HiCAD 3D

HiCAD 3D 2027 - What's New?

All news at a glance

Version 2027

Date of issue: 23/11/2026

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Discontinuations

Discontinuation of the Plant Engineering/Layout Planning module

From HiCAD 3D 2027 (Version 32.0) onwards, the **Plant Engineering/Layout Planning** Extension Module will no longer be supported. Customers with a maintenance contract for the Plant Engineering/Layout Planning module will automatically receive the **Pipeline Planning** Extension Module when they update. Additional features from the Plant Engineering/Layout Planning module will be incorporated into Pipeline Planning.

Discontinuation of Windows® 10

Microsoft® has discontinued support for the Windows® 10 operating system in October 2025. For compatibility reasons, HiCAD 2025 SP2 and HELiOS 2025 SP2 were the last versions of our CAD or PDM system to support Windows© 10. HiCAD 2026 and HELiOS 2026 no longer run under Windows© 10.

Discontinuation of various HiCAD modules

Starting with HiCAD 2026 (Version 31.0), the following HiCAD modules will no longer be supported:

- **HiCAD classic** and **HiCAD creator**

  The extension modules that previously required HiCAD classic or HiCAD creator now require HiCAD solution. Customers with a maintenance contract for one of the two modules will automatically receive HiCAD solution when they update.

- **HiCAD Sheet Metal suite**

  Customers with a maintenance contract for HiCAD Sheet Metal suite will automatically receive Sheet Metal suite premium when they update.

- **Metal Engineering suite classic**

  Customers with a maintenance contract for HiCAD Metal Engineering suite classic will automatically receive HiCAD solution when they update.

- **Report Manager professional**

  The additional functionality of the Report Manager professional has been integrated into the Report Manager.

From HiCAD 2026 onwards, the following change to the module structure applies:

- **Steel Engineering suite premium**

  The extension module Sheet Metal professional has been integrated into the Steel Engineering suite premium.

Discontinuation of Steel Engineering grid

From HiCAD 3D 2027 onwards, the **Steel Engineering grid** function will no longer be available.

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 Microsoft SQL Server 2012 and 2014

Microsoft® discontinued support for **SQL Server 2012** in July 2022 and no longer provides security updates. As a result, **SQL Server 2012** will no longer be supported from **HELiOS 2025** onwards.

Also, Microsoft® discontinued support for SQL Server 2014 in July 2024. Therefore, SQL Server 2014 will no longer be supported from HELiOS 2026 onwards.

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

Major Release

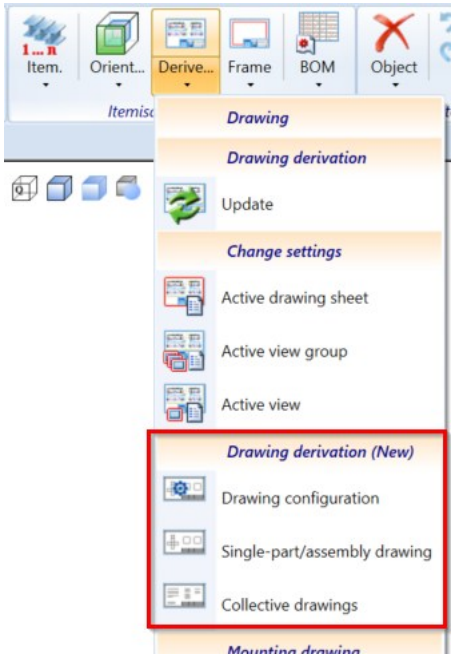
HELiOS becomes HiCAD PDM

From version 2027 onwards, HiCAD is the collective name for our product family. The CAD system is called **HiCAD 3D**. The PDM system, previously known as HELiOS, is now called **HiCAD PDM**. Both products remain unchanged; what has changed are the names and the places where these names appear: program files, the Start menu, directories and settings. You can find further information on HiCAD 3D here and on HiCAD PDM here.

Drawing derivation

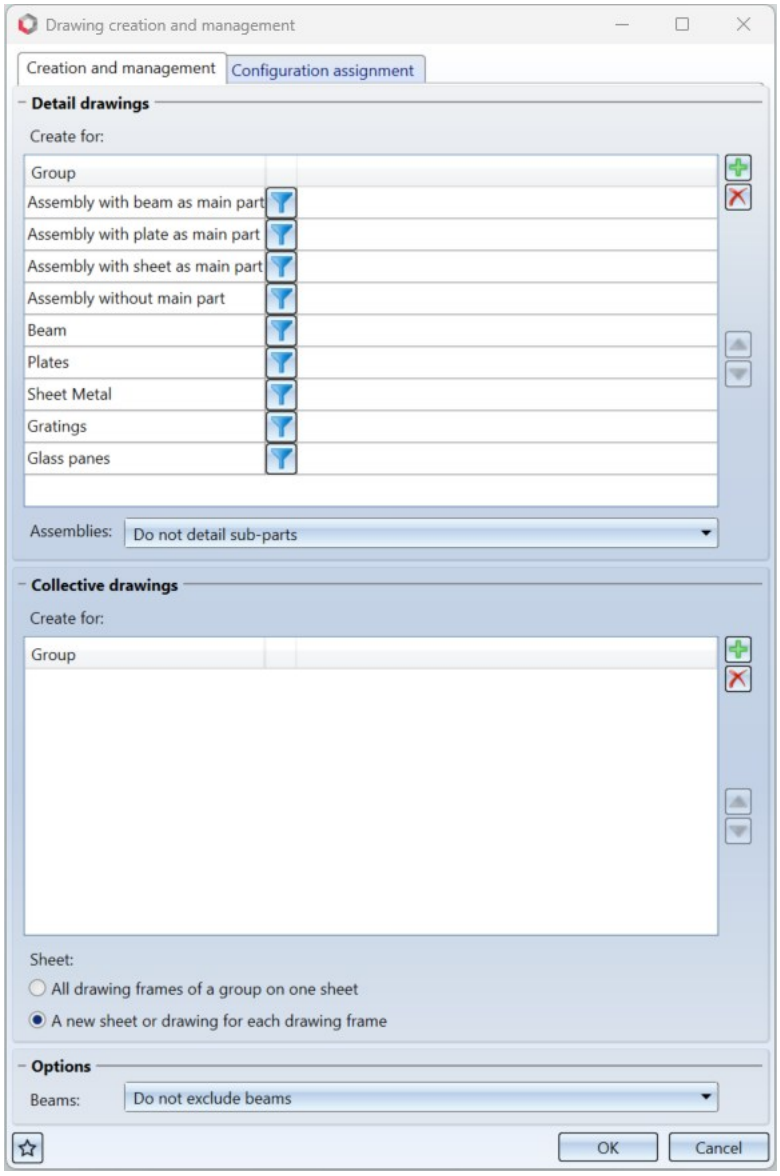
Drawing derivation (New)

The drawing derivation function has been revised and expanded to include three new features: **Drawing configuration**, **Part/Assembly drawing** and **Collective drawings**. You can find these in the ribbon at **Drawing > Itemisation/Detailing > Derive... > Drawing derivation (New)**. These functions allow you to derive drawings from the design, whereby the default settings do not need to be set individually for each derivation but are instead configured centrally once.



Drawing configuration

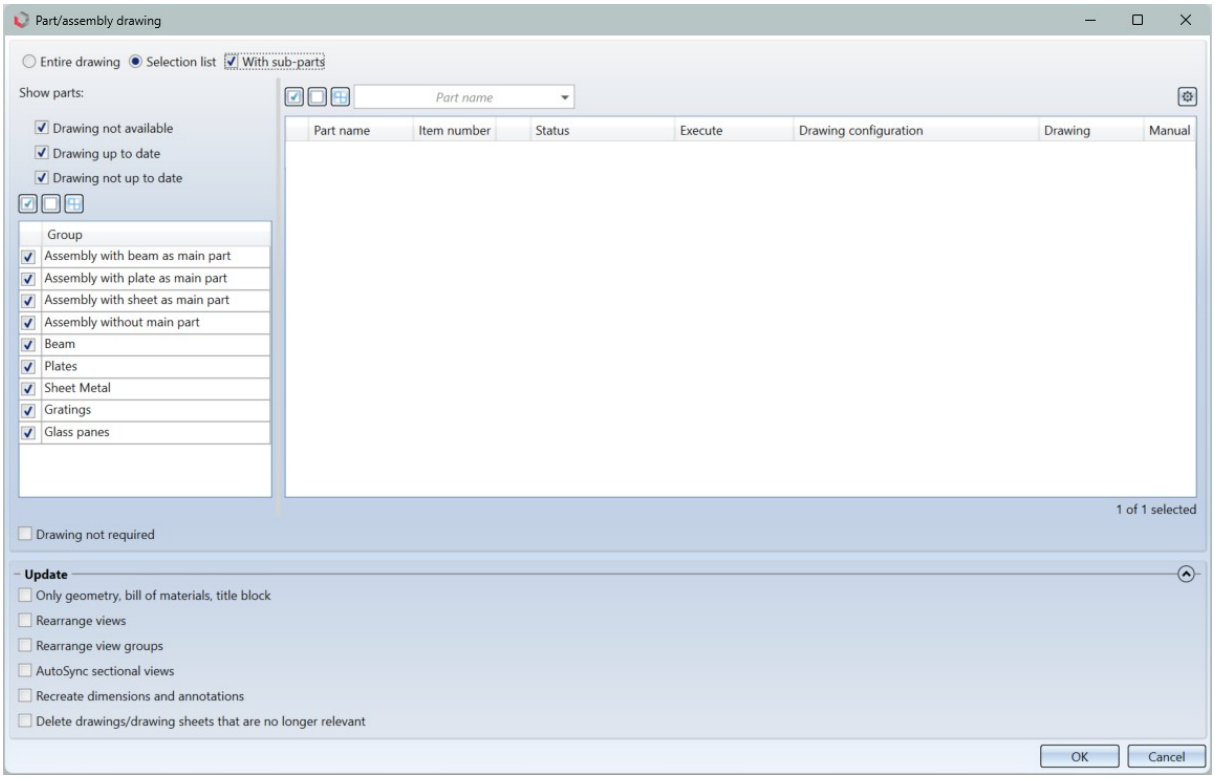
The **Drawing configuration** function allows you to define the settings for drawing derivation centrally for the current drawing. The associated dialogue window is called **Drawing creation and management**. On the **Creation and management** tab, you specify for which groups individual part and collective drawings are to be generated. You can load the default settings via Favourites or define the groups yourself.



On the **Configuration assignment** tab, you will find predefined groups containing individual parts, each of which is assigned to a drawing configuration. You can also add your own entries or folders and assign drawing configurations to them.

Part/assembly drawings and Collective drawings

The **Part/assembly drawing** and **Collective drawing** functions use a similarly structured dialogue window. On the left side, you select the groups; the part list is in the middle, and the update options are below it. For collective drawings, the **Rearrange view groups** option is also available.



Once you have selected the groups, the part list displays every part and assembly covered by the group definitions in the **Drawing configuration** function. In the part list, you can add or remove individual parts and assemblies. Each row is filled in automatically, but the action shown in the **Execute** column can be changed subsequently. The **Drawing configuration** column displays the configuration specified in the **Drawing configuration** function on the **Configuration assignment** tab. You can also adjust these settings at a later stage. If sheets already exist, you can use the **Drawing** column to adjust the sheet so that the drawing derivations of several parts are placed on a single sheet. When creating a drawing for the first time or replacing an existing one, you can also customise the drawing configuration entirely manually by clicking the icon; this opens the **Drawing configuration – "Name of the preset drawing configuration"** dialogue window.

Hinweis:

In the current version, the **Drawing derivation (New)** dialogue window exclusively supports the creation of production drawings within the existing drawing. The creation of drawings in external designs is not yet part of the new dialogue box. For this reason, the new functions for creating production drawings within the existing drawing are currently available as an option in the relevant sub-menu. An extension of the **Drawing derivation (New)** dialogue box to include functions for creating drawings in external designs is planned for one of the upcoming versions.

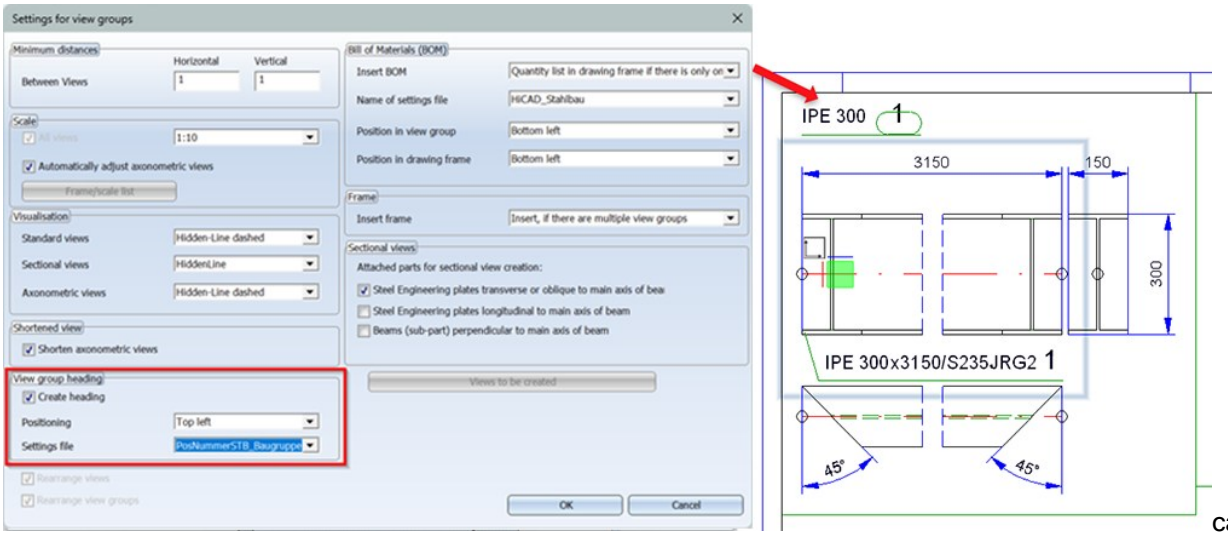
Headings for view groups in drawing derivation

View group headings (aka captions) can now be generated using the new **Part/assembly drawing** and **Collective drawings** functions, as well as the existing **Drawing derivation** function.

To create the captions using the new functions, use the Manual Adjustment of Drawing Configuration. There, on the **View groups** tab in the **View group caption** section, you will find the required settings.

If you are using the existing **Drawing derivation** function, you will find the relevant settings in the **Settings for view groups** dialogue window in the new **View group caption** section:

- **Create heading**: Use this checkbox to specify whether the view group should have a caption.
- **Positioning**: Here you can specify the position of the caption in relation to the view group, e.g. top left.
- **Settings file**: Here you select the ftd file that defines the content and formatting of the heading.

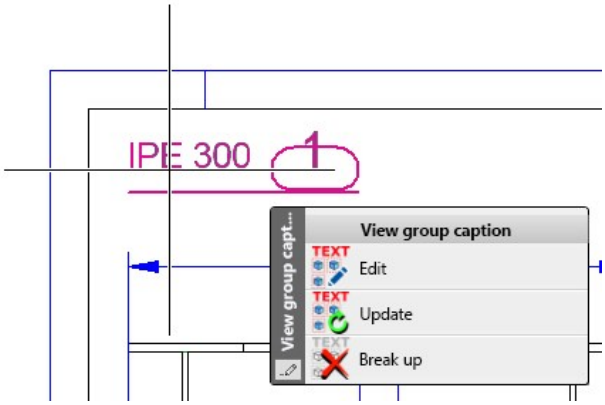


caption

In the Configuration Editor, the relevant parameters are available as default settings at **Automatic drawing derivation > Production drawing > Usage-dependent >** *name* **> View group** in the **View group captions** section. Here, name refers to the name of the respective usage.

A generated heading can be edited in the drawing at a later stage:

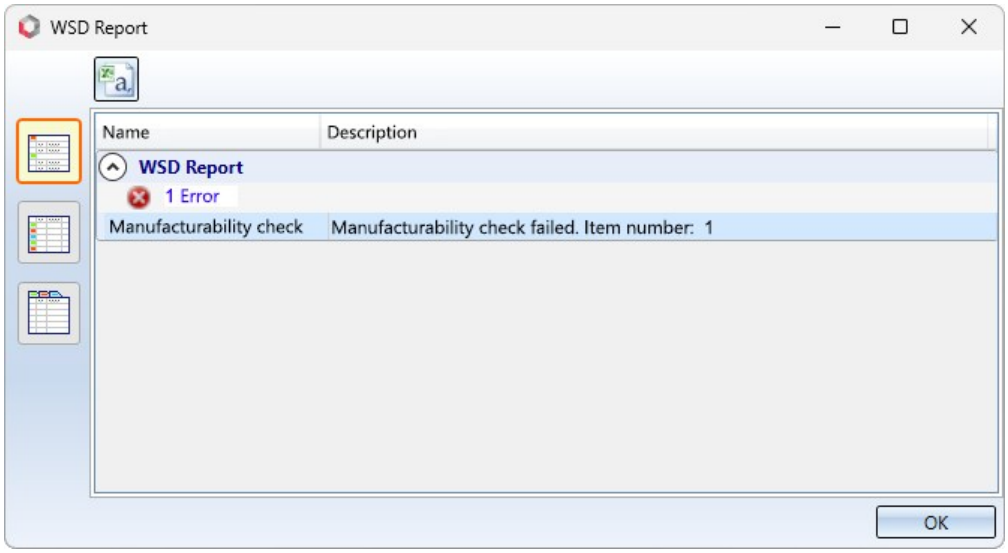
- Double-click on the heading to edit it.
- The heading can be moved around the sheet using drag and drop. The positioning mode then changes to **Manual**.
- The context menu for the heading contains the functions **Edit**, **Update** and **Break up**. Select **Break up** to delete the heading. The view group remains intact.



Manufacturability check during automatic drawing derivation

In the Configuration Editor, at **Sheet Metal > Manufacturability check**, the new parameter **Perform a manufacturability check when creating automatic drawing derivations** is available. It controls whether a manufacturability check is performed for the sheet metal parts contained in workshop drawings during automatic derivation (creation and updating).

If the parameter is activated, each sheet metal part in the part list is checked for manufacturability based on the criteria set at **Sheet Metal > Manufacturability check**. If a sheet is identified as unmanufacturable, a corresponding message appears in the drawing derivation report window.



The check is performed for all selected sheet metal parts–regardless of whether a (new) view or development pattern is actually generated for the respective sheet metal part. This ensures that even an unchanged sheet metal part is checked during an update; otherwise, it would remain unchecked and would be incorrectly identified as manufacturable.

If the parameter is deactivated (default setting), no manufacturability check takes place during drawing derivation.

New annotation template for bores with the “Site” version

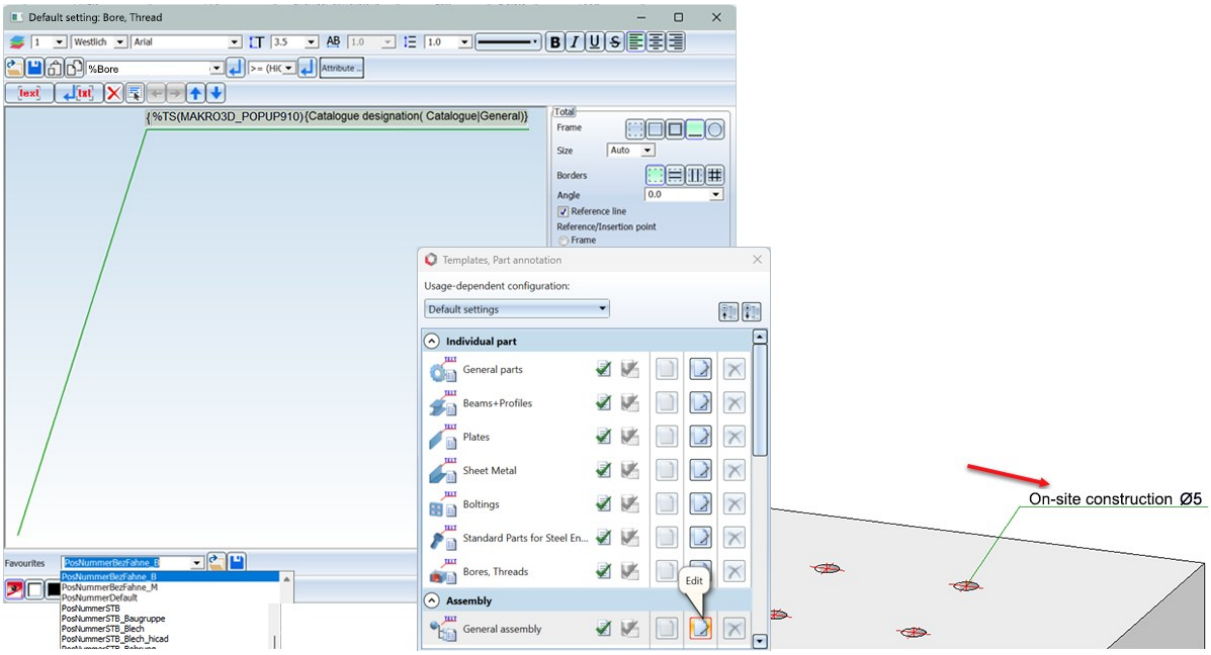
For standard processings with the “Site” version, the new predefined annotation template **PosNummerBezFahne_B.ftd** is now available; this template automatically retrieves the production type and prepends it to the annotation.

The template can be loaded in the Configuration Editor at **Drawing > Annotations > Annotation template** for the parameter **Annotation template of the bore type for site** by entering z:**PosNummerBezFahne_B.ftd** there.

You can then select the template in the **Templates, Part annotation** dialogue window:

To do this, click **Edit** next to the entry **Bore, Thread** and, in the editor that appears, select the annotation template **z:PosNummerBezFahne_B.ftd** in the **Favourites** field.

When you subsequently annotate standard processings, e.g., using the **Standard designation** function, the annotation will be prefixed with the remark **On-site construction**.


Default setting: Bore, Thread
%Bore
{%TS(MAKRO3D_POPUP910){Catalogue designation(Catalogue|General)}
Total
Frame
Size
Auto
Borders
Angle
0.0
Reference line
Reference/Insertion point
Frame
Templates, Part annotation
Usage-dependent configuration:
Default settings
Individual part
General parts
Beams+Profiles
Plates
Sheet Metal
Boltings
Standard Parts for Steel En...
Bores, Threads
Edit
Assembly
General assembly
Favourites
PosNummerBezFahne_B
PosNummerBezFahne_M
PosNummerDefault
PosNummerSTB
PosNummerSTB_Baugruppe
PosNummerSTB_Blech
PosNummerSTB_Blech_hicad
On-site construction Ø5

HiChat: New AI tool for searching through the Online Help

The **HiChat** Extension Module is now available.

In the docking window, you can ask questions about using HiCAD via chat. The questions are answered using AI based on the online help.

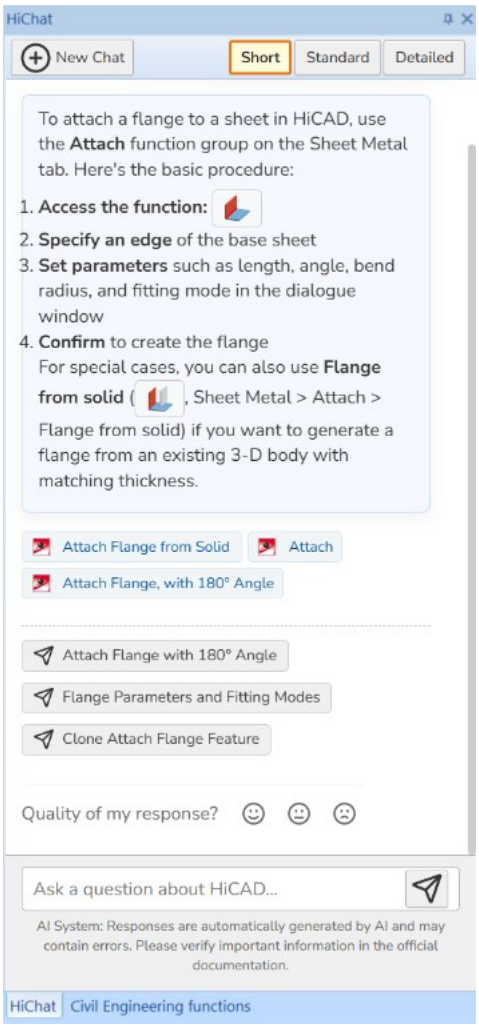
You can adjust the level of detail in the answer using the buttons in the upper-right corner of the window. The available options are **Short**, **Standard**, and **Detailed**.

Clicking the  button clears the chat history. It is recommended to do this before changing topics.

If the generated answer refers to specific functions, you can access them directly via the chat by clicking the function icon.

At the end of the response, source references appear as links to the corresponding pages in the online help that were used to answer the question, along with suggestions for follow-up questions on the topic.

You also have the option to rate the response on a three-point scale (**Good**, **Medium**, **Unusable**) and submit a comment on the response.



HiChat is available in German, English, Italian, and French. In the Polish version, **HiChat** is in English.

To use the chat, you need the corresponding HiCAD Extension Module, HiChat. For questions regarding HiCAD PDM, you also need the HiCAD PDM Extension Module starting with version 32.0.

To ensure that **HiChat** takes the Drawing Management module into account when answering questions, steel engineering drawing management must be activated in the Configuration Editor. To access the P&ID (RPA), P&ID, Plant Engineering, and Variant Editor modules, the corresponding licenses for piping design must be available.

Hinweis:

- AI can make mistakes. You should verify important information in the official documentation.
- HiChat is classified as a low-risk system under the EU AI Act. For more information on how HiChat works, data processing, and related server operations, see the How to Use the Help chapter in **HiChat**.

Redesign of the Properties window in the ICN

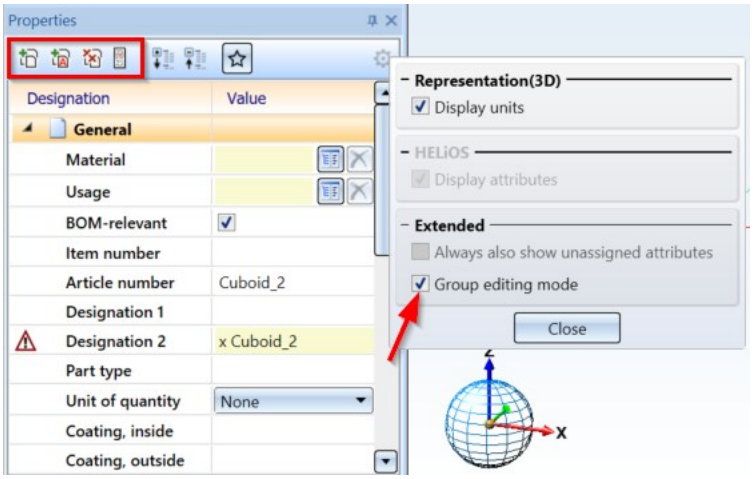
The **Properties** window in the ICN, which displays the properties and attributes of the currently active part(s), has been redesigned. In addition to a modernized design, part properties are now organized into groups. This improves clarity, making it easier to find important information quickly.

Another new feature is the group editing mode, which allows you to configure the view according to your preferences.

When editing mode is enabled, the following options are available:

- create new groups,
- combine existing part properties and attributes into a new group,
- automatically generate new groups,
- set the visibility of attributes for each individual part type,
- move properties and entire groups via drag & drop.

You can activate the **Group editing mode** using the **Settings** button in the **Properties** window toolbar. Once you have finished editing the view, you can save your settings as a favourite via .

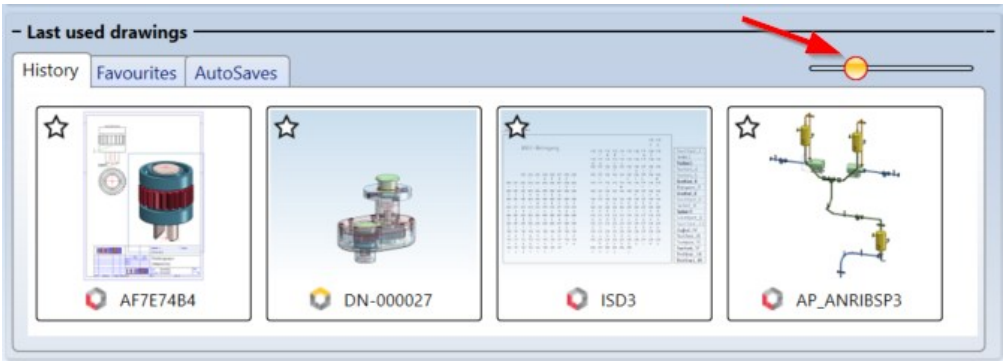


Please note that HiCAD PDM attributes are only displayed when you are working with PDM. Otherwise, these attributes are hidden in the **Properties** window.

Start Centre

Adjustable preview image size

The **Last used drawings** section in the Start Centre dialogue now includes a slider that allows you to adjust the size of the preview images. Alternatively, you can adjust the size by holding down the CTRL key and using the mouse wheel while the cursor is over the preview images.



Restoring interim backups

HiCAD 3D automatically creates interim backups of your drawings whilst you work. From now on, the Start Centre will offer these backups for loading on the new **AutoSaves** tab the next time you start the program, whether following a programme crash or after a normal exit.

For each backup, the name of the drawing is displayed along with its original save location, the date and time of the backup, and a preview image of the drawing's state at that time. Several older versions are retained for each drawing, depending on the settings in the Configuration Editor (**System settings > Data save > Number of data saves**) .

Clicking on the preview image opens the selected drawing. The drawing appears under its original name. If the original drawing is already open, or if several versions of the same drawing are being loaded, the name is automatically appended, e.g. *Hall(2)*.

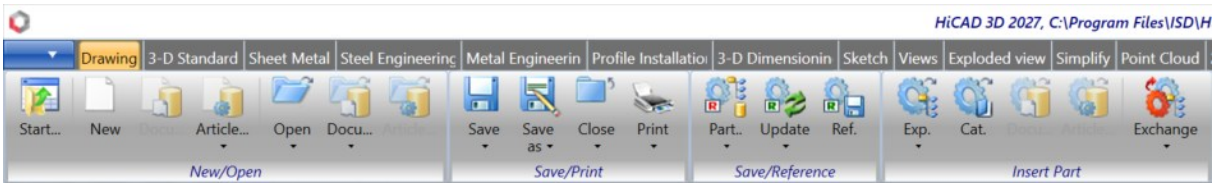
The first time you save a restored drawing, HiCAD 3D always asks for the name and save location. The original name and directory are suggested. This ensures that the original file is never overwritten without your knowledge. You decide whether to replace it or save the version under a new name.

Backups of crashed sessions are retained until HiCAD 3D is closed properly, even across multiple crashes.

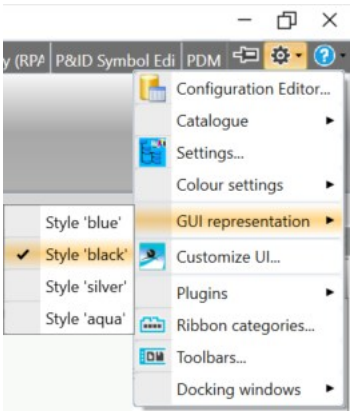
The previous option "Keep drawing-related data in cache when switching drawings" in the Configuration Editor at **System settings > Miscellaneous** is now called **Automatically create a backup when switching drawings** and can be found under **System settings > Data save**. If this option is enabled, a complete, restorable backup is created every time you switch drawings. This may cause the switching process to take slightly longer. If it is disabled by default, the backups created during the switch are not suitable for loading and will not be offered. Existing settings are automatically retained upon updating.

