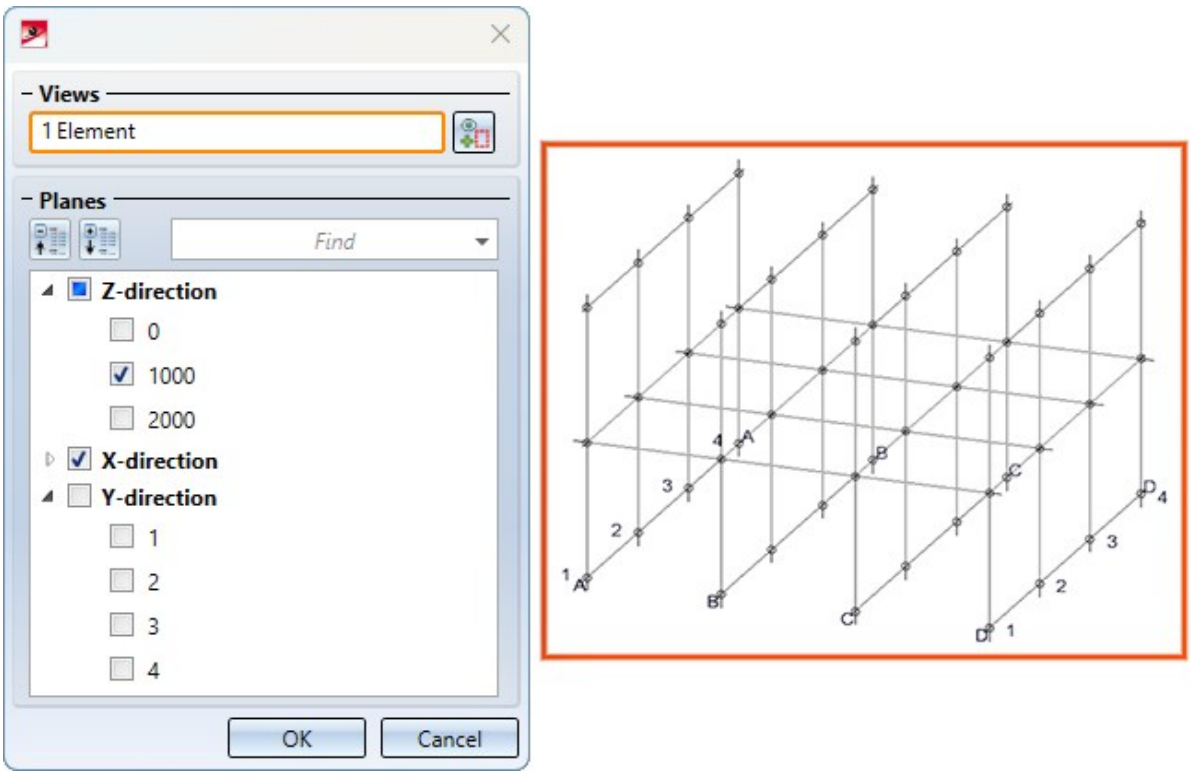
You can use the  and  icons to add new axes or delete individual axes. If the grid consists of more than one plane, you can specify which planes individual axes should or should not be displayed on. The generation of axis annotation has also been updated. In the additional dialogue window, **Designation**, you can assign a **Prefix**, **Postfix** and **Suffix for subsequently inserted axes** in addition to the consecutive axis annotation. In addition, you can specify an angle by which the axes of a direction are rotated in an anti-clockwise direction.

An annotation can also be given to planes in the new dialogue. Like axes, planes can be added or deleted later.

In the lower part of the dialogue, the size of the grid currently being created is displayed in the form of length x width x height. You can also assign an article number to the grid directly here. It is also possible to save the grid as a referenced part. The settings made can be saved as favourites. The dialogue window **Representation parameters** can be opened using the cogwheel. The settings for changing the text and line parameters of the grid and the grid annotation have been summarised here.

The context menu of the axis grid contains the function **Ansichtsweise Ebenensichtbarkeit**. This is where you can define the visibility of the individual axes within the planes. The default setting is as shown in the figure above: *All X- and Y-axes are visible on all created planes, the Z-axes are hidden everywhere.*

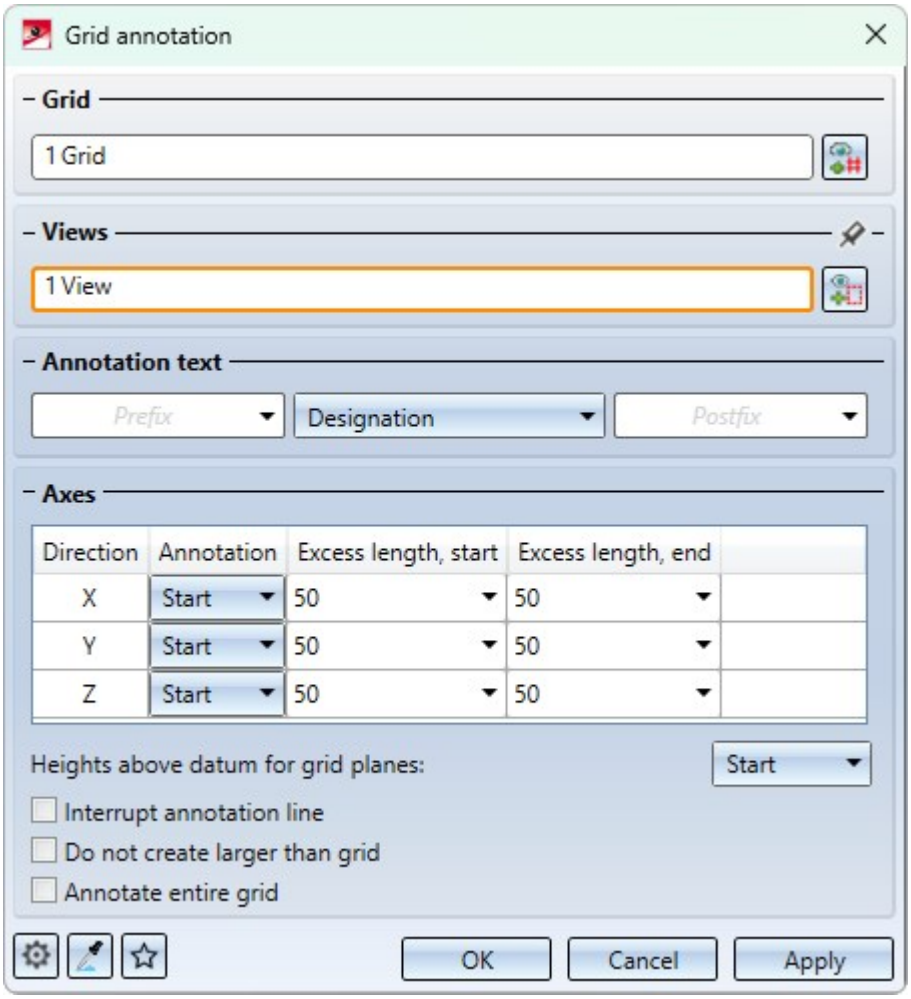
In the example image below, all YZ planes are displayed and the X axes are only shown on the middle Z plane (**1000**):



The option of creating grid sub-systems has not yet been integrated into the new grid function. The old **Steel Engineering grid function** is still available for this purpose. It has been moved from the menu item **Drawing** > **Others** > **Extras** to **3-D Standard** > **New** > **Others**.

Revision of the grid annotation functions

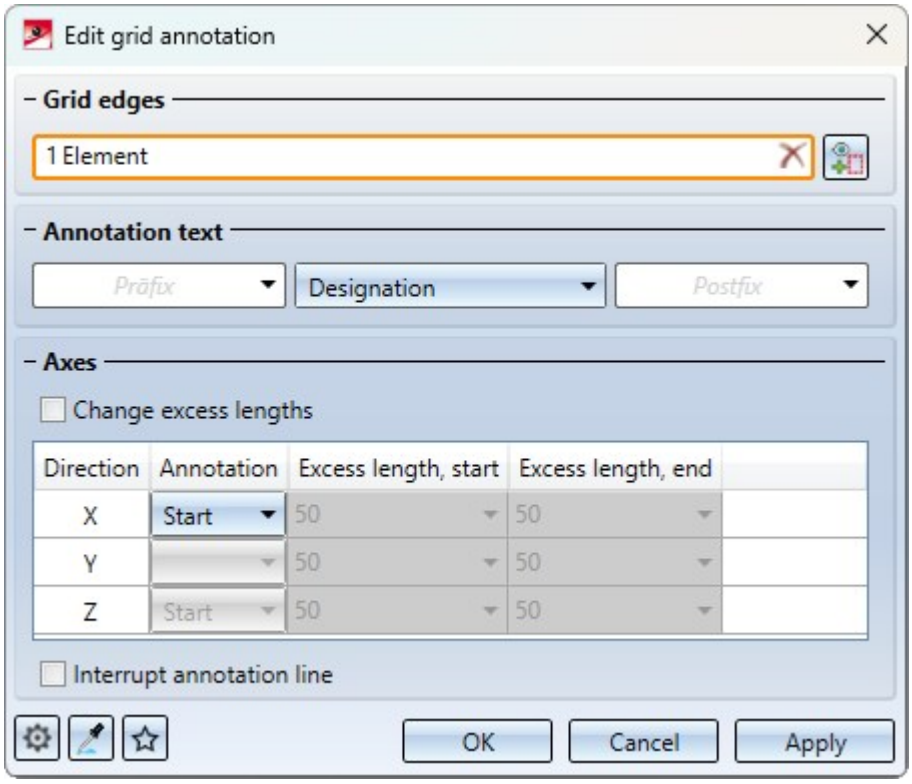
The dialogue for the function **3-D Dimensioning + Text** > **Symbols** > **Create grid annotation** has been completely revised. As part of this, many processing functions from the underlying pull-down menu and the context menu have been combined and revised:



The new dialogue window allows you to select and annotate several grids at once. You can also select several views. The process of creating texts for axis annotation has been simplified and is no longer part of a separate dialogue window. Annotations can now be created at both ends of the axis, not just at the start or end of the axis. Height markers can be created for the grid levels. The checkbox **Interrupt annotation line** can be used to create annotations without lines. The settings made can be saved as favourites. The pipette symbol has been used to integrate the functions from the **Reference parameters** section into the new function dialogue and therefore remove them from the pull-down menu.

If you only want to annotate individual axes within a grid, use the **Create grid annotation, Individual** .

The new function **Edit grid annotation** has been added to the pull-down menu:



The functions from the **Change parameters** section and the function **Align grid annotation** were integrated in the dialogue and therefore removed from the pull-down menu. You can still access the **Edit parameters, individual** function under the tab **Up to HiCAD 2024**.

Use the dialogue to select one or more grid edges that you want to edit. It does not matter whether they belong to the same direction or the same grid. The previous settings of the selected grid lines are included in the function dialogue and can then be changed and accepted. If the selected grid lines have different settings at certain points, this is indicated by a message in the dialogue. A new, uniform setting can then be made for the selected lines.

.

Delete grid annotations

The functions for deleting grid annotation lines have been combined into the **Delete selection of annotation lines** function. This function allows you to select and delete individual lines manually using the left mouse button or to make multiple selections using the right mouse button to delete several lines at once. The multiple selection offers the following options:

- **Delete, Rectangle**
- **Delete, View**
- **Delete, in all visible views**
- **Delete, Drawing**

The context menu for grid annotations contains the functions **Delete, Individual** and **Delete, Total**.

New function for creating a structure assembly

The new functions **Create structure assembly** and **Create structure assembly as sub-part** have been added to **3-D Standard** > **New** > **Assembly** for creating structure assemblies. In addition, the new functions have been added to the **Context menu for drawings** under **New 3-D part** and to the **Context menu for assemblies** under **New Part** > **Main part** or **New part** > **Sub-part**. The functions behave like the other assembly functions and create an assembly that is not BOM-relevant from the **Part typeStructure assembly**.

Until now, there was no direct function for creating structure assemblies. To mark an assembly as such, the **Part type** had to be set to **Structure assembly** either manually or after creation.

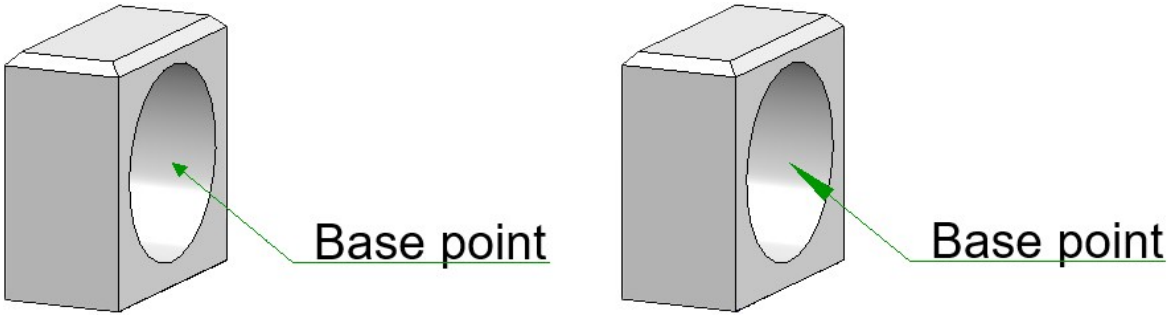
The settings for assemblies in the Configuration Editor were previously divided into the areas **Steel Engineering** > **Assembly** and **Modelling** > **Part creation** > **Assembly** and have now been combined under **Modelling** > **Part creation** > **Assembly**. As part of the new functions for structure assemblies, the settings have been extended to include the item **Article number for structure assemblies**.

Remove references to an active part list

The functions **Break up referencing, Individual+identical parts** and **Break up referencing, Active part+sub-parts** under **Drawing** > **Save/Reference** > **Update referenced identical parts** can be applied not only to an active part or an active assembly, but also to an active part list. To do this, select the desired parts in the ICN while holding down the CTRL key and then right-click to select the desired function for cancelling referencing.

Annotation - base point symbol arrow

For 3-D annotations, different base point symbols can be selected. From SP1, the arrow, arrow (filled) and arrow (open) symbols are always displayed in the screen plane. This means that the symbols remain in the predefined size. Previously, the symbols were spatially distorted, causing the size to vary.

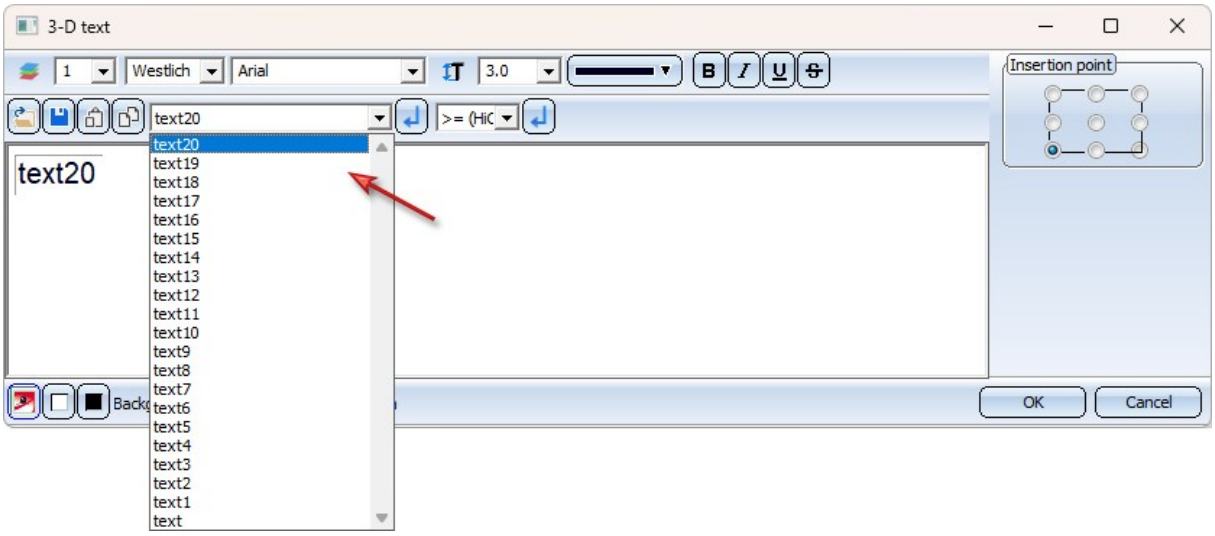


Undo when moving views

Moving a view in the **Sheet area** is stored by the **Undo** function. By executing this function, moving a view can be undone. However, if the perspective of a view is changed, for example with the function **Rotate view dynamically about 3-D point**, the view can only be changed back to the previous view with the function **Undo last view transformation**.

Selection list of the entered texts in the Text Editor

The Text Editor has a list box in which used texts are stored temporarily. It has been increased from 10 entries to 100 entries.

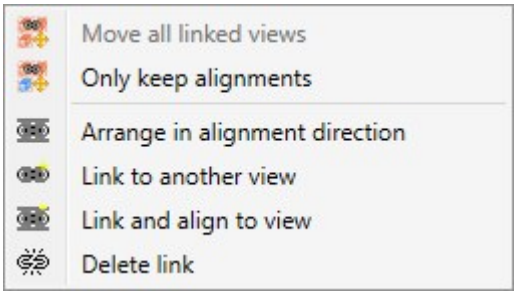


Moving views in alignment

The context menu that opens when you right-click while dragging a view offers two options for how the linked views behave.

The **Move all linked views** function is selected by default. This function moves all views that are linked to the selected view. The second function is **Only keep alignments**. This function only moves the views that are aligned with the selected view.

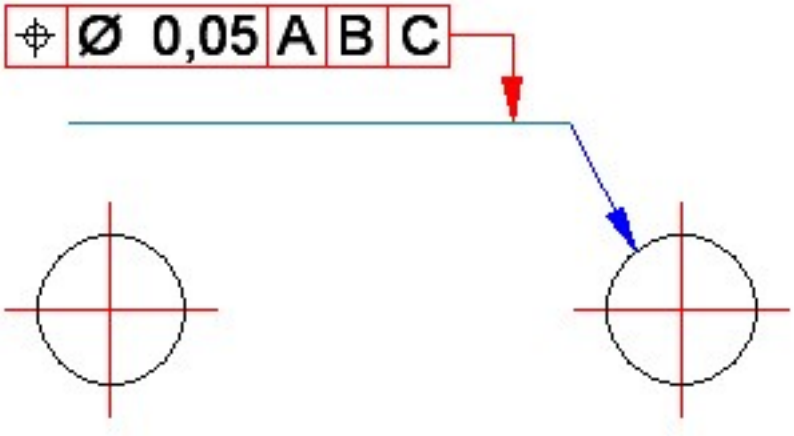
In addition to these two options for the behaviour of the linked views, there are two functions for linking the selected view to another view. The first option, **Link to another view**, links the view to another view without conditions. The second option, **Link and align to view**, allows you to link the view to another view in alignment. For views that are linked to master views, two additional link functions are available.



The first is **Arrange in alignment direction**. This function allows you to move the selected view freely without aligning it. The second function is **Delete link**. This function allows you to delete a link for the selected view.

Auxiliary lines for form and positional tolerances

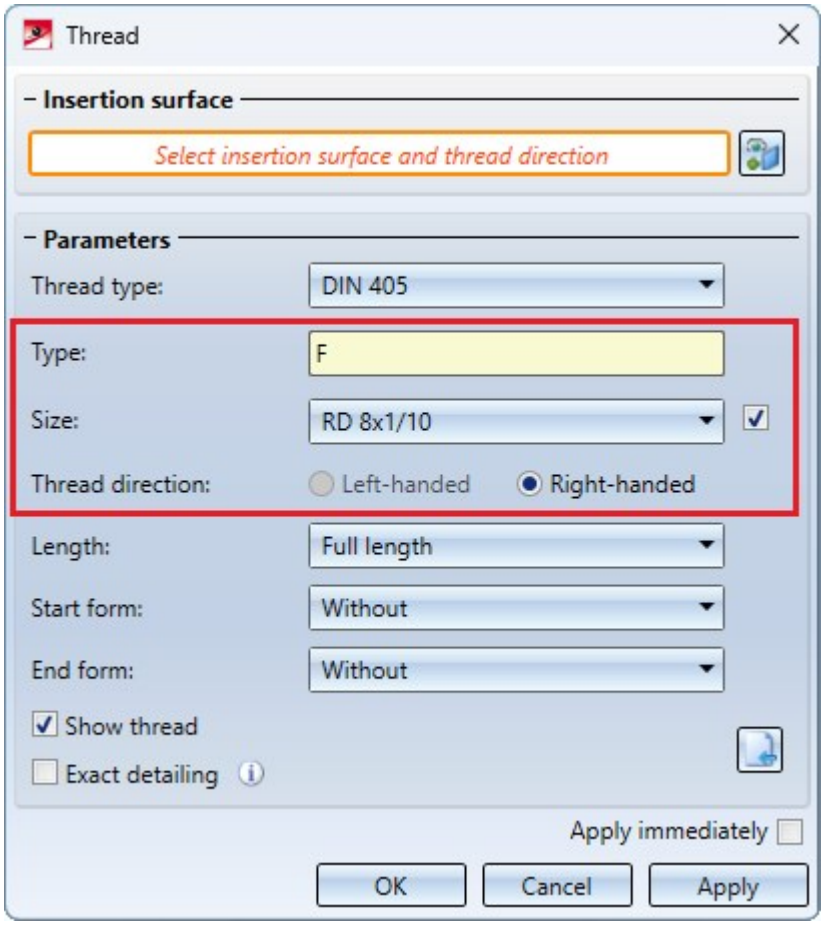
In the **Form/Positional tolerances** function, you can add an auxiliary line for the form and positional tolerance to be created. To do this, activate the **Create with auxiliary line** checkbox.



Major Release

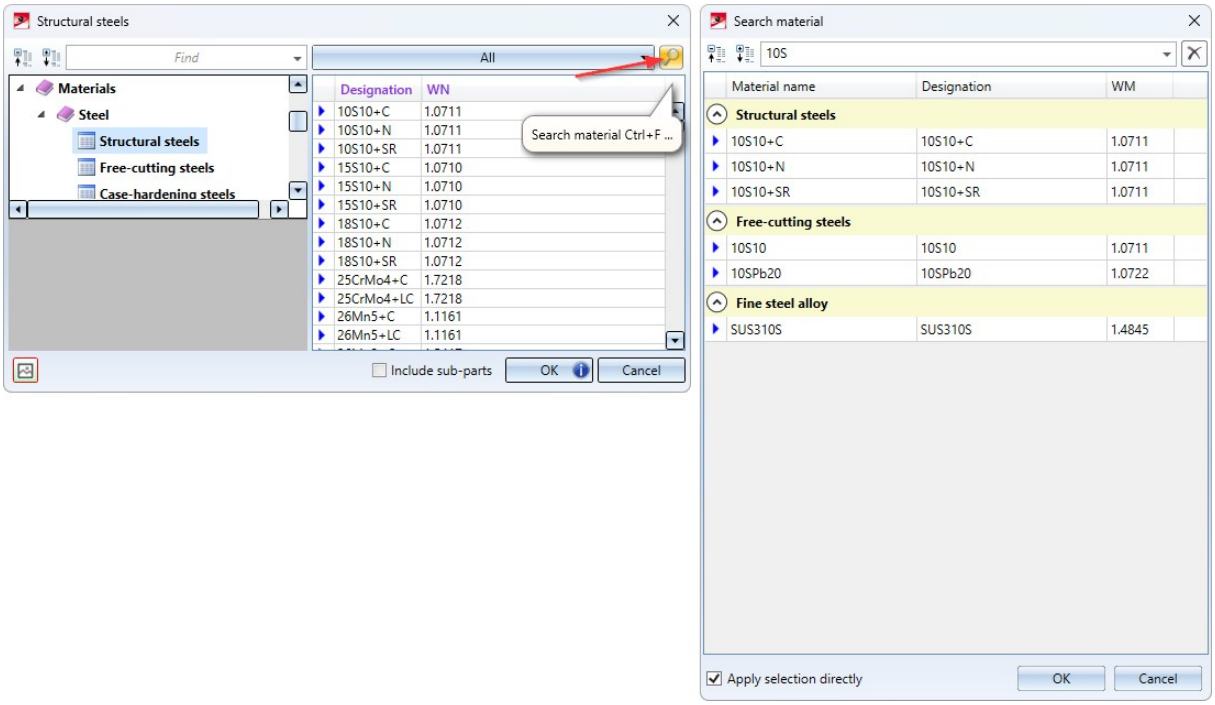
Internal/External threads - Dialogue change

The dialogue of the **Internal/External threads** function has been changed slightly. The term **Orientation** has been replaced by the term **Thread direction**, which is commonly used in practice, and the order of the parameters has been adjusted.



Search material

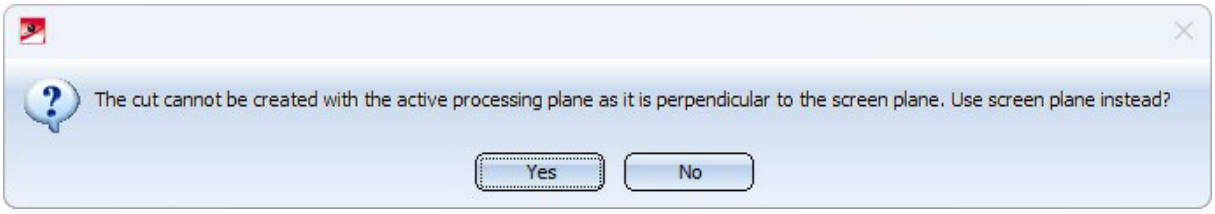
The **Material** function now also supports the **Search material** function. After clicking on the icon, you can use the search bar to search for exact material names or for those that contain the specified search term. The materials are displayed sorted by material group.



Alternatively, the material search can be called up using the key combination **CTRL + F** if the material dialogue is open.

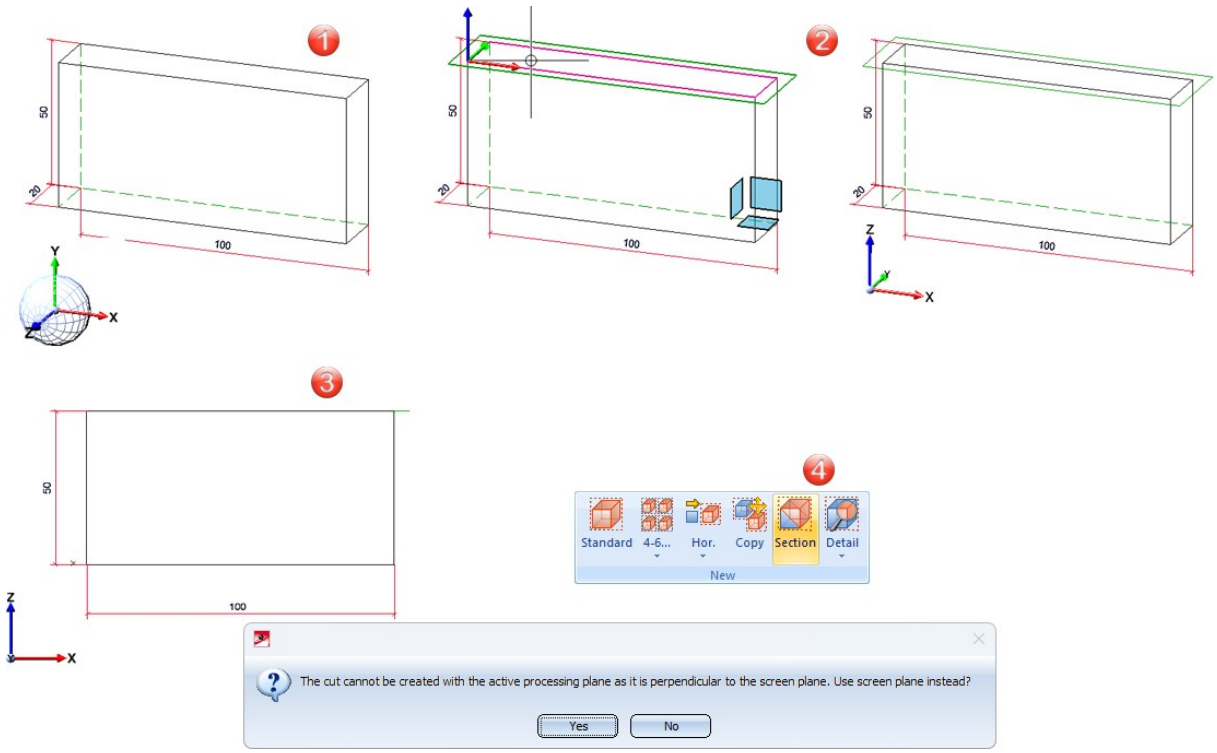
Create sectional view - Processing plane

If you call up the **New sectional view** function with an active processing plane, the processing plane is applied. However, if the processing plane is perpendicular to the screen plane, a message will now appear. This message informs you that the processing plane is perpendicular to the screen plane and that the cut cannot be created. You will then be asked whether the screen plane should be used for the cut instead.



Example:

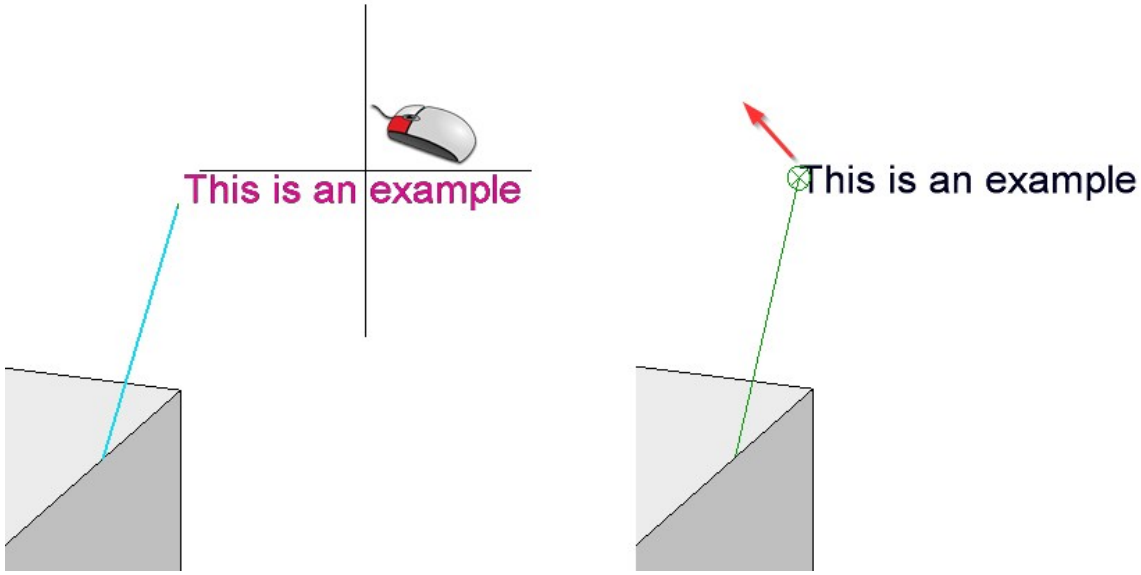
In the drawing (1), a processing plane (2) is first defined and the system switches to the top view (3). If the **New sectional view** function is then called up again (4), the message shown above appears.



Drag & drop of annotations - Reference point

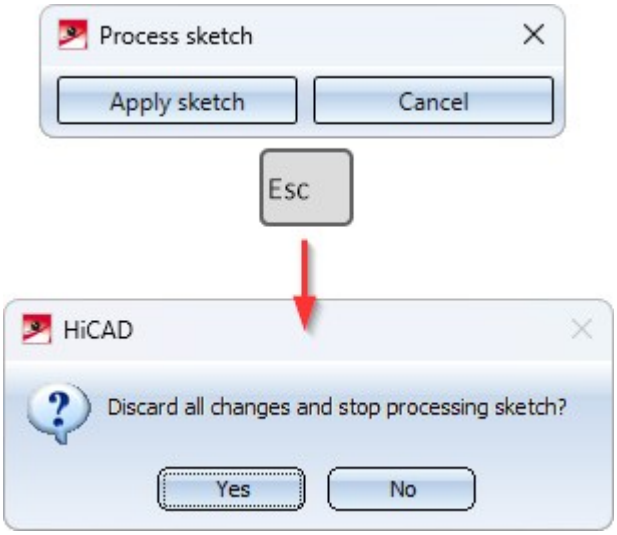
Annotation tags can be easily moved using drag & drop. This applies to both the reference line and the annotation including the leader line. The part that is closer to the cursor is always moved. Previously, the behaviour was such that selected annotations always automatically jumped to the current cursor position. This made it difficult to move the annotation so that the direction was maintained.

From HiCAD 2025, the cursor automatically jumps to the reference point of the annotation instead. This new behavior applies to part annotations, form/positional tolerances, weld seam annotations, 3-D texts and chamfer dimensions.



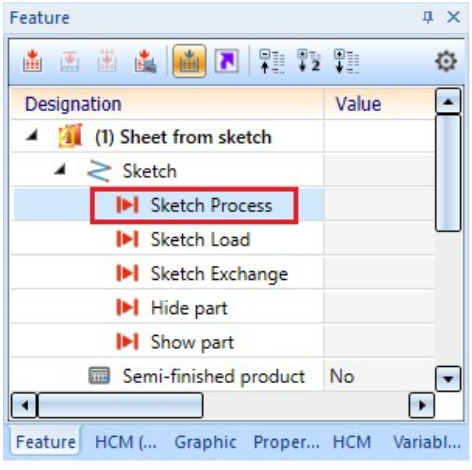
ESC key in **Process Sketch** mode

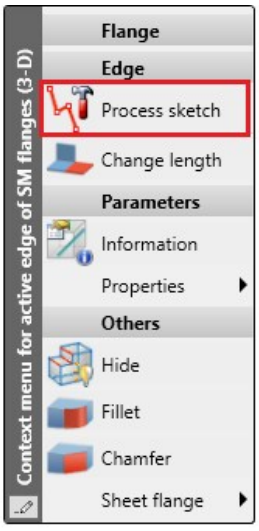
In **Process sketch** mode, the changes were previously discarded and the mode ended when the **ESC** key was pressed. As of HiCAD 2025, a prompt is displayed that offers you the option of not discarding the changes after all. This provides security, for example, if you have accidentally pressed the ESC key because you want to end a sketch function that has already been ended.

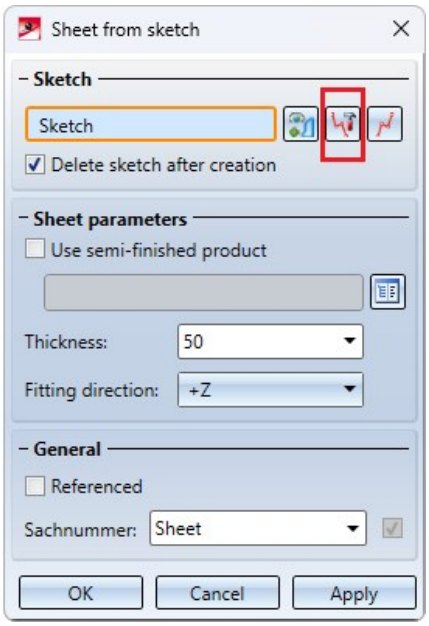


The message appears when the ESC key is pressed in the following cases:

- Calling up **Process sketch** in the feature log,
- Calling up **Process sketch** in the context menu of a geometry and
- Calling up **Process sketch** by clicking on the  symbol in a function dialogue, e.g. **Sheet from sketch**.



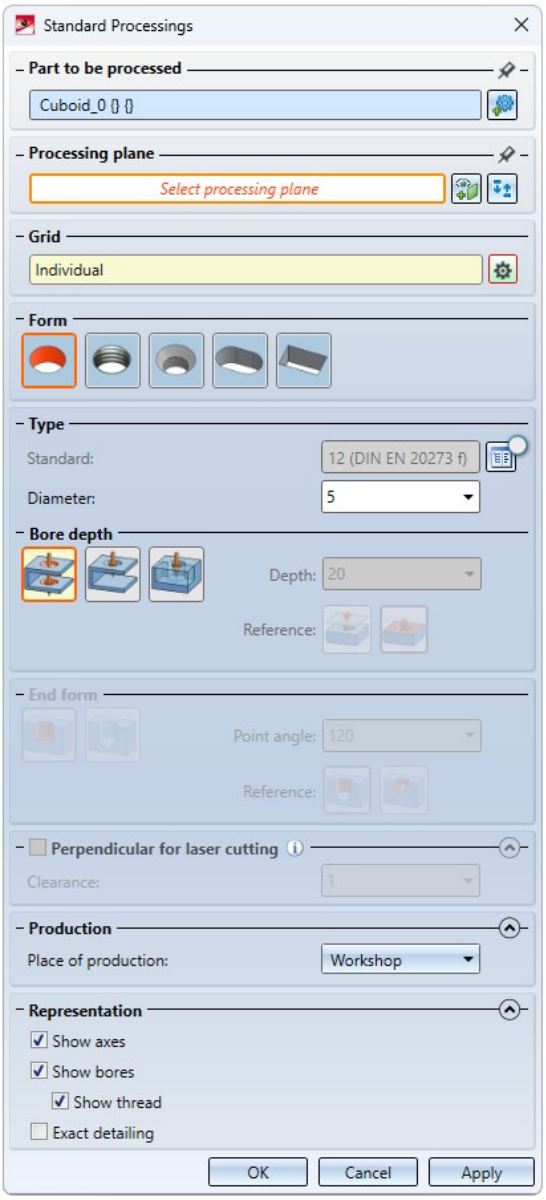


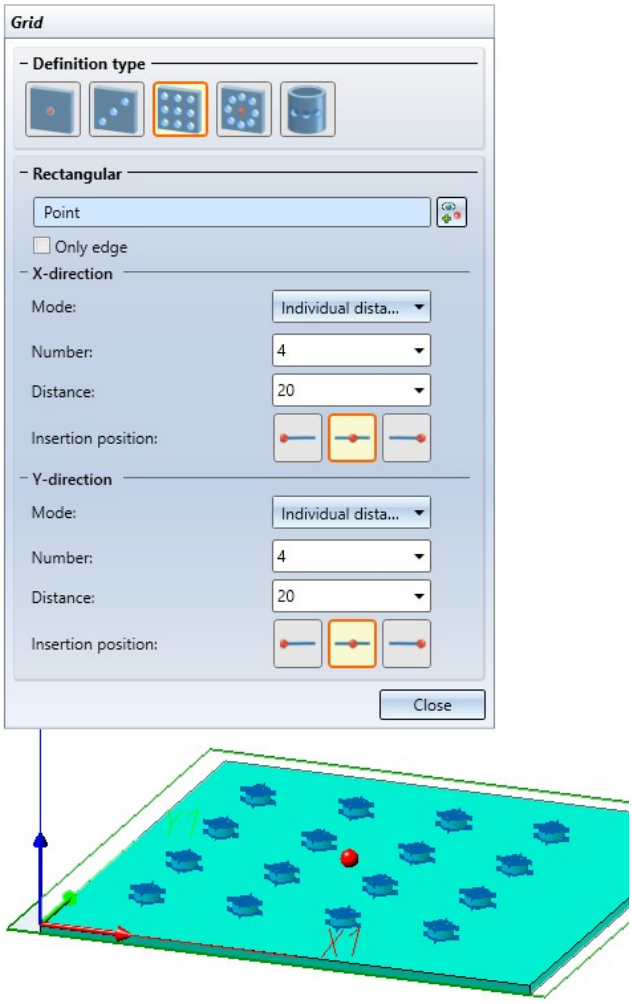


Standard processings

New standard processing

The new function **Bores, Countersinks, Threads**  has been added to the **Standard Processings** area.





In the **Form** area, you can choose from **Bore**, **Threaded hole**, **Counterbore**, **Slot** and **Rectangle**. In combination with the **Type** area, which is tailored to each individual form, almost all previous standard processing functions can also be carried out using the new function.

Whereas the old functions for standard processing required many dialogue windows to be filled out one after the other, the advantage of the new standard processing is that all processing steps are combined in one dialogue. The **Bores, Countersinks, Threads** function includes the following new features:

- The **Settings for bores** dialogue window has been integrated into the new dialogue so that these settings can be changed later while the part is being processed.

- In the areas **Part to be processed**, **Processing plane** and **Grid**, you can change the part to be processed, the processing plane or the grid settings at any time.
- The design of the configuration window for the grid settings has been completely modernised.
- The catalogue system for bores, threads and counterbores has also been integrated into the new dialogue and can be accessed at any time for the **Bore**, **Threaded hole** and **Counterbore** types via the catalogue symbol in the **Type** area.
- The new dialogue contains the **Perpendicular for laser cutting** option. If the inserted bores are to be cut with a laser, you can set the laser to be perpendicular to the surface during cutting by activating the checkbox and also specify the size of any clearance. This option is not available for counterbores and threaded bores, or for bores with a set drilling depth.
- For the previous functions from the Standard Processings area standard processing, it is possible to subsequently specify in the feature log whether the processing is to be carried out in the **Workshop** or on **Site**. This selection option has also been added to the dialogue for the new standard processing in the **Production** area.

You will find the new function in the pull-down menu of the function **3-D Standard > Standard processings > Bores/Threads**  .

Settings for standard parts and processings in the Configuration Editor

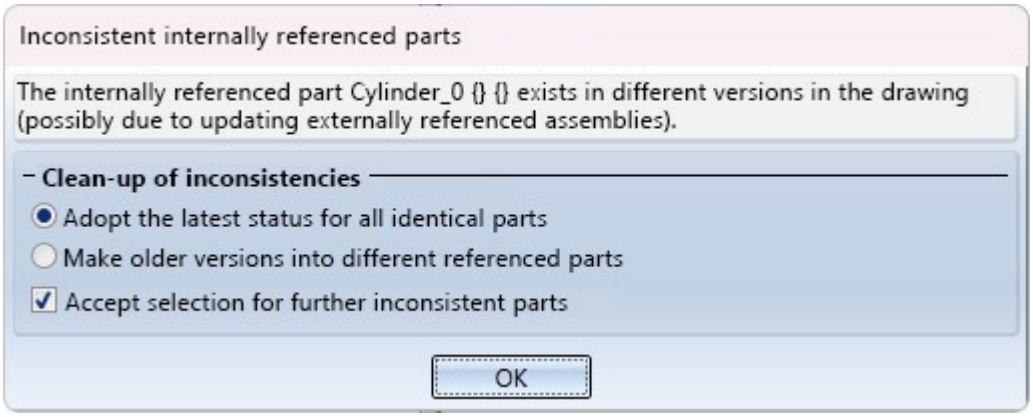
In connection with the new function **Bores, Countersinks, Threads**  , the settings for standard parts/standard processing in the Configuration Editor have been changed as follows:

- The path **System settings > Standard parts** has been renamed to **System settings > Standard parts and processings**.
- The parameter **Colour no. mounting bore** has been renamed to **Colour of construction site bore / mounting bore**.
- The parameter **Required minimum material thickness for blind holes** is not evaluated by the new standard processing. The setting can now be found under **Compatibility > Standard parts and processings up to HiCAD 2023**.
- The settings for 2-D functions have been moved to the path **System settings > Standard parts and processings > Standard parts and processings, 2-D** .

Internally referenced parts in externally referenced assemblies

If internally referenced parts are inserted as sub-parts of externally referenced assemblies, it is possible that these are modified differently in external drawings. For this reason, internally referenced parts are now given a time stamp when their superordinate externally referenced assemblies are saved or when the drawing in which they are installed is saved. In this way, it is possible to trace which copy of identical parts was saved last.

If conflicts are detected, the following dialogue is displayed:

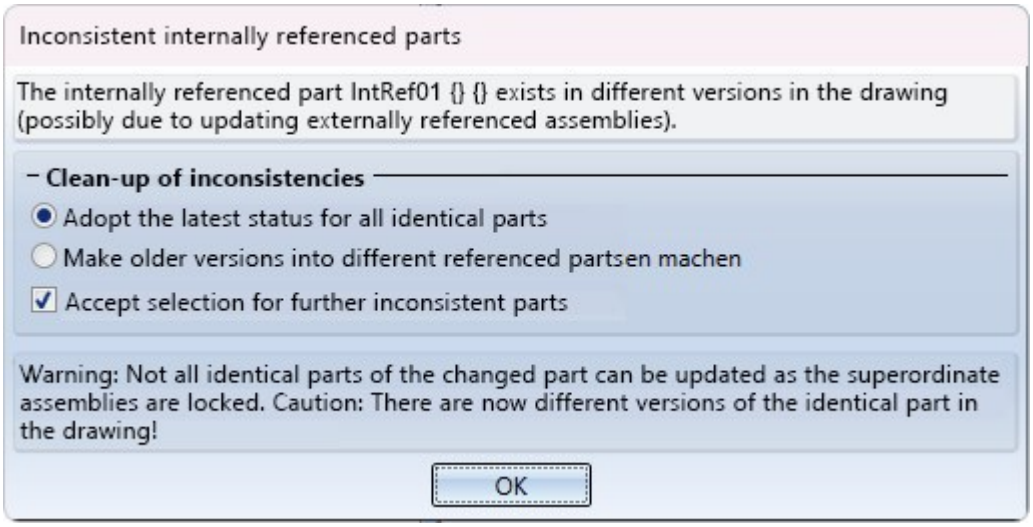


You can therefore decide whether all identical parts should be replaced with the latest version or whether different identical parts should be treated as different referenced parts from now on. Changes are then no longer synchronised between these parts.

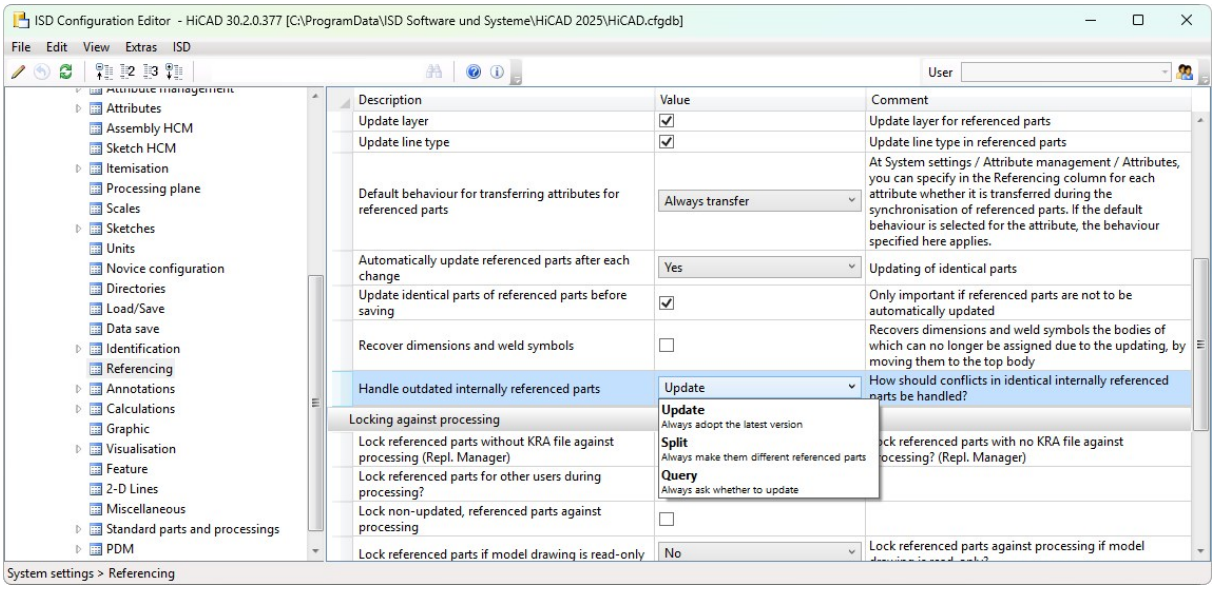
The test for conflicts is also carried out when loading drawings from older HiCAD versions (before HiCAD 2025). This means that the message can already appear when loading older drawings, even if no externally referenced parts have been updated. If the drawings are saved (from HiCAD 2025), the conflicts are cleaned up automatically. After that, conflicts can only arise due to competing changes in externally referenced parts - in which case the query shown above appears again.

Hence the recommendation:
Please do not install the same internally referenced parts as sub-parts of different externally referenced assemblies. This can lead to conflicts!

If these conflicts are below released externally referenced assemblies (which accordingly may no longer be changed), the following message appears:



If you want HiCAD to always use the latest version or always create different reference parts without a dialogue appearing, you can change the behaviour in the Configuration Editor. There you will find the parameter **Handle outdated internally referenced parts** under **System settings > Referencing**.



The ISD default setting **Query**.

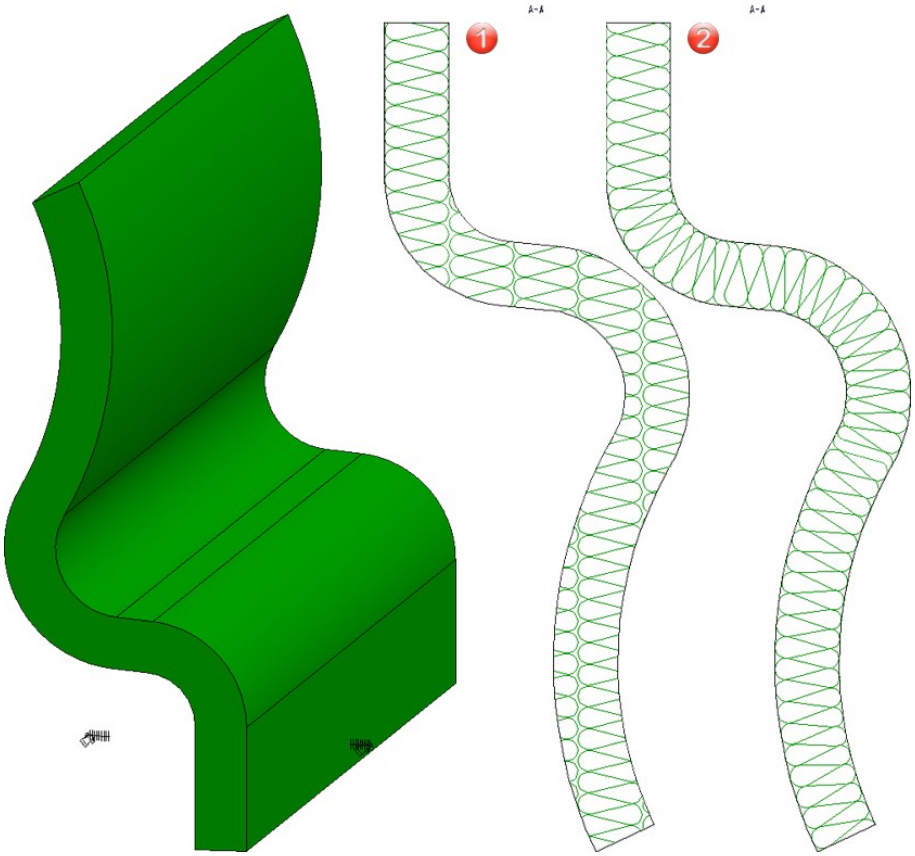
Improved algorithm for insulation hatching

The algorithm for insulation hatchings has been significantly improved. From HiCAD 2025, the insulation hatching follows the geometry of the part. The requirements for the part to be hatched are:

- The polyline has only tangential transitions.
- The thickness of the part is constant.
- The boundary consists exclusively of circle and straight line segments.

Example:

The drawing contains a part created with the **Insulation along sketch** function. (1) shows the sectional view with insulation hatching in HiCAD 2024, (2) the sectional view in HiCAD 2025.



Please note:
When loading drawings created with an earlier version of HiCAD, the insulation hatching is not updated automatically. This only takes place when the part is recalculated or when the sectional/detail views or the cut-out are updated.

Dimensioning in loop

When using the **Linear dimensioning, parallel**, **Linear dimensioning, direct**, **Angle dimensioning** and **Arc dimension** functions, another dimensioning can be created after pressing the **MMB** for the first time. The selected function remains active. The function is cancelled by pressing the **ESC** key or by pressing the **MMB** again.

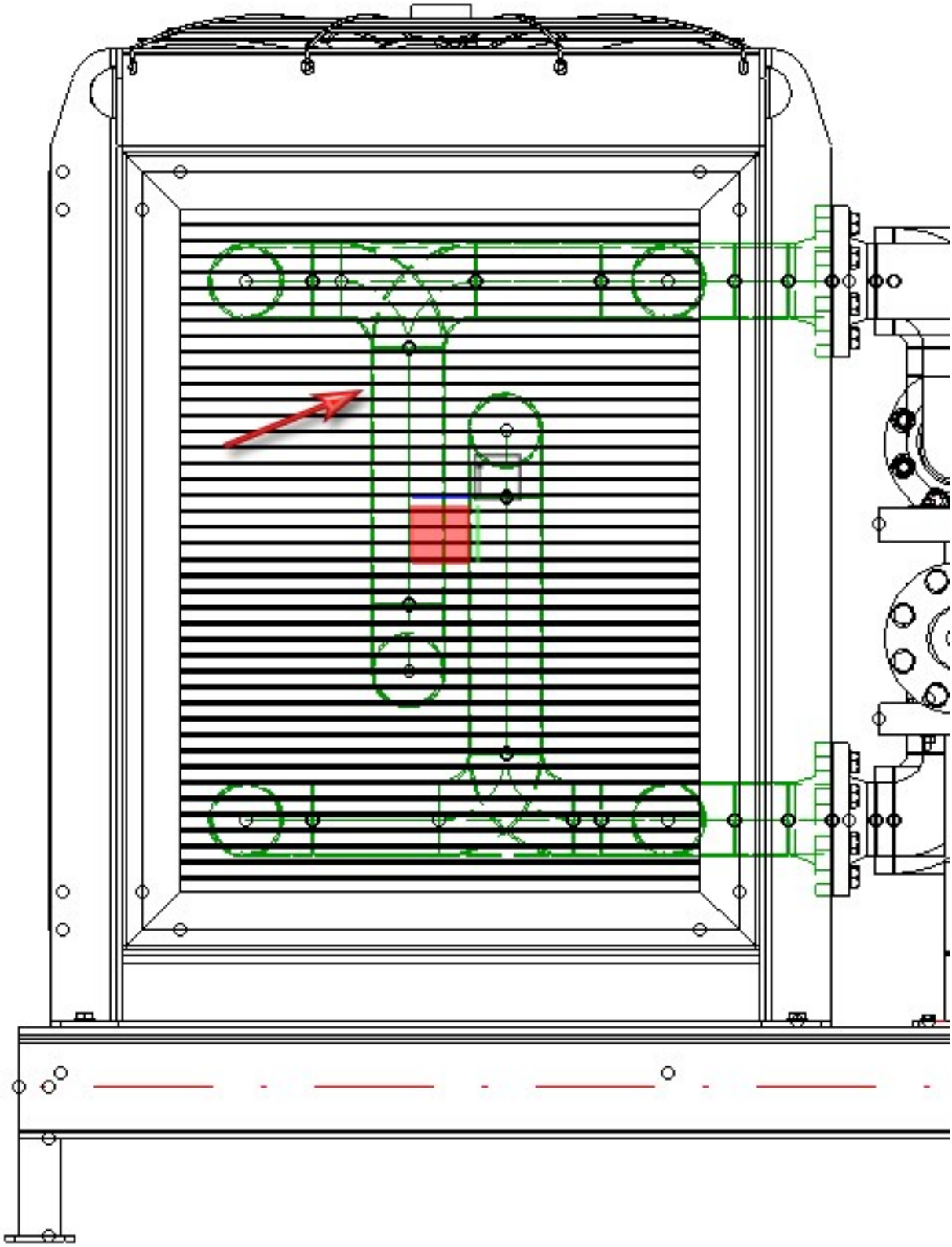
Display hidden lines of individual parts

Two new functions are available in the context menu of 3-D parts:

 **Hidden edges dashed**

 **Hidden edges normal**

The **Hidden edges dashed** function can be used to display the hidden lines of individual parts in a view. For example, if the **Hidden Line** display mode is active for the view, the Hidden Line dashed display mode can be selected for individual parts of the view. With the **Hidden edges normal** function, the display mode of the view is selected again to display the part.



The functions can also be selected in shaded views. However, the effects only become apparent when the display mode is switched to **Hidden Line**.

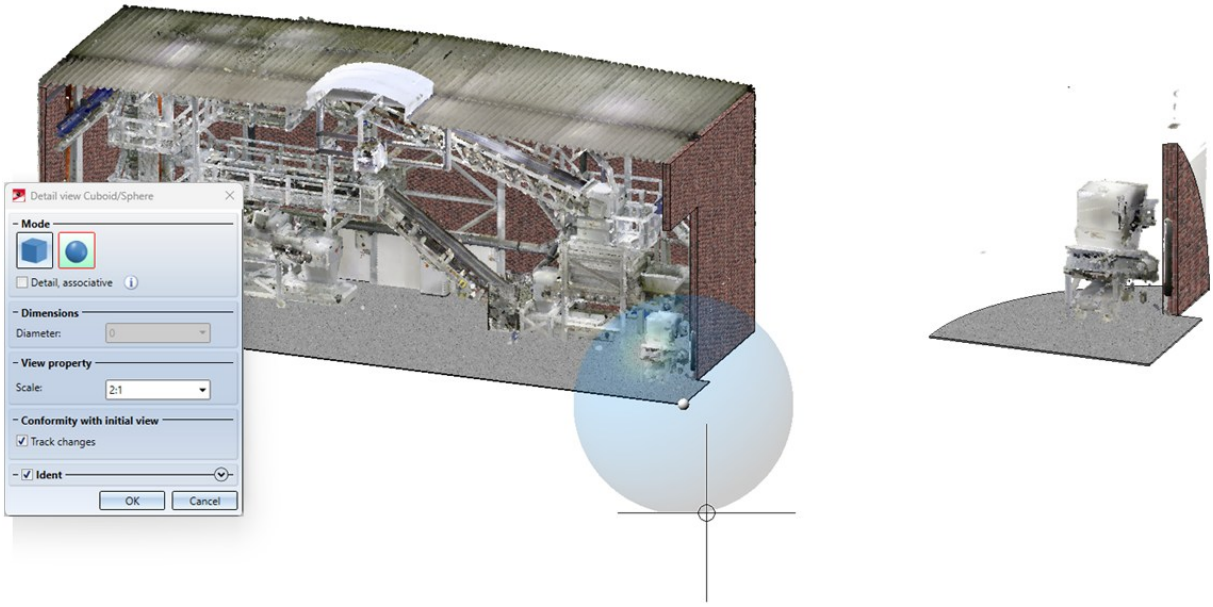
Point clouds

Major Release

Point clouds in sectional and detail views

Point clouds are now also taken into account when creating and editing sectional and detail views. The following restrictions apply:

- BIn sectional views, the section path can consist of one or more line elements. The section path must be open.
- If the **Surface intersection** option is selected when creating/processing the sectional view, point clouds are ignored.
- In the case of details from sectional views, any section boundary clipping planes that may be present are ignored.



Example of a detail view

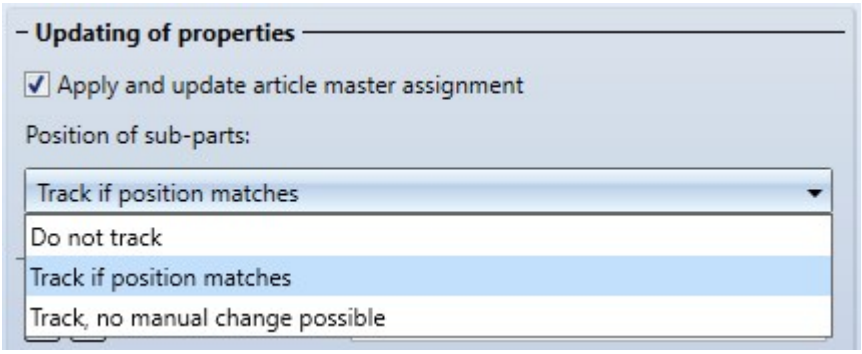
Feature Technology

Service Pack 2

Part structure of dependent parts

Starting with HiCAD 2025 SP2, it is no longer possible to change the part structure of assemblies that have been marked as dependent using the **3-D Standard** > **New** > **Dependent part** function.

Transforming sub-parts into dependent assemblies



In the dialogue of the function **3-D Standard** > **New** > **Dependent part**, the option **Track geometrical position of sub-parts** has been specified. The checkbox has been replaced by the list box **Position of sub-parts** with the following options:

- **Do not track**: The geometrical position of the sub-parts in the dependent assembly does not change when sub-parts are transformed in the original assembly. This setting corresponds to the previous behaviour when the checkbox was deactivated.
- **Track if position matches**: The geometrical position of the sub-parts in the dependent assembly changes exactly when the position last matched. This means that as soon as you move a part in the dependent assembly, movements of the part in the original assembly no longer affect it. This setting corresponds to the previous behaviour when the checkbox was activated.
- **Track, no manual change possible**: No manual changes can be made to the sub-parts of a dependent assembly. When updating, the position of the sub-parts is restored so that they match the sub-parts of the original assembly.

Major Release

New feature functions

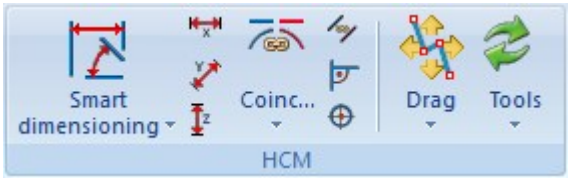
In HiCAD there is now the new feature function **catalogue_item_id_or_zero** in addition to the existing **catalogue_item_id** function. It returns the first catalogue record ID that matches the column values searched for. If no ID is found, **0** will be returned. A list of the current functions can be found here.

HCM

Service Pack 2

Optimisation of icons in the 3-D C-edge HCM Ribbon

The icons in the Ribbon of the 3-D C-edge HCM have been slightly changed to make them look simpler overall.



Service Pack 1

Automatic constraint search during dragging of 3-D Part HCM removed

Previously, an automatic constraint search was started using the function **3-D-Standard > HCM > Dragger, Motion Simulation**  . This means that when dragging the moved 3-D part, possible HCM constraints are suggested as soon as the cursor hits points, edges or surfaces of other objects. The constraint search has been removed from the function, so that the function only moves a part or assembly along the screen plane while adhering to the assigned HCM constraints.

As a result, the option **Preserve Drag constraints** has been removed from the 3-D Part HCM settings at **3-D Standard > HCM > Tools > Settings**  .

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.

Service Pack 2

Support of the new Report Manager

The new class

- ISD.CAD.BOM.ReportManagerExport

is available for the Report Manager.

Point clouds

The HiCAD API has been expanded to include the **PointCloud** class for working with point clouds.

Attach bend zone without flange

To improve user-friendliness, the **Attach**  dialogue was revised and extended in HiCAD 2025. This results in the new command for attaching bend zones without flanges:

```
var sheet = Context.CreatePart(new BaseSheetCreator2(100, 150, 2));
var connectingEdge = sheet.Edges.ElementAt(1);
sheet.Apply(new AttachFlange2(connectingEdge, 1, 90));
```

Referencing property

The existing class for reading the attribute properties is to be extended by the properties of the referencing transfer.

This particularly changes the designations **References**. The HiCAD API now provides the following command:

- **AttributeSettings.GetSpecifications()**, extended by Referencing-Property.

Drawing Management

In the ICN, drawings that have been created and edited with the Drawing Management are identified as follows:

	There is no database connection.
	The drawing has not yet been saved in HELiOS and no product structure is available, or The drawing contains objects that have not yet been saved in HELiOS and the product structures in HiCAD and HELiOS are not identical.
	The drawing has been saved, but the data in the database is not up to date (e.g. if you have used the Save without HELiOS update function).

To determine the current status, the HiCAD API provides the following command:

- Scene.GetDrawingManagementICNState()

Updating drawings

The HiCAD API provides the following command for updating drawings:

- Scene.ShowAttributeDialog()

DXF/DWG export of Sheet Metal parts

The following command is now available for the DXF export of a list of non-itemised sheet developments:

- DXFSheetDevelopment3DSettings.AllowInvalidItemNumbers

Service Pack 1

Insert new beam

For the **Insert new beam**  function from HiCAD 2023, there is a new class in the HiCAD API:

- ISD.CAD.Steel.BeamCreator?2

Feature "Create new" for Element Installation

The context menu of the **Element Installation** feature includes the function **Create new**  . This function recalculates all installed elements based on the selected parameters.

Since the function takes longer to recalculate larger drawings, the HiCAD API provides the following script:

- RecreateElemInstall.cs

Major Release

Into assembly

Use the **Into assembly**  function (RMB > Assembly/Assembly/Part structure) to convert **Dummy parts**  into assemblies. The HiCAD API now provides the following command for this purpose:

- Node.ConvertToAssembly()

Update processed bolted/riveted elements

After making changes to the bolted objects, you can use the **Update**  function to update the bolting/riveting accordingly, for example if you have subsequently changed the thickness of bolted plates. The function can be used for individual boltings/rivetings, all boltings/rivetings of the active part or all boltings/rivetings of the drawing.

The HiCAD API now provides the following command for this purpose:

- ISD.CAD.Data.Boltings.Update()

Delete all item numbers

The HiCAD API now also supports the deletion of all item numbers  with the following command:

- ISD.CAD.Steel.Itemization2.DeleteData

If the itemisation mode **By top level assemblies** is active, only the numbers of the parts that were itemised in this mode are deleted.

Standard part - Free weld seam symbol

You can use the **Free weld seam symbol**  function to individually configure the annotation with weld seam symbols on any lines (in accordance with DIN EN ISO 2553). The HiCAD API provides the following class for this purpose:

- Class WeldSeamSymbolCreator

Hatching in sectional view and cut-out

The **Hatching in section and cut-out**  function can be used to define an individual hatching for each part and view for those cut surfaces that result from sections, cut-outs or detail cut-outs. The command **Node.Properties.SetCutHatching** is available in the HiCAD API for this function. New for the properties of the insulation hatching is the parameter

- InsulationHatching.

Interfaces

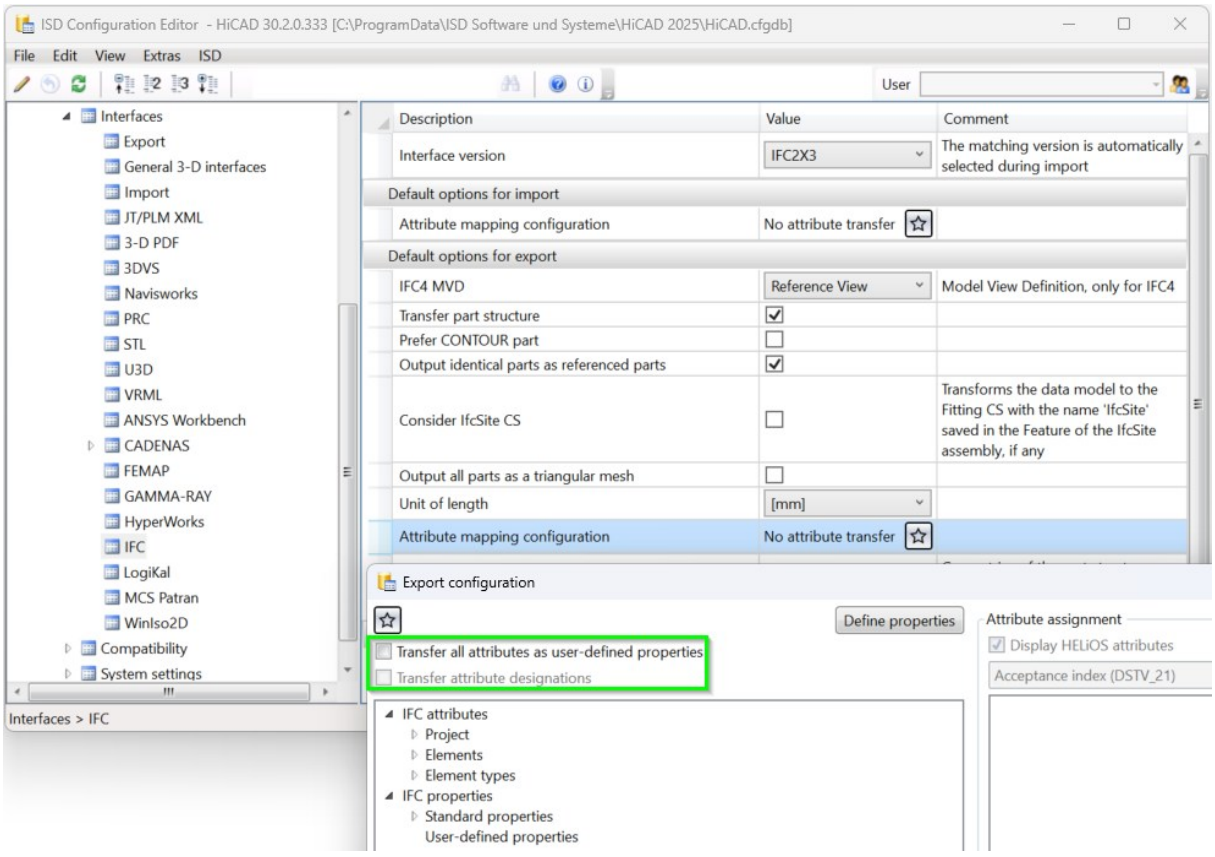
Service Pack 2

IFC interface

Transfer all attributes as user-defined properties

At the top of the **Export configuration** dialogue window for the IFC attribute mapping, you will find two new check-boxes:

- **Transfer all attributes as user-defined properties**: Activate this check box if you want to automatically write all HiCAD attributes to the IFC file without having to make any further export configurations.
- **Transfer attribute designations**: If this option is active, the additional checkbox **Transfer attribute designations** can be used to specify that the language-dependent designations of attributes are to be exported instead of the keys of user-defined properties.



Export of grids

The new HiCAD planning grids are also taken into account when exporting to the IFC format.

Increased performance and improved import

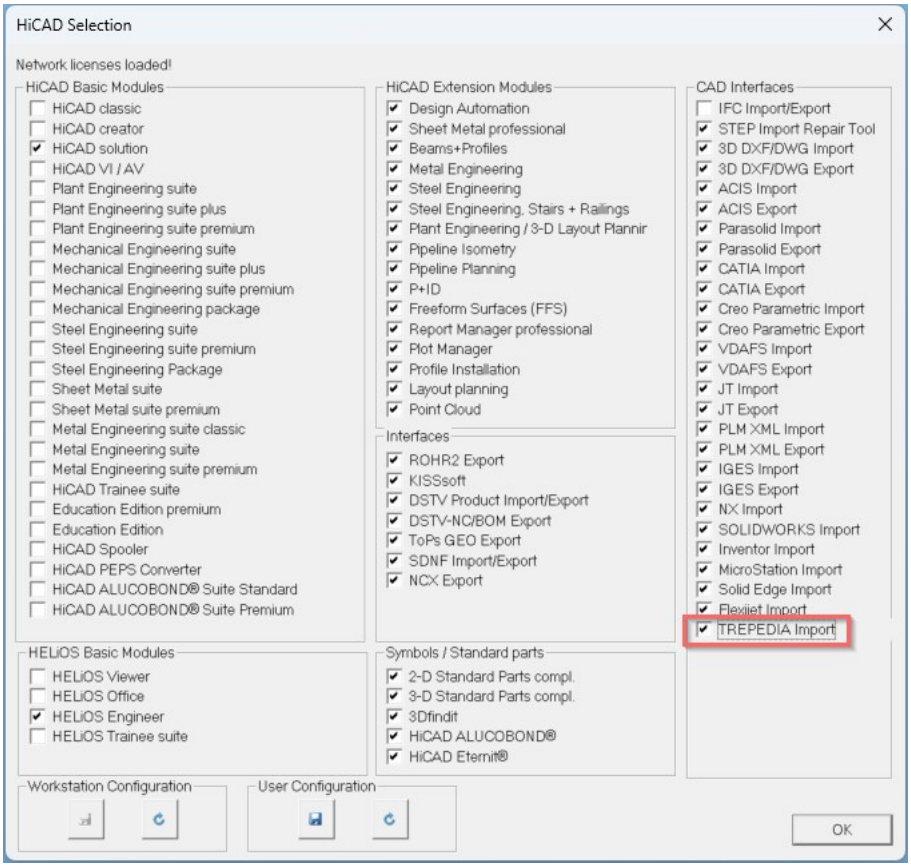
A significant performance improvement for importing IFC files has been achieved by an adjustment in the area of feature calculation.

In addition, structuring parts are now always created as an assembly during import, instead of as a dummy part.

