HiCAD

UNLIMITED CAD PERFORMANCE **DEVELOPED BY ISD**

What's new?

Version 2023

News Overview

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THE WORLD OF CAD AND PDM SOLUTIONS

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Basics

Discontinuations

Discontinuation of Windows® 7 and Windows® 8

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

Discontinuation of "old" HiCAD itemization

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

Discontinuation of "old" OpenGL versions

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

Discontinuation of old figure format (FIG)

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

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

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

Discontinuation of the "old" Create detail drawing function

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

Discontinuation of HELiOS 32-Bit and HiCAD Viewer 32-Bit

As of HELiOS 2022 (Version 2700.0), a 32-Bit version for HELiOS and the HiCAD Viewer will no longer be available. However, the interface to 32-Bit applications such as Office will still be possible and is not affected by the discontinuation of the 32-Bit installation of HELiOS.

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.

Service Pack 2 2023 (V 2802)

Automatic data save with higher performance

As of SP2, when using the automatic data save, the polygon models of parts are no longer saved, but are only recalculated when a drawing is restored. This improves the performance of automatic data save for drawings with very many parts

The behaviour can be influenced in the Configuration Editor at **System settings > Data save** via the checkbox **Quick data save** The checkbox is active by default

Please note the following:

- The polygon models of reduced loaded parts are always saved - regardless of whether the checkbox is active or not.

- In rare cases it can happen that users change the display accuracy of individual parts. In this case, the representation would have to be re-adjusted after the restoring. If you work specifically and frequently with display accuracy, you should deactivate the checkbox.

Itemisation

Itemisation - minor text changes
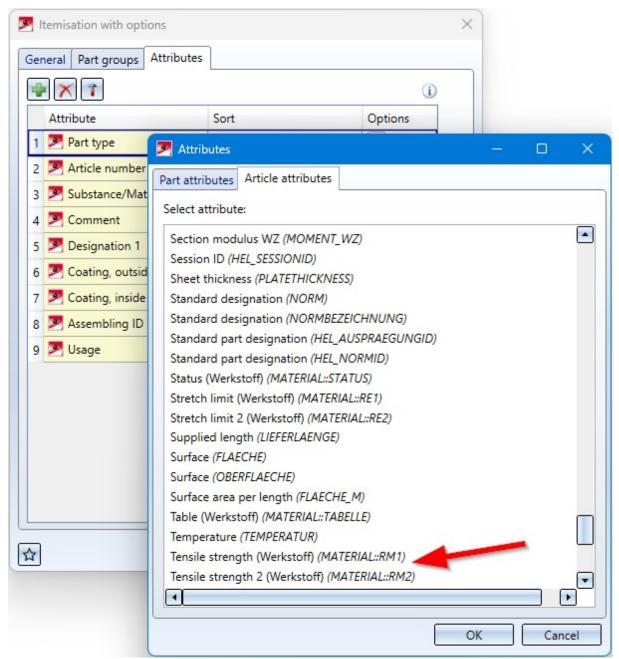
The term "Itemisation mode Standard" has been replaced in SP2. In this context, the function "Switch to standard itemisation" has been renamed **Switch from itemisation up to HiCAD 2017**.

In addition, texts have been adapted in Configuration Editor at **System settings > Itemisation** or **System settings > Itemisation > Migration**:

up to HiCAD 2023 SP1	as of HiCAD 2023 SP2
Presettings that are entered as drawing properties when fixing the itemisation mode Standard..	Default settings are entered in the drawing properties in the first situation in which they become relevant.
When switching to the itemisation mode Standard, use the stored default settings.	When switching from the Itemisation up to HiCAD 2017, do not perform a conversion, but use the stored default settings.

Itemisation - HELiOS Article reference attributes

As of SP2, the article reference attributes defined in HELiOS can also be used as a sort criterion in itemisation, for example Tensile strength. These are offered in the selection list on the **Attributes** tab when adding and editing attributes.

In practice, frequently used article reference attributes are

- Customer,
- Material and
- Manufacturer.

Note that if you have not already done so, you must first define these as reference attributes in HELiOS.

Itemisation - Sort numbers

In practice, a sorting is often desired in which the numbers are not sorted as text, i.e. alphabetically, but as numbers. Therefore, as of SP2, it is possible to specify for many attributes how the sorting should be done for the respective attribute. For this purpose, the **Attributes** tab has been adapted accordingly. New is the column **Options**, in which you can select the sorting mode by clicking on the corresponding symbol:

Compare digits like letters

Example: IPE 100 < IPE 80

Compare numbers as a whole

Example: IPE 80 < IPE 100

> The default setting as of SP2 is **Compare numbers as a whole** .