High-contrast colour scheme in the Ribbon’s dark design

The Ribbon’s color scheme has been revised in the dark design. The appearance has been adjusted to provide a darker, higher-contrast look that more closely resembles the previous design.



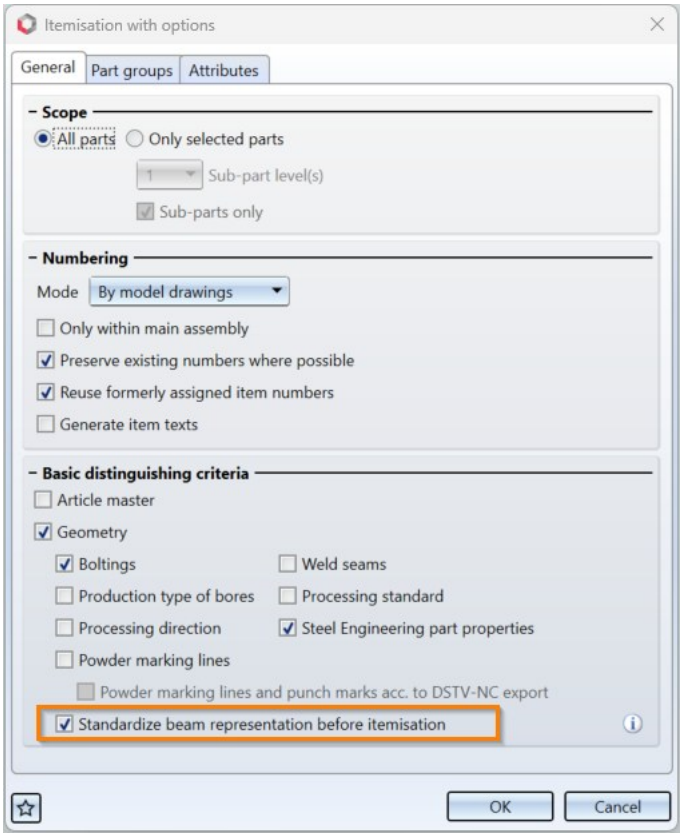
You can select the design via **Settings** > **GUI representation > Style 'Black'**.



Itemisation - Standardize beam representation

In the **Itemisation** options, on the **General** tab inthe **Basic distinguishing criteria** section, the new checkbox **Standardize beam representation before itemization** is available.

If the checkbox is selected, profiles displayed differently are converted to a uniform representation before itemisation. This ensures that identical beams receive the same item number– regardless of their representation. If the checkbox is deselected, beams displayed differently always receive different item numbers. The checkbox is selected by default in the provided favorites and for existing data.



Basic module HiCAD 3D Solution

Starting with HiCAD 3D 2027, the Basic module**HiCAD 3D Solution** includes the **Save variant for element installation**  function.

Dynamic zoom: New default settings and permanent storage

The default settings for the dynamic zoom have been adjusted, and the way settings are saved has been improved further.

You can still set the **zoom parameters**—magnification factor and aspect ratio—using the transparent toolbar (right-click on **View all > Zoom parameters**). These settings are now saved permanently on a per-user basis and are retained even after restarting HiCAD 3D. These parameters are no longer available in the Configuration Editor.

In addition, the default settings for the zoom have been changed:

- Magnification factor: 20
- Aspect ratio: 0.75

Filling in title blocks: Attributes of the part displayed on drawing sheet

When automatically filling in the title block, part attributes could previously only be taken from the highest BOM-relevant part in the entire drawing. You can now also evaluate the highest BOM-relevant part that is shown on the drawing sheet containing the title block. This is particularly useful for multi-sheet layouts: each title block refers to the sheet on which it is located.

The new data source is available in both configuration methods:

- **Complete title block (from DB):** In the title block text, use the new source V to select the part displayed on the sheet, e.g. **@*@20,V.$01@*@**. Source **K** evaluates the highest BOM-relevant part of the entire drawing, as befores.
- **Complete title block (from attributes):** In the files *SchrFe_ausAttr.dat* (general call) and *wsd_titleblock.dat* (derived drawings), the first column contains the new identifier **3** for the highest BOM-relevant part displayed on the sheet. The identifier **0** is reserved for the highest BOM-relevant part of the entire drawing.

The reference sheet is the drawing sheet on which the title block is located. If no sheet can be assigned to the title block, the active drawing sheet is used. If there are several parts of equal rank that are BOM-relevant at the highest level, or if no part BOM-relevant is displayed, the field remains empty.

Further information can be found in the topic Variables/Attributes in the Title Block.

3-D

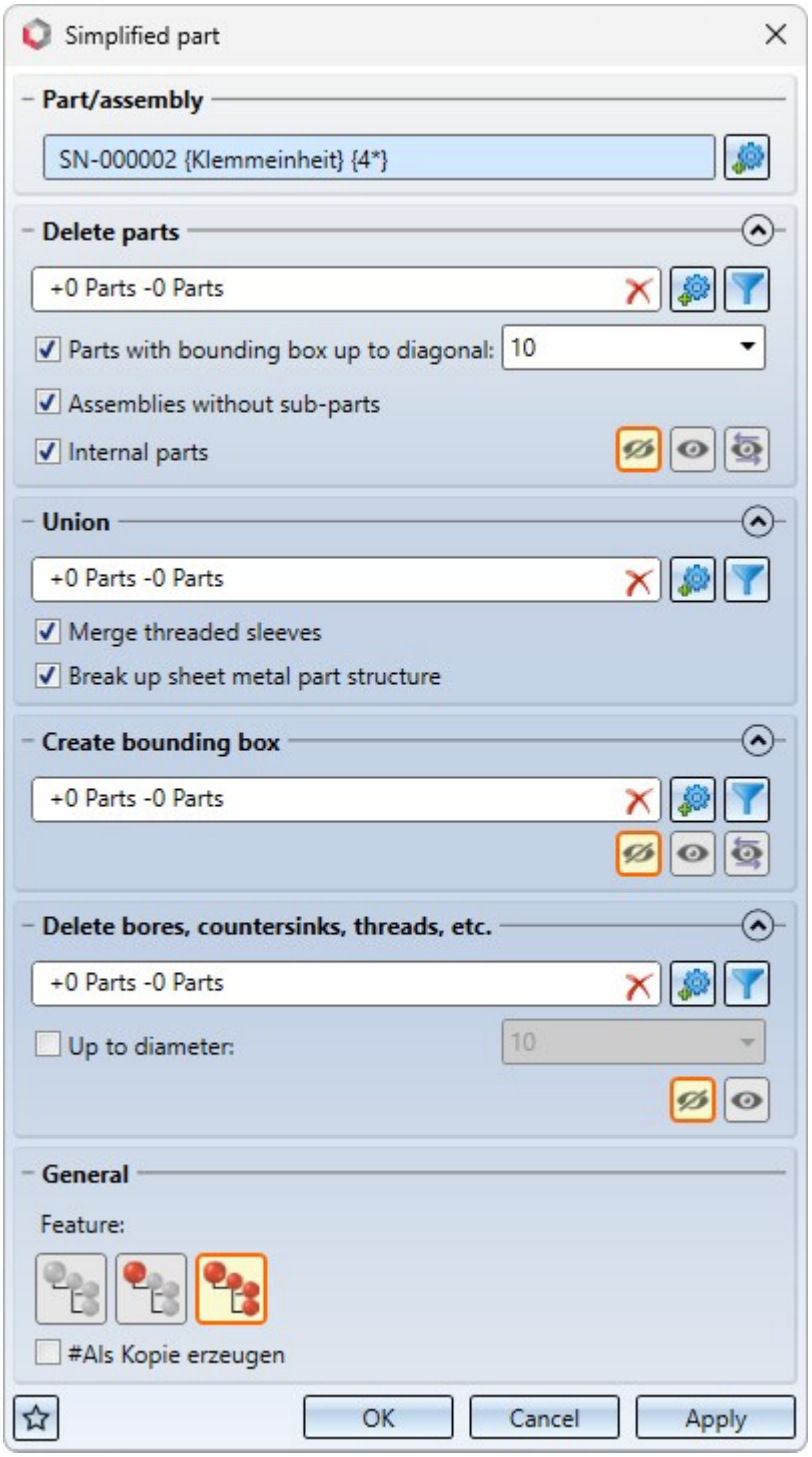
Major Release

New function: Simplified part

With the new **Simplified part**  function, you can replace a part or assembly, along with its sub-parts, with a simplified version. This is useful, for example, for improving performance when working with large drawings or for reducing the amount of data during export. The part or assembly must be referenced for this to work.

Simplification may involve deleting or merging individual sub-parts from a main part, creating a bounding box, or deleting standard processing steps. To do this, work through the dialogue window step by step to configure the desired simplification. You can select the individual parts to be simplified directly by clicking on them in the graphics or via a part filter, and exclude individual parts that are not to be simplified. You can then save your settings as favourites.

Once the function has been successfully executed, a new entry called **Simplified part** is created in the feature log, which can be used to revise the simplification at a later stage.



You can find the new function under **3-D Standard** > **New** >**Dependent part** > **Simplified part** or under **Simplify** > **Create** > **Simplified part**. To use this function, you will need the **Simplify** extension module.

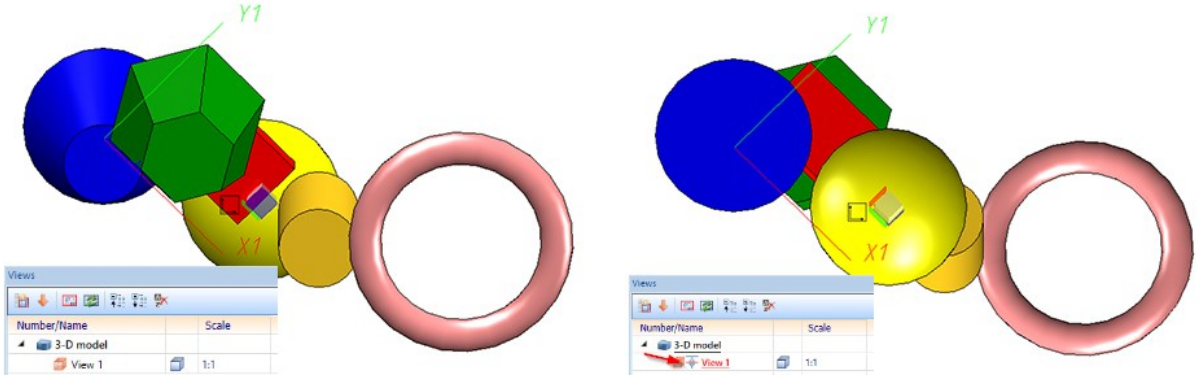
Reflected ceiling plan

Under **Views** > **Projections** > **Standard AXO** > **Reflected ceiling plan**, you can use the new **Activate** and **Deactivate** functions to switch the active view to a reflected ceiling plan (RCP, also known as an

engineering view) and switch back again.

The orientation of the view remains unchanged; a top view therefore remains congruent with the floor plan. However, the occlusion is calculated as if you were looking at the drawing from the opposite side. This means that the element closest to the ceiling is visible. The function is available for all views with orthographic projection and in all display modes.

Identifying elements, dimensioning and annotations, and dynamic rotation all work as usual in the mirrored ceiling view. You can also derive section and detail views from it.



(1) Original view, (2) Reflected ceiling plan activated

Dimensioning

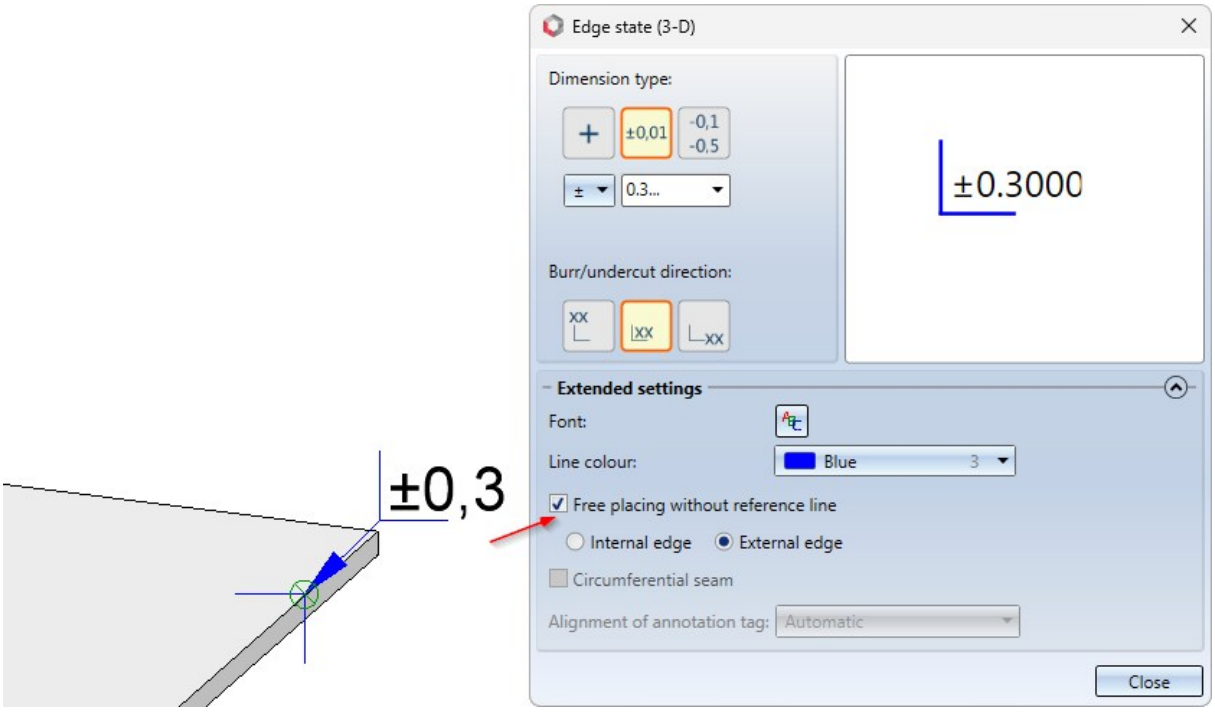
Edge State without annotation line

The **Edge state** 

function under **3-D Dimensioning + Text** > **Symbols** > **Edge state** has been enhanced to allow edge states to be marked without an annotation line. Previously, an annotation line was mandatory for every annotation.

If the annotation is created without an annotation line, you must also specify whether the information refers to an internal or external edge.

In addition, the dialogue has been revised so that edge states are now created while the dialogue is open.

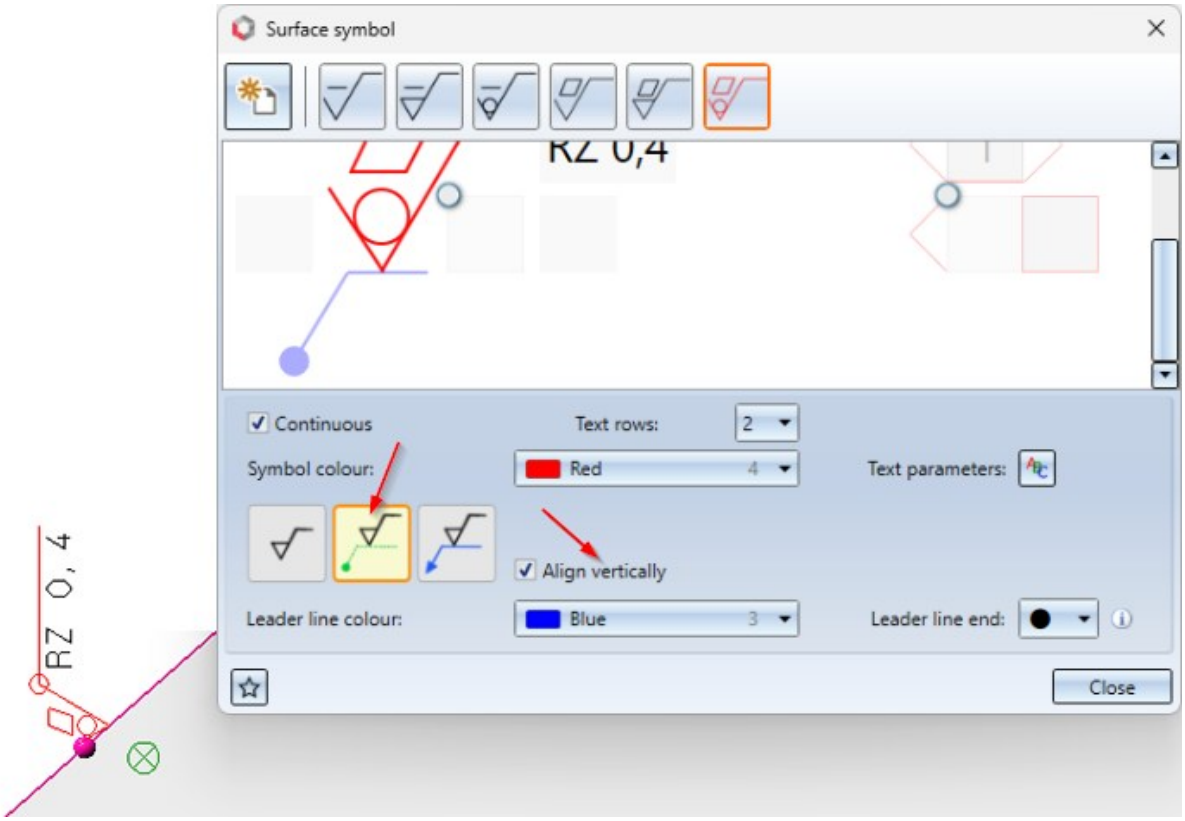


Surface symbols: Improvements to the dialogue window

The **Surface finish (3-D)** function under **3-D Dimensioning + Text** > **Symbols** > **Surf...** has been revised so that you can now create surface symbols whilst the dialogue is open, in the same way as for **Form/Positional Tolerances**.

Once you have set the desired options in the dialogue window, switch to the drawing and select the reference point. The surface symbol is displayed as a preview at the cursor and can be positioned with a further click. If you change the settings in the dialogue window while doing this, these changes are applied immediately to the preview. The dialogue window remains open throughout, allowing you to define and place further surface symbols straight away.

In addition, a new option has been introduced for the **Preferred without leader line** mode to align surface symbols vertically as well.



Dimension assignment

From now on, the **Dimension assignment** is set globally for all drawings in the Configuration Editor under **System settings** > **Annotations** > **Dimensioning, 3-D** > **Interactive dimensions**. The **Dimension assignment** option in the function dialogue under **3-D Dimensioning+Text** > **Tools** > **Dimensions** > **System** has been removed.

Previously, it was possible to specify which parts a dimension was assigned to within the drawing itself using the **Dimensions** function on the **System** tab. The setting in the configuration served only as a default for new drawings.

Dimensions and annotations can be dropped on silhouettes

Previously, **Form/Positional tolerances**, **Surface symbols**, **Edge states** and weld seam symbols could only be dropped on the lateral surface or the axis of cylindrical parts. These annotation objects can now also be dropped on the silhouette of cylindrical surfaces:

To do this, in selection mode, use the RMB to open the element type selection menu and select the **Surface** filter. If you then move the mouse pointer over a cylindrical surface, its silhouette will be highlighted and can be selected as a reference point for the annotation.

This is still not possible for 3-D texts; such text can still only be dropped at points.

Dimensions in exploded views

In exploded views, it is now also possible to set dimensions using the functions under **3-D Dimensioning + Text**. Previously, all point options were disabled there, meaning that no dimensions could be set. It is also possible to position a processing plane in the view and reassign base points.

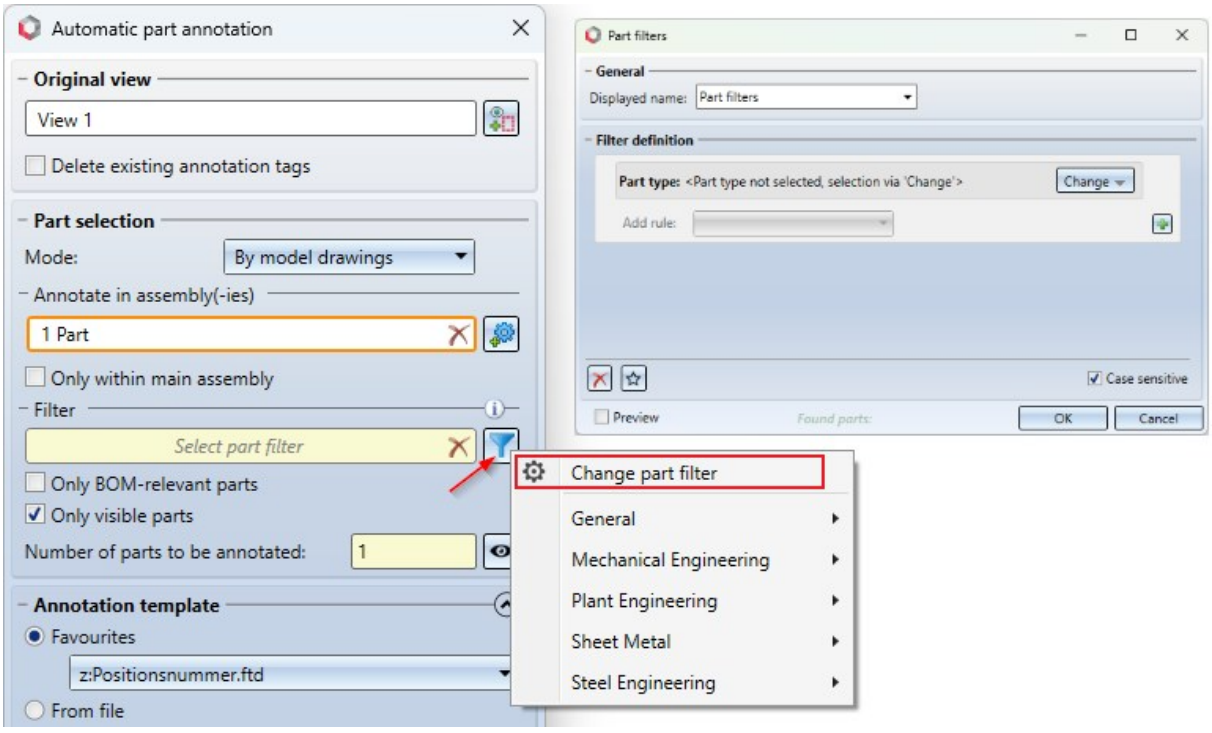
You can also set new annotation tags in exploded views without having to be in **Exploded view** mode.

Automatic part annotation revised

The **Automatic part annotation** 

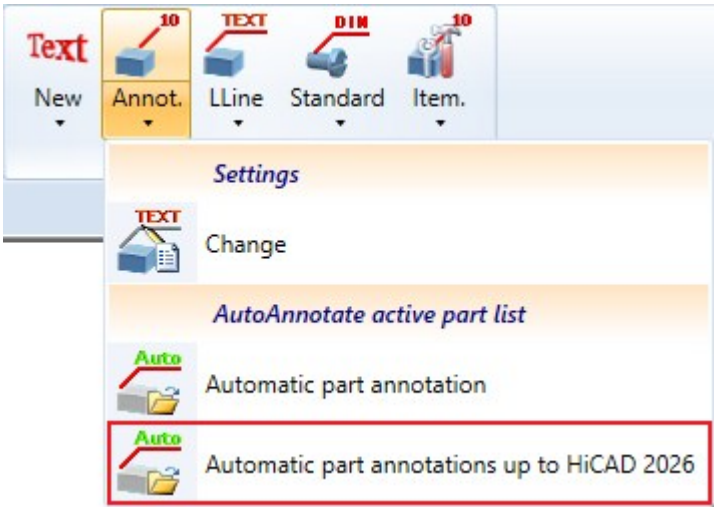
feature under **3-D Dimensioning + Text** > **Text** > **Annot.** has been enhanced in terms of annotation placement and the dialogue:

- Previously, only favourites could be loaded via the filters in the dialogue box. Individual filters first had to be created using the 'Part Filter' function in the transparent toolbar and could then be loaded via the configuration. In all areas of the dialogue window where you can select filters, the **Change part filter** option has been added alongside the existing favourites, allowing you to create manual filters directly via the dialogue window.



- **Annotation without a leader line:** This is now supported for all part types. You can use a part filter to specify which parts are annotated without a leader line. Assemblies are excluded from this; multi-part components such as sheets are annotated on their main body. The distance from the geometry is smaller than for annotations with a leader line.
- **Collision exclusion** In the new section, you can use a part filter or manual part selection to subtract parts from the collision check, e.g. a concrete slab on which parts are annotated.
- **New Apply current part list mode:** The list box in the **Part selection** area has been extended to include the **Part list** mode. The part list selected when the function is launched is applied. The part selector and filter are hidden in this mode.

- **Grouping of identical parts:** If the **Combine identical parts** checkbox is ticked, you can specify a **Maximum distance (2D)** aup to which identical parts are grouped together.
- **Previous function available again:** The previous function, which was replaced in HiCAD 3D 2027, is once again available under the name **Automatic part annotation up to HiCAD 2026** . You can find the function in the pull-down menu under **3-D Dimensioning + Text** > **Text** > **Annot.**, directly below the new part annotation option.



View properties

Colour tangential edges

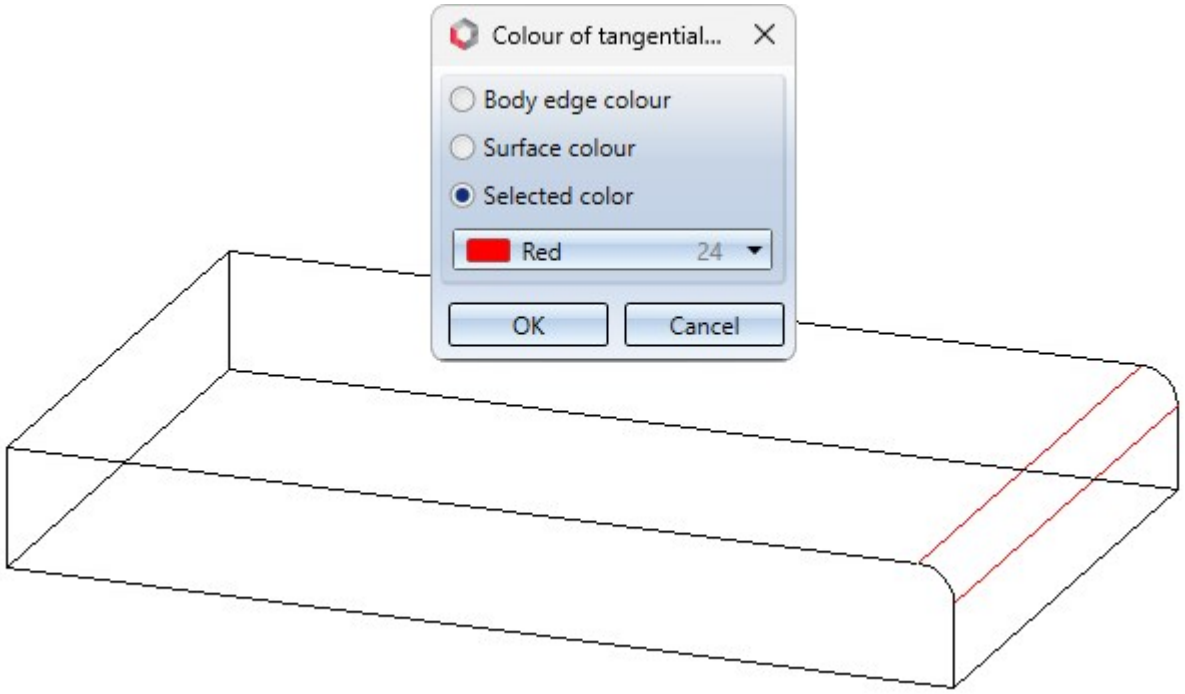
Tangential edges are edges where two surfaces merge smoothly into one another. In HiCAD 3D, they occur, for example, on fillets or at the transition between a flange and the bend zone of a sheet.

With the new **Colour of tangential edges**  function under **Views** > **Properties** > **Elements**, you can assign a fixed **Selected colour**, the **Surface colour** or the **Body edge colour** of the respective part to each view.

Previously, tangential edges could only be shown or hidden.

The function affects views in the **Glass Model**, **HiddenLine**, **HiddenLine dashed** and **Shaded with HiddenLine** display modes.

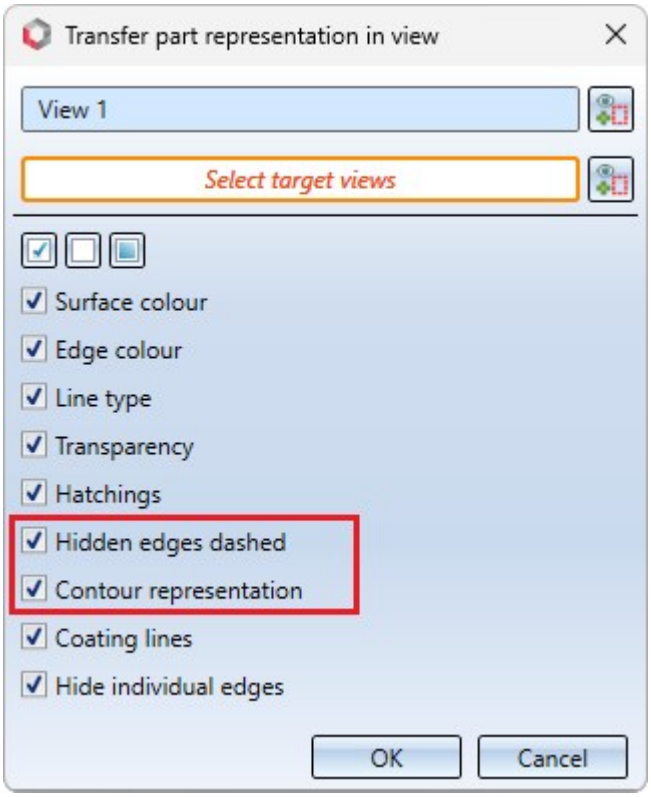
The setting selected in the function dialogue window is saved with the drawing and is applied when a view is copied. In the Configuration Editor, you can set a default under **Drawing** > **Views** > **Representation of special lines** > **Tangential edges**.



Apply hidden edges as dashed lines and contour display to other views

The **Transfer part representation in view** function in the context menu for 3D views has been expanded to include the options **Hidden edges dashed** and **Contour representation**.

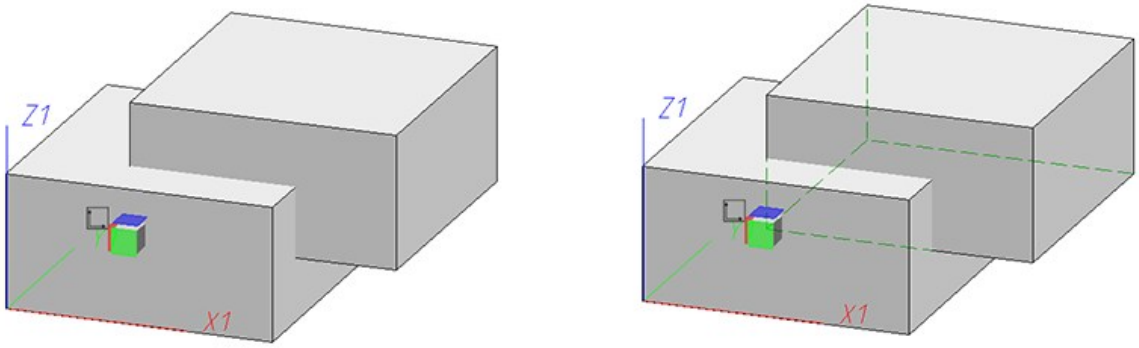
This function allows you to transfer a part's display settings, such as surface or edge colour, from the original view to other views.