TREPEDIA import interface

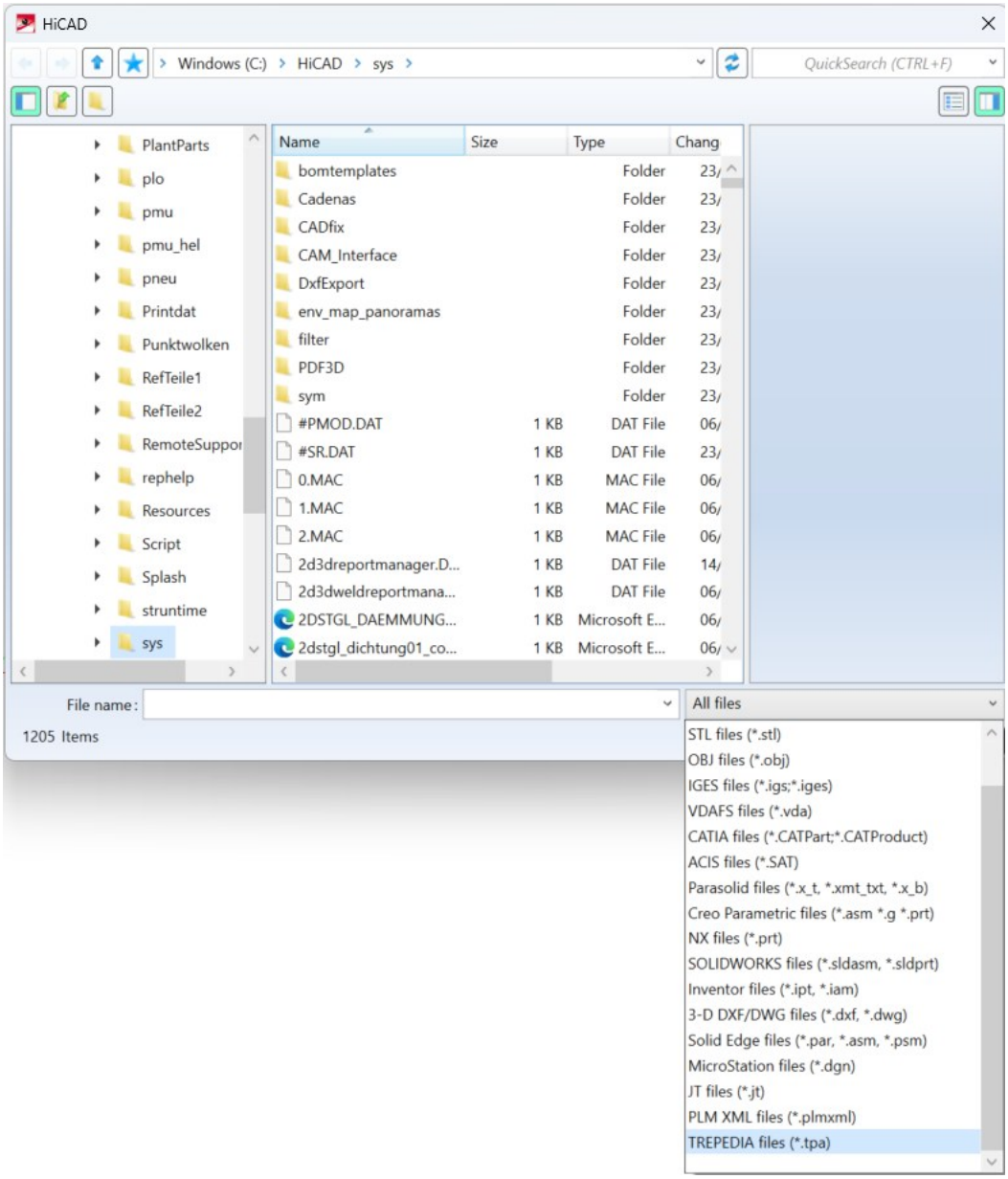
The TREPEDIA import interface allows TREPEDIA models (stairs, railings) to be transferred to HiCAD in IFC format without using the HiCAD IFC import/export interface. The IFC models from TREPEDIA are exported in IFC format with the file extension **.tpa**.

The new **TREPEDIA Import** interface selection is available for this purpose when you start HiCAD.



TREPEDIA files are imported via **Drawing > Insert Part > Exp.** **> 3-D Import**  .

Select the file type **TREPEDIA files (*.tpa)** from the pull-down menu of the import dialogue window at the bottom right.



Please note:

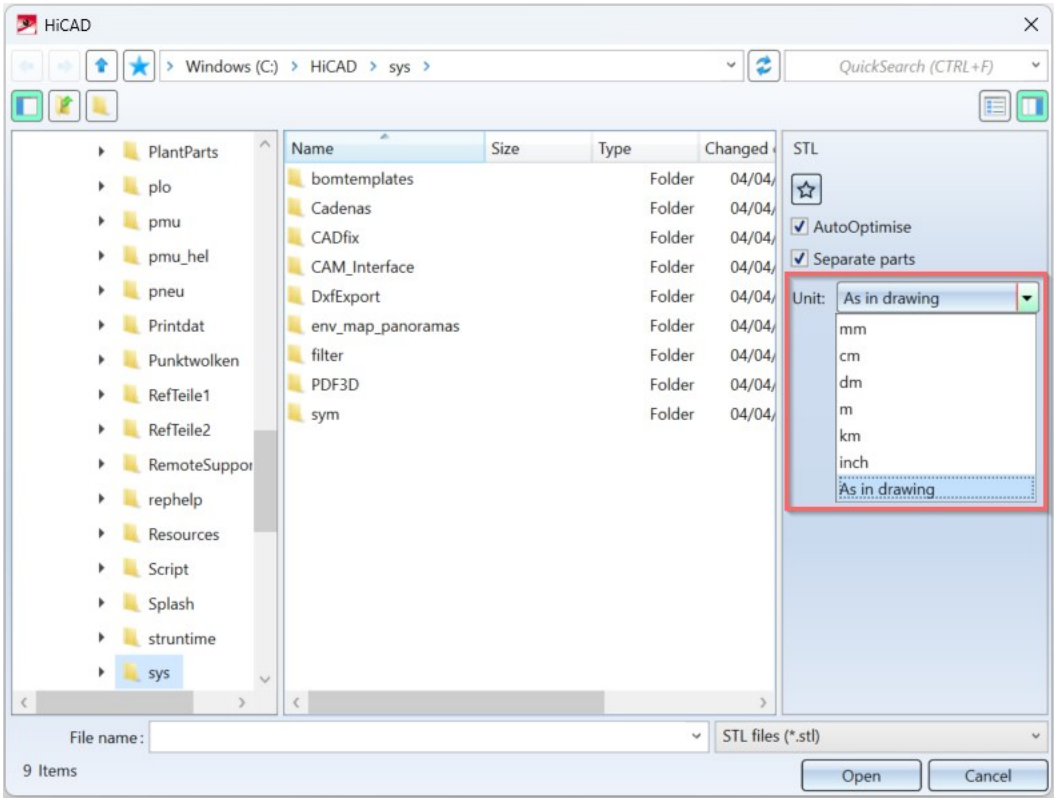
In addition, TREPEDIA models that are exported in IFC format with the extension .tpa using the TREPEDIA button "Direct to HiCAD" can also be transferred directly to HiCAD in this way.

STL and OBJ import

Selection of units

To react to different scenarios when importing STL or OBJ files, Service Pack 2 of HiCAD 2025 makes it possible to select the unit in which the data is to be interpreted in the corresponding dialogue.

By default, the data is read in the current construction unit. You can select a different unit of measurement in the new **Unit** drop-down menu.

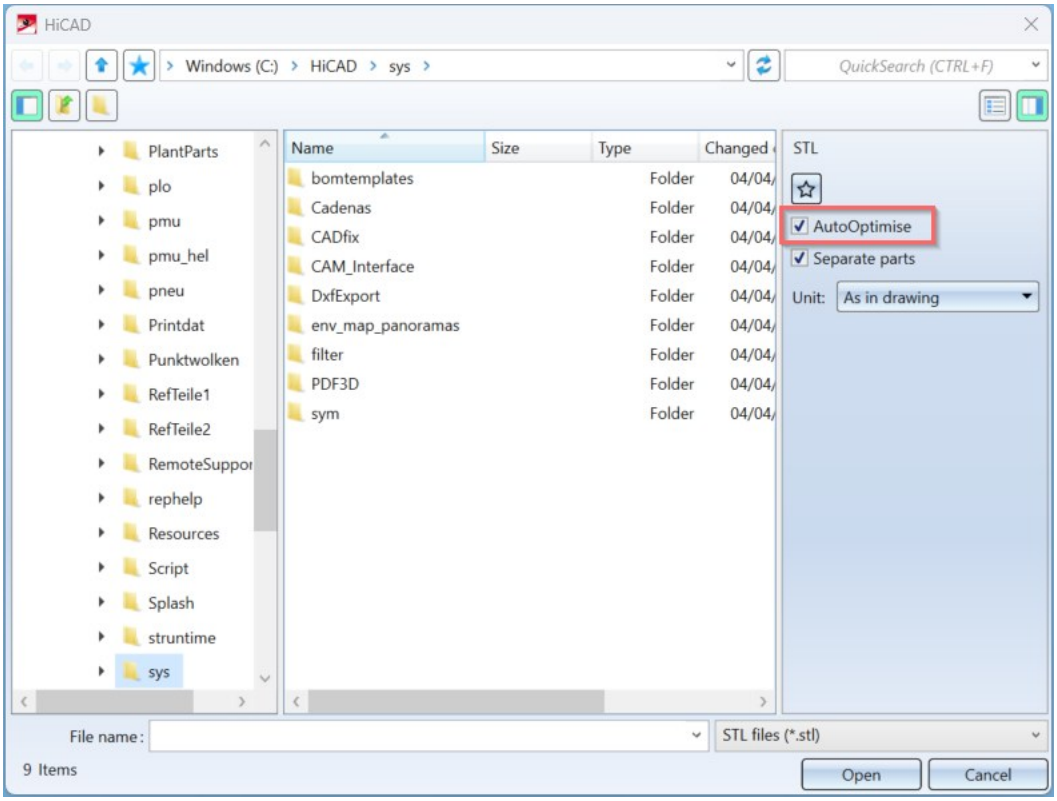


The unit settings can also be saved as Favourites.

Automatic optimisation as default setting

The option **AutoOptimise** is active by default as of Service Pack 2 for both STL and OBJ imports.

This corresponds to the HiCAD function **3-D-Standard > Tools > Surface > Further > Optimise** and performs a merging of (divided) edges and surfaces during import.



FlexiCAD XML: Support of further 3-D objects

The XML interface of Flexijet 3D and FlexiCAD has been expanded to support additional object definitions.

The following new elements have been added:

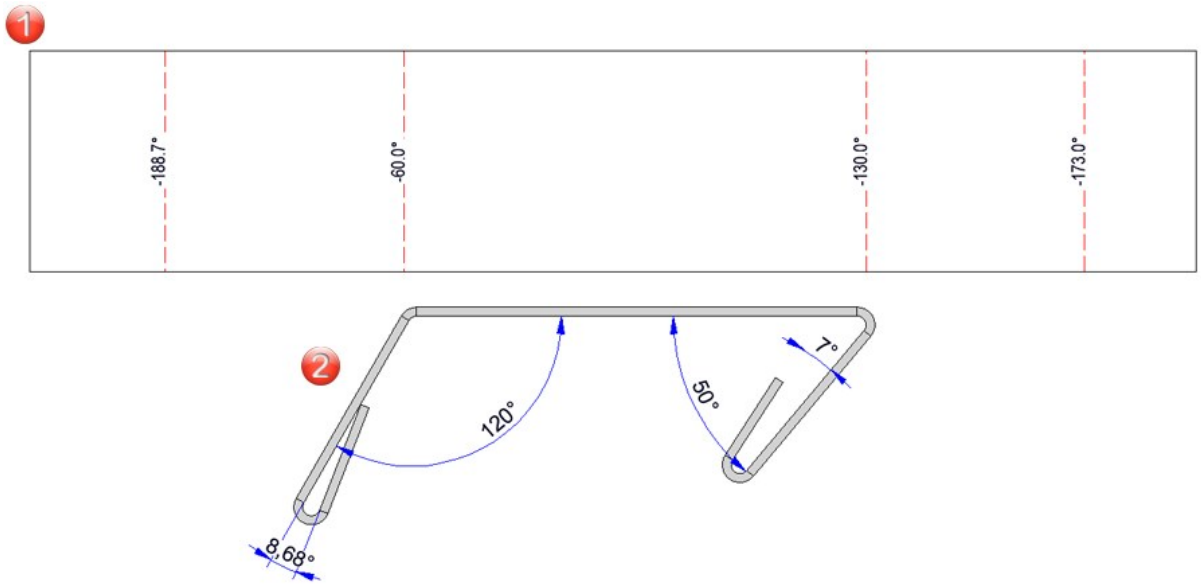
- **<UserWall?> (User-defined wall)**:
  Enables the definition of walls that have been created manually by the user and do not originate from the room measurement module. These walls can have complex geometries, such as pentagonal or trapezoidal shapes, which are defined by a variable number of corner points. The coordinates of these corner points are specified precisely in the XML file.
- **<ObjectType? Value="Void"> (Subtraction body)**:
  Used to define subtraction bodies that the user can already define in FlexiCAD.
- **<ObjectType? Value="3dObject"> (3-D solids)**:
  Enables the integration of 3-D solids that the user has created in FlexiCAD.
- **<ObjectType? Value="Surface"> (Surfaces)**:
  Enables the definition of user-defined surfaces.

TopsGeo Export: Bend angle

When exporting sheet developments with the **ToPsGEO interface**, the CAMInt_HICAD_GEO.ini file is evaluated. This file has been extended so that you can now also output border lips with a positive angle. All folds with an opening angle of < 30° are interpreted as border lips.

Extension in the CAMInt_HICAD_GEO.ini:

Angle (1 = Opening angle, -1 = neg. Opening angle, 2 = HiCAD angle, -2 = neg. HiCAD angle, 3 = Opening angle, also pos. angle for bending angle > 180°, -3= neg. Opening angle, also neg. angle for bend angle > 180°)



In this example, all angles with a negative sign are output when exporting with the setting **-3**. This seems wrong because the 7° angle (on the right in the picture) corresponds to a bend angle of 173° while the 8.68° angle (on the left) corresponds to a bend angle of 188.7°. However, this is the intended result when the setting **-3= neg. opening angle, also neg. angle for bend angle > 180°** is used.

STEP format

Import and export of materials

Materials of parts are taken into account when importing and exporting the STEP format.

For import, the corresponding materials must already be available in the HICAD catalogue.

For export, the density listed in the catalogue is also exported so that the material can be processed in other systems. You can also control whether the material designation or the material number should be written as the material name.

Performance increase for exporting sheets

When exporting unioned sheets, e.g. to IFC or STEP format, longer processing times could occur in the past. With the update to HiCAD 2025 Service Pack 2, a considerable performance increase has been achieved in this area.

Service Pack 1

DXF/DWG interface

Spaces in layer names for DXF export

In earlier versions of HiCAD, spaces in layer names were replaced by underscores for DXF/DWG export.

Since AutoCAD supports spaces in names, this restriction has been removed.

Direction of the outer contour during DXF export in sheet developments

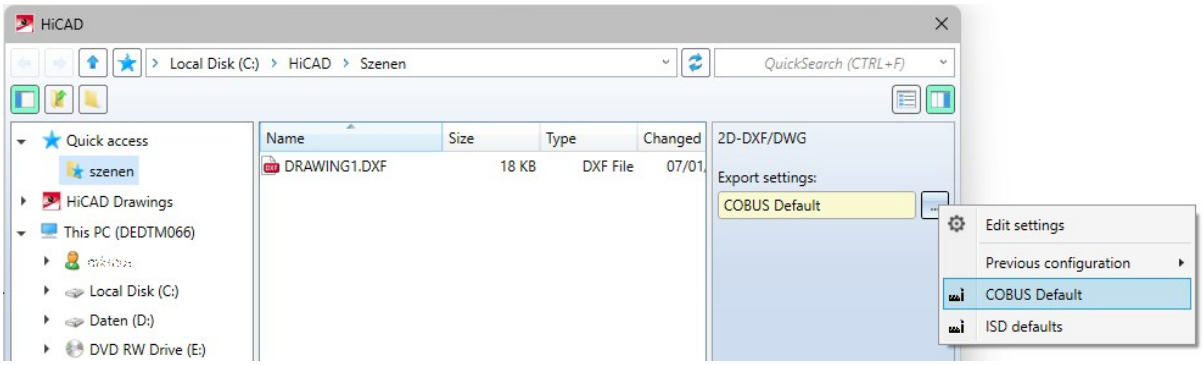
To avoid damaging the material during sheet development, the start point should not be located in an inner corner (concave corner) of the outer contour when exporting the DXF file.

In HiCAD, the edges of closed contours are therefore resorted when exporting DXF via **Sheet Metal > Sheet development > Export** so that the start point is not located in a concave corner.

COBUS Default

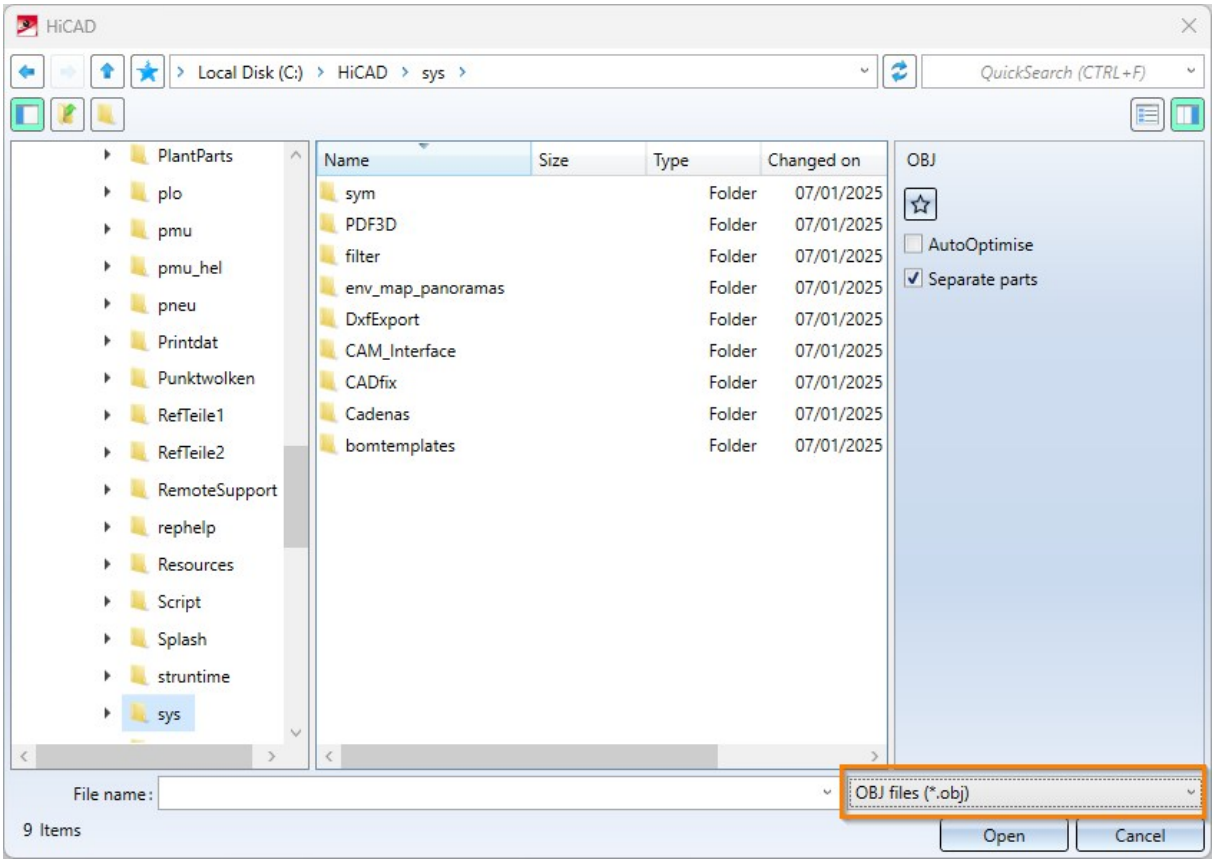
The **COBUS Default** configuration is now available for the DXF interface for Sheet Metal parts in the export dialogue under **Export settings**.

It replaces the old HCADACAD_COBUS.DAT file. The configuration cannot be changed.



OBJ format

With the update to Service Pack 1, OBJ files (.obj) can also be imported in HiCAD via **Drawing > New/Open > Open > 3-D Import**.



OBJ files can also be imported via Drag & Drop.

Navisworks: Export of GUIDs

The Navisworks export now also assigns GUIDs to parts.

This allows you to use quantification functions (quantity determination) in Navisworks.

The GUIDs are stored in the SZA file from which the export is carried out. If another export is carried out, the same GUIDs are assigned again.

HiCAD-Trepedia interface

In order to simplify the interface between HiCAD and the Trepedia program, the **Trepedia** function has been added to the **Extras** pull-down menu in the **Further functions** function group of the **Steel Engineering** Ribbon.

Selecting the function starts Trepedia. After the corresponding stairs have been generated, the resulting model can be inserted into HiCAD as an IFC or STEP file.

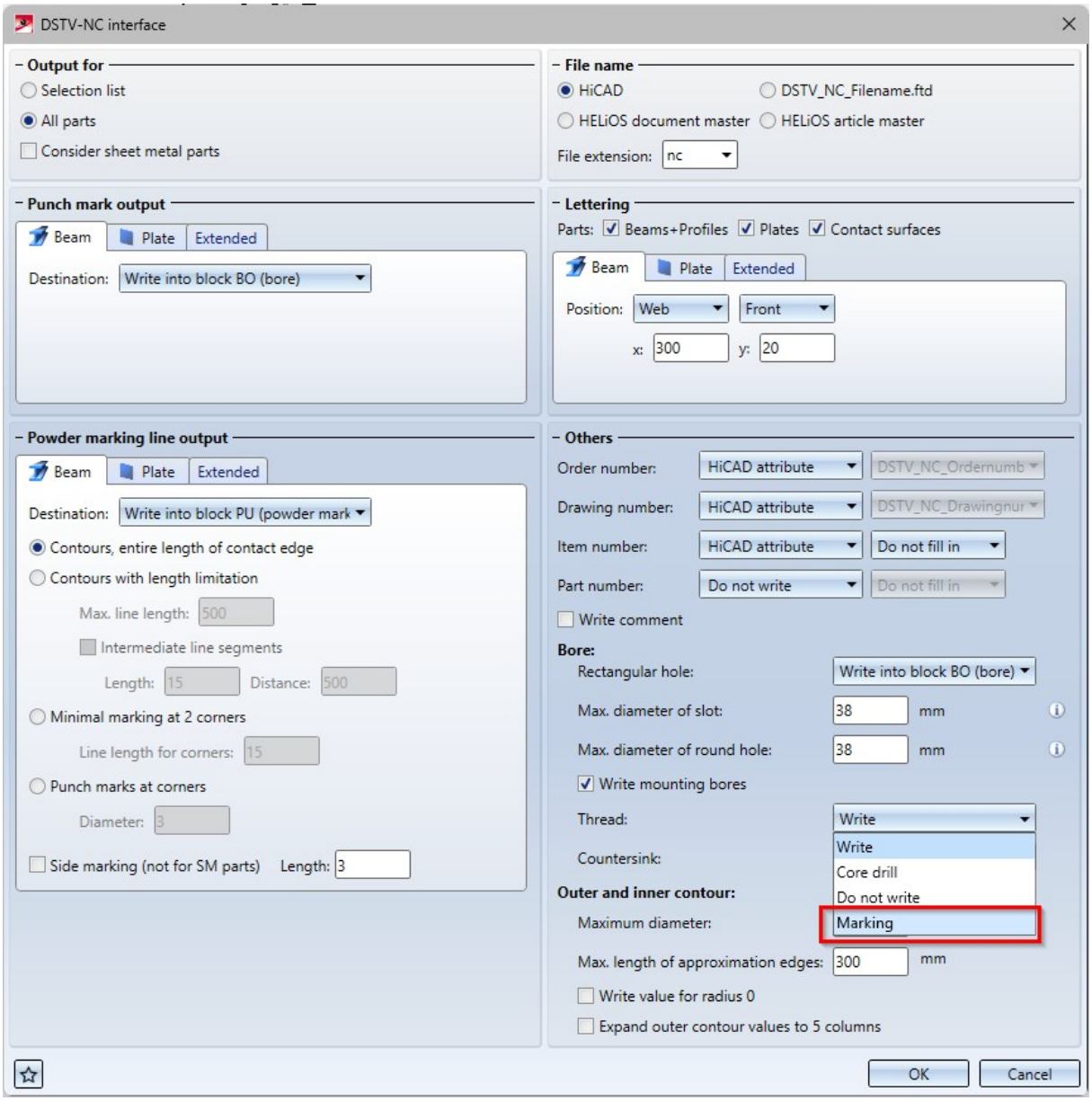
KISSsoft 2024

HiCAD supports the new version of the KISSsoft plugin.

Major Release

DSTV-NC - Output thread as marking

The dialogue window of the DSTV-NC output has been extended in the **Others** section for threads. Previously, threaded holes could only be written, not written or marked as core holes. As of HiCAD 2025, it is now also possible to output threaded holes as **Markings**. In this case, the bores - as previously the punch marks - are given the identifier **m**.

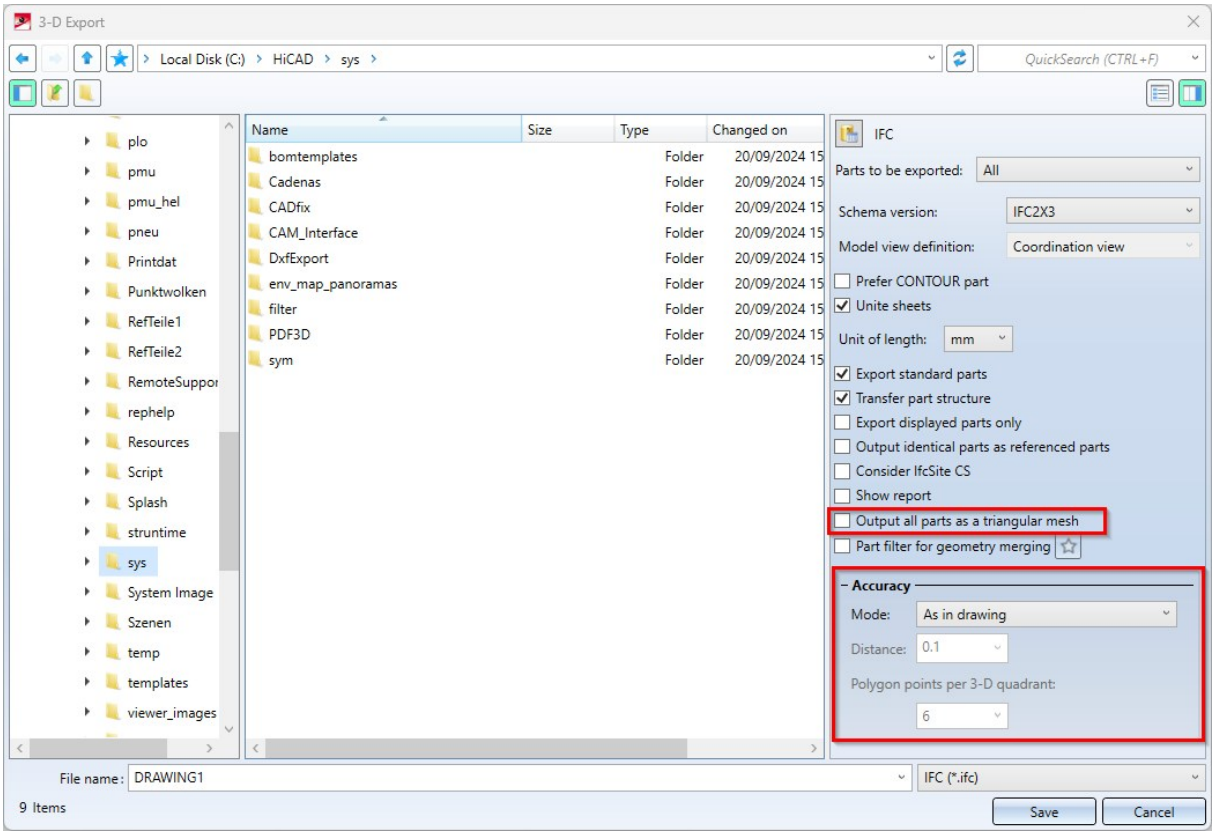


An example:

Setting	BO block
Marking	BO v 728.63o 69.20m 0.00 0.00 v 437.56o 74.23m 0.00 0.00 v 104.60o 76.75m 0.00 0.00
Write	BO v 728.63o 69.20g 12.00 0.00 v 437.56o 74.23g 12.00 0.00 v 104.60o 76.75g 12.00 0.00

IFC export - output as surface model

For IFC export, you now have the option of exporting all parts as surface models. The dialogue window has been expanded accordingly.



If the **Output all parts as triangular mesh** checkbox is activated, all parts are exported as triangular meshes. This can significantly speed up the export process, especially for drawings with many profiles and their edits. However, it should be noted that the profiles cannot then be processed as such by other systems. The accuracy for the surface approximation can be defined under **Accuracy**. If **As in drawing** is selected here, the accuracy that you set with the **Surface approximation** function is used. The other settings correspond to those of the Surface approximation function. The value selected here affects the visualisation of the parts. The higher the value, the cleaner and more accurate the shading. However, this also increases the file size!

Sheet Metal

Service Pack 2

Manufacturability check

The manufacturability check in the Design Checker can now also be performed when creating developments. To do this, activate the **Perform manufacturability check when creating developments** parameter in the Configuration Editor at **Sheet Metal > Manufacturability check**.

The following new checks are now available:

- **Minimum flange length**

  The minimum length of flanges is measured directly on the outside or directly on the outer tangent, as with the Attach function. Only the outside of the cover surface is measured; chamfers can reduce this. A distinction is made between acute and obtuse angles.

- **Minimum bend radius**

  To avoid overstretching the material, a certain bend radius should not be exceeded.

- **Distance between processings**

  To check the distance between processings in the manufacturability check, activate the parameter **Check minimum distance between processings** in the Configuration Editor.

- **Minimum diameter for standard bores**

  This test only applies to standard bores.

- **Minimum Z-fold height**

  The comparison values for this test are loaded from the catalogue. You must therefore first fill in the columns **Min. Z-fold height (<90°)** and **Min. Z-fold height (>=90°)** of the corresponding tables.

New design variant for panels

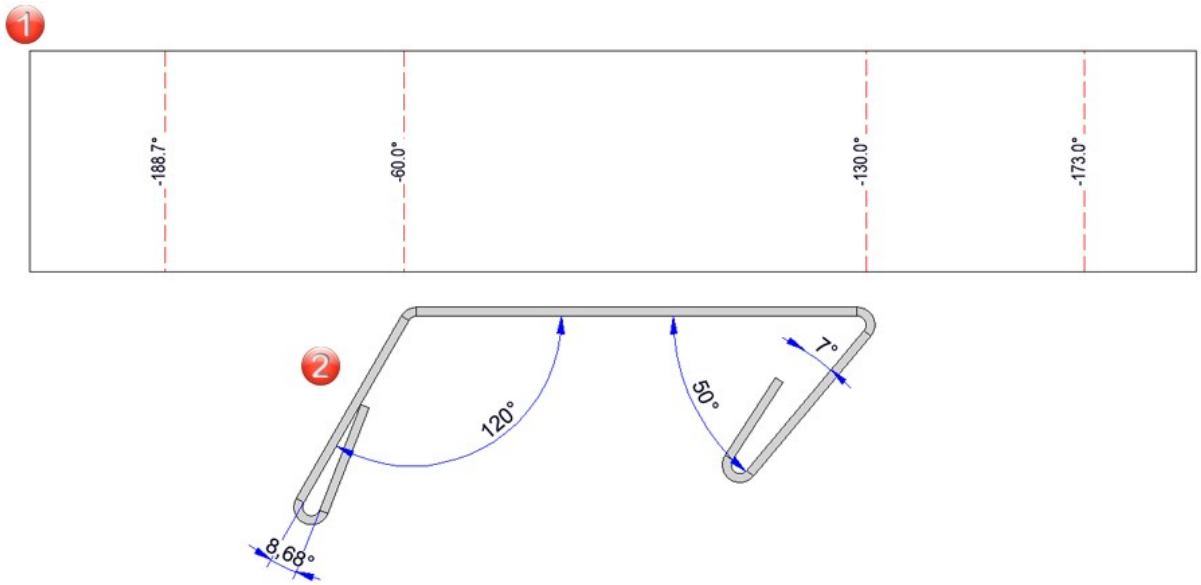
To assist you in designing facades, HiCAD offers the **Panel** design variant. This function allows you to create panels in any shape using the Sketch technology or to fit the panels directly into a frame by selecting beams/profiles. You can select the material for the surface from the catalogue according to a semi-finished product for the glass pane and the sheet.

TopsGeo Export: Bend angle

When exporting sheet developments with the **ToPsGEO interface**, the CAMInt_HICAD_GEO.ini file is evaluated. This file has been extended so that you can now also output border lips with a positive angle. All folds with an opening angle of < 30° are interpreted as border lips.

Extension in the CAMInt_HICAD_GEO.ini:

Angle (1 = Opening angle, -1 = neg. Opening angle, 2 = HiCAD angle, -2 = neg. HiCAD angle, 3 = Opening angle, also pos. angle for bending angle > 180°, -3= neg. Opening angle, also neg. angle for bend angle > 180°)



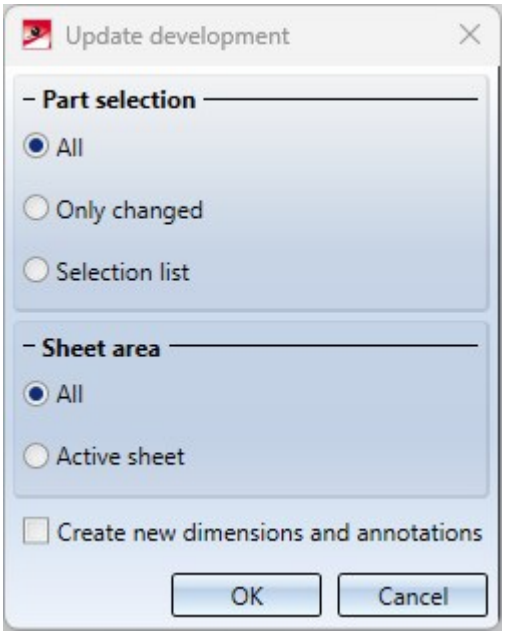
In this example, all angles with a negative sign are output when exporting with the setting **-3**. This seems wrong because the 7° angle (on the right in the picture) corresponds to a bend angle of 173° while the 8.68° angle (on the left) corresponds to a bend angle of 188.7°. However, this is the intended result when the setting **-3= neg. opening angle, also neg. angle for bend angle > 180°** is used.

Service Pack 1

Synchronize all sheet developments and generate new dimensions

The current function **Sync all developments** has been moved to the **Up to HiCAD 2016** area in the **Update** menu of the **Sheet development** function group. This function can be used to update the 2-D and 3-D developments as before.

With the new function **Sync all developments** replacing the old one (**Sheet Metal > Sheet development > Update > ...**) you can use a dialogue to update only 3-D developments (sheets and surfaces).



In the dialogue, you can use the **Part selection** options to determine which parts should be included in the update. If you want to update all sheets, select the **All** entry under **Sheet area**. If you only want to update the active sheet, select **Active sheet**.

Rearrange dimensions and annotations

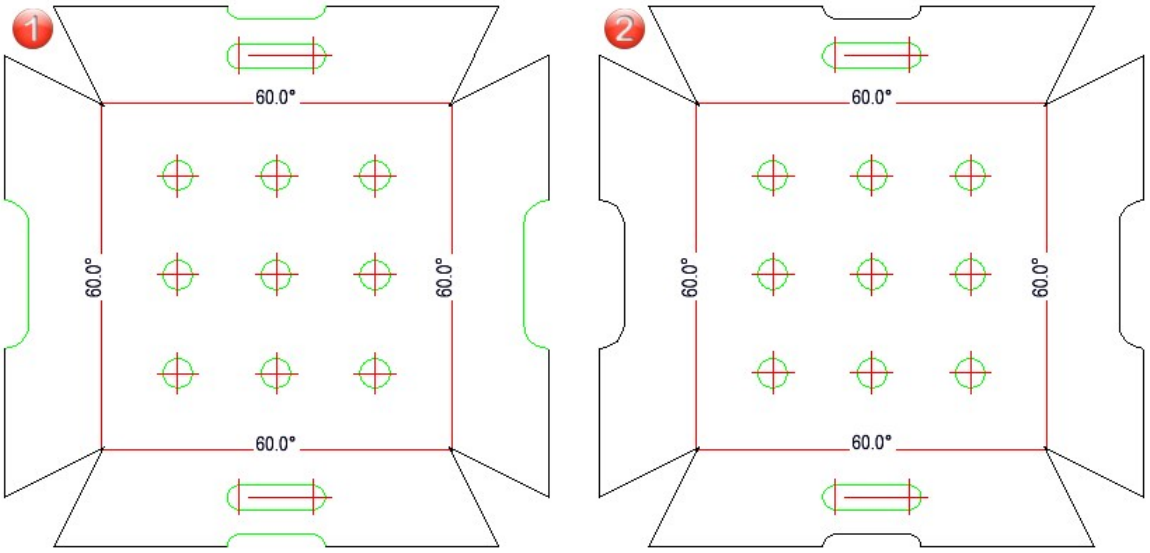
If you have created or arranged dimensions or annotations manually in developments, then you can specify that these dimensions/annotations and their arrangement are retained when the development is updated by deactivating the **Create new dimensions and annotations** checkbox. However, the contents are automatically corrected if necessary.

Please note:

When editing or processing the drawing, new dimension base points or annotations may be created or others may be removed. In such cases, it is not possible to retain the corresponding dimensions and annotations. This means that these dimensions and annotations will be recreated and you will have to adjust them again afterwards.

Colouring of standard processings in the development

Different colours can be defined for the outer contour and the standard processings in the development settings. If a standard processing is located on the edge, the lines belong simultaneously to the standard processing and to the outer contour. In this case, the colour of the outer contour is applied to the standard processing.



(1) Display of the outer contour before HiCAD 2025 SP1, (2) From SP1 onwards, standard parts on the contour are given the colour of the development edges.

Weight attribute

For sheet developments, the "simplified" weight of the minimal bounding rectangle is written to the **§CW**attribute (moved from **General** to **Sheet**). The §CW attribute evaluates the parameter **Consider recessed corners** at **System settings > Attribute management > Attribute calculation**. The value of **§CW** can change as a result.

Removal of the technology parameters

The function **Technology parameters**  (previously in the menu bar under New > Base) has been removed from the Sheet Metal module, as all parameters can be set directly in the dialogue or via the feature (under Clearance).

DXF export

Manufacturability check for Sheet Metal constructions

In the Configuration Editor, you can configure the test for the manufacturability of Sheet Metal constructions. The check can then be carried out when exporting sheet developments, or in the Design Checker. The following aspects can be checked:

- Distance of processings to bend zones
- Distance of processings to edge
- Maximum sheet dimensions
- Collisions in developments (with and without bend zones).

If you have activated the parameter **Carry out manufacturability check** at **Sheet Metal > Manufacturability check**, the checks selected under this parameter will be performed when developments are exported.

Any problems detected will be marked with a  symbol. The export will not be affected by this.

Direction of the outer contour when exporting DXF files

In order to avoid material damage, the start point should not be located in an inner corner (concave corner) of the outer contour when exporting the DXF file. In HiCAD, the edges of closed contours are therefore resorted during DXF export in such a way that the start point is not located in a concave corner.

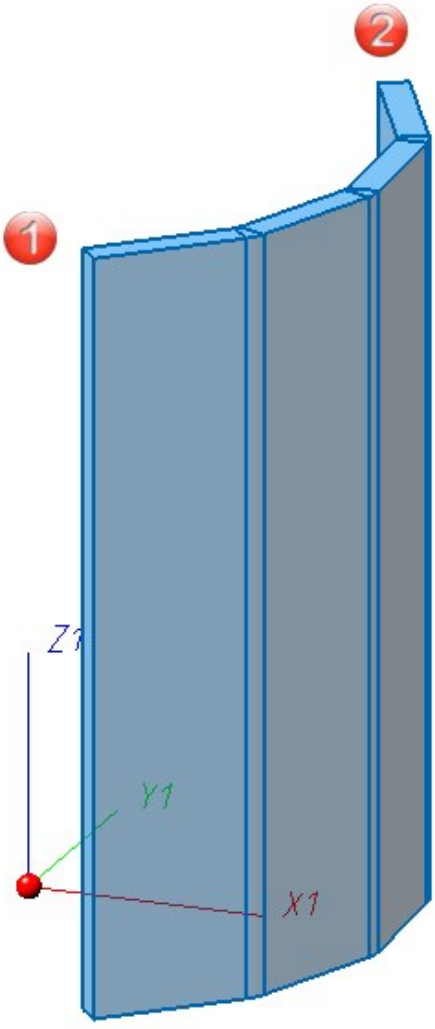
COBUS Default

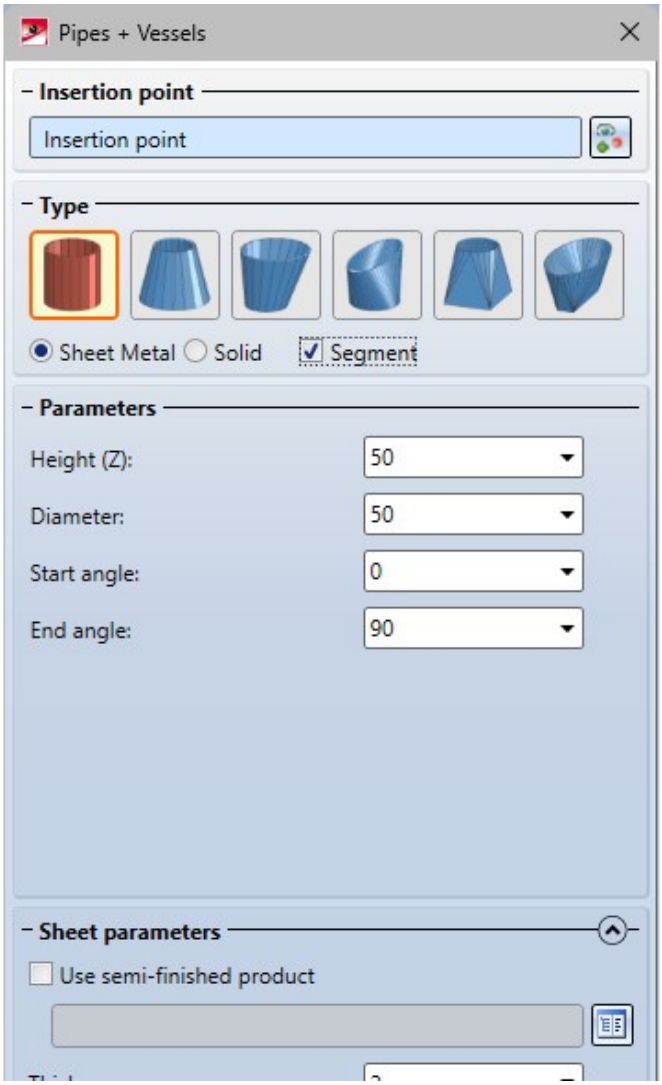
The configuration COBUS Default is now available at Settings for the DXF interface. It replaces the old HCADACAD_COBUS.DAT file. The configuration cannot be changed.

Major Release

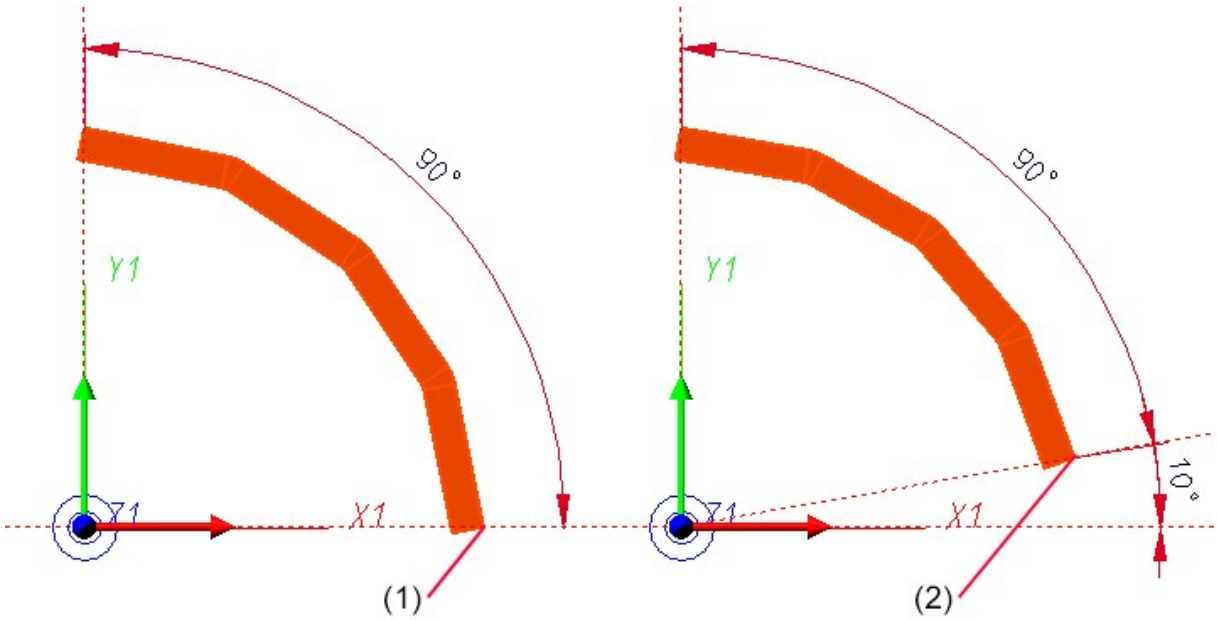
Segments for pipes

You can now use the **Pipes + Vessels**  function to determine pipe segments to be drawn via circles and ellipses. To do this, the **Sheet Metal** and **Segment** option must be activated under **Type**. An input window for the **Start angle** and **End angle** of the segment is then available in the Parameter area.



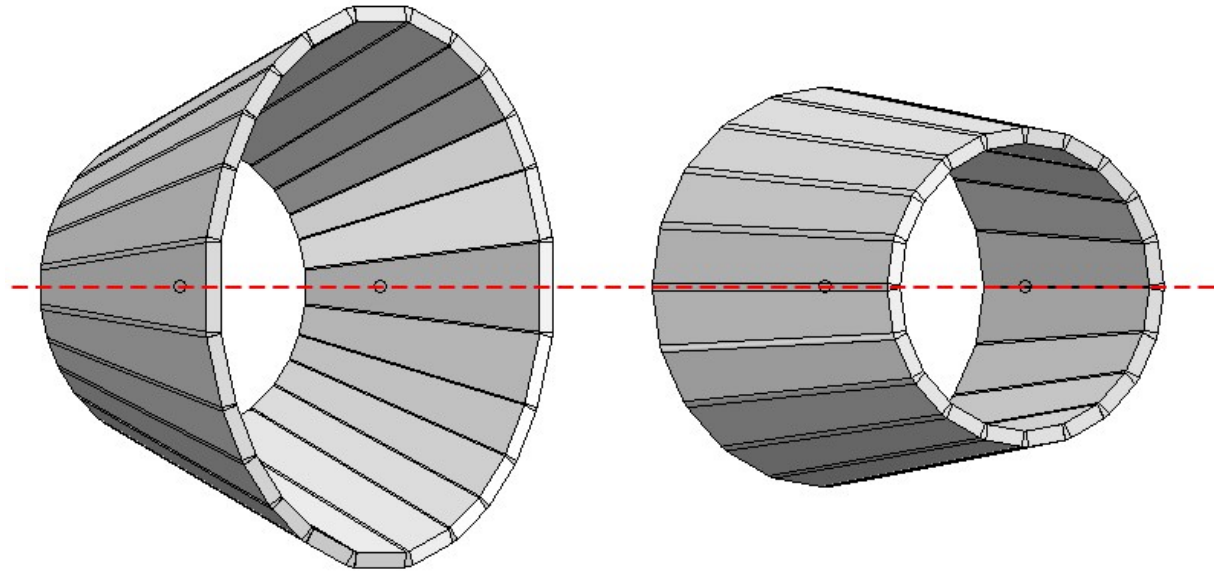


(1) Start angle 30°, (2) End angle 130°



(1) Segment start point 0°, (2) Start point 10°

For pipes and vessels with two fillets, either the flange is centred on the quadrant point or the centre of a bend zone.



(1) Position of pipes before HiCAD 2025, (2) as of HiCAD 2025

Please note:

The function **Pipes + Vessels** for versions before HiCAD 2024 SP1 was removed.

Multiple selection of Sheet Metal parts

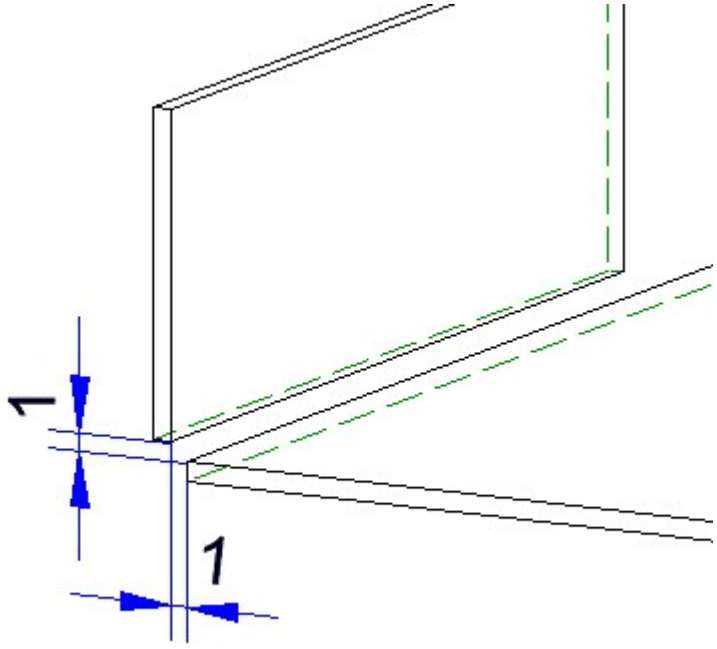
From HiCAD 2025, the sheet metal main part is included in the part list instead of the flange and bend zone when making multiple selections (Ctrl + LMB). If the flange or the bend zone of a sheet metal main part is selected, both sheet metal main parts are completely transferred to the part list when another sheet metal part is identified with Ctrl + LMB. The only exception is if the sheet metal part does not have a feature log. In this case, you can continue to transfer flanges and bend zones to the part list.

Universal attaching of flanges

To improve user-friendliness, the **Attach flange** dialogue has been revised and the following functions have been added:

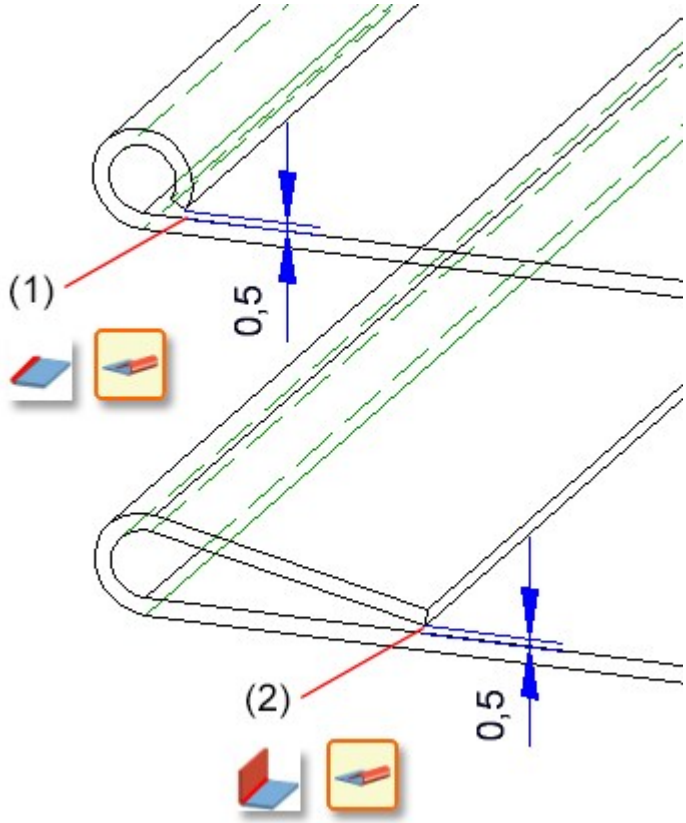
- Attach bend zone , Attach bend zone (2 pts.)
- Flange, without bend zone , Flange, without bend zone (2 pts.)
- Create border lip , Border lip (2 pts.)
- Hinge , Hinge (2 pts.)

The option **Without flange** is available in the upper area. Only the bend zone is created here by identifying the connecting edge. By activating the option for the **1st point**, the width of the new bend zone can be freely selected, regardless of the connecting edge. If you have selected the 1st point in the drawing, you can set the width and the alignment of the width; alternatively, you can identify a 2nd point. The **No bend zone** option has been added to the **Mode** area, which can be used to add flanges and sketches without a bend zone, taking the clearance into account.



Clearance 1

With the new option for the bend angle **Distance to connecting flange**  , you can create border lips in conjunction with the connection type **Attach flange**  , and hinges in conjunction with the connection type **Without flange**  . The width of the attachment is determined by the connecting edge or 2 points.



(1) Bend zones without flanges, Distance to connecting surface 0,5
(2) Flange attached with a distance of 0,5 to the connecting surface

By merging the various Attach functions, the following functions can now be activated directly in the menu bar:

- **Z-fold** 
- **Attach flange to surface** 

Selection behaviour changed

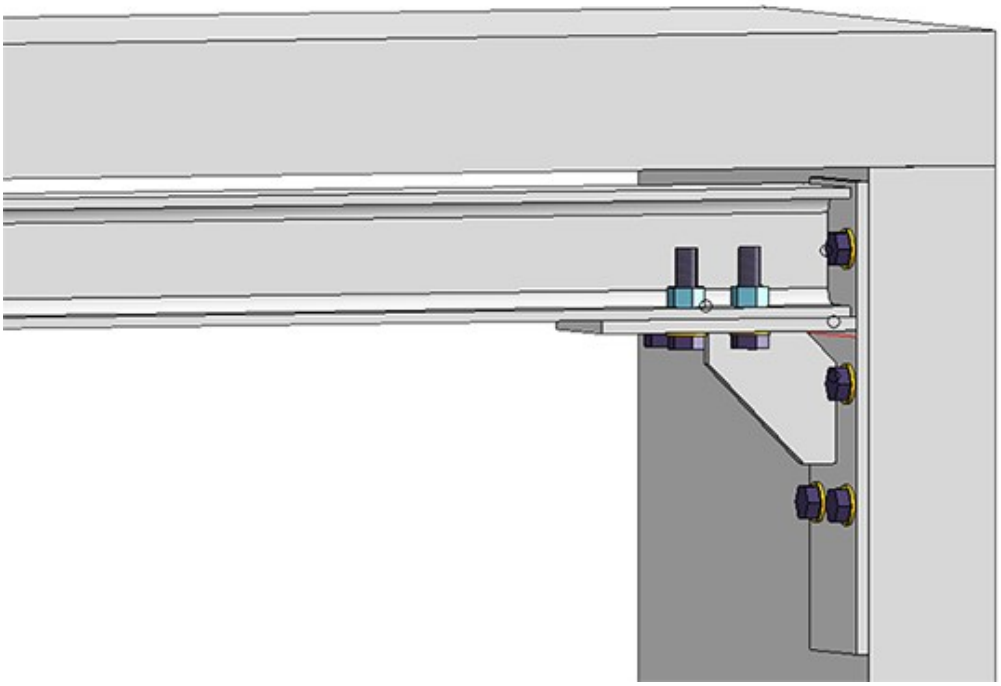
The selection behaviour for the **Attach**, **Z-fold**, **Insert bend zone**, Attach flanges along sketch and **Flanges along sketch, to surface** functions has been adapted to user requirements. This means that after selecting a connecting edge, for example, you do not have to select the  icon in the dialogue to change it. You can directly select a changed connecting edge as long as you do not switch to the dialogue. This behaviour applies to edges, points, surfaces and sketches.

Steel Engineering

Service Pack 2

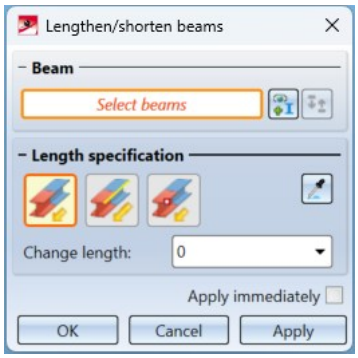
New Steel Engineering connection: Bracket (2202)

The functions for Steel Engineering in the "Civil Engineering functions" docking window have been extended to include a bracket connection. The **Bracket (2202)** function can be used to attach a beam to the substructure or a wall. It can be applied to two beams or to a beam and another part. The fastening is achieved using a total of three plates. These are then combined with any weld seams that may have been created to form the **Bracket** assembly.



Revision of the functions for lengthening beams

The dialogue window of the function **Steel Engineering > Lengthen > Change length**  has been revised and modernised:



- The functions **Change beam length, via new total length**  and **Change beam length, to point**  have been embedded in the dialogue and can be selected in the Length specification area.
- In addition, it is possible to use the  icon to adopt a new total length from a reference beam.
- The new dialogue allows you to select several beams at once and change their length.

Manufacturability check in the Design Checker

The **Manufacturability check** in the Design Checker is a summary of several checks for Sheet Metal parts that you can compile in the Configuration Editor at Sheet Metal > Manufacturability Check.

You can also use the following checks to investigate Steel Engineering plates:

- **Distance of processings to edge**,
- **Maximum sheet dimensions**,
- **Collisions in developments**,
- **Distance between processings**,
- **Minimum distance for standard bores**.

For the reference values of Steel Engineering plates, the catalogues at **Semi-finished products > Plates** have been expanded to include the corresponding columns.

Bar list: Non-numeric values for item texts

When creating bar lists, the item number and length of the beam are displayed on the beams by default. However, other attributes can be displayed instead of the item number. Previously, an error message was displayed for attributes whose value was text rather than a number. From HiCAD 2025 SP 2 onwards, it is also possible to select attributes that contain text as annotations.

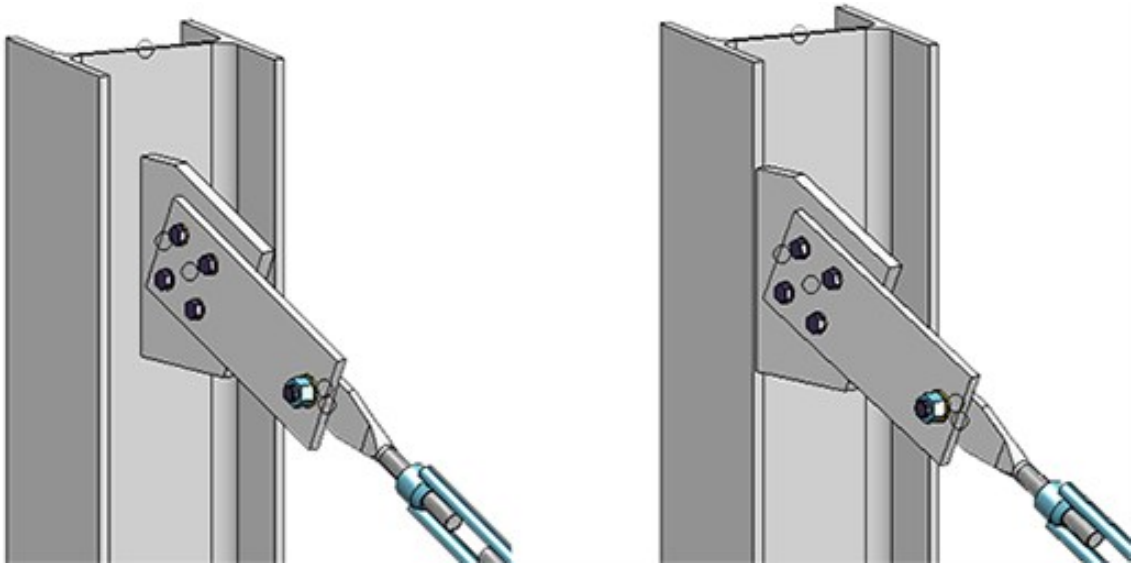
Steel Engineering favourites in imperial units

If **Imperial** is selected under **Regional setting** in the **Parameter configuration** during the HiCAD installation, the Steel Engineering favourites provided will be adjusted accordingly in ProgramData\ISD Software und Systeme\HiCAD 2025\FAVOURITES\STEELENGINEERING.

Service Pack 1

Extension for the turnbuckle with blade screw for Cross-bracing 2601

For the civil engineering function **Steel Engineering > Connections > Cross bracing (2601)**, it is now possible to attach the turnbuckle with blade screws to the connecting points using the gusset plates on the flange. After creating the turnbuckles, activate the **Mount to flange** checkbox in the **Gusset plate** area on the **Connecting points** tab of the dialogue window.



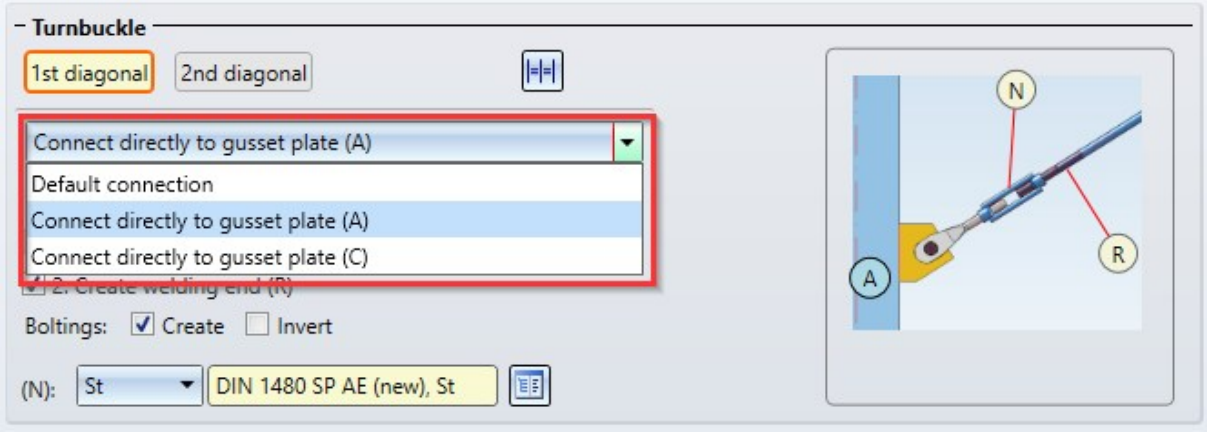
Left: Gusset plate mounted to web, right: Gusset plate mounted to front flange

You create the turnbuckle on the **Fixing** tab in the **Turnbuckle** area. In the **Diagonals** area, you can set on this tab whether the gusset plate should be attached to the front or rear flange.

Cross-bracing 2602

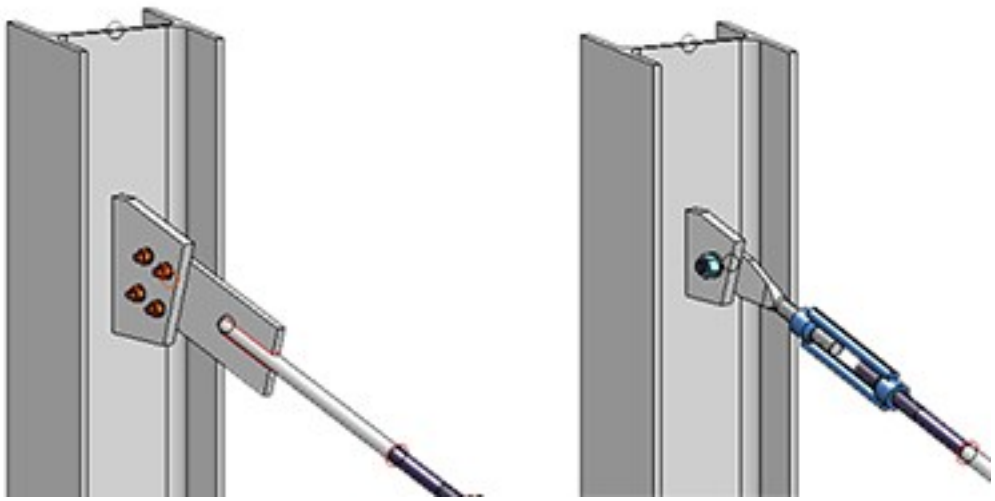
Extension for the turnbuckle with blade screw

For the civil engineering function **Steel Engineering > Connections > Cross-bracing (2602)**, a new option has been added for the turnbuckle with a blade screw:



Previously, the turnbuckles' blade screws were connected to the gusset plates via a connecting plate. A selection list has been added to the **Fixing** tab, where you can choose to connect the blade screw to the gusset plate on one side of the two diagonals without a connecting plate, in addition to the **Default connection**. On the other side of the diagonal, the blade screw is still connected to the gusset plate via a connecting plate.

For the new variant, select the **Connect directly to gusset plate (A)** or **(C)** option for the 1st diagonal and the **Connect directly to gusset plate (B)** or **(D)** option for the 2nd diagonal on the **Fixing** tab, under **Turnbuckle**.

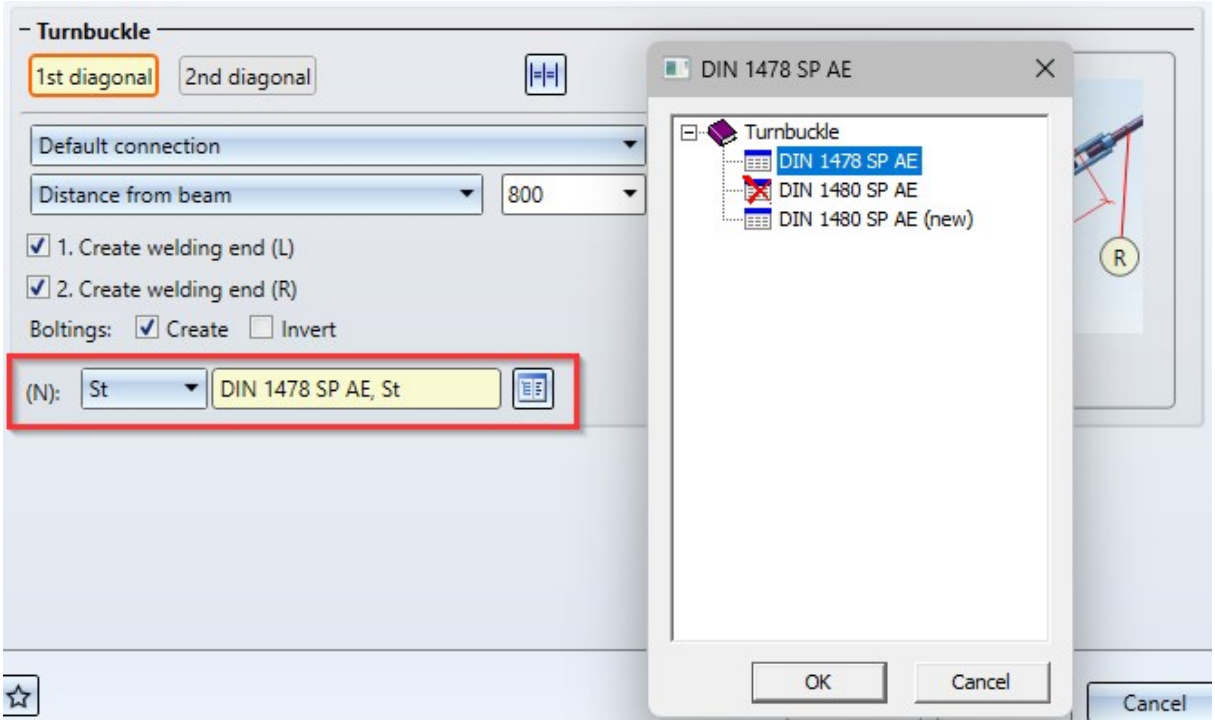


Left: **Default connection** option, Right: **Connect directly to gusset plate** option

On the **Connecting points** tab, you can make further settings for the turnbuckle variant. Under **Chamfers + Fillets**, you can set whether the connecting plates used should be filled. Under **Connection**, you can determine for each individual connecting point (A to D) that the gusset plate is connected to the beam at the flange instead of at the web by selecting the **Mount to flange** checkbox.

Closed turnbuckle form

In addition to the open form of the turnbuckle nut DIN 1480, the closed form of the turnbuckle nut DIN 1478 has been added to the catalogue on the **Fixing** tab, under **Turnbuckle**.

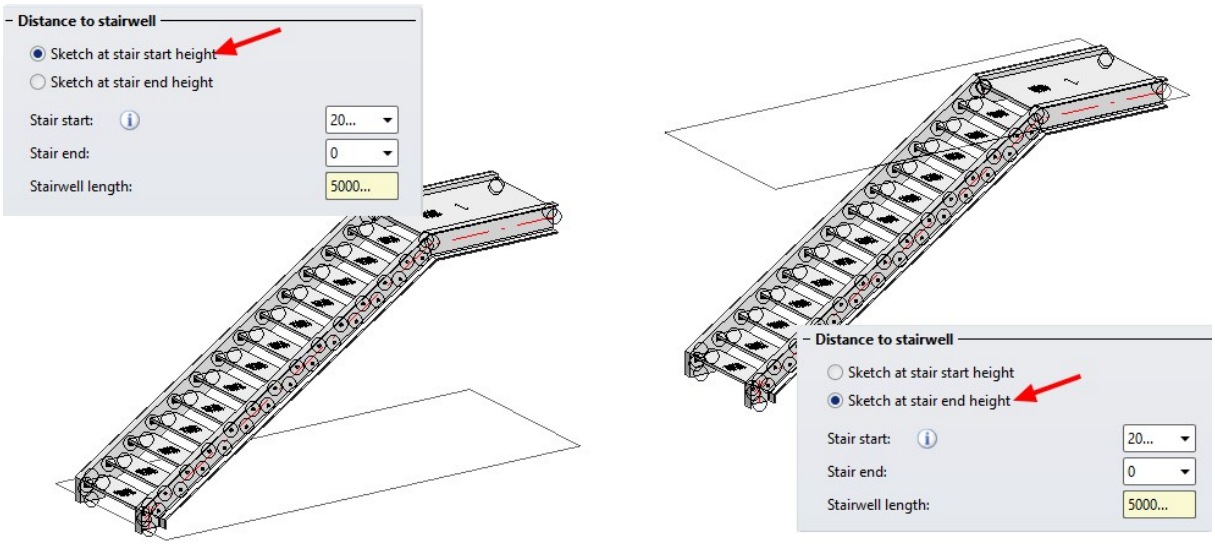


HiCAD-Trepedia interface

In order to facilitate access to the interface of HiCAD with the **Trepedia** program, the **Trepedia** function has been added to the **Steel Engineering > Further functions** function group, in the pull-down menu of the **Change representation, individual** function. Selecting the function starts Trepedia. After the corresponding stairs have been generated, the resulting model can be inserted into HiCAD as an IFC or STEP file.

New setting for the stairwell contour in the Staircase Configurator

When inserting stairs using the Staircase Configurator with a stairwell contour, the position of the sketch of the stairwell contour was previously automatically set to the height of the stair start. The **Calculation** tab of the dialogue window has been extended in the **Distance to stairwell** area to include the options **Sketch at stair start height** and **Sketch at stair end height**. The option **Sketch at stair end height** sets the position of the sketch to the height of the stair end.