A simple example:

Please note:

- The setting of the sorting mode **Compare numbers as a whole** is lost if the drawing is opened and saved with HiCAD 2023 Version 2801.X or 2800.X.

- If a drawing is loaded that contains itemisations from an earlier HiCAD version (before HiCAD 2023 SP2), the settings used there are active.

Itemisation - User-defined sorting for HELiOS attributes with Predefined lists

Item numbers can also be sorted by HELiOS attributes with Predefined lists. Until now, however, the order resulted from the alphabetical order of the language-dependent texts. As of SP2, it is now possible to change the order individually. For this purpose, two new icons are available on the **Attributes** tab.

Sort alphabetically

This icon is only displayed for HELiOS attributes with **Predefined** lists. These are multilanguage attributes, such as the part type (COMPONENT_Type) and - if defined accordingly in HELiOS - string or integer attributes. With a click

on the icon you switch from alphabetical sorting to user-defined sorting. The icon representation changes to   . In

this case, the **Determine user-defined order**   icon is also displayed.

Determine user-defined order

With a click on the symbol you can individually change the sortting order for HELiOS attributes with **Predefined** lists, e.g. for the article attribute **Part type (COMPONENT_TYPE)**:

Behaviour of BOMs when updating derived drawings

BOMs that are aligned to the drawing frame (option Quantity list in drawing frame.../Structure list in drawing frame...) are only rearranged when updating derived drawings if the views are also rearranged. This means that the checkbox **Rearrange views** must be active.

If no rearrangement takes place, HiCAD still ensures that the fixed point of the BOM corresponds to the settings of the drawing derivation.

Example:

If the BOM is arranged in the top right corner, then the top right corner of the BOM will still be in the top right corner of the view frame. If the BOM has become longer or wider, then - to ensure this - it is moved if necessary.

Dimensioning Rules Editor - Development view

In the Dimensioning Rules Editor, as of SP2, the rules for the development view of Sheet Metal parts can also be defined. For this purpose, the dialogue window has been extended by a corresponding tab.

> **Please note:**
> In the drawing derivation, these rules are only taken into account if the options for dimensioning are activated accordingly in the settings for sheet developments, e.g. like this:
>
>

New dimensioning rules for agraffes

For the two dimensioning rules, an isolated point was added to the agraffe in the bore pattern. This point serves as a dimensioning point and has the designation **Dim**.

163: Dimension agraffes in Sheet Metal parts via dimensioning point
Chain dimension,   Top
Outer sheet/plate edges,   Parallel to sheet/plate X-axis

163: Dimension agraffes in Sheet Metal parts via dimensioning point
Chain dimension,   Right
Outer sheet/plate edges,   Parallel to sheet/plate X-axis

3016: Dimension agraffes in Sheet Metal **development** via dimensioning
Chain dimension,   Top
Outer sheet/plate edges,   Parallel to sheet/plate X-axis

3016: Dimension agraffes in Sheet Metal **development** via dimensioning
Chain dimension,   Right
Outer sheet/plate edges,   Parallel to sheet/plate X-axis

(1) Dimensioning point of the agraffe in the sheet, side view from left; (2) Dimensioning point of the agraffe in the sheet development

ICN Part structure display with grid lines

To further emphasize the levels of the part structure, grid lines can be displayed by activating this checkbox.

Left: with grid lines, Right: without grid lines

Customise context menus

The Maximum number of buttons per row in the context menus for parts etc. is now preset to 12. This only applies to new installations. Customised menus do not change during an update installation.

Change assembly coordinate system

Like every part, an assembly also has a part coordinate system. Until now, this coordinate system could either be determined automatically by HiCAD or by you when creating the assembly. A corresponding parameter is available in the Configuration Editor at **Modelling > Part creation > Assembly**.

As of HiCAD 2023 SP2 you can also move the part coordinate system of assemblies with the new function **Change assembly coordinate system** (Drawing > Others > World CS ▼ > ...).

> Note that this can also change the geometry and position of the parts in the drawing. Referenced parts will change at all points of use.

Please also see the examples for Feature and Parametrics.

Extension of the Design Checker for Sheet Metal parts

By setting the part orientation you can determine which view of a part should be the front view or the top view. This is also evaluated when creating the production drawings. Since SP1 it is possible to check for Sheet Metal parts during the creation of the production drawings whether the part orientation has been set accordingly. To do this, the check-box **Check part orientation for Sheet Metal parts** must be activated in the Configuration Editor at **Automatic drawing derivation > Production drawing**.