Hidden edges shown as dashed lines in the ‘Shaded with HiddenLine’ view

Using the **Hidden edges dashed**  and **Hidden edges normal**  functions from the context menu of 3-D parts under **Properties** > **Properties in view**, you can selectively make individual parts ‘show through’, even though the **HiddenLine** display mode normally suppresses all hidden edges. This allows you, for example, to reveal the position of an assembled part in the drawing. The setting applies per part and per view; all other parts in the view remain unchanged.

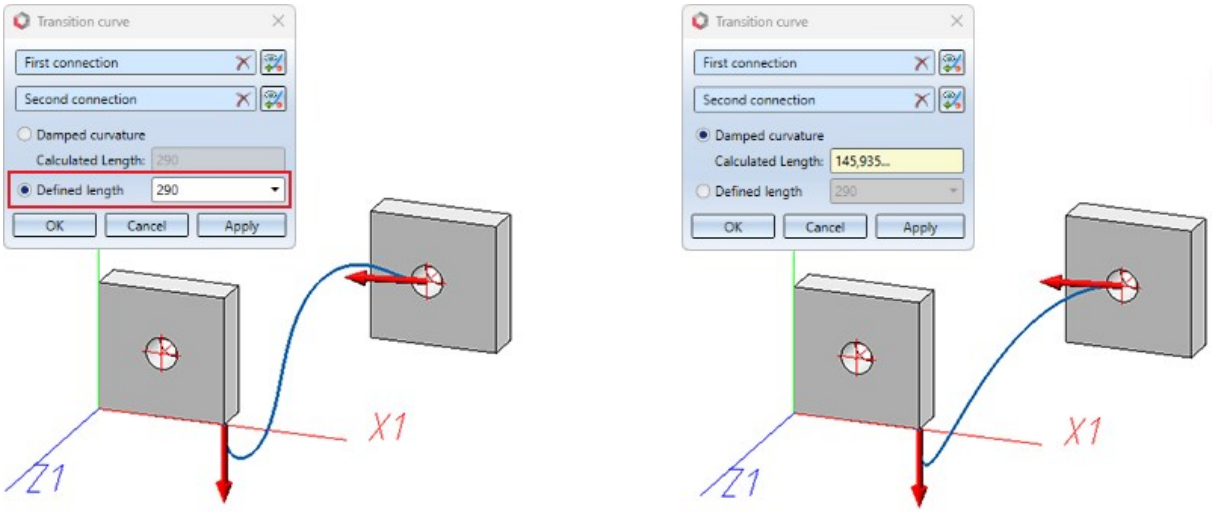
Previously, this special display only worked in the **HiddenLine** display mode. It now also works in the **Shaded with HiddenLine** display mode: the parts in question show their hidden edges as dashed lines superimposed on the shading.



(1) Hidden edges normal, (2) Hidden edges dashed

Transition curve with fixed length

The **Sketch** > **Draw** > **Freehand** > **Transition curve** function has been enhanced to allow the arc length of the curve to be specified as a fixed value. The existing tangency conditions are retained, while the length of the curve is added as a further condition in the HCM model.



To specify the length of the curve, select the **Defined length** option. For the **Damped curvature** option, which creates the curve with the maximum possible curvature between the two specified connections, the **Calculted length** of this curve will now also be displayed.

Extended line selection when defining auxiliary geometry

Previously, when using the **Sketch** > **Tools** > **Aux...** > **Set/Delete auxiliary geometry** function, lines could only be selected one at a time. You now have the same selection options available as when deleting lines, both when setting and when resetting the auxiliary geometry. Using the context menu, you can select **Connected edges**, **Tangentially connected lines or edges** or **Lines within a rectangle**. You can also cancel the selection and exit the function.

Exchange parts: Multiple selection

The functions **Exchange 3-D part**, **Exchange 3-D part, via document master** and **Exchange 3-D part, via article master** under **Drawing** > **Insert part**, as well as the corresponding functions in the context menu for 3-D parts, now also support multiple selection. This allows you to exchange multiple instances of a part in a single operation.

The following rules apply:

- The functions are only available if all selected parts are referenced and they are different instances of the same referenced part. Otherwise, a message will appear.
- Fitting points are queried only once for a reference part. The resulting transformation is applied to all other instances.

- If you launch the function via the context menu whilst multiple parts are selected, the part under the cursor is the reference part. If you launch the function via the ribbon, select the reference part yourself from the multiple selection.

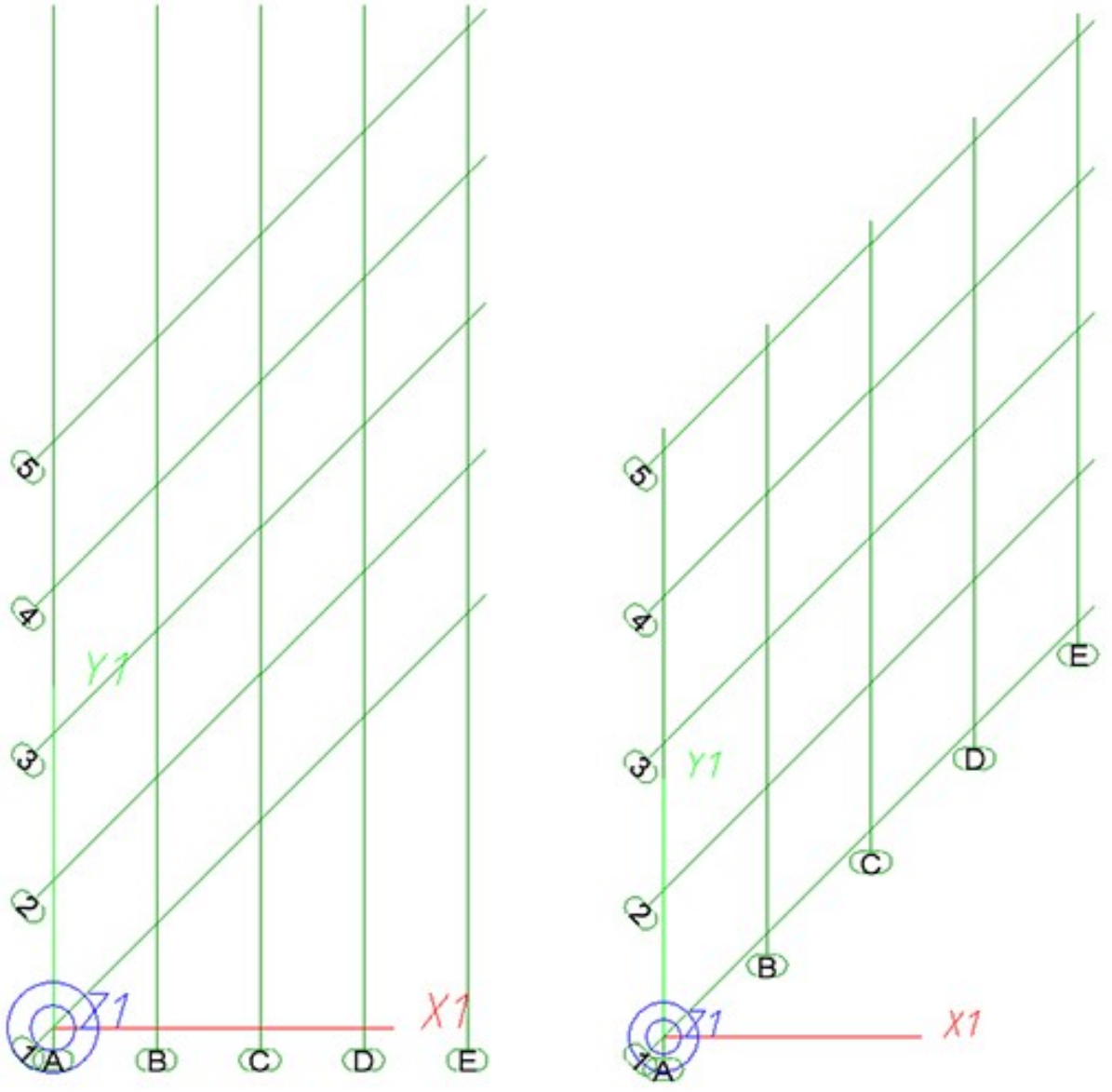
Planning grid

New method for extending grid annotations

The algorithm for **generating grid annotations** for a complete grid has been optimised.

Whereas previously a bounding box was placed around the geometry of the entire grid and the annotation lines were generated so that they extended beyond the bounding box by the same amount in all directions, with the new algorithm the lengths of the annotation lines are aligned with the last line in the orthogonal direction and are then lengthened evenly until all lines have left the geometry.

This means that the length of the lines is determined independently in all four directions. The new algorithm results in a more regular appearance, particularly with more complex grid shapes.

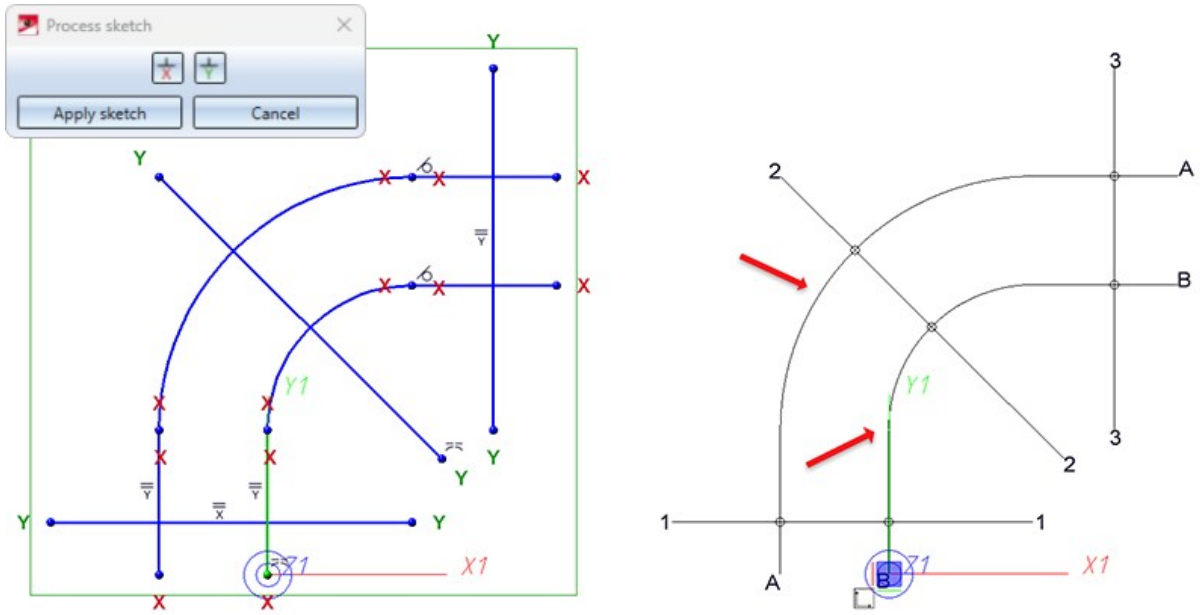


Old and new behaviour in the automatic generation of grid annotations

Drawn axes consisting of several lines

Previously, in a **Planning grid** sketch, it was only possible to create drawn axes consisting of a single line. Now, a drawn axis can also consist of several connected lines, i.e. a polyline.

If you draw a polyline consisting of several connected lines in the sketch, an axis is created for each polyline. The direction definition (X or Y) applies to the entire polyline. The planning grid annotation texts are generated at the start and end points of the polyline respectively.

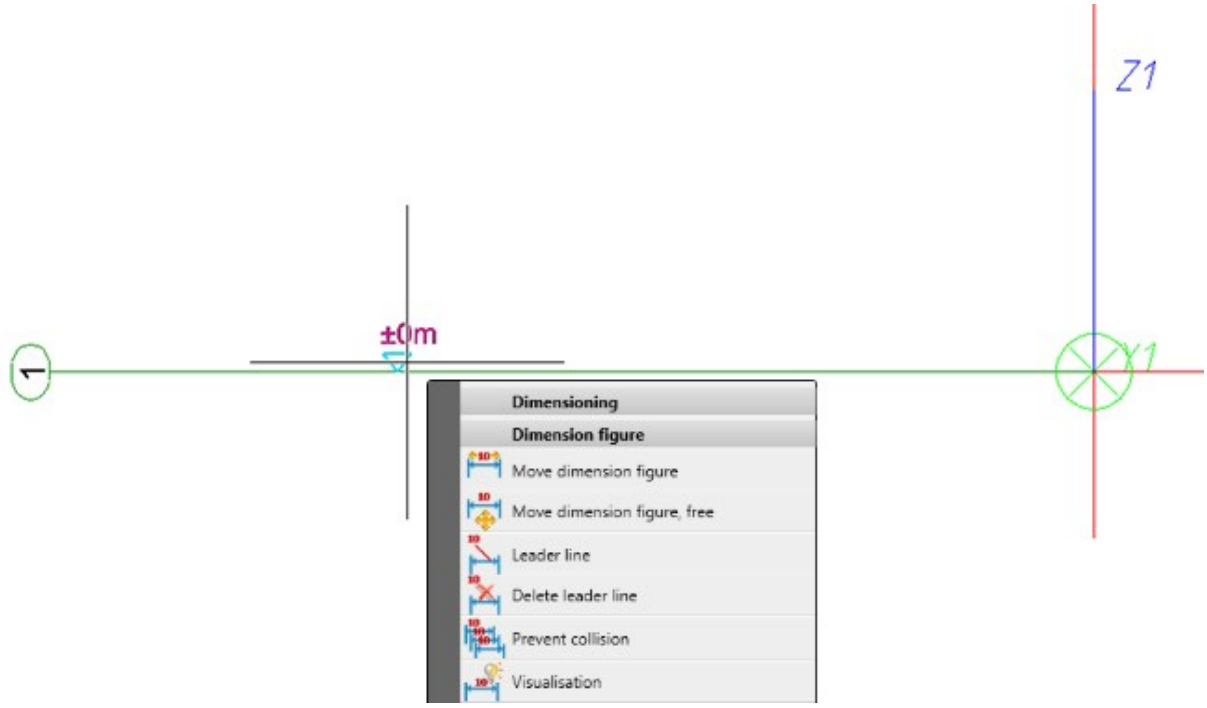


Please note: If you use individual edges of a drawn polyline axis as external references, these references are retained even after changes are made to the sketch, e.g. if the polyline is extended or if edges are lengthened, shortened or moved.

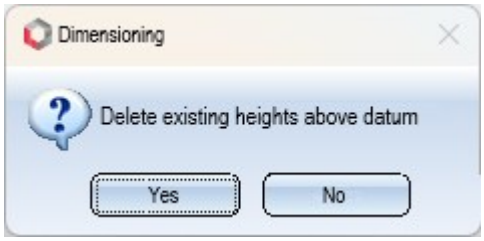
Polylines that contain both regular lines and auxiliary geometry are not permitted. Similarly, all lines within a polyline must have the same direction definition. In both cases, an error message will appear when you apply the sketch.

Heights above datum from grid annotations behave like manually created heights above datum

Heights above datum created via **Grid annotations** behave like manually created heights above datum once they have been created. They are treated as dimensions; this means that when you open the context menu for a height above datum, the context menu for dimensions appears.



If you delete a grid annotation that contains a height above datum , a message will appear asking whether the height above datum should also be deleted.



Old Steel Engineering Grid function removed

In addition to the modernised **Planning grid** function, used to create 3-D grids, the old **Steel Engineering Grid** function was previously still available under **3-D Standard** > **New** > **Others** > **Up to HiCAD 2024**.

It was removed in this major release.

Divide tab removed from the Subtract function dialogue

In the **Process with sketch** dialogue, which is opened via the **Subtract, with translation** , **Subtract, with translation+depth** and **Subtract, with rotation** functions under **3-D Standard** > **Process with sketch**, the **Divide** tab has been removed..

The options for dividing are now fully covered by the **Divide with sketch** function.

Save to catalogue

Save part as catalogue entry: Improved handling

The **Save part as catalogue entry** function has been improved. When saving to sub-catalogues within the factory standards, the selected catalogue path is retained.

Furthermore, thumbnails are now generated and displayed correctly when saving to sub-catalogues. The file name of the thumbnail can be specified manually if required.

Thumbnails are now saved as PNG files when saving to the catalogue

When saving parts to the catalogue using the **Save part as catalogue entry** and **Save part as variant table** functions, the thumbnail is now generated as a PNG file by default. These files are significantly smaller than the BMP files previously used. If required, the BMP format can still be selected manually.

Optimisations when changing the perspective

When changing the projection, e.g. using the**Front view** or**Rotate active XY plane to match the screen plane** functions, the view could previously move, in some cases arbitrarily and by a considerable distance. This has been optimised: when using these and similar functions, the view now remains in its previous position.

Move operations resulting from rotation using the mouse or the SpaceMouse are still detected and compensated for: if the view is subsequently rotated to a new perspective using one of the functions mentioned, it returns to the position in the drawing where it was before the rotations, regardless of how often or in what combination the view was rotated and the perspective changed in the meantime.

Clean up intersections

The **Clean up intersections** function is now also available for all objects for which a bending simulation is possible – including complex sheet metal shapes and arbitrarily curved profiles.

Hollow profiles where the intersection involves more than one plane surface cannot be cleaned up.

Uniquely update the article master for referenced parts

If, during referencing, the article master is transferred **only for sub-parts**, the same referenced part may have different article masters in different assemblies. The article master stored in the reference file was then not unique, which in some cases led to changing article masters during the update.

The part now retains its own article master and, during the update, checks whether this is linked to the document of the referenced part. If this is not the case and exactly one article master is assigned to the document, this is adopted. If several article masters are assigned, the previous one is retained and you receive a message prompting you to assign the article master manually. Nothing changes regarding the transfer **for main and sub-parts**.

Further information can be found under General Notes on Referencing .

Point clouds

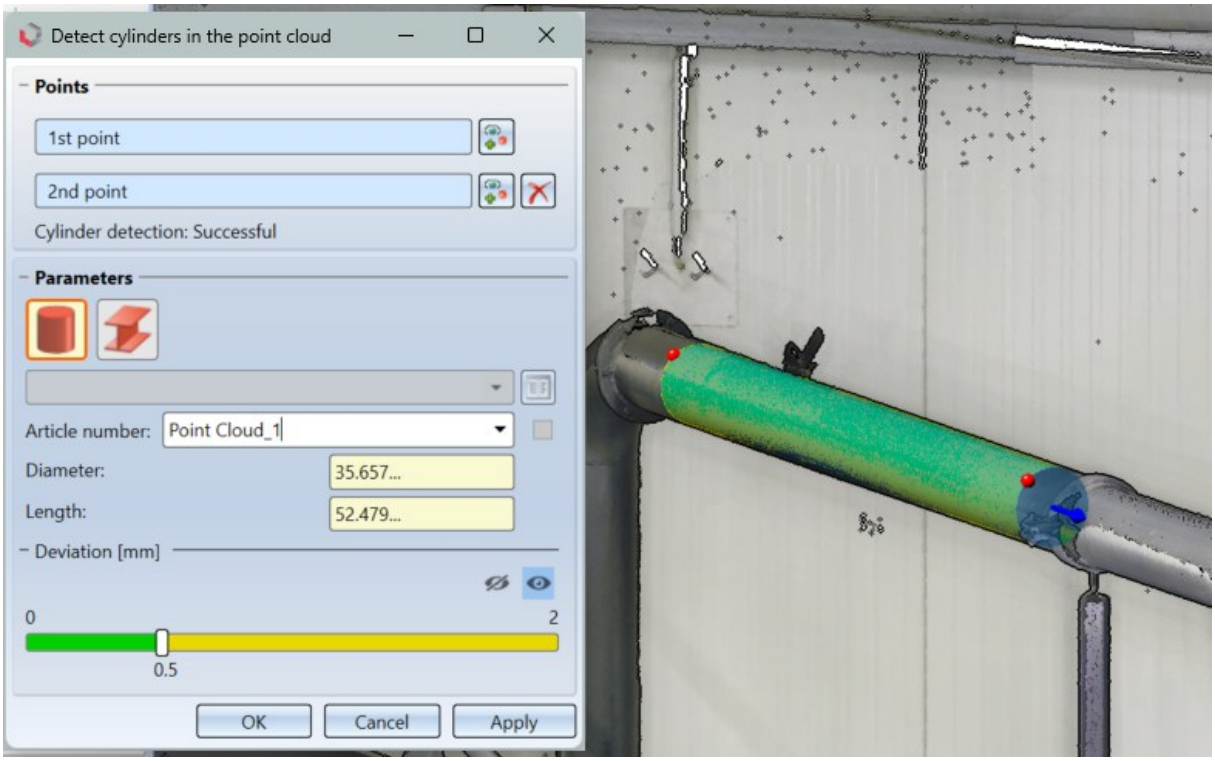
Major Release

Automatically detect cylinders and pipes in point clouds

Using the new **Detect cylinders in the point cloud**  function, cylindrical areas within a point cloud can be automatically detected and incorporated as parts into the drawing. You can find the new function in the ribbon at **Point cloud > Geometry recognition > Cylinder**.

By selecting two points on the scanned cylinder, a matching cylinder is generated and displayed as a preview. The diameter and length are shown in the dialogue window.

You decide whether to create a cylinder or a profile. If you choose a profile, select the appropriate pipe from the catalogue and apply it.



Feature Technology

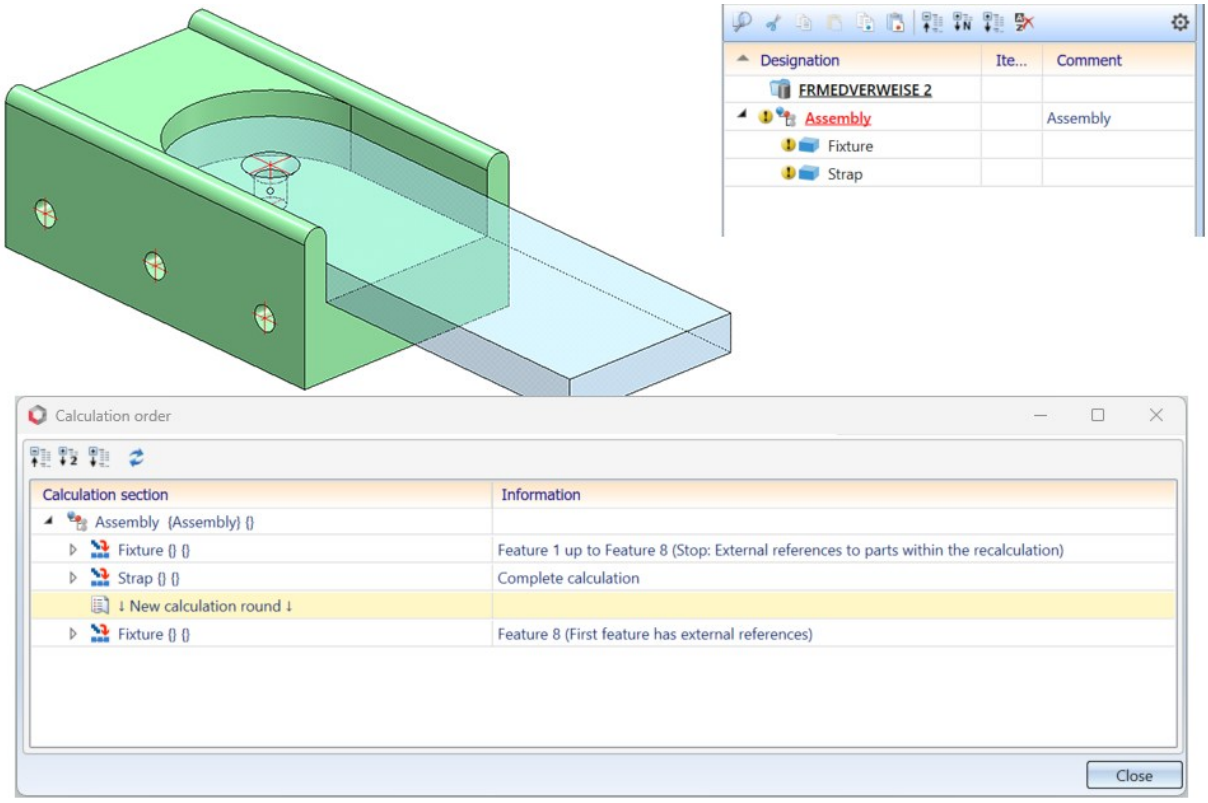
Major Release

Show calculation order of an assembly

When recalculating an assembly, the parts are not simply calculated one after the other. HiCAD 3D first analyses the dependencies between the parts and uses this to generate a calculation order, i.e. a list of calculation sections, which is then calculated from top to bottom.

This calculation order can now be displayed in a dialogue box. To do this, right-click in the empty area of the assembly's feature log and select **Show calculation order**.

The dialogue box shows, for each calculation section, the part, the calculated features, the next feature, and the external references that led to the division. Clicking on an entry highlights the associated part, the feature in the drawing, or the feature and the parameter in the feature log. This allows you, for example, to identify circular dependencies between parts, which can lead to unexpected results when an assembly is recalculated.

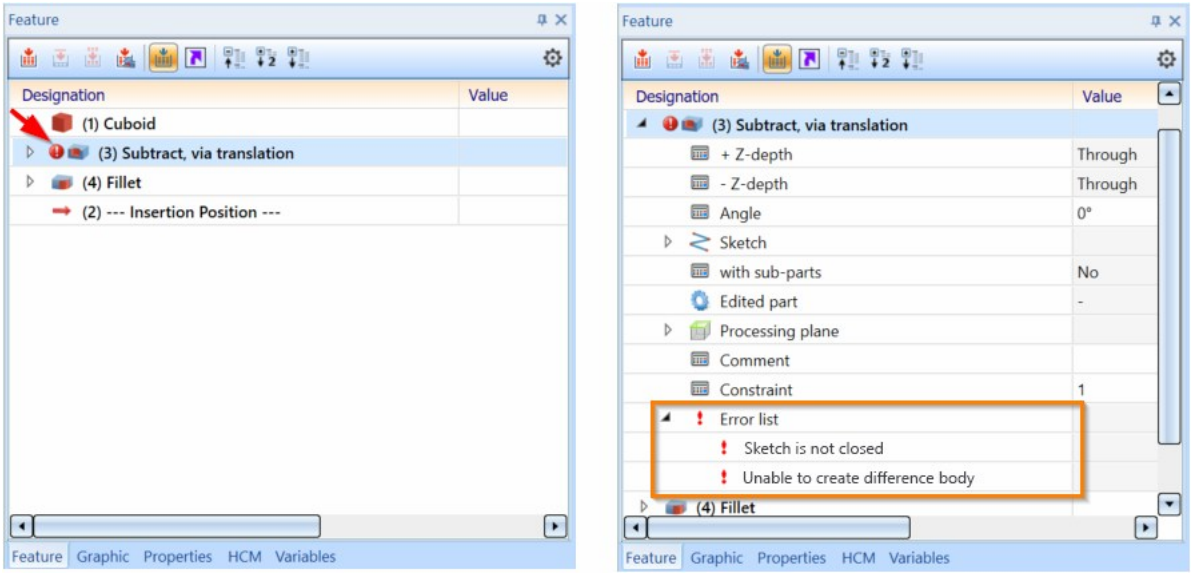


Optimized error display in the feature log

If an error occurs in a feature, clicking the error message expands the feature log to the parameter containing the error.

This behaviour has now been further improved:

- The affected parameter itself is also expanded, so that the error text is immediately visible.
- The feature log automatically scrolls so that the error text is within the visible area.



(1) Collapsed feature log with error message; (2) Clicking the error message in the collapsed feature expands the log to the error text

Replace formula with value function added to the context menu of the feature log

The **Replace formula with value** function was previously only available in the context menu of individual feature entries. It can now also be accessed via the context menu of the feature log and is then executed for all features in the log.

To open the context menu of the feature log, right-click on an empty area in the log.

Automation

Discontinuation

Discontinuation of the ISD.PDM.API

Before carrying out a HELiOS update (HiCAD PDM from 2027 onwards) for an older HiCAD version (HiCAD 3D from 2027 onwards), 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.

Major Release

For comprehensive automation of HiCAD 3D, existing features can now not only be created but also modified via the API.

For all features that can be created via the API, it is therefore also possible to modify the feature parameters retrospectively. The modification is carried out via a common write layer: the respective input value is assigned to the appropriate feature parameter, set via the typed parameter API, and the part is then recalculated. Thus, a single, common interface is sufficient for all features.

Not all parameters can be captured – for example, where a catalogue or a plant engineering environment is required; such cases are subtracted, with justification provided. Individual limitations of the API (such as angles stored internally as negative values or list parameters that cannot be uniquely addressed) are known and documented.

Drawings without updating referenced parts

The API now allows a drawing to be loaded without updating the referenced parts it contains. To do this, set the option

SceneLoadOptions.ReferencedPartsUpdateDialogMode to the value

ReferencedPartsUpdateDialogMode.DontShowAndUpdateNone.

Die nicht aktualisierten Teile werden dabei weiterhin als veraltet gekennzeichnet.

The parts that have not been updated will still be marked as out of date.

If the parameter **Lock non-updated, referenced parts against processing** is activated in the Configuration Editor at **System settings > Referencing**, these parts cannot subsequently be edited.

New API function for sectional views

The new API function **ViewCreator.CreateSectionalView** allows sectional views to be generated in a practical and programme-controlled manner:

ViewCreator.CreateSectionalView(Sketch sectionSketch, View sourceView, double distance, SectionalViewDepth depthType, Extent? depth = default)

A sectional view is generated from a section sketch for the specified original view. An overview of the parameters:

- **sectionSketch**: Sketch containing the section line(s).
- **sourceView**: Original view from which the sectional view is derived.
- **distance**: Distance of the sectional view.
- **depthType** (**SectionalViewDepth**): Type of section depth.
- **depth** (**Extent?**, optional): Section depth.

If the sketch contains a cut line, you can specify which side of the line is to be cut away. Furthermore, the perspective is set automatically: the viewing direction is perpendicular to the original view and aligned with it.

New API functions for the view area

With the new API functions, key view settings can now also be configured programmatically:

- View captions: **View.Caption** is used to set or retrieve the text of the view caption; **View.ShowCaption** is used to show or hide the caption; and **View.DeleteCaption** is used to remove it.
- Required part list: **View.RequiredPartList** defines the parts currently visible in a view as the required part list and restores it (in HiCAD: Load required part list).
- Part properties per view (Properties in view): **Node.Properties.Get/SetHiddenEdgesDashed** controls the Hidden edges dashed property; **Node.Properties.Get/SetContourRepresentation** enables or disables contour representation – for individual parts within a view, without altering the 3D model itself.

New API functions for standard part properties

With the new API functions, the properties of standard parts can now also be modified programmatically.This corresponds to the **Change representation** function (accessible via a standard part's context menu), in whose dialogue window you can specify, amongst other things, the **Representation** and, under **Production**, the **Factory assembly** and **Site assembly** options:

- Boltings: **Boltings.Get/SetConnectionSettings** reads or sets the display and manufacturing settings for a bolting connection – including the switch between factory assembly and site assembly.
- Single/grid standard parts: **StandardParts.Get/SetPartSettings** reads or sets the representation and production settings of standard parts – including switching between factory assembly and site assembly.