Attribute calculation

In the HiCAD Configuration Editor, you could previously set whether the accurate calculation (e.g. with notches or subtractions) or the calculation of the minimum bounding rectangle (length x width x thickness) should be displayed for the standard attributes (§01 Weight, §10 Surface area, §18 Commercial Weight, §20 Volume). From HiCAD SP1, new geometry attributes are available for calculating the bounding rectangle at **System settings > Attribute management > Attributes**, so that the various calculation results are available at the same time

For beams and profiles there are the new attributes:

- **§WBL**, Weight by length
- **§CBL**, Commercial weight by length
- **§SBL**, Surface area by length
- **§VBL**, Volume by length

and for plates and sheets:

- **§CBA**, Commercial weight from rectangular area
- **§SBA**, Surface area from rectangular area
- **§VBA**, Volume from rectangular area

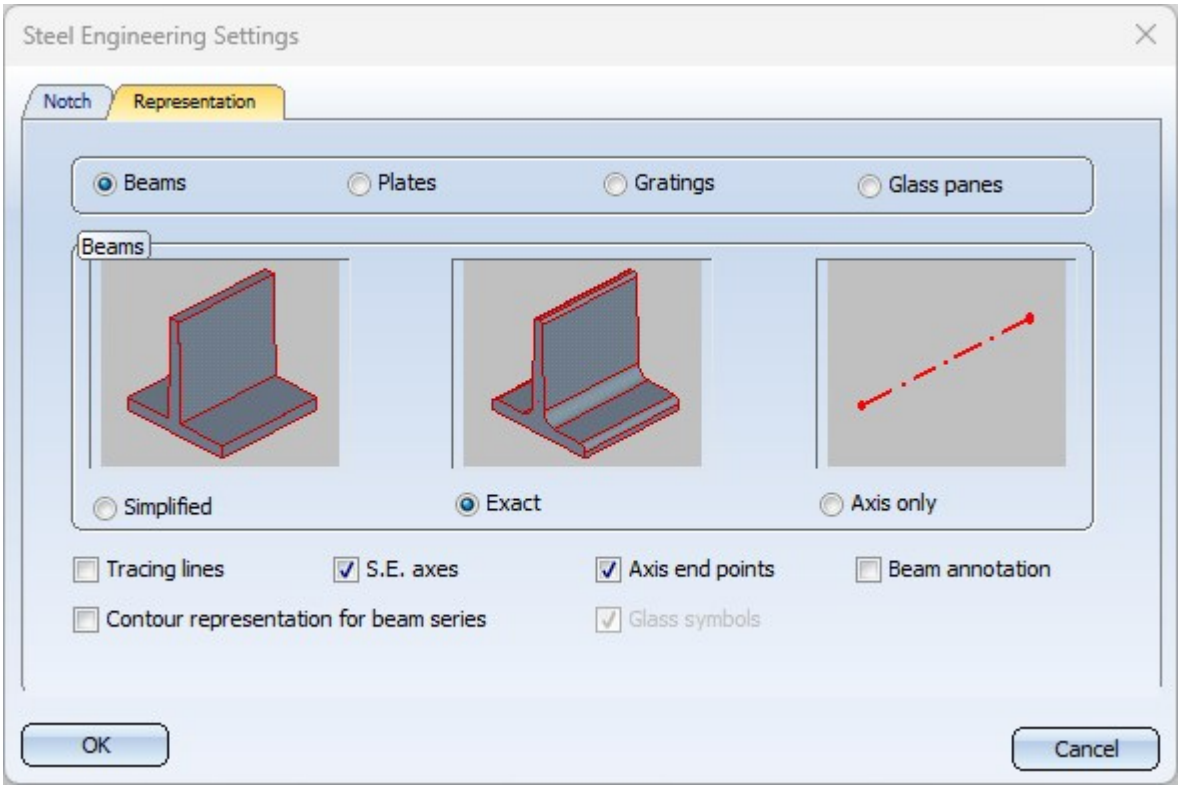
In the Configuration Editor at **System settings > Attribute management > Attribute calculation**, the following parameters have been changed as follows:

- **Type of weight calculation for SE beams** to **Transfer geometry attributes by length to**
- **Type of weight calculation for SE plates** to **Geometry attributes from rectangular area transferred to**

Here you have the option to transfer the calculation of the minimum bounding rectangle not only to the new geometry attributes (§WBL, §CBL, §SBL, §VBL or §CBA, §SBA, §VBA), but also to the standard attributes (§01, §10, §18, §20).

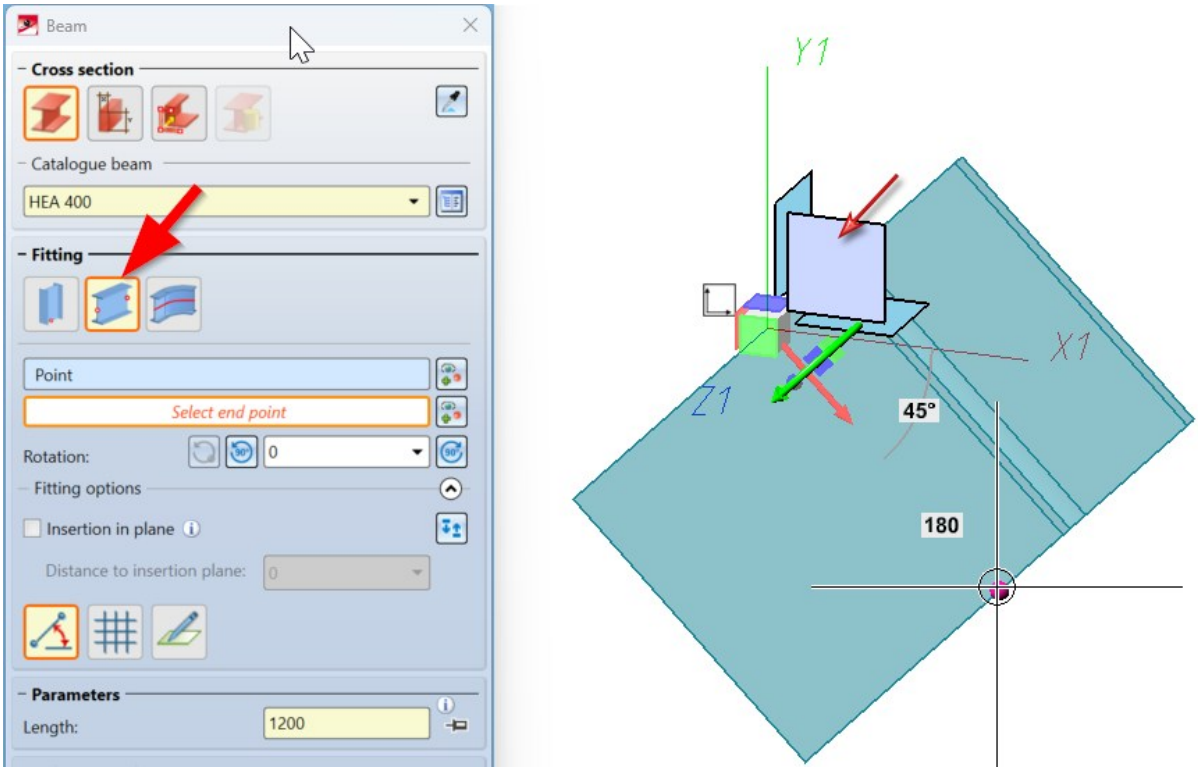
Due to the extensions in the Configuration Editor, the **Weight calculation** tab of the **Steel Engineering Settings** dialogue (**Steel Engineering > Further functions > Settings** ) was no longer needed and was removed. The settings on the tab only had a temporary effect until the next restart of HiCAD.

The dialogue window now looks like this:



Insert new beam with processing plane suggestion

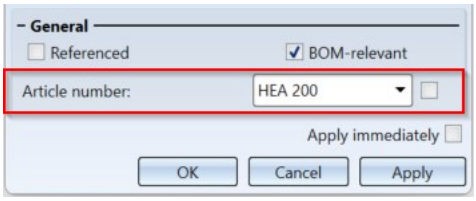
The insertion method **Insertion via 2 points** of the function **Insert new beam** has changed in that a processing plane selection is available after the start point has been determined. These three processing planes represent the current coordinate system. Selecting a processing plane changes the axis in which you can insert the beam.



Manual change of the article number for semi-finished products

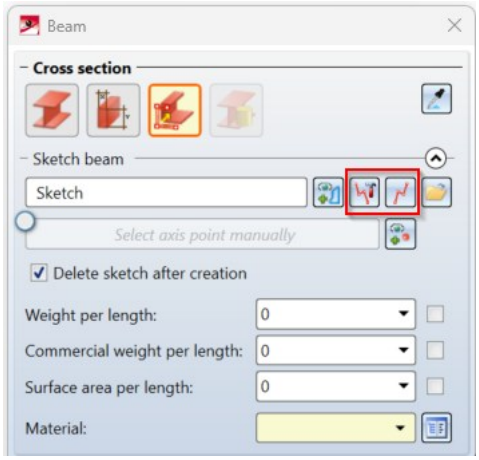
Previously, when a feature was recalculated, the manual change to the article number of beams, profiles, plates and sheets in Steel Engineering was overwritten with the attribute **BZ**, the catalogue designation. From now on, a manual change to the article number will be retained even after the feature has been recalculated.

In this context, an input field for manually changing the article number has been added in the **General** area of the **Insert new beam** dialogue, namely, for the Cross-sections **Prototype beam**, **Beam from (multi-part) sketch**, and for the **Rectangular plate** function. If the corresponding checkbox is active, the article number will adopted from the catalogue.



Further options for beam insertion from sketches

When using the **Insert new beam** function, you can now create a new sketch or change the selected sketch when the dialogue window is open for the cross-section **Beam from (multi-part) sketch**. The buttons **New sketch in plane** and **Process sketch** have been added for this purpose.

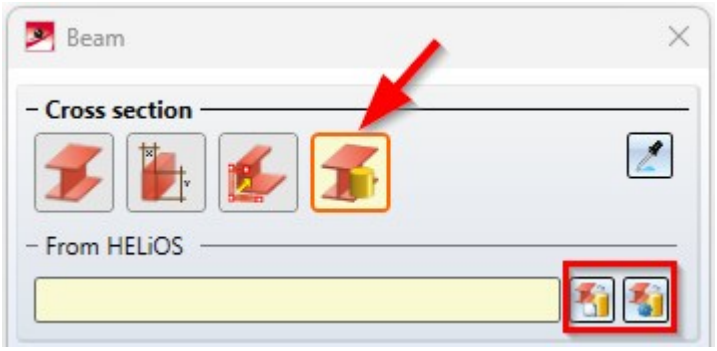


Major Release

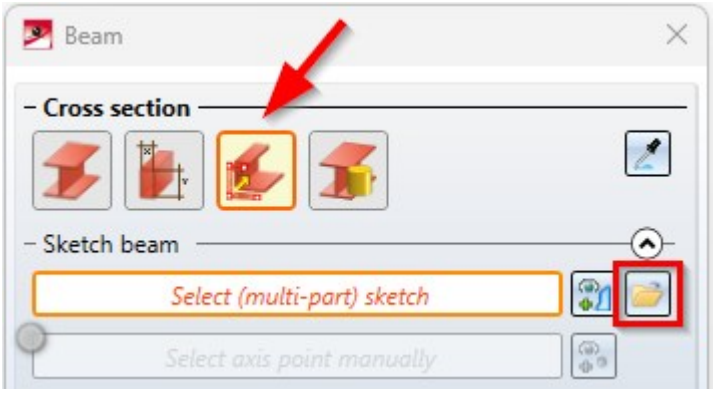
Redesigned beam insertion dialogue

The functions for inserting series beams via **Steel Engineering > New** have been added to the dialogue of the **Insert new beam** function:

- The functions **Insert series, via DB document master** and **Insert series, via DB article master** can be found in the dialogue window of the **Insert new beam** function, in the **Cross section** area under the **From HELiOS** option. The buttons **Select HELiOS document** and **Select HELiOS article** are located to the right of the selection list.

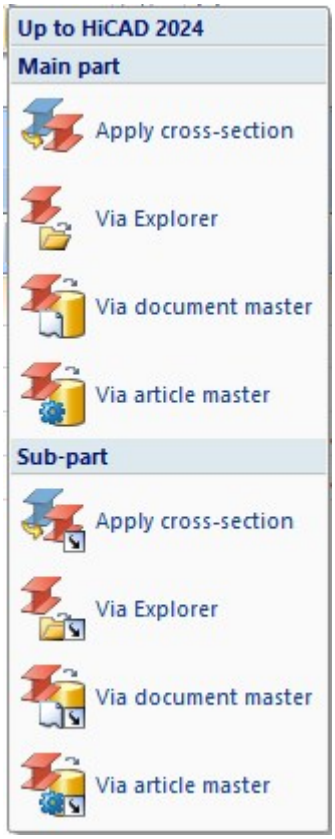


- The function **Insert series, via Explorer** can be found in the **Cross section** area via the option **From (multi-part) sketch**. The button **Load sketch from file** has been added next to the button **Select (multi-part) sketch**.



- The icon of the option **Adopt cross-section from reference beam** was changed from a selection symbol to a pipette symbol.

The old functions have been removed from the location **Steel Engineering > New** and can now be found in the pull-down menu of the **Insert new beam** function in the sub-menu **Up to HiCAD 2024**:



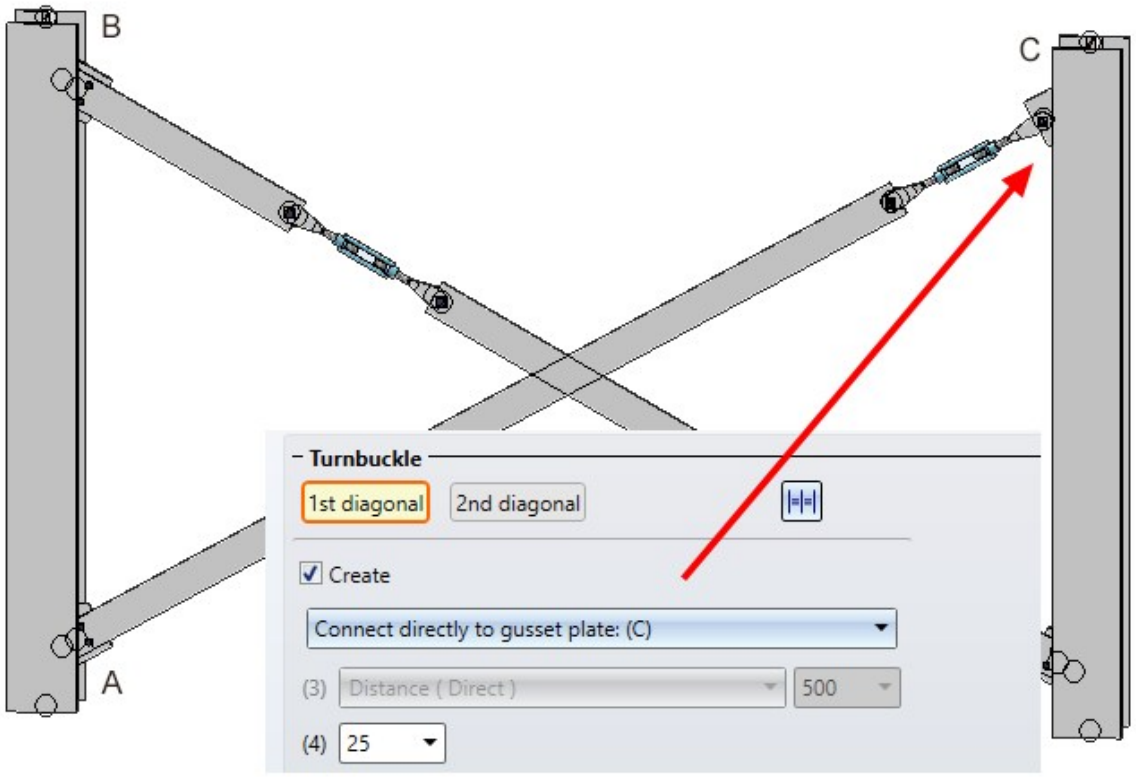
The **Up to HiCAD 2022** section in the drop-down menu of the function **Insert new beam** was removed. This section contained old HiCAD functions that have been integrated in the dialogue of the new function Insert new beam since HiCAD 2023 SP2 and HiCAD 2024. This affects the functions **Insert standard beam**, **Multi-part standard beams**, **Insert prototype beam, value input via dialogue**, **Prototype beam from catalogue**, **Series beam from catalogue**, **Elongated plates**, **Beam from sketch** and **Multi-part beam from sketch**. A table with short descriptions of how to access the respective functions via the new dialogue window can be found in the topic **Insert New Beam** under the paragraph **Functions for beam insertion up to HiCAD 2022 and 2024**.

New connection option for turnbuckle in Cross-bracing (2601)

When using the Civil Engineering function **Steel Engineering > Connections > Web/flange to web/flange > Cross-bracing (2601)**, you can now select whether the blade screw of the turnbuckle should be connected directly to the gusset plate or, as before, connected to the gusset plate via an additional connection plate when inserting turnbuckles.

To do this, select the desired variant in the dialogue window of the function in the **Fixing** tab under **Turnbuckle**:

- **Default connection**: The turnbuckle is connected to the gusset plate at both ends of the diagonal via a connecting plate.
- **Connect directly to gusset plate (A)**: The turnbuckle is connected directly to the gusset plate at point **A** via the blade screw and to the gusset plate at point **C** via a connecting plate (the options ending on **(B)**, **(C)** and **(D)** behave in the same way).



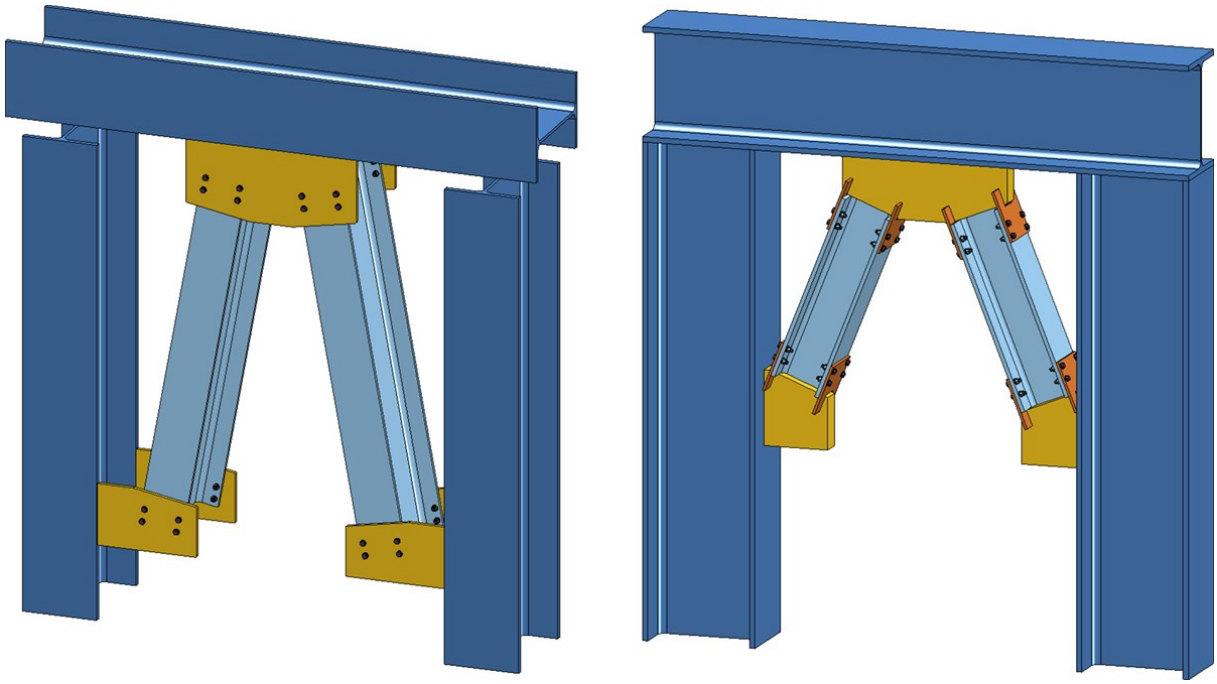
The turnbuckle of the first diagonal is connected directly to the gusset plate at point C, and the turnbuckle of the second diagonal is connected at both ends via a connecting plate.

K-bracing with I-beams

A new **K-bracing** variant is available in the **Civil Engineering functions** at **Steel Engineering > Connections > Web/-flange to web/flange**. For bracings between three I-beams (columns and beams), this variant can be used to install K-bracings consisting of gusset plates, I-beams and - if selected - connecting plates (slotted plates).

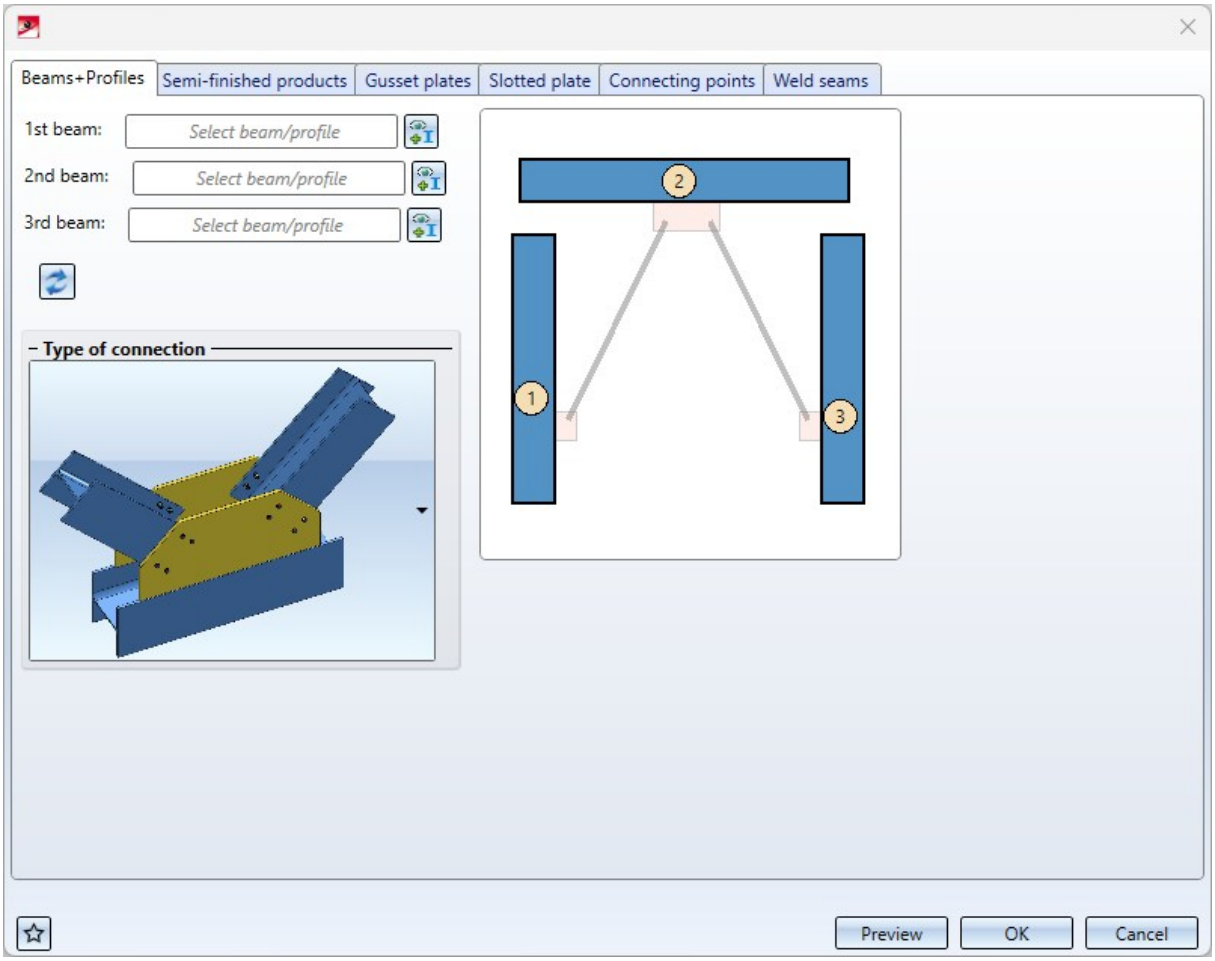
Depending on the position of the beams to be connected, different types of connection are available:

- The bracing members are welded to the gusset plate.
- A slotted plate is bolted to the bracing members and welded to the gusset plate.
- The bracing members and gusset plates are bolted together.



Left: Bolting of bracing members and gusset plates, Right: Bracing with slotted plates

After calling up the function, the **K-bracing** dialogue window is displayed:

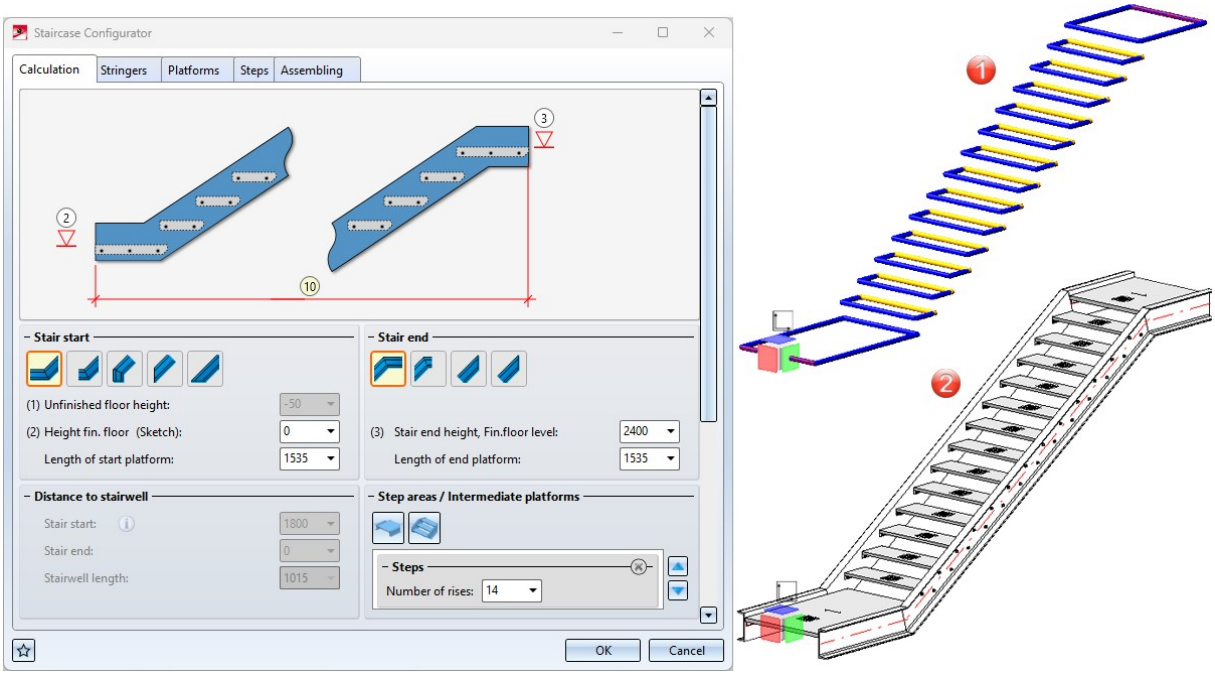


Define grid

The **3-D Grid** function that was previously available via **Steel Engineering > Further functions > Settings > 3-D Grid** has been moved. The function has been completely revised and can now only be accessed via **Drawing > Others > Extras > 3-D Grid**  .

Staircase configurator - Simplified preview

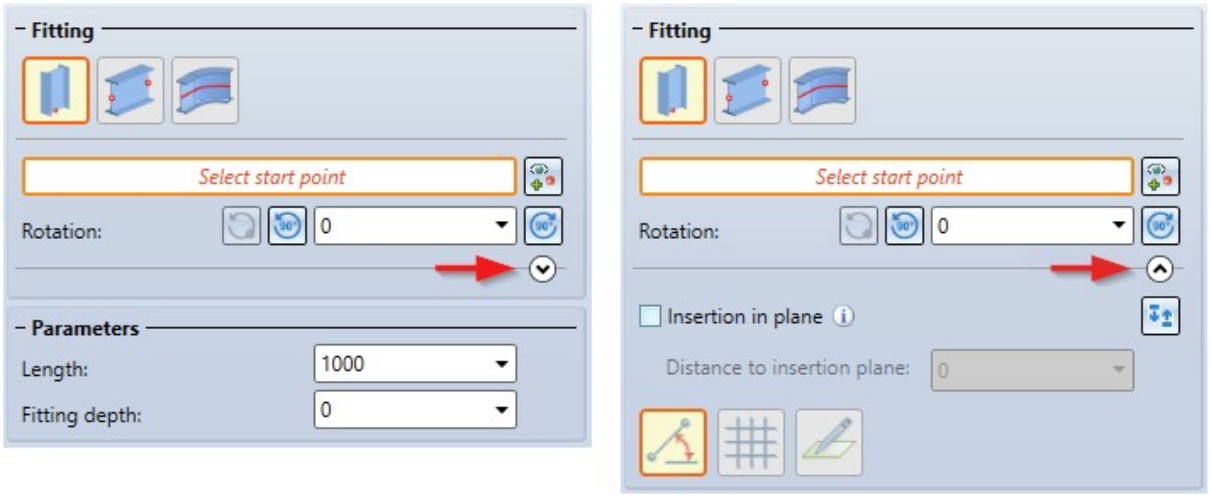
When inserting stairs with the **Staircase Configurator**, a simplified preview of the staircase is now displayed while the dialogue is open- depending on the current settings. This applies both for insertion with and without selecting the stairwell. When working without a stairwell, the preview is placed in the origin of the world coordinate system.



(1) Simplified preview, (2) Result

Beam insertion - Revised dialogue

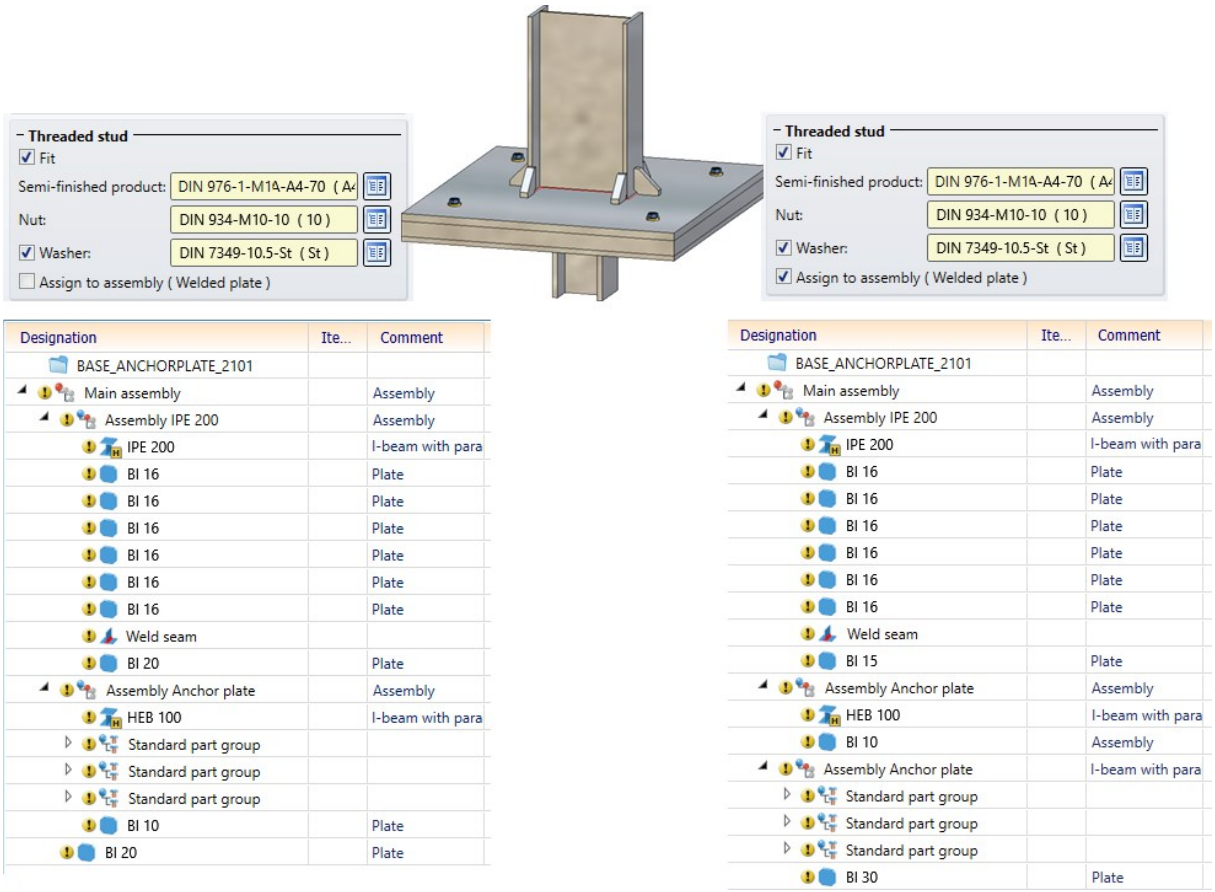
The dialogue window of the **Beam** function has been changed in the **Fitting** area. The fitting options can now be collapsed or expanded by clicking on and .



Base plate + Anchor plate (2101)

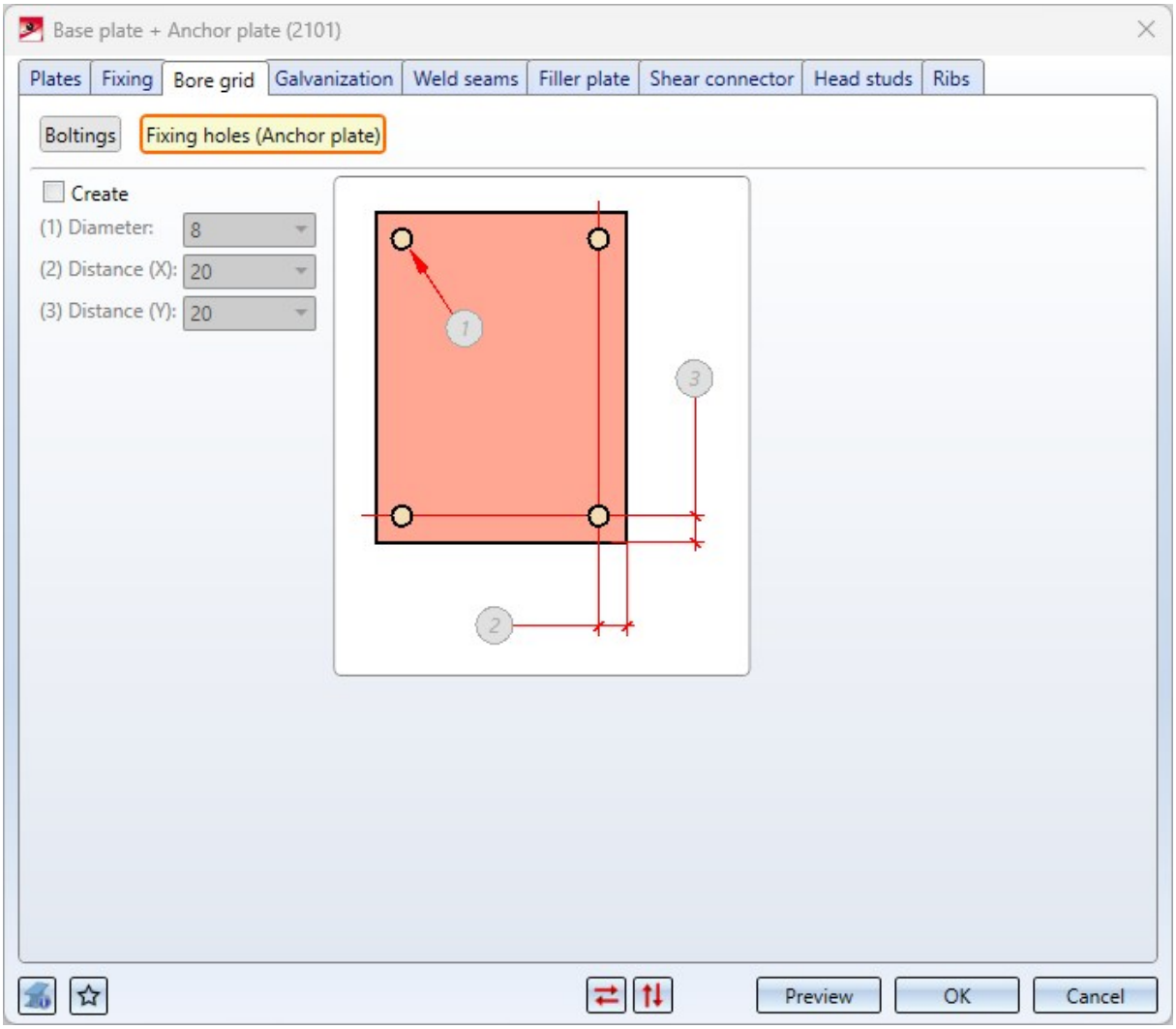
Threaded stud

If the base plate and welded plate are connected to each other by threaded studs, the various standard parts (studs, nuts, washers) are each combined in an assembly with the name **Standard part group**. These assemblies were previously always assigned to the assembly with the anchor plate. As of HiCAD 2025, it is now possible for continuous threaded studs (e.g. DIN 976-1) to assign the corresponding standard part groups to the assembly of the welded plate. A corresponding checkbox has been added to the **Fixing** tab for this purpose. If the checkbox is deactivated, the welded plate is installed as an individual part.



Fixing holes

Fixing holes can only be created for the anchor plate. As of HiCAD 2025, the four fixing holes are positioned relative to the edges of the plate. Specify the diameter as well as the horizontal and vertical distance between the centre of the hole and the edge of the plate.



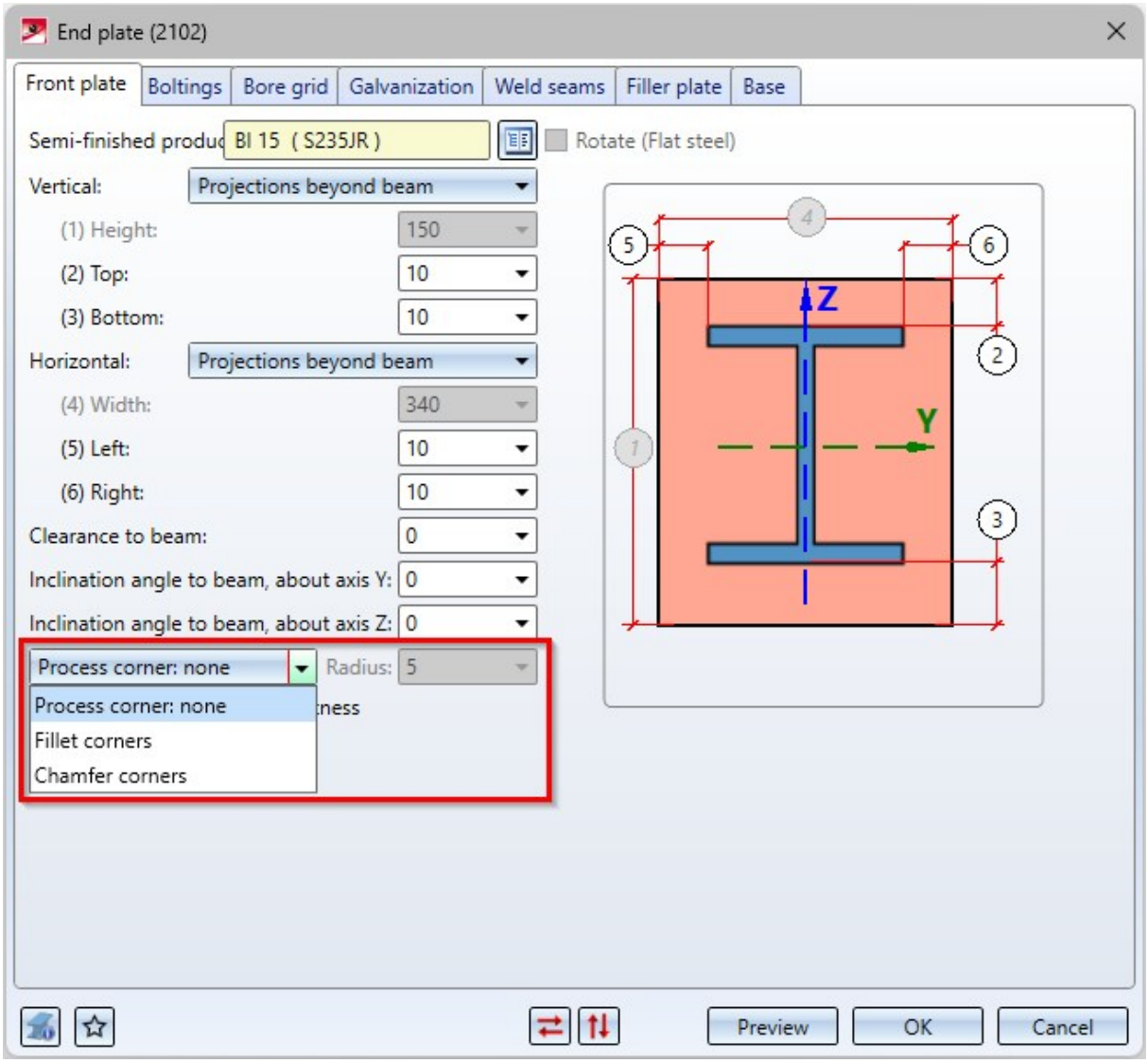
Weight from rectangular area of development

As of HiCAD 2025, the attribute **Weight from rectangular area of development** (§CW) is also calculated for Steel Engineering plates and elongated plates in the same way as for Sheet Metal parts, if the setting is set accordingly in Configuration Editor at **Modelling > Part properties** in the **Sheets or plates** section.

This does not apply to gratings and glass panes.

End plate (2102) - Chamfered front plate

For the End plate (2102), chamfering is now also possible as corner processing of the end plate. A corresponding selection box has been added to the **Front plate** tab for this purpose.



The chamfer angle is 45°, the length can be selected freely.

Drawing Management

Service Pack 1

HELiOS settings in the Configuration Editor

In order to increase user-friendliness, the HELiOS settings for Drawing Management have been restructured.

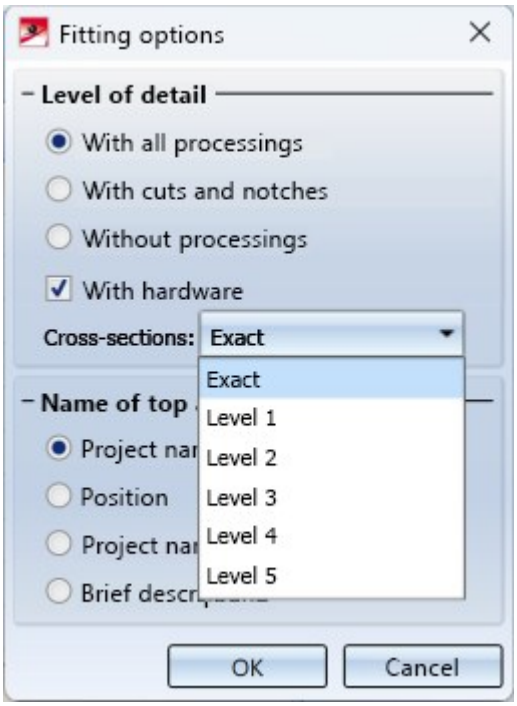
- The directory **PDM** can no longer be found directly under **Active configuration (Base configuration)**, but under **System settings**.
- The settings under **PDM** > **HiCAD - HELiOS interface** and under **System settings** > **HELiOS** have been combined at **System settings** > **PDM** > **HiCAD - HELiOS interface**.
- The settings under **PDM** > **HiCAD - HELiOS interface** > **Product structure** can now be found at **System settings** > **PDM** > **HiCAD - HELiOS interface**.
- The setting **Transfer product structure attributes to part attributes when updating HELiOS attributes** can be found at **Compatibility** > **HiCAD - HELiOS interface**. There you can also find the new setting **Allow loading/saving with interrupted HELiOS connection**.

Metal Engineering

Service Pack 2

LogiKal-Import: Detail levels of cross-sections

In the **Fitting options** window for importing from LogiKal, you can specify the level of detail for items:



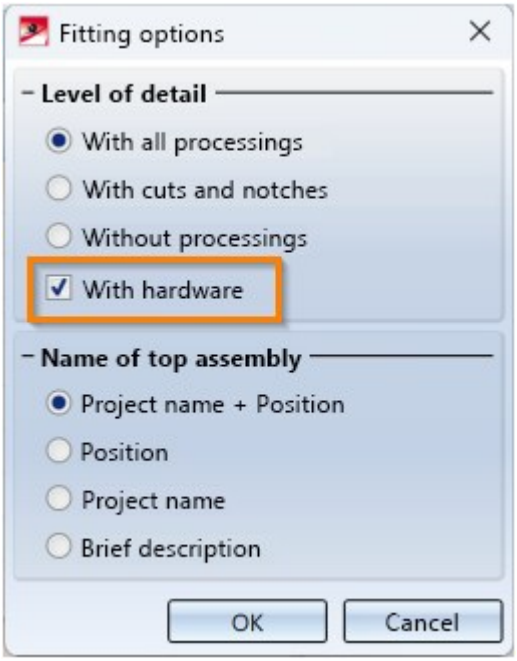
In addition to an exact representation, you can choose between five further detail levels in the pull-down menu in LogiKal.

In earlier versions of HICAD, it was possible to configure the level of detail, but this has now been simplified for the user by integrating it into the fitting dialogue.

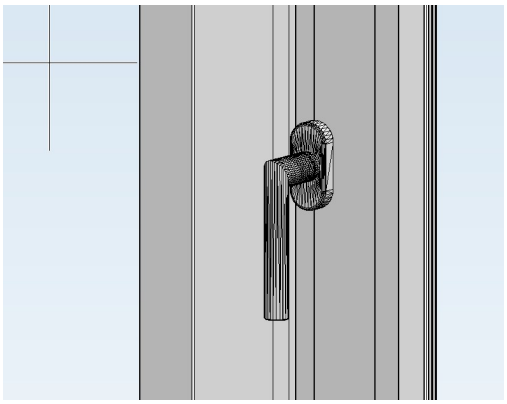
Service Pack 1

LogiKal import, with hardware

The **With hardware** option has been added to the **Fitting options** dialogue window for importing from LogiKal.



If it is active, the fittings, i.e. the handles plus accessories, are also generated from the corresponding .OBJ files from LogiKal in HiCAD:



LogiKal version 12.4. or higher is required for this.

Major Release

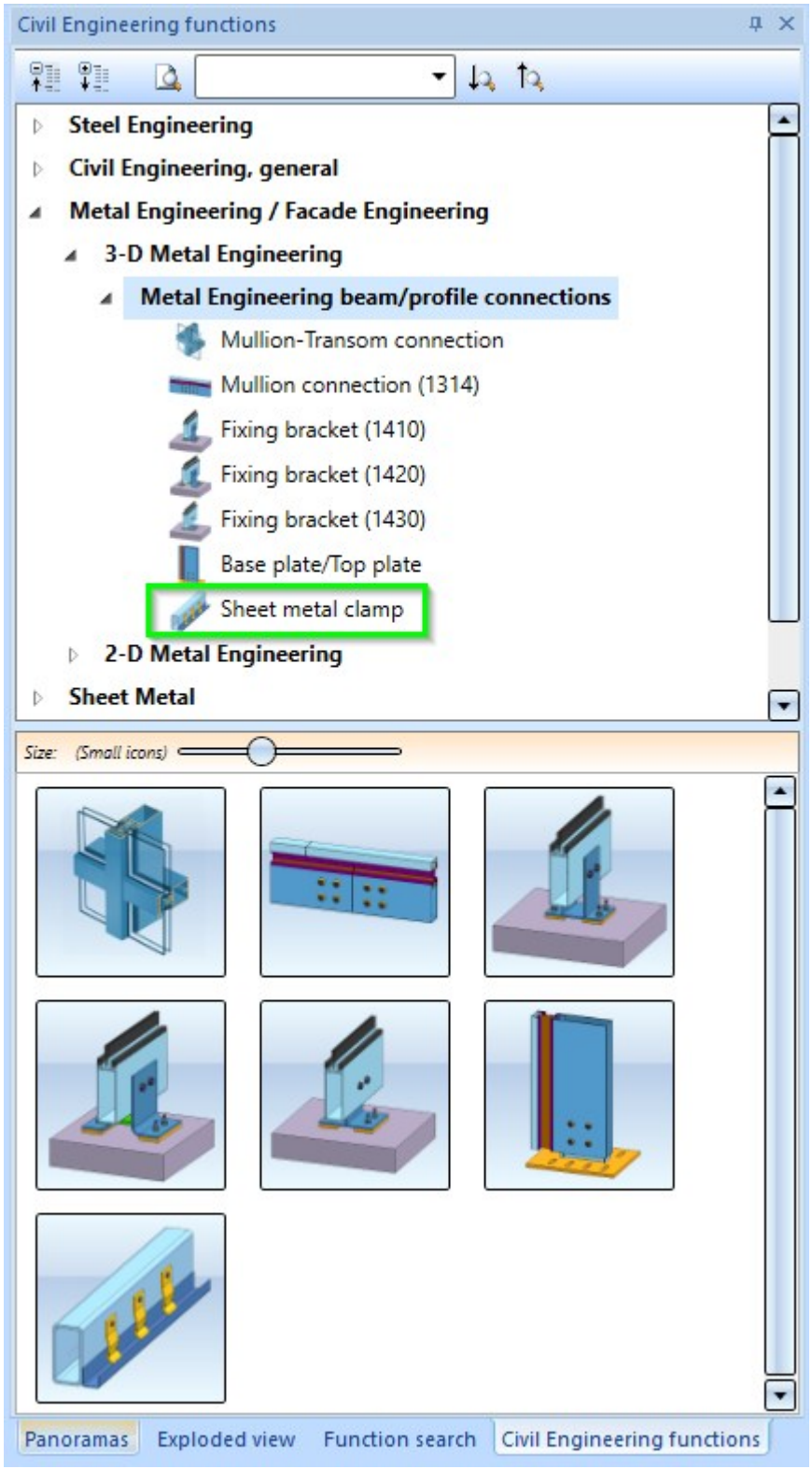
Improved algorithm for insulation hatching

The algorithm for insulation hatching has been significantly improved. From HiCAD 2025, the inset hatching follows the geometry of the part.

For further information please read the notes in the 3-D News.

Sheet metal clamp

The Civil Engineering function **Sheet metal clamp**, which was previously only available in the Steel Engineering area in earlier HiCAD versions and is mainly used for facade engineering, is now also available if you work with the Metal Engineering module when you update to HiCAD 2025.

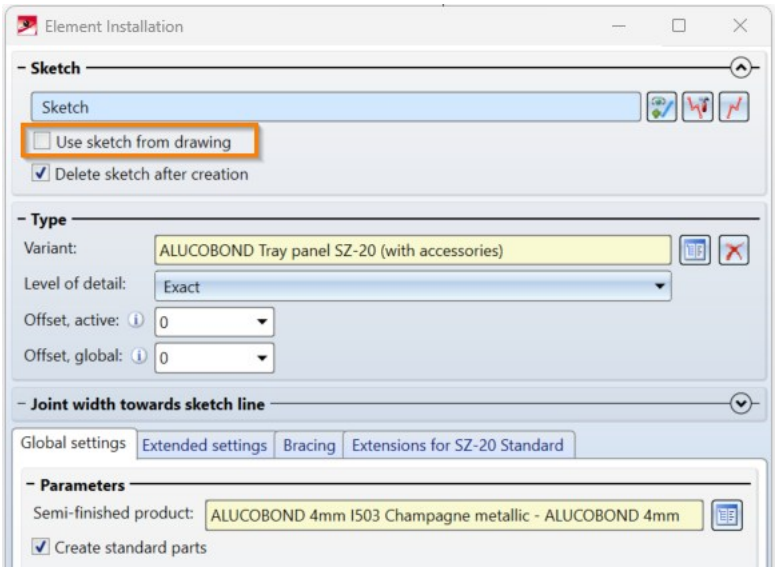


Layout Planning

Service Pack 2

External references for the sketch

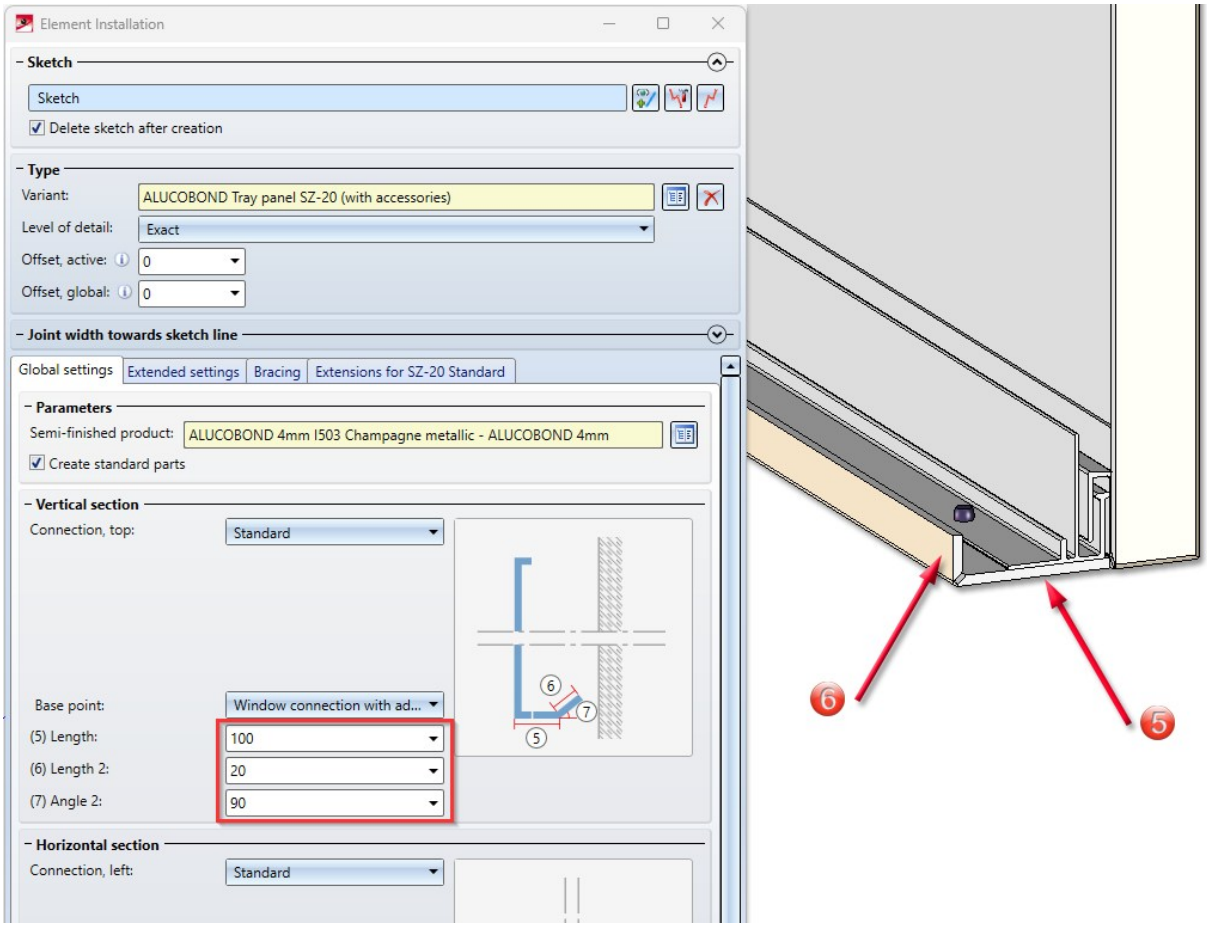
Until now, sketches with external references could not be edited using the **Element installation** or **Sub-structure** feature. From HiCAD 2025 SP2 onwards, changes to sketches that result from editing external references are taken into account if the new option **Use sketch from drawing** is activated. If, for example, the feature **Element installation** is recalculated, the changed sketch is then used to calculate the installed elements.



Service Pack 1

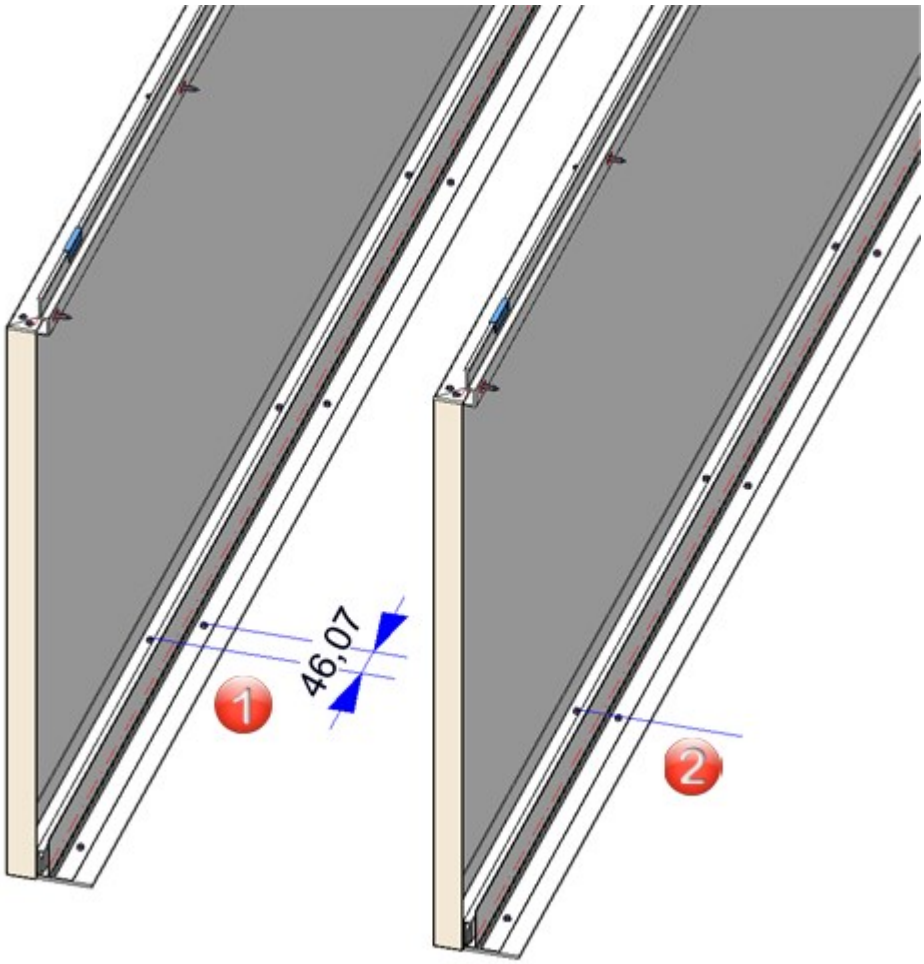
Window connection with additional sheet metal - Base point

For ALUCOBOND® SZ 20 tray panels, the **Window connection with additional sheet** has been extended at the **Base point** by a second sheet of metal. If you would like to fold a second sheet of metal, enter a length and the bending angle. If you enter 0 for the length, no second sheet will be attached.



Window connection with connecting plate

With ALUCOBOND® SZ-20 tray panels, the window connection with additional plate from SP1 is carried out in a practice-oriented manner with a uniform alignment of the rivets.

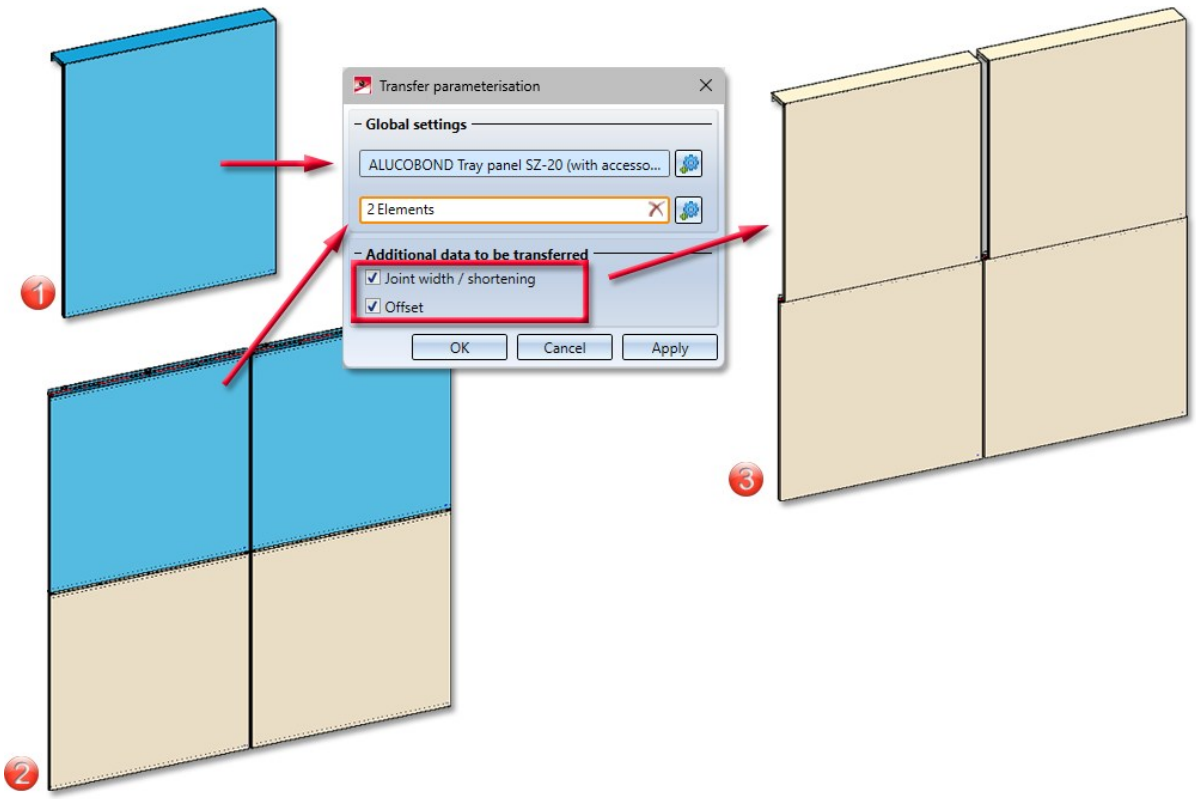


(1) Before HiCAD 2025 SP1, (2) Alignment of the rivets from SP1 in one line

Major Release

Transfer parameterisation

You can use the **Transfer parameterisation** function to apply the parameters of one installation element to another installation elements. This tool has been extended so that the element's **Joint width** to the sketch line and the distance of the element to the sketch plane (**Offset**) can now also be transferred. However, this is only possible if the installation elements are based on the same variant.



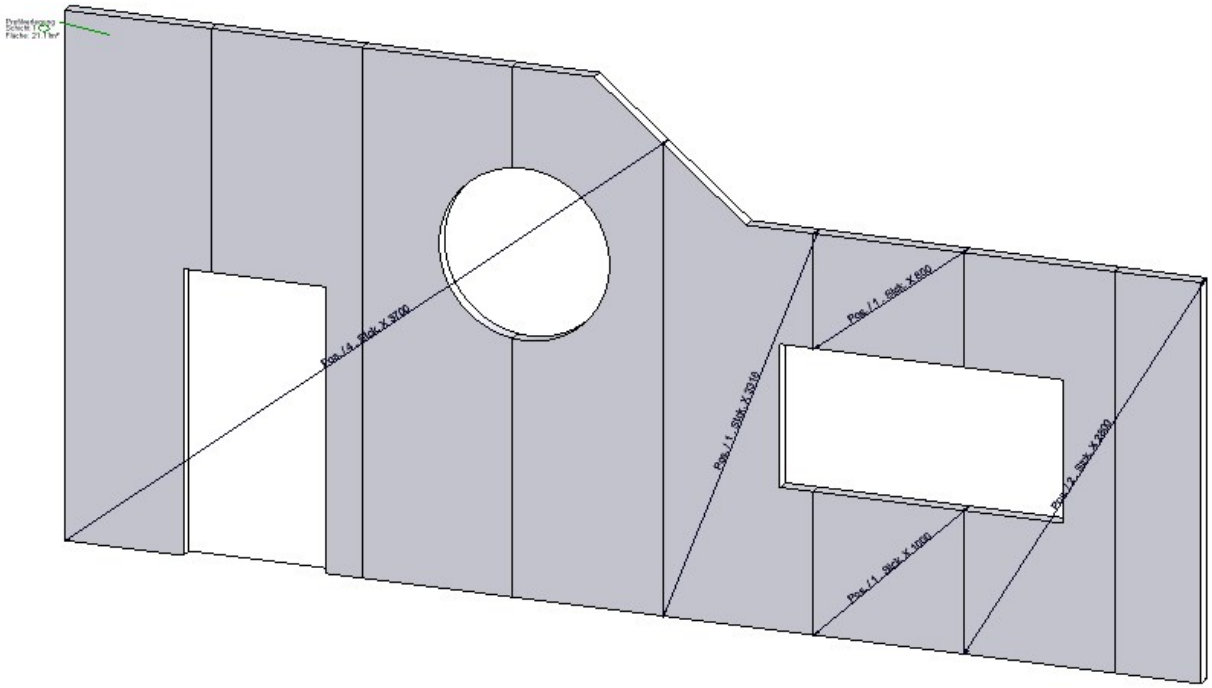
(1) Source:ALUCOBOND Tray panel SZ-20 (with accessories), with **Attic, long**,
(2) Target: ALUCOBOND SZ-20, **Standard**,
(3) Elements after transfer of **Attic, long**, **Joint width** and **Offset**

Profile Installation

Service Pack 2

Annotations on exact representation

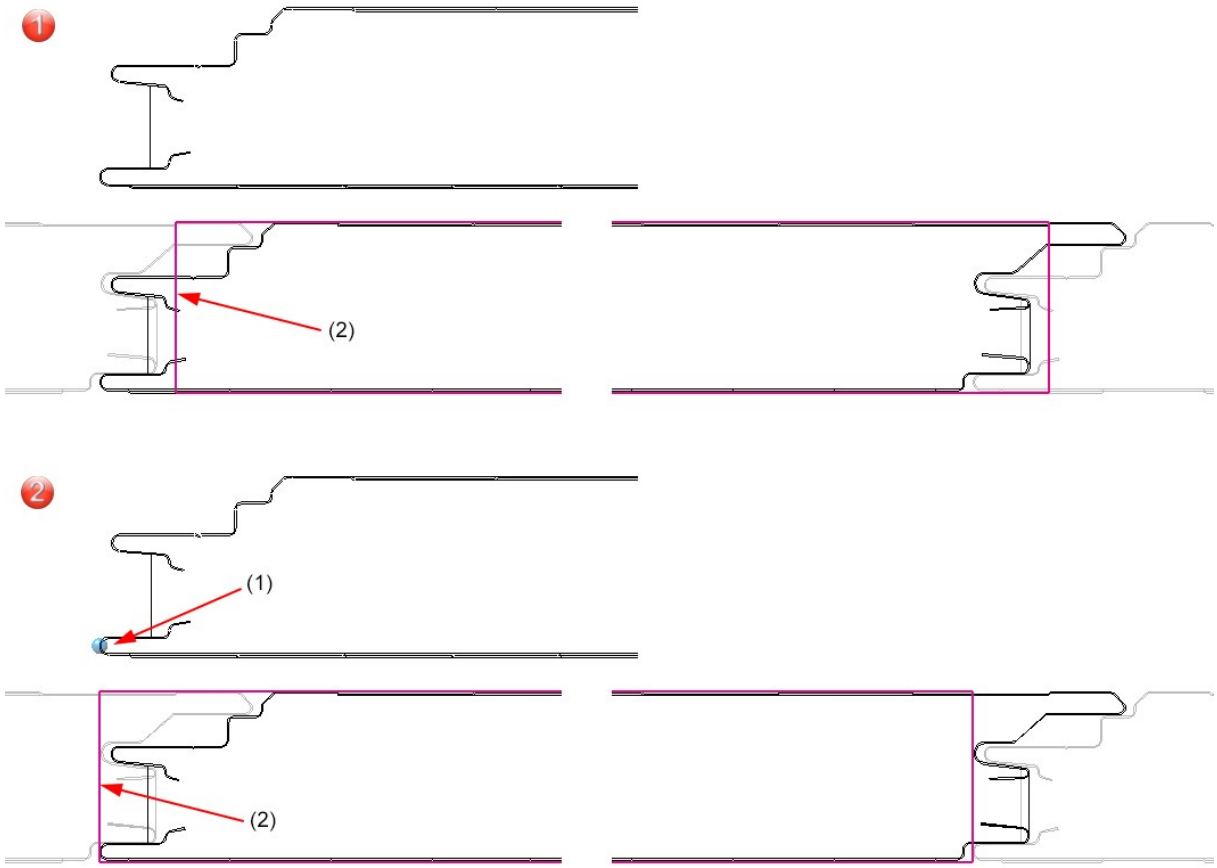
To display more than just the annotations on uncut profiles in a drawing, there is now the new function **Add designations, cut contour view**  .



After selecting the function, the cut contour is displayed. Adjacent profiles of the same length are indicated by diagonal lines. The edits are also displayed.

Fitting point for contour

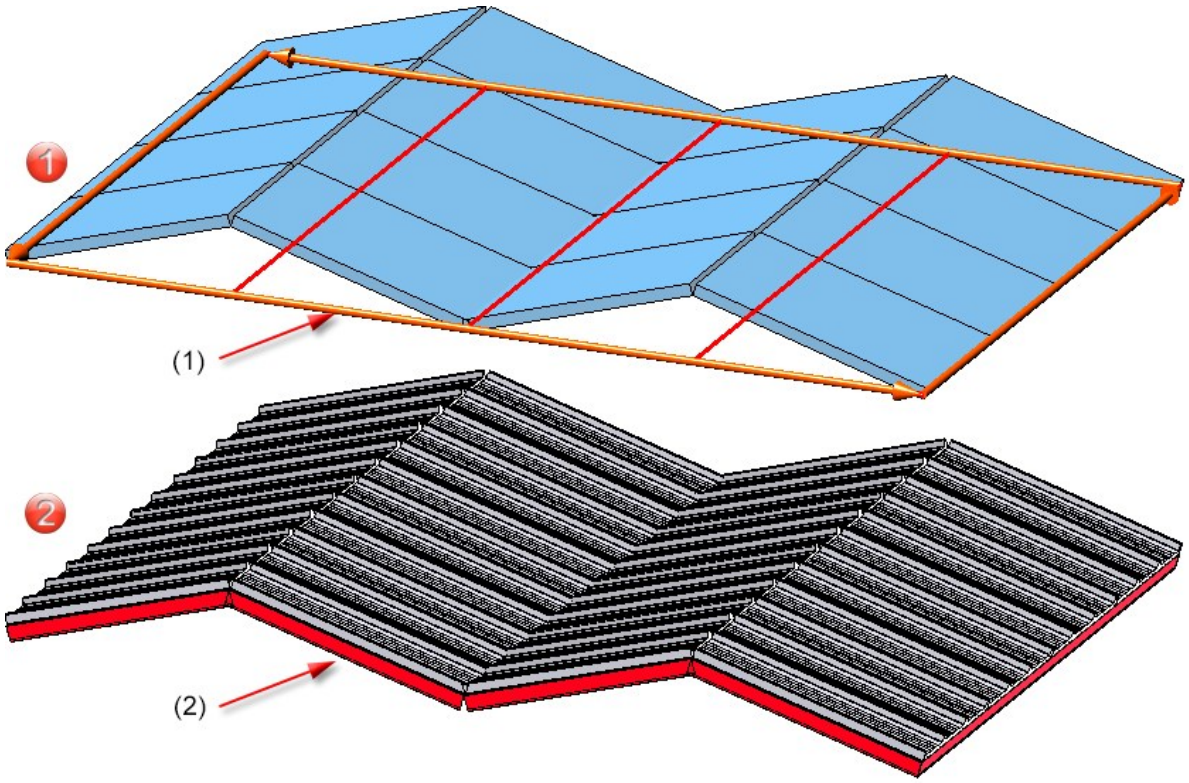
The fitting point for the **Profile installation**  function now not only refers to the exact profile but also to the contour of the profile. Thus, when setting a fitting point, the uncut contour is aligned left-aligned to the fitting point. If no fitting point is set, the contour is centred.



Example 1 without fitting point: The contour (2) (magenta) is centred in the exact profile. Example 2 with fitting point: The contour is aligned left-aligned to the fitting point.

Edge profiles for roof inclinations

For inclined roofs, the edge profiles were previously projected onto the horizontal surface. From SP2 onwards, when an edge is selected that runs over several roof sections, a profile with the correct inclination is automatically generated for each roof section.