As of SP2, this can also be checked via the Design Checker - independent of the setting in Configuration Editor. The Design Checker has been extended accordingly.

Updating mounting drawings

When updating mounting drawings that were created with the setting

 New view group via active assembly(-ies)

only the parts of the view group that belong to the selected assemblies are taken into account from SP2 and not all parts that lie within the bounding box.

Support of the new SpaceMouse®

When working on 3-D drawings, you can use a SpaceMouse®in addition to the standard mouse. HiCAD supports the current models from 3Dconnexion. To connect to HiCAD, install the current software 3DxWare from 3Dconnexion GmbH on your computer.

By pressing, pulling, rotating and tilting the controller cap (control knob), you can rotate, move and enlarge or reduce your 3-D drawing in HiCAD, regardless of the SpaceMouse type. The navigation in the **Object Mode** of the SpaceMouse® behaves as if you were reaching for the 3-D part and holding it in your hand.

The rotation point for stabilising the rotation is automatically placed on the nearest part. If you move the standard mouse, the rotation point also changes.

The **MENU** button on the SpaceMouse® or **3Dconnexion Home** (on your Desktop) will take you directly to the **3Dconnexion Settings**. Via **Advanced Settings** you can configure the default behaviour of the controller cap in HiCAD.

Some HiCAD functions (such as Save drawing) are pre-assigned to the buttons of the SpaceMouse®. With the SpaceMouse® Enterprise, the HiCAD functions are shown with text and icon in the display of the mouse. To customise the function keys for your company, call up the **3Dconnexion Settings**. Then click on **Buttons**. Click on the arrow next to the function name and select a different HiCAD function. Click on **Close** to save the new assignment of the function button.

HiCADGUIReset - Restore defaults

The functions of **HiCADGUIReset** can now also be executed automatically by specifying various options, i.e. without displaying the dialogue window.

Analogous to the functions of the dialogue window, the following options are possible:

Option	Effect	Example
-Save *Dateiname*	Save GUI settings	HiCADGUIReset -Save D:\GUI\GUI1.hicadgui
-Load *Dateiname*	Load GUI settings	HiCADGUIReset -Load D:\GUI\GUI1.hicadgui
-ResetAll	Restore DUI delivery state	HiCADGUIReset -ResetAll
-ResetIDs	Reset button IDs	HiCADGUIReset -ResetIDs

What's new?

Service Pack 1 2023 (V 2801)

Product extension

The **Coating** function is now also part of the basic HiCAD creator and HiCAD solution models. This means that the function is also available in all industry suites.

Part filter extension

The **Find** ▼ function in the transparent toolbar has been extended to include filters for profile installation. With the new rule **Properties > Profile sub-part type (Profile Installation)** the sub-part types
- Contour, cut,
- Contour, uncut,
- Sandwich panel, outside sheet,
- Sandwich panel, inside sheet,
- Insulation and
- Others, e.g. sealing chords.

can be easily selected.

Units in the Configuration Editor

In the Configuration Editor, under **Extras**, you can set whether the values specified there for length and weights should be displayed in metric units (millimetres and kilograms) or in imperial units (inches and pounds).

The numbers are displayed rounded, with the unit following. If the values are edited, no rounding will occur, but the numbers will be displayed exactly.

Examples:

If the setting under **Extras** is changed, the values will be converted accordingly, e.g.

The setting of the units of measurement under **Extras** only affects the display in the Configuration Editor. When drawing in HiCAD, the values are always converted to the unit of measurement of the drawing.

Sheet name - Placeholder during printing

In the printer output there is now a placeholder for the **Sheet name** called **<SheetName>** which can be used in the XPS and the ISD FilePrinter output to generate the file name. For this purpose, the context menu under **Post-processing Command line** has been extended on the **PS/XPS Settings** tab.

This concerns the functions

Drawing > Save/Print > Print > Active drawing and

Drawing > Save/Print > Print > Selected model drawings.

Examples:

- C:\Program Files\gs\gs9.54.0\bin\gswin64c -dNOPAUSE -dBATCH -dPDFA -sDEVICE="pdfwrite" -sOut-putFile="<FilePath><SheetName>.pdf" <CurrentFile>

- C:\Program Files\gs\gs9.54.0\bin\gswin64c -dNOPAUSE -dBATCH -dPDFA -sDEVICE="pdfwrite" -sOut-putFile="<FilePath><FileName><SheetName>.pdf" <CurrentFile>

.