New API functions for imported parts

Parts imported via a 3D interface  are marked with this symbol  in the part structure in HiCAD, and additional import data is stored (e.g. the path to the import file). With the new API functions, this information can now be retrieved and an import updated:

- Retrieve properties: **Node.GetImportInfo()** returns a part's import information – whether it is an imported part and the stored data (e.g. the path of the import file).
- Update import: **ISD.CAD.IO.ExternalImportUpdate.Update()** updates an imported part, i.e. replaces it with a modified import file. This corresponds to the **Update imports**function via the 3D interface (right-click on the  symbol).

New API function for element installation: Renew dialogues

The API already provided the functions **Recreate()** and **UpdateVariants()** for editing element installations. In addition to these, the following is now also available:

- Renew dialogues: **RenewDialogs()** updates the user dialogue for variants that have already been installed – in the same way as the function of the same name, **Renew dialogues**, in the context menu of the **Element Installation** feature. If the user dialogue for a variant has been subsequently changed, this function can be used to bring the dialogue displayed when editing the Element installation up to date.

If only the dialogues need to be updated, **Renew dialogues** is a quicker alternative to **Recreate**.

Interfaces

Major Release

Update to CADfix 14

Following the update to **CADfix 14**, the following new file formats are now available for import:

- **ACIS**: R1 - 2026 R1
- **CATIA**: V5-6 R2026
- **Inventor**: 11 - 2027
- **Solid Edge**: V18 (2006) - 2026
- **Parasolid**: 9 - 38.1
- **Creo Parametric**: Pro/E 2001 - CREO 12.0
- **SOLIDWORKS**: 98 - 2026
- **NX**: 1 - 2512

DSTV-NC export for steel pipes, round beams and round steels

Profiles with a circular cross-section, such as steel pipes, round beams and round steels, can now also be exported via the **DSTV-NC** interface.

%PIPE_LINE_NAME attribute in IFC export

The %PIPE_LINE_NAME attribute (pipeline name) is now also included in the **IFC** export.

New interface: IDEA StatiCa

A new interface to **IDEA StatiCa** is now available. This allows the geometry of a drawing – i.e. the **Parts**, their **Contact surfaces** and the **Connections** formed by weld seams and boltings – to be transferred to IDEA StatiCa. The loads are not calculated from HiCAD 3D, but are entered directly into IDEA StatiCa.

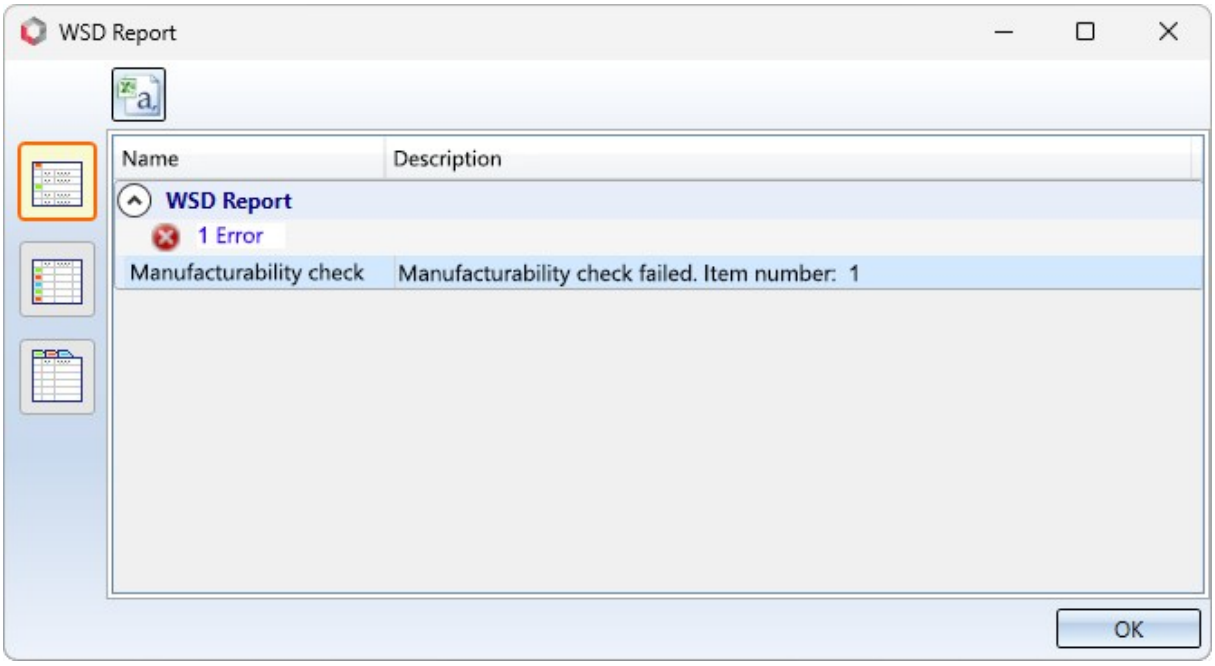
Sheet Metal

Major Release

Manufacturability check during automatic drawing derivation

In the Configuration Editor, at **Sheet Metal > Manufacturability check**, the new parameter **Perform a manufacturability check when creating automatic drawing derivations** is available. It controls whether a manufacturability check is performed for the sheet metal parts contained in workshop drawings during automatic derivation (creation and updating).

If the parameter is activated, each sheet metal part in the part list is checked for manufacturability based on the criteria set at **Sheet Metal > Manufacturability check**. If a sheet is identified as unmanufacturable, a corresponding message appears in the drawing derivation report window.



The check is performed for all selected sheet metal parts—regardless of whether a (new) view or development pattern is actually generated for the respective sheet metal part. This ensures that even an unchanged sheet metal part is checked during an update; otherwise, it would remain unchecked and would be incorrectly identified as manufacturable.

If the parameter is deactivated (default setting), no manufacturability check takes place during drawing derivation.

Export of flat steels and wide flat steels

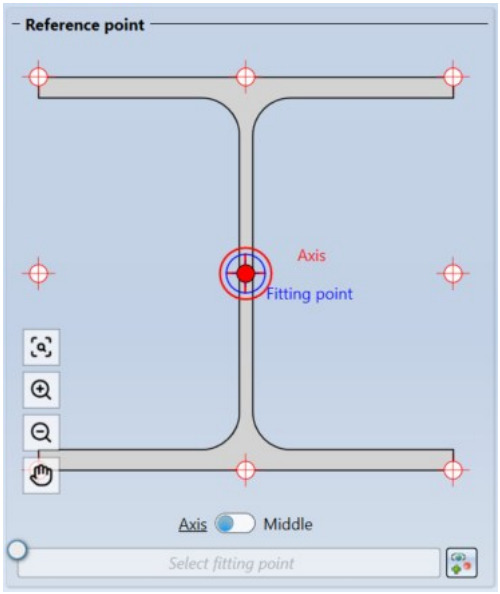
The **Develop and export sheets** function for DXF/DWG and STEP export has been expanded: In addition to sheets and steel plates, **Flat steels** and **Wide flat steels** can now also be exported. The X-Y plane of the part coordinate system is used as the plane of the sheet or plate.

Steel Engineering

Major Release

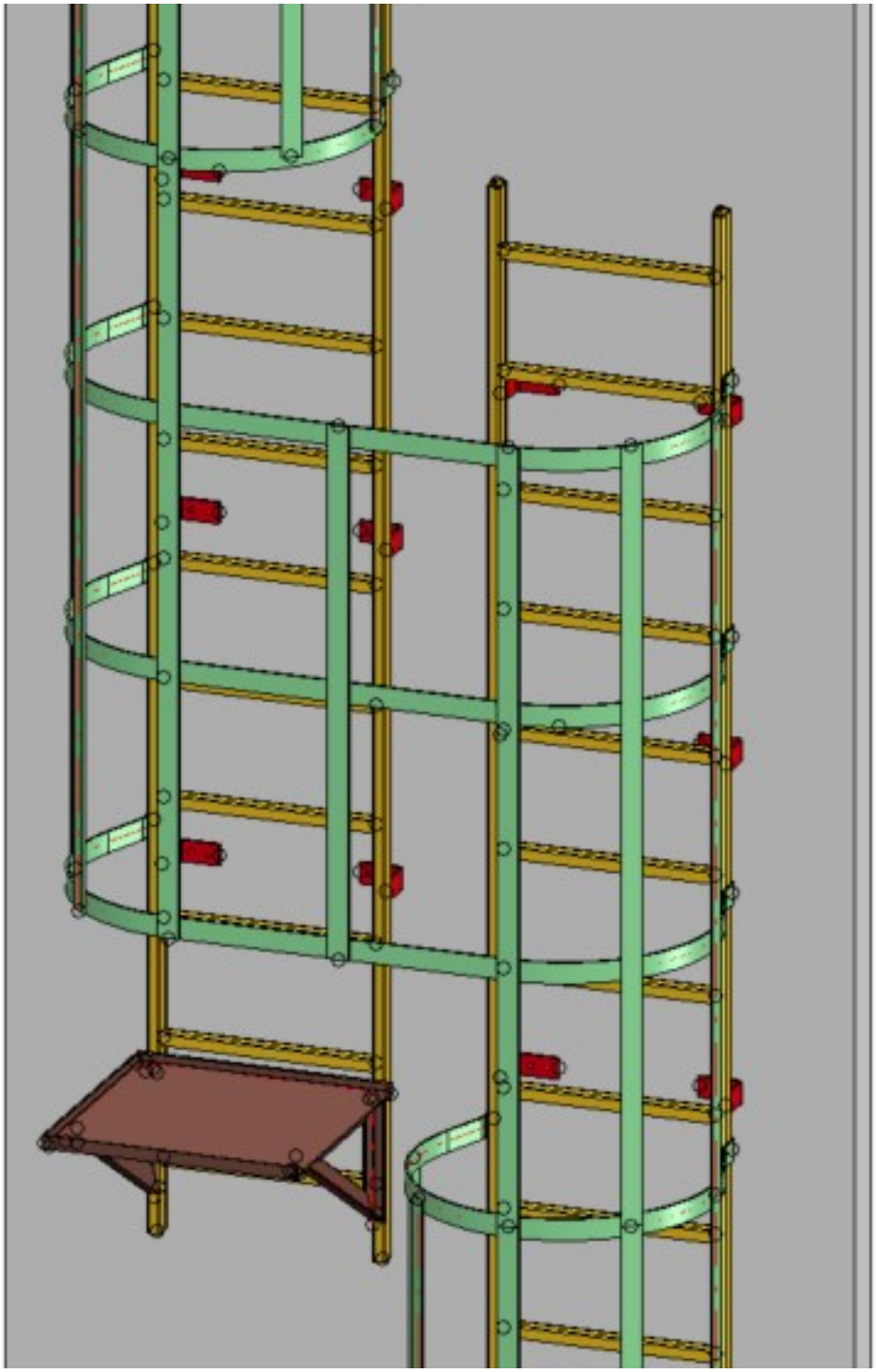
Insertion of new beams: Enhanced display of beam cross-section

In the dialogue window for the **Insert new beam** function, the display options for the beam cross-section in the **Reference point** area have been expanded. Using the icons on the left-hand side, you can adjust, enlarge, reduce, and move the cross-section display. These functions are available for all beam cross-sections, allowing you to see even the details of complex series beams.



New function: Ladder (new)

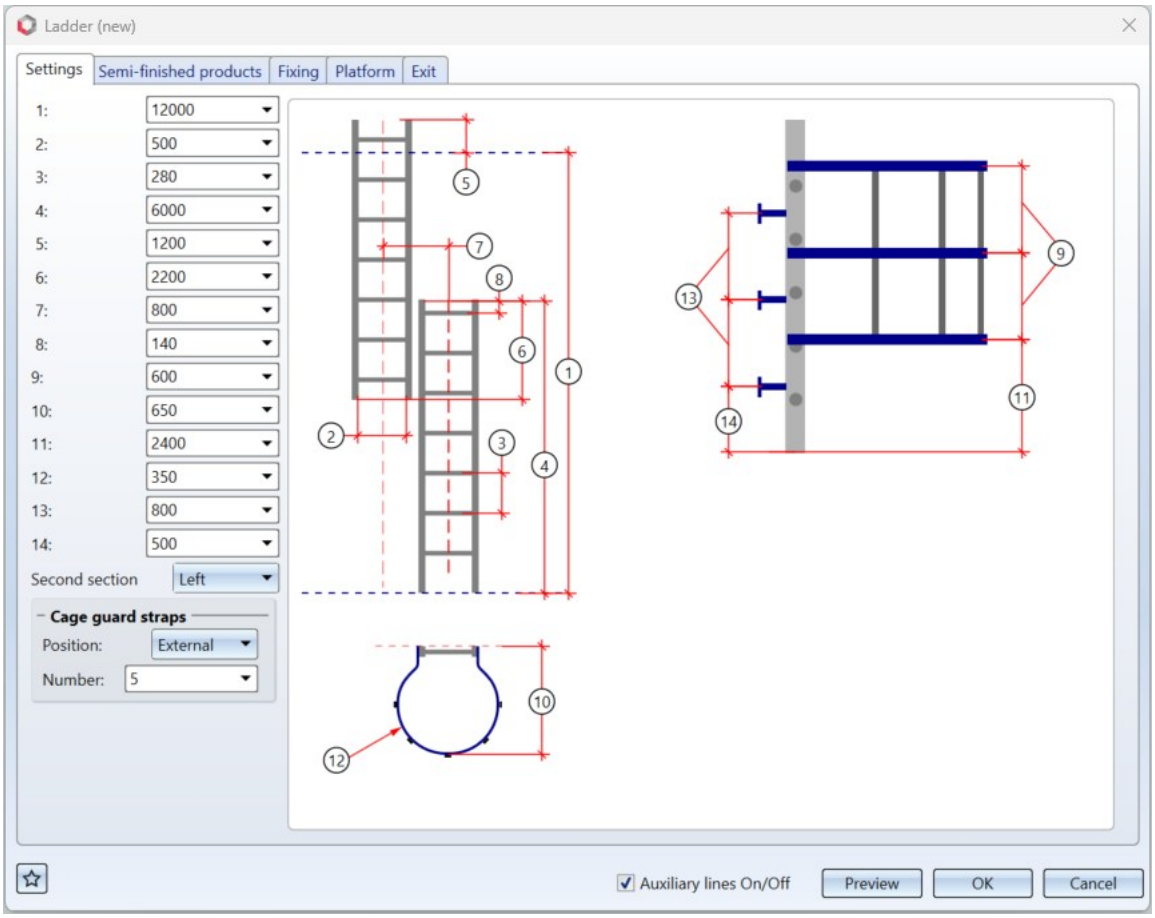
In the **Civil Engineering functions** docking window at **Steel Engineering > Stairs + Railings > Stairs**, the new **Ladder (new)** function has been added. This feature allows you to create ladders parametrically and in accordance with DIN standards.



After calling up the feature, the **Ladder** dialogue window appears. It consists of the **Settings**, **Semi-finished products**, **Fixing**, **Platform**, and **Exit** tabs.

In the **Settings** tab, you define the dimensions and geometry of the ladder, as well as the position of a second section and the parameters of the guardrail. In the **Semi-finished products** tab, you define all parts used via the catalogue and specify the usage for the assemblies.

In the **Fixing** , **Platform**, and **Exit** tabs, you specify the wall fastening, an optional platform, and the type and dimensions of the exit.



New interface: IDEA StatiCa

A new interface to **IDEA StatiCa** IDEA StatiCa is now available. This allows the geometry of a drawing – i.e. the **Parts**, their **Contact surfaces** and the **Connections** formed by weld seams and boltings – to be transferred to IDEA StatiCa. The loads are not calculated by HiCAD, but are entered directly into IDEA StatiCa..

Drawing Management

Major Release

Logos in drawing frames updated

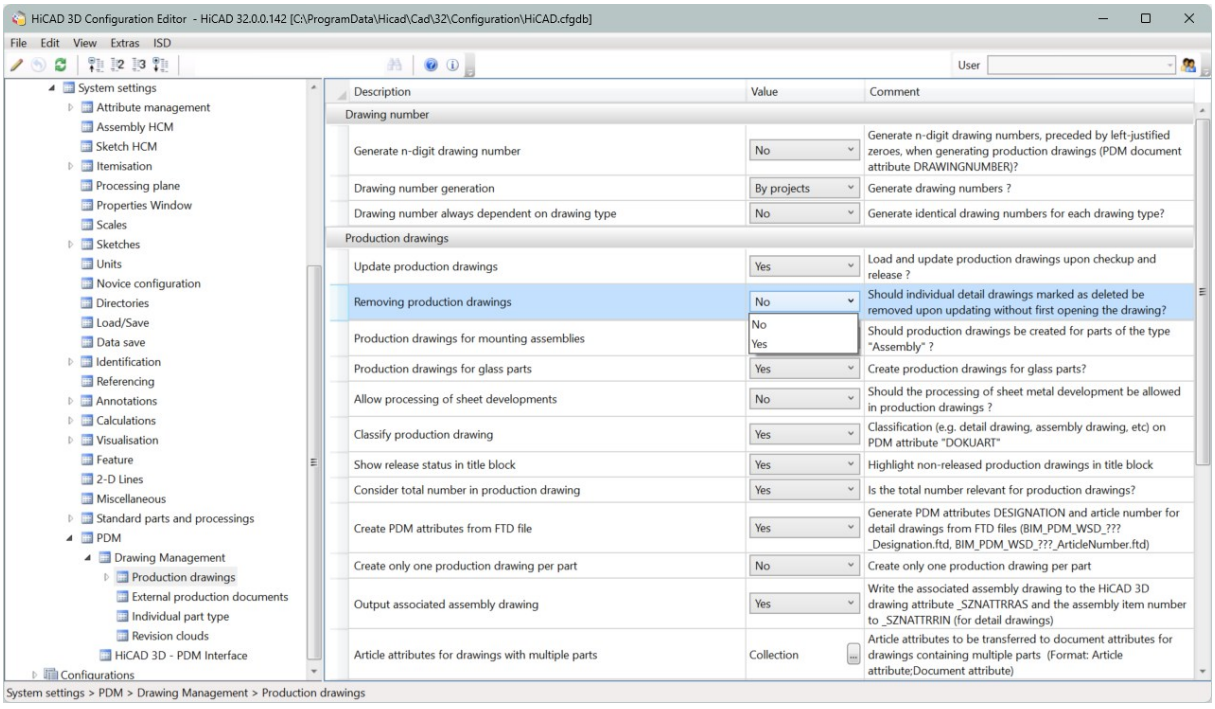
The drawing frames supplied with HiCAD 3D now feature the new logo in the title block. This change has been applied to all supplied drawing frames. The updated drawing frames can also be used in the previous version..

Directly remove documents for deleted parts when updating

When you delete parts from a drawing that was managed with the **Drawing Management** module, the associated records are assigned the link type **Deleted** when the drawing is saved. Previously, when updating the drawings, the associated documents first had to be loaded into **HiCAD 3D** and then selected and removed.

This process can be simplified using the new **Remove production drawings** parameter in the **Configuration Editor** at **System settings > PDM > Drawing Management > Production drawings**. If the parameter is set to **Yes** via the drop-down menu, detail drawings marked as deleted are removed directly during the update, without having to open the drawing first.

The parameter only applies to detail drawings for individual parts, i.e. drawings of individual parts and drawings of individual assemblies in which no sub-parts are detailed. Collective drawings and drawings containing sub-parts will continue to be handled as before.



Importing standard masks for Drawing Management via the HiCAD PDM Desktop

You can now import the standard templates for Drawing Management directly via the HiCAD PDM Desktop. The **Import standard package** function can be found in the HiCAD PDM Options at the bottom of the window

via the Manage button. Once you have selected a setting package – such as HiCAD 3D Drawing Management – the associated standard templates will be imported into your system.

This procedure replaces the previous method of importing standard templates via the **ParameterConfigAdministrator.exe** tool. For further information, see Installation / Parameter Configuration.

Metal Engineering

Major Release

LogiKal functions with the unit of measurement "inch"

The LogiKal functions in HiCAD 3D can now also be used when the unit of measurement "inch" is set in HiCAD 3D. This applies to the replacement of LogiKal profiles as well as the import of LogiKal mullions and transoms into a metal framework.

This requires that the unit "inch" is also set in LogiKal. Items comprising a single pane of glass, where the glass thickness is specified as a whole-inch value, are supported.

Installation Planning

Major Release

Basic module HiCAD 3D Solution

From HiCAD 3D 2027 onwards, the basic module **HiCAD 3D Solution** offers the **Save variant for element installation**  function.

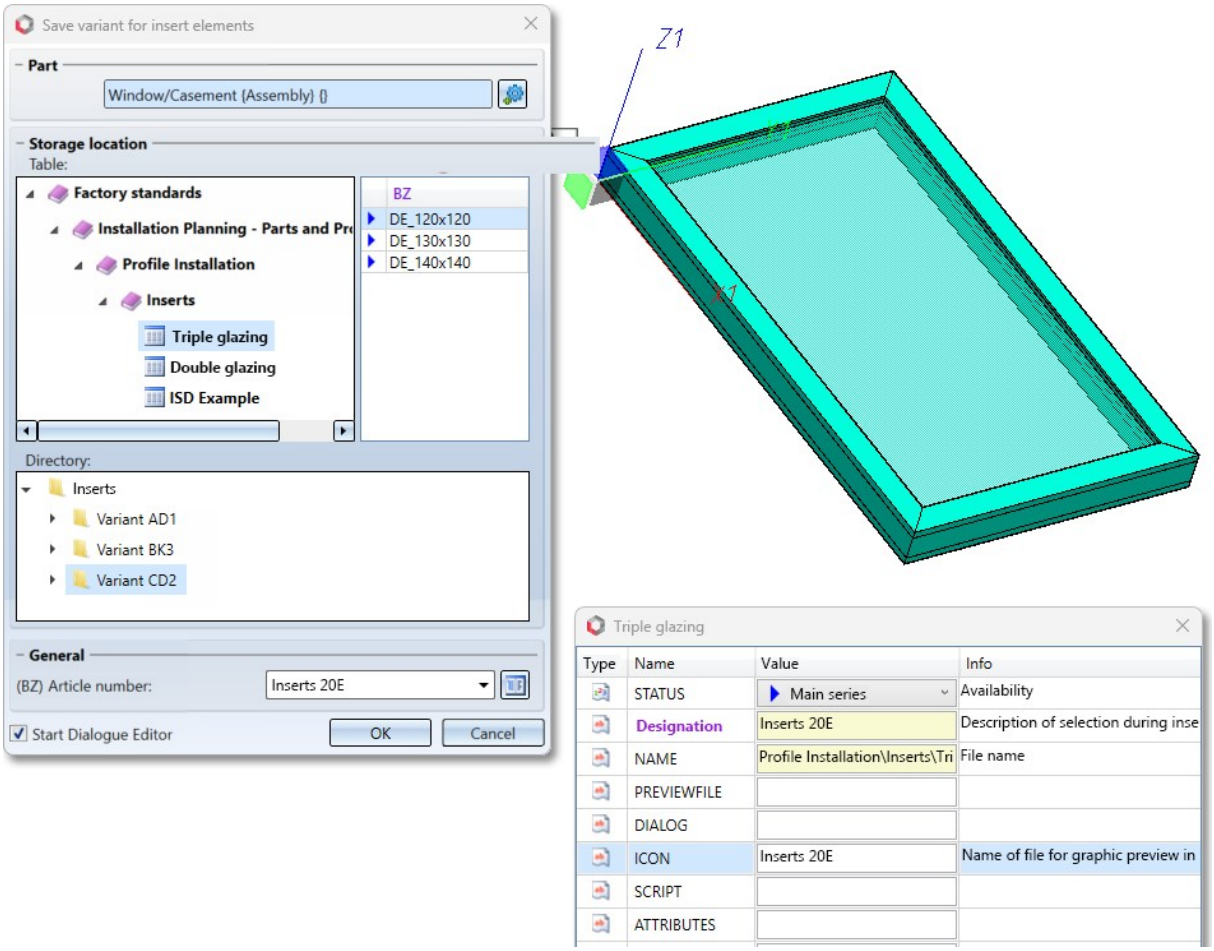
Profile Installation

Major Release

Save variant for inserts

Customised inserts for a profile installation can now be created directly from the **Profile Installation** menu bar using the new function **Save variant for inserts** . To do this, you must have designed and parameterised the relevant insert and defined a fitting coordinate system for the variant.

The only location offered for saving is the associated profile installation catalogue at **Factory standards > Installation Planning – Parts and Processes > Profile Installation > Inserts**.



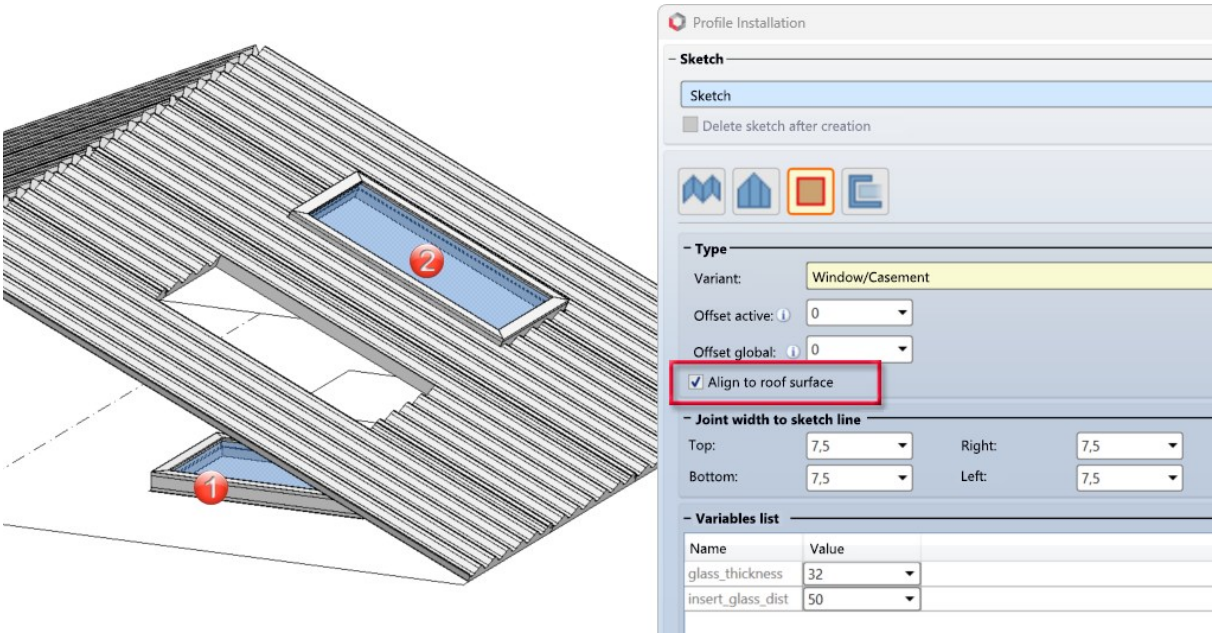
Align inserts to roof surface

Previously, when dealing with a roof inclination, inserts were always projected flat onto the horizontal surface (sketch plane). There is now a new option, **Align to roof surface**, on the **Inserts** tab.

- If this option is activated, the insert element lies in the plane of its roof section rather than flat in the sketch plane.
- The insert dimensions are stretched in the roof direction in accordance with the roof inclination, as the sketch is a flat plan view.
- This option is set per insert (not globally).
- In existing drawings, the insert will continue to be build in flat. However, you can change this via the feature.

This option is only available if the opening can be unambiguously assigned to a roof section and its position is defined by a catalogue entry. The relevant area is that of the roof section bounded in the roof direction, not the outer contour – an opening may extend beyond the outer contour perpendicular to the roof direction.

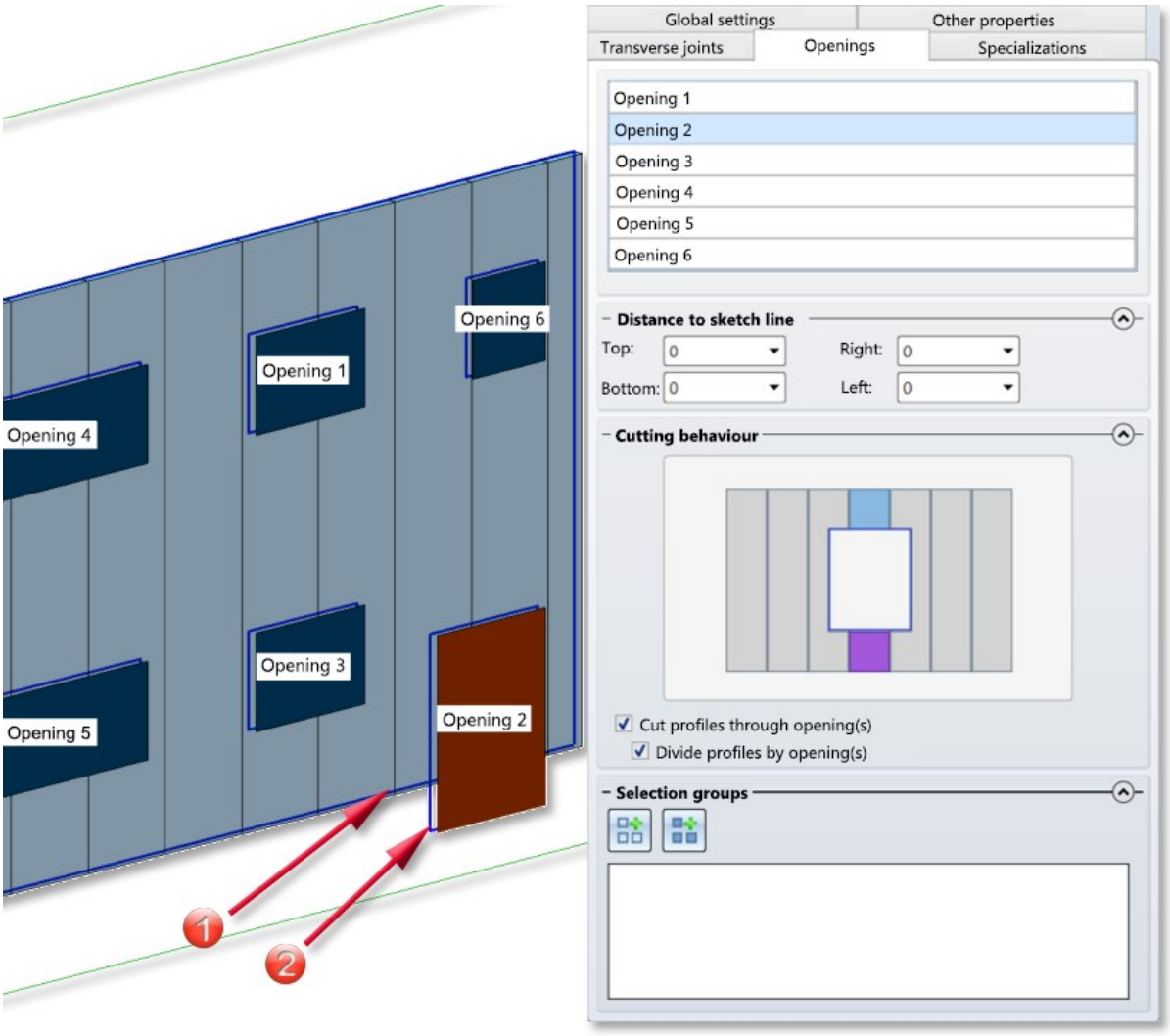
If, after the switch has been set, an opening subsequently extends across several roof surfaces (e.g. following a sketch modification), an error symbol appears next to the option. In this case, deselect the option or correct the sketch.