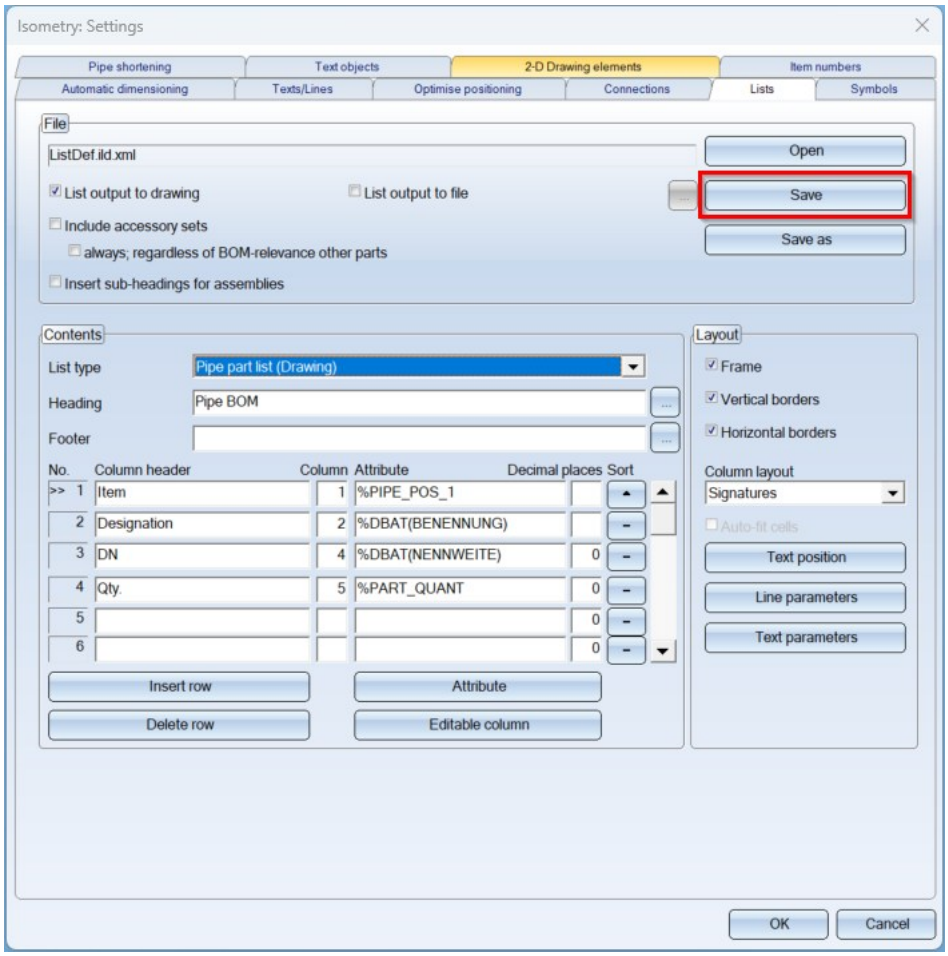
(1) Select the edges for the edge profiles using the sketch (orange arrows), (2) Edge profiles are adapted to the roof inclination

Plant Engineering

Service Pack 2

Save settings for lists

In addition to the **Save as...** button, the List settings  for isometries and pipe spool drawings now also feature the **Save** button.

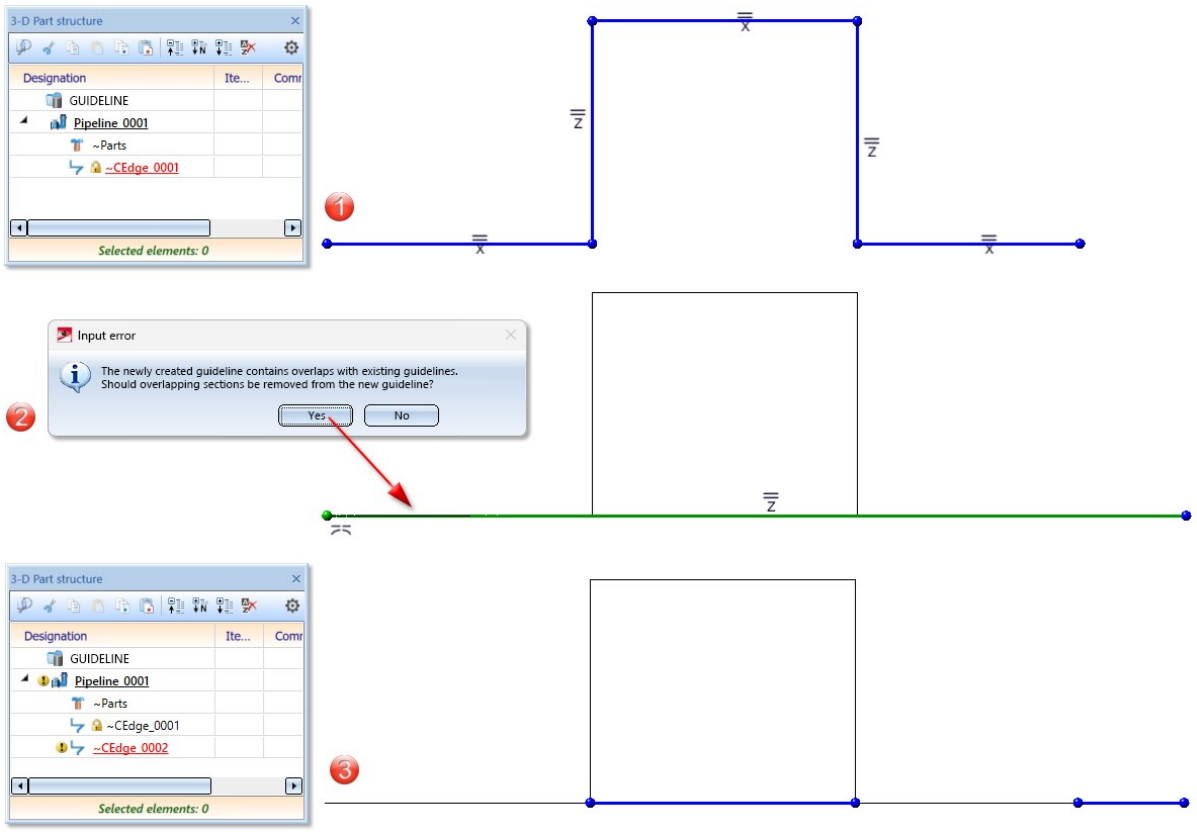


Correct guidelines after creation

From HiCAD SP2, newly drawn guidelines are checked for conflicts with existing guide lines or pipe parts after sketching. If a conflict can be identified beyond doubt, you will be offered the option of automatically shortening the new guide line. Conflicts include overlaps of the new guideline with existing guidelines and existing pipe parts that are arranged in the drawing in such a way that they would cover the new guideline.

Please note:

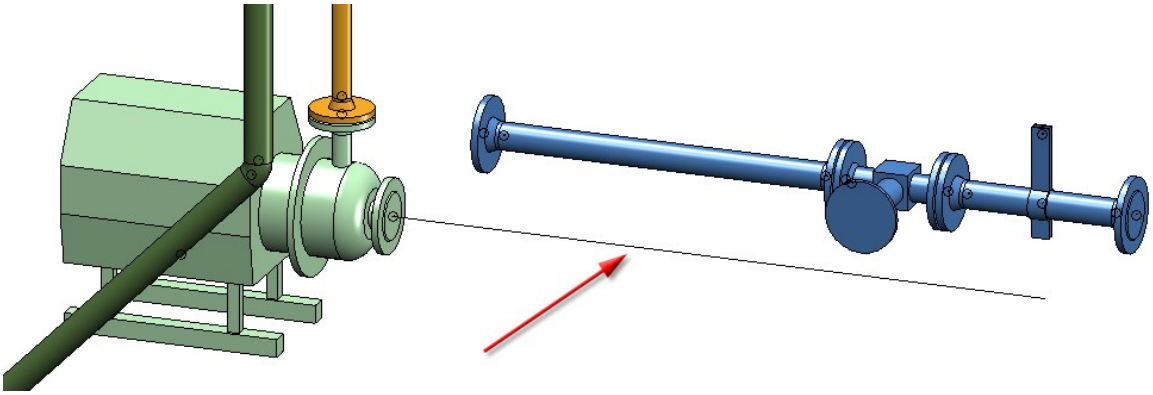
Overlaps of a guideline with itself are not corrected.



(1) First guideline, (2) overlapping second guideline, (3) overlaps are omitted

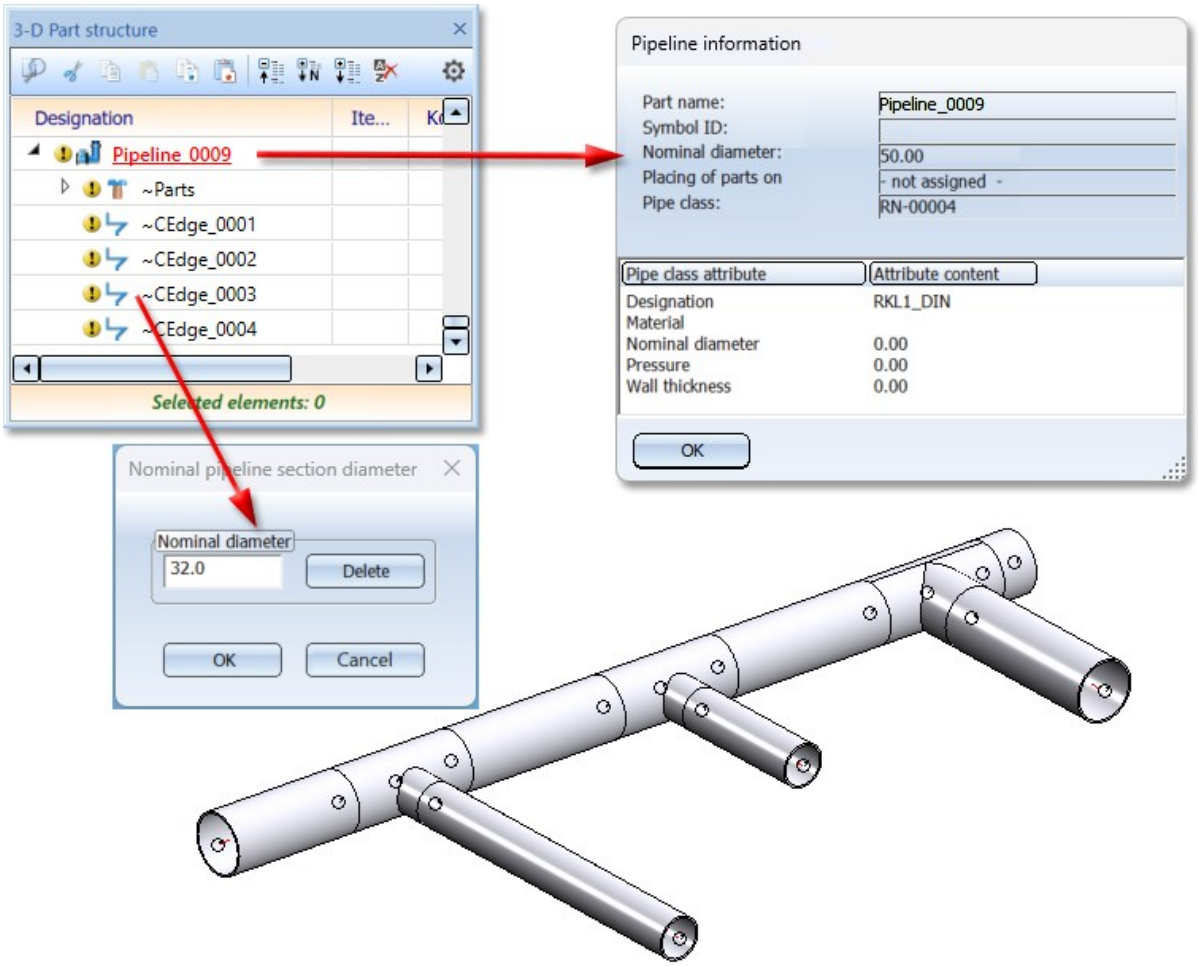
Make guidelines visible after moving ~Parts and ~CEdge

It sometimes happens that pipe part nodes (~Parts) or guidelines (~CEdgeXXXX) are moved by mistake. Until now, the unassigned guideline was then invisible. To make the unwanted displacement obvious, unassigned guidelines are made visible again after such an action.



Defining nominal guideline diameters that differ from the nominal pipeline diameter

From HiCAD 2025 SP2 it is possible to assign a nominal diameter to guideline sections even if a nominal diameter has already been defined in the superordinate pipeline. The input dialogue of the Specify nominal pipeline section diameter function for setting and changing nominal pipeline section diameters remains unchanged. A previously defined section diameter can be deleted by confirming the empty input field.



If there is a nominal diameter on both the guideline and the pipeline, the nominal diameter of the guideline applies. This is particularly relevant for the functions for part insertion, automatic part placement, and part exchanging.

Furthermore, the nominal diameter of a guideline will in future also be saved under the Nominal diameter attribute. Until now, the nominal diameter has been saved under the Quantity 1 attribute. This assignment is to be removed in the next Major Release.

Standardised context menus

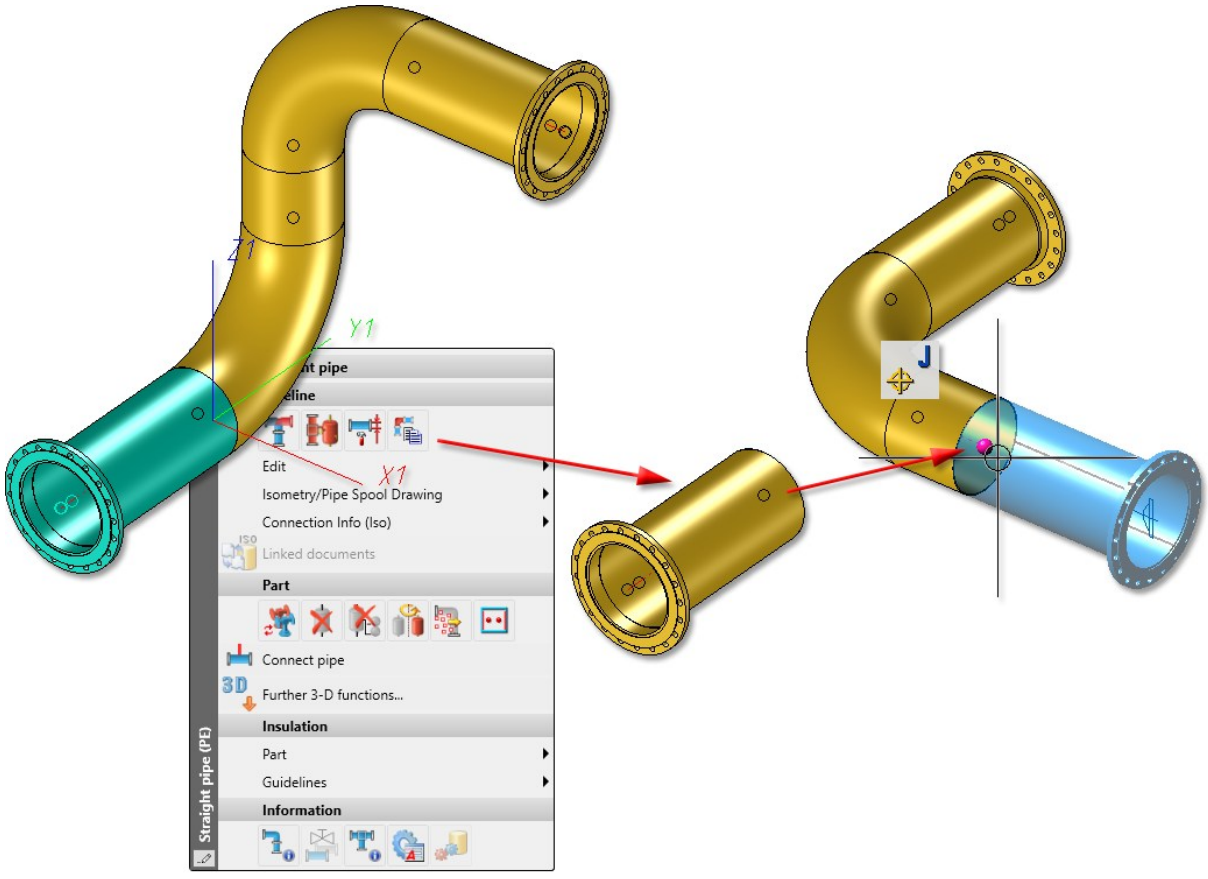
The Plant Engineering context menus in the ICN have been standardised with those in the drawing, and supplemented with functions for referenced parts for both drawing and ICN.

Specifically, the change affects the context menus of **Straight pipes**, **Elbows**, **Components**, **Component connections**, **Flange boltings**, **Duct parts**, **Other Plant Engineering parts**, and **Guidelines**.

Improved copying of pipe parts

Copying pipe parts with the **Copy pipe parts**  function can be a quick method for inserting new parts into a pipeline. This function has therefore been added to the context menus of pipe parts. The **Copy pipe parts**  and the **Move pipe parts**  function have been added to the context menu of multiple selections.

The behaviour of the copy function has been changed so that when copying a pipe part with a guideline below it to a target pipeline without guidelines, no new guideline is created in the target pipeline.



Pipe parts copied from Pipeline_0001 (1) with guideline to Pipeline_0002 (2) without guideline

Straight pipes according to EN 10216

The following variants have been added:

Variant file	Designation	Part type
EN10216-1-R2.VAA	Seamless tube	Straight pipe
EN10216-1-R3.VAA	Seamless tube	Straight pipe
EN10216-2-R2.VAA	Seamless tube	Straight pipe
EN10216-2-R3.VAA	Seamless tube	Straight pipe
EN10216-3-R2.VAA	Seamless tube	Straight pipe
EN10216-3-R3.VAA	Seamless tube	Straight pipe
EN10216-4-R2.VAA	Seamless tube	Straight pipe
EN10216-4-R3.VAA	Seamless tube	Straight pipe
EN10216-5-R2.VAA	Seamless tube	Straight pipe
EN10216-5-R3.VAA	Seamless tube	Straight pipe

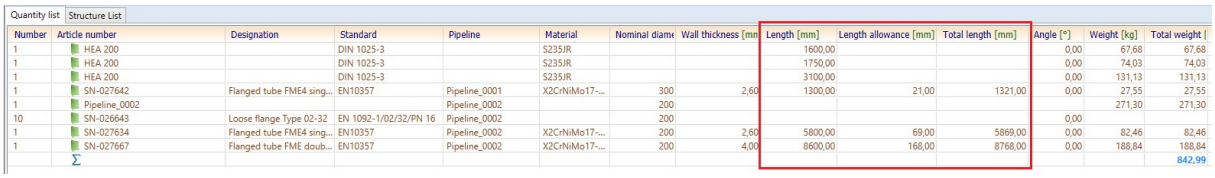
Length allowances in Bills of Materials

Length allowances are now listed separately in the Bills of Materials for pipeline planning.

The configuration files for BOMs included in the scope of delivery

- anlagenbau_rl.rm_settings (Used by Plant Engineering > Evaluation > Bill of Materials, for entire drawing )
- anlagenbau_szn.rm_settings (Used by Plant Engineering> Evaluation > Bill of Materials, for active pipeline )
- HiCAD_Anlagenbau.rm_settings (Used by Drawing > Itemisation/Detailing > Bill of Materials, for entire drawing )
- HiCAD_PipeBook.rm_settings (Used by Isometry + Pipe Spool Drawing > Edit lists > EditPL... > Pipe book )

now contain the columns **Length allowance** and **Total length** in addition to the **Length** column:



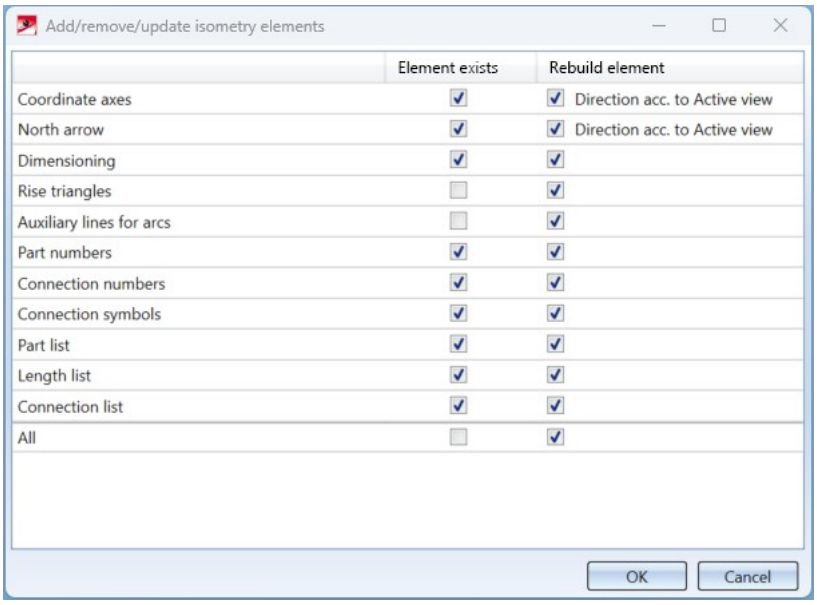
Number	Article number	Designation	Standard	Pipeline	Material	Nominal diam	Wall thickness [mm	Length [mm]	Length allowance [mm]	Total length [mm]	Angle [°]	Weight [kg]	Total weight [
1	HEA 200		DIN 1025-3		S235JR			1600,00			0,00	67,68	67,68
1	HEA 200		DIN 1025-3		S235JR			1750,00			0,00	74,03	74,03
1	HEA 200		DIN 1025-3		S235JR			3100,00			0,00	131,13	131,13
1	SN-027642	Flanged tube FME4 sing...	EN10357	Pipeline_0001	X2CrNiMo17-...	300	2,60	1300,00	21,00	1321,00	0,00	27,55	27,55
1	Pipeline_0002			Pipeline_0002		200						271,30	271,30
10	SN-026643	Loose flange Type 02-32	EN 1092-1/02/32/PN 16	Pipeline_0002		200					0,00		
1	SN-027634	Flanged tube FME4 sing...	EN10357	Pipeline_0002	X2CrNiMo17-...	200	2,60	5800,00	69,00	5869,00	0,00	82,46	82,46
1	SN-027667	Flanged tube FME doub...	EN10357	Pipeline_0002	X2CrNiMo17-...	200	4,00	8600,00	168,00	8768,00	0,00	188,84	188,84
	Σ												842,99

The change makes the **Combine pipe lengths in BOMs** parameter from the Configuration Editor under **Plant Engineering** > **Bills of Materials** more transparent. Previously, this parameter ensured that length allowances were also included in the total lengths. Now, the geometric lengths and length allowances are summarised separately. The additional column **Total length** now always shows the sum of **Length** and **Length allowance**.

Isometry and pipe spool drawing

Merge Add/Remove elements and Update dialogues

The functions **Add/Remove elements** and **Update all** have been combined into combined dialogues into the **Update** function for both pipe spool drawing and isometry.



The combined dialogues simplify updates of elements that were not included in the old update dialogue (dimensioning, rise triangles, part numbers and auxiliary lines for bent pipes). These were previously updated using the **Add/Remove** dialogue.

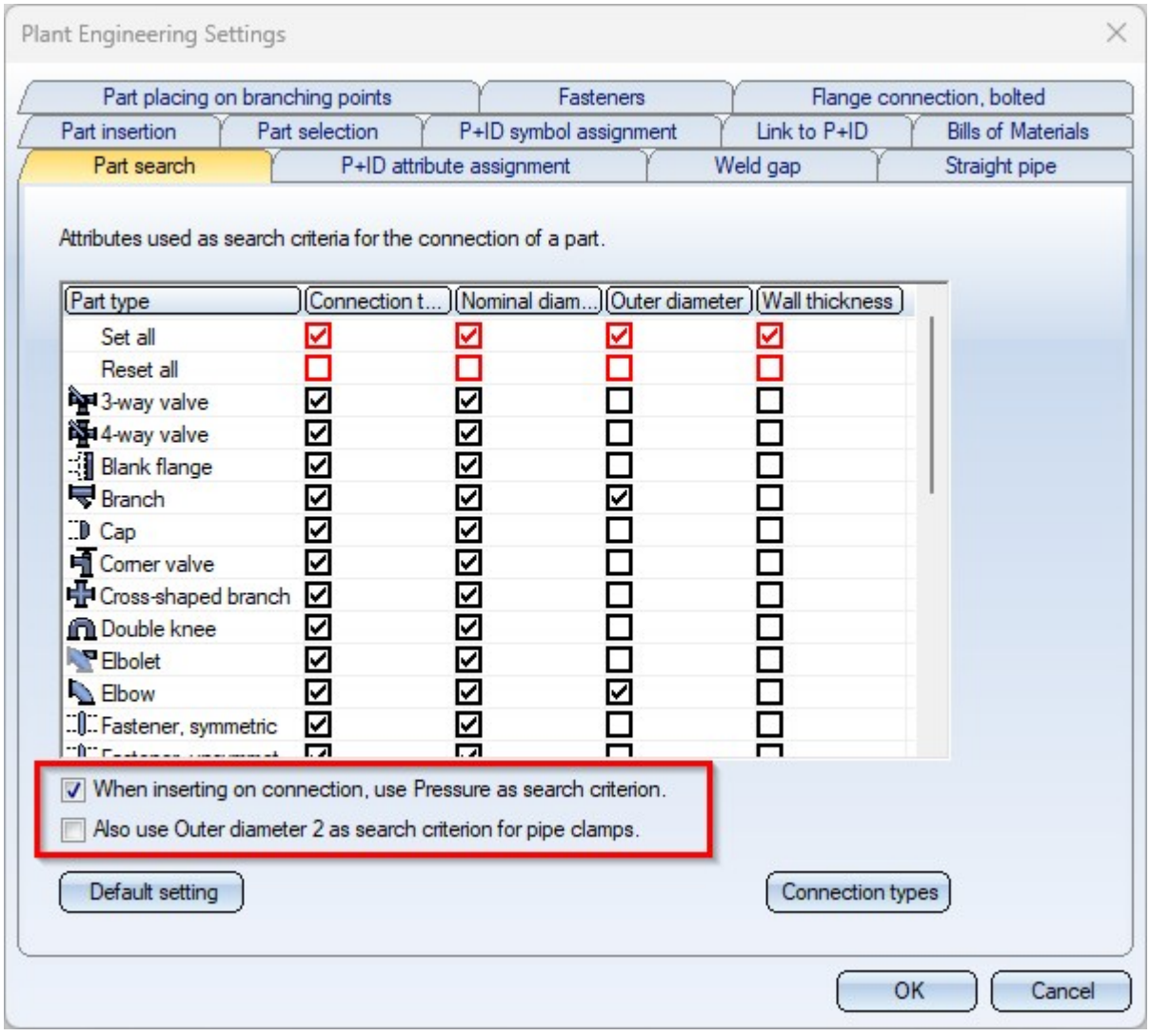
Service Pack 1

VarDbAttConfig deleted

the file VarDbAttConfig.exe is no longer needed and has therefore been deleted.

Plant Engineering Settings: Part search optimised

The **Part search** tab of the Plant Engineering Settings has two options less from HiCAD 2025 SP1.



From now on, the following applies:

- If the part search is unsuccessful, the search conditions will always be displayed again.
- The outer diameter of pipes is no longer used as a search condition for loose flanges.

Transfer of ROHR2 standard designations to ROHR2

ROHR2 uses standard designations that partly differ from those used in HiCAD. In the **PlantParts** folder, there is a sub-folder **Rohr2** containing the file rohr2normen.csv. This file describes which ROHR2 standard designation corresponds to a HiCAD standard designation. The header of the file contains a short description of its structure:

Format:

Part type identifier; Part type; Standard as in HiCAD archive; Standard as in ROHR2

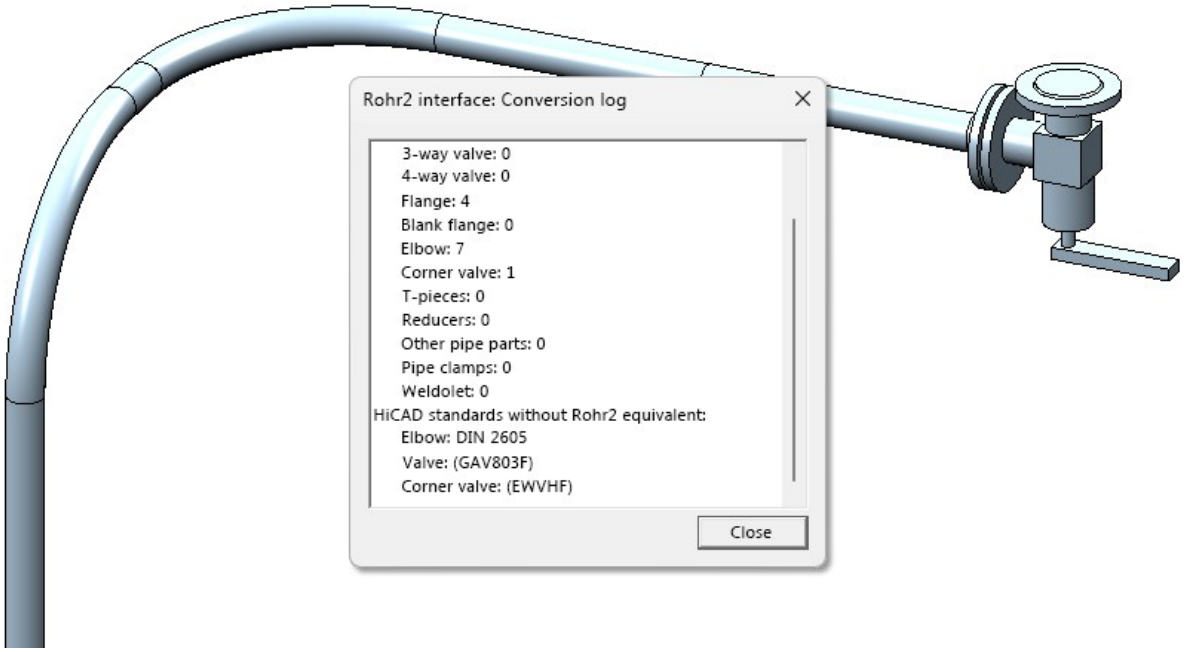
#

5100010;Flange;ASME SoFlange 150;ASME B16.5 150 Slip on

etc.

The formatting of rohr2normen.csv is UTF-16 LE BOM.

The log displayed after a transfer to ROHR2 contains a list of the HiCAD standards for which no ROHR2 equivalent could be found:



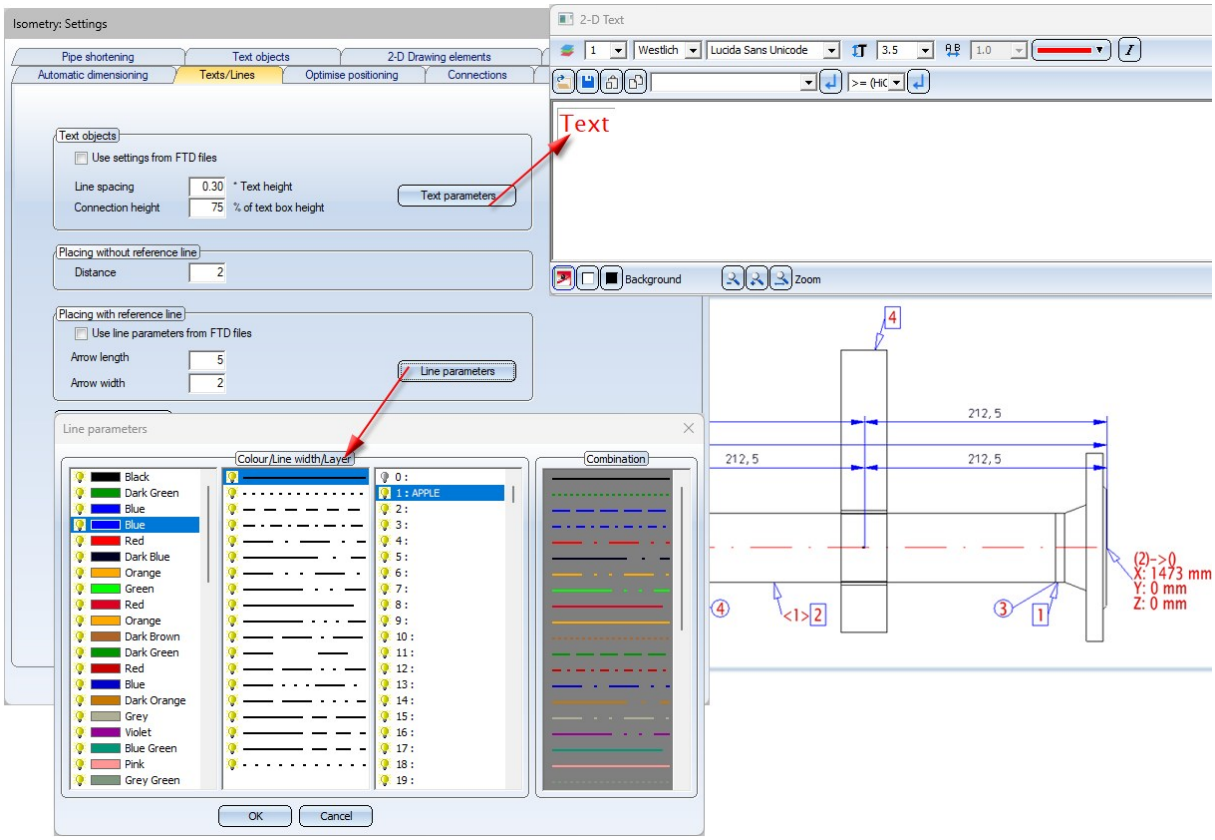
You should check whether any relevant standards are missing and, if necessary, enter them in the ROHR2NORMEN.CSV. As a first point of reference, it can be said that ROHR2 only defines standards for straight pipes, elbows, flanges, caps, reducers and T-pieces (as of ROHR2 version 33.1). Standard designations of other part types are therefore probably of secondary importance. If in doubt, consult your ROHR2 documentation.

Isometry and Pipe spool drawing

Update text parameters

After changing the text parameters in the Isometry settings, it is now easy to apply the changes to the tags. From HiCAD 30.1, the **Complete update** function also updates the text and line parameters of the text tags.

The parameters updated are those that have no (practical) influence on the position of the tags. These are the font and font colour, but also the display of the reference line, e.g. whether it is dashed.

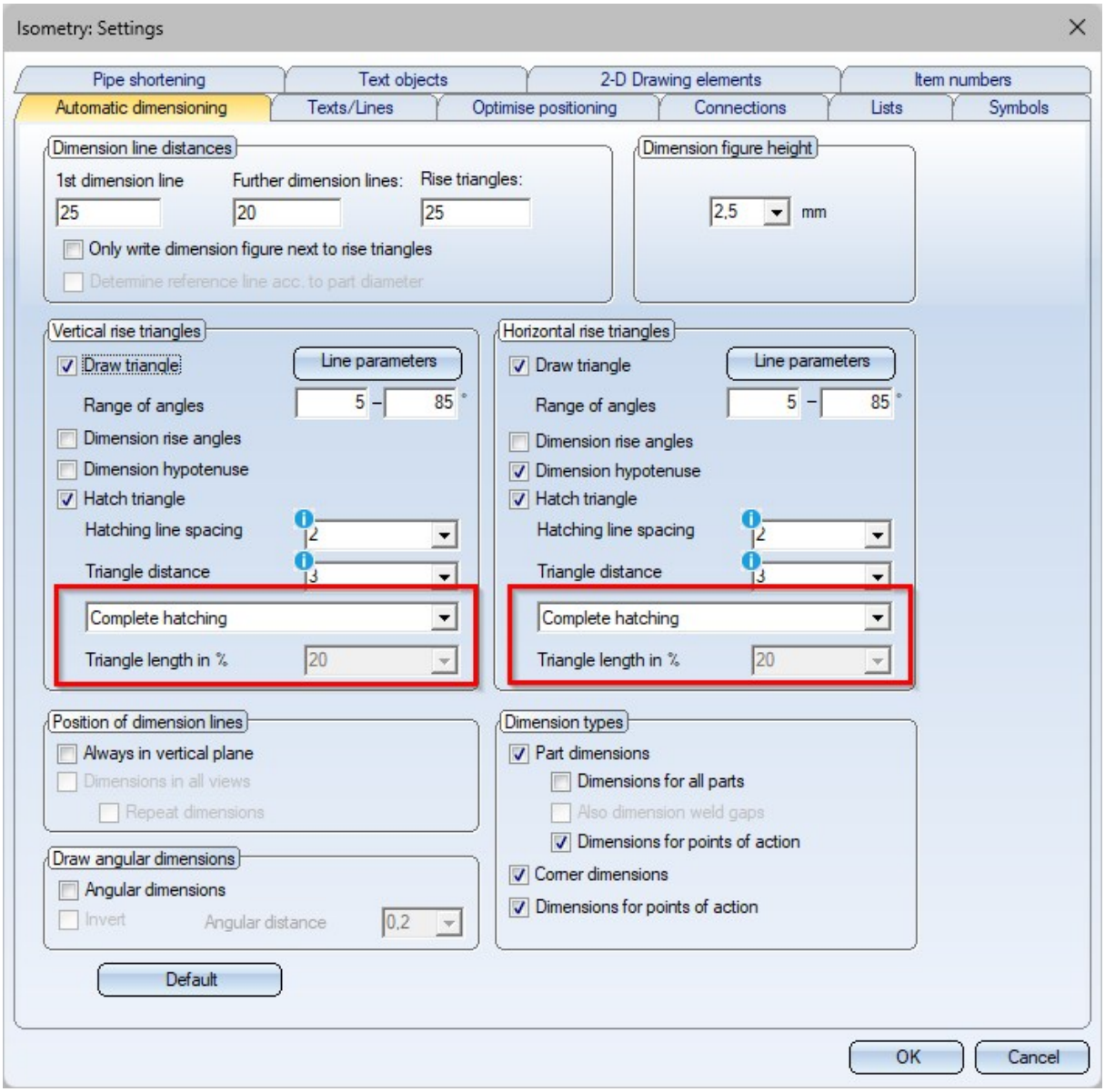


The main use case for this update is when changing the global text and line parameters on the **Text/Lines** tab of the **Isometry/Pipe Spool Drawing Settings** function.

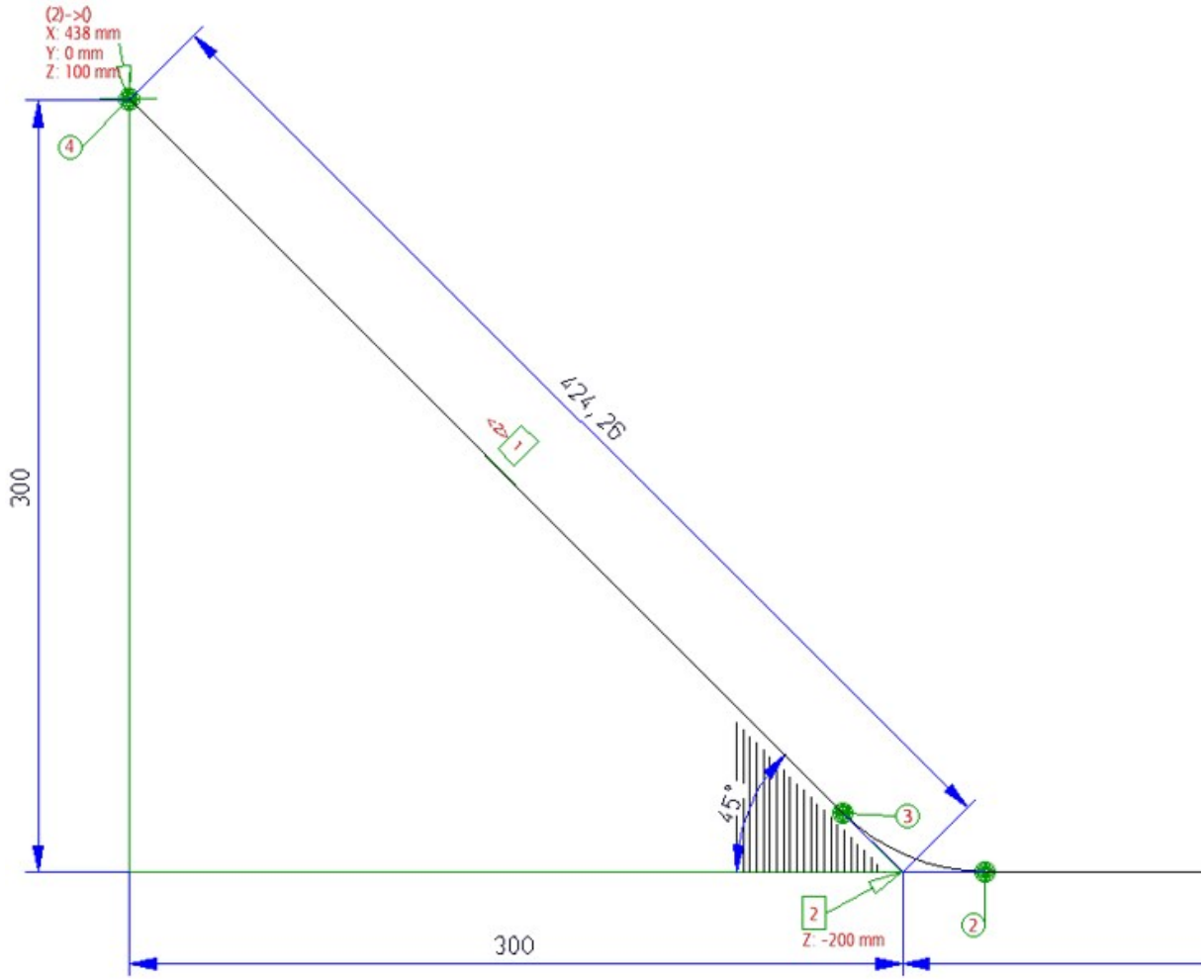
Please note the following restriction for old drawings if you subsequently activate the Use text settings from FTD files or Use line settings from FTD files options on the **Text/Lines** tab: In old constructions, only the correct FTD file can be determined for position tags from the tag itself, but not for the other tag types. Thus, only position tags can be updated. In constructions from HiCAD 30.1 onwards, this restriction does not apply, so that connection symbols, connection coordinates, warning signs, subdivision points, etc. can also be updated. Please also note that only one font/font colour is ever set for the entire tag.

Isometry: Adjusting the parameters for rise triangle hatching

The settings for the rise triangles of the isometry have been slightly adjusted.



When a rise triangle is partially hatched, the hatched area is specified exclusively via the length of the base side of the rise triangle, and this is done proportionally in % to the length of the base side of the entire rise triangle. Before HiCAD 2025 SP1, the specification was in the length unit of the drawing.



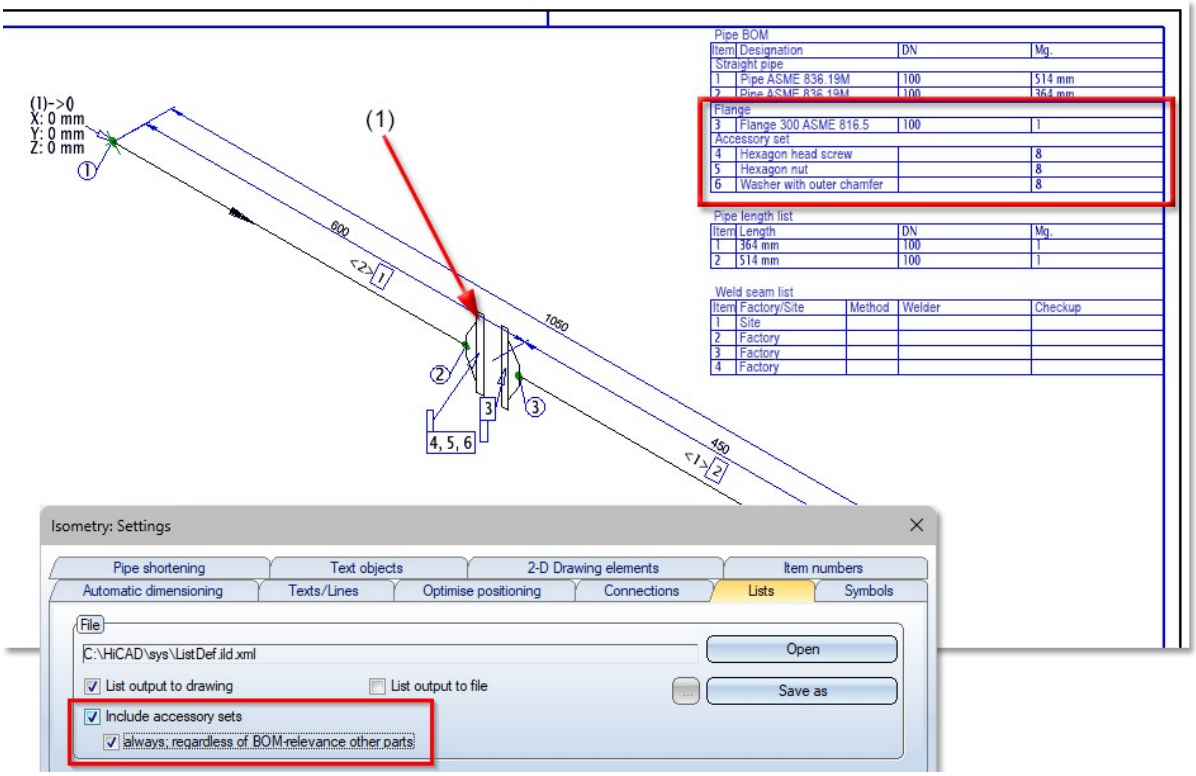
Important:

If you had previously selected your settings using the triangular area, this setting will become a complete hatching in HiCAD 30.1.

Isometry/Pipe spool drawing: Accessory sets of flange connections in BOMs and itemisation tags

Especially in the case of existing pipelines, it may be necessary to create an isometric drawing that does not show the parts themselves, but their accessory sets in text tags and BOMs. Prior to HiCAD 2025 SP1, however, it was not possible to output an accessory set independently of the BOM-relevance of the part it is an accessory of. Now you can force such accessory sets to appear in the BOMs of isometric and pipe spool drawings as well as in the text tags.

For this purpose, the Isometry and Pipe spool drawing settings now include the sub-option **always; regardless of BOM-relevance of other parts** under the main option **Include accessory sets** on the **Lists** tab. If this option is selected, pipe parts that are not BOM-relevant but have an accessory set are given a text tag. In addition, all accessory kits are listed in the pipe BOM, regardless of whether their respective pipe part is BOM-relevant.



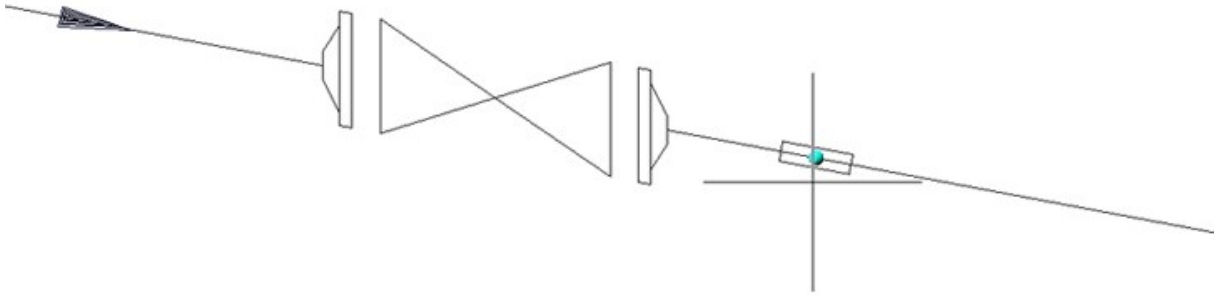
(1) The left flange is not BOM-relevant but has an accessory set. The right flange is BOM-relevant and has no accessory set.

Only one flange is shown in the pipe BOM. The accessory set of the non-BOM-relevant flange is shown in both the pipe BOM and the text tag.

Isometry: Arrow functions

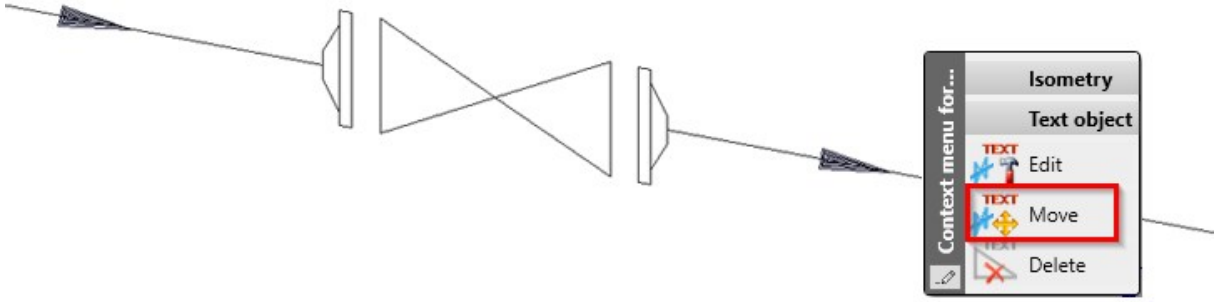
From HiCAD 2025 SP1, flow direction arrows can be edited in the isometry. You can add, move and delete flow direction arrows.

To add flow direction arrows, start by clicking this icon  in the **Isometry + Pipe Spool Drawing** tab. You will then be prompted to select a reference point. While selecting the reference point, a rectangle indicates where the flow direction arrow will be drawn.

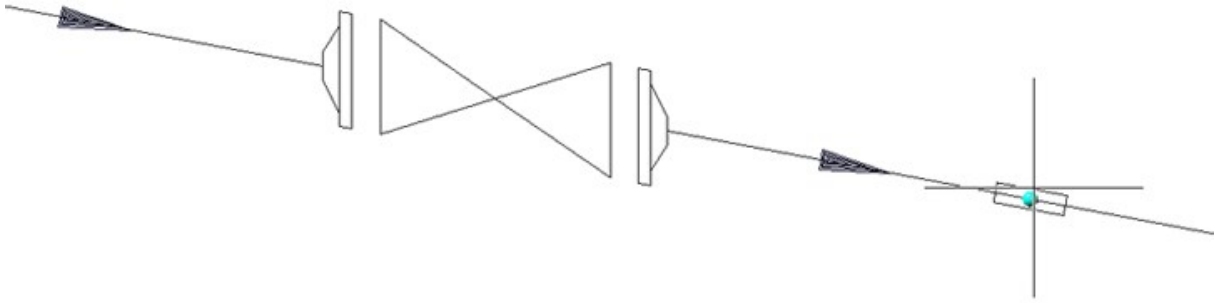


You can continue adding flow direction arrows until you cancel the function with **ESC** or **MMB**.

You start moving flow direction arrows via the context menu of the flow direction arrow.

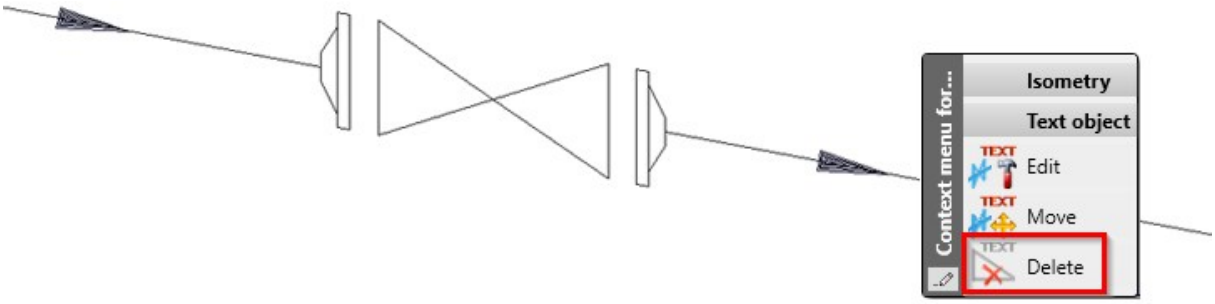


As with adding flow direction arrows, you will then be asked to select a new reference point for the flow direction arrow:



Clicking on the desired location ends the function.

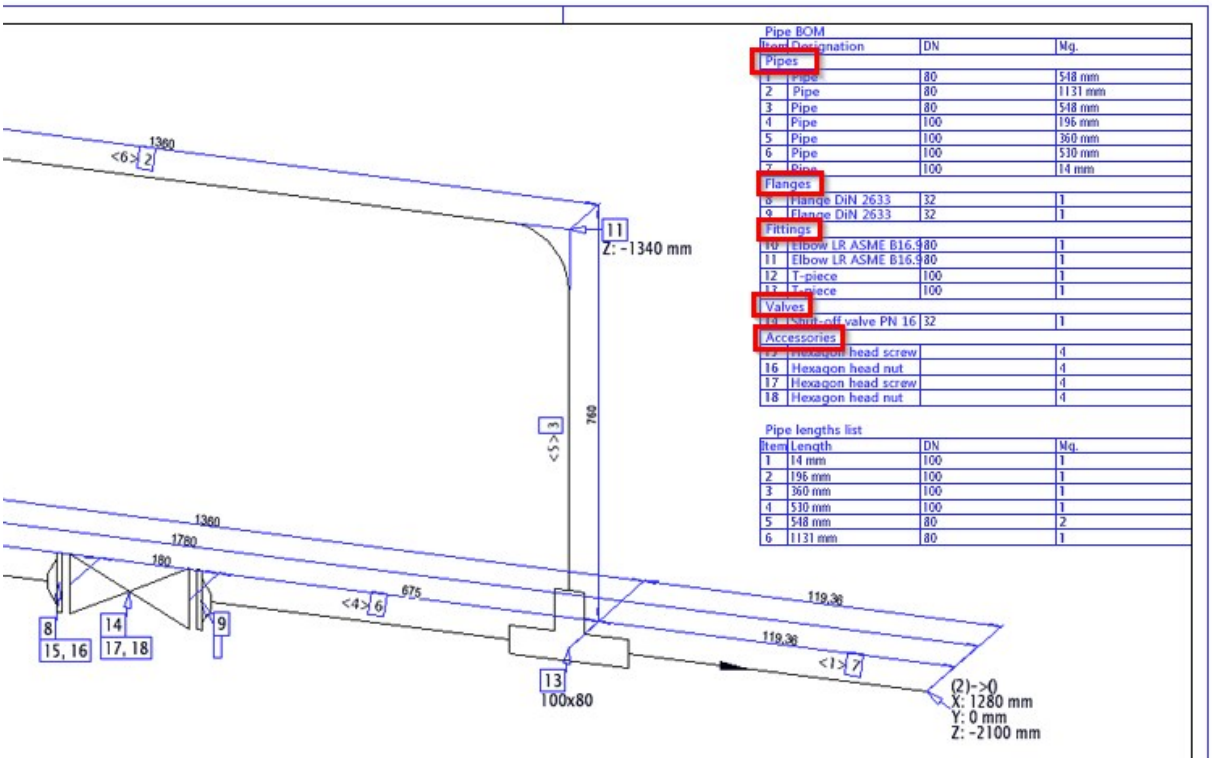
Deleting a flow direction arrow can also be done via the context menu.



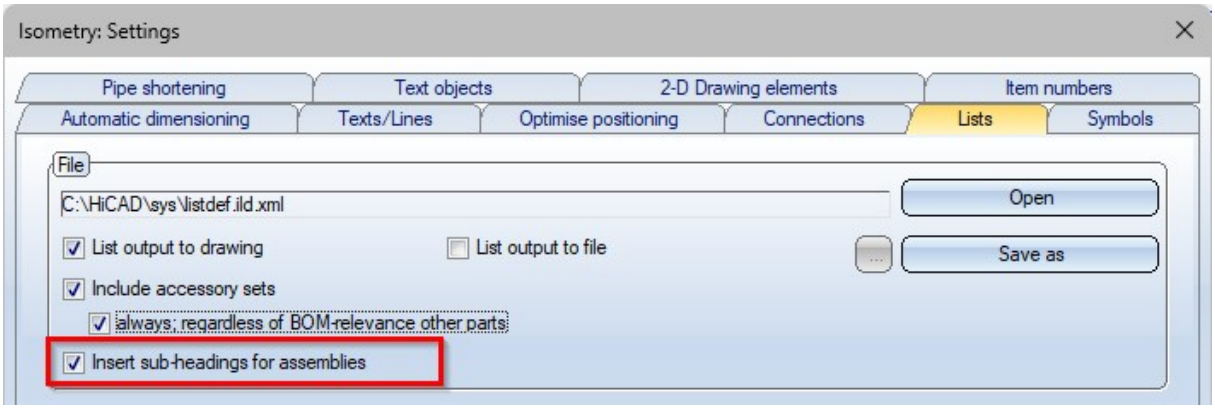
After selecting **Delete**, the flow direction arrow is removed and the function ends.

Isometry: Pipe part type headings in BOMs

From HiCAD 2025 SP1, the isometry and pipe spool drawing BOMs can have subheadings for part types:



You activate the subheadings on the **Lists** tab of the Settings dialogue for isometries and pipe spool drawings:



The subheadings are created by assigning headings to the pipe part types from which the rows of the BOMs have been created. Rows with the same heading are grouped together.

Accordingly, this option only makes sense if you create your pipe parts list in such a way that pipe part types that belong together appear in the appropriate order in the BOM. This is the case in the default settings from HiCAD 2025 SP1.

Important: In particular, the part hierarchy in the Item numbers tab of the Settings dialogue for isometries and pipe spool drawings, as well as the sorting order in the column Definitions of the Lists tab, have a significant effect on the order in the BOMs. Assigning a separate item number to each individual part also does not work well with intermediate headings.

The configuration of the headings themselves is done via a text file for isometry and pipe spool drawing. The text file for the isometry is located under Install\PlantParts\IsoConfig\BOMHeadings.txt, and for the pipe plan under PlantParts\SpoolConfig\BOMHeadings.txt.

Both files are structured in the same way. First, headings are defined, e.g.:

H2: Flanges

The heading “H2” is therefore flanges. The colon is essential to distinguish the designation of the heading from the heading itself. You can then assign the heading to pipe part types, for example, like this:

5100010;Flange;H2

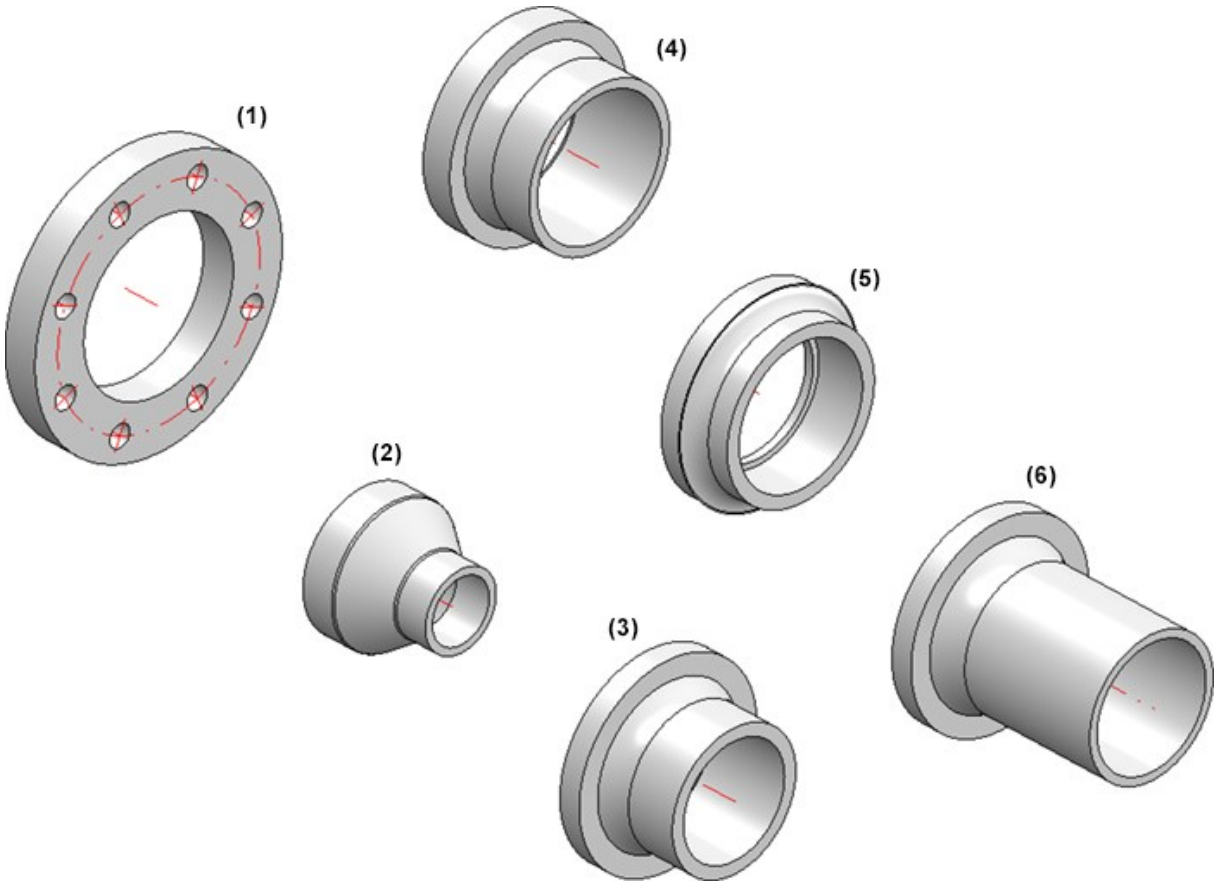
Flanges would thus be grouped under the subheading “Flanges”. When assigning, it is important to separate the part type identifier, the part type and the heading designation with semicolons.

Major Release

ELGEF / PROGEF - Further variants

The following parts have been added to the ELGEF/PROGEF variants:

	File	Designation	Type
(1)	ECOFIT_LOOSEFLANGE.VAA *1	Loose flange PP-V flange	Flange
(2)	ECOFIT_REDUCER_SDR11.VAA	Butt welded reducer SDR 11	Reducer, concentric
(3)	ECOFIT_STUB_FLANGE_SDR11.VAA	Butt welded collar SDR11	Welding neck
(4)	ECOFIT_STUB_FLANGE_SDR17.VAA *2	Butt welded collar SDR17	Welding neck
(5)	ECOFIT_STUB_FLANGE_SDR17_ GROOVED.VAA	ecoFIT collar bushing PE100 SDR17	Welding neck
(6)	ELGEF_STUB_FLANGE_SDR17.VAA	Butt welded collar SDR17	Welding neck

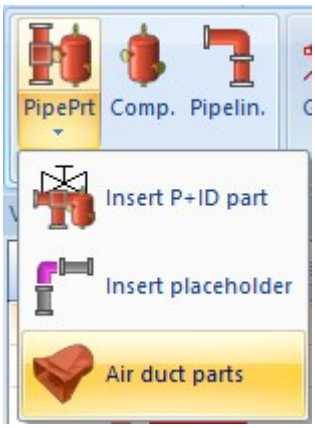


*1 The previous ECOFIT_LOOSEFLANGE_SDR11 and ECOFIT_LOOSEFLANGE_SDR17 variants are summarised in the ECOFIT_LOOSEFLANGE.VAA file.

*2 The previous variants ECOFIT_STUB_FLANGE_A_SDR17 and ECOFIT_STUB_FLANGE_B_SDR17 are summarised in the ECOFIT_STUB_FLANGE_SDR17.VAA file .

Air duct parts

The insertion of air duct parts is now possible again. The **Air duct parts** function is available for this at **Plant Engineering > New > PipePrt > ...**.



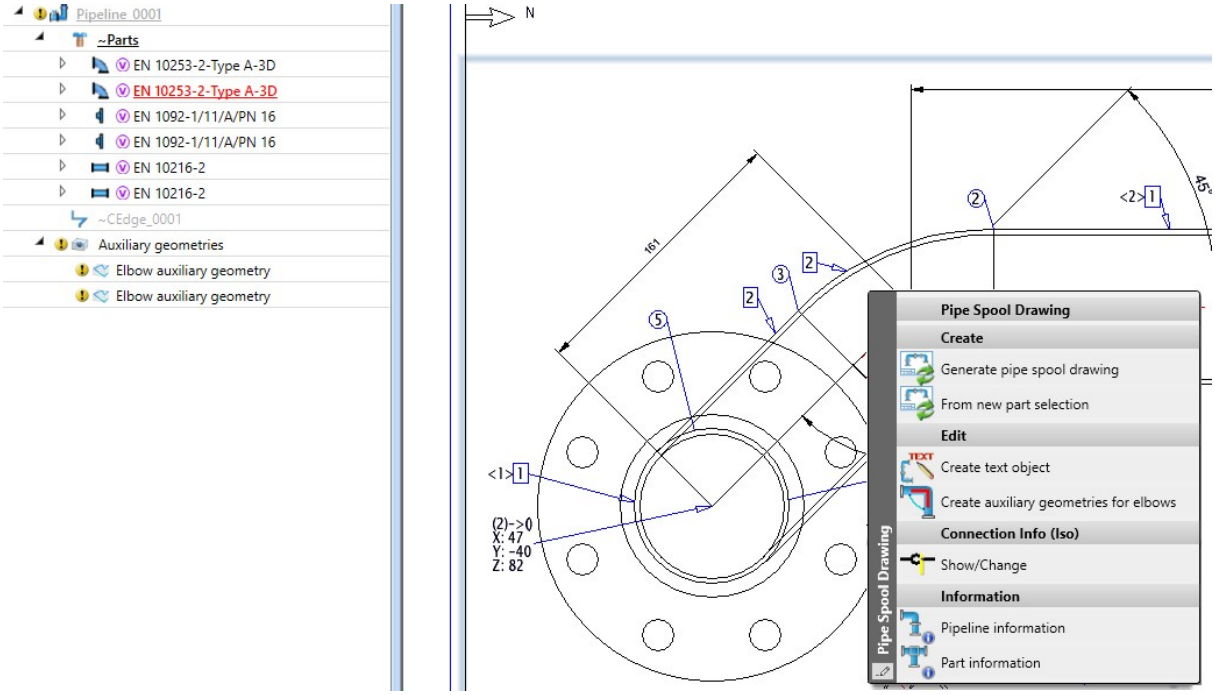
Pipe spool drawing - Auxiliary geometries for elbows

A pipe spool drawing is dimensioned almost entirely automatically. In practice, however, there is often a desire to add your own dimensions to the automatic dimensions.

In particular, if these dimensions are to refer to the corner points of elbows, it sometimes becomes difficult because there is usually no geometry there that can be picked directly.

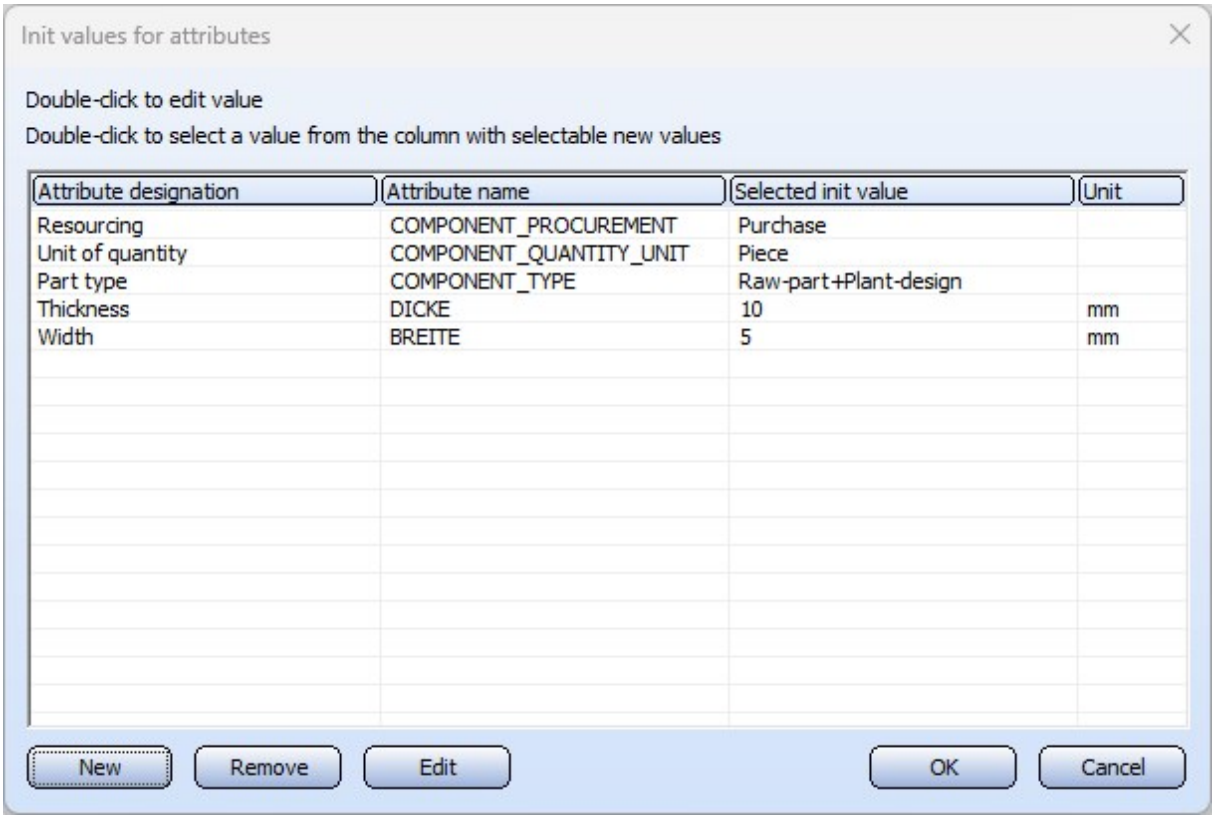
To make these cases easier, the function **Create auxiliary geometries for elbows**  is available. This function can be found in the context menu of a pipe spool drawing. To access it, right-click an element of the pipe spool drawing.

After calling up the function, HiCAD prompts you to select an elbow. A part named **Elbow auxiliary geometry** is then created below the pipeline under **Auxiliary geometries**. This part contains a sketch consisting of two lines that connect the connecting points of the elbow beyond the corner point.



DBPlantDataImport

DBPlantDataImport enables the initialization of attributes. Usually, the part type or the document type are initialized here, for example. However, it is also possible to specify unit-related attributes. To make this more comprehensible, DBPlantDataImport displays the unit in which the entered value is accepted in an additional column in HiCAD 30.0.



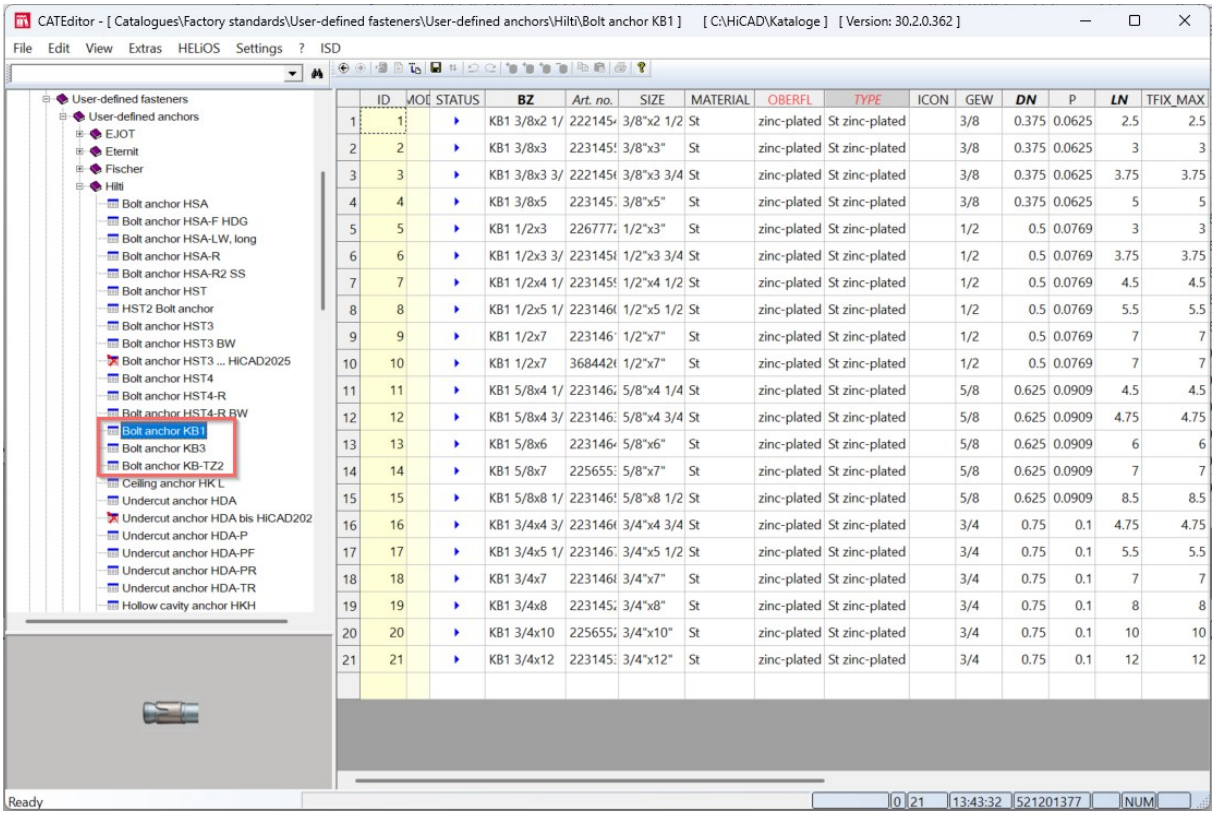
The unit is fixed and cannot be changed.