Regarding the **Sheet name**, i.e. the placeholder **<SheetName>**, please note the following:

- All not allowed characters are replaced by an underscore _ in the sheet names. Not allowed characters are:\:/\"<>?*|.

- If the sheet names are the same, a new name will be generated automatically, e.g. MySheet -> MySheet(2) -> MySheet(3) ...

- For merged PS files, the name is composed of the output sheet names (separated by commas). The length is limited to 32 characters. E.g. MySheet1,MySheet2,MySheet3.pdf

Hatching settings

New under **Drawing > Properties** is the **Hatching settings** function.

Use this function to define the default settings for the hatching of cut surfaces created by sectional views, cut-outs and detail views. The settings apply to hatchings in the current drawing and are saved together with the drawing.
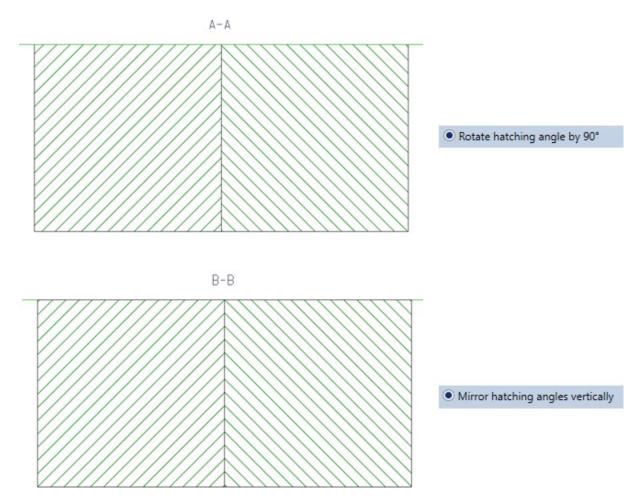
The previous functions

- **Acc. to material catalogue**,
- **Parallel lines** and
- **Scale-dependency**

have been integrated into the new function.
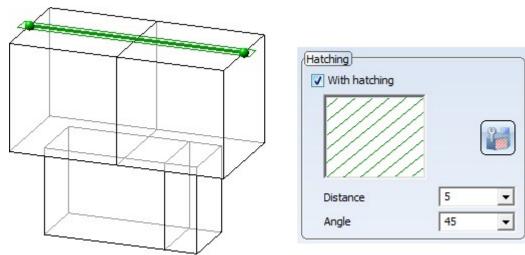
A new feature is that it is now possible to specify how neighbouring parts are to be visually delimited from each other in a sectional view. This can be done by rotating the hatching angle or by vertical mirroring.

A–A

● Rotate hatching angle by 90°

B–B

● Mirror hatching angles vertically
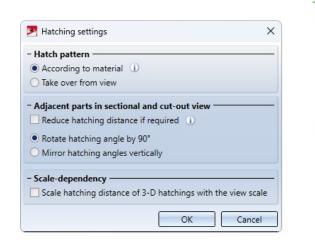
Sometimes, however, changing the hatching angle is not sufficient to clearly delimit neighbouring parts from each other. This can be the case, for example, when several parts touch each other (see example). If the **Reduce hatching distance if required** checkbox is active, the hatching distance is also used for delimitation. This means that the hatching distance of individual parts is reduced.
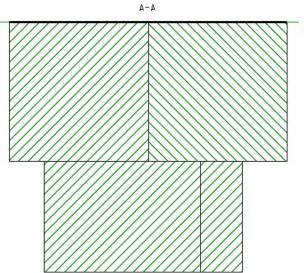
Example

We consider four cuboids that touch each other. We want to create a sectional view with the sketch shown as the section path.

Hatching
☑ With hatching

Distance 5
Angle   45
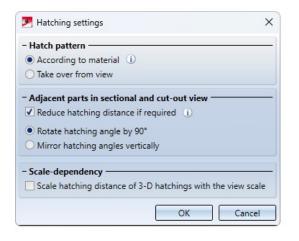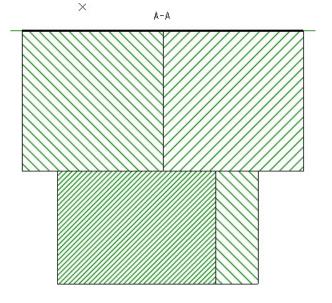
The following image shows that not all parts can be clearly distinguished from each other using only the hatching angle.
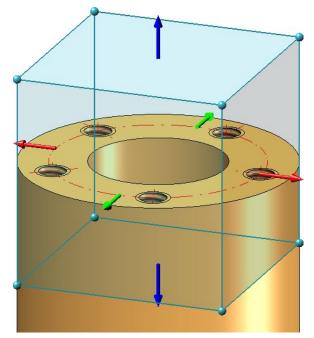
In this case, it is recommended to activate the **Reduce hatching distance if required** checkbox.