(1) Insert not aligned with the roof surface, (2) Insert aligned with the roof surface

Openings may extend beyond the installation area

Openings (e.g. door openings) that cross the installation area are common in practice. Previously, however, the rectangle representing the opening in the installation sketch was not permitted to extend beyond the outer contour of the installation area, and the outer contour could not be interrupted – such situations could only be implemented manually. From HiCAD 3D 2027 onwards, openings may touch or cross the outer contour of the installation area.



(1) Outer contour of the laying area, (2) Opening extending beyond the outer contour

Hinweis:

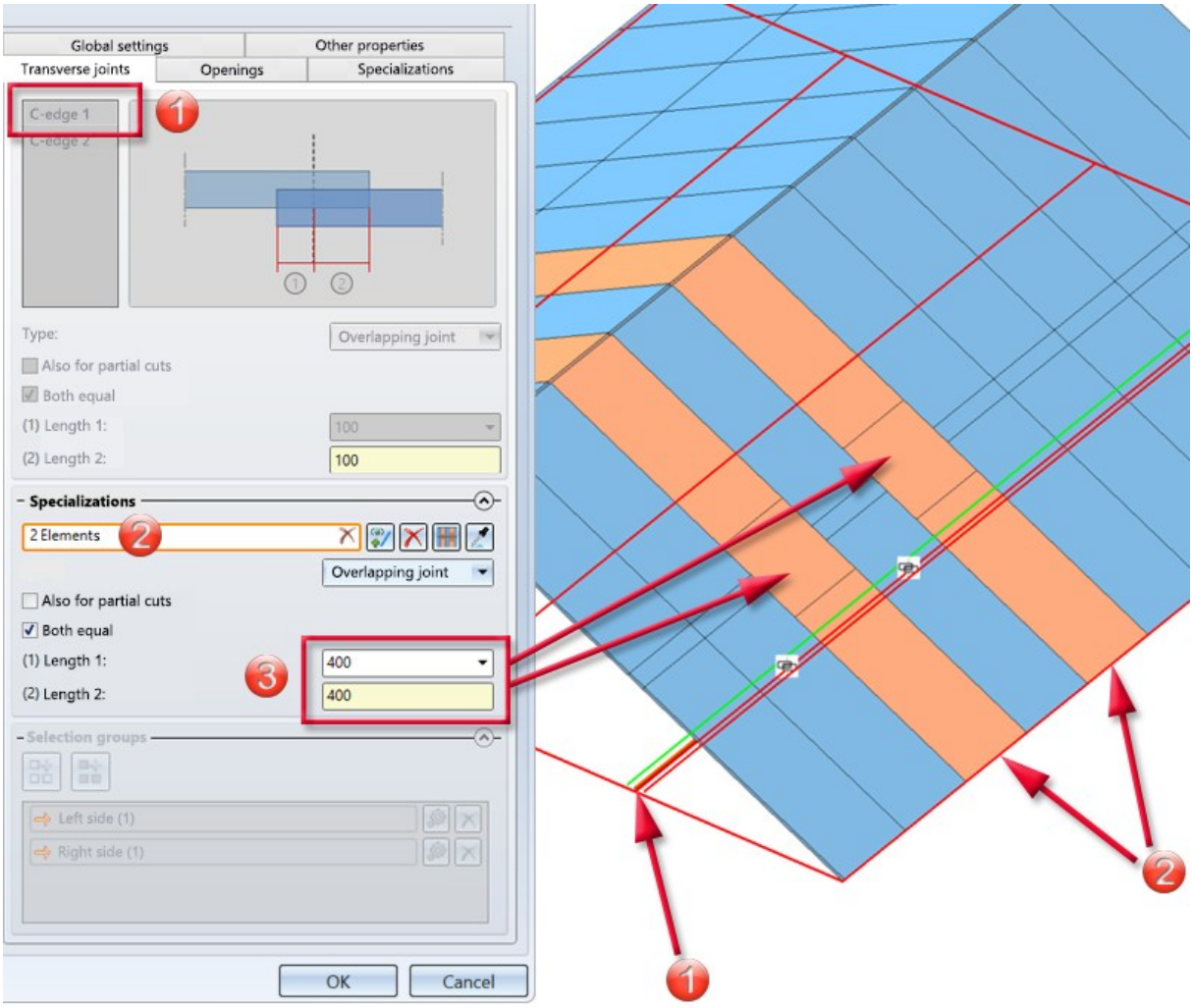
Existing drawings remain unchanged. Exception: Previously, it was not possible to create an opening that crossed the outer contour (an error message appeared); such an opening is now created following recalculation.

Defining transverse joints for individual profile paths

Previously, the settings for a transverse joint always applied to the entire layer. On the **Transverse joints** tab, there is now a new **Specializations** section where the transverse joint settings can also be defined differently

for individual profile paths. This allows you, for example, to arrange profiles in a staggered pattern to improve structural stability.

To do this, select one or more transverse joints and then select the affected profile paths  for which different values are to apply. You can then change the settings. Use the pipette symbol  to apply the specializations to the selected profile paths. Tracks with specializations are marked by symbols at the relevant points (visible only on the **Transverse joints** tab).



(1) Composite edge of the transverse joint, (2) profile paths to be modified, (3) modified overlap values

Specialized profiles when changing the installation width

On the **Specializations** tab, you can assign profile types from the catalogue to individual profiles in a layer that differ from the **Global settings** for the installation area. If you then change the **Installation width** in the same layer on the **Other properties** tab to a value that differs from the catalogue default, these specializations will now be retained.

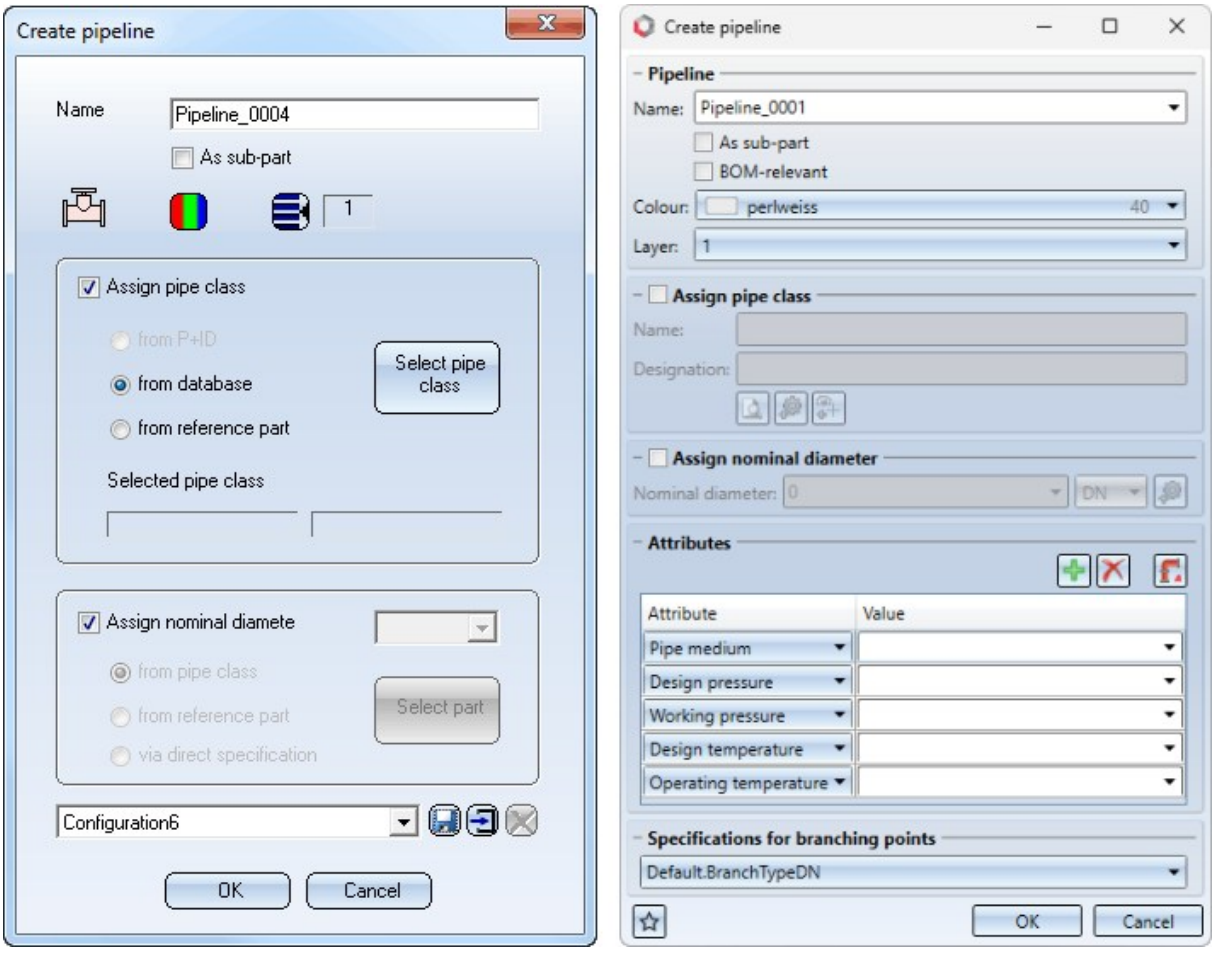
Previously, the profiles defined on the **Specializations** tab were deleted when the installation width was changed. This behaviour has been adjusted so that specialized profiles are retained even when the installation width is changed.

Plant Engineering

Major Release

New Pipeline Dialogue

In HiCAD 3D 2027, the dialogues for creating and editing pipelines have been revised. The dialogue for creating pipelines is still available under **New** > **Create pipeline** .



Before HiCAD 3D 2027, from HiCAD 3D 2027 onwards

The following properties can be specified in the **Create pipeline** dialogue window

1. Pipeline designation
2. BOM-relevance
3. Create pipeline as a sub-part of the active part
4. Colour
5. Layer

6. Pipe class (selectable via the database, a reference pipe part or an P&ID symbol from the linked P&ID schema)
7. Nominal diameter (selectable via direct input or a reference component)
8. Settings file for branching points
9. Further customisable plant engineering attributes for the pipeline

The first seven properties behave as they did in the previous dialogue window.

The procedure for settings files for branch points has been changed. The creation and editing of these files can no longer be initiated directly from the pipeline dialogue, but must be carried out via the Plant Engineering settings. In the pipeline dialogue, only the files present in the relevant directory are offered for selection.

A new feature is the option to define additional plant engineering attributes or custom attributes for the pipeline. This allows existing attributes to be applied directly to the pipeline.

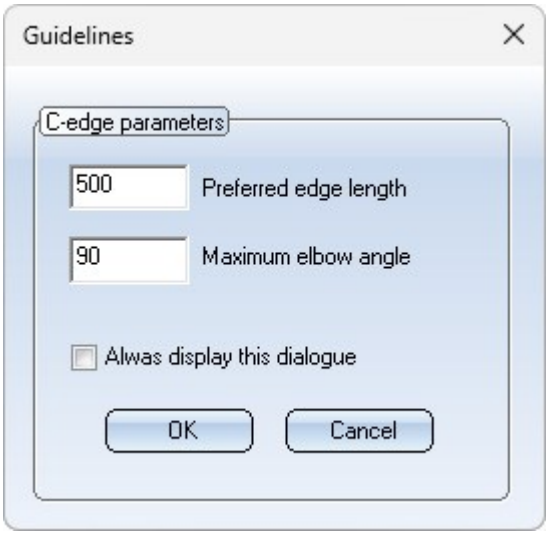
To edit an existing pipeline, the new **Edit pipeline**  function is available under **Plant Engineering** > **Pipeline Tools**. This function allows you, amongst other things, to change the pipeline's name, colour and layer. The previous functions for changing these properties have therefore been removed from the menu. They remain available via the context menu for pipe parts.

The functions **Assign pipe class and nominal diameter** and **Change pipe class and/or nominal diameter of all parts** have also been removed. The assignment or modification of pipe class and nominal diameter can now also be carried out via **Edit pipeline** .

If the pipeline already contains parts, changing the nominal diameter and pipe class is still only possible under certain conditions. For example, the pipeline must not contain any pipe parts without a guideline.

Guideline via end points without a minimum length

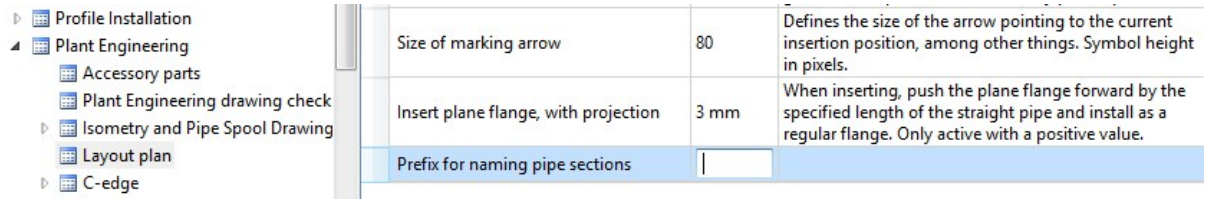
When calculating a guideline, the set minimum length is now treated as the preferred length.



Previously, edges shorter than the specified minimum length were discarded. From HiCAD 3D 2027 onwards, shorter edges are also taken into account where required by the geometry. The specified minimum length will still be given preference where it can be adhered to.

Configure naming of pipeline sections

For pipeline sections, the prefix of the automatically assigned name can now be configured. The setting is located in the Configuration Editor under **Plant Engineering** > **Layout plan** > **Prefix for naming pipe sections**. If no prefix is specified, the default name is **Pipeline_section_000X**.

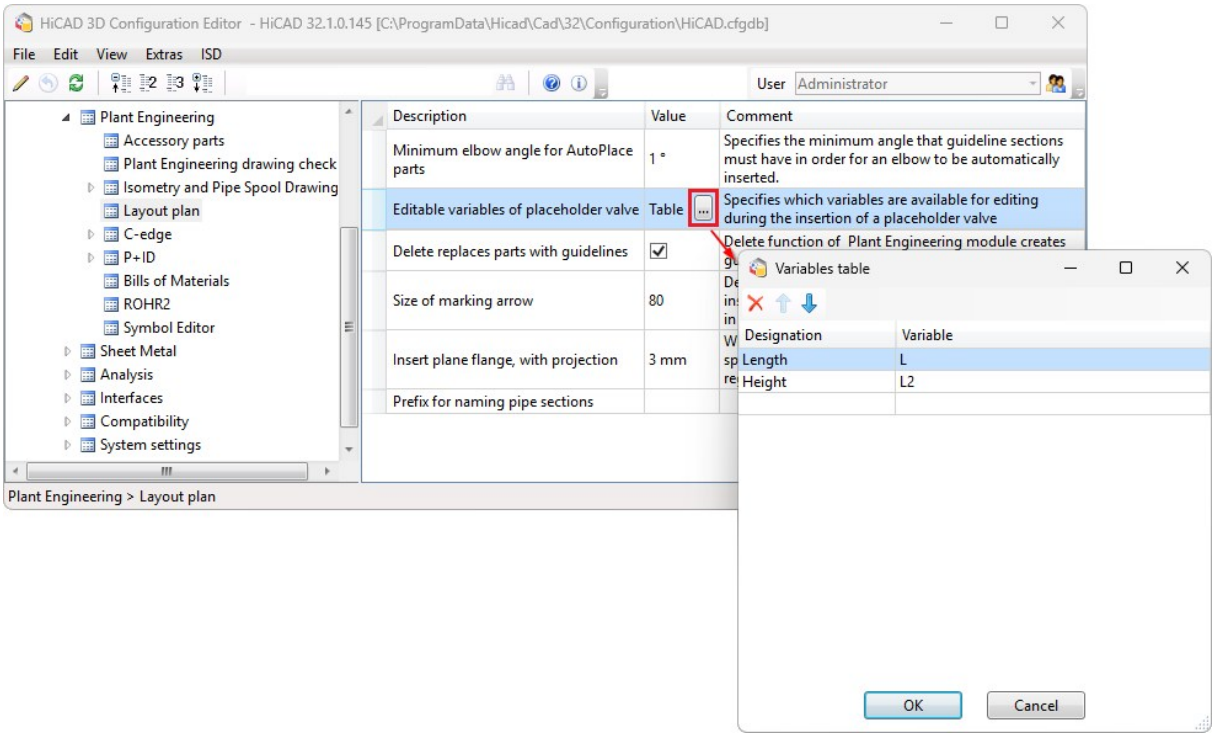


Automatically add blind flanges during AutoPlace process

The function **Plant Engineering** > **Guideline Tools** > **AutoPlace parts on guidelines** now automatically adds blind flanges to free pipeline ends in a single operation. Previously, blind flanges were only inserted if there was already a flange at the free pipeline end. f a flange had first been created using the **Flanges on free ends** option, the function had to be run again to insert the blind flange. From HiCAD 3D 2027 onwards, blind flanges are also added to flanges created during the same Auto-Place process. This eliminates the need for an additional step.

Extend placeholder fittings

For placeholder fittings, it is now possible to specify which variables can be edited during the insertion of the fitting. The setting **Editable variables of placeholder valve** is available for this purpose in the Configuration Editor under **Plant Engineering** > **Layout plan**.



The default setting is tailored to the placeholder fitting supplied with HiCAD 3D.

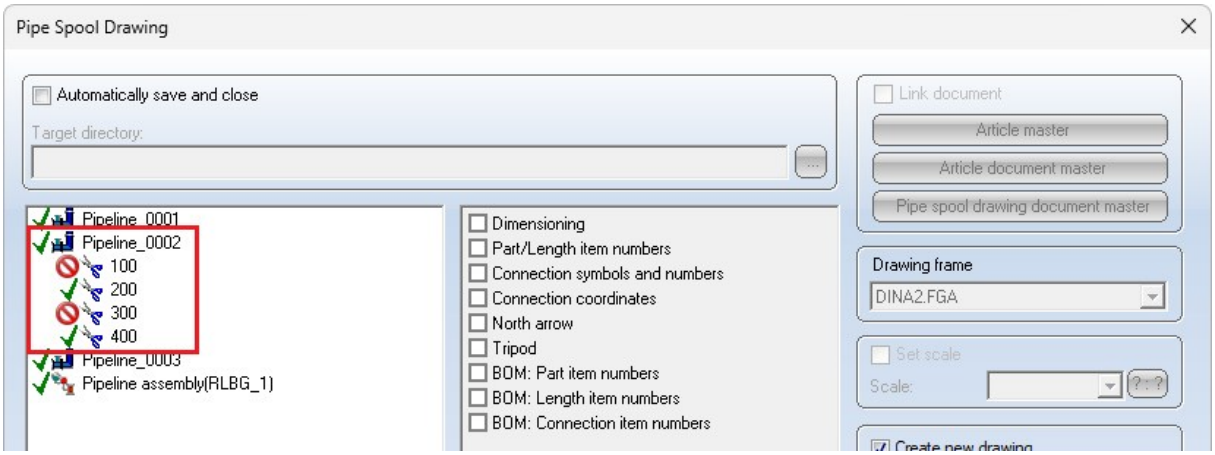
Pipeline isometry / Pipe spool drawing

Pipeline breakdown on separate sheets

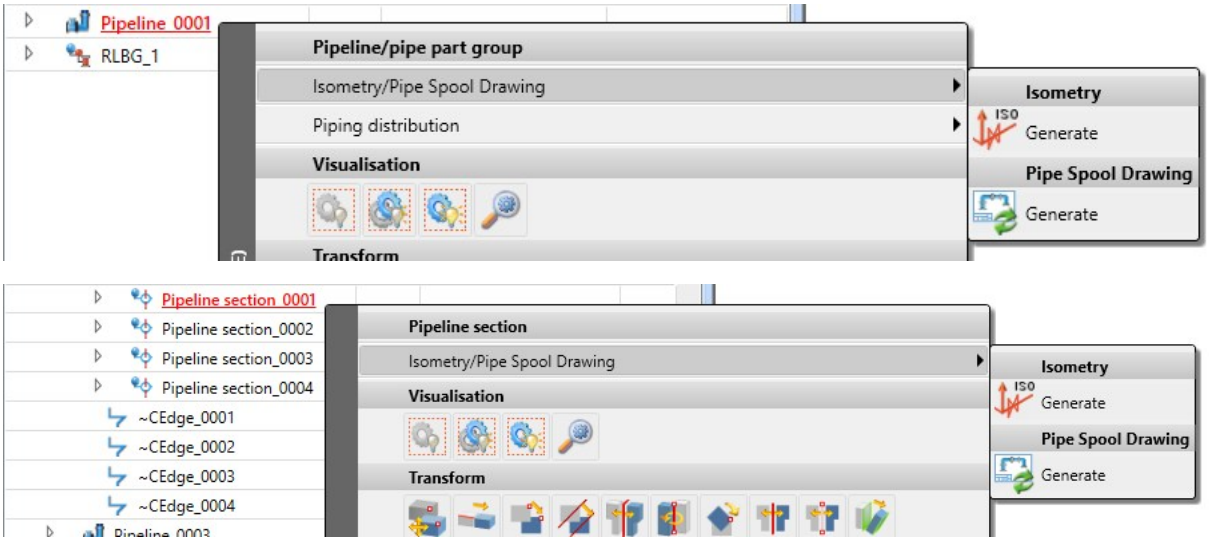
Previously, pipe spool drawings for parts of a pipeline were created by selecting individual parts before generating the drawing. This selection was then only visible on the pipe spool drawing sheet and had to be repeated each time a new drawing was generated.

From HiCAD 3D 2027 onwards, the pipework sections already used for isometric views can also be utilised for creating pipe spool drawings. This means that it is no longer necessary to select individual parts beforehand. The option **Part selection before displaying pipe spool drawing dialogue** has been removed from the Configuration Editor.

In the pipe spool drawing dialogue, pipelines and pipe sections can be enabled and disabled independently of one another. This allows both pipe spool drawings of the entire pipeline and of individual pipe sections to be created within a single drawing. This is also possible for split isometrics.



Separate context menus are also available for pipelines and pipe sections. These include an option to start creating the pipe spool drawing or isometric drawing directly.



Hinweis:

For split pipelines, the setting **Generate complete isometric drawing of split pipelines** in the Configuration Editor can be used to specify whether an isometric drawing of the entire pipeline should also be generated.

Isometric BOMs: %PART_COUNT in addition to %PART_QUANT

The **%PART_QUANT** attribute allows you to specify the quantity of pipe parts in BOMs. Straight pipes are treated differently here, as their total length is displayed rather than their count.

From HiCAD 3D 2027 onwards, **%PART_COUNT** is also available. This attribute does not apply the special treatment to straight pipes; these are therefore simply counted.

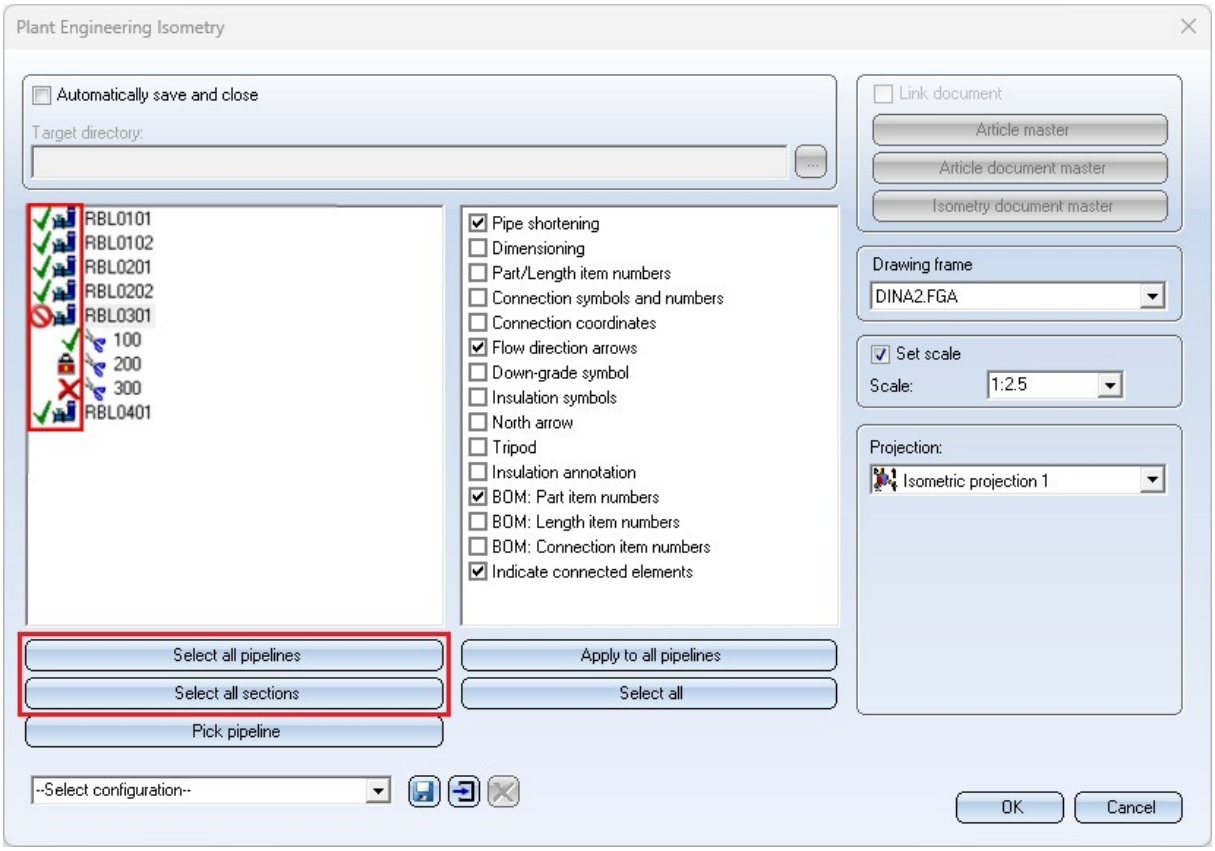
Updating pipe spool drawings and isometries for referenced pipelines

Referenced pipelines are now reliably updated in pipe spool drawings and isometrics following changes to the part structure. If parts are added to or deleted from a pipeline, the generated sheet areas are updated accordingly. In the case of split pipelines, the parts of the respective pipeline section are taken into account.

Update individual sections/sheets separately for split isometries/pipe spool drawings

For isometries and pipe spool drawings of split pipelines, from HiCAD 3D 2027 onwards, individual sheets can be regenerated without altering the status of the other sheets in the drawing.

To this end, the previous checkboxes used to select or deselect drawings or drawing sheets for pipelines, pipeline sections and pipeline assemblies have been replaced by four icons. Sheets from existing drawings can be removed or regenerated. An additional option allows the respective sheet to remain unchanged. The fourth icon corresponds to the previous empty checkbox.



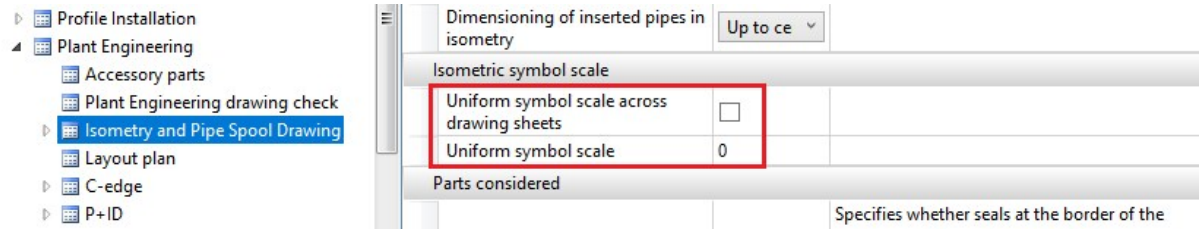
In addition, a button has been added to select or deselect all sections of the current pipeline selection.

The available options depend on the drawing from which the dialogue is opened:

Symbol	Option	Availability		
		Layout plan with document link	**Layout plan without document link**	**Isometry/pipe spool drawing**
	Create (new) sheet	all	all	all
	Do not create sheet/drawing	Pipelines, Assemblies	Pipelines, Sections, Assemblies	-
	Keep sheet	Sections	-	Sections
	Delete/do not create sheet	Pipelines, Sections, Assemblies	-	Pipelines, Sections, Assemblies

Uniform symbol size in isometries

From now on, the symbol size can be standardised across all sheets of an isometry. This applies to both the selected scale and the pipe symbols. For the symbol scale, two new settings are available: **Uniform symbol scale across drawing sheets** and **Uniform symbol scale** in the Configuration Editor under **Plant Engineering** > **Isometry and Pipe Spool Drawing**.



Selecting the appropriate setting defines the scale for all isometric sheets. It is then no longer possible to set a different scale in the isometry dialogue. If only **Uniform symbol scale** is activated but no value is specified, all sheets that would previously have been scaled automatically will be scaled at a ratio of 1:1.

For elbows, a uniform elbow symbol size can be set whilst using pipe shortening. To do this, the setting **Adjust size of elbow symbols to the base unit of the pipe shortening** is available in the Configuration Editor under **Plant Engineering** > **Isometry and Pipe Spool Drawing**. If this is enabled, the elbow symbol size for all elbows, regardless of their size or curvature, is scaled to a radius of *[base unit of pipe shortening]* × *[proportions of other parts]* .

Do not include gaskets at the end

With HiCAD 3D 2027, gaskets at the ends of pipelines, pipeline sections or pipeline assemblies can be excluded when creating isometrics and pipe spool drawings. The **Exclude border gaskets** setting is available for this purpose in the Configuration Editor under **Plant Engineering** > **Isometry and Pipe Spool Drawing**.

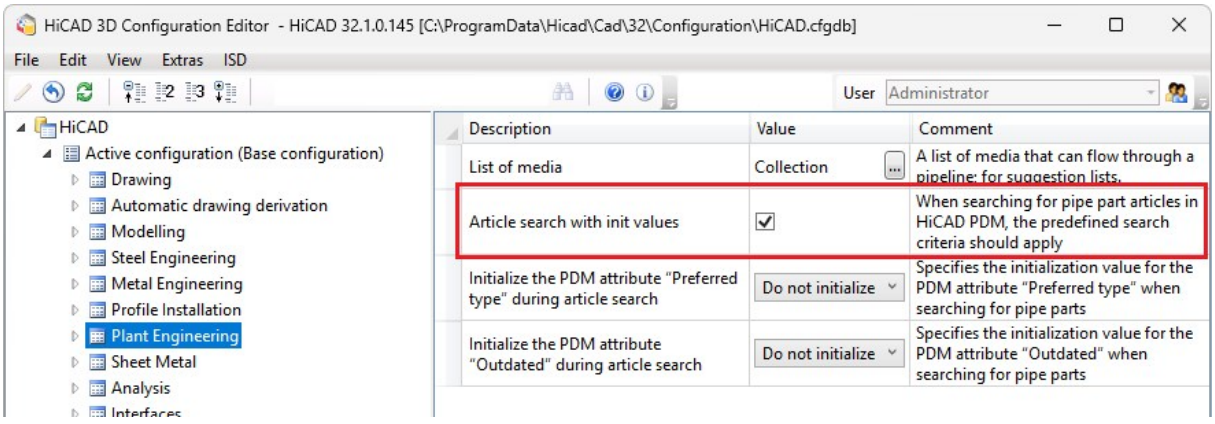
With this setting, border gaskets are handled as follows:

- Border gaskets are not subject to dimensioning.
- The connection coordinate is set to the adjacent connection.
- Border gaskets are not visible in the pipe spool drawing.
- Border gaskets do not appear in BOMs. Exception: border gaskets that are a part of a set of accessories.
- Border gaskets continue to be assigned item numbers.