Catalogue Editor

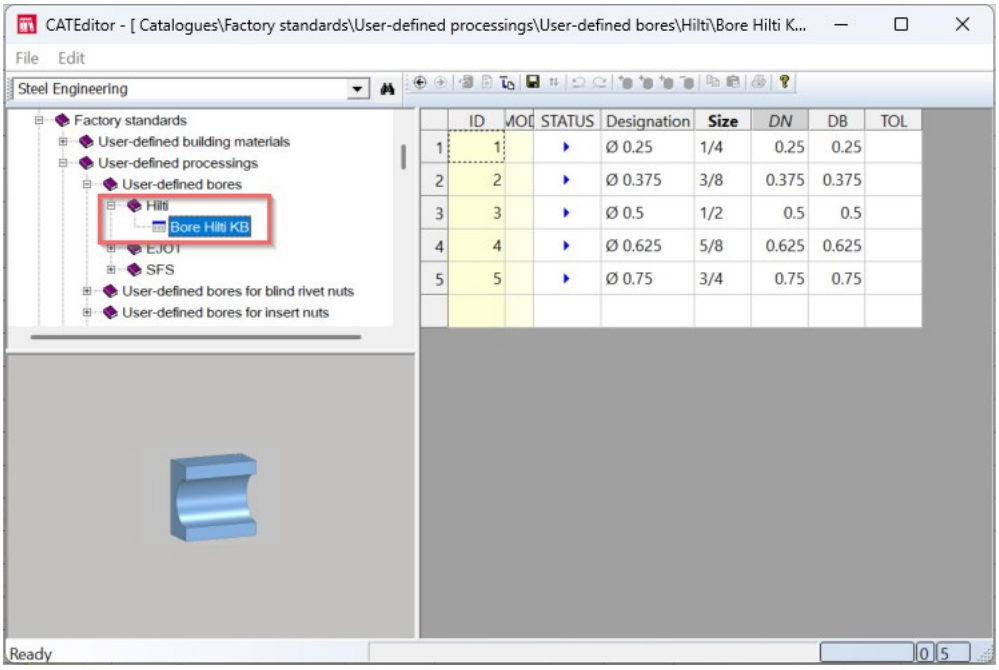
Service Pack 2

Catalogue: US anchors

Three bolt anchors from Hilti have been added to the catalogue. These bolt anchors have imperial units of measurement. You can find them at **Factory standards > User-defined fasteners > User-defined anchors > Hilti > Bolt anchor KB1, Bolt anchorKB3** and **Bolt anchorKB-TZ2**.

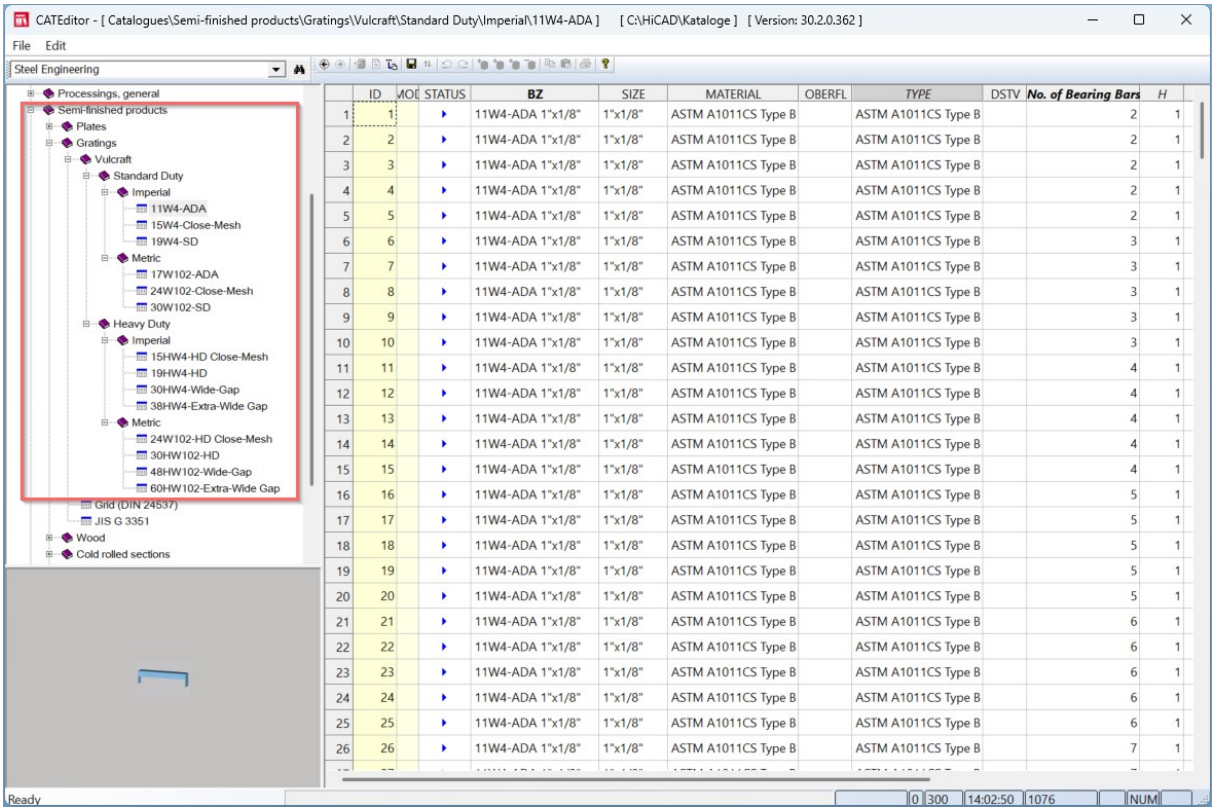


In addition, a table for Hilti bores has been added. You can find these at **Factory standards > User-defined processings > User-defined bores > Hilti > Bore Hilti KB**.



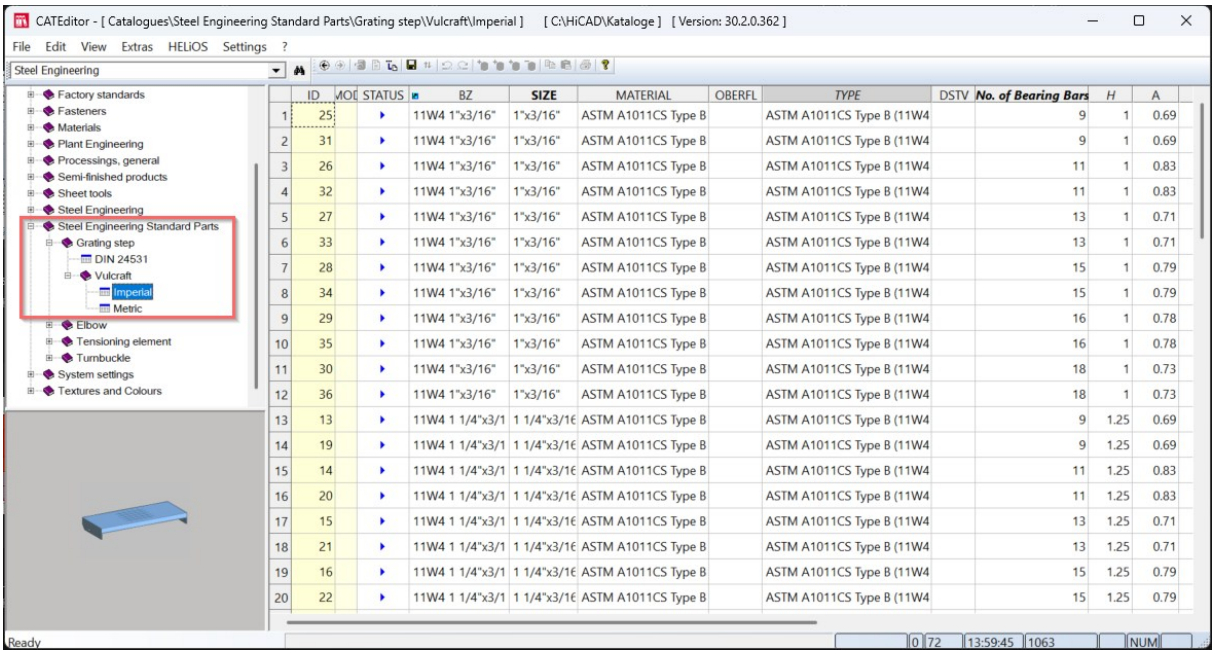
Gratings (Steel Bar Grating)

Grating steps have been added to the catalogue by Vulcraft. You can find them in the table of semi-finished products at **Semi-finished products > Gratings > Vulcraft > Load Tables > ...** Here you will find the **Standard Duty** and **Heavy Duty** versions. Both are available in **Imperial** and **Metric** units.



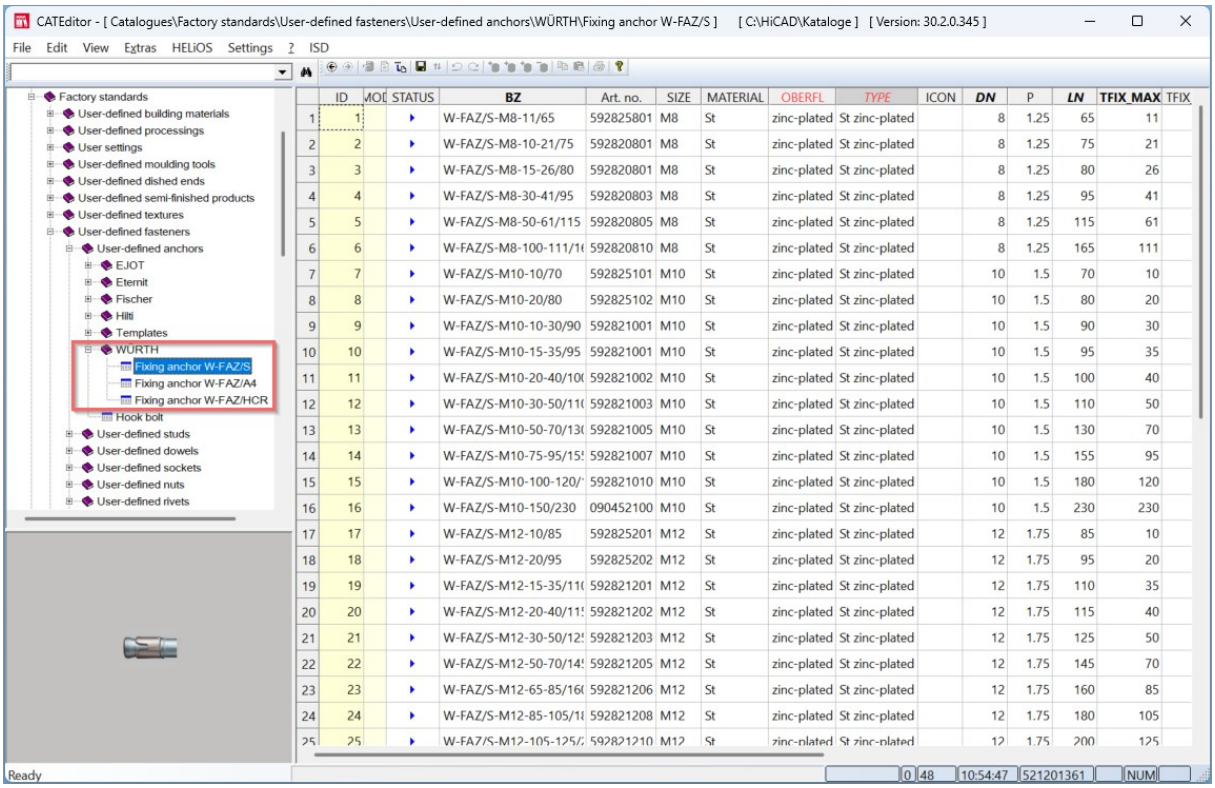
Grating steps (Steel Stair Treads)

A new table has been created for grating steps from Vulcraft. You can find them at **Steel Engineering Standard Parts > Grating step > Vulcraft > ...**. The grating steps are available in **Imperial** and **Metric** units.



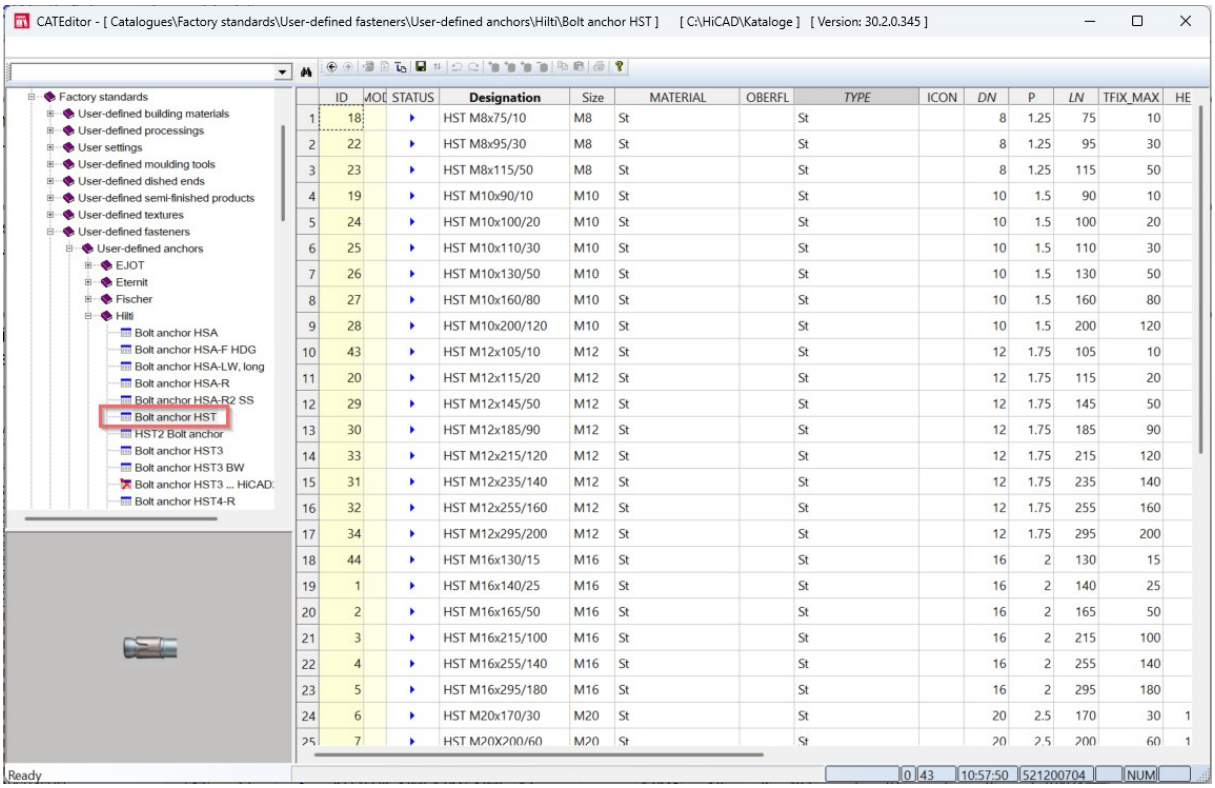
Würth fixing anchor added

Würth added three tables of various fixing anchors of the type W-FAZ to the catalogue. These three fixing anchors can be found at **Factory standards > User-defined fasteners > User-defined anchors > WÜRTH**.



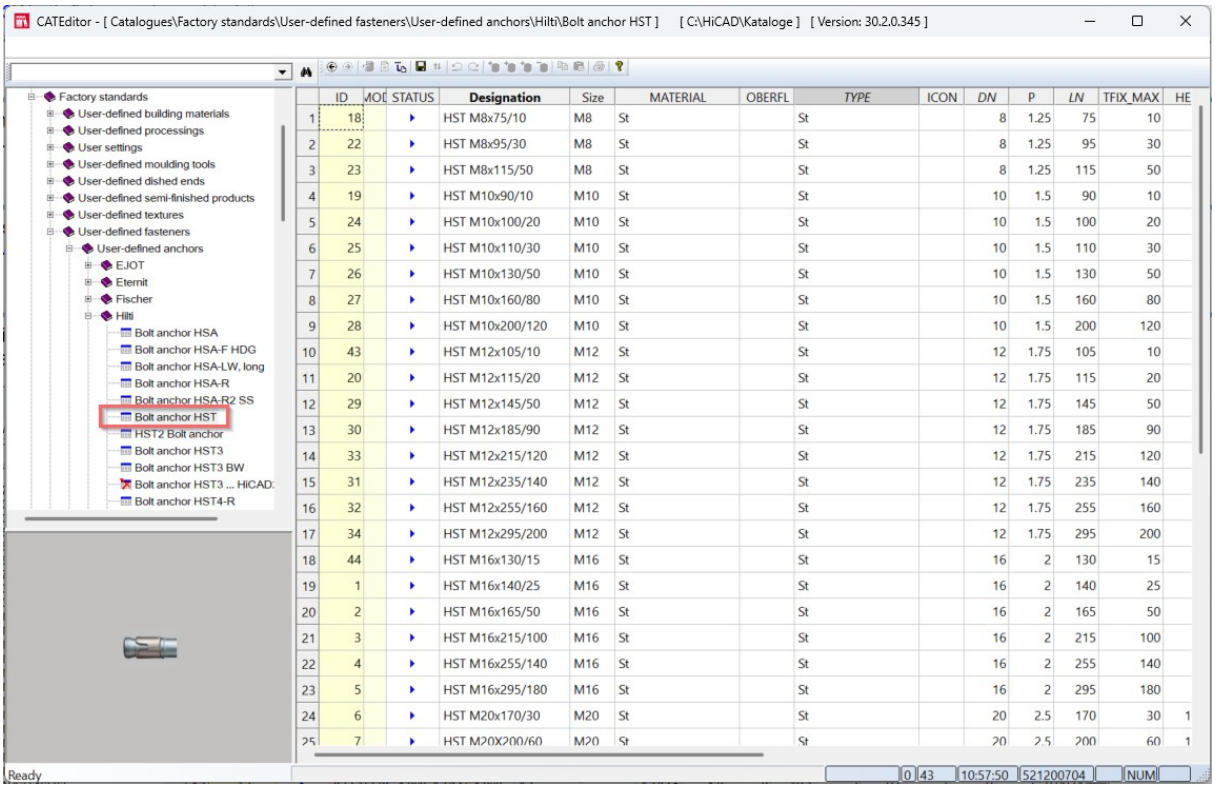
Hilti anchor added

Hilti added the HST-R and HST-HCR bolt anchors. These bolt anchors can be found at **Factory standards > User-defined fasteners > User-defined anchors Hilti > Bolt anchors HST**.



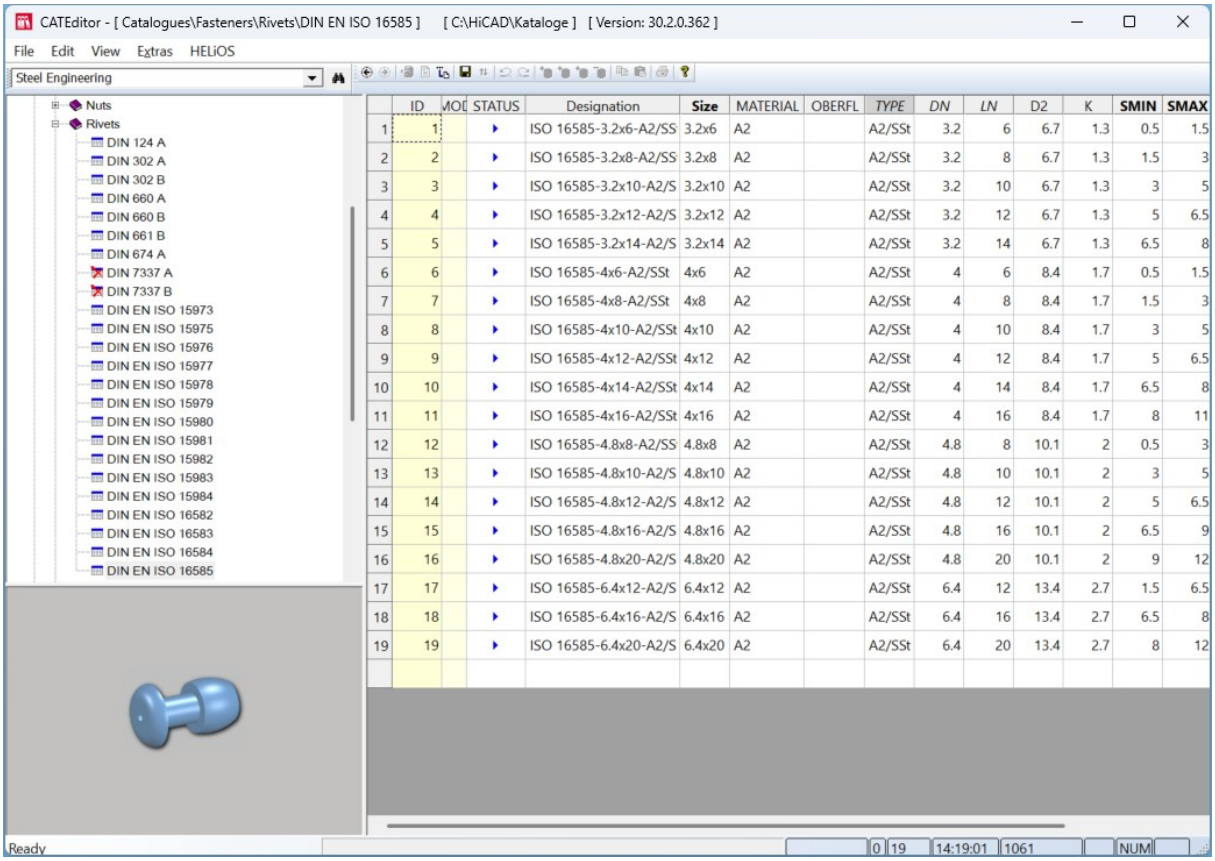
EJOT screws added

A self-drilling screw from EJOT has been added to the catalogue. This self-drilling screw can be found at **Factory standards > User-defined fasteners > User-defined bolts + screws > EJOT > JT3-12**.



Blind rivet added to catalogue

The blind rivet DIN EN ISO 16585 has been added to the **Fasteners** catalogue . You can find it at **Fasteners > Rivets > DIN EN ISO 16585**.



Access to purchased parts and factory standard parts via the Bolting function

If you have saved fasteners in the **Purchased/Factory standard parts** catalogue, you can access this catalogue in the **New bolting/riveting**  function dialogue window and insert the corresponding elements into the drawing as bolting.

Templates for anchors

In the **Factory standards** catalogue, template tables for bolt anchors and undercut anchors have been added at **User-defined fasteners > User-defined anchors > Templates**. You can use these templates to create your own anchors. Each of the two tables has a table file ISD_VORLAGE_BOLZENANKER.IPT or ISD_VORLAGE_HINTERSCHNITTANKER.IPT and a corresponding KRA file, which contains the parameterised 3-D part. The files are located in the HiCAD subdirectory **Catalogues > Factory standards**.

To use a template, you must first create a copy of the corresponding table file. A copy of the KRA file with the same name is also created automatically. You can then customise the anchor template according to your individual requirements, either by extending the records in the table file or by editing the KRA file in HiCAD.

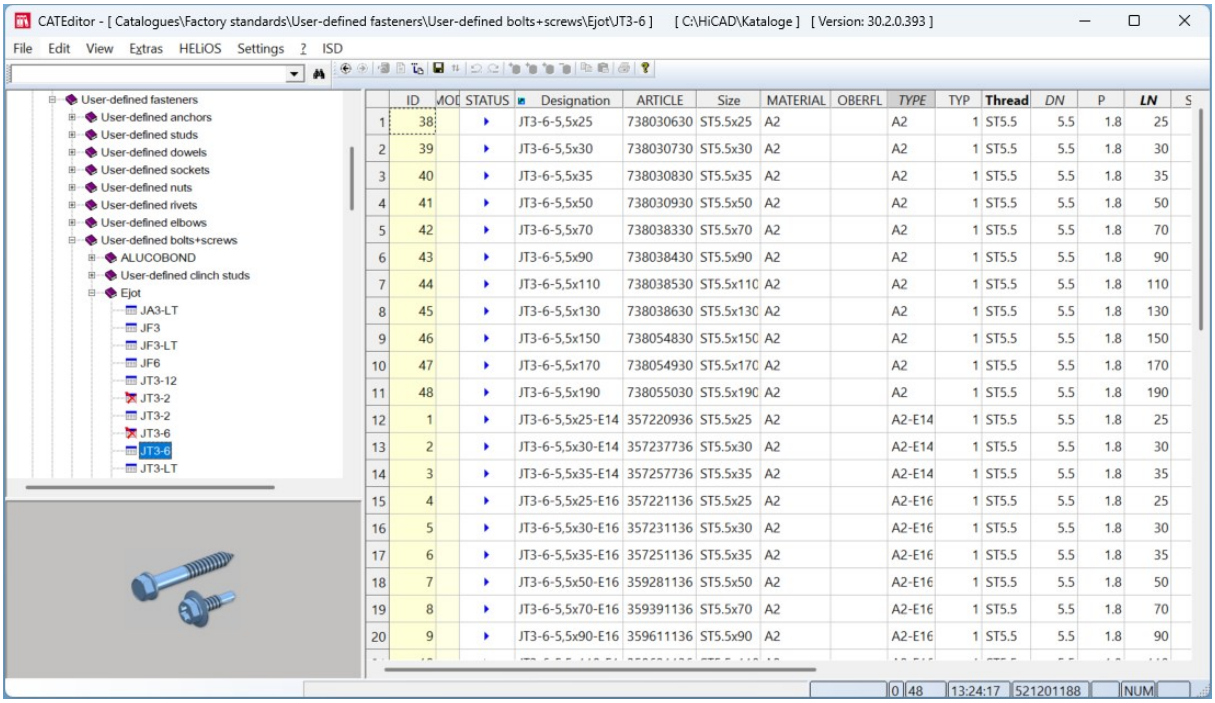
Further information on creating your own anchors using the template tables can be found here.

Corrections in the Hilti anchor catalogue

In the **Factory standards > User-defined fasteners > User-defined anchors > Hilti > HST 3 bolt anchors** catalogue, the same designations were used for anchors with different thread lengths. When inserting anchors using the function **3-D Standard > Standard Parts > New bolting/riveting**  , this meant that not all available anchors were available for selection. The duplicate table entries have been removed and the bolting dialogue for anchors has been expanded so that the selection of an anchor can be specified using the options **Anchoring depth** and **Thickness of filler plates**.

EJOT tapping screws without sealing washer

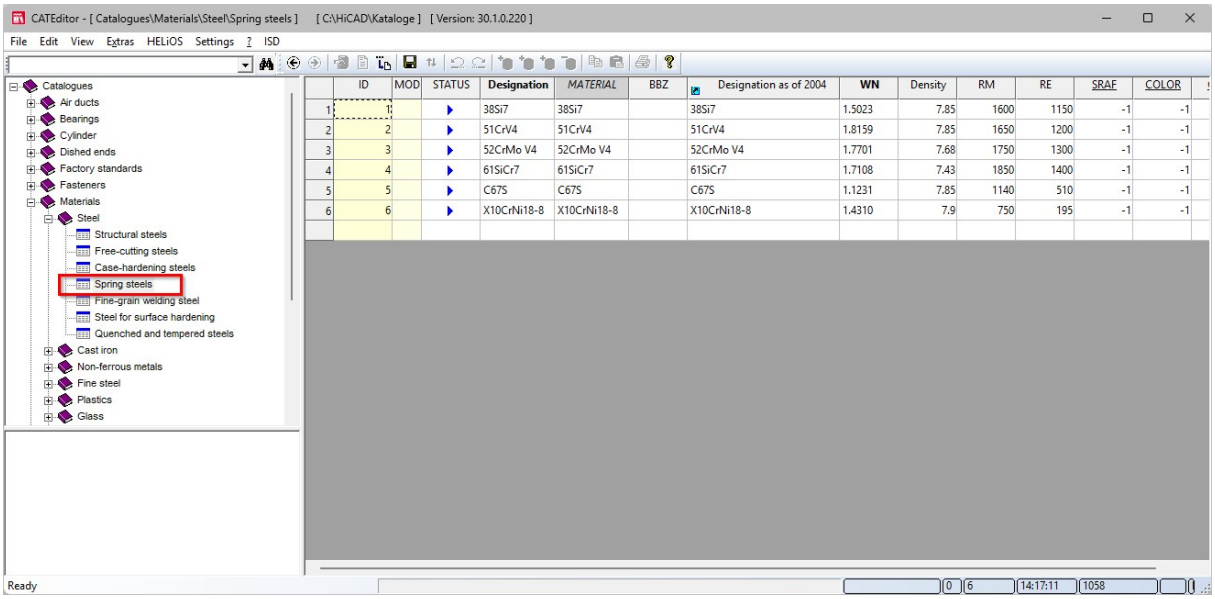
The screw JT3-6-5.5 by EJOT is also available without sealing washer. this screw has been added to the catalogue. You can find them at **Factory standards > User-defined fasteners > User-defined bolts+screws > EJOT > JT3-6**.



Service Pack 1

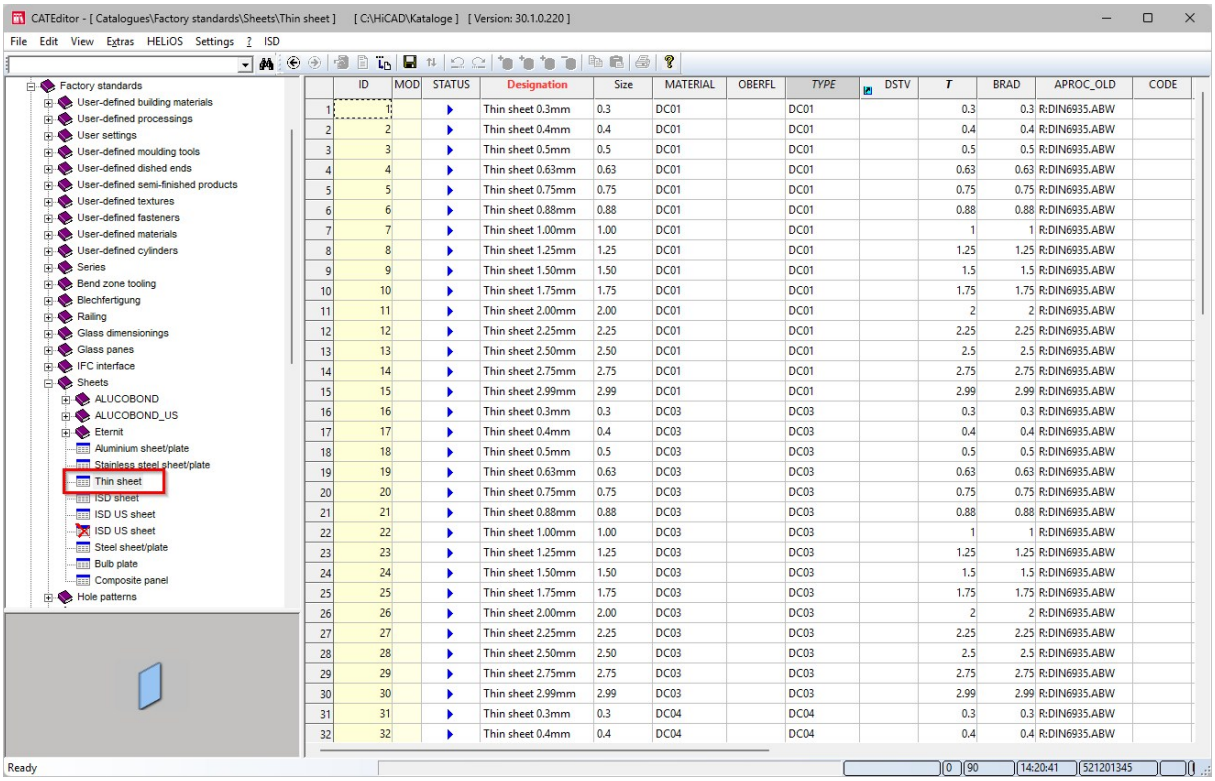
Spring steels in the catalogue

In the **Materials** catalogue, the **Spring steels** table with six spring steels has been added to the **Steel** folder.



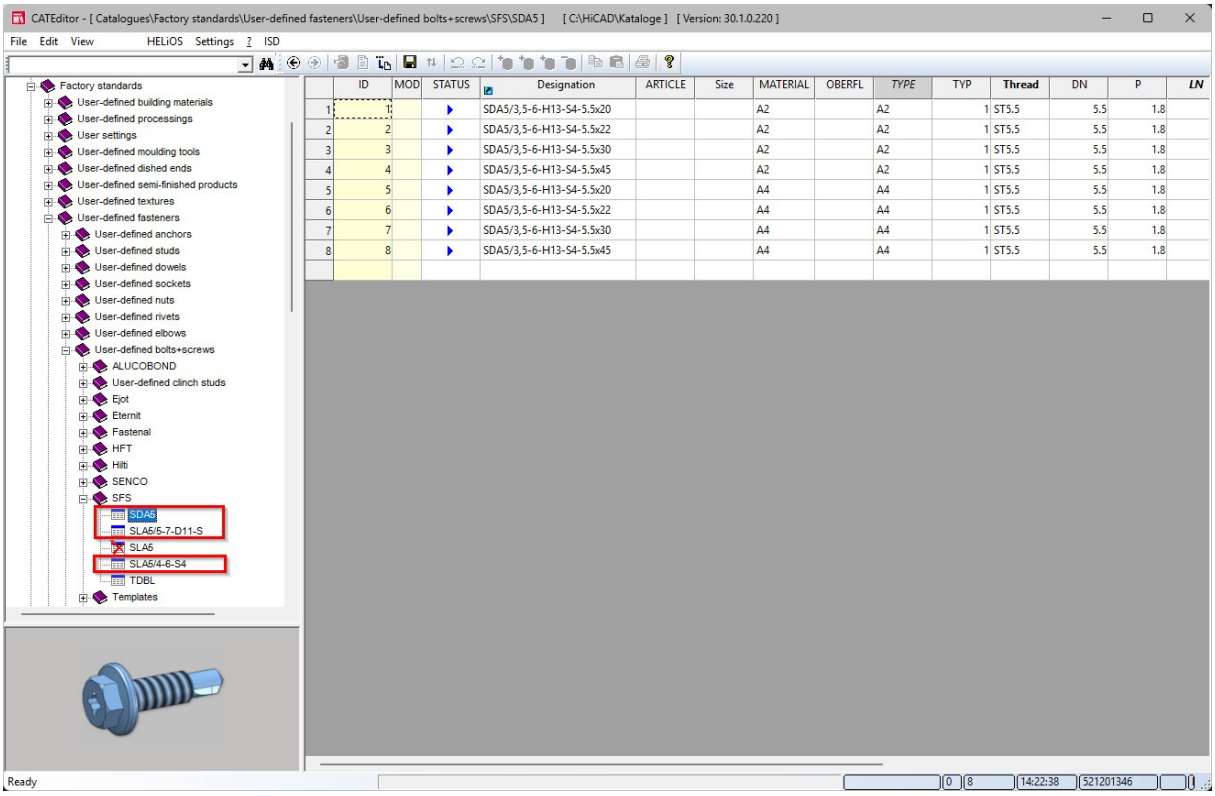
Thin sheets in the catalogue

A new table, **Thin sheet**, has been added to **Factory standards > Sheets**.



ALUCOBOND SFS screw

The **Factory standards** catalogue has been expanded. Three new tables with two **SLA5** screws and one **SDA5** screw have been added to **User-defined fasteners > User-defined bolts+screws > SFS**.

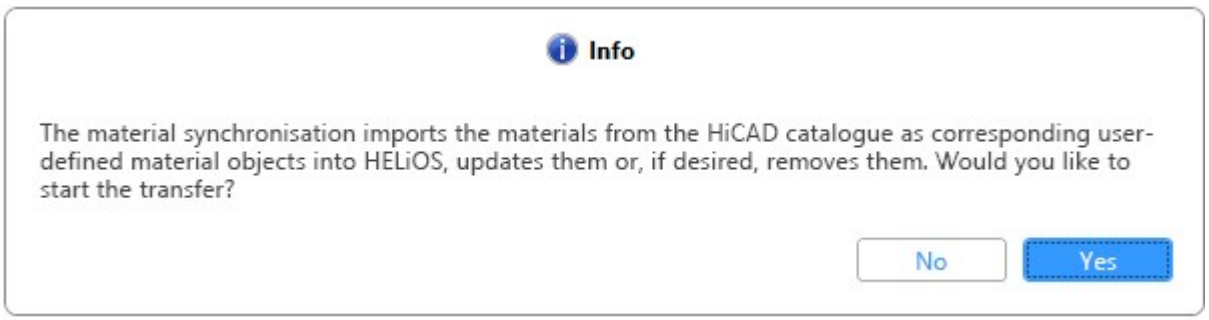


Additional thicknesses for sheets

In the **Factory standards** catalogue, several tables in the **Sheets** folder have been modified. This affects the tables **ISD sheet**, **Bulb plate**, **Steel sheet/plate** and **Stainless steel sheet/plate**, which have now been supplemented by several sheet thicknesses.

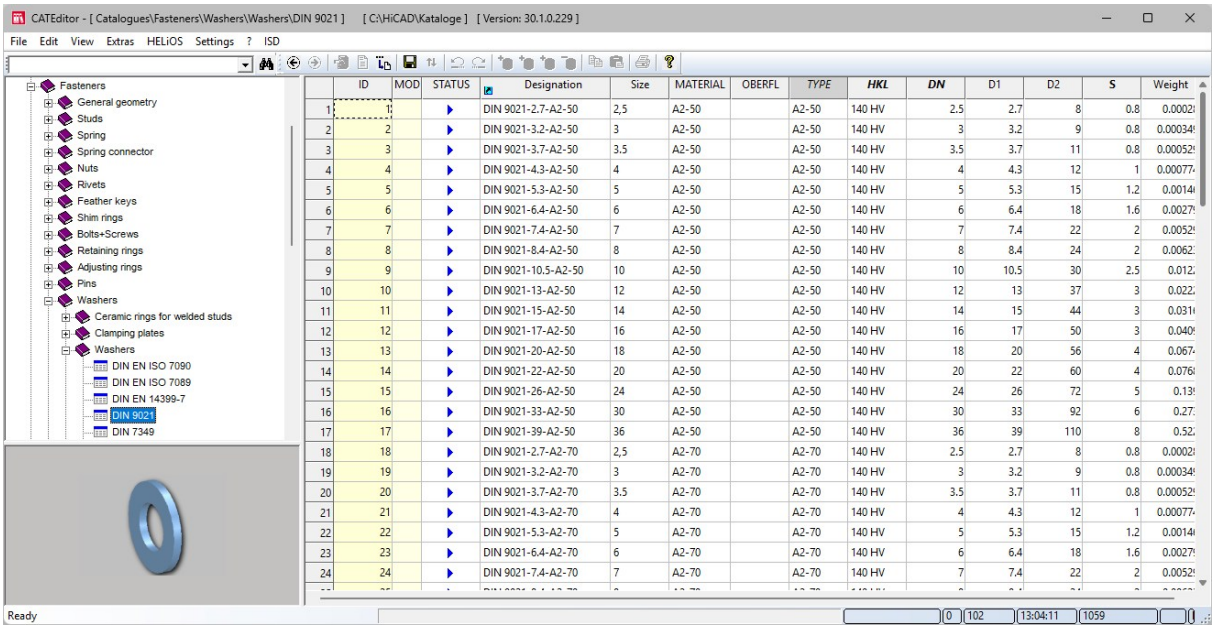
Query for material sync

When you call up the Material synchronisation, from Service Pack 1 onwards, a security prompt appears first asking whether you really want to carry out the process.



Washer DIN 9021

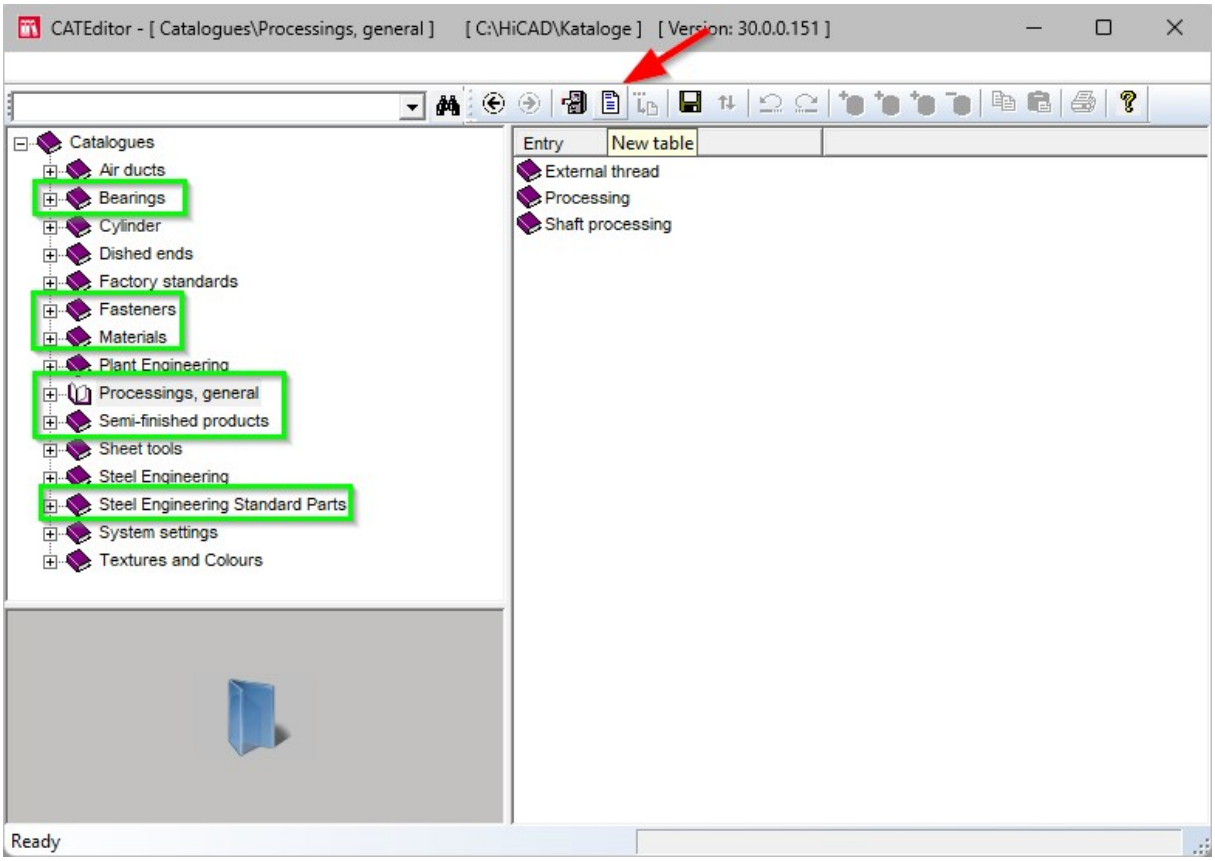
Washers according to DIN 9021 have been added to the catalog. These can be found at **Fasteners > Washers > Washers > DIN 9021**.



Major Release

Create tables in catalogues

It is now possible to create new tables in the following catalogues: **Semi-finished products**, **Processings, general**, **Steel Engineering Standard Parts**, **Fasteners**, **Bearings**, **Materials**.



Attribute management now in the Configuration Editor

Previously, the attributes were managed in the catalogue under System settings in the System attributes table. From HiCAD 2025 onwards, the table of HiCAD attributes can be found in the Configuration Editor at **System settings > Attribute management > Attributes**. The **ISDConfigEditor** function has been added to the catalogue under **Extras**, so that the Configuration Editor can be started directly from the catalogue.

Column comments for countersinks

In the catalogue, the column comment has been changed to "Countersink angle" in all tables with countersinks that contain the ALFA column.

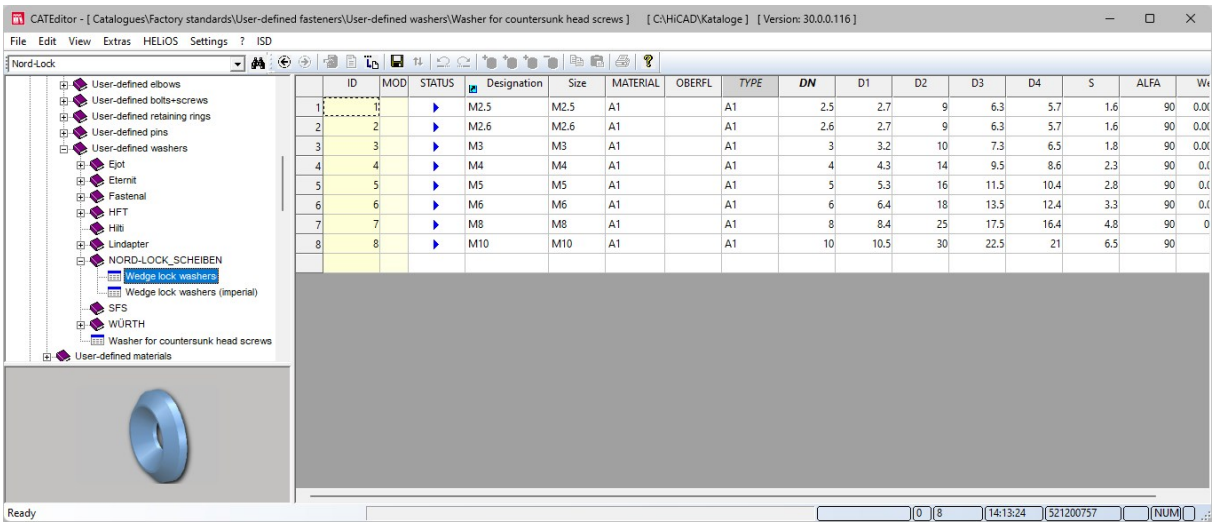
	ID	MOD	STATUS	Designation	Size	TYPE	DN	DB	DS	T	ALFA	TOL	WTOL+-	TOPSYMBOL
1	1			ISO 15065-1.6	1.6	M	1.6	1.8	3.6	0.95				
2	2			ISO 15065-2	2	M	2	2.4	4.4	1.05				
3	3			ISO 15065-2.5	2.5	M	2.5	2.9	5.5	1.35				
4	4			ISO 15065-3	3	M	3	3.4	6.3	1.55				
5	5			ISO 15065-3.5	3.5	M	3.5	3.9	8.2	2.25	90	H 13	1°	
6	6			ISO 15065-4	4	M	4	4.5	9.4	2.55	90	H 13	1°	
7	7			ISO 15065-5	5	M	5	5.5	10.4	2.58	90	H 13	1°	
8	9			ISO 15065-6	6	M	6	6.6	12.6	3.13	90	H 13	1°	
9	10			ISO 15065-8	8	M	8	9	17.3	4.28	90	H 13	1°	
10	11			ISO 15065-10	10	M	10	11	20	4.65	90	H 13	1°	
11	12			ISO 15065-2	2.2	ST	2.2	2.4	4.4	1.05	90	H 13	1°	
12	13			ISO 15065-3	2.9	ST	2.9	3.4	6.3	1.55	90	H 13	1°	

Countersink angle

#Datentyp: Fließkommazahl (ISD ONLY)
#Wertkategorie: Angle (ISD ONLY)
#Werteeinheit: ° (ISD ONLY)

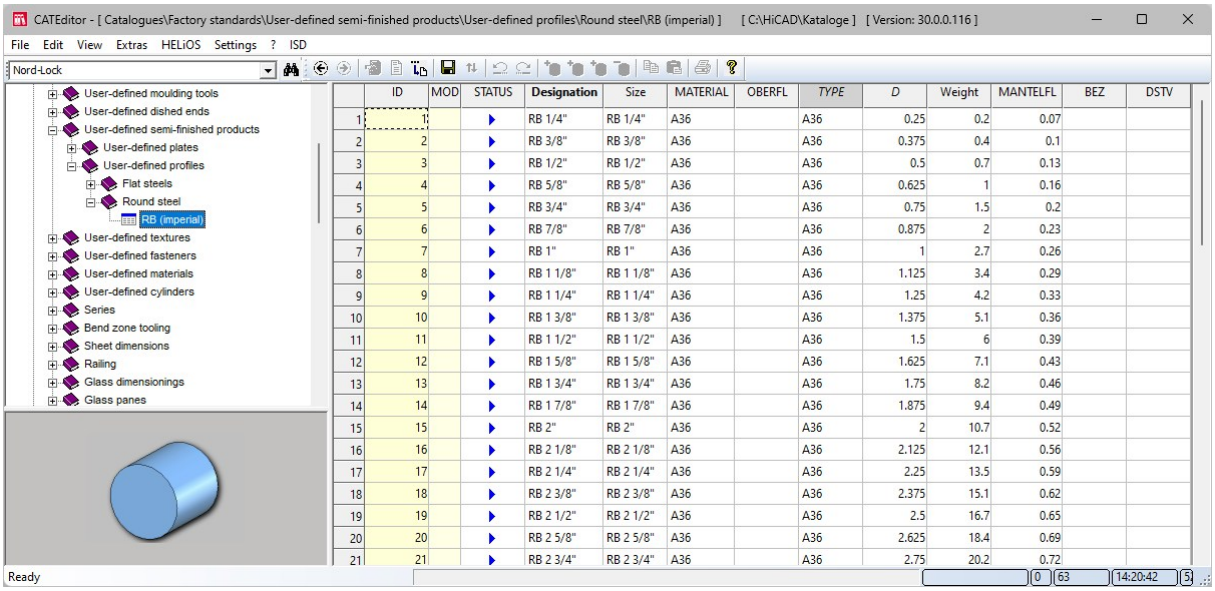
Wedge lock washers added to the catalogue

New in the catalogue **Factory standards > User-defined fasteners > User-defined washers** are the entries **Nord-Lock** and **Würth**. These contain **Wedge lock washers** from the two manufacturers Nord-Lock and Würth.



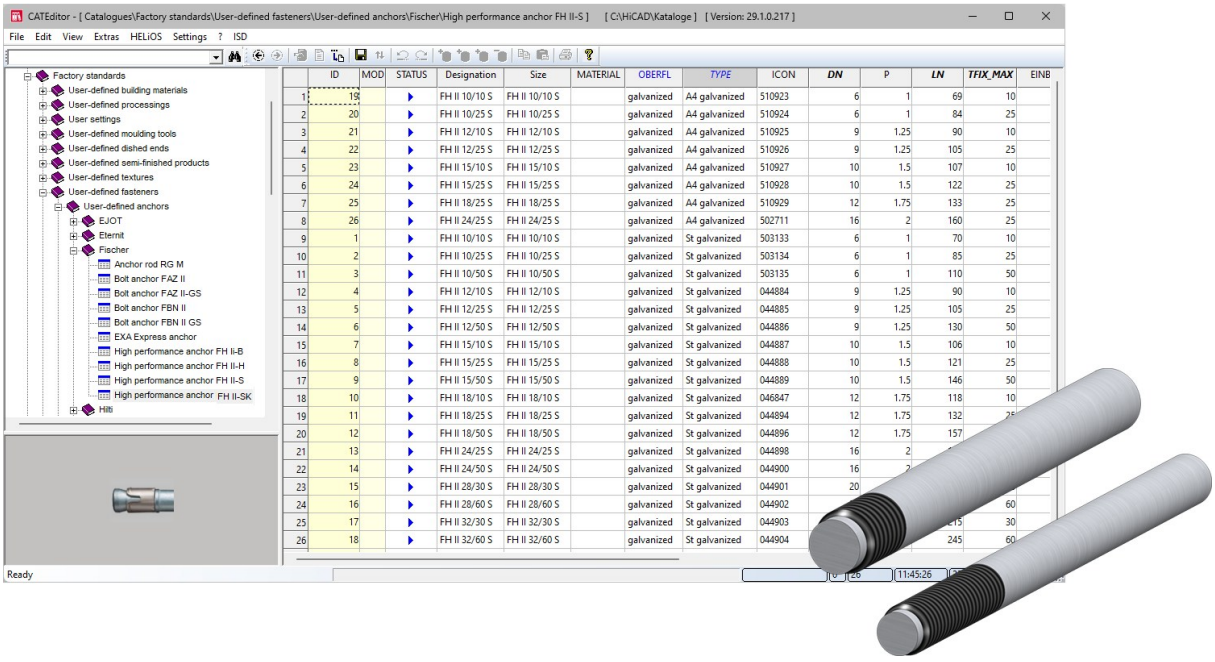
Round steel added for the US market

New in the catalogue **Factory standards > User semi-finished products > User-defined profiles** is **Round steel** for imperial use from the supplier **Dlubal**.



Fischer High performance anchor FH II with countersunk head

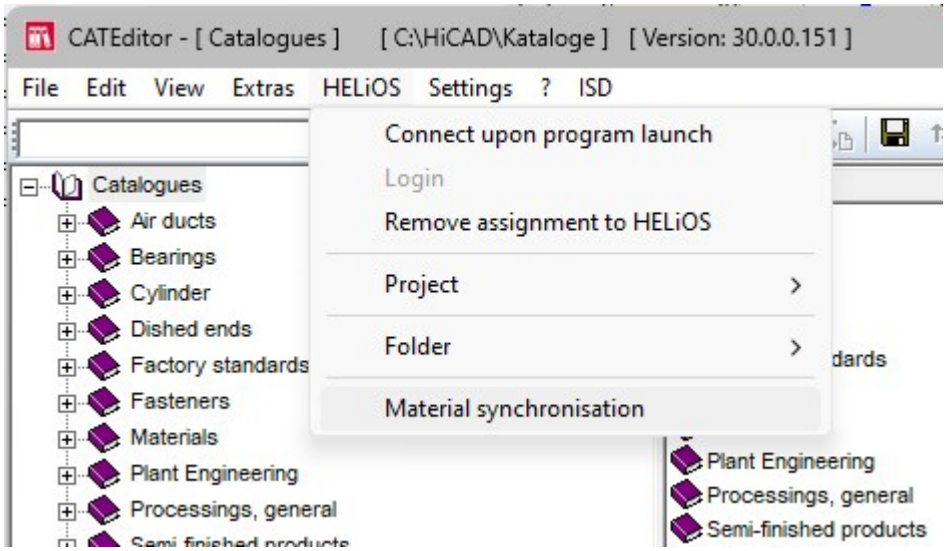
The catalogue **Factory standards\User-defined fasteners\User-defined anchors\Fischer** has been extended by the table **High performance anchor FH II-SK**. The table contains anchors of type Fischer High performance anchor FH II with countersunk head.



Material synchronisation

The **Material synchronisation** function is used to compare and, if necessary, synchronise the HELiOS Database with the materials created in the HiCAD Catalogue.

From Version 2025, you can find it in the Catalogue Editor via the menu item **HELiOS > Material synchronisation**.



In the past, the HELiOS material table was emptied and refilled when the materials were transferred again using the Materialmanager.exe program. With the update, the individual materials are updated. The re-recognition is based on the table and item ID.

Please note the following in this context: With the update to HELiOS 2025, some attributes that are no longer needed on the HELIOS side have been removed from the material database (UT_WERKSTOFF).

This means that these attributes are no longer present in the delivery state of the standard database of HELiOS 2025 (or newer).

If you are already working with a database and HELiOS masks/result lists in which these attributes are used and filled with values, nothing will change.

The attributes in question are: BEZ, BIM, WSD, COLK, COLOR, CUSTOM1, CUSTOM2, CUSTOM3, CUSTOM4, CUSTOM5, ID, LART, MOD, RE2, RM2, SCHI, SRAF.

Bill of Materials / Report Manager

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.

Service Pack 2

New attributes for the Report Manager

The new attributes for Steel Engineering beams and profiles and Steel Engineering plates have been integrated into the Report Manager export files**rm_h_exportpart.HDB** and **rm_db_exportpart.HDB**.

Attributes for Steel Engineering beams and profiles:

- §WBL, Weight by length
- §CBL, Commercial weight by length
- §SBL, Surface area by length
- §VBL, Volume by length

and Steel Engineering plates:

- §CBA, Commercial weight from rectangular area
- §SBA, Surface area from rectangular area
- §VBA, Volume from rectangular area

Activate BOM-relevance

When you use the following functions for creating parts in the **3-D Standard** Ribbon tab:

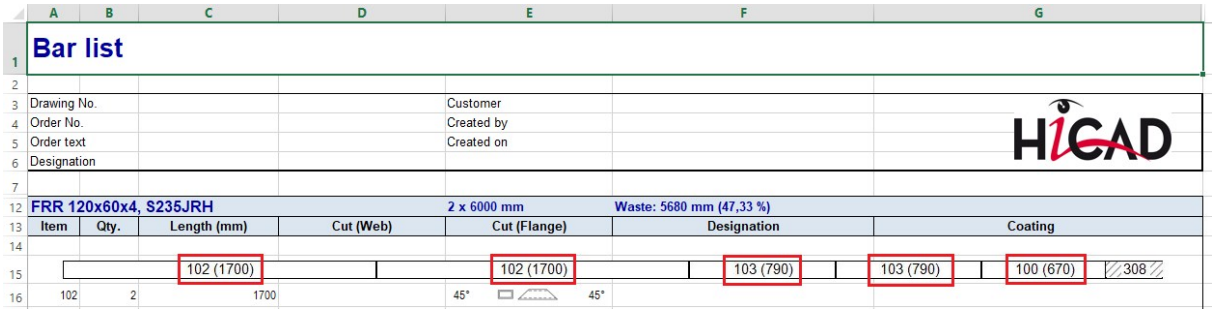
- **Assembly**
- **Solid primitive**
- **Extruded part**
- **Revolved part**
- **C-edge sweep**
- **Part from 3-D sketch**

the **BOM-relevance**  option is now active when you start HiCAD for the first time. If you change this setting in the individual functions, the changed setting is retained even after you restart HiCAD.

The default settings can be restored using HicadGUIReset.exe.

Bar list: Non-numeric values for item texts

When creating bar lists, the item number and length of the beam are displayed on the beams by default, as shown in the image below. However, other attributes can be displayed instead of the item number. This can be set in the corresponding script HiCAD_Stahlbau.30.0.cs in the HiCAD SYS directory. Previously, an error message was displayed for attributes whose value was text rather than a number. From HiCAD 2025 SP 2 onwards, it is also possible to select attributes that contain text as annotations.



Service Pack 1

Configuration of product structure

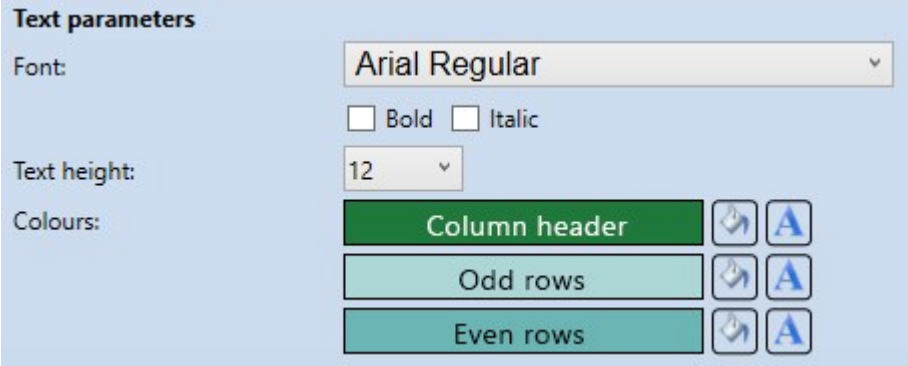
When the product structure is transferred to the Report Manager, a configuration file is evaluated in which the transfer of attributes is stored. If you want to use a different configuration file, you can set this in the **HELiOS Options** under the new menu item **Product structures > Report Manager**.

Major Release

Coloured HTML tables

Column colour

You can now also design the HTML tables in colour. The **Text parameters** area has been added to the **Settings**.



There you can set the text font, the text style (bold, italic) and the text height. The colours for the tables, separated by **Column header**, **Even rows** and **Odd rows**, are defined below. A text and background colour can be selected in each case.

Level	Part name	Material	Item	Qty.	Designation 2	Part type	Cut (Flange)	Cut (Web)	Designation
1	635347842		1	1	1x Railing	Assembly			Railing
2	635347873		1	1	1x Segment	Assembly			Segment
3	635349055		1	3	3x Infill	Assembly			Infill
4	422617710	S235JR	1	21	7x Fl 20x5-848.56323 S235JR	Flat steel	32.9° /--------I	I--------I	Fl 20x5
4	422535814	S235JR	2	3	1x Fl 30x6-1161.44709 S235JR	Flat steel	I--------I	I--------I	Fl 30x6
3	635350511		2	1	1x Infill	Assembly			Infill
4	422617710	S235JR	1	1	1x Fl 20x5-850.18048 S235JR	Flat steel	32.9° /--------I	I--------I	Fl 20x5

Print coloured BOMs

The **Print (Settings)**  function provides the same options for the colour design of the print output as in the HTML table.

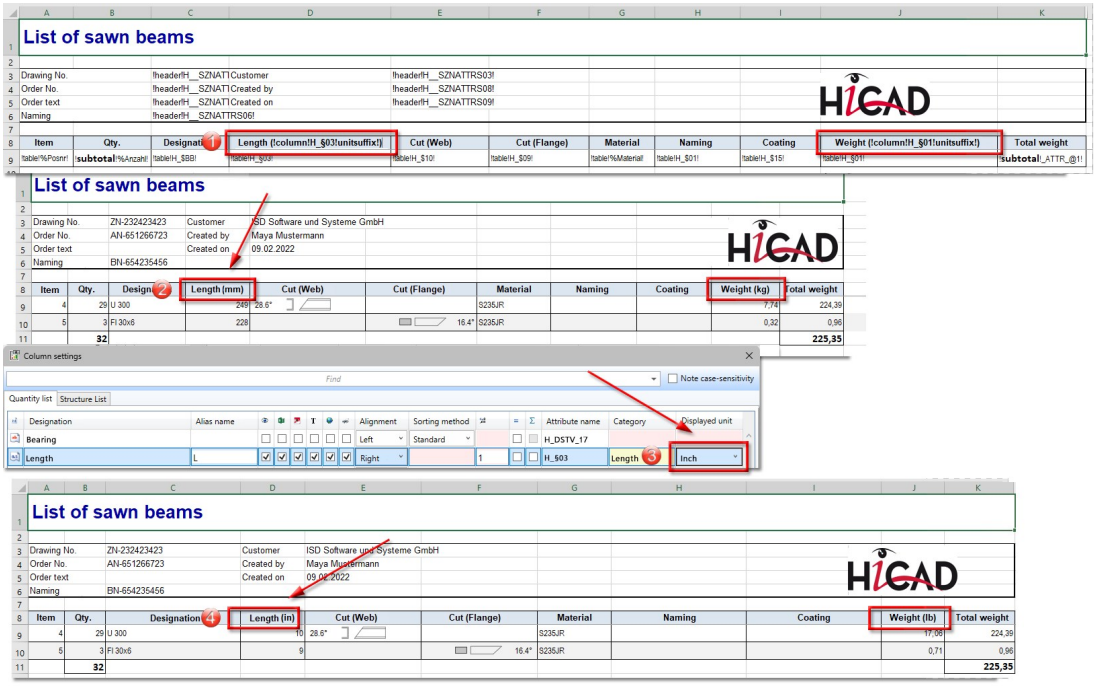
Units in Lists of sawn beams

The displayed units (mm, inch) of the values (e.g. length) are taken from the List of sawn beams as well as the other lists from the Report Manager. If you now want to output inches instead of millimetres, use the **Column settings**  function to set the display unit for the length to inches for the Quantity list . The column header may then need to be adjusted in the Excel template. To do this, open the XLSX file and place a placeholder in the column header. If you then change the unit in the Report Manager, the column headers in the column list will also be changed.

The headers for non-metric units are prepared in the following templates:

- BOMTemplates/HiCAD-DB_Anlagenbau.EN.30.0.xlsx
- BOMTemplates/HiCAD-DB_Mechanik.EN.30.0.xlsx
- BOMTemplates/HiCAD-DB_Stahlbau_BIM.EN.30.0.xlsx
- BOMTemplates/HiCAD_Anlagenbau.EN.30.0.xlsx

- BOMTemplates/HiCAD_Blech.EN.30.0.xlsx
- BOMTemplates/HiCAD_Mechanik.EN.30.0.xlsx
- BOMTemplates/HiCAD_Metallbau.EN.30.0.xlsx
- BOMTemplates/HiCAD_PipeBook.EN.30.0.xlsx
- BOMTemplates/HiCAD_Stahlbau.EN.30.0.xlsx
- BOMTemplates/Proflex.EN.30.0.xlsx



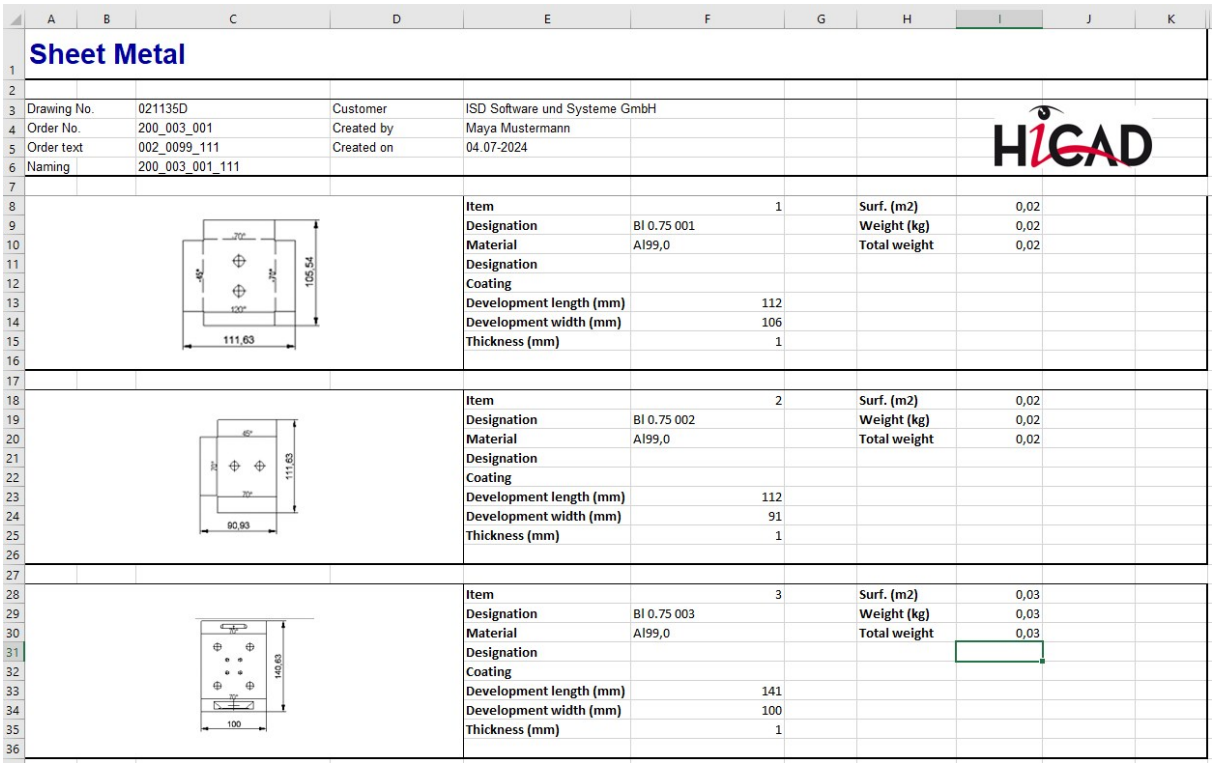
(1) Template HiCAD-DB_Stahlbau_BIM.EN.30.0.xlsx, (2) List of sawn beams, (3) Changing the displayed unit in the Report Manager, (4) Automatically changed column header

Sheet Metal parts with development

In the Excel template HiCAD_Stahlbau.EN.30.0.xlsx, there is now a new sheet in Excel, **Sheet Metal parts with development**. This list is deactivated by default, as generating the developments can be time-consuming. If you need this list, open the **Settings** sheet of the HiCAD_Stahlbau.EN.30.0.xlsx file and set the value in the Generate column from **false** to **true**.

Name	Create	Structure list
Structure list	true	true
Quantity list	true	false
Total profiles list	true	false
Profile structure list	true	false
Packaging list	true	false
Packaging list 2	false	false
Quantity list, Profile installation	true	false
List of sawn beams	true	false
Bolt|screw list	true	false
Bolt|screw list		
Shipping list	true	true
Shipping list, short	true	false
Shipping list, short, w. image	true	false
Plates	true	false
Sheet Metal	true	false
Sheet Metal parts with image	true	false
Sheet Metal parts with image 2	false	false
Sheet Metal parts with development	true	se
Unprocessed sheets	true	false
Unprocessed SM prts w. img	false	false
Steel plates and metal sheets	true	false
Steel plates and metal sheets with im	true	false
Gratings	true	false
Glazing	true	false
Bar list	true	false
Bar - Summary	true	false
	false	false
	false	false
	false	false

Template HiCAD_Stahlbau.EN.30.0.xlsx, **Settings** sheet



Sheet Metal

Drawing No.	021135D	Customer	ISD Software und Systeme GmbH
Order No.	200_003_001	Created by	Maya Mustermann
Order text	002_0099_111	Created on	04.07-2024
Naming	200_003_001_111		

Item	1	**Surf. (m2)**	0,02
Designation	Bl 0.75 001	**Weight (kg)**	0,02
Material	Al99,0	**Total weight**	0,02
Designation			
Coating			
Development length (mm)	112		
Development width (mm)	106		
Thickness (mm)	1		

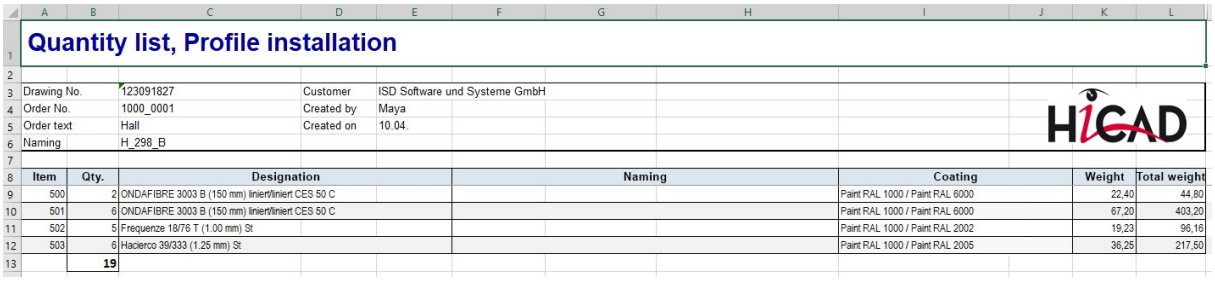
Item	2	**Surf. (m2)**	0,02
Designation	Bl 0.75 002	**Weight (kg)**	0,02
Material	Al99,0	**Total weight**	0,02
Designation			
Coating			
Development length (mm)	112		
Development width (mm)	91		
Thickness (mm)	1		

Item	3	**Surf. (m2)**	0,03
Designation	Bl 0.75 003	**Weight (kg)**	0,03
Material	Al99,0	**Total weight**	0,03
Designation			
Coating			
Development length (mm)	141		
Development width (mm)	100		
Thickness (mm)	1		

Output as Excel document with template; **Sheet Metal parts with development** sheet

Quantity list for profile installation

In the Excel template HiCAD_Stahlbau.EN.30.0.xlsx there is now a new sheet with a quantity list for profile installation. For this purpose, the attribute %PI, i.e. the so-called dispatch item number, is evaluated during identical part search. The prerequisite for this is that the parameter **Carry out dispatch itemisation** is activated in the Configuration Editor at **Profile Installation > Dispatch itemisation**. In this way, a dispatch itemisation is assigned in addition to the item number based on the uncut elements.



Quantity list, Profile installation

Drawing No.	123091827	Customer	ISD Software und Systeme GmbH
Order No.	1000_0001	Created by	Maya
Order text	Hall	Created on	10.04.
Naming	H_296_B		

Item	Qty.	Designation	Naming	Coating	Weight	Total weight
500	2	ONDAFIBRE 3003 B (150 mm) liniert/liniert CES 50 C		Paint RAL 1000 / Paint RAL 6000	22,40	44,80
501	6	ONDAFIBRE 3003 B (150 mm) liniert/liniert CES 50 C		Paint RAL 1000 / Paint RAL 6000	67,20	403,20
502	5	Frequenze 18/76 T (1.00 mm) St		Paint RAL 1000 / Paint RAL 2002	19,23	96,16
503	6	Hacierco 39/333 (1.25 mm) St		Paint RAL 1000 / Paint RAL 2005	36,25	217,50
	19					

Configuration Management

Service Pack 2

Attribute management

Attribute calculation

The texts at System settings > Attribute management > Attribute calculation have been revised for better understanding. In addition, the attributes are now listed with their names in the comment column (e.g. attribute Weight (§01) is assigned).