HiCAD PDM

Enabling form pre-populations in plant engineering

The pipe part search in plant engineering now takes into account form pre-populations from HiCAD PDM. Previously, form pre-populations were ignored when searching for pipe parts. From HiCAD 3D 2027 onwards, they are used by default. This behaviour can be disabled in the Configuration Editor under **Plant Engineering** > **Article search with init values**.

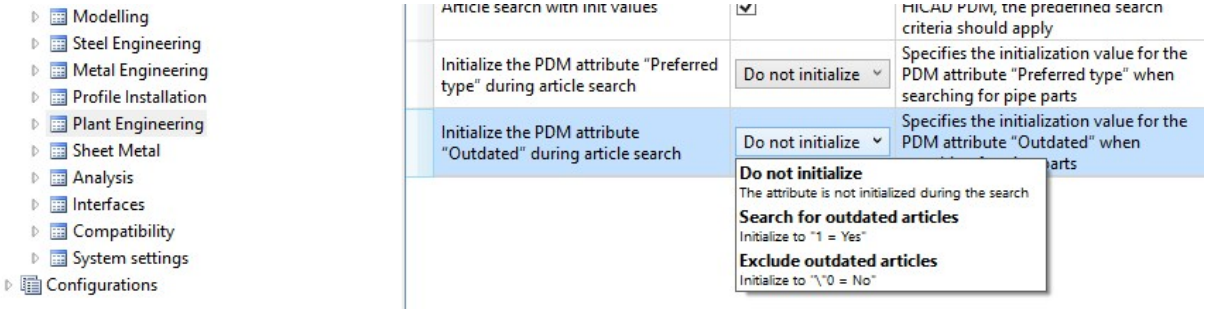


This setting only affects the search for pipe parts in plant engineering, for example during part insertion or auto-assignment. The general article search in HiCAD 3D remains unchanged.

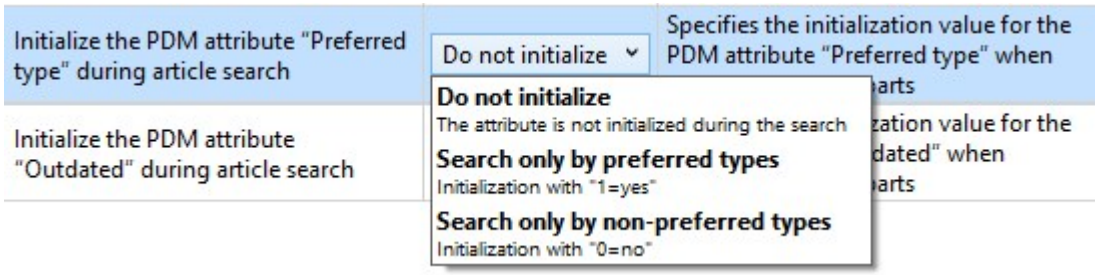
Marking pipe part articles as deprecated in HiCAD PDM

If you manage the articles for your pipe parts in HiCAD PDM, there may be instances where certain articles are still required to remain in the database but are no longer to be used in day-to-day work in HiCAD 3D. From HiCAD 3D 2027 onwards, the new **Outdated** (PLANT_DEPRECATED) attribute will therefore be added when preparing HiCAD PDM for plant engineering. This allows you to identify articles as deprecated in the article masks.

For part searches, you can specify in the Configuration Editor under **Plant Engineering** how the attribute should be taken into account during the item search. To do this, you can select a corresponding initialization or specify that the attribute should not be taken into account during the search.



Another new feature is a drop-down list for the **Preferred type** (VORZUGSTYP) attribute:



Previously, the preferred type could only be used during manual searches in HiCAD PDM. It can now also be automatically included in pipe part searches.

Importing standard packages for Plant Engineering

With the update to HiCAD PDM 2027, the previous method of importing standard templates for plant engineering via ParameterConfigAdministrator.exe is no longer required. Instead, the standard templates can be imported via the PDM Desktop. The **Import standard package...** function is available for this purpose.

A PDM administrator can make a standard package available to all or selected users as a central setting package.

PAA Editor

Insert FGA directly into PAA from the Symbols folder

In the **PAA Editor**, you can now assign an FGA file to a PAA as a symbolic representation. This requires that the FGA file and the associated EMF file are located in the relevant folder, for example in the *PlantParts/Symbols* folder.

Part data synchronization

Detect discrepancies between VAA/PAA files and HiCAD PDM

With the new PlantPartsPdmCompare.exe programme, you can compare the data in VAA/PAA files with the corresponding articles and documents in the HiCAD PDM database. To do this, the programme checks the VAA/PAA files referenced by the respective document masters and identifies discrepancies between the data.

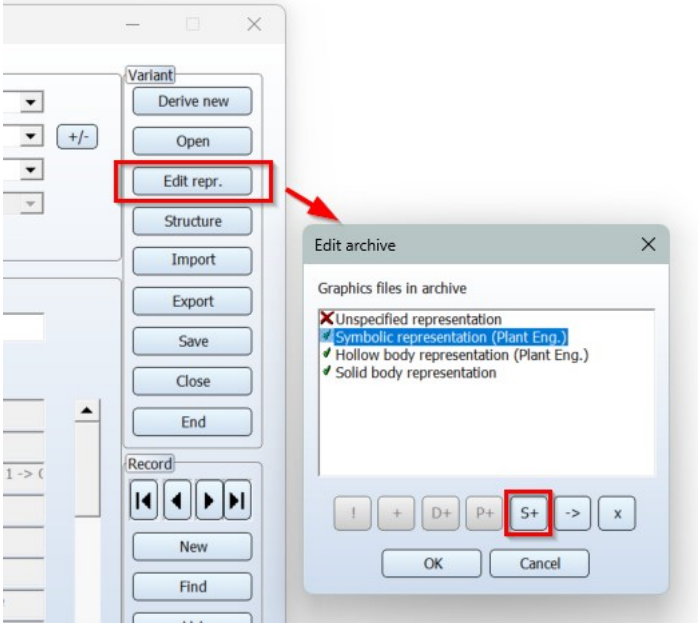
The results of the check are summarised in the console and also output in detail in a CSV file. The CSV file contains, amongst other things, details of the affected file, the record, the severity level and the differing attribute values, and can be analysed using Excel, for example. A text report is also generated. If required, the check can be restricted to a single VAA/PAA file or to a list of files.

Variant Editor

Major Release

Inserting FGA directly into VAA from Symbols folder

In the Variant Editor, you can now assign an FGA file to a VAA as a **Symbolic representation**. This requires that the FGA file and the associated EMF file are located in the relevant folder, for example in the *PlantParts/Symbols* folder.



Catalogue Editor

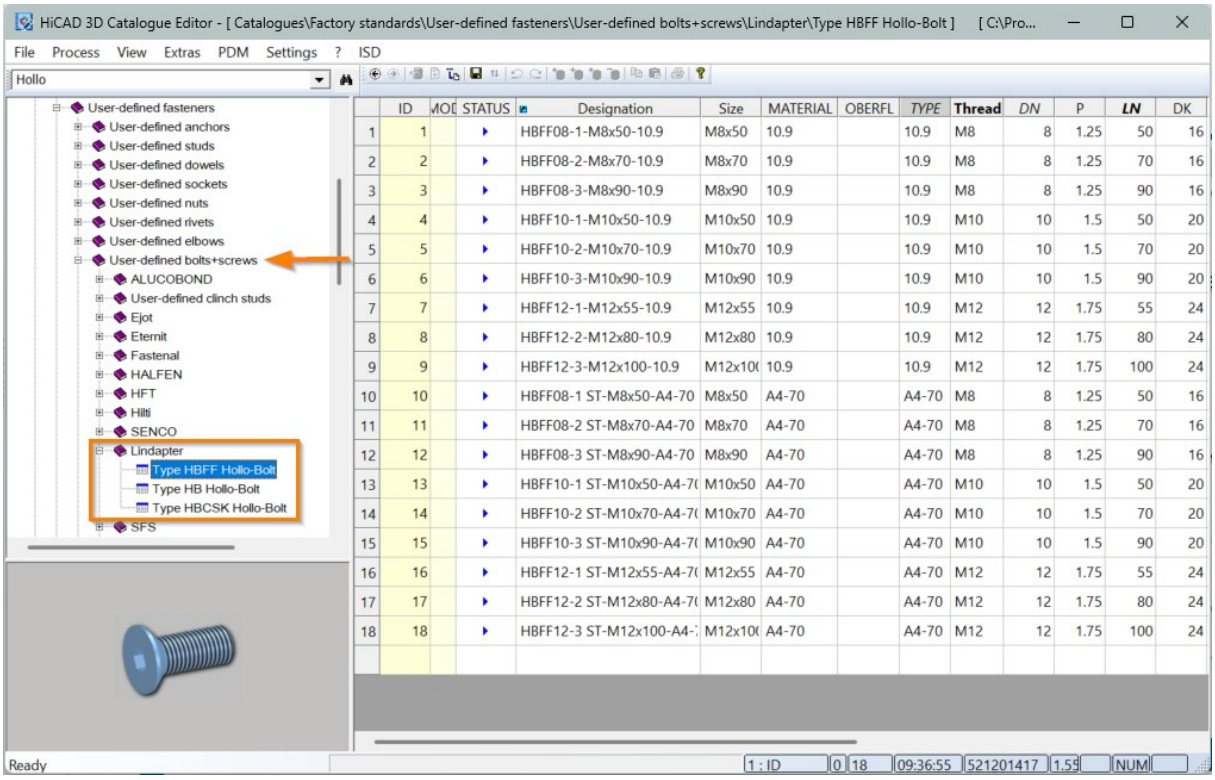
Major Release

Catalogue update: Lindapter Hollo-Bolt fasteners

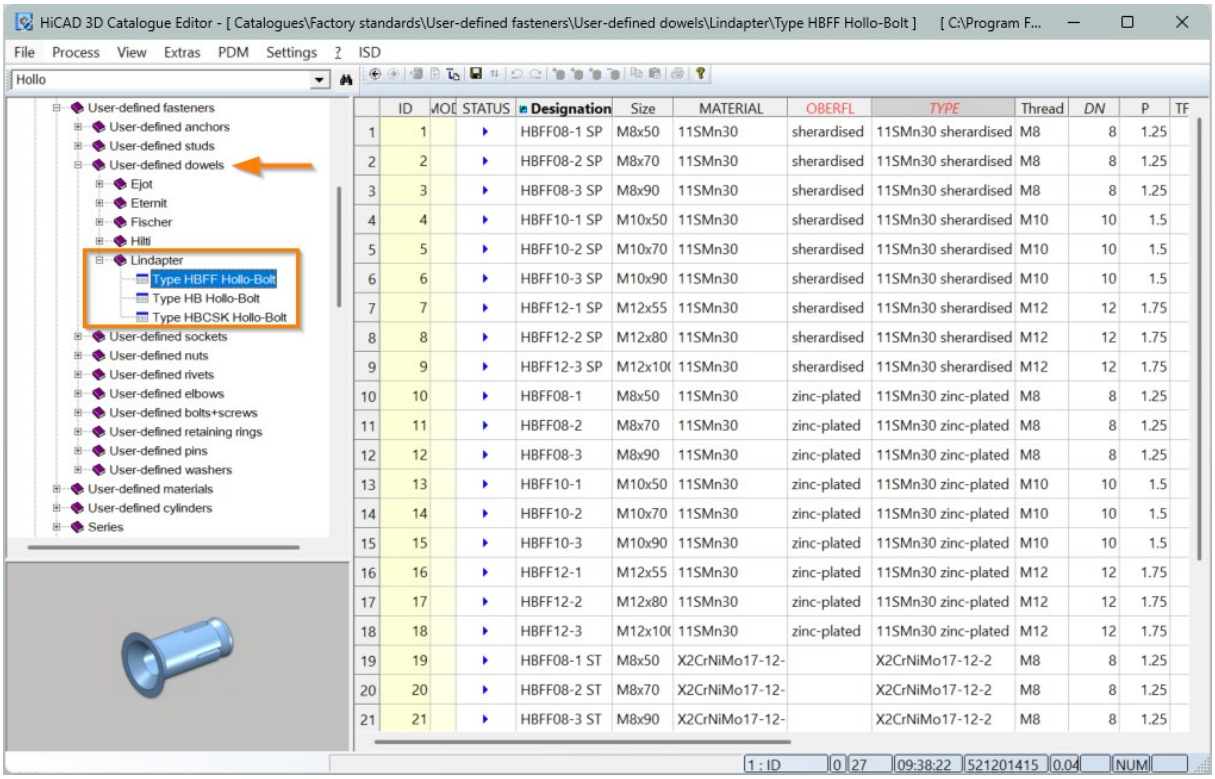
Lindapter's **Hollo-Bolt** fasteners have been added to the catalogue and can be inserted using the Bolting dialogue window. The three types available are HBFF, HB and HBCSK. The correct anchor is automatically selected for each bolt.

The bolts can be found at **Factory standards > User-defined fasteners > User-defined bolts+ screw > Lindapter**:

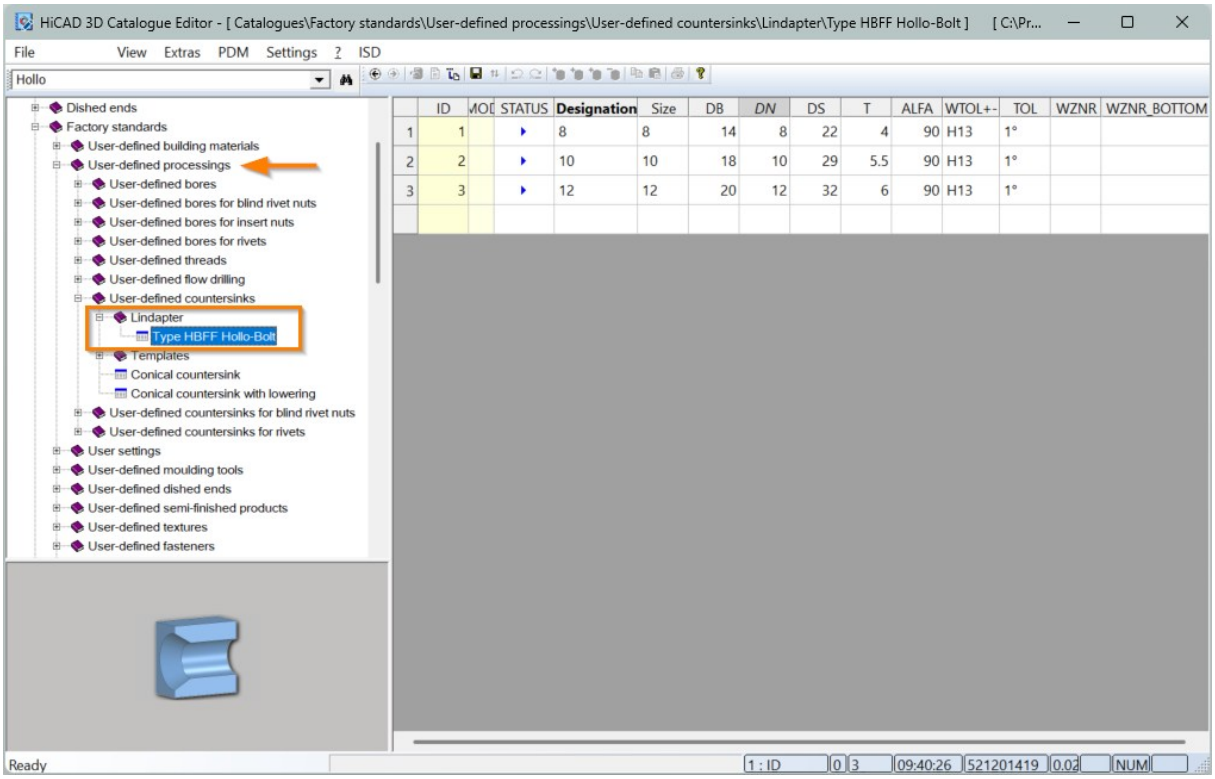
- Type HBFF Hollo-Bolt
- Type HB Hollo-Bolt
- Type HBCSK Hollo-Bolt



The corresponding dowels are available at **Factory standards > User-defined fasteners > User-defined dowels > Lindapter** and are automatically selected when the relevant bolt is chosen.



In addition, the Type HBFF Hollo-Bolt countersink has been added at **Factory standards > User-defined processings > User-defined countersinks > Lindapter**.



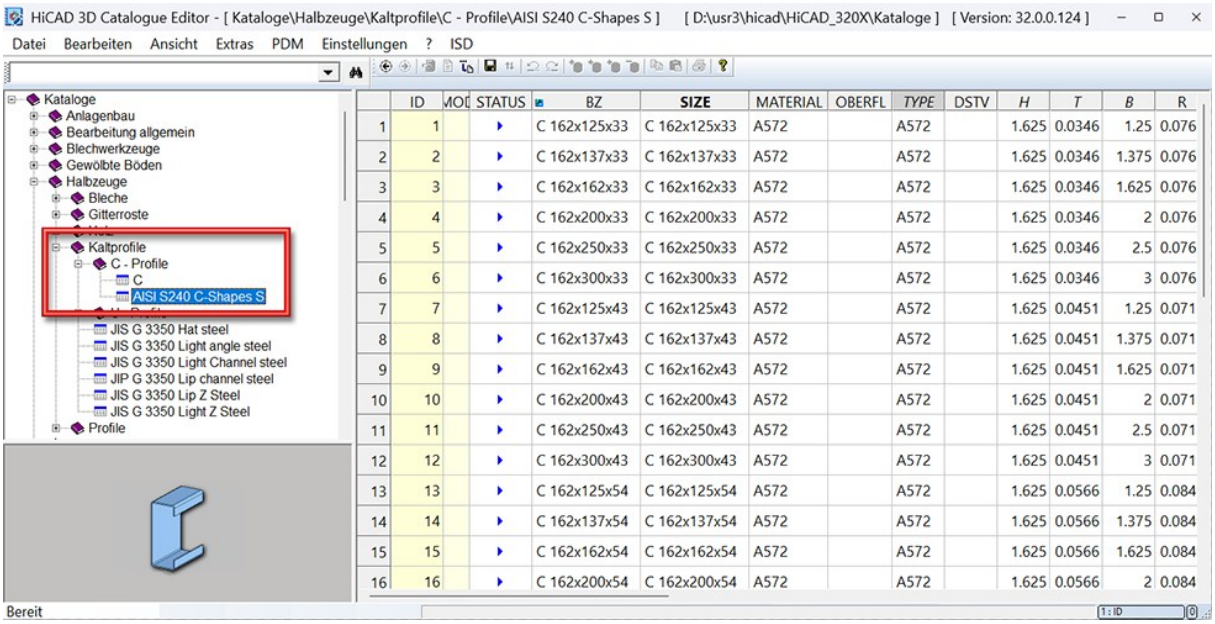
Preview images now as PNG

When saving parts to the catalogue, preview images are now generated as PNG files by default. Compared to the previous BMP format, these files require significantly less storage space. If required, the BMP format can still be selected manually.

Catalogue update: North American C-sections (AISI S240)

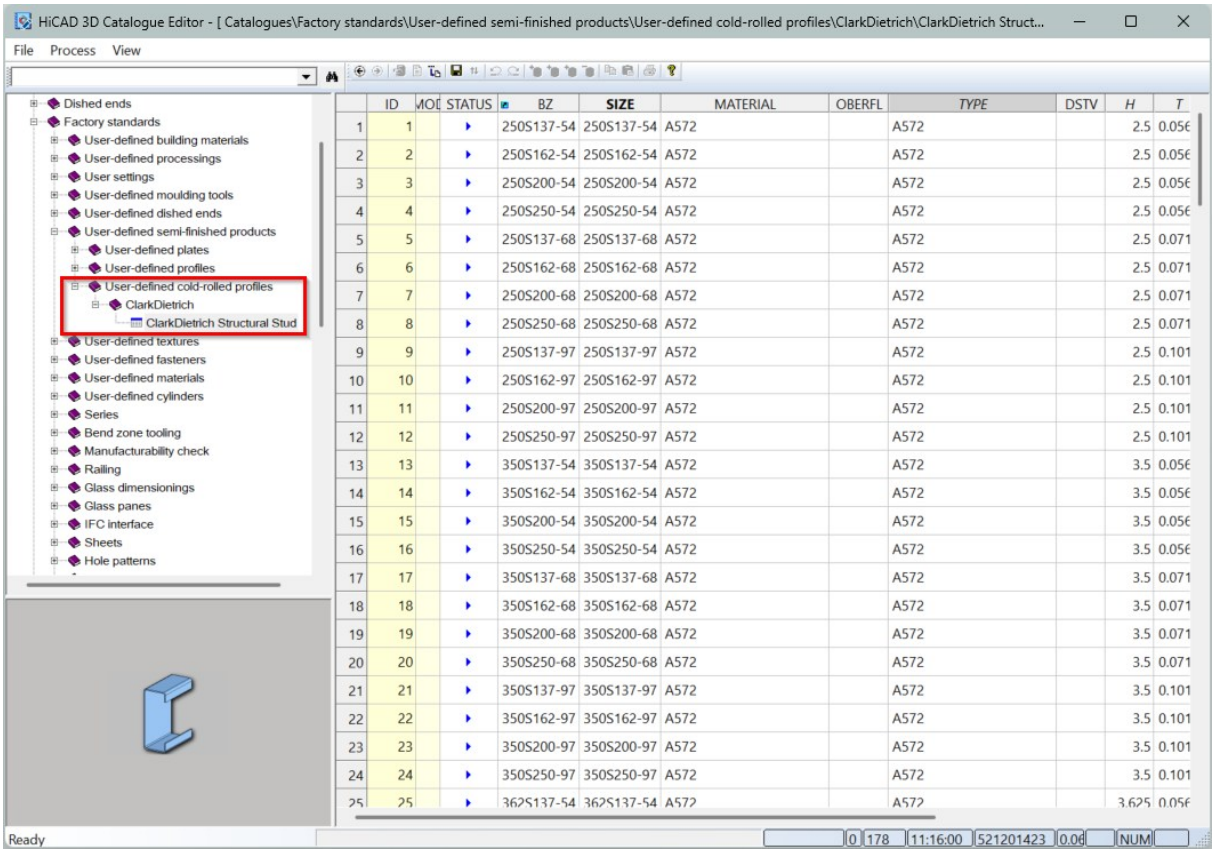
Für die nordamerikanische Kaltprofil-Bauweise (Structural Framing) wurden zwei neue Profiltabellen in den Katalog aufgenommen.

Die genormten C-Profile nach **AISI S240** (Cold-Formed Steel Structural Framing) inklusive der zugehörigen Bemessungswerte finden Sie unter **Semi-finished Products > Cold-rolled sections > C-beams > AISI S240 C-Shapes S**.



	ID	MO	STATUS	BZ	SIZE	MATERIAL	OBERFL	TYPE	DSTV	H	T	B	R
1	1			C 162x125x33	C 162x125x33	A572		A572		1.625	0.0346	1.25	0.076
2	2			C 162x137x33	C 162x137x33	A572		A572		1.625	0.0346	1.375	0.076
3	3			C 162x162x33	C 162x162x33	A572		A572		1.625	0.0346	1.625	0.076
4	4			C 162x200x33	C 162x200x33	A572		A572		1.625	0.0346	2	0.076
5	5			C 162x250x33	C 162x250x33	A572		A572		1.625	0.0346	2.5	0.076
6	6			C 162x300x33	C 162x300x33	A572		A572		1.625	0.0346	3	0.076
7	7			C 162x125x43	C 162x125x43	A572		A572		1.625	0.0451	1.25	0.071
8	8			C 162x137x43	C 162x137x43	A572		A572		1.625	0.0451	1.375	0.071
9	9			C 162x162x43	C 162x162x43	A572		A572		1.625	0.0451	1.625	0.071
10	10			C 162x200x43	C 162x200x43	A572		A572		1.625	0.0451	2	0.071
11	11			C 162x250x43	C 162x250x43	A572		A572		1.625	0.0451	2.5	0.071
12	12			C 162x300x43	C 162x300x43	A572		A572		1.625	0.0451	3	0.071
13	13			C 162x125x54	C 162x125x54	A572		A572		1.625	0.0566	1.25	0.084
14	14			C 162x137x54	C 162x137x54	A572		A572		1.625	0.0566	1.375	0.084
15	15			C 162x162x54	C 162x162x54	A572		A572		1.625	0.0566	1.625	0.084
16	16			C 162x200x54	C 162x200x54	A572		A572		1.625	0.0566	2	0.084

In addition, the manufacturer-specific C-profiles ‘Structural Stud’ from ClarkDietrich are available as a user table at **Factory standards > User-defined semi-finished products > User-defined cold-rolled profiles > ClarkDietrich > ClarkDietrich Structural Stud**.



	ID	STATUS	BZ	SIZE	MATERIAL	OBERFL	TYPE	DSTV	H	T
1	1		250S137-54	250S137-54	A572		A572		2.5	0.056
2	2		250S162-54	250S162-54	A572		A572		2.5	0.056
3	3		250S200-54	250S200-54	A572		A572		2.5	0.056
4	4		250S250-54	250S250-54	A572		A572		2.5	0.056
5	5		250S137-68	250S137-68	A572		A572		2.5	0.071
6	6		250S162-68	250S162-68	A572		A572		2.5	0.071
7	7		250S200-68	250S200-68	A572		A572		2.5	0.071
8	8		250S250-68	250S250-68	A572		A572		2.5	0.071
9	9		250S137-97	250S137-97	A572		A572		2.5	0.101
10	10		250S162-97	250S162-97	A572		A572		2.5	0.101
11	11		250S200-97	250S200-97	A572		A572		2.5	0.101
12	12		250S250-97	250S250-97	A572		A572		2.5	0.101
13	13		350S137-54	350S137-54	A572		A572		3.5	0.056
14	14		350S162-54	350S162-54	A572		A572		3.5	0.056
15	15		350S200-54	350S200-54	A572		A572		3.5	0.056
16	16		350S250-54	350S250-54	A572		A572		3.5	0.056
17	17		350S137-68	350S137-68	A572		A572		3.5	0.071
18	18		350S162-68	350S162-68	A572		A572		3.5	0.071
19	19		350S200-68	350S200-68	A572		A572		3.5	0.071
20	20		350S250-68	350S250-68	A572		A572		3.5	0.071
21	21		350S137-97	350S137-97	A572		A572		3.5	0.101
22	22		350S162-97	350S162-97	A572		A572		3.5	0.101
23	23		350S200-97	350S200-97	A572		A572		3.5	0.101
24	24		350S250-97	350S250-97	A572		A572		3.5	0.101
25	25		362S137-54	362S137-54	A572		A572		3.625	0.056

Report Manager

Discontinuation of the Report Manager Professional

The additional functionality of Report Manager Professional has been integrated into the 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.

Major Release

Isometry BOMs: %PART_COUNT additionally to %PART_QUANT

The expression **%PART_QUANT** allows you to specify the quantity of pipe parts in bills of materials. Straight pipes are treated differently, as the total length is displayed for them rather than the number.

From HiCAD 2027 onwards, **%PART_COUNT** is also available. This expression does not apply the special treatment to straight pipes; these are therefore simply counted.

Configuration Management

Major Release

Dynamic zoom: Parameters removed from the Configuration Editor

The parameters for the dynamic zoom and zoom increments are workstation-specific. They are therefore no longer managed in the Configuration Editor. At **System settings > Visualization**, the following parameters have been removed:

- **Zoom factor**
- **Zoom size**
- **Incremental zoom factor**
- **Incremental translation speed**

Instead, you can set these values as before using the transparent toolbar (right-click on total view > **Zoom factor** or **Zoom parameters**). These settings are now saved permanently and on a per-user basis and are retained even after restarting HiCAD 3D.

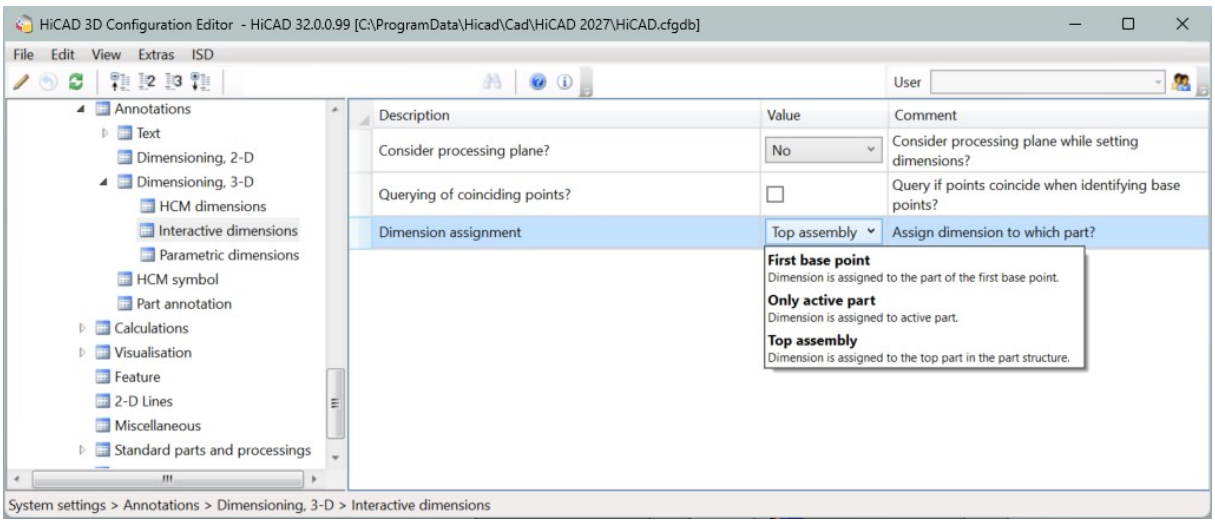
In addition, the zoom default settings have been adjusted:

- Enlargement factor: 20
- Aspect ratio (magnifier size): 0.75

The factors for step-by-step zooming and moving remain unchanged.

Dimension assignment

Previously, Dimension assignment in the Configuration Editor only affected new drawings. Starting with HiCAD 3D 2027, however, it applies globally to all drawings..