Attributes and referencing

The settings for synchronizing attributes during referencing can now be found at System settings > Attribute management > Attributes. For this reason, the Referencing column has been added to the list. This eliminates the attribute list at System settings > Referencing > Synchronization of attributes. New attributes no longer need to be edited in different places in the Configuration Editor. You can make the following settings in the Referencing column:

- Do not transfer: Each occurrence of a part can have different values.
- Always transfer: Each occurrence of a part has the same value.
- Transfer for sub-parts: Different occurrences of an assembly have the same value on their sub-parts.
- Standard behaviour: The behaviour set at System settings > Referencing > Standard behaviour for transferring attributes for referenced parts applies. Before SP2, this parameter was called "Synchronise item number/part attributes when updating".

Attribute description as tooltip

The table with the attributes (at System settings > Attribute management > Attributes) has been expanded to include the Description column. In this column, you can enter multilingual texts to explain the attributes. These texts are then displayed as tooltips in the HiCAD Annotation Editor when you select an attribute.

Duplicating attributes in attribute management

You can now duplicate attributes using the context menu in the attribute management. To do this, right-click on the attribute you want to duplicate and select Copy. Then right-click again to open the context menu and select Paste. This creates another row in the attribute table with the attribute as a user attribute. You can find the table at System settings > Attribute management > Attributes.

Attribute calculation upon change

If changes are made to externally referenced sub-parts of locked assemblies or parts, the value of the **Weight**. attribute may also change. The weight is only updated for locked assemblies if you activate the **Weight calculation ignores lock for assemblies** parameter in the Configuration Editor under **Compatibility** > **Attribute calculation**.

In this context, the text in the selection box has been changed for several parameters under **Attribute calculation** (System settings > Attribute management). In addition to **Do not auto-calculate** and **Manual / When itemising**, there is now the option **With every change** instead of **Always**.

Manufacturability check

The manufacturability check in the Design Checker can now also be performed when creating developments. To do this, activate the **Perform manufacturability check when creating developments** parameter in the Configuration Editor at **Sheet Metal > Manufacturability check**.

The following new checks are now available:

- **Minimum flange length**

  The minimum length of flanges is measured directly on the outside or directly on the outer tangent, as with the Attach function. Only the outside of the cover surface is measured; chamfers can reduce this. A distinction is made between acute and obtuse angles.

- **Minimum bend radius**

  To avoid overstretching the material, a certain bend radius should not be exceeded.

- **Distance between processings**

  To check the distance between processings in the manufacturability check, activate the parameter **Check minimum distance between processings** in the Configuration Editor.

- **Minimum diameter for standard bores**

  This test only applies to standard bores.

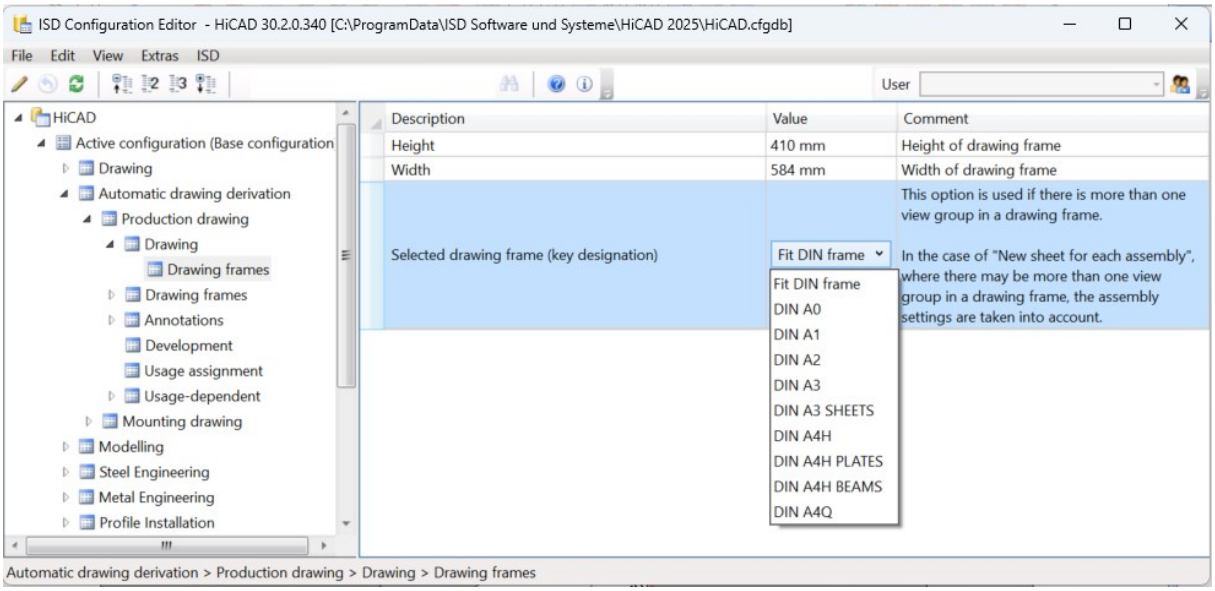
- **Minimum Z-fold height**

  The comparison values for this test are loaded from the catalogue. You must therefore first fill in the columns **Min. Z-fold height (<90°)** and **Min. Z-fold height (>=90°)** of the corresponding tables.

Default setting for drawing frame

In the Configuration Editor, you can now select the size of the drawing frame via a listbox at **Automatic drawing derivation > Production drawing > Drawing > Drawing frame > Selected drawing frame**.

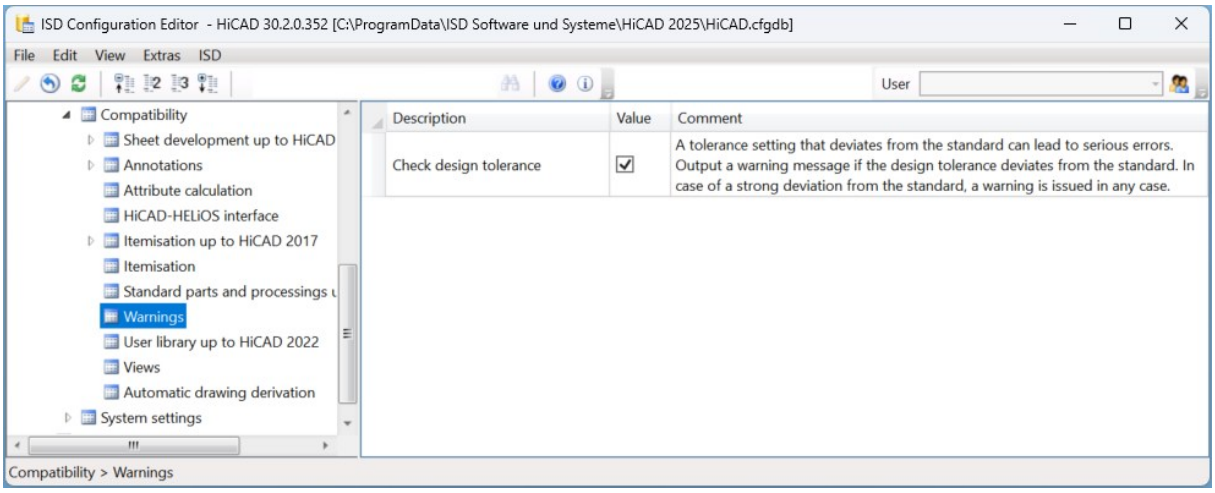
If a certain frame is preselected here, it can be adopted when creating a production drawing. To do this, activate the **From configuration** checkbox in the **Drawing derivation** dialogue window under **Drawing parameters**.



Design tolerance

HiCAD stores all coordinates with the greatest possible accuracy. Due to rounding errors, numerical values, e.g. point coordinates, can be checked for equality up to a certain tolerance size. The default value is 0.001. This means that two points are considered identical if their distance is less than a thousandth of a millimetre. If you change the tolerance, a warning message appears when you load the drawing.

You can deactivate the message in the Configuration Editor at **Compatibility > Warnings > Check design tolerance**, provided that the value is between 0.0001 and 0.01.



Service Pack 1

Manufacturability of Sheet Metal parts

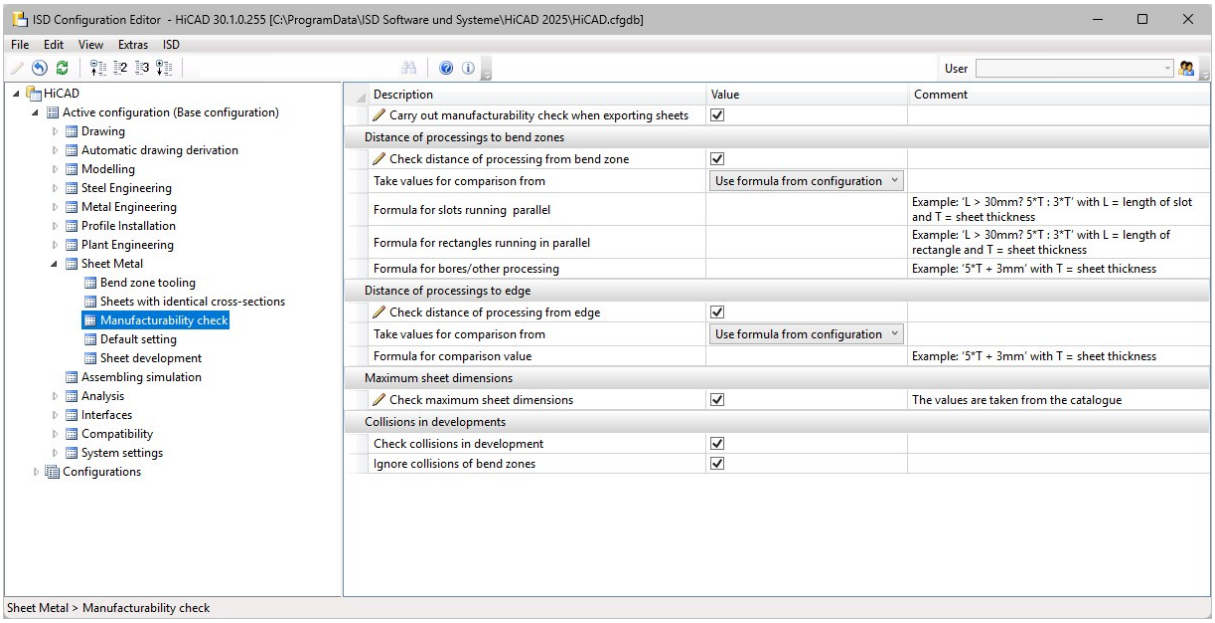
In Configuration Editor, you can configure the new check for the manufacturability of Sheet Metal parts. The check can then be carried out when exporting sheet developments, or in the Design Checker. The manufacturability check combines the existing checks

- Check maximum sheet dimension
- Check collisions in development (with and without bend zone)

and the new checks

- Check distance of processing from bend zone
- Check distance of processing from edge

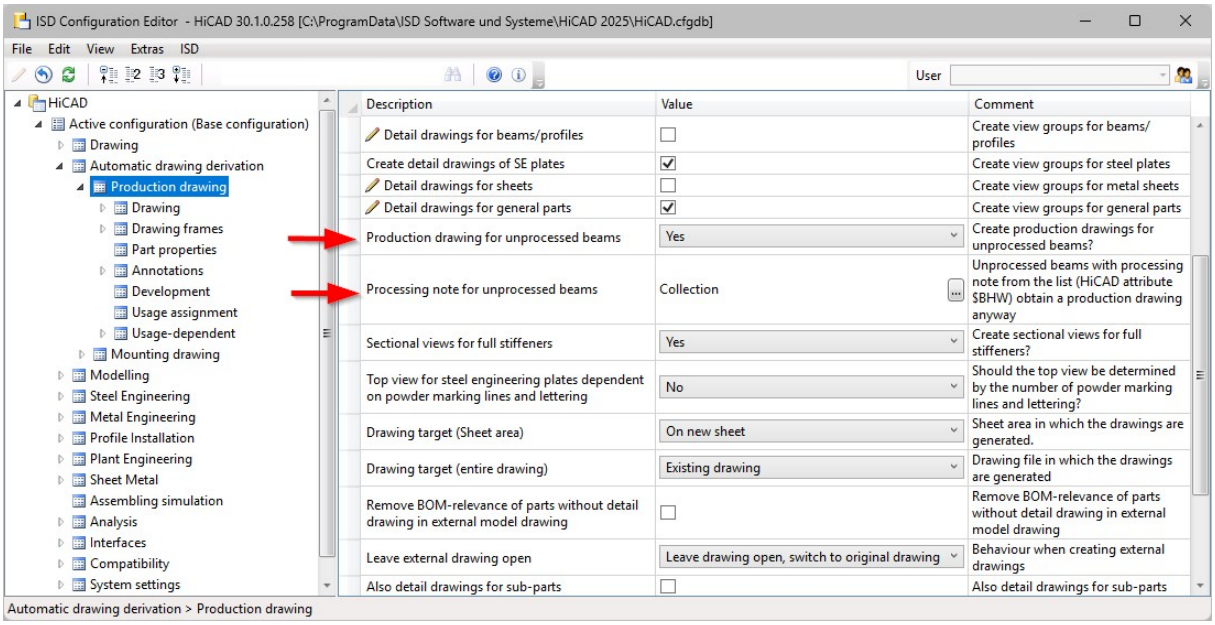
You can find the parameters at **Sheet Metal > Manufacturability check**.



Production drawing / processing notes for unprocessed beams and profiles

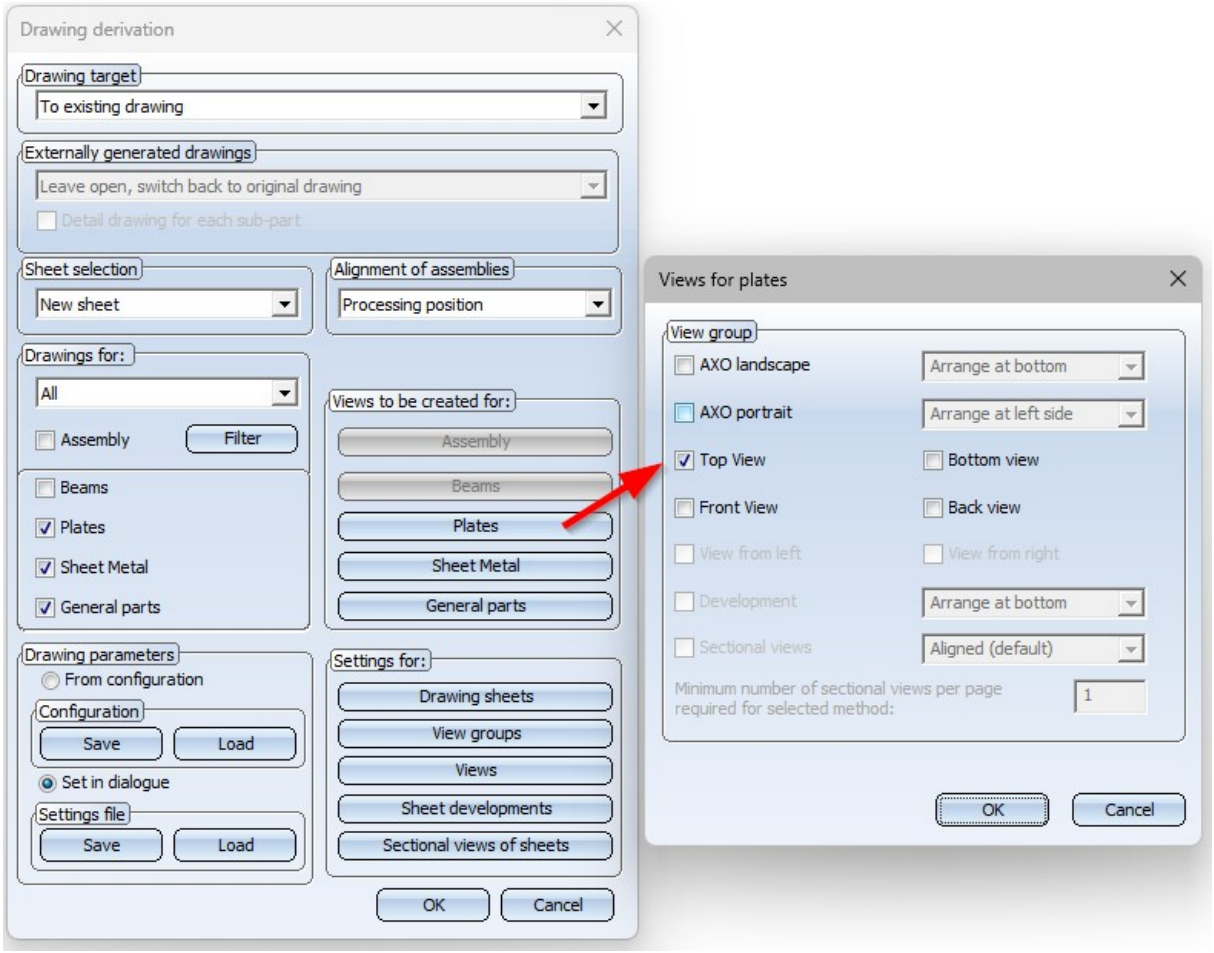
In the drawing, a distinction is made between processed and unprocessed beams and profiles based on the feature entries. As of SP1, this is not only taken into account for Drawing Management but also in drawing derivations.

Therefore, the two parameters **Production drawing for unprocessed beams** and **Processing note for unprocessed beams** can now be found at **Automatic drawing derivation > Production drawing**.



Setting for Steel Engineering plates in drawing derivation

For the drawing derivation of Steel Engineering plates, the parameter **Top view for steel engineering plates dependent on powder marking lines and lettering** has been added to the Configuration Editor (at **Automatic drawing Derivation > Production drawing**). This parameter is read out when you activate Top view during the drawing derivation of Steel Engineering plates.



If you select **Yes**, the top side is determined depending on the powder lines and lettering when the top view is created. The side with powder lines is selected as the top side. If there are powder lines on both sides of the plate, the side with the higher number of lines is on top. If the plate has both powder lines and lettering, the side with the lettering is on top.

If the default setting **No** is selected, one side of the plate will be chosen at random as the top side.

Attributes

In the HiCAD Configuration Editor, you could previously set whether the accurate calculation (e.g. with notches or subtractions) or the calculation of the minimum bounding rectangle (length x width x thickness) should be displayed for the attributes (§01 Weight, §10 Surface area, §18 Commercial Weight, §20 Volume). From HiCAD SP1, new attributes are available for calculating the bounding rectangle at **System settings > Attribute management > Attributes**, so that the various calculation results are available at the same time

For beams and profiles there are the new attributes:

- **§WBL**, Weight by length
- **§CBL**, Commercial weight by length
- **§SBL**, Surface area by length
- **§VBL**, Volume by length

and for plates and sheets:

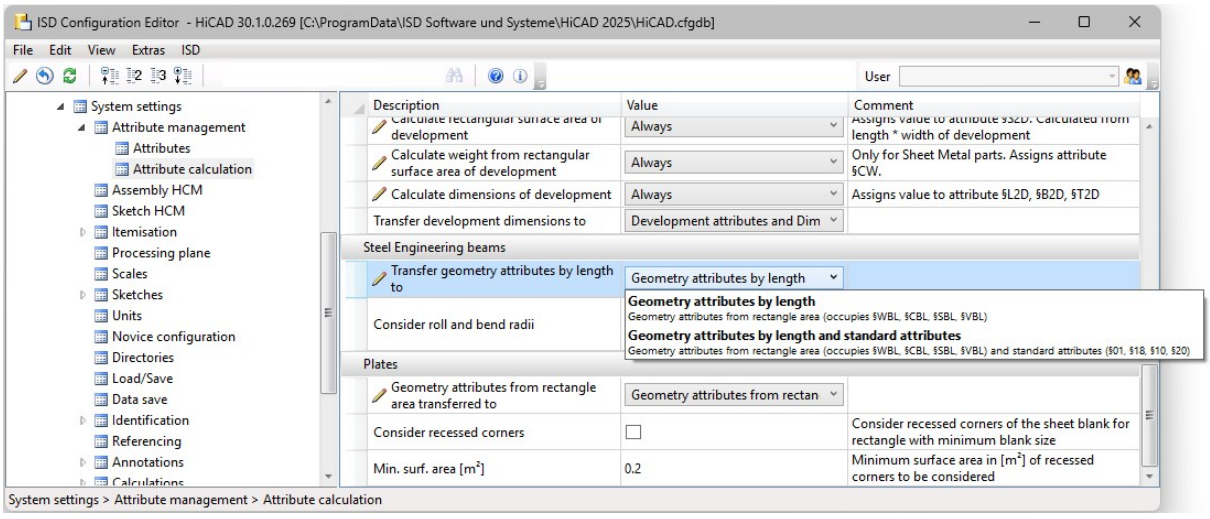
- **§CBA**, Commercial weight from rectangular area
- **§SBA**, Surface area from rectangular area
- **§VBA**, Volume from rectangular area

In the Configuration Editor at **System settings > Attribute management > Attribute calculation**, the following parameters have been changed as follows:

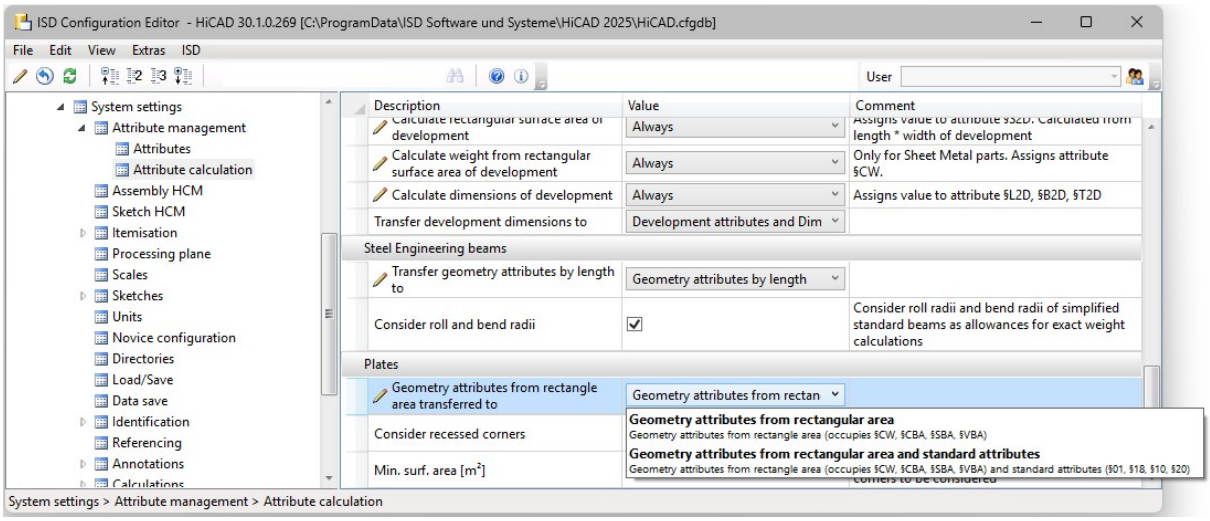
- **Type of weight calculation for SE beams** to **Transfer geometry attributes by length to**
- **Type of weight calculation for SE plates** to **Geometry attributes from rectangular area transferred to**

Here you have the option to transfer the calculation of the minimum bounding rectangle not only to the new geometry attributes (§WBL, §CBL, §SBL, §VBL or §CBA, §SBA, §VBA), but also to the standard attributes (§01, §10, §18, §20).

For sheet developments, the "simplified" weight of the minimal bounding rectangle is written to the **§CW** attribute (moved from **General** to **Sheet**). The §CW attribute evaluates the parameter **Consider recessed corners** at **System settings > Attribute management > Attribute calculation**. The value of **§CW** can change as a result.



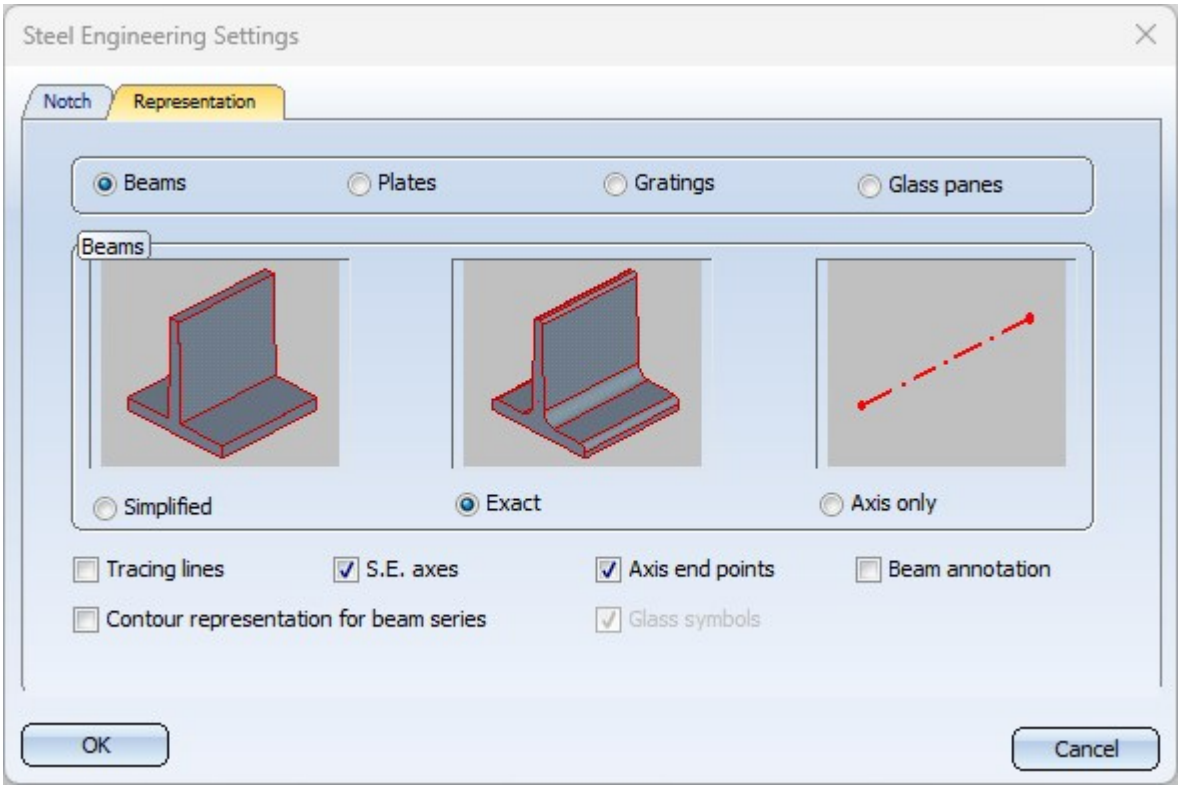
Settings for beams and profiles



Settings for plates and sheets

Due to the extensions in the Configuration Editor, the **Weight calculation** tab of the **Steel Engineering Settings** dialogue (**Steel Engineering > Further functions > Settings**)  was no longer needed and was removed. The settings on the tab only had a temporary effect until the next restart of HiCAD.

The dialogue window now looks like this:



HELiOS Settings in the Configuration Editor

In order to increase user-friendliness, the HELiOS settings for HiCAD have been restructured and summarised.

- The PDM directory is no longer located directly under Active configuration (Basic Configuration), but at **System settings**.
- The settings at **PDM > HiCAD - HELiOS interface** and at **System settings > HELiOS** can now be found combined at **System settings > PDM > HiCAD - HELiOS Interface**.
- The settings at **PDM > HiCAD - HELiOS interface > Product structure** can now be found at S**ystem settings > PDM > HiCAD - HELiOS interface**.
- The setting **Transfer product structure attributes to part attributes when updating HELiOS attributes** can be found at **Compatibility > HiCAD - HELiOS interface**. This is also where you will find the new setting **Allow loading/saving with interrupted HELiOS connection**.

New function for creating a structure assembly

Previously, there was no direct function for creating structure assemblies. As of SP1, the functions **Create structure assembly** and **Create structure assembly, as sub-part** , are available via **3-D Standard > New > New assembly**.

The settings for assemblies in the Configuration Editor were previously located at Steel Engineering > Assemblies and can now be found at **Modelling > Part creation > Assemblies**. As part of the new functions for structure assemblies, the settings have been extended to include the new entry **Article number for structure assemblies**.

P+ID

The settings in the Configuration Editor for P+IDs can now be found at **Plant Engineering > P+ID > P+ID (RPA)**.

Major Release

Attribute management

As of HiCAD 2025, the system attributes are no longer managed in the catalogue but in the Configuration Editor at **System settings > Attribute management**. When installing an update, the catalogue update transfers the attribute information from the catalogue to the Configuration Editor.

The definition of the drawing and part attributes can be found at **System settings > Attribute Management > Attributes**. **New** and **Delete** buttons can also be used to create and edit new, customer-specific attributes.

At **System settings > Attribute management > Attribute calculation**, you will now find the settings for calculating attributes, which were previously found under "Modelling > Part properties".

Settings for standard parts and processing

In the Configuration Editor, the settings for standard parts and standard processings have been changed as follows:

The path "System settings > Standard parts" has been renamed to **System settings > Standard parts and processings**.

The parameter "Colour number of mounting hole" has been renamed to **Colour of construction site bore / mounting bore**.

The parameter **Minimum material depth for blind hole** is not evaluated by the new standard processing. The setting can now be found at **Compatibility > Standard parts and processings up to HiCAD 2023**.

The settings for 2-D functions have been moved to **System settings > Standard parts and processings > Standard parts and processings, 2-D**.

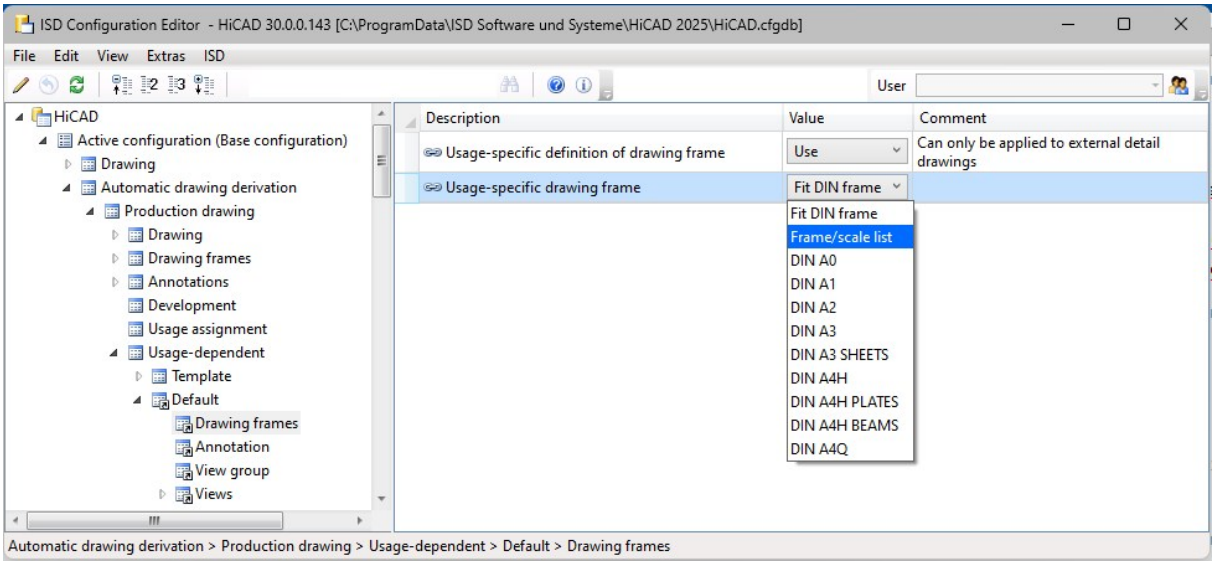
Novice configuration - Dialogue change

The dialogue of the **Novice configuration** function in HiCAD and the corresponding path in Configuration Editor have been adjusted. The path **System settings > Start configuration** has been renamed to **System settings > Novice configuration**. The Modelling section has been expanded to include the **Query plane when applying standard processings** parameter.

Activate frame / scale list

The option "Use frame/scale list" (**Automatic drawing derivation > Production drawing > Usage-dependent >... > View group**) for creating a dimension scale list during drawing derivation in HiCAD has been removed.

Instead, you can now select the **Frames/scale list** in the Configuration Editor using the option **Usage-dependent drawing frames** (**Automatic drawing derivation > Production drawing > Usage-dependent > ... > Drawing frames**):



Save referenced detail drawings as itemised source models

As of HiCAD 2025, detail drawings that are created using the **Reference part, Save, Detail drawing** function are no longer automatically saved as itemised source models; if you still want to save detail drawings as itemised source models, you can set this in Configuration Editor at **System settings > Itemisation > Make detail drawings itemised source models when creating them**.

Internally referenced parts in externally referenced assemblies

If internally referenced parts are inserted as sub-parts of externally referenced assemblies, it is possible that these are modified differently in external drawings. For this reason, internally referenced parts are now given a time stamp when their superordinate externally referenced assemblies are saved or when the drawing in which they are installed is saved. In this way, it is possible to trace which copy of identical parts was saved last.

If you want HiCAD to always adopt the latest status or always create different reference parts without a dialogue appearing, you can change the behaviour in Configuration Editor. There you will find the parameter **Handle outdated internally referenced parts** at **System settings > Referencing**. The default setting is **Query**.

Notes on HELiOS Updates

Microsoft SQL Server

The SQL Server Native Client (often abbreviated to SNAC) has been removed by Microsoft from SQL Server 2022 (16.x) and SQL Server Management Studio 19 (SSMS). It is recommended to use the latest version of the Microsoft ODBC Driver for SQL Server instead.

Further information can be found on the Microsoft website or in the installation instructions for Microsoft SQL Server 2022.

HELiOS Workspaces: Conversion of the system directories

When updating from an older version to HELiOS 2025 (Version 30.0.0) or higher, please note that the directory structure will change. Since an automated migration is not possible, all users have to check out all data and empty their workspaces before installing the update to avoid data loss.

In previous versions, the workspaces were located at %localappdata%. This meant that different workspaces could be located on one system. To prevent this, the update to HELiOS 2025 will move the workspaces to the **%programdata%** directory.

Checked-out files are then stored at **%programdata%\ISD Software und Systeme\HELiOS Workspace\(...)*** (*plus Location ID and User ID). The workspace databases are stored version-dependently at **%programdata%\ISD Software und Systeme\HELiOS <Version>\Location-ID**.

Notes on update installations

For an update installation of a HELiOS version older than HELiOS 2020 (version 2500), a central update of the supplied HELiOS database must be carried out..

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 2

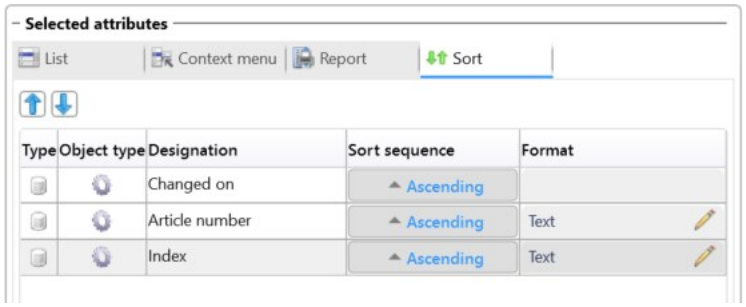
Result lists

Sort

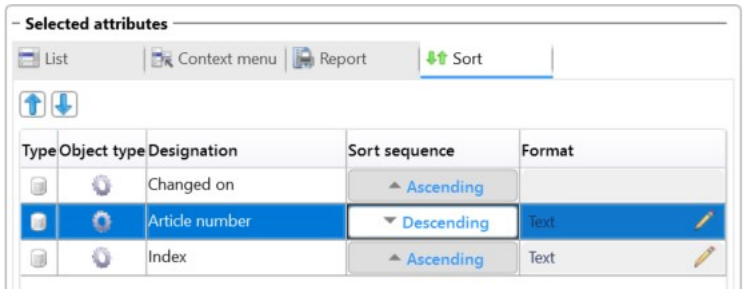
In the dialogue window for configuring attributes for result lists, you will find the new **Sort** tab.

In this submenu item, you can configure the default sorting for the respective result list. This is then applied both during the initial setup and each time a result list is refreshed (e.g. via F5).

Similar to the other tabs in the window, you can create a list of attributes that are then used to sort the result list in the specified order:

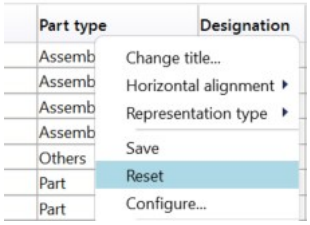


For each attribute, you can specify whether the attribute should be sorted in ascending or descending order (numerical, alphabetical). For example, always by the highest index (descending), but first by date from oldest to newest.



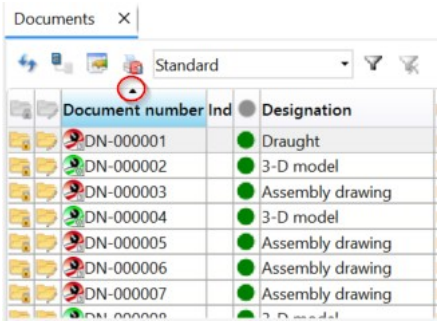
The attributes used for sorting do not necessarily have to be configured as visible columns in the result list in order to sort them.

Resetting result lists via the **Reset** function of the context menu also resets the sorting criteria.



Sorting arrow

At the top of result lists, a sort arrow icon above the corresponding column indicates whether the result list is sorted by only one attribute (by clicking on the header of a column or by defining only one sort attribute in the result list configuration) in ascending or descending order.



Subsequent sorting

Subsequent or repeated sorting of result lists has been optimised so that the result list does not have to be loaded completely for sorting. This was previously necessary and resulted in a progress bar.

Apart from the fact that the loading of result lists, for example for UI and VA attributes, has been accelerated in general, they can now be sorted very efficiently for database attributes. Compared to HELiOS Version 30.0, performance in this area has improved by a factor of 20.

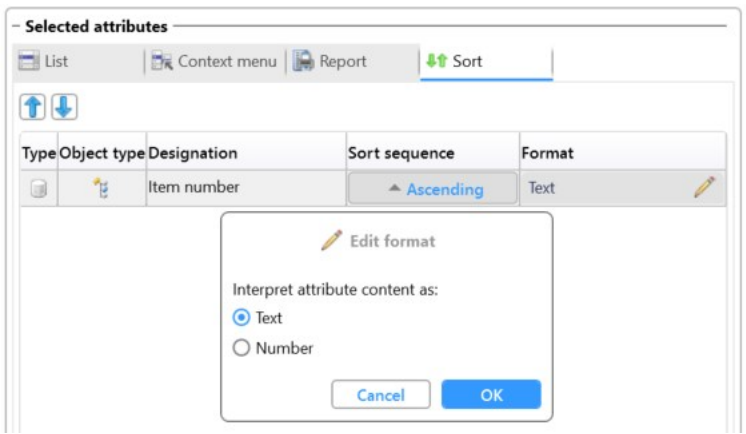
In earlier versions, when multiple sorts were performed in succession, the second sort was very fast (< 1 second) because the result list had already been loaded completely. This still applies to the sorting of UI attributes. When sorting by database attributes, however, each sort requires the time needed for the performance increase mentioned above.

The 'break-even' point in terms of time required is around 3 sorts, i.e. if the user were to perform many sorts in succession, the old method would be advantageous after this number of sorts.

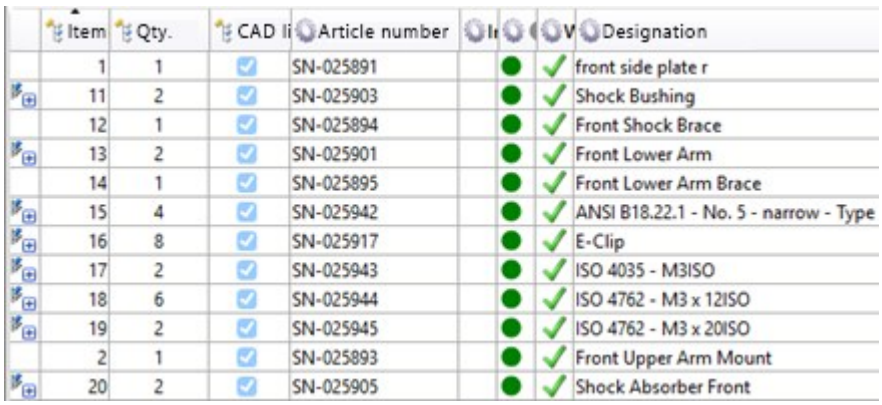
For a certain transition period, we are offering the option of deactivating the subsequent sorting of result lists using the database server. If you require more information, please contact ISD Consulting.

VARCHAR attributes as Text or Numbers

When sorting result lists, you can also configure attributes of type VARCHAR to be interpreted as **Numbers** instead of **Text**:

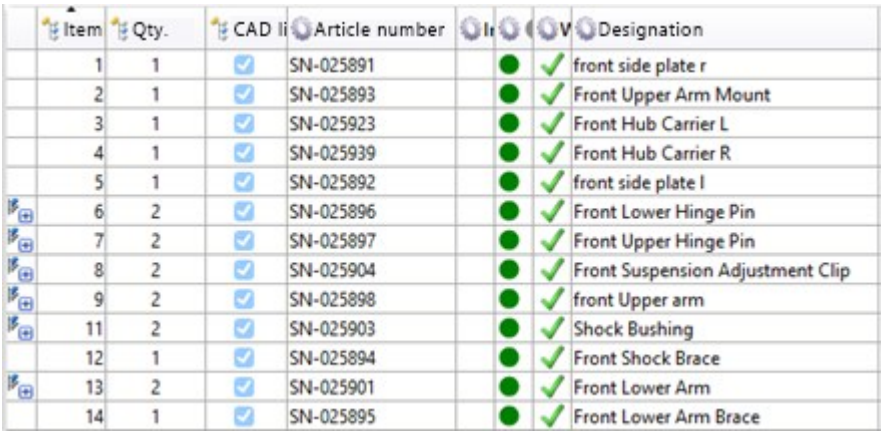


In some cases, lexicographical sorting can cause a sort to not be displayed exactly as desired. See the example of the Item number as **Text**:



Item	Qty.	CAD li	Article number	Designation
1	1	☑	SN-025891	front side plate r
11	2	☑	SN-025903	Shock Bushing
12	1	☑	SN-025894	Front Shock Brace
13	2	☑	SN-025901	Front Lower Arm
14	1	☑	SN-025895	Front Lower Arm Brace
15	4	☑	SN-025942	ANSI B18.22.1 - No. 5 - narrow - Type
16	8	☑	SN-025917	E-Clip
17	2	☑	SN-025943	ISO 4035 - M3ISO
18	6	☑	SN-025944	ISO 4762 - M3 x 12ISO
19	2	☑	SN-025945	ISO 4762 - M3 x 20ISO
2	1	☑	SN-025893	Front Upper Arm Mount
20	2	☑	SN-025905	Shock Absorber Front

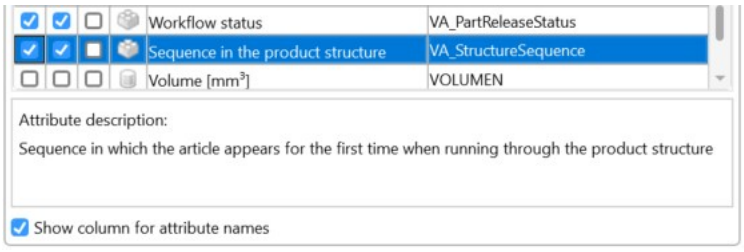
If the format is changed to **Number**, the sorting will look as follows:



Item	Qty.	CAD li	Article number	Designation
1	1	☑	SN-025891	front side plate r
2	1	☑	SN-025893	Front Upper Arm Mount
3	1	☑	SN-025923	Front Hub Carrier L
4	1	☑	SN-025939	Front Hub Carrier R
5	1	☑	SN-025892	front side plate l
6	2	☑	SN-025896	Front Lower Hinge Pin
7	2	☑	SN-025897	Front Upper Hinge Pin
8	2	☑	SN-025904	Front Suspension Adjustment Clip
9	2	☑	SN-025898	front Upper arm
11	2	☑	SN-025903	Shock Bushing
12	1	☑	SN-025894	Front Shock Brace
13	2	☑	SN-025901	Front Lower Arm
14	1	☑	SN-025895	Front Lower Arm Brace

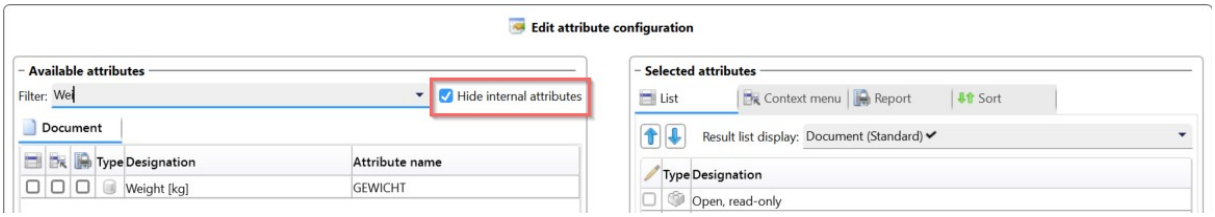
Utilized articles: Sequence in the product structure

A new virtual attribute **(VA_StructureSequence)** has been created for the **Utilized articles** result list in a product structure context, which specifies the order in which an article appears in the product structure.



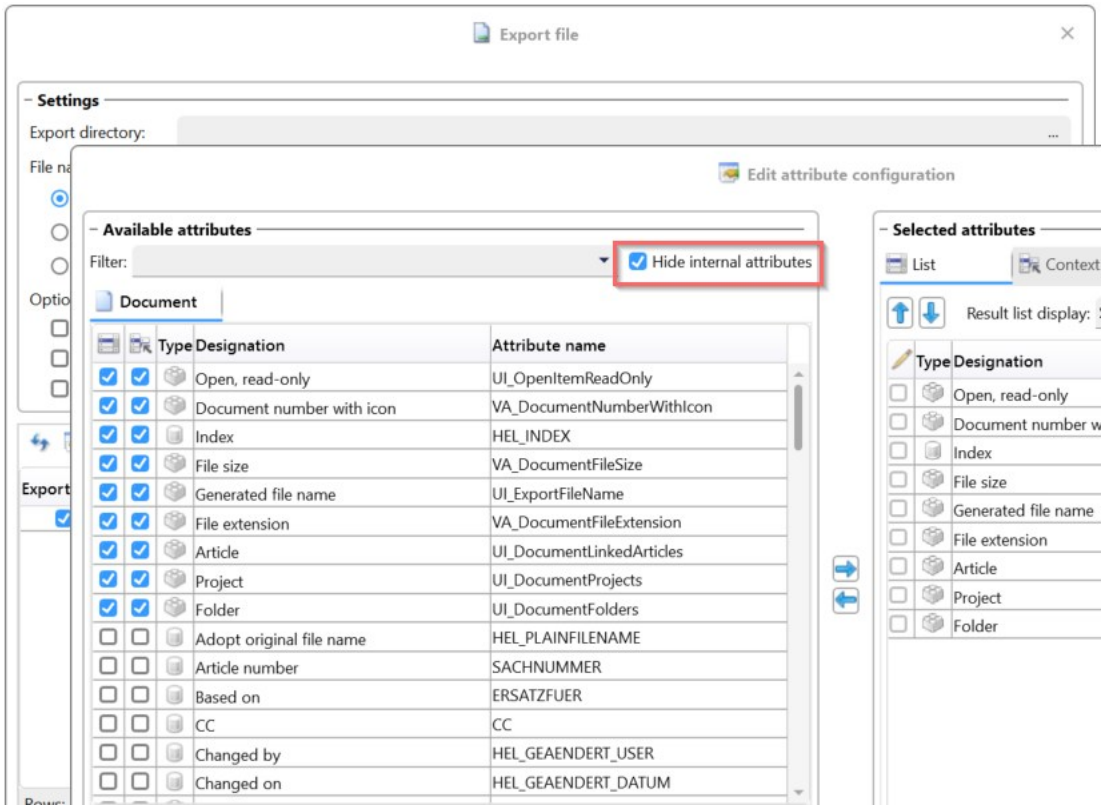
Hide internal attributes

In the dialogue window for configuring attributes for result lists, you will find the new checkbox **Hide internal attributes**.



So-called internal attributes are HELiOS system attributes that are generally not relevant for end users when sorting result lists. They are therefore excluded from the display in the attribute configuration for result lists by default, but can be included by deactivating the checkbox.

You will also find this option in some submenu windows for attribute selection, such as for configuring a mask or defining attribute assignments in various contexts.



Please note:

The actual result list display (i.e. the sum of all attributes configured for the result list) is not changed by this attribute classification. Internal attributes can also be displayed independently of this in the HELiOS interface by subtracting them from the list of available attributes using the active checkbox.

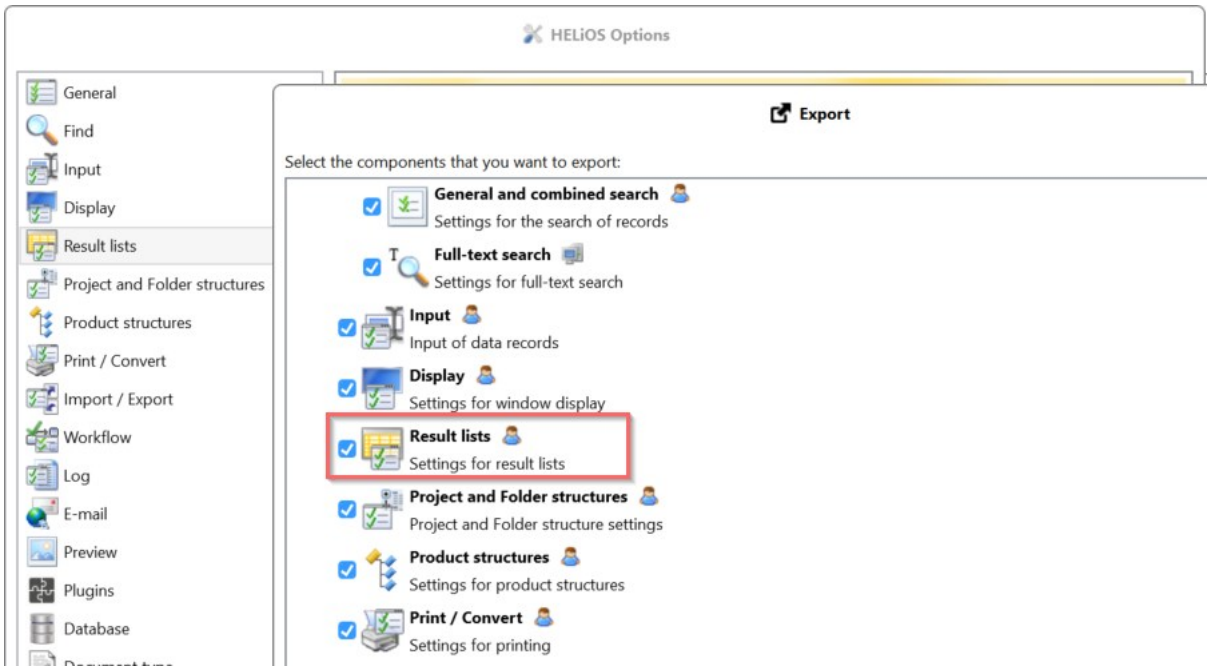
Transfer of old options during update installation and option to import/export

In earlier HELiOS versions, the **Result lists** tab of the **HELiOS Options** offered the possibility to specify the default sort order for result lists.

These settings are automatically transferred by the HELiOS update and are now available in the configuration of the result list.

The older option for sorting according to occurrence in the product structure for **Utilized articles** corresponds to the new virtual attribute for the **Sequence in the product structure** (VA_StructureSequence, see above), which is taken into account and implemented accordingly during the update.

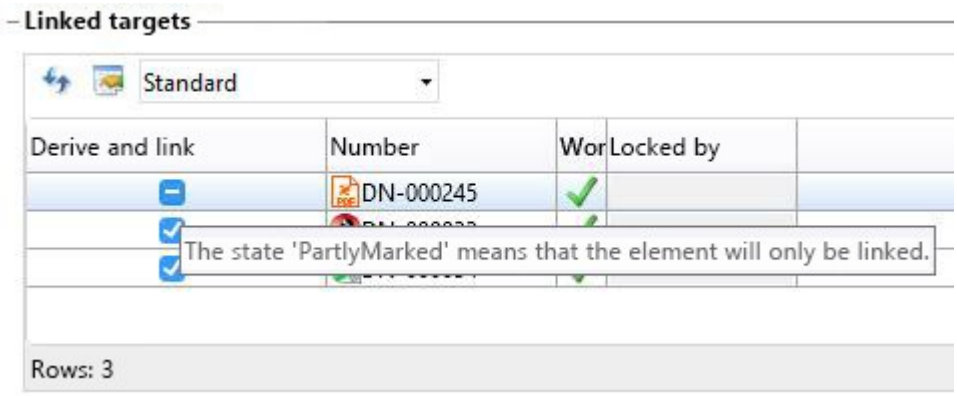
In addition, the sorting criteria are also managed in the **Import/Export settings** window that you open via the **Manage...** button on the **Result lists** tab of the **HELiOS Options**:



Partially activated checkboxes

During processes such as the derivation with links, activated checkboxes (checkmark symbol) mark the linked objects that are derived (and remain linked) with the derivation.

The dash symbol indicates objects that are linked to the derivation but are not derived themselves



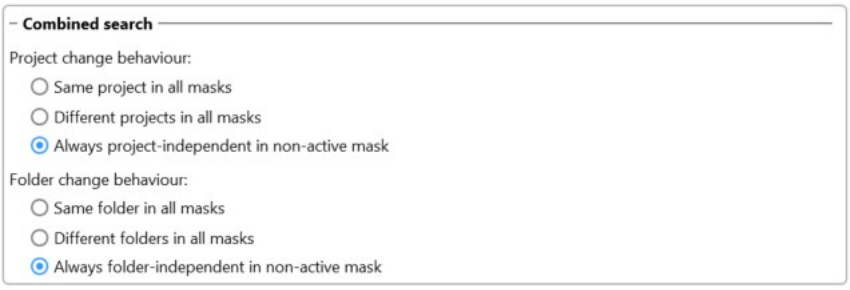
In earlier HELiOS versions, partially activated objects were indicated by a light grey border.

Default of project and folder change behaviour

In the **HELiOS Options**, at **Find > Combined search**, you can make settings for the project change behaviour and folder change behaviour in the context of the so-called combined search:

Should changing a project or folder selection in a search mask also affect the other masks of the combined search?

The default setting for this is set to Project-independent or Folder-independent as of Service Pack 2. This means that a project or folder selection in the active mask does not automatically affect the corresponding setting in the inactive search mask. However, if this is desired, you can of course continue to set the behaviour in the options to basically adopt the project/folder change of the active mask in the inactive one as well.



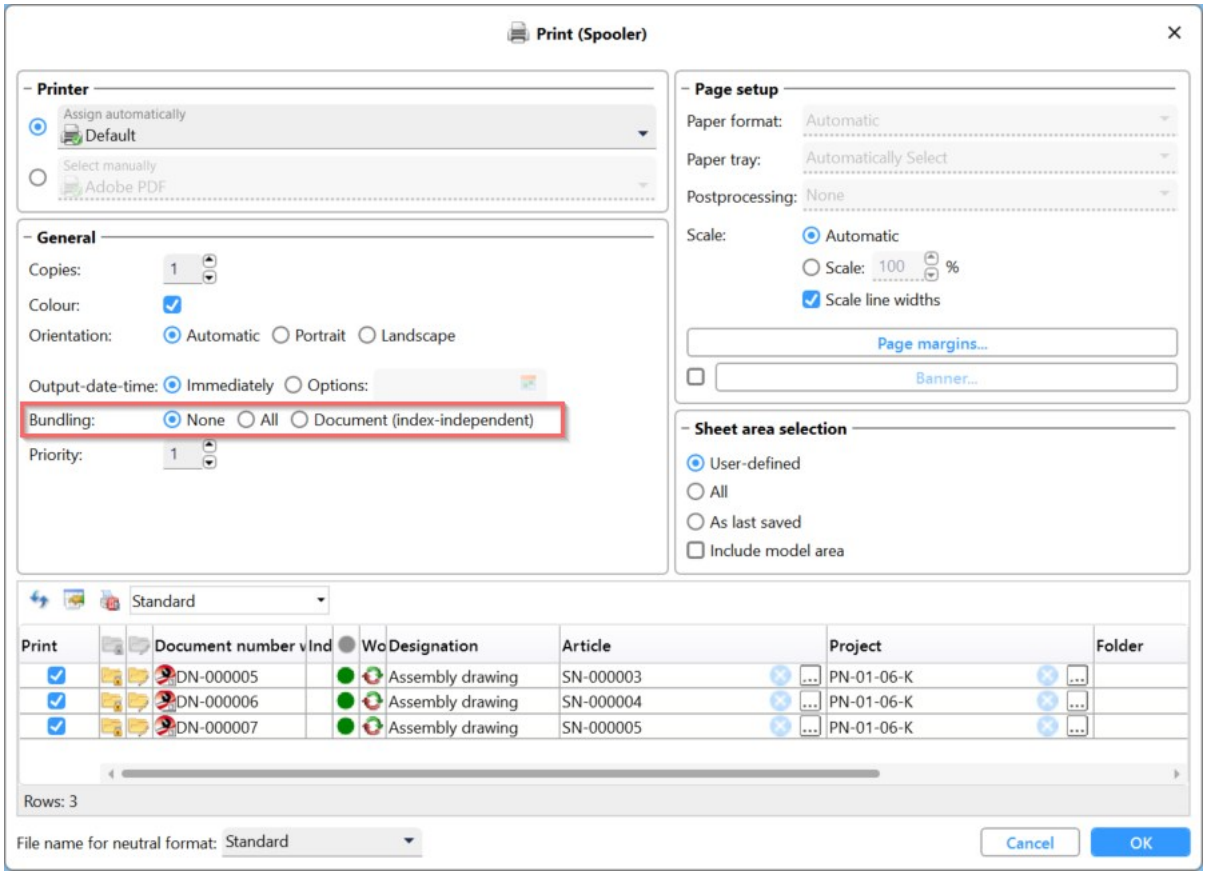
Print (Spooler)

Bundling

When printing documents via the **Print (Spooler)** function, the bundling option has been expanded:

To process any series of print jobs sent from your client in a single group without them being "mixed" with print jobs from other clients, you can activate the corresponding checkbox to set one of the following procedures for bundling the print job:

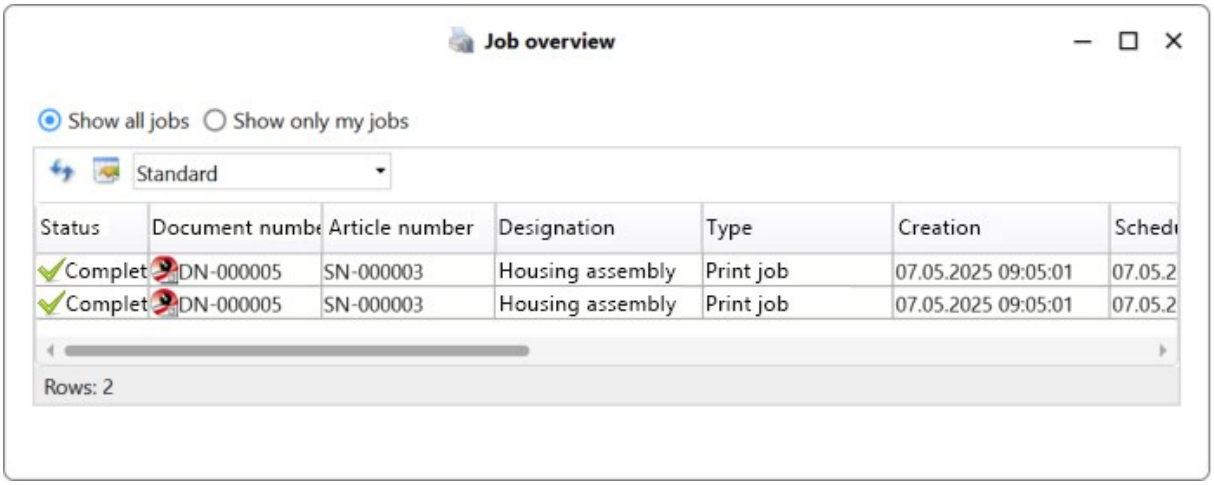
- **All**: If this checkbox is activated, series of print jobs sent from your client are sent and processed as a group. Different index levels of a document that have been selected for output are not necessarily processed 'separately' (i.e. other documents can be output in the processing sequence of the bundle"'between" the different index levels of a particular document).
- **Document (index-independent)**:If this option is activated, HELiOS documents with multiple sheet areas are converted into separate documents, with one document containing each HELiOS sheet area on a separate page. Different index levels of a document are processed in batches, i.e. output one after the other.



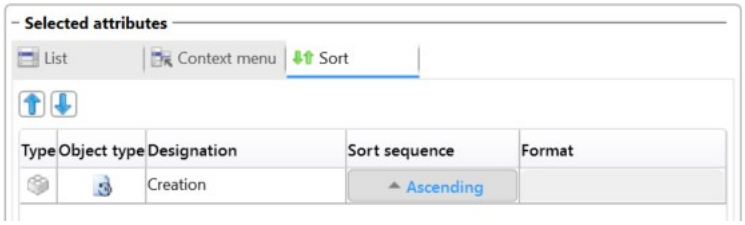
Job overview

With the update to Service Pack 2 for HELiOS 2025, you will find the new menu item **Job overview**  in the **Extras** ribbon of the HELIOS Desktop.

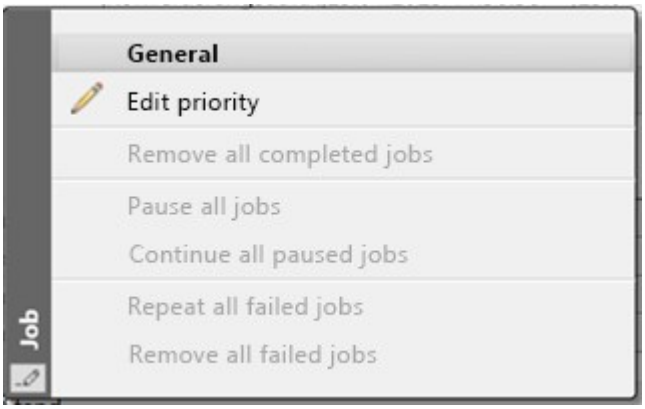
This displays an overview of all print jobs that have been started, are still in development or have been completed, similar to the **HELiOS Spooler Admin Tool**.



The result list configuration options of the HELIOS Desktop, such as the new definition of default sorting criteria, are also available.



In addition, you can influence the processing order in a queue of several print jobs that have not yet been completed using the context menu function **Edit priority**.



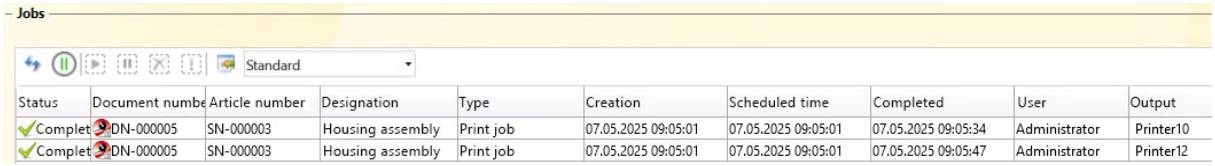
Output different formats of a print job on different printers

If a file, such as a PDF or a HiCAD drawing (SZA), contains sheets with different formats (e.g. A4 and A3), the HELiOS Spooler used to base the printer assignment on the first sheet of the file. This could result in only excerpts of the following sheets with larger formats being printed.

Starting with Service Pack 2 of HELiOS 2025, the HELiOS Spooler also takes into account different paper formats within the print job on a page-by-page basis.

This means that if you print a drawing file containing different paper formats via the spooler, these will be output to different printers according to the printer assignments in the HELiOS Spooler Admin Tool, provided that the appropriate assignments for the respective formats exist.

In the job overview under **Output**, you can see how the print jobs for such documents are distributed to different printers, for example:

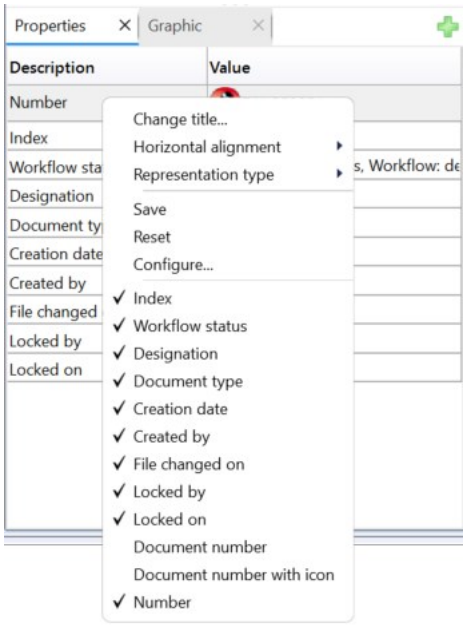


For file formats such as PDF, you can see in the log file of a print job that different pages of a document have been sent to different printers.

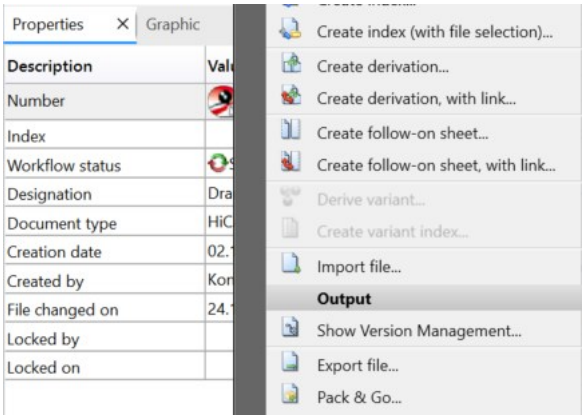
Context menu in the Value column of the Properties window

The **Properties** window displays the context-related data that you entered when creating the object (Project, Folder, Product, Class). Here, too, you can configure the selection of the displayed attributes.

Previously, only a context menu was available in the entire area for configuring the current **Properties** window itself.

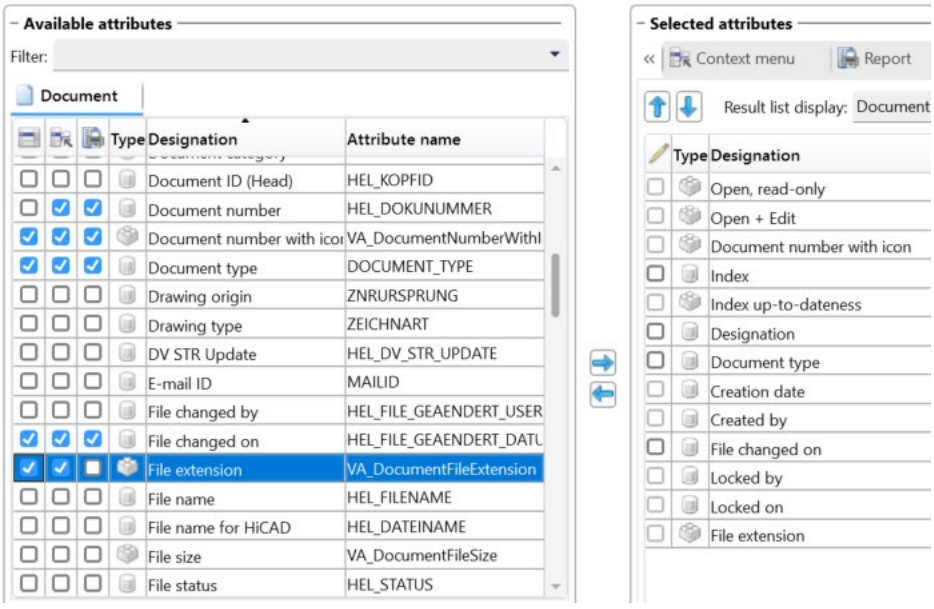


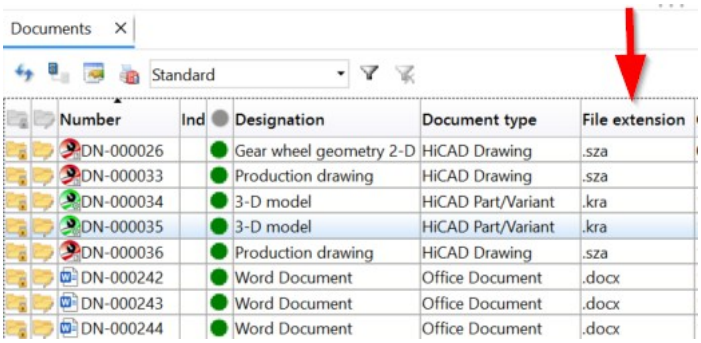
As of Service Pack 2, a right-click on the **Value** column opens the object type-related context menu for the HELiOS object currently displayed in the **Properties** window.



Virtual document attribute: File extension

The new virtual database attribute **VA_DocumentFileExtension** (Designation: **File extension**) displays the extension for the associated file of a document in the document context (e.g.: '.sza', '.pdf', '.txt' etc.).



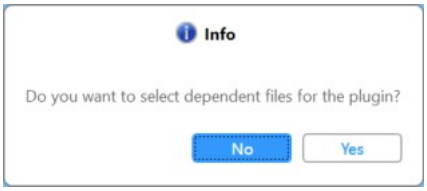


Plugin management in the HELiOS Options

With the update to HELiOS 2025 Service Pack 2, customer-specific plugins are managed in the new Options submenu at **HELiOS-Optionen > Plugins**.

By clicking on  a new plugin can be added to the **List of available plugins**. The Windows Explorer file selection window opens, where you must select the corresponding DLL of the desired plugin.

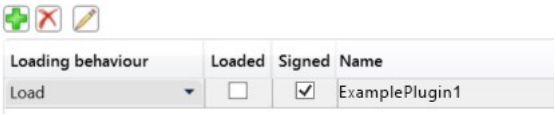
After selecting the DLL, a query appears asking whether the plugin includes additional files:



When you click **Yes**, another file selection dialogue box opens, in which you can select one or multiple files belonging to the plugin.

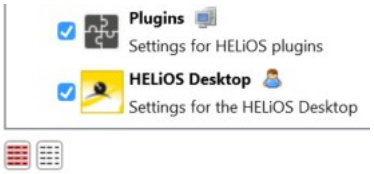
These may be additional DLLs required by the plugin or settings files that are essential for the plugin.

After selection, the new plugin is displayed in the list:



You can also influence the **Loading behaviour** of plugins there.

The settings for the plugins are stored on a computer-specific basis, i.e. every HELiOS user logged on to a computer will see the same plugins with the same options regarding loading behaviour when starting HELiOS. The settings can be transferred to other computers using the export and import mechanism for HELiOS Options:



 Note that customer-specific plugins currently entered in HELiOS.exe.config will not be automatically transferred to the new mechanism when updating to 30.2.0 or higher. They must therefore be re-entered once. In future versions, a HELiOS update will then automatically update the settings for the plugins stored in the HELiOS Options. Only the plugins themselves may need to be updated across release cycles.

Please also note that, starting with version 30.2.0, plugin management via the HELiOS options replaces the manual adjustments to the HELiOS.exe.config file that were necessary in earlier versions, and that a HELiOS installation therefore no longer contains a Plugins folder at (...)\exe\.

When updating a HELiOS Desktop in standalone mode, the update automatically creates a backup of the old plugin directory. Updating a HiCAD/HELiOS installation does not do this, as the installation cannot distinguish between HiCAD and HELiOS plugins.

If a plugin has been copied to the \exe\ folder of HELiOS by mistake, you will receive an error message asking you to remove the plugin DLL from the invalid location and restart HELiOS.

Activate functionality for HiCAD Plant Engineering

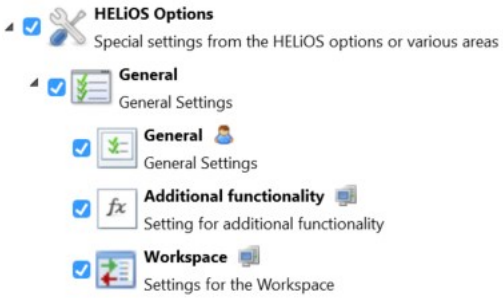
With the update to Service Pack 2 of HELiOS 2025, you will find the new submenu item **Additional functionality** at **HELiOS Options > General.**

There, you can enable the functionality for HiCAD Plant Engineering (handling pipe classes and variants) for the HELiOS Desktop by activating the checkbox.

The new option replaces the activation/deactivation of the functionality via HELiOS.exe.config.

The setting is saved on a computer-specific basis and can be transferred to other computers using the export and import mechanism for HELiOS Options.

There, it is offered in the selection menu, under **General**, with the **Additional functionality** checkbox:



When updating from a lower HELiOS version to (version 30.2 or higher, or when reinstalling HELiOS Service Pack 2), this checkbox must be activated once if the corresponding plugin (Helios.Custom.HicadPlugin) was previously active.

HELiOS Mail Proxy

Until now, emails were sent via HELiOS using the SMTP protocol with plain text authentication.

In order to take into account other security requirements for system architectures and future developments in the area of Exchange Online, a new server application will be available from HELiOS 2025 SP 2 with the **HELiOS Mail Proxy**, which regulates the sending of mail from HELiOS clients to a configured mail server.

Nothing will change for running systems at this point, i.e. switching to the **HELiOS Mail Proxy** is not mandatory at this time.

For more information, please refer to the HELiOS installation manual.

Asynchronous execution of viewers

HELiOS now also supports viewers integrated into HELiOS that support asynchronous execution.

For users, this means that HELiOS remains operable even when a preview is being built.

Improvements and performance increases

Optimised processes and improvements have been achieved for HELiOS 2025 with Service Pack 2 in several areas.

This includes the error output process during update installations.

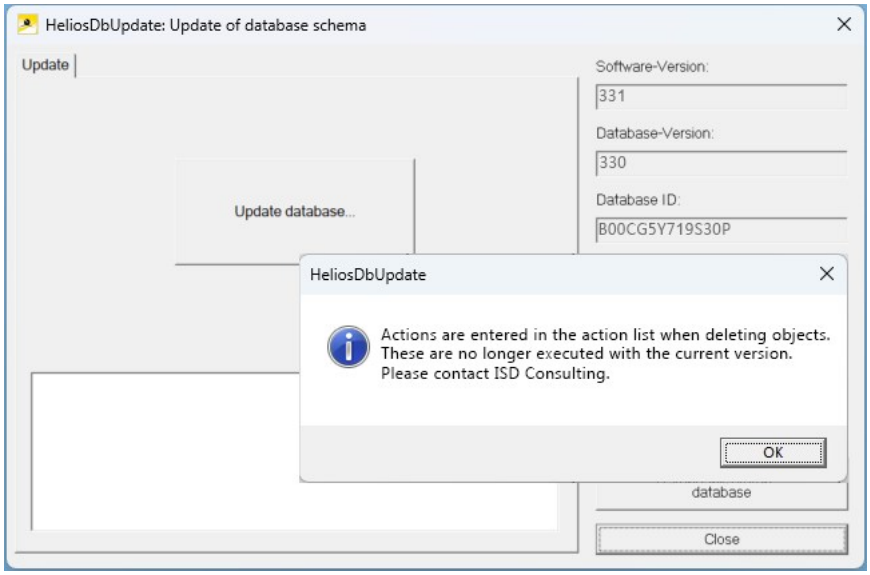
A further significant performance increase has been achieved when deleting multiple HELiOS objects at once.

In the event of a serious error (e.g. connection to the database is lost), the HELiOS Client writes the files to be deleted to the log file ArticleDeletion.log under %APPDATA%\ISD Software und Systeme\<ProductVersion>\Logging\Core\ .

These entries can be used to repeat the deletion later (manually) and look as follows:

```
17.04.2025 14:54:36
Konstrukteur2;Head: B004URKMOE3KZ300003JKL, Rev: 0;
```

Please also note at this point that deletion processes via workflow action lists are no longer possible due to the restructuring. If such an action is still configured in HELiOS, the following information message appears during the database version update (the update is still carried out):

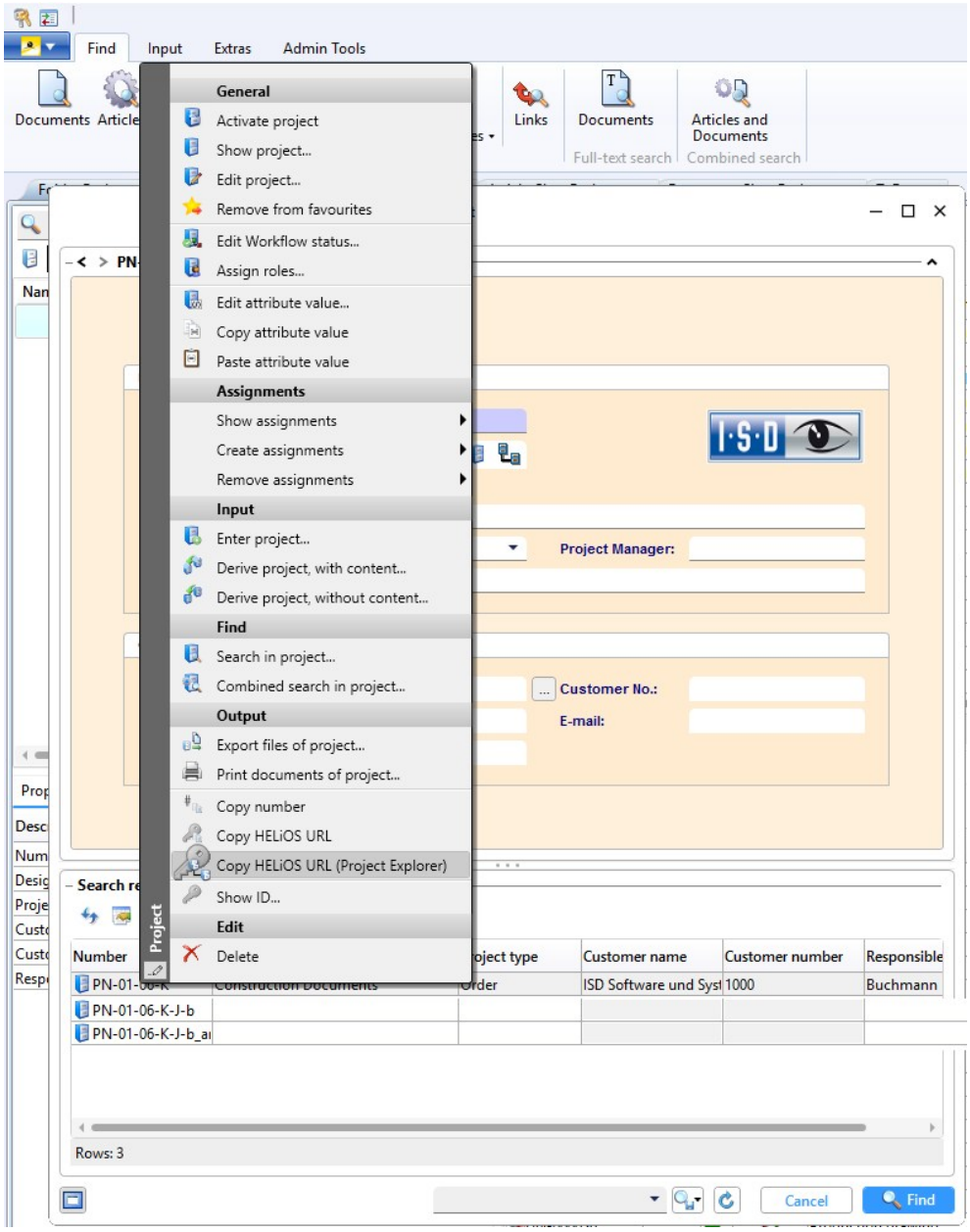


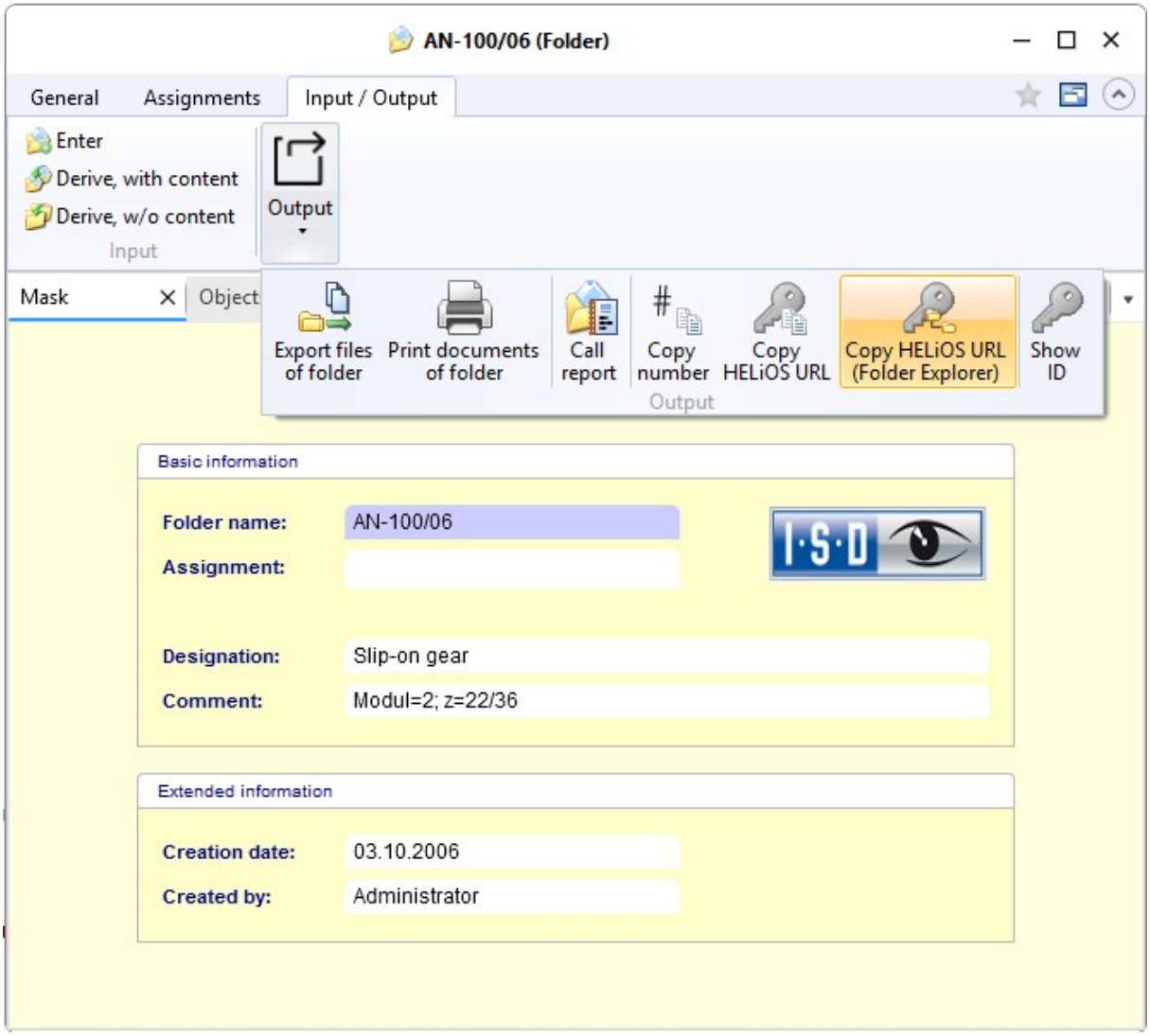
Service Pack 1

Project Explorer / Folder Explorer: Copying the HELiOS URL, Revised UI

The context menus for projects and folders, as well as the Ribbon menus for the project and folder detail masks, now include the new functions **Copy HELiOS URL (Project Explorer)**  and **Copy HELiOS URL (Folder Explorer)**  .

These functions can be used to copy the HELiOS URL of a project or a folder, including a call-up in the corresponding Explorer area to the clipboard.





In addition, the user interface behaviour in HELiOS Desktop and coupled applications has been further adapted and improved:

The Explorer areas of HELiOS can also be opened in several instances at the same time, in order to perform drag & drop actions between them, etc.

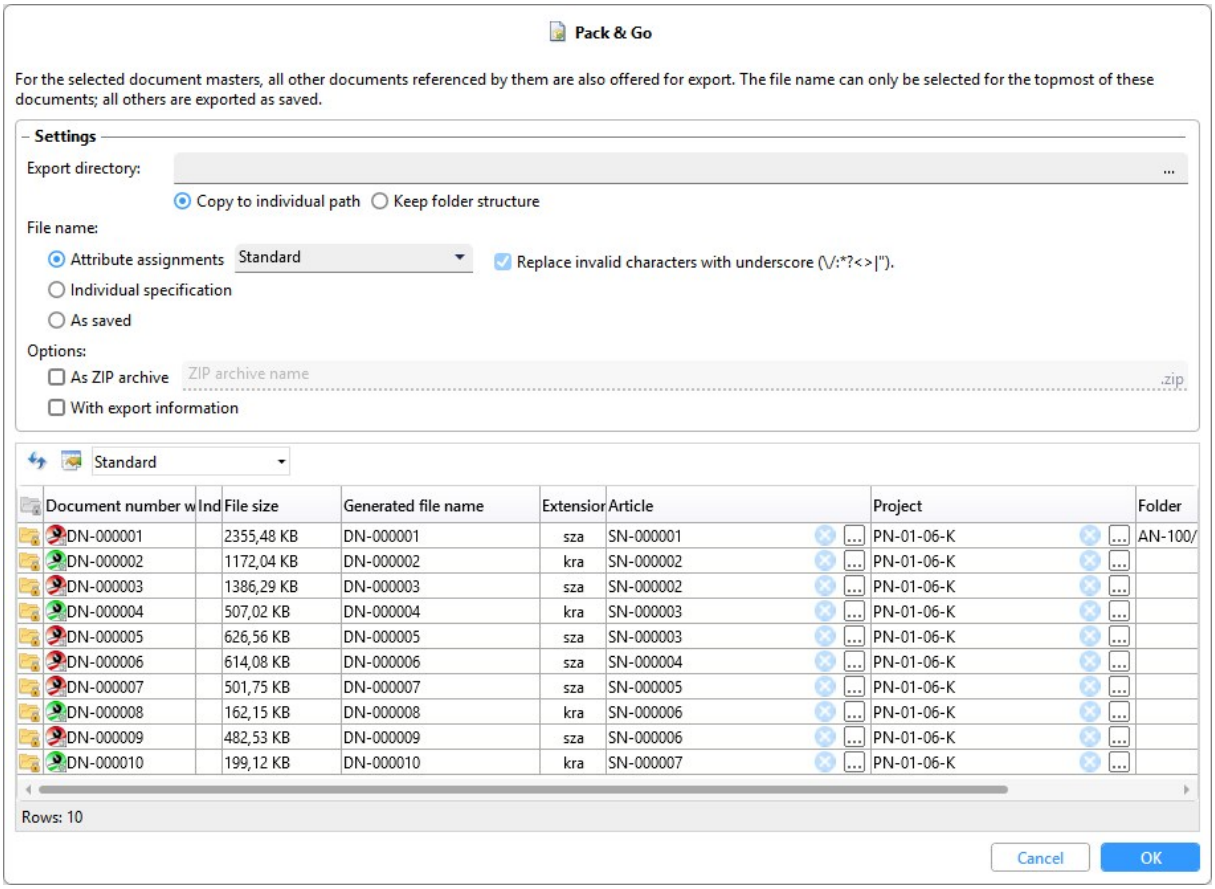
Export file and Pack & Go

With Service Pack 1, the dialogue windows for exporting files or sending HELiOS documents as e-mail attachments have been revised.

The additional settings dialogue has been removed.

Another new feature is the **Pack & Go**  document output function..

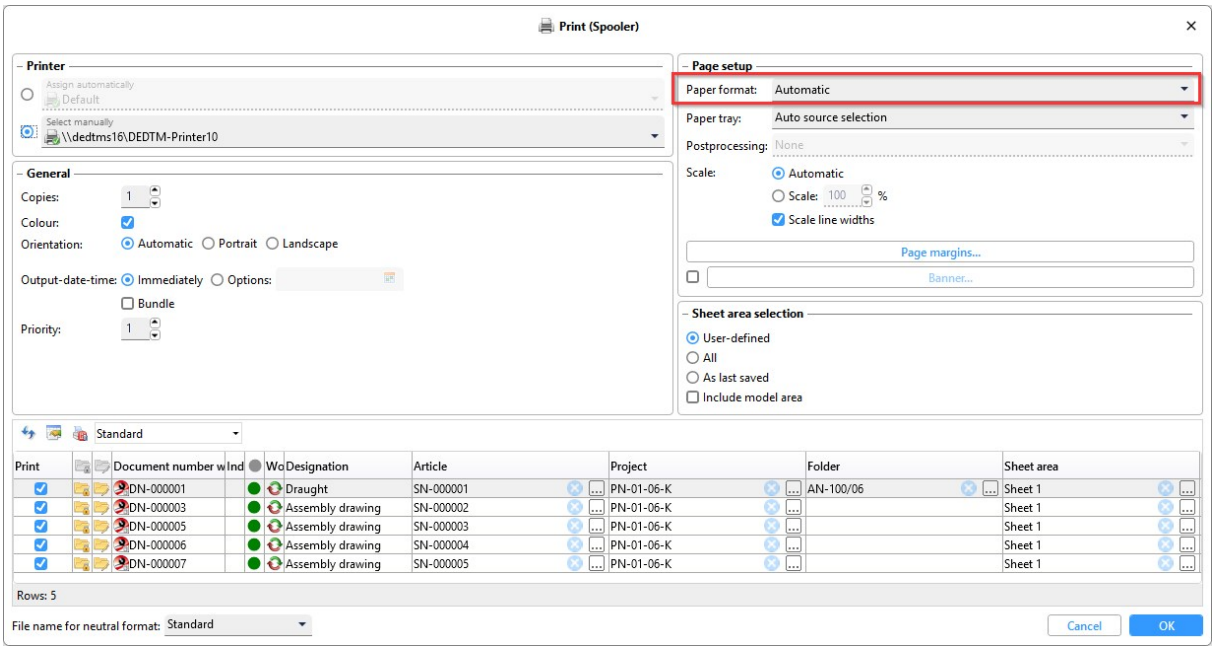
In contrast to the Export file function from HELiOS, the output via **Pack & Go** is designed to export related subsets of data, such as CAD structures, while taking referenced documents (from the document/model structure) into account.



Print (Spooler)

Automatic selection of paper format

When printing HELiOS documents via **Output > Print (Spooler)** ,  the default setting for the **Paper format** is now set to **Automatic**..

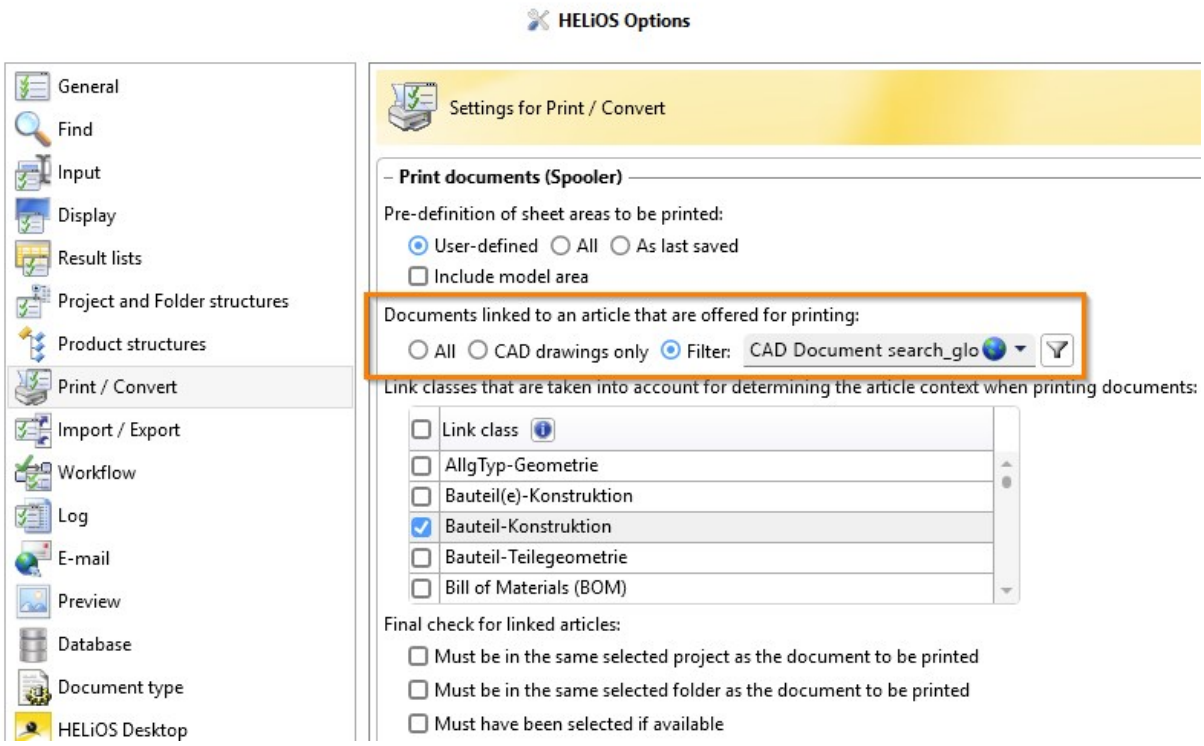


HELiOS Options: Documents linked to an article that are offered for printing

In the **HELiOS Options**, on the **Print/Convert** tab, you can use the new sub-menu item **Documents linked to an article that are offered for printing** to set which documents are included in the context of articles when the **Print linked documents** function is used.

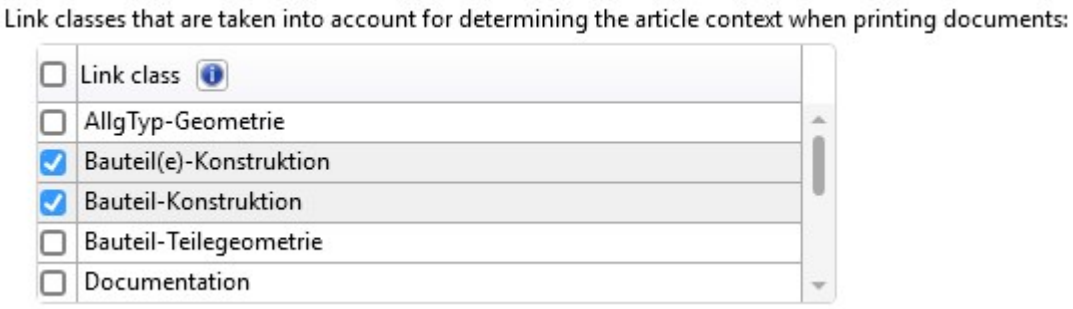
You can choose between three options:

- **All**: With this option (default setting), all documents linked to the article are included.
- **CAD drawings only**: If this checkbox is activated, only system links with which CAD drawings are linked to an article in HELiOS by default are taken into account.
- **Filter**: By activating this option, you can select a global, user-defined document search template that is used to determine the relevant documents.



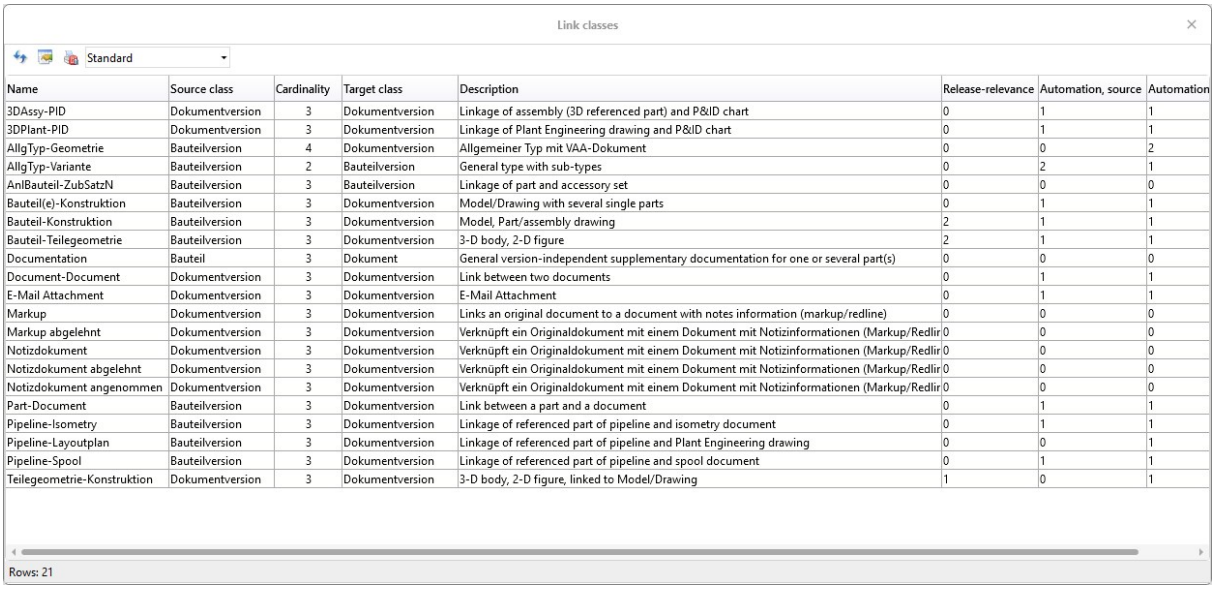
HELiOS-Options: Link classes that are taken into account for determining the article context when printing documents

Another new menu entry at HELiOS Options > Print/Convert allows you to set the linkage classes that are used to determine the item context when printing documents.



By default, the link classes relevant for determining the part context are activated here. In principle, you can add or remove further links by clicking the checkboxes. You can also deselect all entries if you prefer a document output without taking the part context into account.

Clicking on in the sub-menu's header displays a result list of all link classes.



Name	Source class	Cardinality	Target class	Description	Release-relevance	Automation, source	Automation
3DAssy-PID	Dokumentversion	3	Dokumentversion	Linkage of assembly (3D referenced part) and P&ID chart	0	1	1
3DPlant-PID	Dokumentversion	3	Dokumentversion	Linkage of Plant Engineering drawing and P&ID chart	0	1	1
AllgTyp-Geometrie	Bauteilversion	4	Dokumentversion	Allgemeiner Typ mit VAA-Dokument	0	0	2
AllgTyp-Variante	Bauteilversion	2	Bauteilversion	General type with sub-types	0	2	1
AnlBauteil-ZubSatzN	Bauteilversion	3	Bauteilversion	Linkage of part and accessory set	0	0	0
Bauteil(e)-Konstruktion	Bauteilversion	3	Dokumentversion	Model/Drawing with several single parts	0	1	1
Bauteil-Konstruktion	Bauteilversion	3	Dokumentversion	Model, Part/assembly drawing	2	1	1
Bauteil-Teilegeometrie	Bauteilversion	3	Dokumentversion	3-D body, 2-D figure	2	1	1
Documentation	Bauteil	3	Dokument	General version-independent supplementary documentation for one or several part(s)	0	0	0
Document-Document	Dokumentversion	3	Dokumentversion	Link between two documents	0	1	1
E-Mail Attachment	Dokumentversion	3	Dokumentversion	E-Mail Attachment	0	1	1
Markup	Dokumentversion	3	Dokumentversion	Links an original document to a document with notes information (markup/redline)	0	0	0
Markup abgelehnt	Dokumentversion	3	Dokumentversion	Verknüpft ein Originaldokument mit einem Dokument mit Notizinformationen (Markup/Redlir	0	0	0
Notizdokument	Dokumentversion	3	Dokumentversion	Verknüpft ein Originaldokument mit einem Dokument mit Notizinformationen (Markup/Redlir	0	0	0
Notizdokument abgelehnt	Dokumentversion	3	Dokumentversion	Verknüpft ein Originaldokument mit einem Dokument mit Notizinformationen (Markup/Redlir	0	0	0
Notizdokument angenommen	Dokumentversion	3	Dokumentversion	Verknüpft ein Originaldokument mit einem Dokument mit Notizinformationen (Markup/Redlir	0	0	0
Part-Document	Bauteilversion	3	Dokumentversion	Link between a part and a document	0	1	1
Pipeline-Isometry	Bauteilversion	3	Dokumentversion	Linkage of referenced part of pipeline and isometry document	0	1	1
Pipeline-Layoutplan	Bauteilversion	3	Dokumentversion	Linkage of referenced part of pipeline and Plant Engineering drawing	0	0	1
Pipeline-Spool	Bauteilversion	3	Dokumentversion	Linkage of referenced part of pipeline and spool document	0	1	1
Teilegeometrie-Konstruktion	Dokumentversion	3	Dokumentversion	3-D body, 2-D figure, linked to Model/Drawing	1	0	1

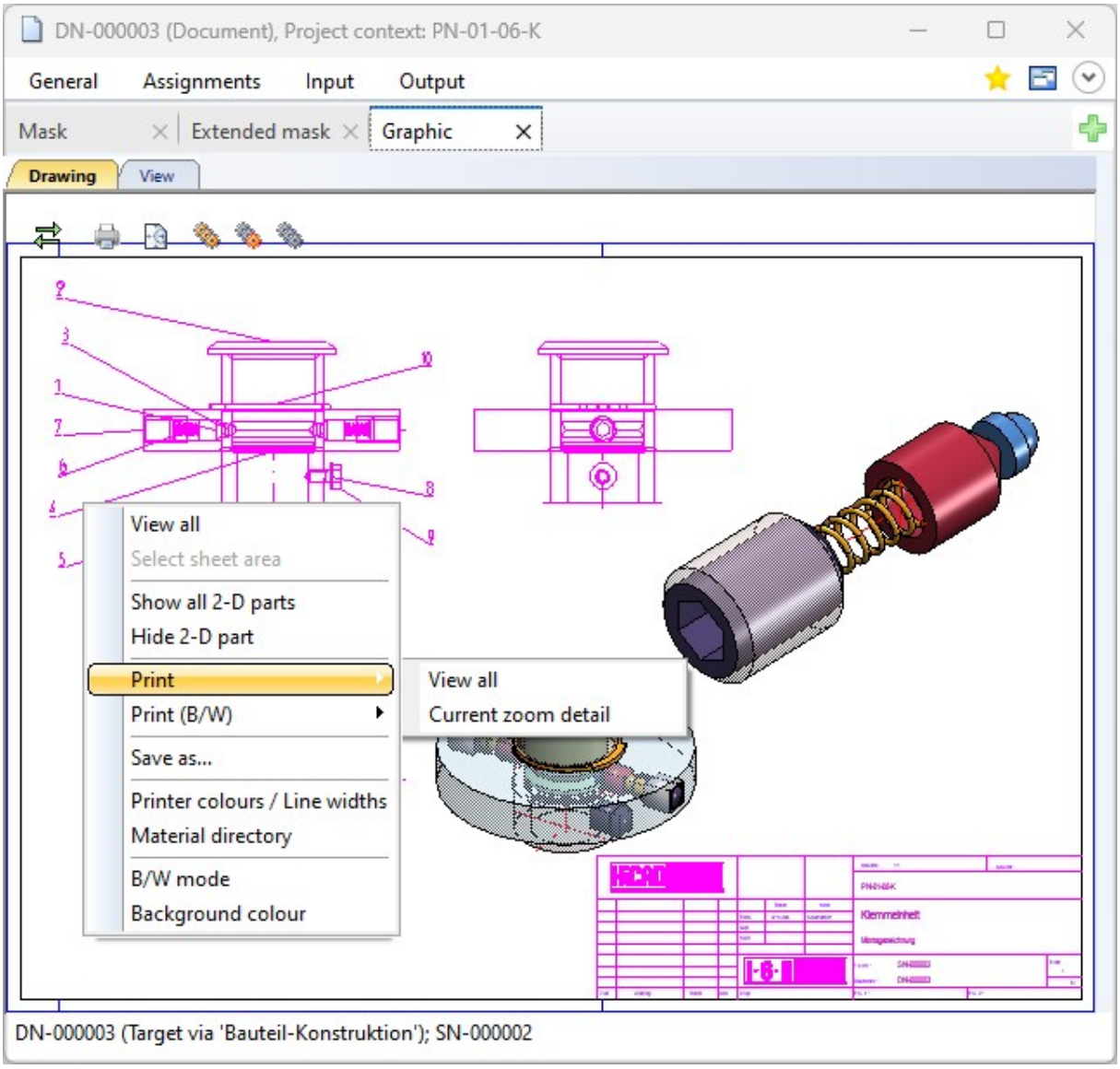
Rows: 21

Improved user guidance

In addition to the above-mentioned innovations, the update to Service Pack 1 also improves user guidance in the print dialog, for example with regard to information and error messages.

HiCAD Viewer: Project and folder context

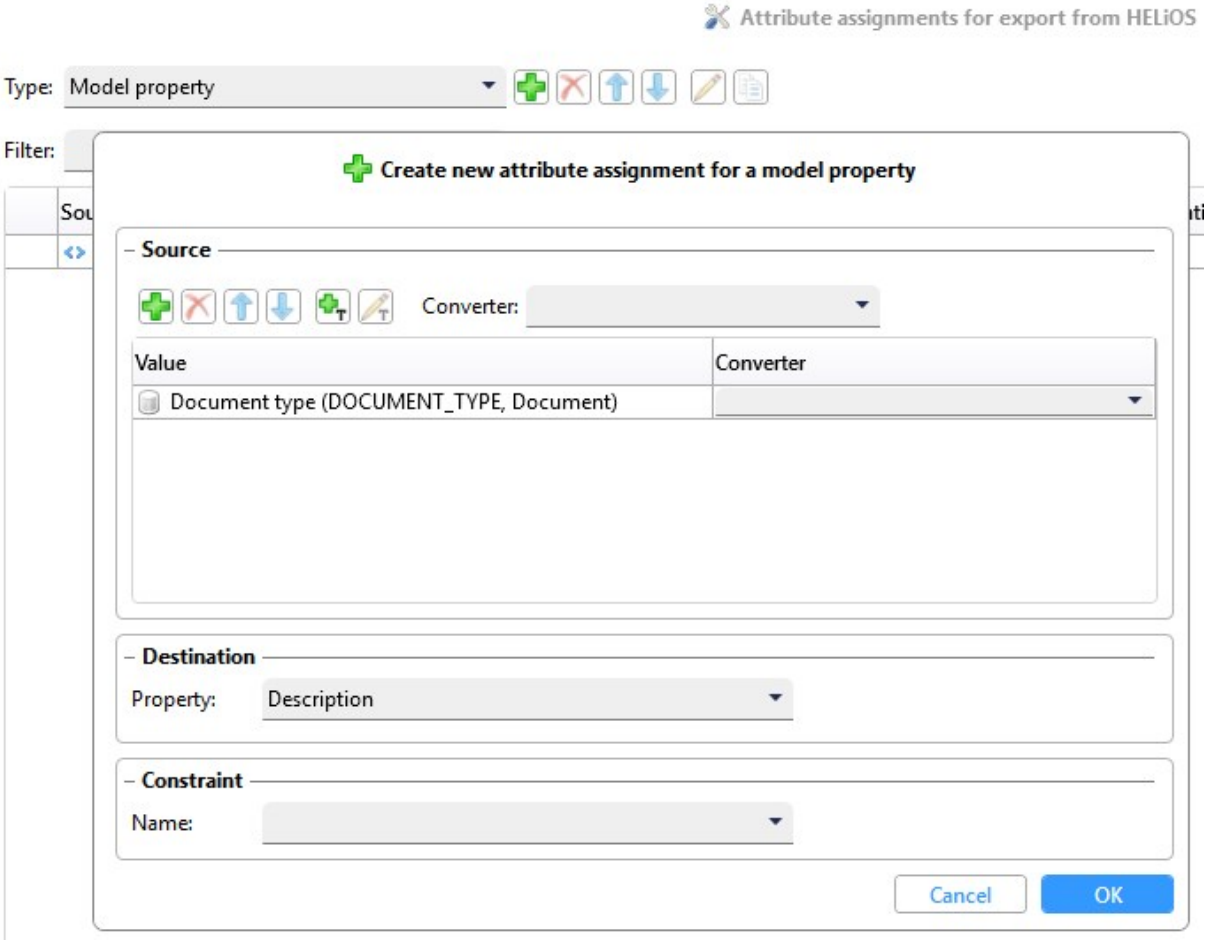
The display of the HiCAD Viewer now takes into account any clear project/folder assignments of HELiOS documents, if available.



HELiOS Options: Attribute assignments for Cadmium Cloud

In the HELiOS Desktop, the new sub-menu area **Attribute asssignments for Cadmium Cloud** can be found at **HELiOS Options > Preview**.

Here, you can define the assignments of HELiOS attribute assignments for the display name and the description of documents in the Cadmium Cloud preview.



In the Workspaces display in the Cadmium Cloud, this corresponds to the columns "Display name" and "Description":

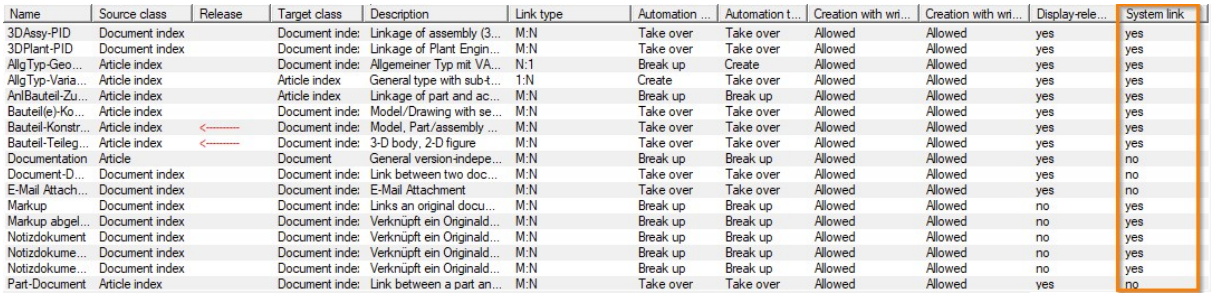
Performance optimisation of the PDF preview

The performance of the PDF preview in HELiOS has been significantly improved.

System link

The new **System link** attribute has been created in the HELiOS database for **link classes**.

A new column has been added to the corresponding overviews:

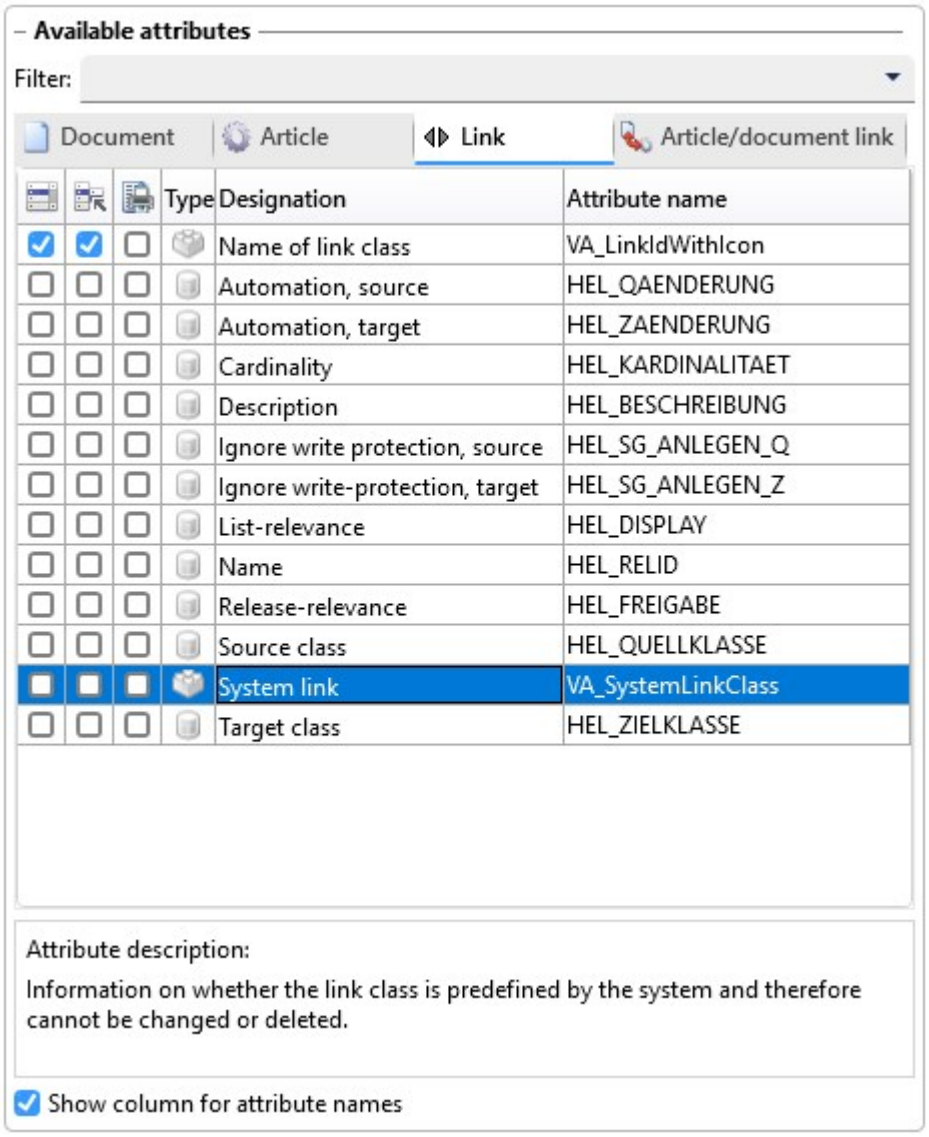


Name	Source class	Release	Target class	Description	Link type	Automation ...	Automation t...	Creation with wri...	Creation with wri...	Display-rele...	System link
3DAssy-PID	Document index		Document inde:	Linkage of assembly (3...	M:N	Take over	Take over	Allowed	Allowed	yes	yes
3DPlant-PID	Document index		Document inde:	Linkage of Plant Engin...	M:N	Take over	Take over	Allowed	Allowed	yes	yes
AllgTyp-Geo...	Article index		Document inde:	Allgemeiner Typ mit VA...	N:1	Break up	Create	Allowed	Allowed	yes	yes
AllgTyp-Varia...	Article index		Article index	General type with sub-t...	1:N	Create	Take over	Allowed	Allowed	yes	yes
AnlBauteil-Zu...	Article index		Article index	Linkage of part and ac...	M:N	Break up	Break up	Allowed	Allowed	yes	yes
Bauteil(e)-Ko...	Article index		Document inde:	Model/Drawing with se...	M:N	Take over	Take over	Allowed	Allowed	yes	yes
Bauteil-Konstr...	Article index	<----------	Document inde:	Model, Part/assembly ...	M:N	Take over	Take over	Allowed	Allowed	yes	yes
Bauteil-Teileg...	Article index	<----------	Document inde:	3-D body, 2-D figure	M:N	Take over	Take over	Allowed	Allowed	yes	yes
Documentation	Article		Document	General version-indepe...	M:N	Break up	Break up	Allowed	Allowed	yes	no
Document-D...	Document index		Document inde:	Link between two doc...	M:N	Take over	Take over	Allowed	Allowed	yes	no
E-Mail Attach...	Document index		Document inde:	E-Mail Attachment	M:N	Take over	Take over	Allowed	Allowed	yes	no
Markup	Document index		Document inde:	Links an original docu...	M:N	Break up	Break up	Allowed	Allowed	no	yes
Markup abgel...	Document index		Document inde:	Verknüpft ein Originald...	M:N	Break up	Break up	Allowed	Allowed	no	yes
Notizdokument	Document index		Document inde:	Verknüpft ein Originald...	M:N	Break up	Break up	Allowed	Allowed	no	yes
Notizdokume...	Document index		Document inde:	Verknüpft ein Originald...	M:N	Break up	Break up	Allowed	Allowed	no	yes
Notizdokume...	Document index		Document inde:	Verknüpft ein Originald...	M:N	Break up	Break up	Allowed	Allowed	no	yes
Part-Document	Article index		Document inde:	Link between a part an...	M:N	Take over	Take over	Allowed	Allowed	yes	no

One system link is predefined by HELiOS and, in contrast to links that have been newly created by the user, cannot be changed or deleted.

This is because system links are connected to HELiOS program logics, which would no longer function if the link were removed.

In result lists with a link context, you can also use the virtual attribute **System link (VA_SystemLinkClass)** to display this information:



HELiOS Options: Product structures

When you call up **Output to Report Manager** or **Output to Report Manager, new window**, a configuration file will be evaluated in which the transfer of attributes of the product structure is stored.

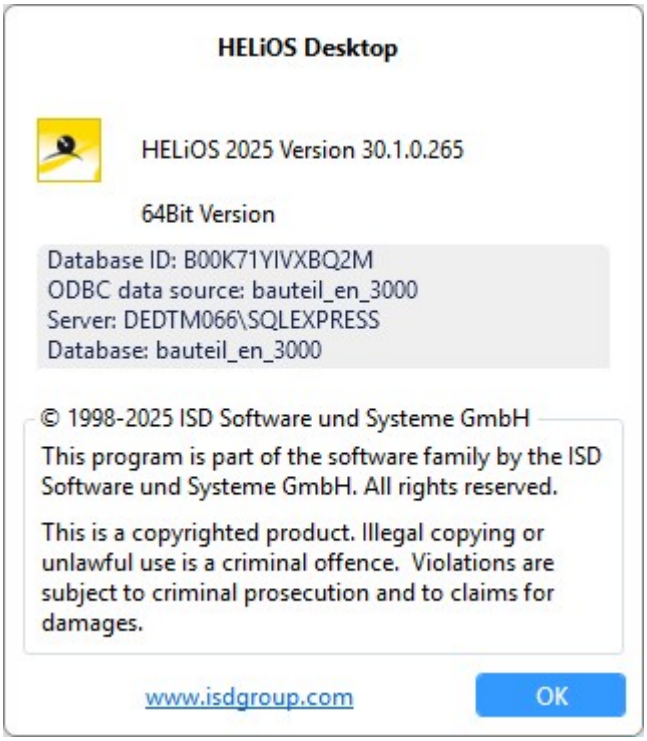
If you want to use a different configuration file than the default one, you can set this in the new tab of the HELiOS Options dialogue at **Product structures > Report Manager**.

Display of additional information about the current Database connection in the HELiOS info window

Via  > Info  you can open an info window in the HELiOS Desktop, which displays information about the HELiOS Desktop, such as the current HELiOS version number (incl. build number) and the number of any installed hotfixes.

With the current service pack, the information has been extended to include details of the database connection: the database ID of HELiOS, the name of the ODBC data source, the computer name of the database server and the SQL server instance, as well as the name of the HELiOS SQL database.

Example:



Locking Manager: Unlocking independent of the host computer

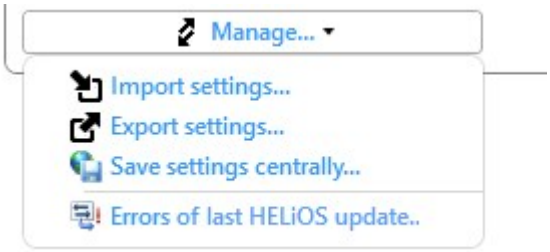
In order to avoid problems that may occur when working remotely with third-party software, the **Locking Manager** for unlocking HELiOS objects has been extended so that HELiOS users working from different workstations can unlock locked HELiOS objects such as CAD drawings from other host computers using the Locking Manager.

In this case, you will receive the query *You are trying to unlock objects that were originally locked on another computer. Are you sure you want to unlock them?* Click **Yes** to unlock the objects or **No** to cancel the unlock operation.

Errors of last HELiOS update

If errors occur during HELiOS update processes, a window with corresponding messages will be displayed when you start the HELiOS Desktop. You can optionally copy further detailed information into the clipboard.

You can view the window at any time via **HELiOS-Options >Manage...>** **Errors of last HELiOS update...**.



Automatic saving of dumps when HELiOS crashes

As of HELiOS 2025 Service Pack 1, installations of HELiOS applications perform a configuration that automatically saves so-called dump files when the program crashes.

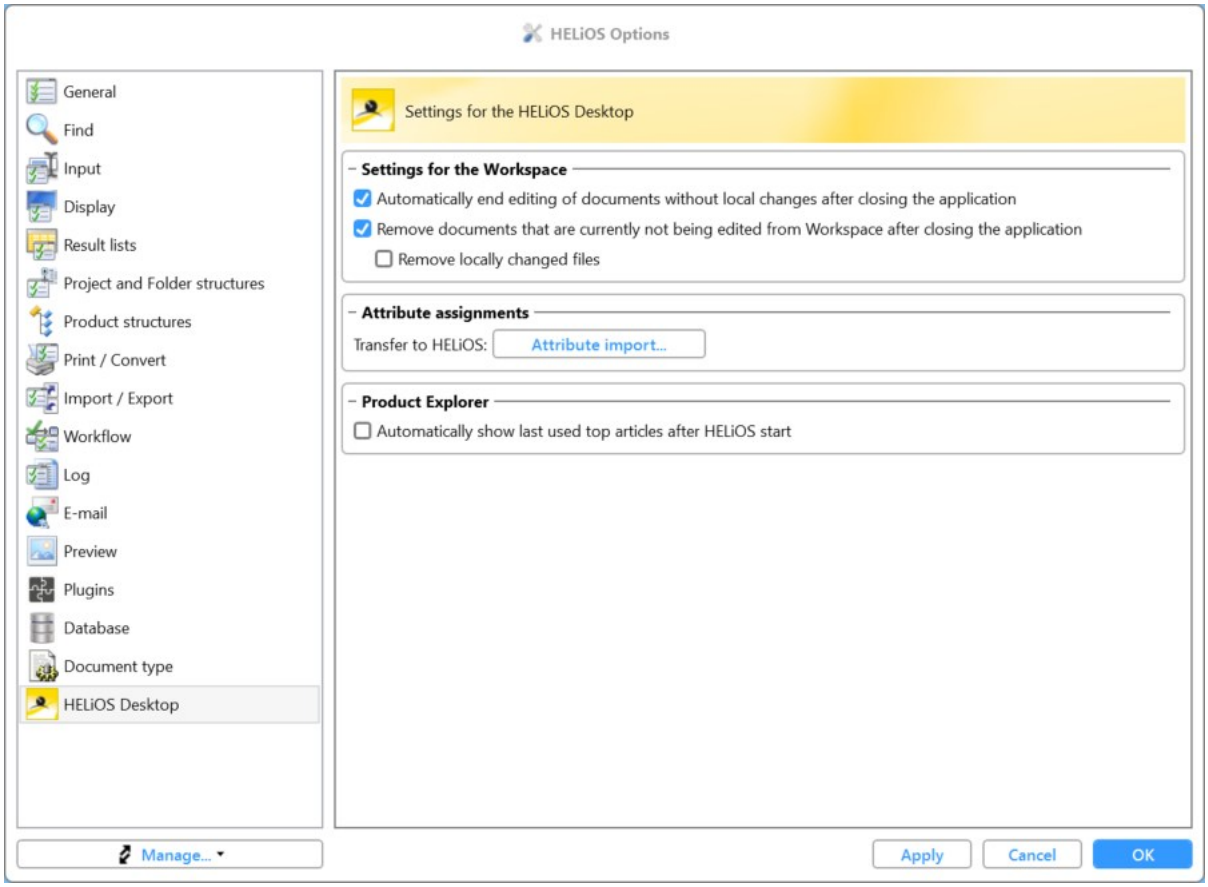
These will then be stored in the directory %LOCALAPPDATA%\ISD Software und Systeme\CrashDumps.

Major Release

Improvements of the UI and user guidance

Improvements to the user interface and user guidance of HELiOS Desktop 2025 can be found in the **HELiOS Options**, among others.

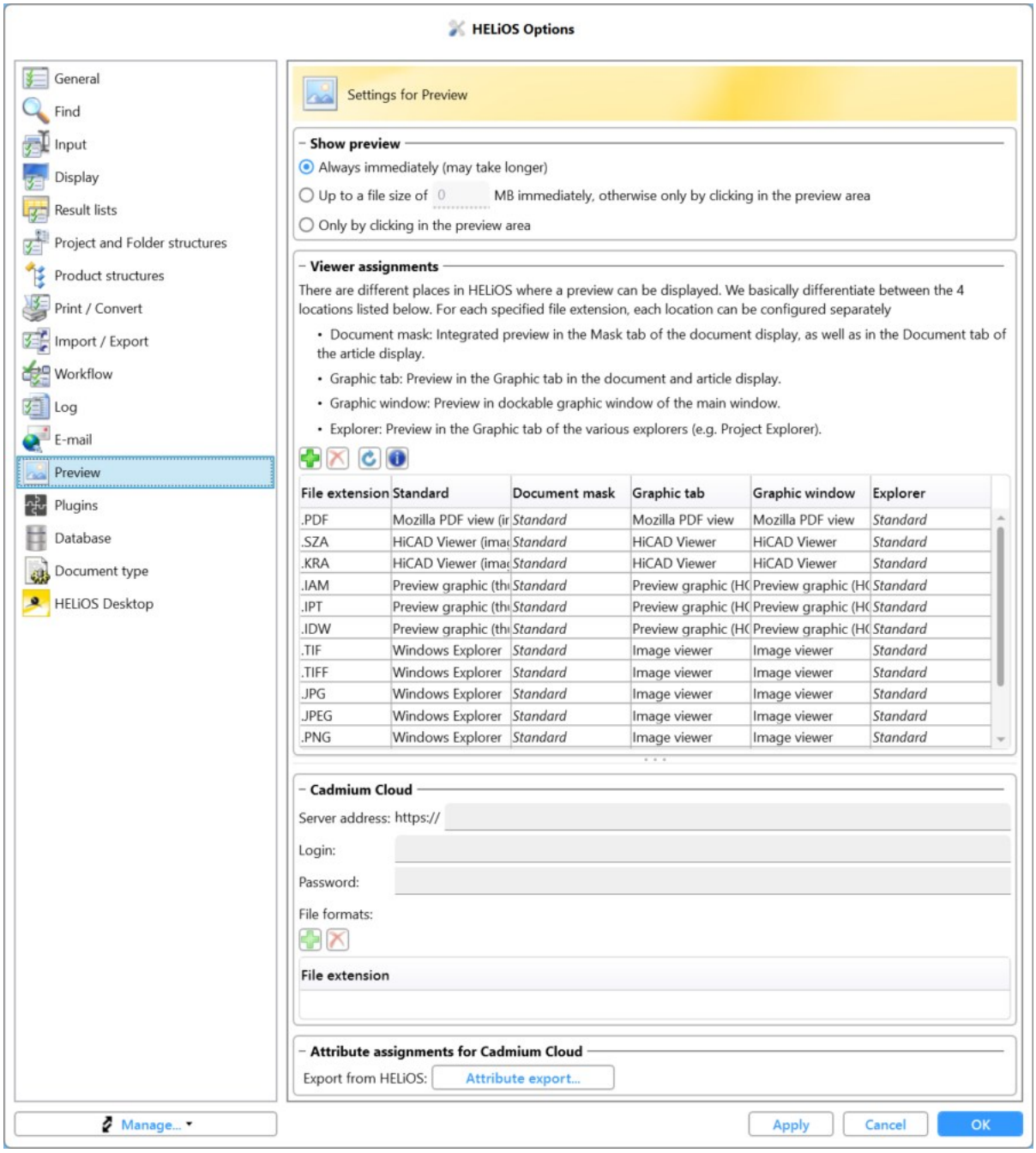
Revisions to the menu structure now make working in the Options window even more intuitive.



Revised preview and viewer integrations

The graphical preview displays of documents in HELiOS have been revised and improved.

In the **HELiOS Options** , you will find the new **Preview** tab, in which you can configure viewer assignments:



There you can configure which viewer should be used for each file format for the different places where a preview can be displayed in HELiOS (e.g. preview graphic in the document detail mask or in the Explorer context).

You can add further file extensions or remove those that are not required.

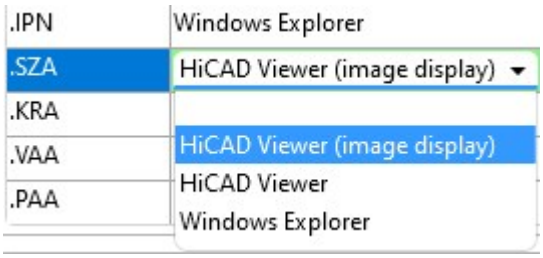
HELiOS generally distinguishes between integrated and external viewers:

- Integrated viewers are viewers that HELiOS provides automatically. A separate installation by the user is not necessary.

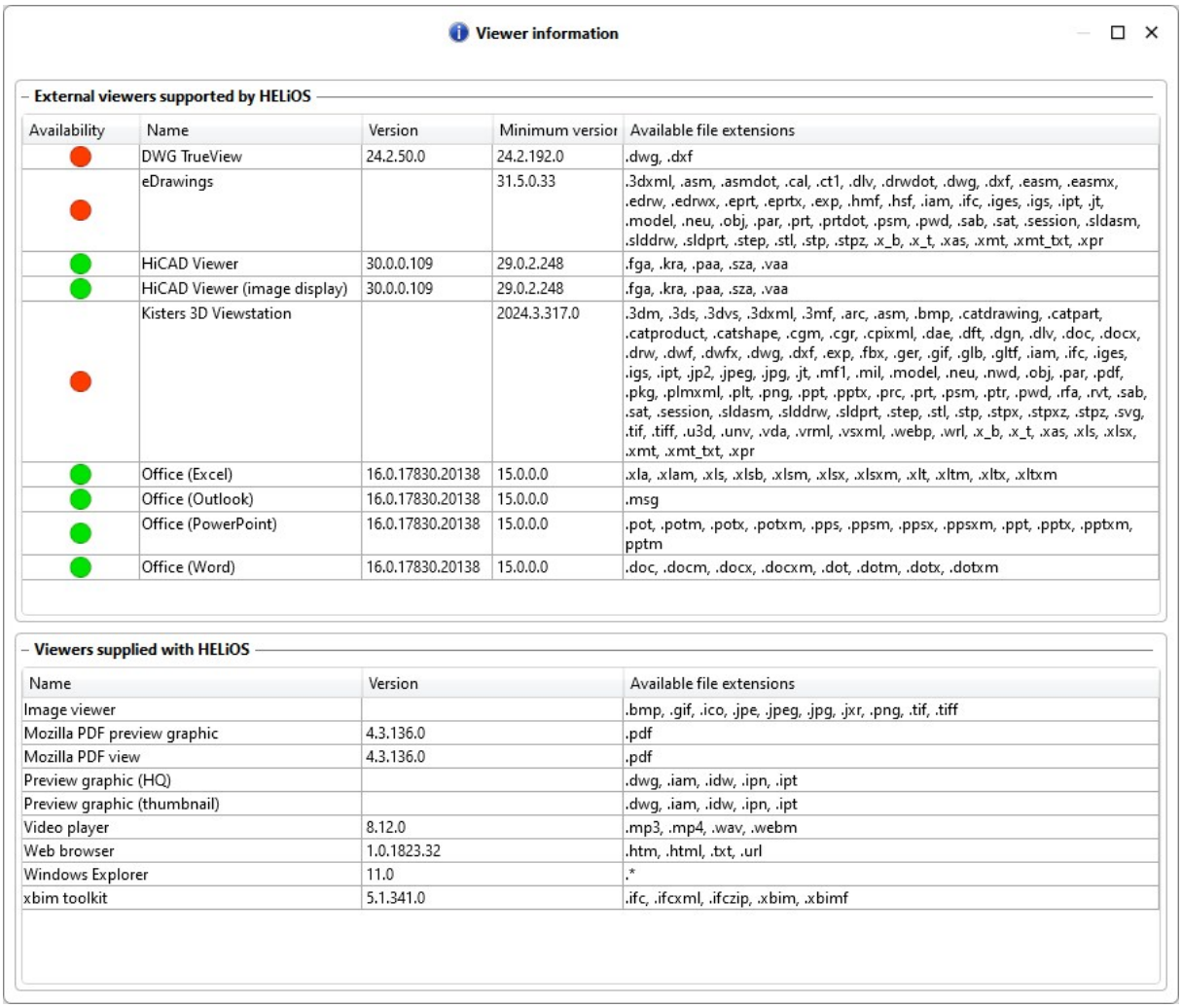
Examples include the **Image Viewer** for Bitmap formats, **Mozilla Pdf** for previewing PDF files or the **xbim toolkit** for IFC files.
Changes to the configuration of the viewer assignments have a direct effect for integrated viewers (i.e. without restarting HELiOS).

- In addition to the integrated viewers, HELiOS also supports several other viewers that you can install yourself.
  Examples of these are the viewers for **Office** documents or, for example, the **Kisters 3D Viewstation**.

When editing the viewer assignment, only those viewers are offered that also support the respective file extension.

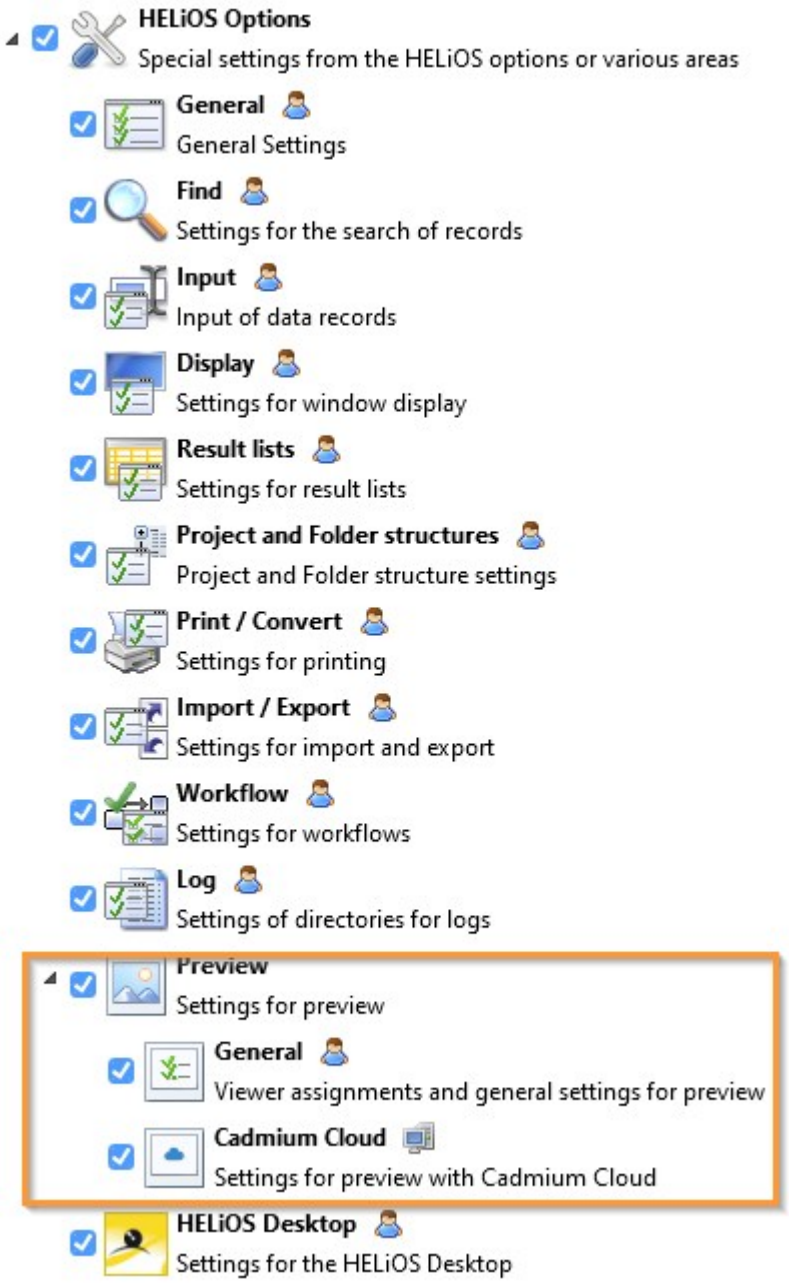


Clicking on the icon in the **Viewer assignments** area of the dialogue window displays a list of all viewers supported by HELiOS with further information:



The new configuration of the preview is saved user-specifically. (In older HELiOS versions, it was identical for all logged-in users).

The **Preview** settings can also be managed via the export/import mechanism of the UI settings in HELiOS:



You can also choose between the general preview settings and those for Cadmium Cloud; both can be individually exported and imported.

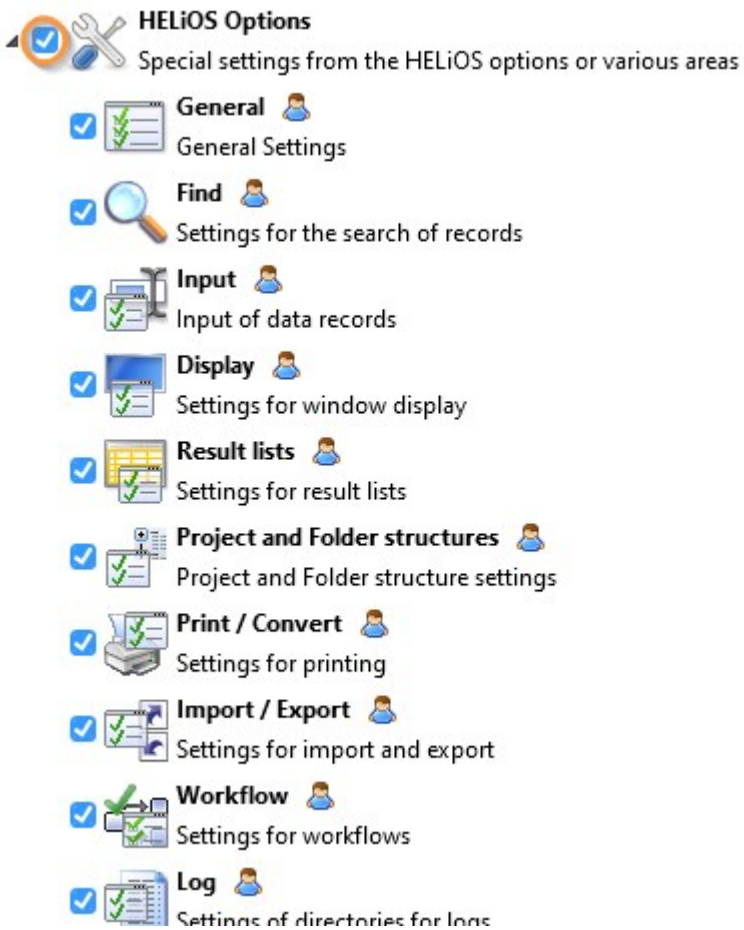
Please note:

The system file hel_preview.ini from earlier HELiOS versions is no longer required as a result of this revision and is no longer created by new installations. The defaults of the new preview configuration are set up

Update installations of older HELiOS versions with HELiOS 2025 (Version30.0.0) or higher create a new configuration file from the existing hel_preview.ini accordingly and leave the old file unchanged in the file system The reason for this is that different users may be able to log in and the corresponding settings are then migrated to the new functionality for each of the (newly) logged in users.

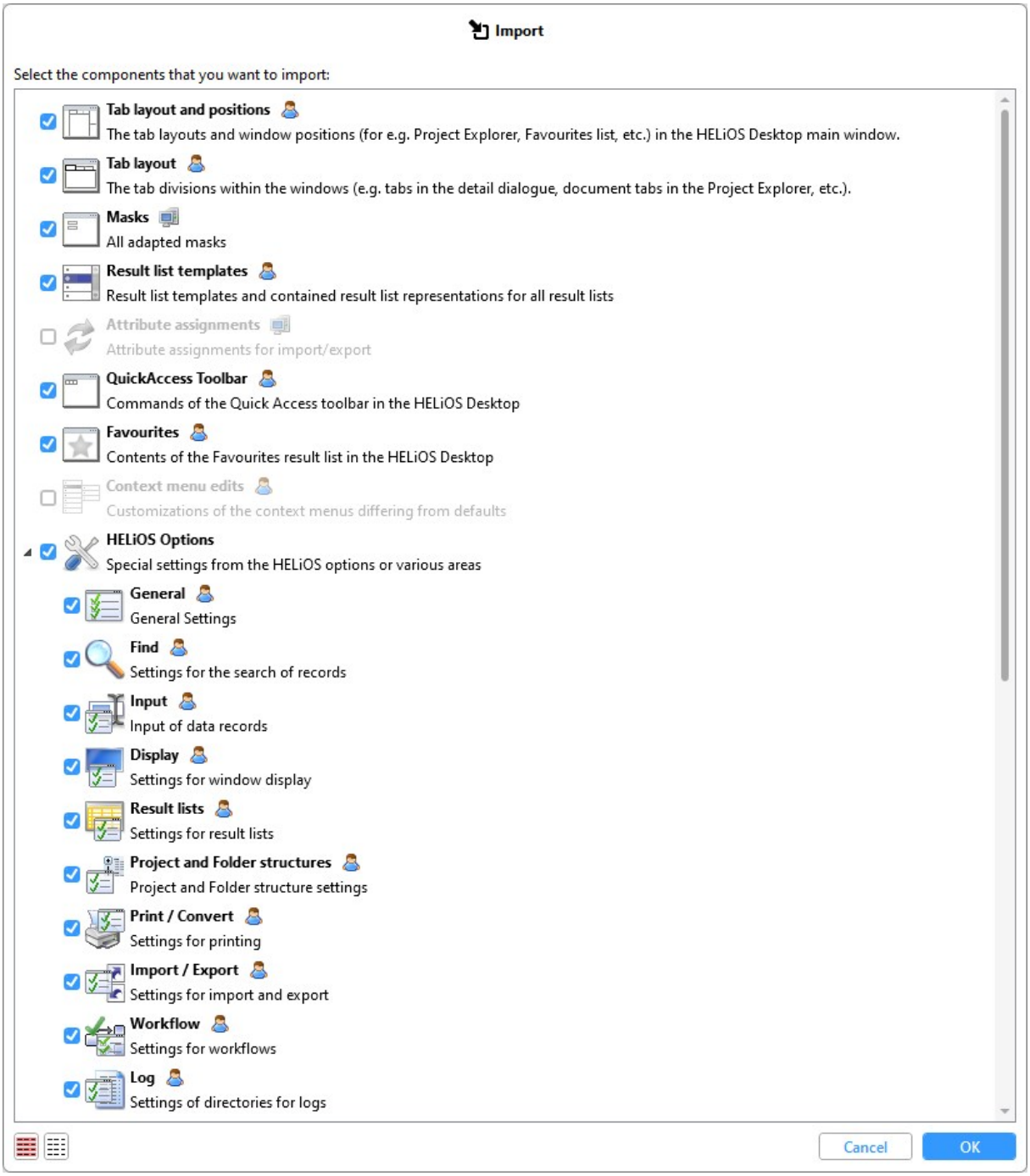
Export and import of UI settings

When exporting HELiOS Options via the export/import mechanism for user interface settings in HELiOS, all settings can be added to or subtracted from the export or import by selecting or deselecting the top entry of the HELiOS Options with a single click on the checkbox.



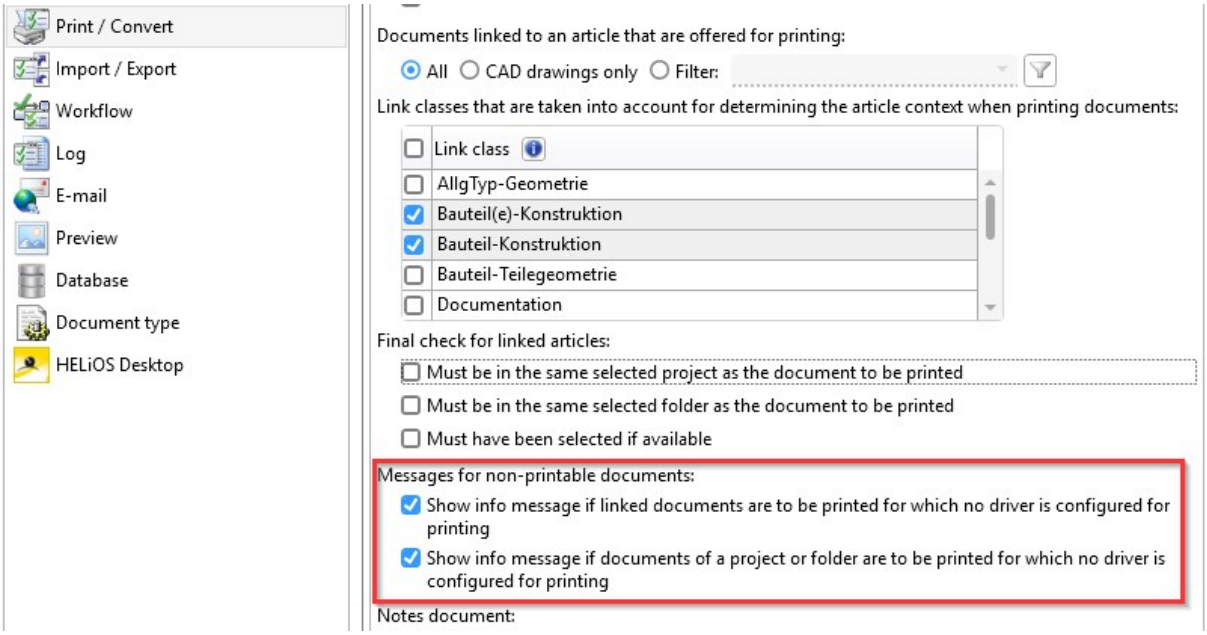
Furthermore, non-exported settings are not offered for selection during import.