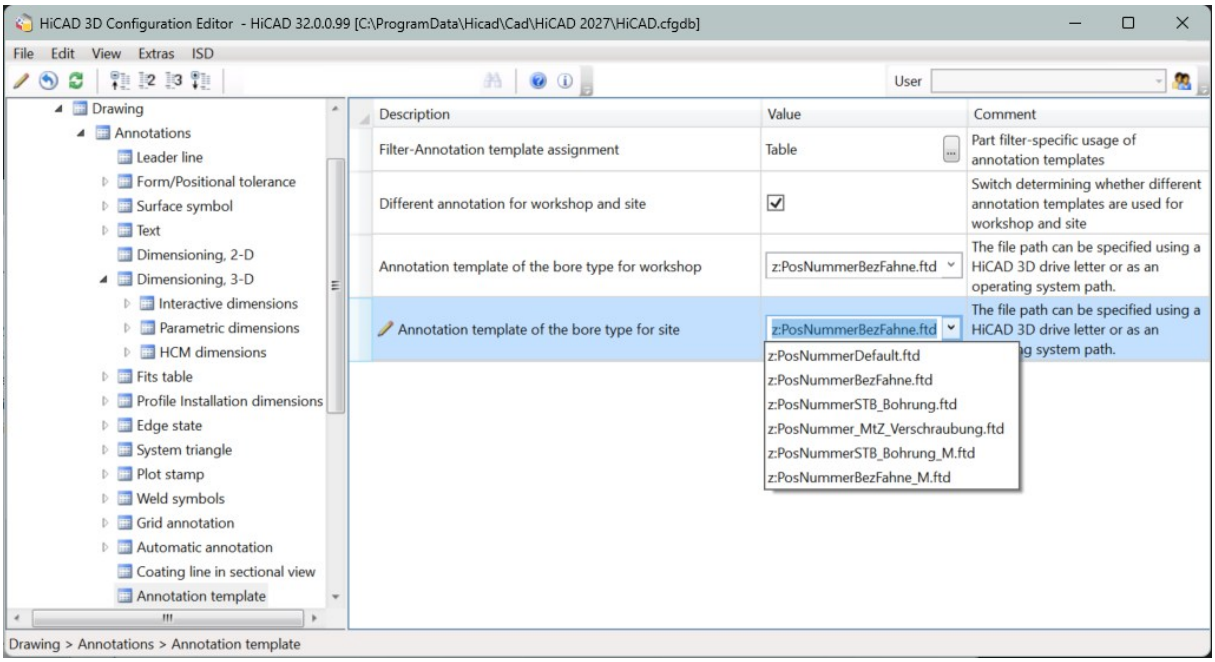
Renaming of the ISDConfigEditor

The EXE file for HiCAD 3D's Configuration Editor has been renamed from CadConfigEditor.exe to CadConfigEditor.exe.

Bore specification for sites

There is a new annotation template for processings of the type **Site**, which reads out the manufacturing type. To load the new template in the Configuration Editor at **Drawing > Annotations > Annotation template** for the parameter **Annotation template of the bore type for site**, enter the drive letter from the FILEGRUP. DAT and the name of the template (*.ftd).

z:PosNummerBezFahne_B.ftd



Direct start of the Configuration Editor

Starting with HiCAD 3D 2027, the HiCAD 3D Configuration Editor (CadConfigEditor.exe) can be launched using the **-select** startup parameter. This opens the Configuration Editor and automatically navigates to the desired setting in the tree structure—you no longer need to search for the node manually.

Command:

CadConfigEditor.exe-select "<node>"

The node is the key name of the setting; the planes of the tree structure are separated by a period, and the path is enclosed in quotation marks.

Example:

"C:\Program Files\ISD\HiCAD\exe\CadConfigEditor.exe"-select "SYSTEM.Referencing"

This command opens the editor and directly highlights the 'Referencing' node under "System Settings." The program is located in the exe directory of the HiCAD installation, by default at C:\Program Files\ISD\HiCAD\exe.

You can drop this command from any program—for example, via an AI assistant that guides you directly to the appropriate setting.

If the specified node does not exist, the editor starts normally without jumping to it.

Manufacturability check during automatic drawing derivation

In the Configuration Editor, at **Sheet Metal > Manufacturability check**, the new parameter Perform manufacturability check when creating automatic drawing derivations is available. It controls whether a manufacturability check is performed for the folded sheet metal parts contained in the drawings when automatically deriving or updating shop drawings.

If the parameter is enabled, each sheet metal part in the part list is checked for manufacturability based on the criteria set at **Sheet Metal > Manufacturability check**. If a sheet metal part is identified as unmanufacturable, a corresponding message appears in the drawing derivation report window.

Colouring tangential edges

Using the new **Colour of tangential edges** function under **Views > Properties > Elements**, you can assign a fixed selected colour, the face colour or the solid edge colour of the respective part to each view.

The setting selected in the function dialogue box is saved with the drawing and is applied when a view is copied. In the Configuration Editor, you can set a default at **Drawing > Views > Display of special lines > Tangential edges**.

Restore interim backups

The previous option "Keep drawing-related data in cache when switching drawings" in the Configuration Editor at **System settings > Miscellaneous** is now called **Automatically create a backup when switching drawings** and can be found at **System settings > Data save**. If this option is enabled, a complete, restorable backup is created every time the design is changed.

Captions for view groups in drawing derivations

If you use the existing drawing derivation function, you will find the relevant settings in the **View group settings** dialogue window, in the new **View group caption** section. In the Configuration Editor, the corresponding parameters are available as default settings at **Automatic drawing derivation > Production drawing > Usage-dependent >** *<Usage>* **> View group** in the **View group captions** section.

Plant Engineering

Making HiCAD PDM search mask initializations usable in Plant Engineering

Up to HiCAD 2026 SP2, the PDM system's search mask initializations were ignored when searching for pipe parts. This prevented initializations from other HiCAD 3D areas – in particular, the steel engineering drawing management – from making it impossible to find pipe parts.

From HiCAD 3D 2027 onwards, the initializations are applied by default, but can be deactivated in the Configuration Editor at **Plant Engineering > Article search with init values**.

Naming of pipeline sections

At **Plant Engineering > Layout plan**, you can configure the **Prefix for naming pipe sections** before the counter. If no value is specified, the naming convention is, for example, **Pipeline_Section_000X**.

Marking pipe part articles as outdated in HiCAD-PDM

From HiCAD 2027 onwards, the new **Outdated (PLANT_DEPRECATED)** attribute is available. You can use this to mark articles as outdated in the article masks. Plant Engineering supports the use of this attribute in the part search.

Therefore, if you wish to exclude articles marked as outdated, select **Exclude outdated articles** from the drop-down list at **Plant Engineering** for the setting **Initialize the PDM attribute "Outdated" during article search**.

Another new addition here is a drop-down list for the setting **Initialize the PDM attribute "Preferred type" during article search**.

Expand dummy fittings

In the Configuration Editor, at **Plant Engineering > Layout plan > Editable variables of placeholder valves**, you can specify which variables should be offered as editable during the insertion of dummy fittings.

Isometry and Pipe spool drawing

Consistent symbol size in isometric drawing

Within an isometric drawing, it is straightforward to standardise the symbol size across all sheets. This applies to both the selected scale and the elbow symbols.

Two new settings for the symbol scale have been created in the Configuration Editor at Plant Engineering > Isometry and Pipe Spool Drawing:

- **Uniform symbol scale across drawing sheets**
- **Uniform symbol scale**

In the Configuration Editor, a setting for elbows has been introduced which allows a uniform elbow symbol size to be achieved whilst using pipe shortening:

- **Adjust size of elbow symbols to the base unit of the pipe shortening**

P&ID

Additional attributes for text objects

From HiCAD 3D 2027 onwards, you can extend the list of text flags with your own expressions. To do this, define them in the Configuration Editor at **Plant Engineering > P+ID > Additional expressions for text objects**.

Viewer

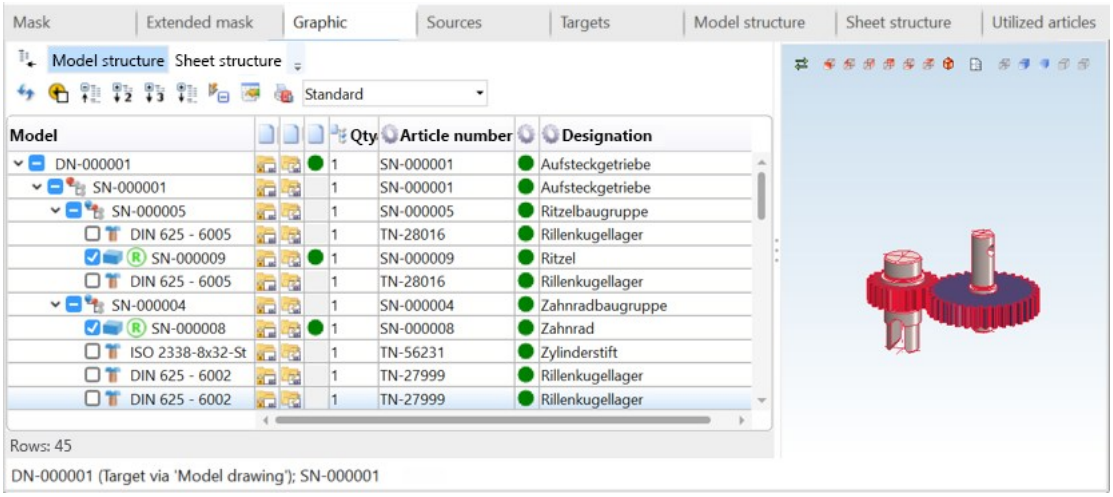
Major Release

Improvements to the PDM interface

The HiCAD 3D Viewer is integrated into HiCAD PDM Desktop, enabling drawings to be assessed quickly and clearly – even within the desktop interface.

With the update to HiCAD 3D 2027 and HiCAD PDM 2027, a number of improvements and adjustments have been made in this area.

By activating or deactivating checkboxes, geometries (individual parts or assemblies) can be shown or hidden in the model structure tree.



It is also possible to show several individual parts or assemblies side by side.

The colours and thickness of the graphical markings in the viewer can be adjusted via the Windows Registry if required.

Menu items not required in the PDM Desktop are also hidden in the Viewer view.

Notes on HiCAD PDM 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 installing an update from an older version to HELiOS 2025 (from 2027: HiCAD PDM, version 32.0.0) or higher, please note that the directory structure will change (again). As automated migration is not possible, all users must (and, when updating to HiCAD PDM 2027, may need to do so repeatedly) check out all data and clear their workspaces before installing the update to avoid data loss.

In earlier versions, the workspaces were located under %localappdata%. This meant that different workspaces could exist on a single system. To address this, the workspaces were moved to the **%programdata%** directory.

Checked-out files are then located under **%programdata%\Hicad\Pdm\Workspace\(...)***\ (*plus Location ID and User ID). Depending on the version, the workspace databases will be located under **%programdata%\Hicad\Pdm\32.0\General\Locations**.

Notes on update installations

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

Since conflict may occur during the update process with **PdmSetup.exe** 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 HiCAD PDM database.
  For the backup, either use the Database Import & Export (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 conflicts occur during the update, they will be recorded in the log file PdmUpdate.txt (located in the system path **%AppData%\Hicad\Pdm\<Major.Minor>\Logging\PdmUpdate**

Please have this file ready if you contact the ISD Hotline to troubleshoot and perform a successful update in the event that the update does not complete without errors.

- **MultiCAD interfaces**

  If you are working with a multi-CAD interface (e.g. the HiCAD PDM-Inventor interface), please note that before installing an update of an older version to HiCAD PDM 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.

- **Silent installations**

  When updating from older versions to HiCAD PDM 2027 or later as part of a so-called silent installation, please note that existing Setup.ini files may need to be amended or recreated.

HiCAD PDM Desktop

Major Release

Templates and Template files

Templates, together with **Template files**and **Templates for attribute value assignment** , form the three building blocks of HiCAD PDM's new initialization concept.

A template can include attribute value assignments (rules that automatically populate attributes when objects are imported into the database), a template file (sample files from which new PDM documents are created), and other initializations (workflow, classification) can be combined so that they can be applied with a single click when entering objects via the template selection

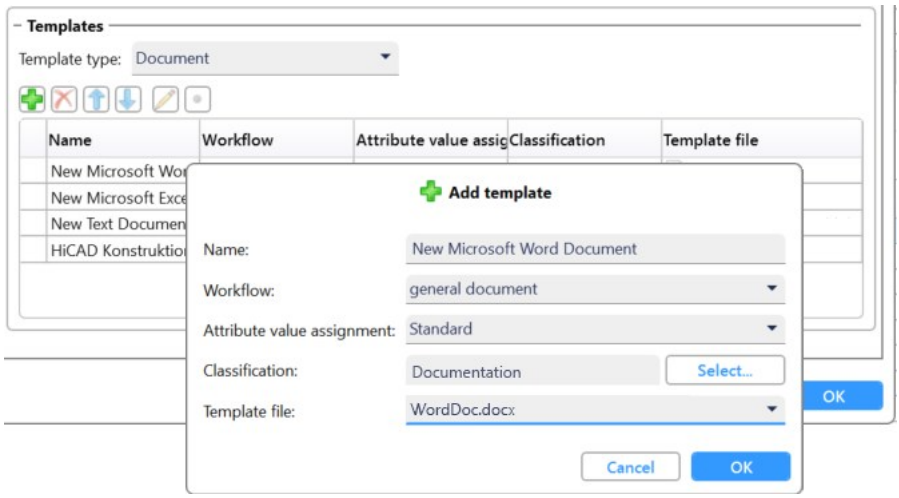
A template file, in turn, is a specific file that serves as a template for new documents. For example, a blank drawing sheet, a Word form, or any other template for a specific application (e.g., HiCAD 3D or MS Office). If a template file contains, for example, a style or a graphic, this is carried over into all documents created in HiCAD PDM based on that template file. A "Document" type template can reference a template file, but does not have to.

Templates and template files, just like attribute mappings, can be created on a user-specific basis or used system-wide with the ability to export and import UI settings and PDM Option settings.

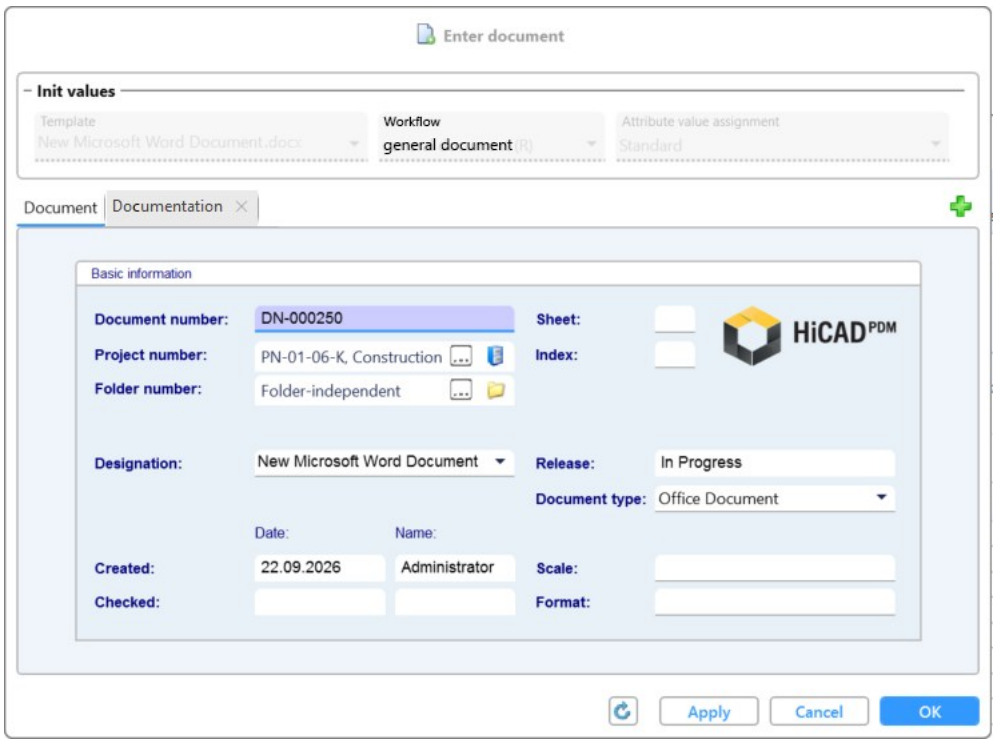
An example:

A template file named "WordDoc.docx" is, in and of itself, simply an empty Word file that has been imported into the PDM system. Only a predefined template—for example, one named "DocumentTemplate_Docx"—that references this file and additionally sets the "Document" workflow (as well as, if applicable, the "Standard" attribute value assignments and a classification) turns it into a complete initialization that you can apply with a single click when entering a document.

In the PDM Options, the template setup looks something like this:



After selecting this sample template during document entry, the initializations are then applied accordingly in the form, including a tab for the configured classification:



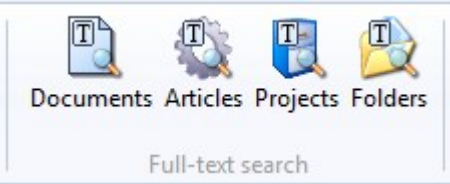
Please also note the instructions on how old “sample files” from HELiOS are converted into templates with template files during update installations.

Full-text search

If you use the **Full-text search**, the update to HiCAD PDM 2027 provides you with a number of enhancements and adjustments.

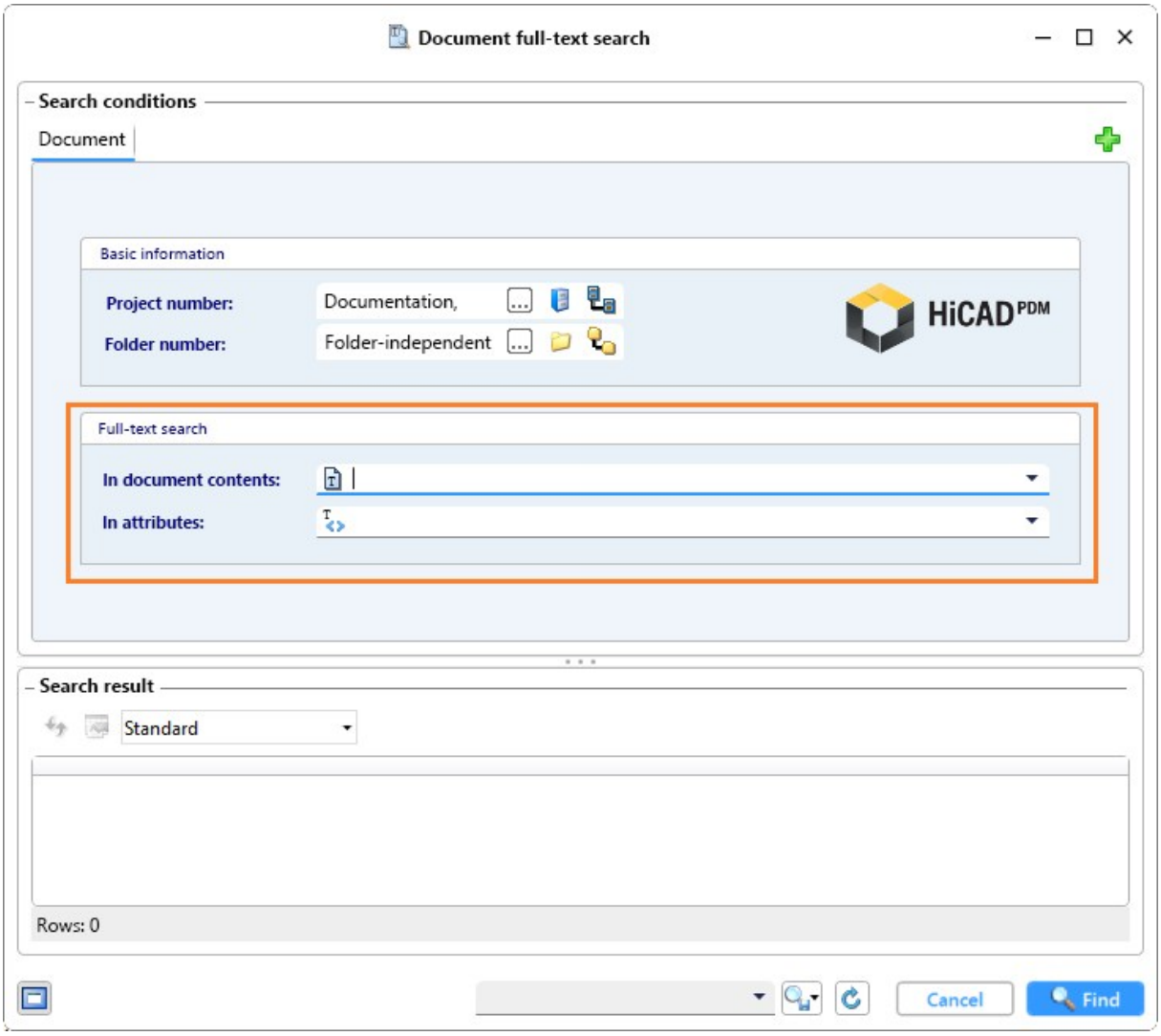
When using the full-text search, automatic spell-checking is disabled to ensure that only exact matches are returned. (However, the internal dictionary continues to be populated during indexing. This means that when spell-checking is subsequently enabled, the indexed terms will be available as possible matches without the need to re-index.)

Full-text search for Articles, Projects, and Folders

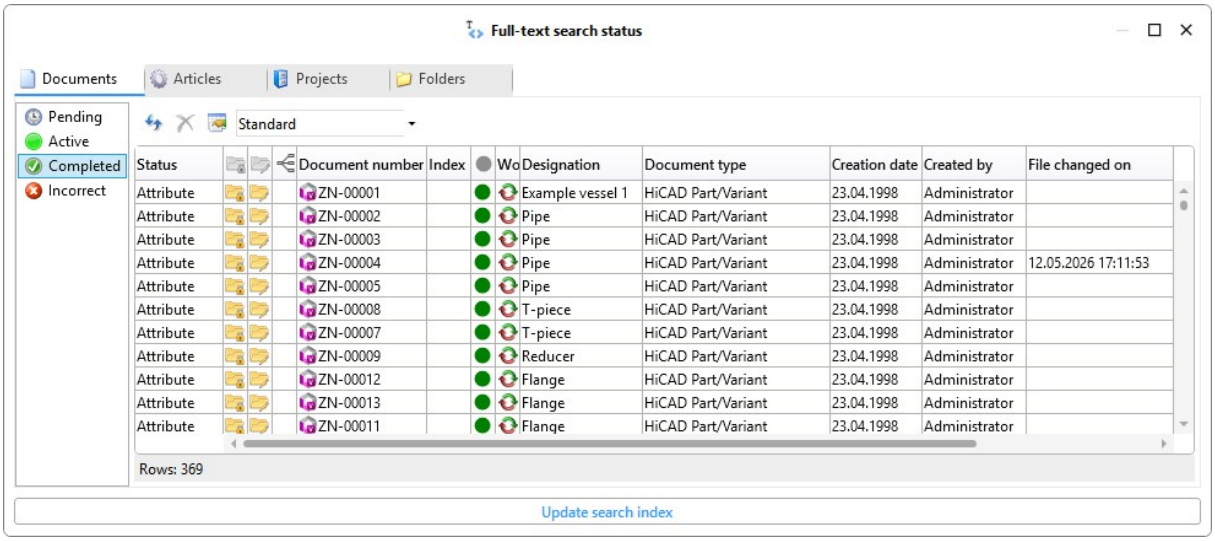


The full-text search mechanism has been expanded to allow searching not only of documents but also of articles, projects and folders within HiCAD PDM. In addition to document file contents, the master data for the respective PDM object types is also indexed by the full-text search.

In the case of documents, a distinction can be made between a full-text search **In document contents** and a **In attributes**:



You will now also find separate tabs for each PDM object type in the window for updating the search index:



Wildcard placeholders

You can also use the wildcard placeholders ***** and **?** when entering search terms. Following an adjustment to the program logic, this works as follows.

- The '*****' wildcard can appear at any position (including at the beginning or end) in the search term to represent zero, one, or more additional characters in the otherwise fully spelled-out search term.

  (Example: A search for “*Document*” would include “Document,” “Documents,” or “(the) Document” just as it would “PDM document search mask.”)

- The wildcard `**?**` stands for exactly one unknown character at that position in the search term.

  ((Example: A search for “Document?” would include “Documents,” but not "Document" or “(of the) document.”)

- It is possible to combine both in a single search query.

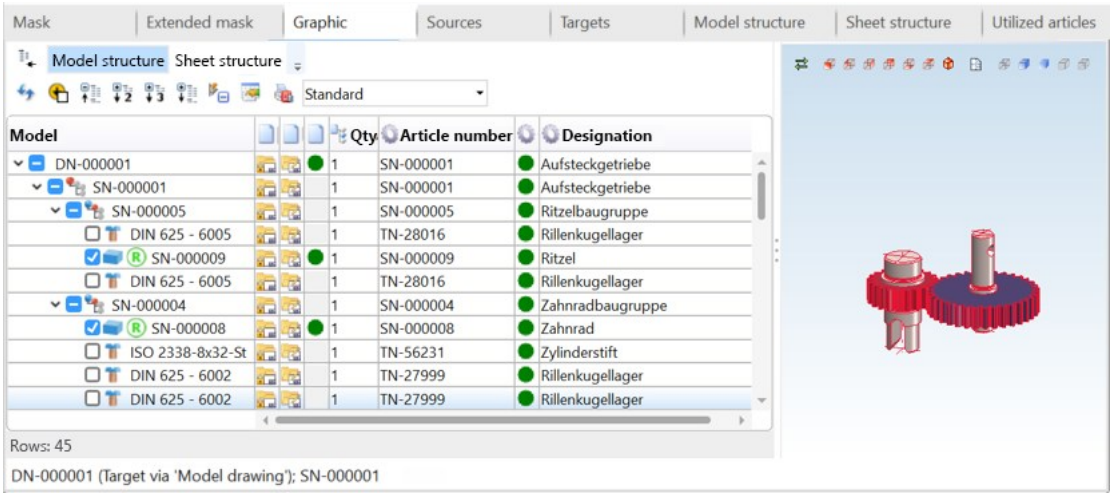
  ((Example: A search for “*Document?” would include "Documents" or “PDM documents,” but not “Document,” “(of the) document,” or “PDM document search mask.”)

HiCAD 3D Viewer

The **HiCAD 3D Viewer** is a tool that allows you to quickly and clearly evaluate HiCAD 3D drawings and models–even when integrated into the HiCAD PDM Desktop interface.

With the update to HiCAD 3D 2027 and HiCAD PDM 2027, several improvements and adjustments were made in this area.

By selecting or deselecting checkboxes, geometries (individual parts or assemblies) can be shown or hidden in the model structure tree.



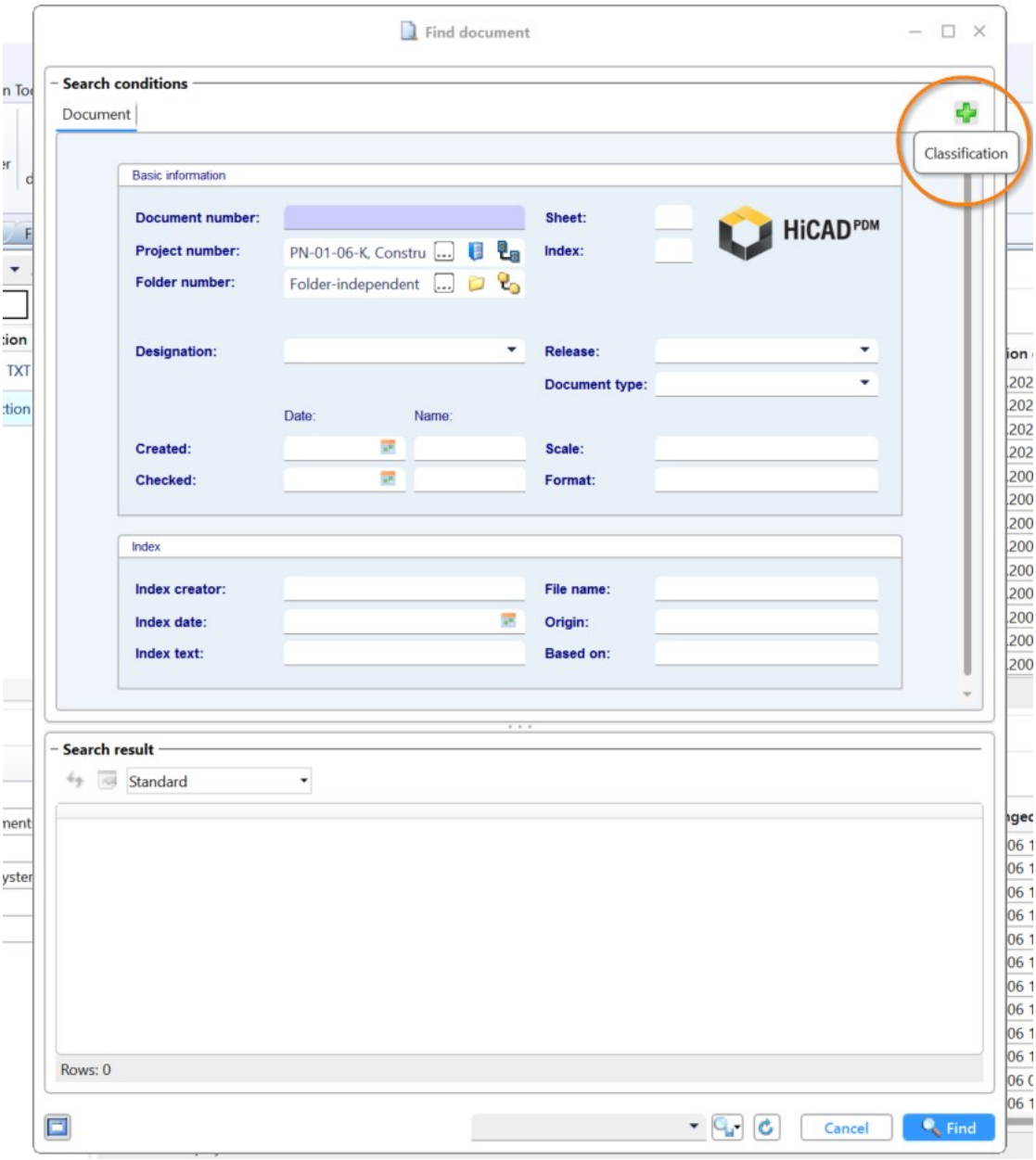
Multiple individual parts or assemblies can also be shown side by side.

The colours and thickness of the graphical markings in the viewer can be adjusted via the Windows Registry if necessary.

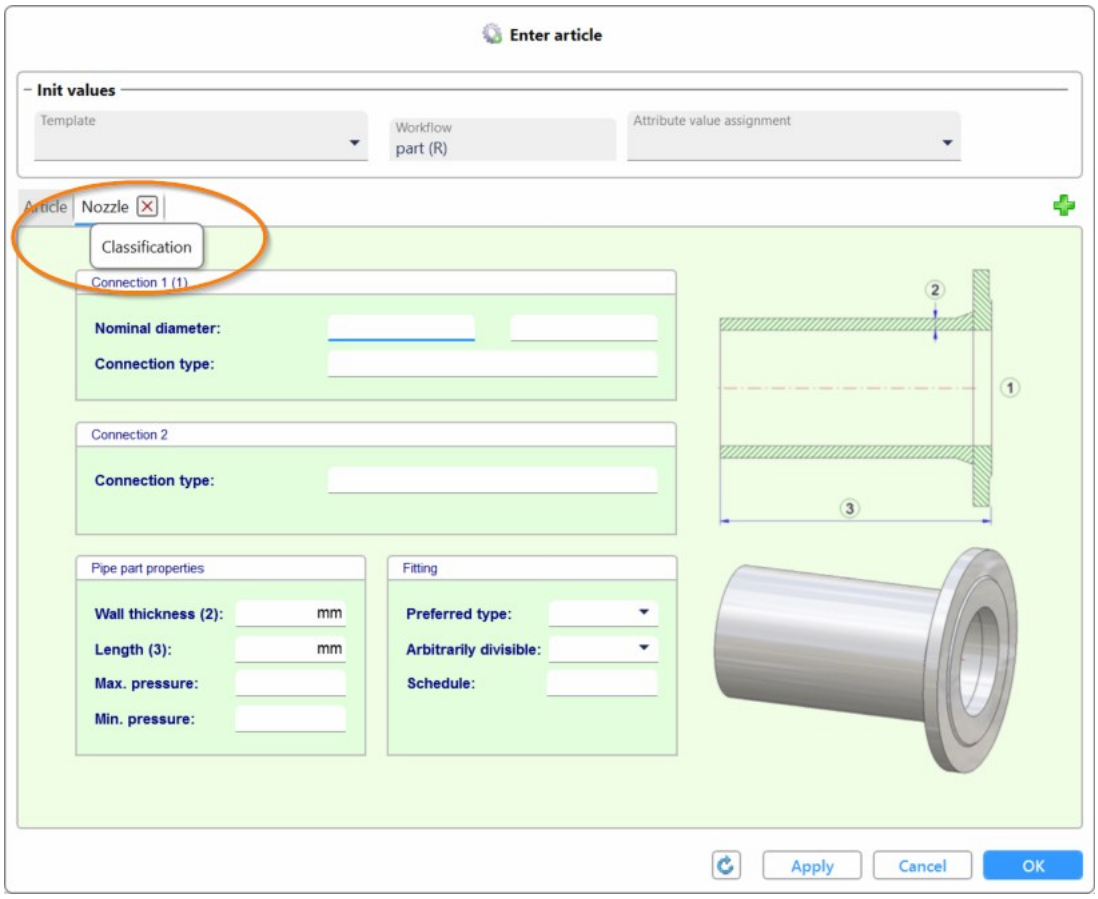
Additionally, HiCAD 3D menu items that are not required in the PDM Desktop are hidden in the viewer.