The symbol indicates user-specific settings, the calculator symbol indicates settings of the local system.



Messages in case of non-printable documents

In the **Print/Convert** tab of the **HELiOS Options** dialogue window, you will find two new options under **Messages for non-printable documents**:



Both options control the handling of documents for which no corresponding driver is configured when printing via the Spooler.

These may be KRA files (HiCAD geometries), for example, which are saved as documents in HELiOS.

If non-printable documents have been selected for printing when calling up when calling up **Print documents of folder** or **Print documents of project** or **Print linked document**, an info message such as the following appear:

If the corresponding option has been deactivated, no info messages appear in scenarios such as those mentioned above. The non-printable documents are filtered out in the background.

Messages that are due to the explicit call of a non-printable document via **Print (Spooler)**, cannot be suppressed:

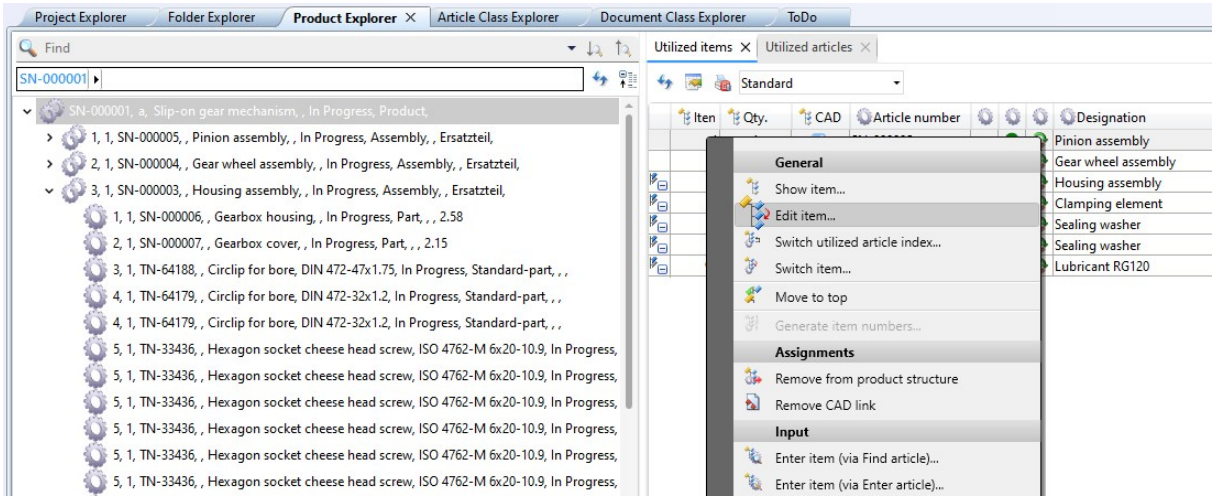
In this case, printing can be cancelled by clicking on **No**. If you click on **Yes**, the non-printable documents are filtered out and no longer listed in the subsequent print dialogue.

Temporary editing of utilized items

In earlier HELiOS versions, when editing a utilizeded item, the user was prompted to manually accept changes to the product structure if the top part of the product structure was not set to "In progress".

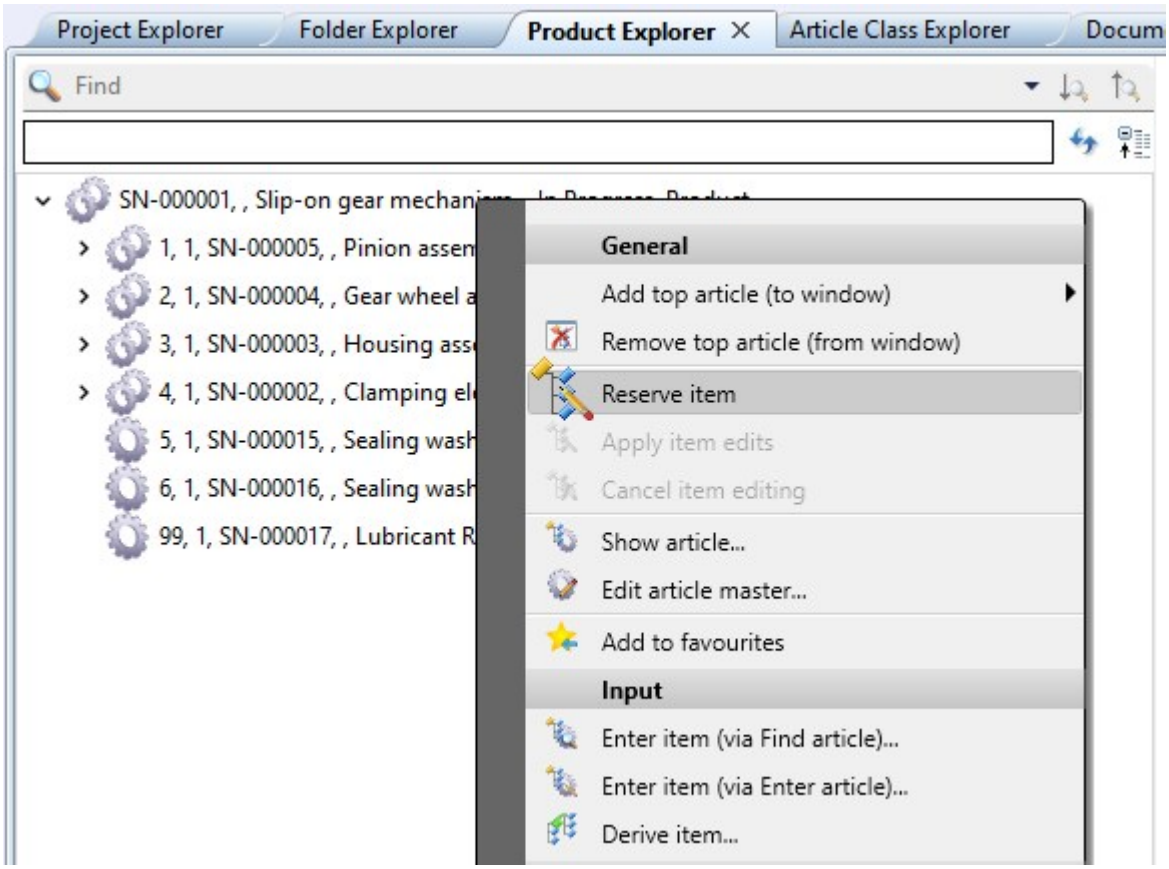
In order to eliminate this intermediate step, the behaviour in such situations has been changed as follows:

- If the top part of the product structure has not yet been set to ‘In progress’, it will be set to this status temporarily for the operation. At this moment, the top part assembly is locked for other users. After completing an edit or entry, the changes are automatically applied.
- If the top part is already being edited, you have the option of ‘reserving’ the top part assembly and can manually apply or cancel the editing at a specific point in time.



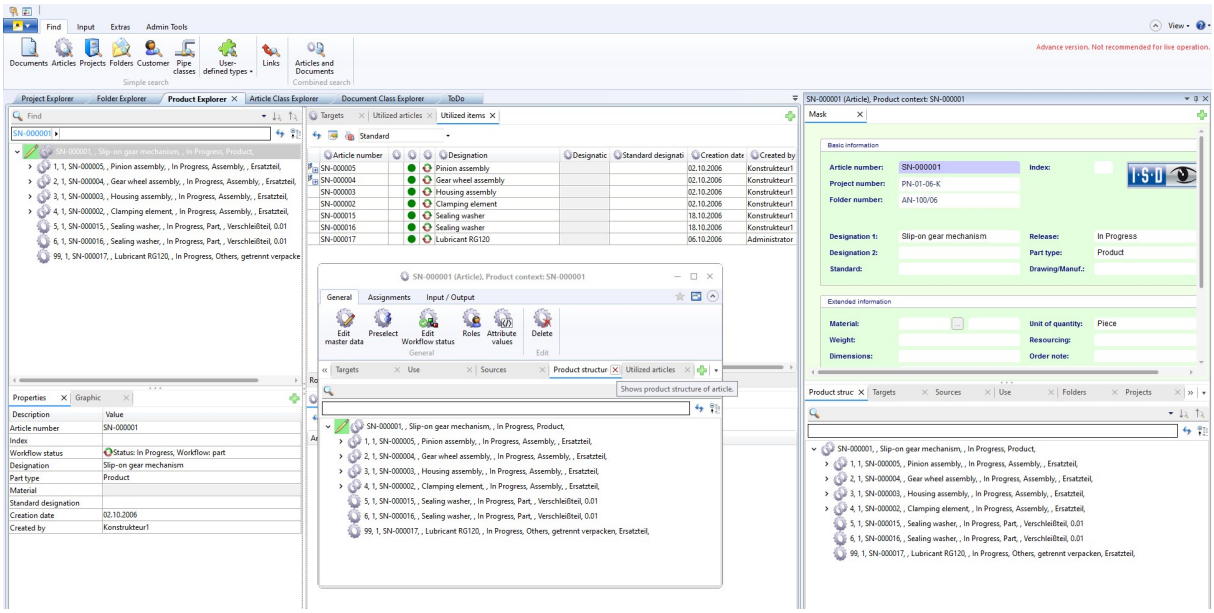
Product structure: Reserve 1-level assemblies for editing

When a direct function such as  **Reserve item** from the Product context menu, or when a product structure is synchronised from a linked CAD application, the product structure's 1-level assemblies are set to **In progress**and are thus locked for others by the corresponding user.



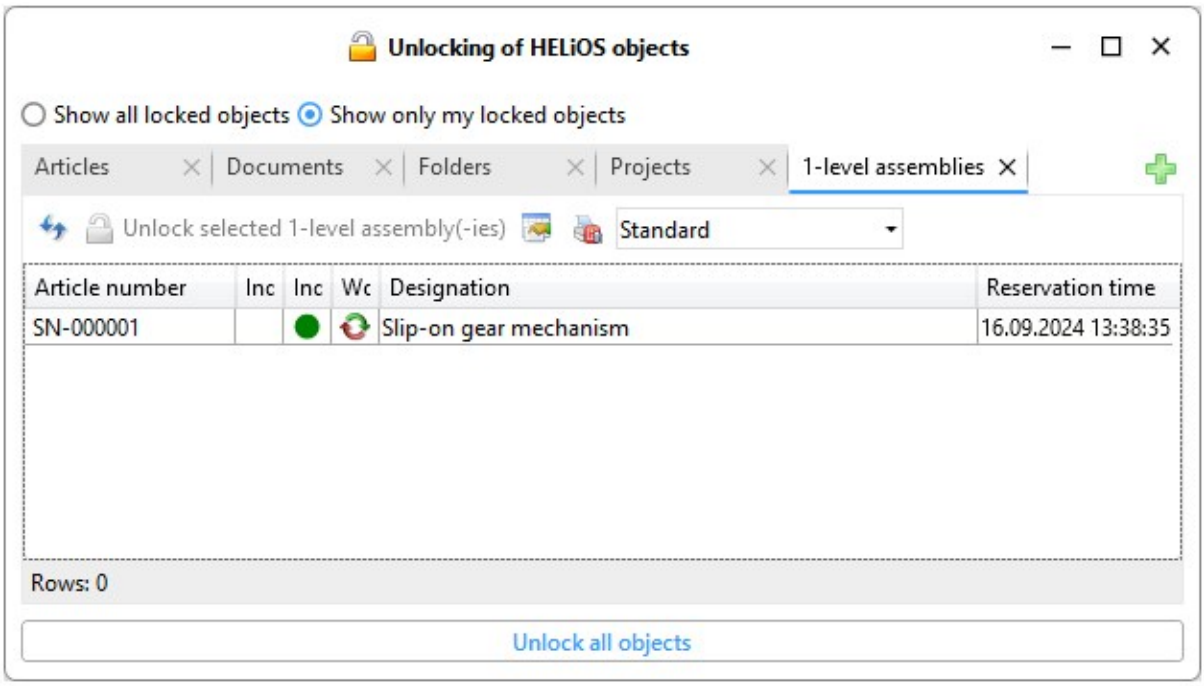
Please note in this context:

The reservation has a global effect in the context of the corresponding HELiOS user. This means that reservations of 1-level assemblies are consistently indicated (within the same application) by a green marking of the header with a pen icon.

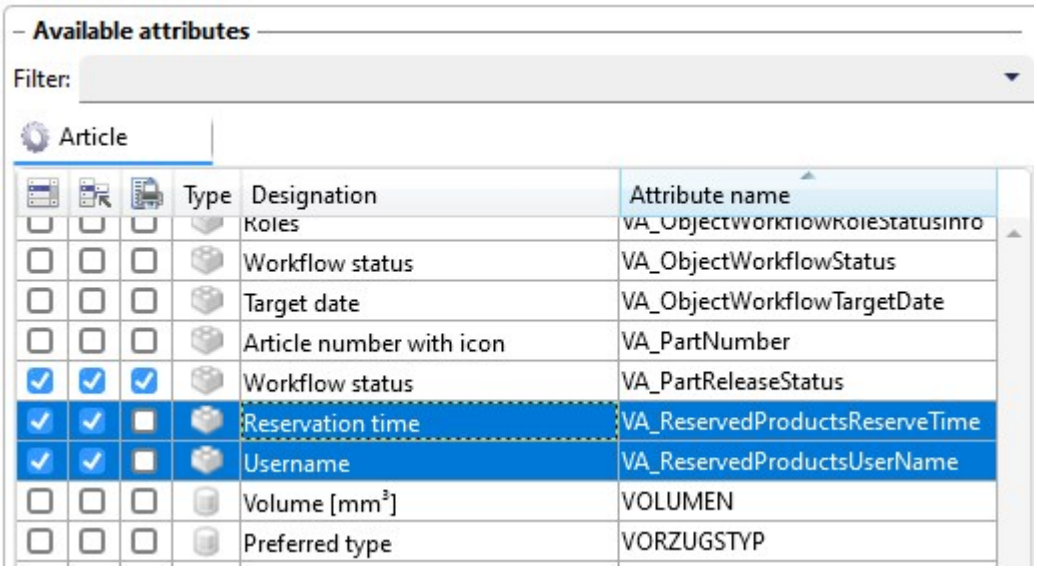


Editing is possible in all dialogue windows with this marking.

In the **Locking Management** tool of HELiOS, the 1-level assemblies reserved for editing are displayed accordingly:



In the result list for 1-level assemblies , virtual attributes are available to you, which can be used to show the start of the reservation (name: **Reservation time**, attribute name: **VA_ReservedProductsReserveTime**) or a display of the corresponding HELiOS user (name: **Username**, attribute name: **VA_ReservedProductsUserName**) in the Locking Management.



HELiOS-URLs

Opening and activating Project and Folder Explorers

Similar to URL calls, which are also possible with the HELiOS Internet Server, you can also open HELiOS objects via URL calls in the context of the Folder Explorer or Project Explorer when working with the HELiOS Desktop.

This means that the corresponding Explorer context for a HELiOS object is also opened via a call specified in this way. The Explorer jumps to the corresponding position.

The new partial commands in a URL row are:

- **explore-and-activate-folder**
- **explore-and-activate-folder-by-id**
- **explore-and-activate-Project**
- **explore-and-activate-project-by-id**

If the HELIOS Desktop is not running when a HELiOS URL is called (from another application), it will be started automatically.

Show newest index

With the extension **&indexStrategy=NewestRevision** in a URL call, it is possible to open the detail mask of the most current index status for a document (show-document) or an item (show-article) of HELiOS.

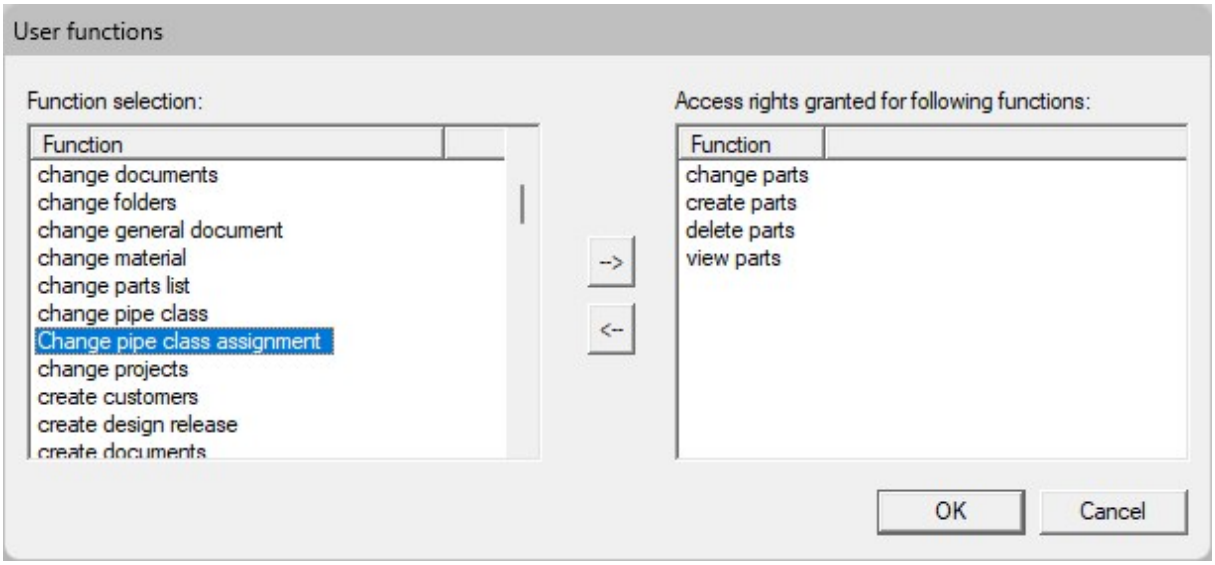
Example of a link:

- **helios://show-article?attributes=%7bHEL_SACHNUMMER:SN-000016%7d&indexStrategy=NewestRevision**

The URL uses '&indexStrategy=NewestRevision' to call up the most current index of an article with the article number 'SN-000016'.

Change pipe class assignment

For working with pipe classes, the special user right **Change pipe class assignment** is now available in the User Management of EdbSetup. This right can be granted to HELiOS Users and Groups:



Only Users and Groups to whom this right has been added can

- Assign parts to a pipe class and
- Remove the assignment of parts to a pipe class.

Please note:

This right is automatically assigned to PDM Administrators of the HELiOS default database during the update, but it must be manually assigned to all other Users and Groups.

Note: Adding the KRA version number for update installations

Since HiCAD Version 2024, version numbers of 3-D parts (KRA files) have been stored in HELiOS.

However, these version numbers may not be available in HELiOS for inventory data from customers who do not work with the Vault Server. Therefore, when updating a database version older than 30.0.0 to a higher version of HELiOS, the version numbers of the KRA files are transferred to HELiOS once.

You will receive a corresponding message during the update process. Further information can be found in the installation manual.

If errors occur during the migration, it is possible to run the migration tool again via the command line.

Important note on update installations: Conversion of system directories of HELiOS Workspaces

Beachten Sie bei der Update-Installation einer älteren Version auf HELiOS 2025 (Version 30.0.0) oder höher, dass sich die Verzeichnisstruktur ändert.

 Since an automated migration is not possible, all users must check in all data and empty their workspaces before an update installation in order to avoid data loss.

In previous versions, the workspaces were located under %localappdata%.

This meant that different workspaces could be located on one system.

To counteract this, the workspaces will be moved to the %programdata% directory with the update to HELiOS 2025.

Checked-out files are then stored at **%programdata%\ISD Software und Systeme\HELiOS Workspace\(...)***

(*plus Location ID and User ID).

The version-dependent workspace databases are stored at **programdata%\ISD Software und Systeme\HELiOS <Version>\Location-ID**.

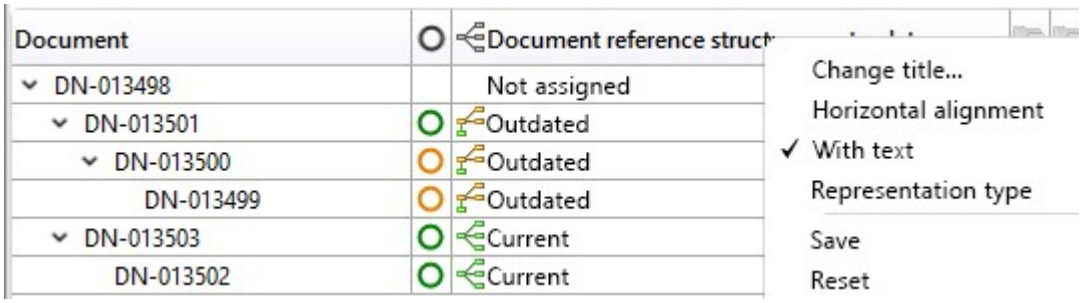
HELiOS in HiCAD

Service Pack 2

Model structure: Document reference structure up-to-dateness in the header

In the **Model structure** tab of the HELiOS document master, the attribute **Document reference structure up-to-dateness (UI_ReferenceTreeIsOutdated)** also displays the up-to-date status of the references in the header of the document structure.

Please also note that parts without any further referenced structures are also marked green and are therefore displayed as up to date ("Current").



Performance increase for derivations

When an article with linked objects is derived in HELiOS and the drawing is then opened in HiCAD, the corresponding part in the drawing is replaced by the derivation. In the past, this could lead to time-consuming recalculations.

With the update to Service Pack 2 of HiCAD 2025, a considerable performance increase could be achieved here.

HELiOS user in HiCAD window caption

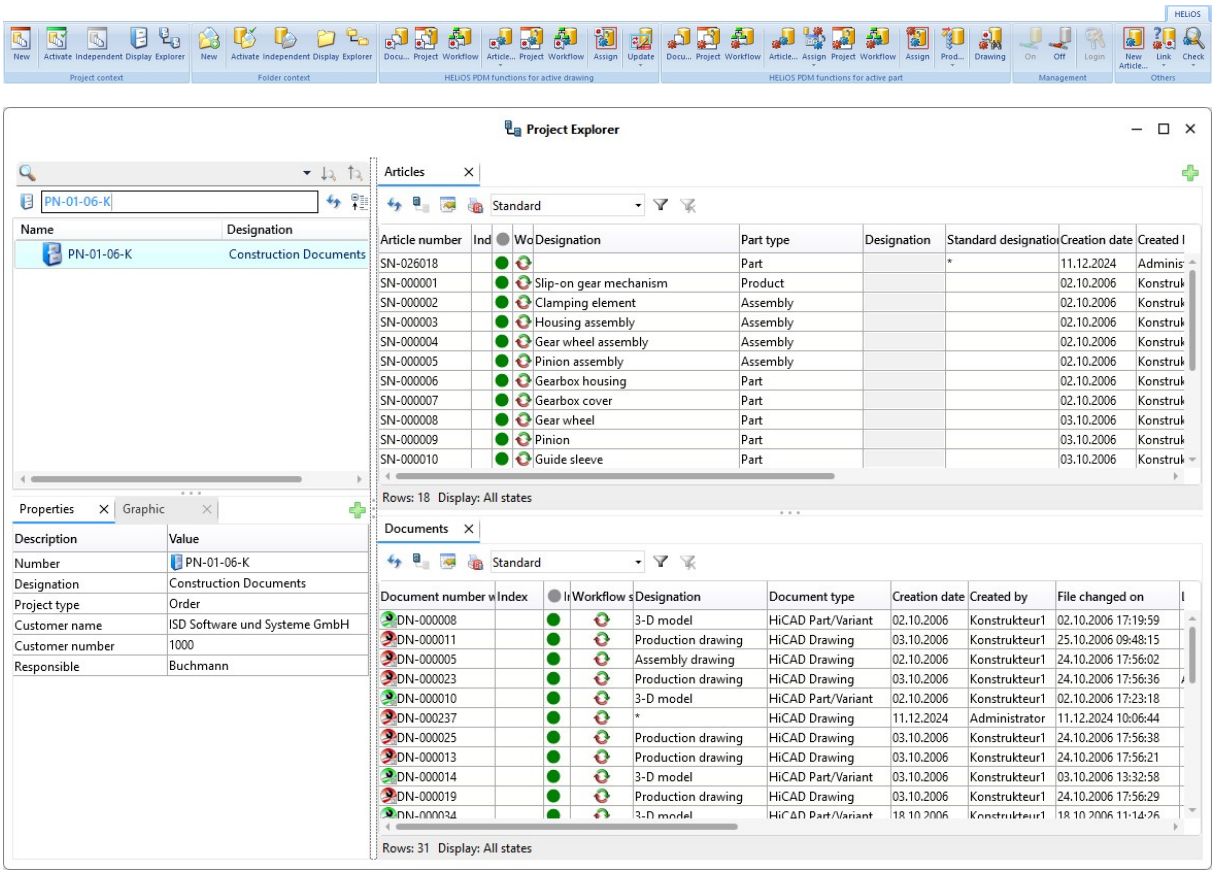
From Service Pack 2 onwards, the logged-in HELiOS **User** and the user **Group** are displayed in the HiCAD window caption (instead of in the HiCAD Properties window).

Service Pack 1

HELiOS Ribbon with new functions

With the update to Service Pack 1, the “HELiOS PDM” Ribbon tab has been renamed to **HELiOS**.

New on this tab are the function groups **Project context** and **Folder context**, which include functions for the HELiOS detail masks of the active project or the active folder, as well as the Project Explorer and the Folder Explorer of HELiOS.



Recursive visualisation of outdated documents in the document structure

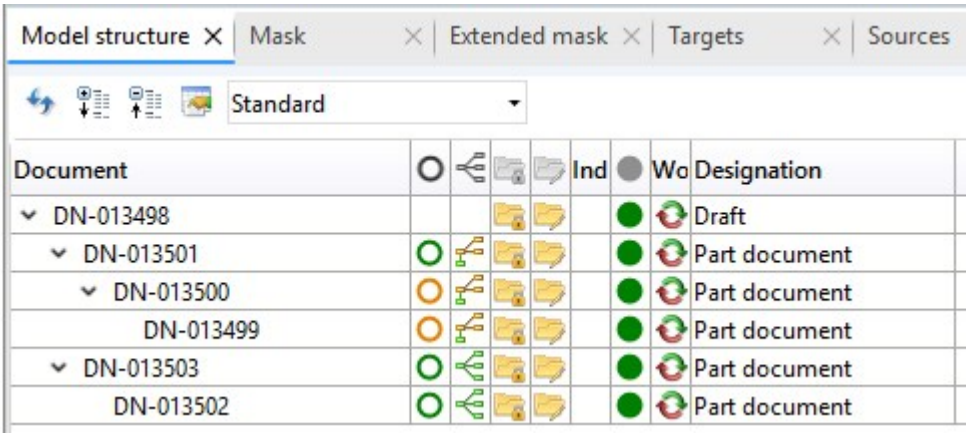
The virtual attribute **Document reference structure up-to-dateness** (UI_ReferenceIsOutdated) is available to indicate referenced documents that have changed since the reference was created, and can be shown in a HELiOS document structure.

Document reference structure up-to-dateness (UI_ReferenceTreeIsOutdated) is a further virtual attribute that has been added, which now also allows a recursive display.

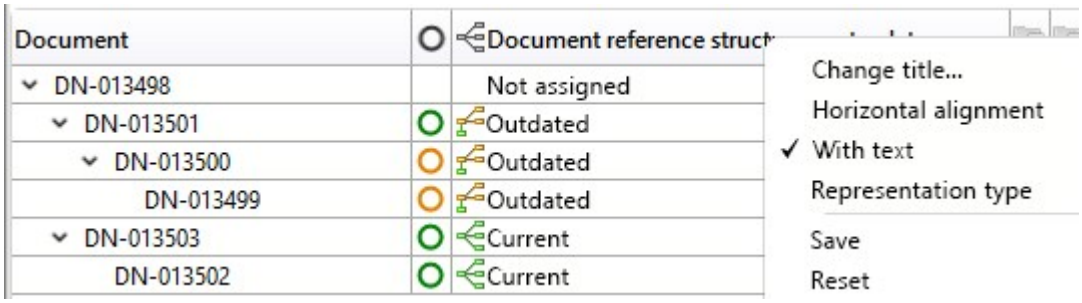
Document reference up-to-dateness	UI_ReferenceIsOutdated
Document reference structure up-to-dateness	UI_ReferenceTreeIsOutdated

This means that the attribute indicates whether there is a referenced document in the corresponding subtree of the structure that has changed since the time of referencing.

The display in the result list of the document structure then looks like this (under with pure icon display):



You can also switch to a display **With text**:



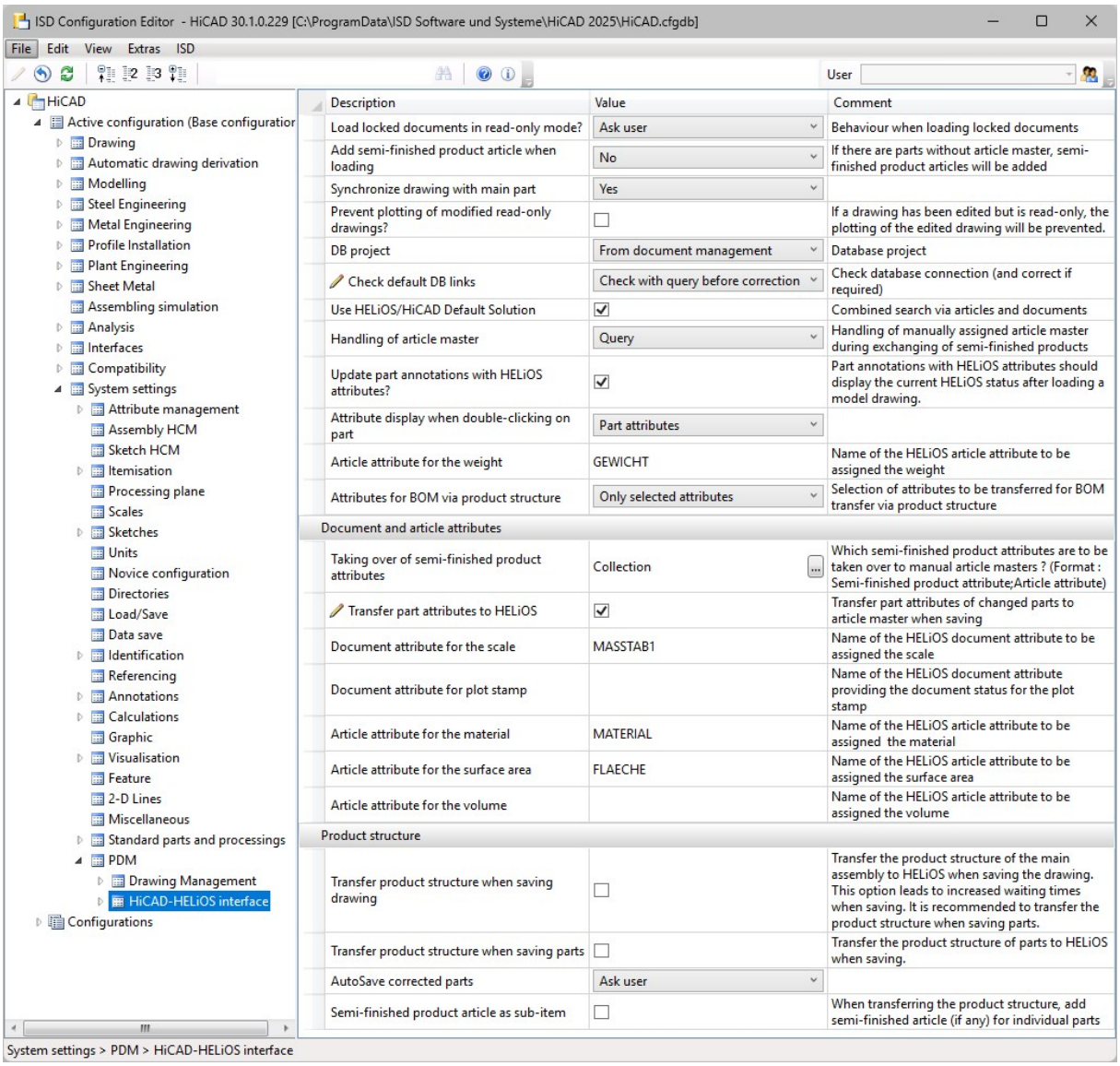
If the documents marked accordingly are opened, they are updated automatically (because they contain non-current references).

You can therefore also use the attribute in the results list display to proactively detect such updates and trigger them if necessary.

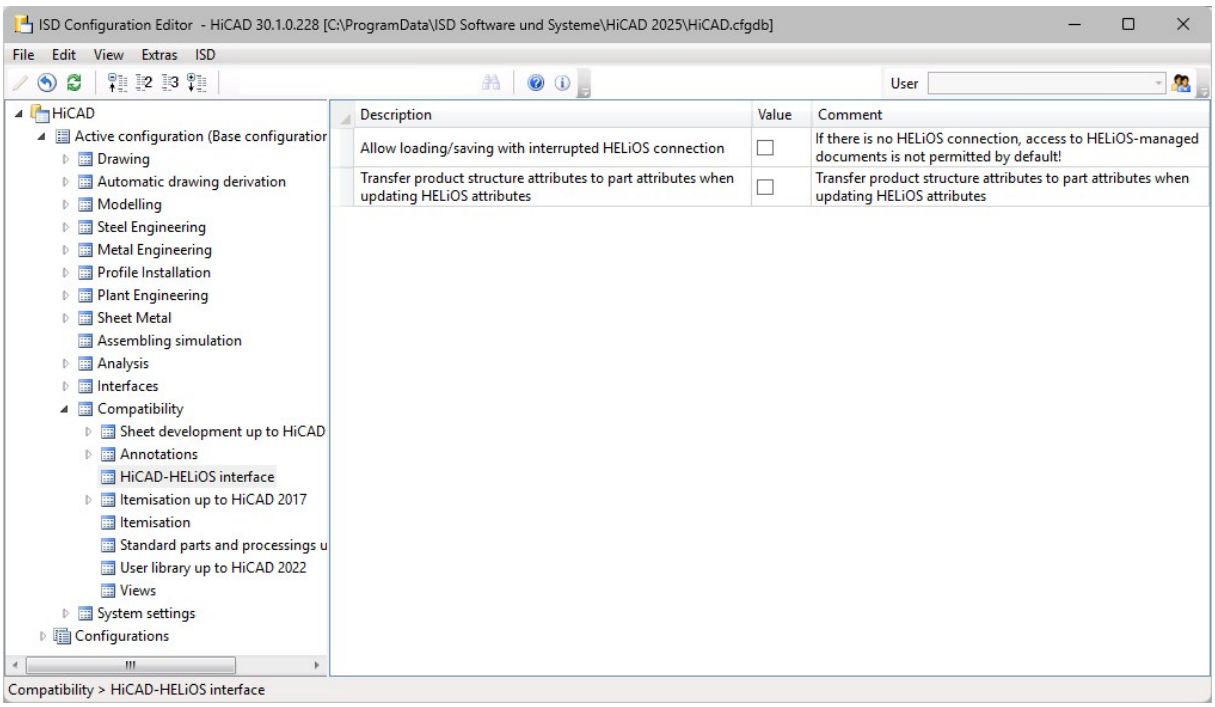
Configuration Editor: HiCAD-HELiOS interface

With the update to Service Pack 1, the interaction between HiCAD and HELiOS has been combined in the **Configuration Editor** (ISDConfigEditor):

Settings concerning the product structure or document and article attributes, which were distributed across different sub-menu paths in earlier versions, have now been combined at **Active Configuration (Base configuration) > System settings > PDM > HiCAD-HELiOS interface**:



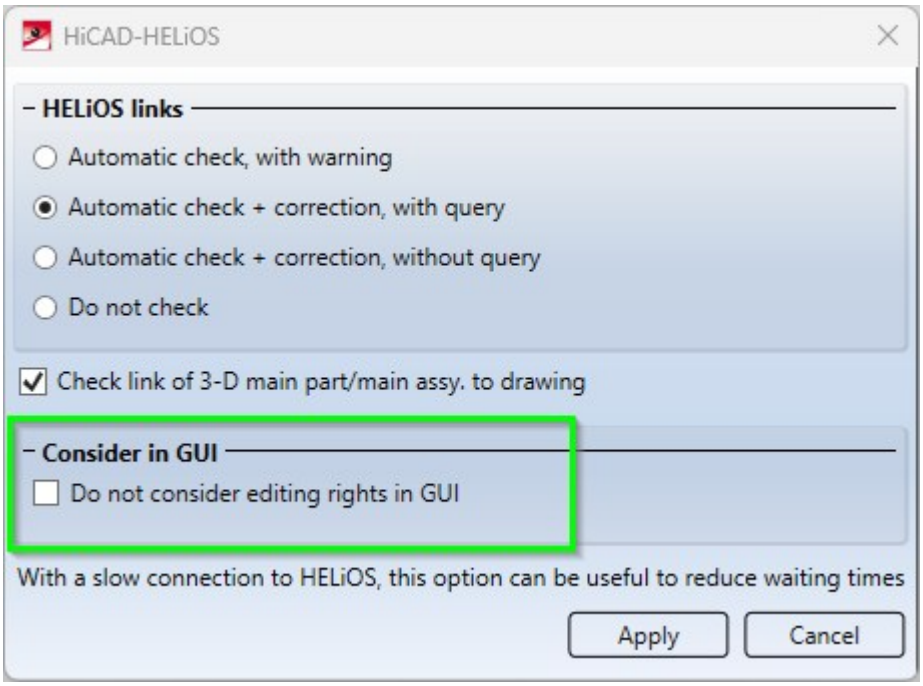
Settings for the database connection and for updating HELiOS attributes for HiCAD can be found at **Active configuration (Base configuration) > Compatibility > HiCAD-HELiOS interface**:



Major Release

Editing rights in the user interface

In the dialogue window that opens when you select **HELiOS > Others > Link > Settings** , you will find the new menu area **Consider in GUI** with the checkbox option **Do not consider editing rights in GUI**.



It is not activated by default.

If the connection to HELiOS is slow, this option can be useful to reduce waiting times.

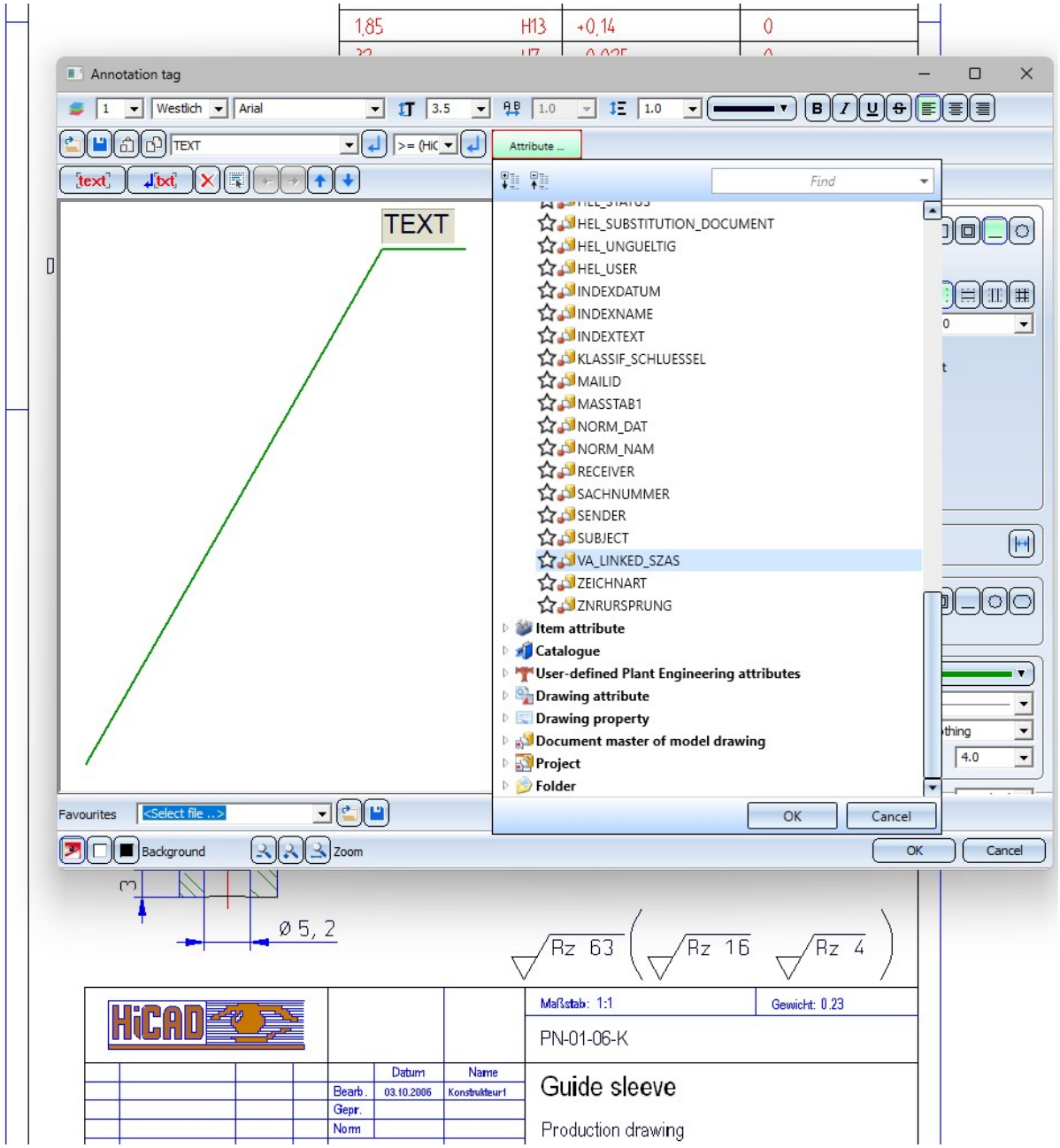
Please note:

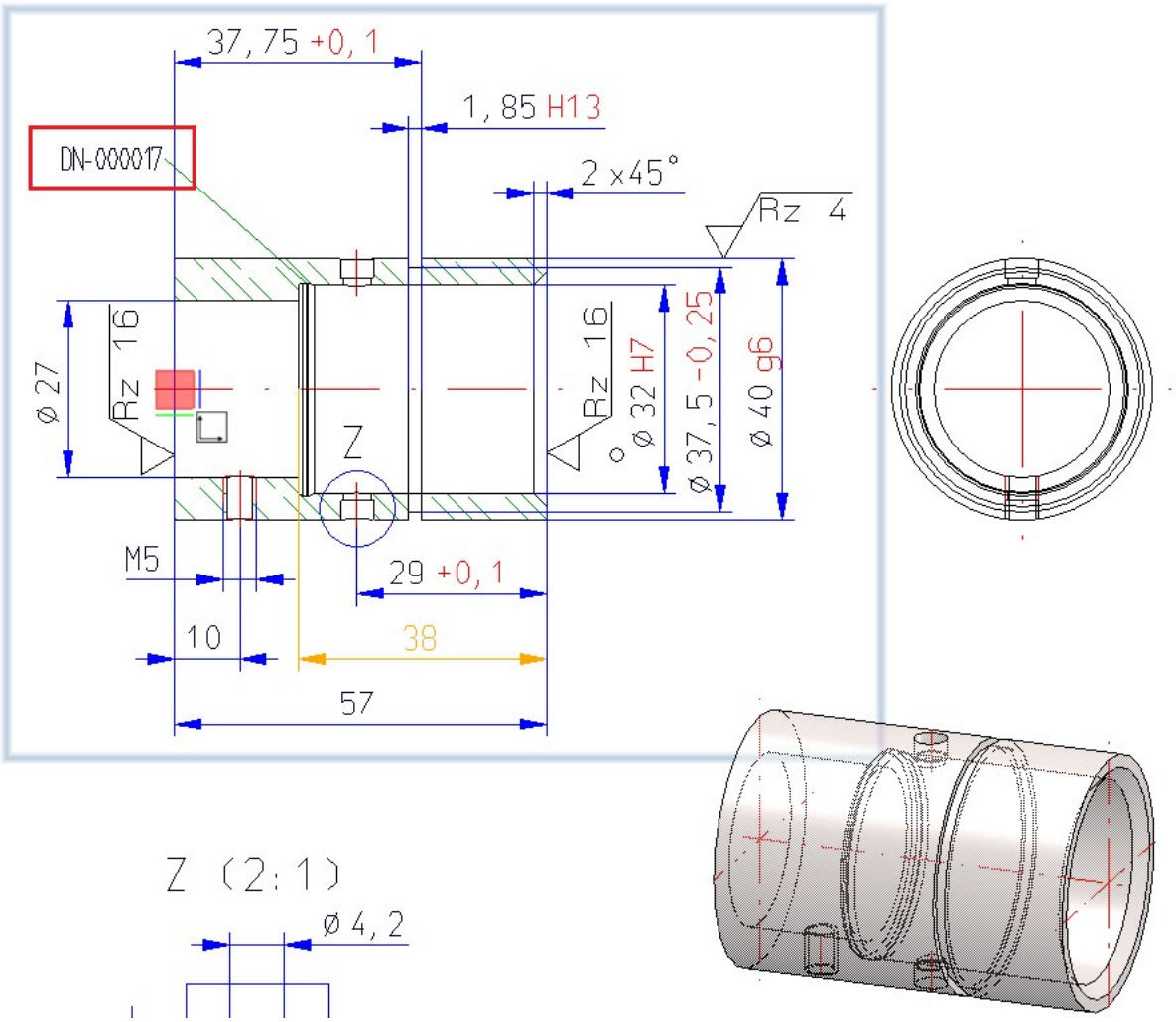
In earlier HiCAD versions, this settings dialogue was called "Automatic 3-D main part and link check in HELiOS functions".

Merging it with other options changes the name.

Document number of drawing

A new HELiOS attribute can be used in HICAD's Text Editor for annotation tags: the virtual database attribute **VA_LINKED_SZAS** (under **Document master of part**) displays the drawing document number of a sub-part in an assembly.




37, 75 +0, 1
1, 85 H13
DN-000017
2 x45°
Rz 4
Rz 16
Rz 16
ø 27
ø 32 H7
ø 37, 5 -0, 25
ø 40 g6
Z
M5
29 +0, 1
10
38
57
Z (2:1)
ø 4, 2

Load/Save behaviour when the HELiOS connection is interrupted

Please note the following changes in the application behaviour when working with HELiOS in HiCAD:

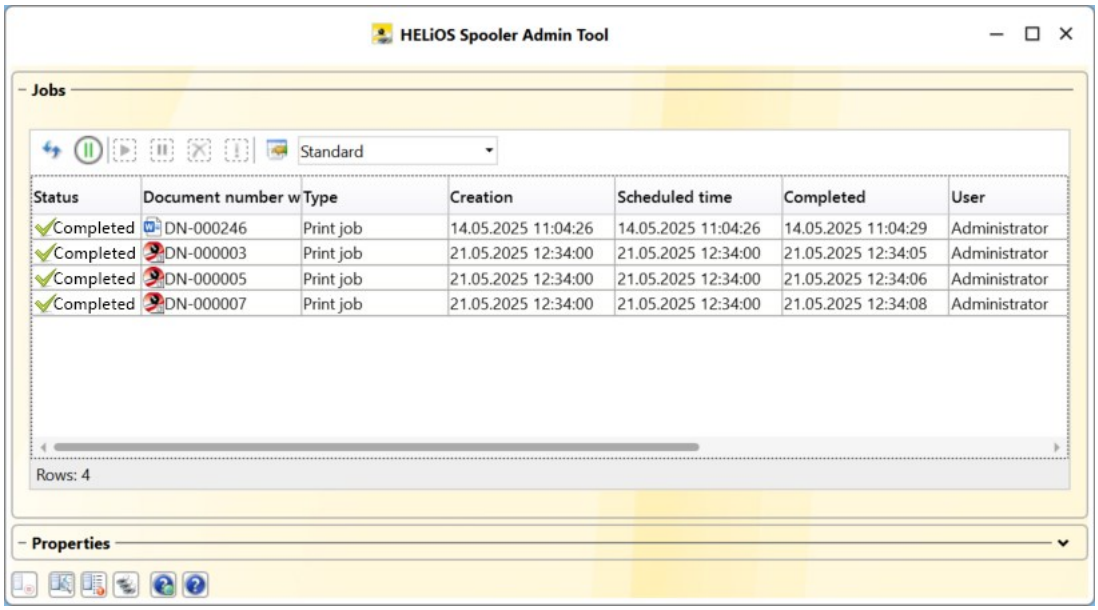
- Files last loaded in HiCAD always indicate whether they were loaded from HELiOS. Such files cannot be reloaded if (the connection to) HELiOS is not active.
- If a HiCAD file (.sza) is loaded from the HiCAD drawings directory (set in FILEGRUP.DAT) via the Windows Explorer, the system checks whether a corresponding HELiOS flag is set in the file header. This prevents the file from being loaded locally. HiCAD files that have been loaded locally from other directories can be opened.
- The same restrictions that apply to SZA files do not apply to KRA files, which can be loaded and saved without restriction.
- If the connection to HELiOS is switched off during a running HiCAD session, all loaded HiCAD files with HELiOS reference (document master ID) are set to 'write-protected'. This means that they can no longer be saved (even if the database connection is re-established). Therefore, you will receive a warning message when switching off, which gives you the opportunity to cancel the switch-off.
- If desired, you can disable this behaviour in the **ISD Configuration Editor** by going to **Compatibility > HiCAD-HELiOS interface** and enabling the **Allow loading/saving with interrupted HELiOS connection** checkbox.

HELiOS Spooler

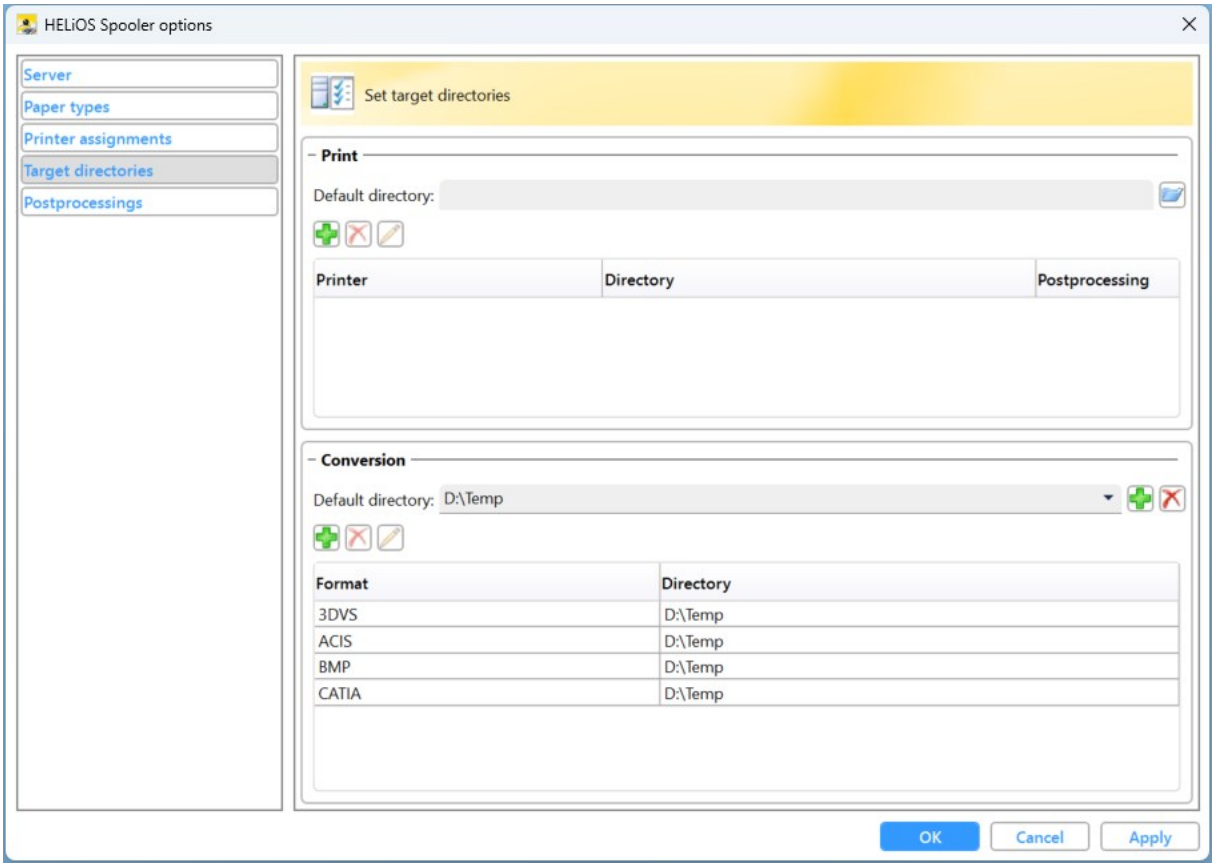
Service Pack 2

UI improvements and target directories

With the update to Service Pack 2 of HELiOS 2025, the **HELiOS Spooler Admin Tool** has also been revised and improved.



At **HELiOS Spooler Options > Target directories**, you now have the option of configuring additional target directories in addition to the default directory.



In earlier HELiOS versions, the system file ServerSettings.xml had to be adjusted manually, which is no longer necessary.

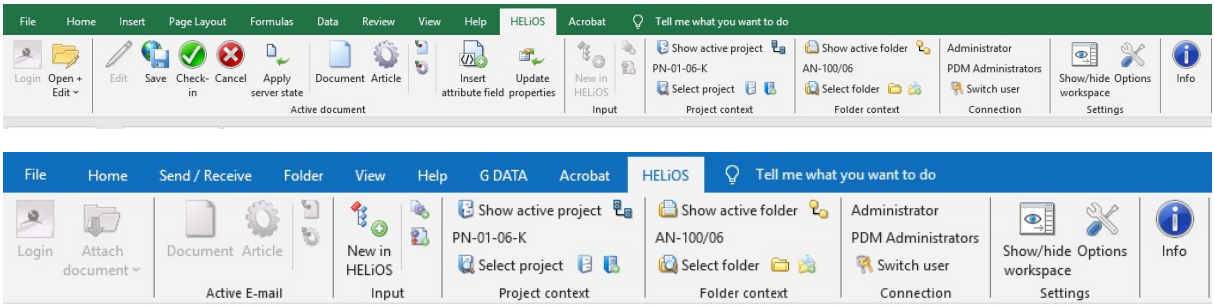
HELiOS MS Office Interface

Service Pack 1

Revised Office interface with an extended range of functions and options

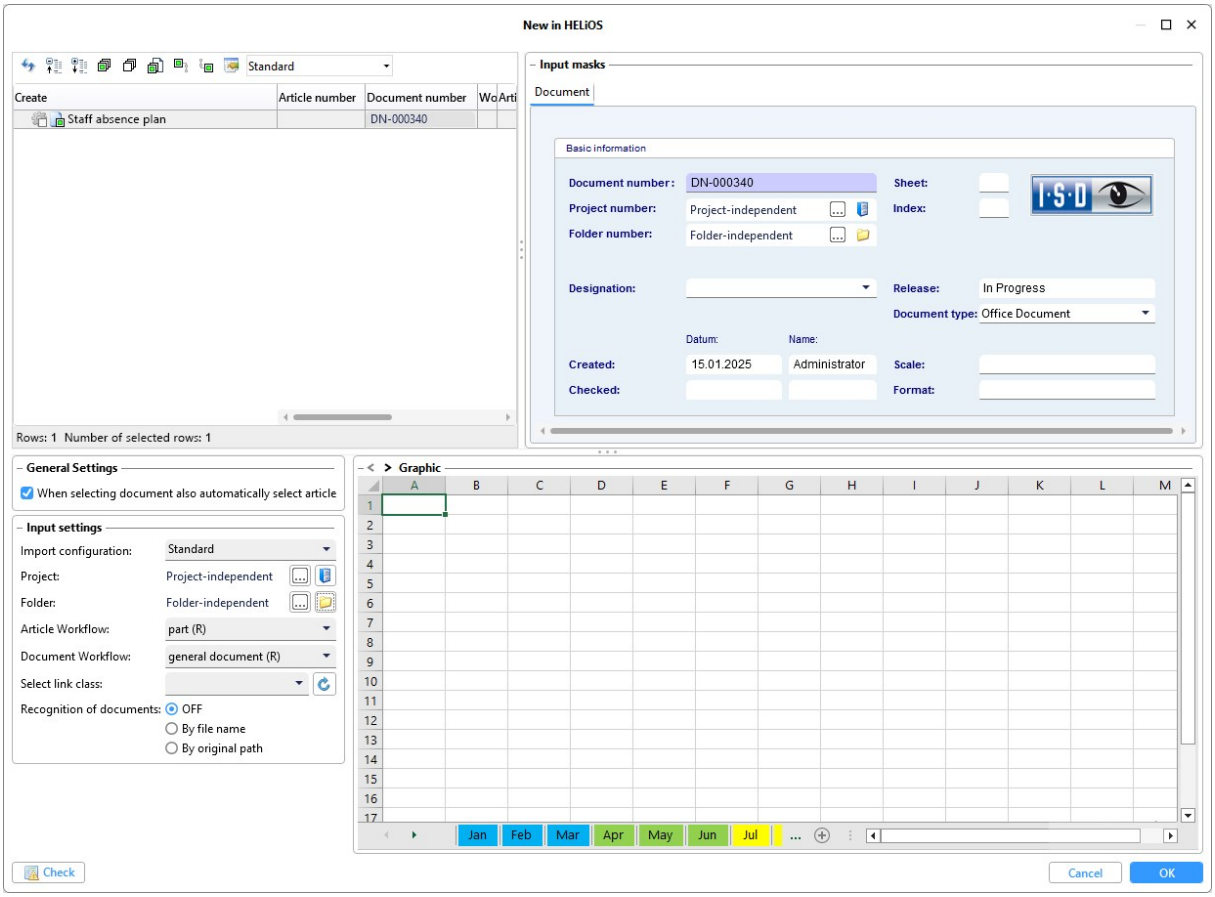
HELiOS-Ribbon

With the update to Service Pack 1, the **HELiOS** Ribbon in linked MS Office applications (Excel, Word, PowerPoint or Outlook) has been revised and adapted.



New in HELiOS

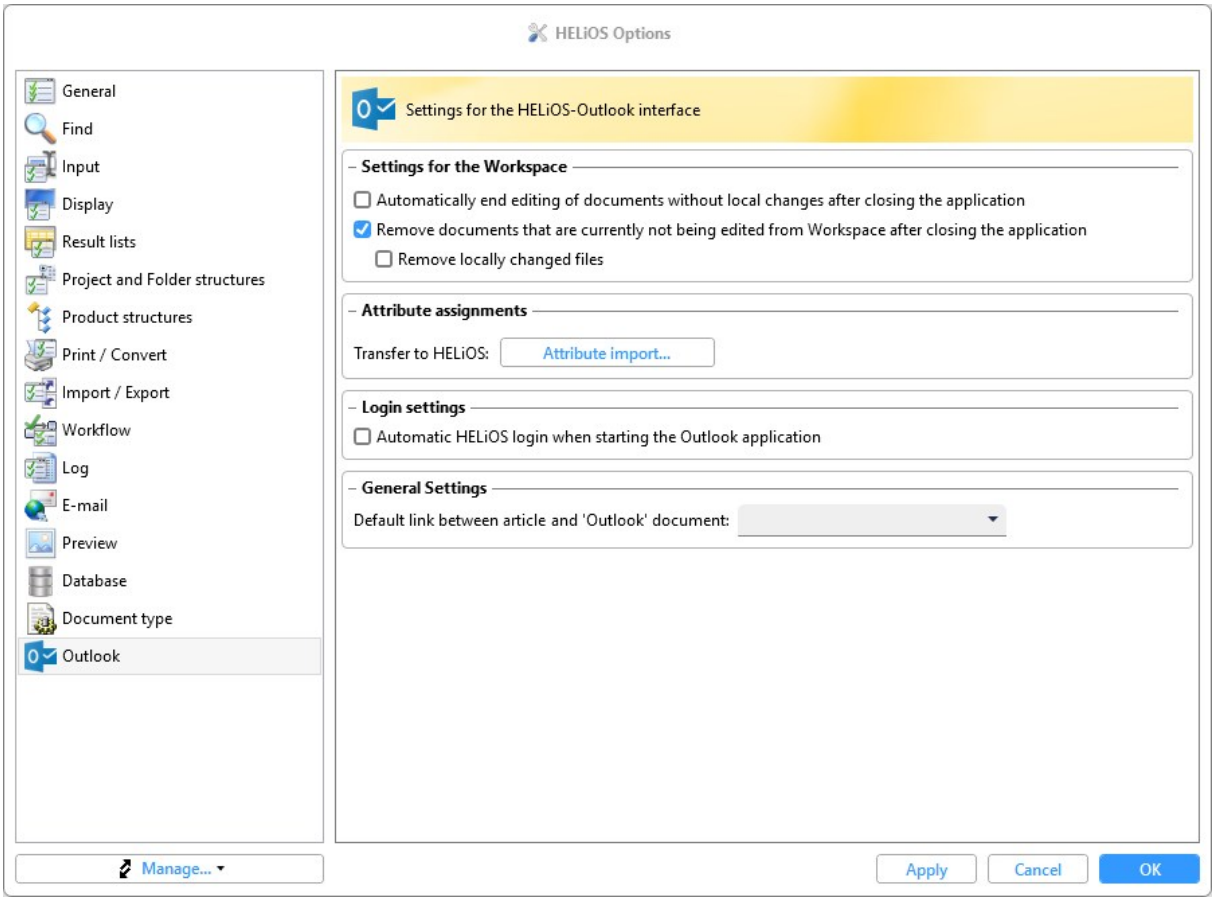
Similar to the multi-CAD functionality of HELiOS, the enhanced **New in HELiOS** dialogue is available for Office applications and for the transfer of e-mails and their attachments to HELiOS, in order to save files in HELiOS.



Please also note the information on the revised transfer mechanism for saving e-mails in HELiOS using drag & drop.

Options

The **HELiOS Options** window, in which you can make advanced settings for your interface with Excel, Word, PowerPoint or Outlook regarding the HELiOS workspace, login behaviour, set default links for HELiOS data and attribute assignments, has also been further adapted and revised with the update to Service Pack 1.

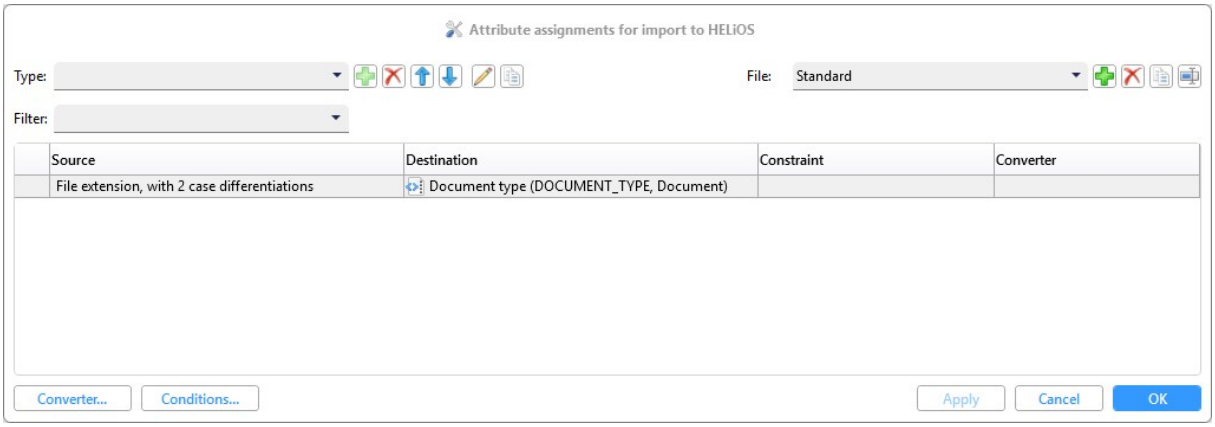


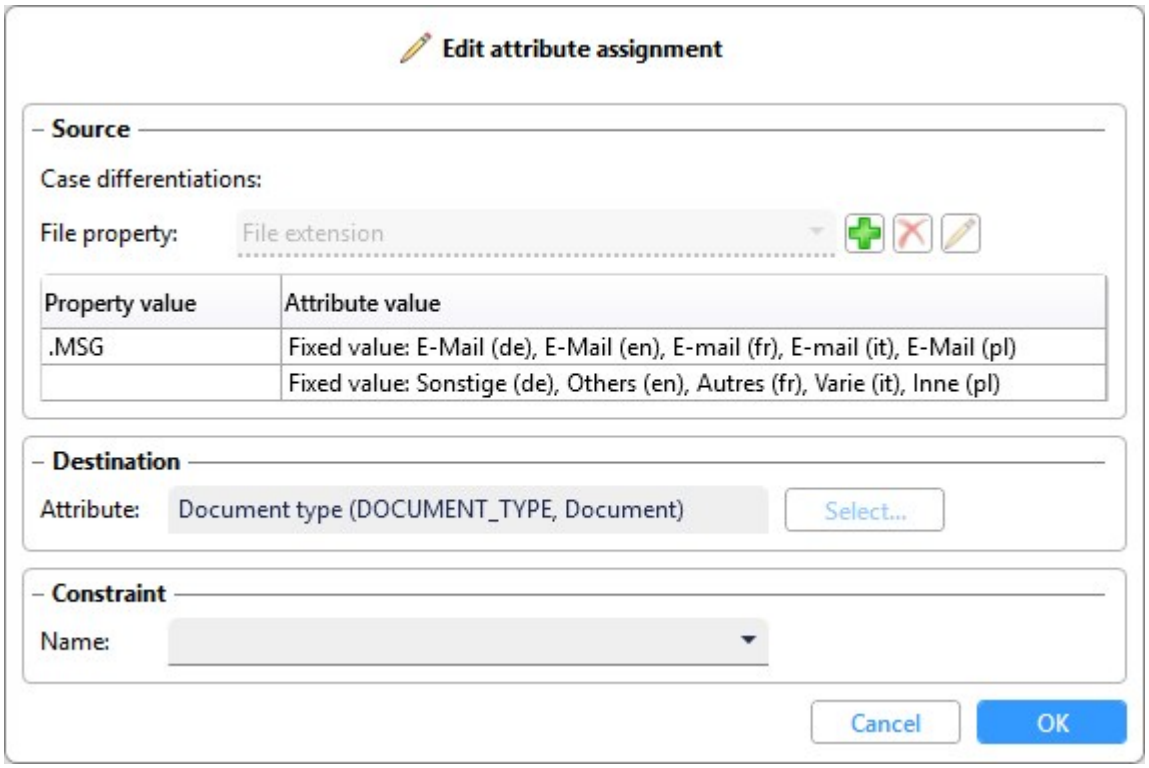
Attribute assignments

The attribute assignments for the transfer of e-mails and attached files to HELiOS have also been revised and improved.

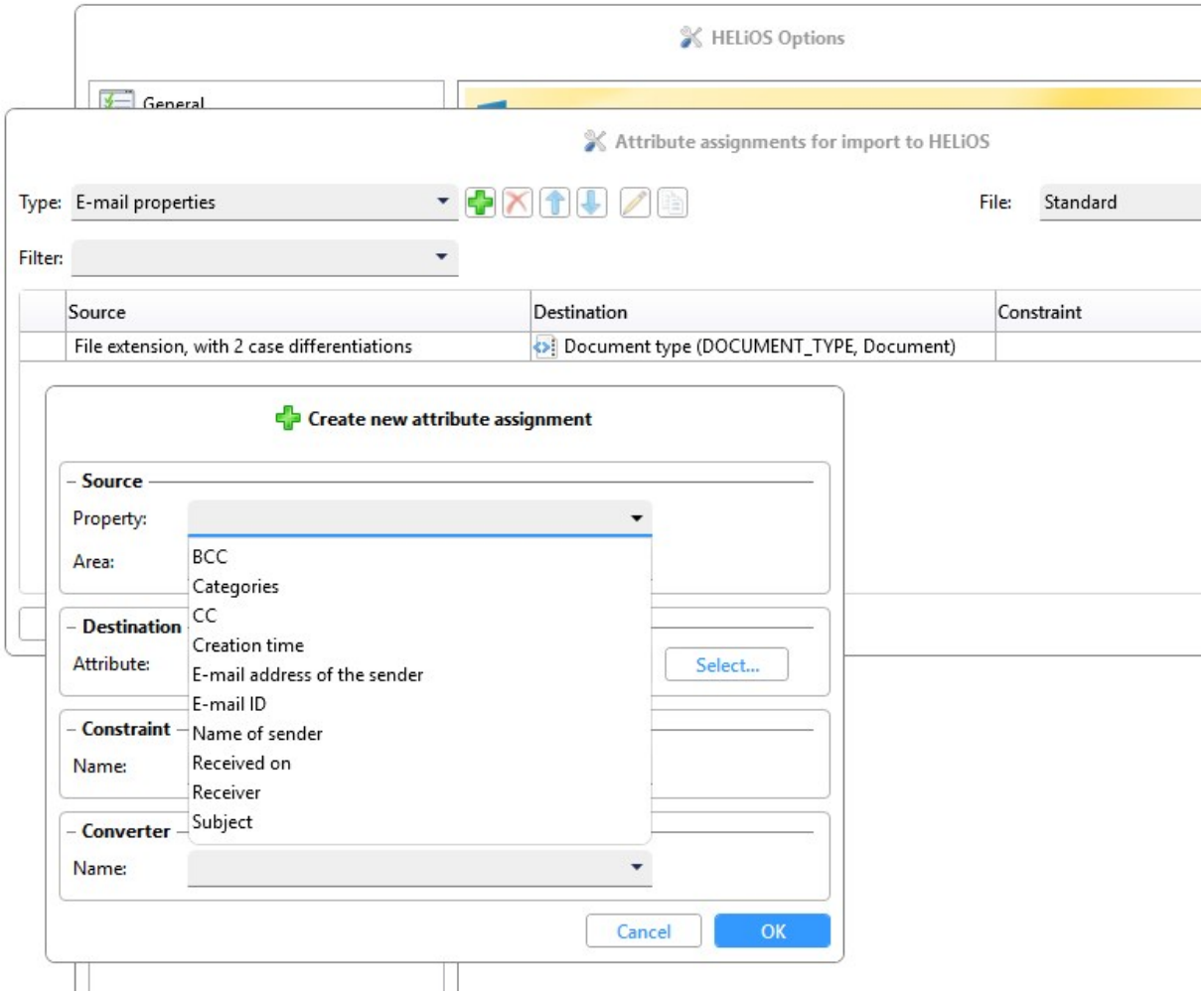
In the **Options** window of your HELiOS-/Outlook interface, you will find the submenu item **Attribute assignments**.

This opens an extended dialogue with a comprehensive range of attribute mapping functions, as you already know them from other areas of HELiOS.





E-mail properties can be mapped with article and document attributes.



HeliosCouplings

An adjustment of the HeliosCouplings tool indicates which installed Office versions may not be supported by HELiOS and subtracts them from the selection options.

Major Release

Important note on update installations: Conversion of system directories of HELiOS Workspaces

Beachten Sie bei der Update-Installation einer älteren Version auf HELiOS 2025 (Version 30.0.0) oder höher, dass sich die Verzeichnisstruktur ändert.

 Since an automated migration is not possible, all users must check in all data and empty their workspaces before an update installation in order to avoid data loss.

In previous versions, the workspaces were located under %localappdata%.

This meant that different workspaces could be located on one system.

To counteract this, the workspaces will be moved to the %programdata% directory with the update to HELiOS 2025.

Checked-out files are then stored at **%programdata%\ISD Software und Systeme\HELiOS Workspace\(...)***

(*plus Location ID and User ID).

The version-dependent workspace databases are stored at **programdata%\ISD Software und Systeme\HELiOS <Version>\Location-ID**.

HiCAD

UNLIMITED CAD PERFORMANCE **DEVELOPED BY ISD**

Your local contact

We attach great importance to the direct contact with our customers and partners, because only a lively dialogue and constant exchange with practice ensure application- oriented software development.

Feel free to contact us! Whether at our headquarter in Dortmund or at one of our branches and subsidiaries in your vicinity - we will be happy to answer all your questions about our products and services. We are looking forward to hearing from you!

Headquarter Dortmund

ISD Software und Systeme GmbH

Hauert 4

D-44227 Dortmund

Phone +49 231 9793-0

info@isdgroup.com

ISD locations worldwide at **www.isdgroup.com**

HiCAD 30.2

This document is generated automatically.