Classification

With the update to HiCAD PDM 2027, you can access the Classification in Input and Find masks via the  icon in the top right-hand corner of the window.



Furthermore, the classification tabs can now be closed very easily by clicking the **x** on the tab in the screens.



Link classes

New system links during database update

When updating the database to the latest version of HiCAD PDM, any missing link classes in existing databases are added as system links.

There are a total of 72 system link classes. Newly added classes are not active initially (see below).

New attributes for link classes: ACTIVE and APPLICATION

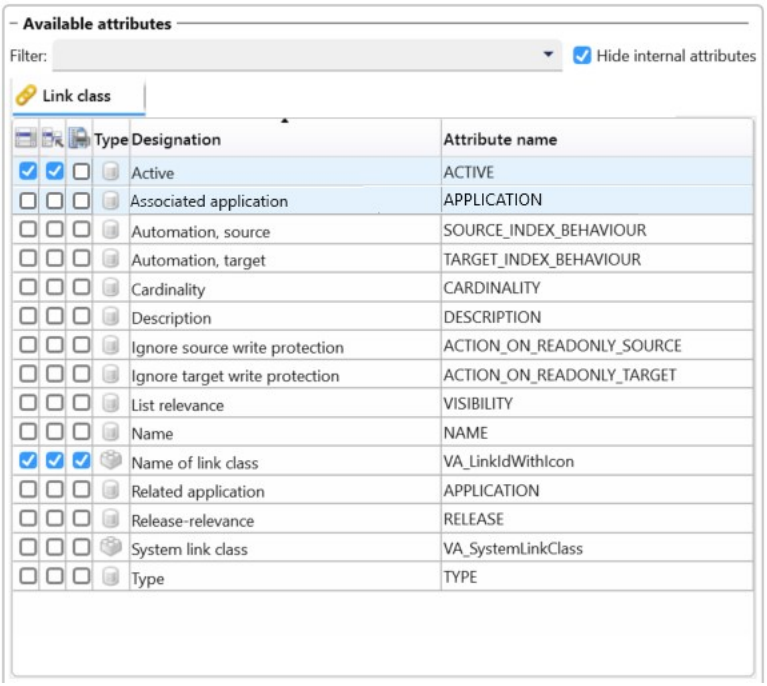
There are two new attributes for these link classes:

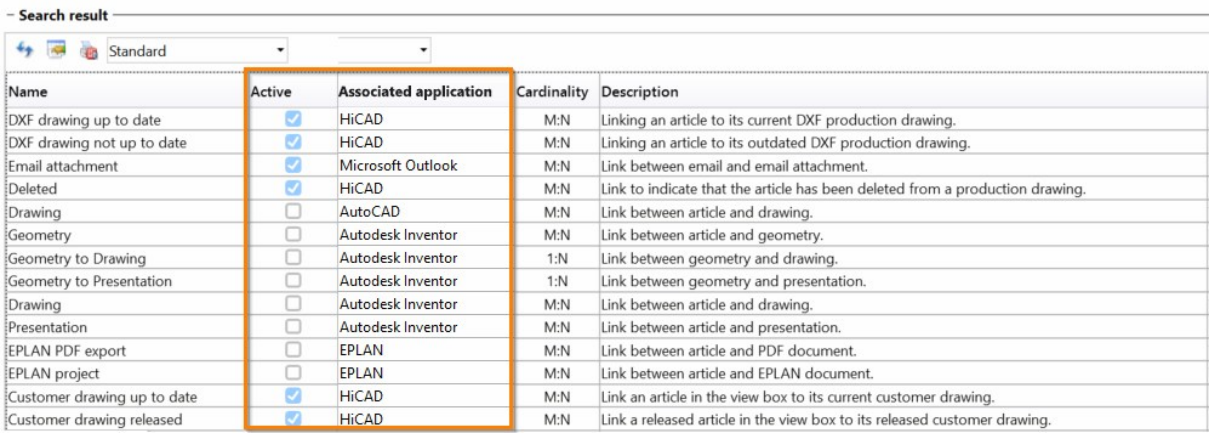
- **Associated application (APPLICATION)** identifies the application for system link classes that uses the corresponding link class. This is automatically assigned by the system and cannot be changed by the user.

  Whether this is displayed in the user interface can be explicitly enabled or disabled in the PDM Options (see below).

- **Active (ACTIVE)** indicates whether a link class should be actively used.

  If this is not the case, the corresponding link class will not be offered as a selection option in the link dialogues (this can be changed by the user. However, link classes that are already in use cannot be set to inactive again).

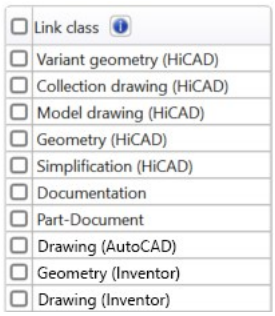




PDM Options > Display > Show associated application

The new PDM Option **Show associated application** on the **Display** tab also allows you to control whether the respective application associated with the link class is displayed when selecting link classes.

The following is an example from a section of the combined search:

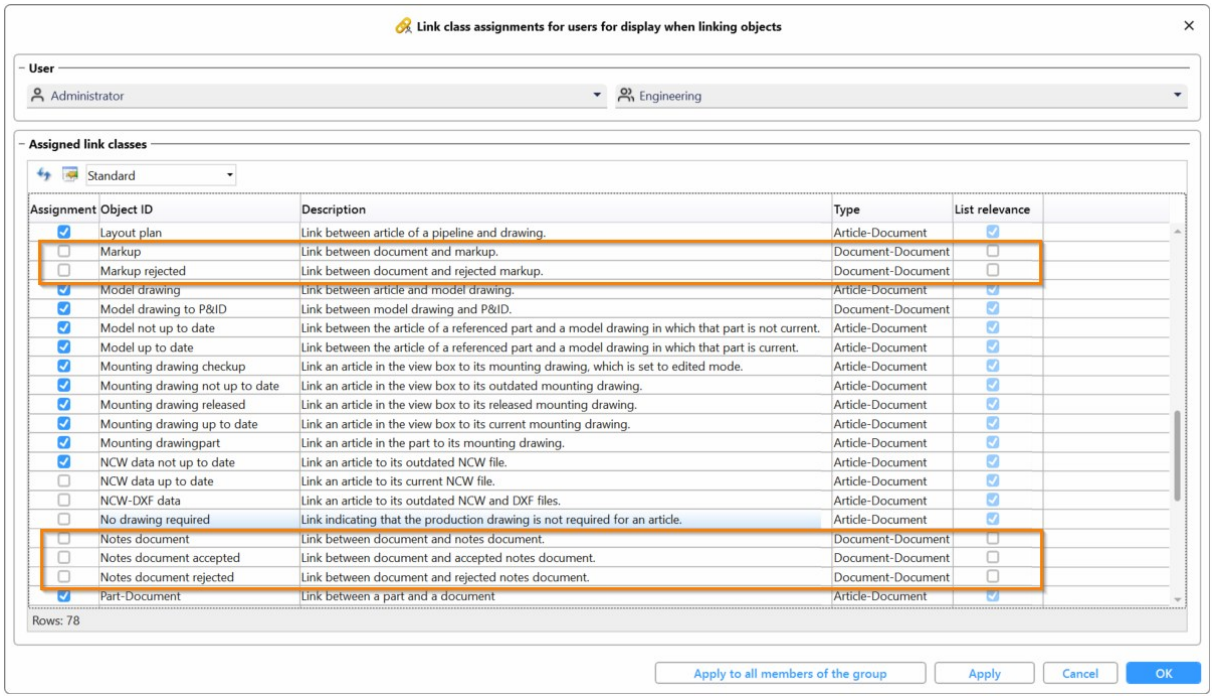


This can be useful, for example, if different link classes have the same displayed name (such as the “Drawing” link class for AutoCAD and Inventor files in the example).

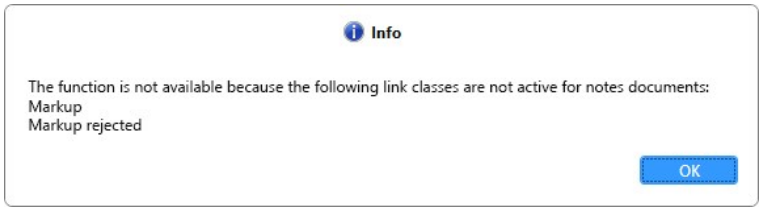
By default, this option is not enabled.

Markup and Notes documents

Please note that when working with the Markup or Notes document functionality, these are provided via link classes.



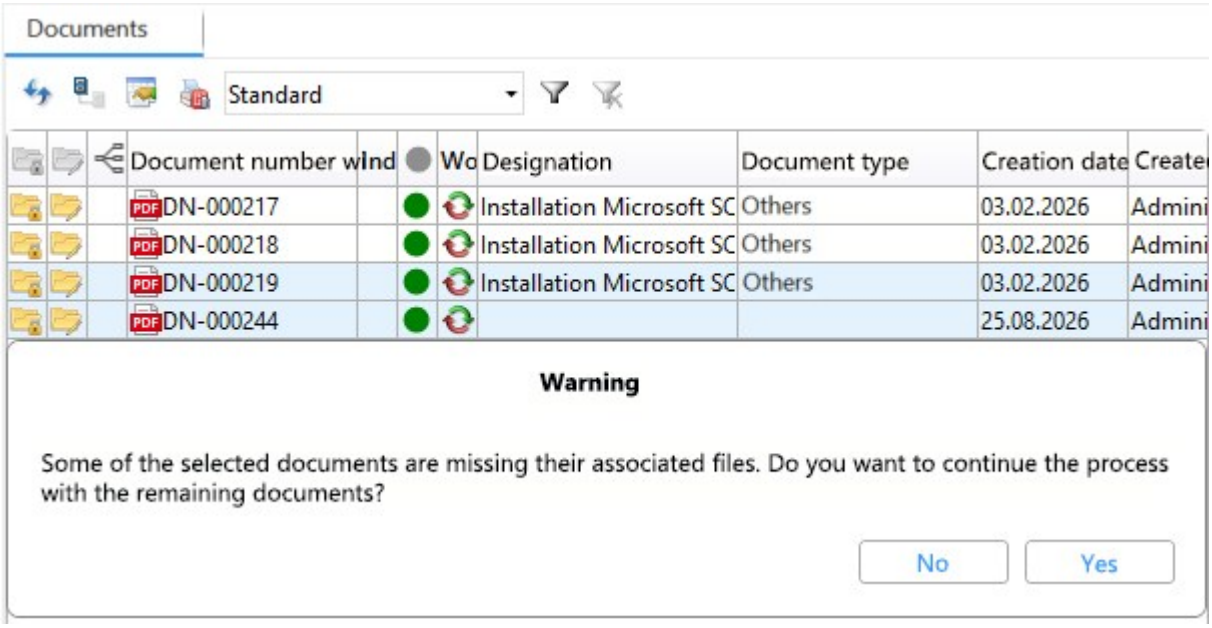
By default, the link classes are inactive in the HiCAD PDM delivery database.



Print and Convert: Query if files are missing

It may happen that there is no corresponding file in the system for certain document entries in the PDM database.

If any such documents are included amongst the selected documents when you call up **Print** or **Convert file** , you will receive a warning message asking for confirmation.



Click **No** to cancel the process, or **Yes** to filter out the documents without files and continue with the remaining documents.

Model structure

In the detail masks of HiCAD PDM document master records, a detailed **Model structure** view is displayed for **HiCAD 3D** and Inventor files.

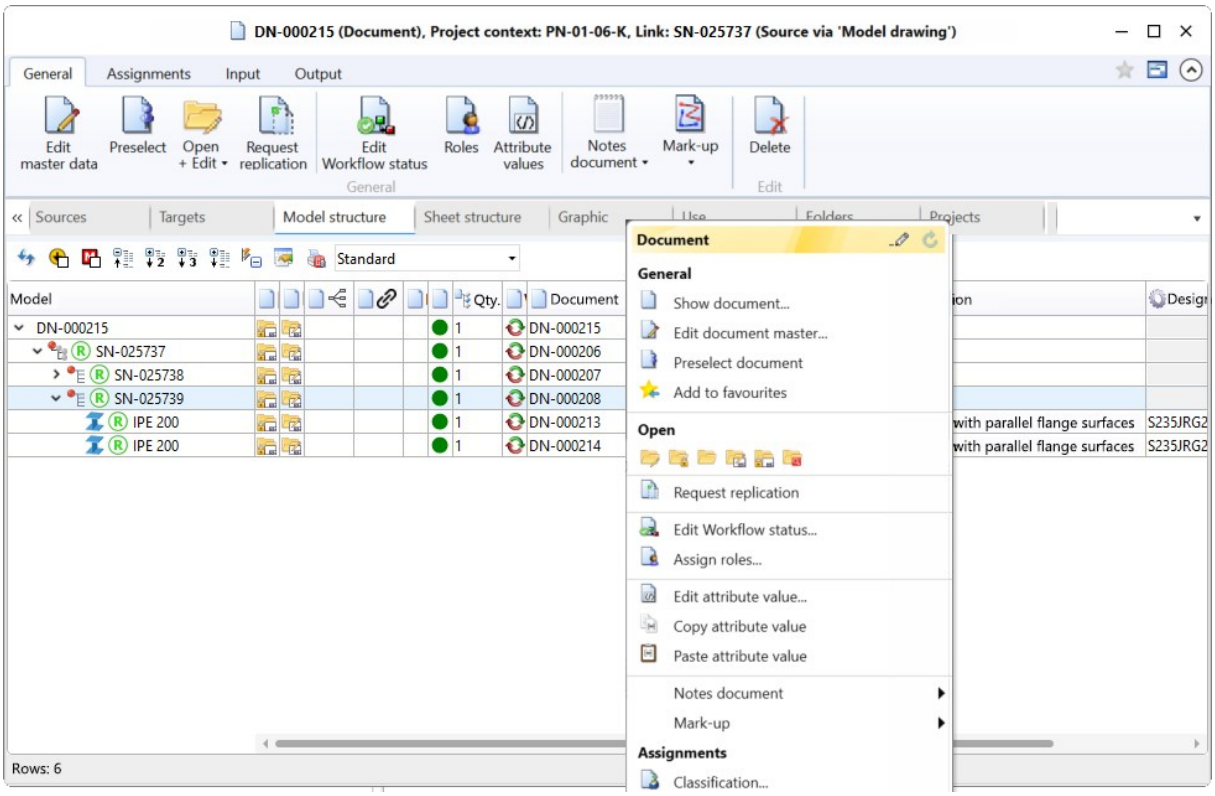
The update to HiCAD PDM 2027 has introduced a number of improvements in this regard:

When saving geometries that contain a referenced part (or its sketch) as a feature in HiCAD 3D, this information is also transferred to HiCAD PDM, where it is stored and displayed in the **Model structure**:

In HiCAD 3D, simplified parts and assemblies are assigned the ‘Simplified Part’ feature. Sub-assemblies of simplified assemblies, in turn, have the ‘Simplified Sub-part’ feature. Both are stored in the model structure and taken into account.

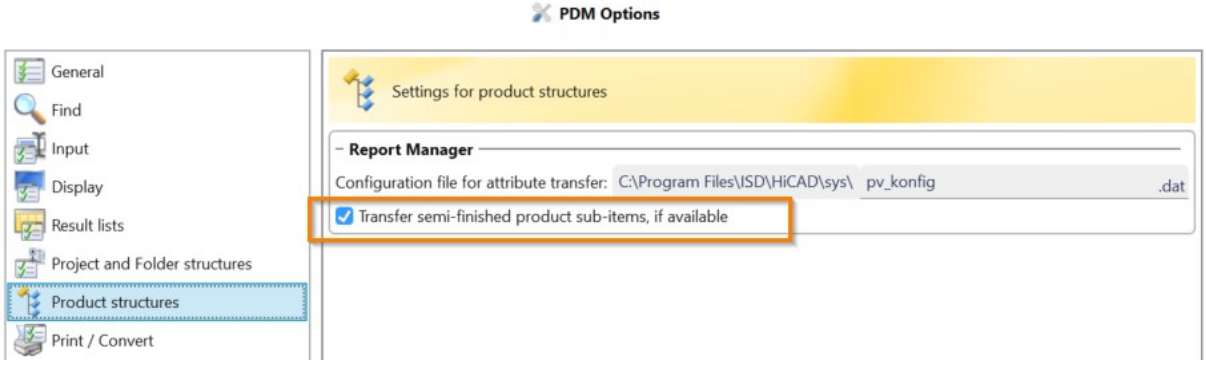
When determining and displaying the current status of base parts, both simplifications and dependencies are also taken into account.

When a document details dialogue is opened from within the model structure, the article context of the top part is also passed along.

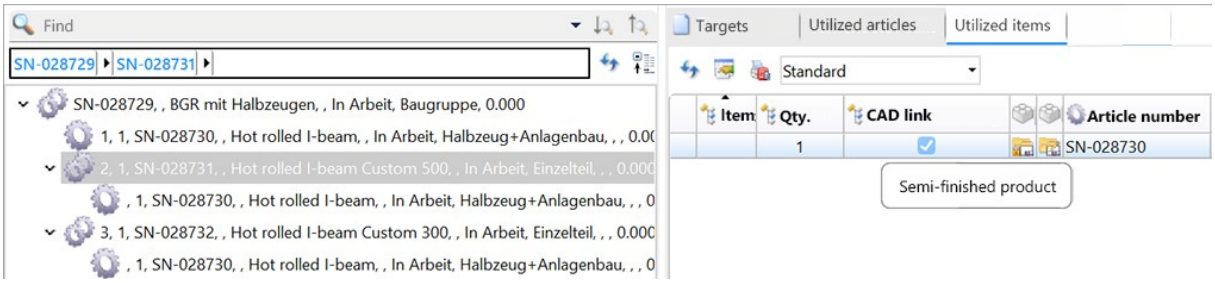


Semi-finished product sub-items

A new setting under **PDM Options > Product structures** can be used to control whether semi-finished product sub-items should, where applicable, always be filtered out when data is transferred to the **Report Manager**.



On the **Utilized items** result list for a product structure, the tooltip for a set **CAD link** may indicate whether the item in question is a semi-finished product sub-item.



Database Options: Classic file name generation

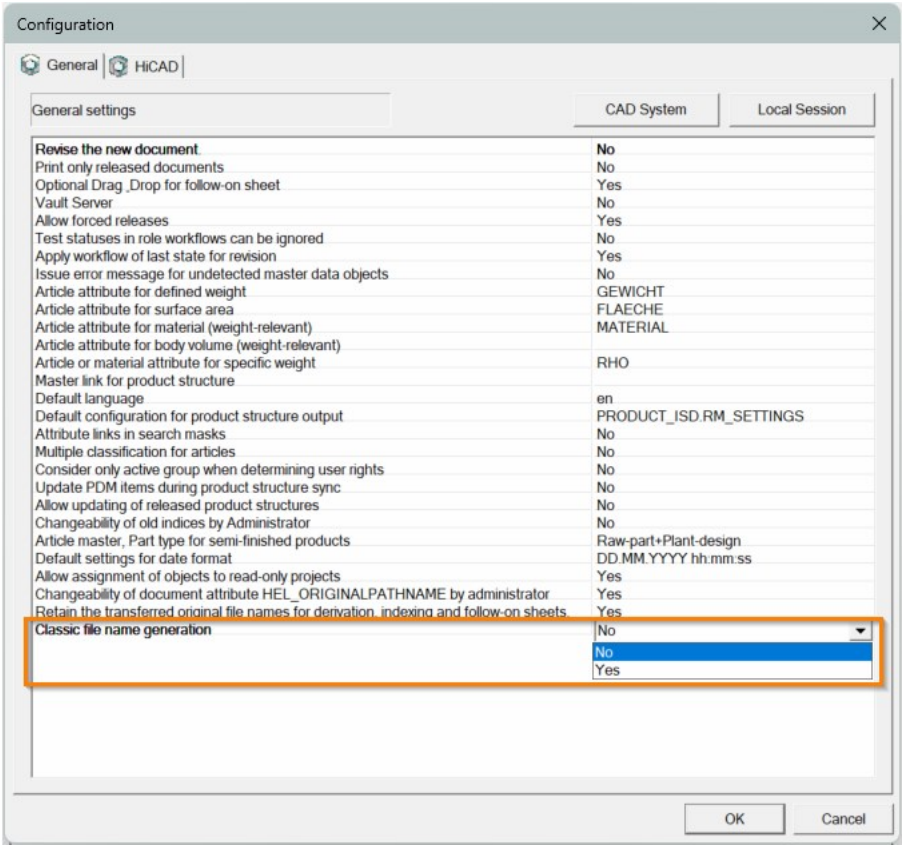
In earlier versions of HELiOS PDM, there were differences in automatic file name generation:

- When working with a Vault Server, the file name is generated according to the following scheme: <Head- ID>_ <Rev- ID>.File extension>.
  Example: B004URKMOE3KZ300003VJC_ 0.text
  As the combination of Head ID and Rev ID is always unique for a document, the file name is also always unique.
- When working without the Vault Server, the file name is generated as a cryptic 8-character string.

  Example: 55AFCE85.txt

In theory, the file name could therefore be ambiguous in the second case if documents were to be created in rapid succession; for this reason, file name generation has been switched by default to the method used with an active Vault Server.

For a transitional period, the naming scheme can be changed to the "classic" method via **PDM Options > Database** by changing the **Classic file name generation** setting from **No** to **Yes**.



Please note: This option is only available when working without a Vault Server.

HiCAD PDM in HiCAD 3D

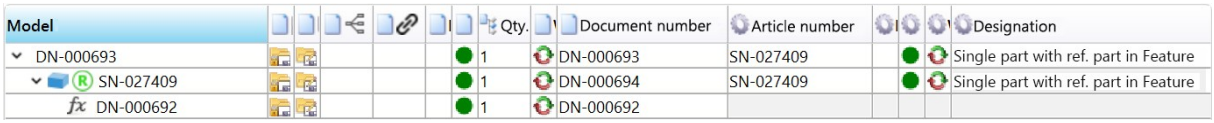
Major Release

Model structure

Referenced parts of the HiCAD 3D feature

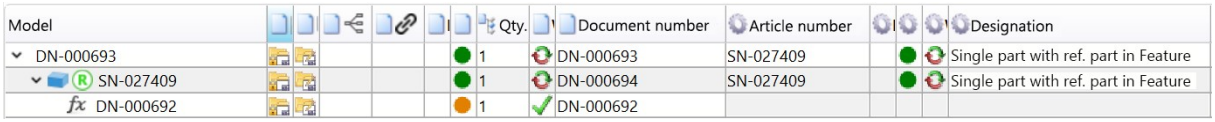
The **Model structure** tab on a HiCAD PDM document master screen displays the document structure of HiCAD 3D files (.SZA, .KRA).

With the update to HiCAD PDM 2027, this functionality has been extended so that when geometries are saved that contain a referenced part (or its sketch) within a feature in HiCAD 3D, this information is also transferred to HiCAD PDM, stored in the **Model structure** and displayed:



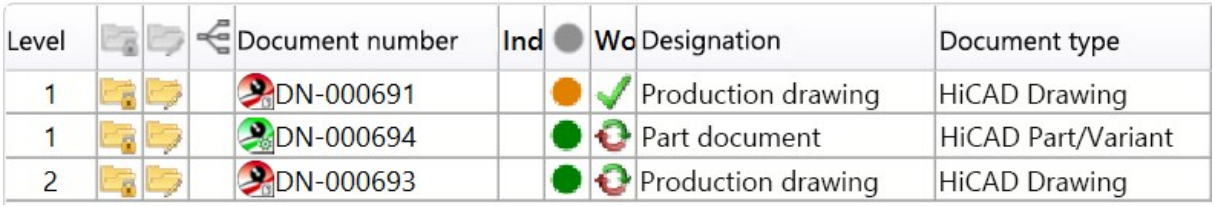
Model	Qty.	Document number	Article number	Designation
DN-000693	1	DN-000693	SN-027409	Single part with ref. part in Feature
SN-027409	1	DN-000694	SN-027409	Single part with ref. part in Feature
DN-000692	1	DN-000692		

As the database also takes into account the KRA version number of the referenced part (or sketch) within the feature, the model structure also indicates whether the version is up to date or not.



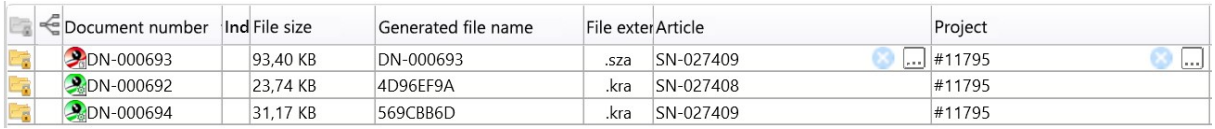
Model	Qty.	Document number	Article number	Designation
DN-000693	1	DN-000693	SN-027409	Single part with ref. part in Feature
SN-027409	1	DN-000694	SN-027409	Single part with ref. part in Feature
DN-000692	1	DN-000692		

When viewing the PDM document, it is also possible to check the **Use** of the referenced part (or sketch):



Level	Document number	Ind	Wo	Designation	Document type
1	DN-000691			Production drawing	HiCAD Drawing
1	DN-000694			Part document	HiCAD Part/Variant
2	DN-000693			Production drawing	HiCAD Drawing

Furthermore, the referenced parts (or sketches) within the feature are also taken into account when exporting the CAD document using the **Pack & Go** function.



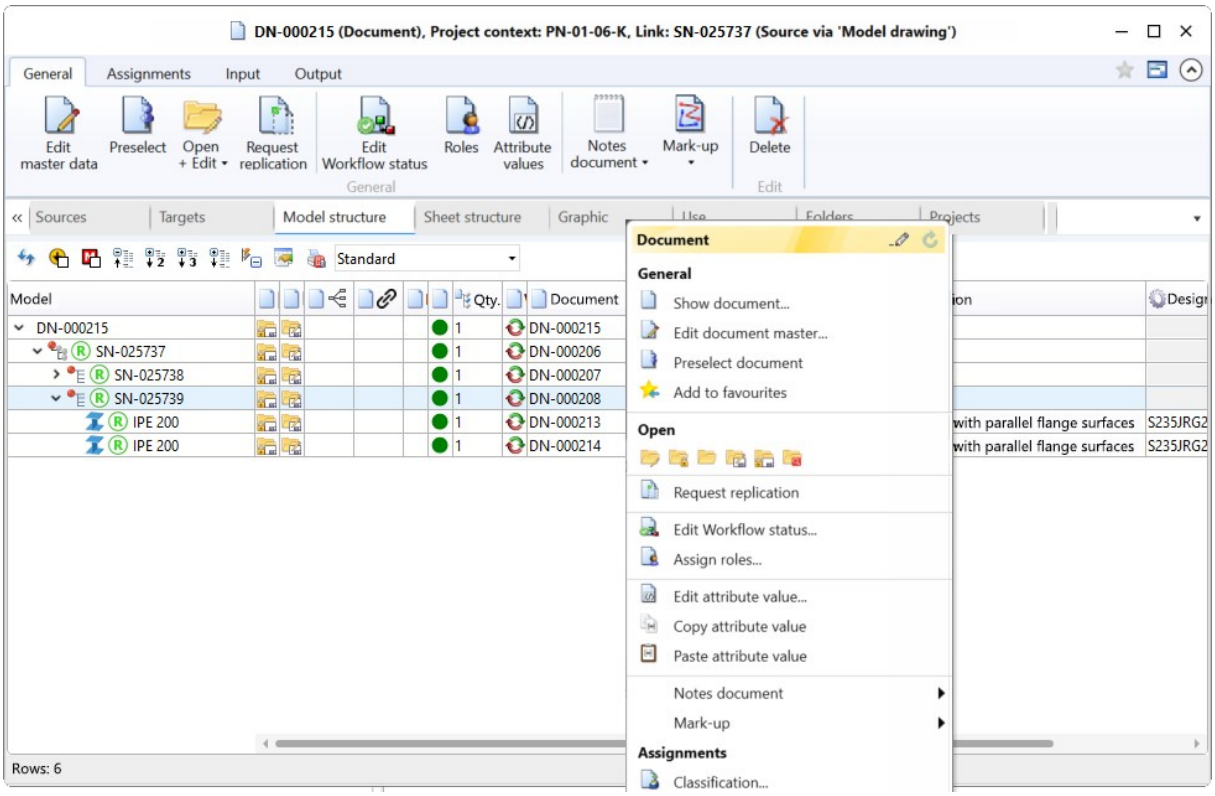
Document number	Ind	File size	Generated file name	File exter	Article	Project
DN-000693		93,40 KB	DN-000693	.sza	SN-027409	#11795
DN-000692		23,74 KB	4D96EF9A	.kra	SN-027408	#11795
DN-000694		31,17 KB	569CBB6D	.kra	SN-027409	#11795

Further optimisations

In HiCAD 3D, simplified parts and assemblies are assigned the Simplified Part feature, whilst sub-assemblies of simplified assemblies are assigned the "Simplified sub-part" feature. Both are stored in the model structure and taken into account.

When determining and displaying the up-to-dateness of base parts, both simplifications and dependencies are also taken into account.

When a document details dialogue is opened from within the model structure, the article context of the top part is also passed along.



HiCAD PDM Vault Server

Major Release

Improvements to the Vault Server installation

Like other HiCAD PDM components that are installed separately, the Vault Server is first uninstalled and then reinstalled as part of an update installation. Should any errors occur or the process be interrupted, the old version can be restored.

New and update installations of HiCAD PDM on the client side "recognise the Vault connection details from previous installations on the system and populate the setup screens accordingly (where they can be amended by the user if necessary).

Furthermore, it is generally possible to install multiple instances of the Vault Server in parallel on a single system (this may be useful for testing purposes, for example).

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All locations and contact details at a glance:

ISD Software und Systeme GmbH

Hauert 4

D-44227 Dortmund

Phone +49 231 9793-0

info@isdgroup.com

HiCAD 3D 32.0

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