Easier identification of plug-in arrows

The plug-in arrows - as used, for example, when defining a detail view of type cuboid or when clipping point clouds - are now easier to identify. The arrows are already identified when the cursor is in the "vicinity".

Mounting drawing - View group above assemblies
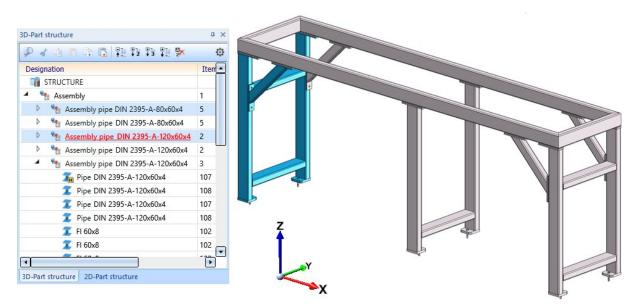
When creating mounting drawings, it is now possible to create the view group above the active assembly or the selected assemblies.
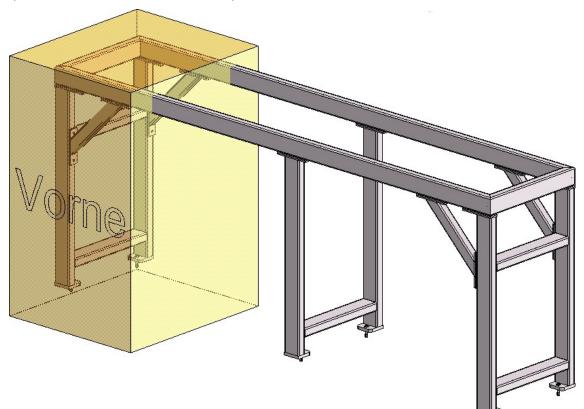
The view group is determined by selecting/defining a processing plane when calling the function for the active assembly or the selected assemblies. The same options are available for selecting/defining the plane as for the **3-D processing plane** function. The box is created by extruding the processing plane to the front and back - considering the offset specified in the **extended settings**. The size of the box is determined here by the active assembly or the selected assemblies when the function is called.

Example:
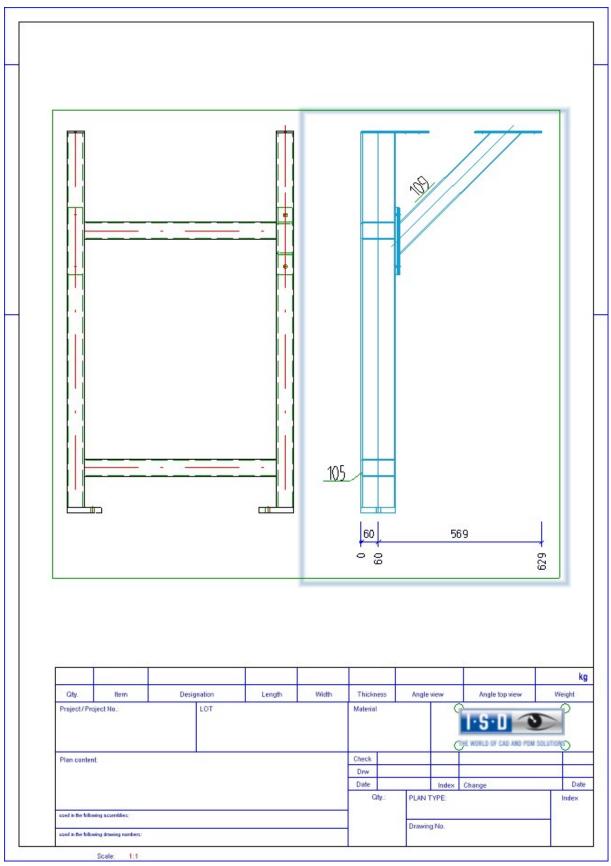
A mounting drawing of the two selected assemblies is to be created.

The XZ plane of the world CS is selected as the plane.

Result:

Derived drawings - Check part orientation for Sheet Metal parts

By setting the part orientation you can determine which view of a part should be the front view or the top view. This is also evaluated when creating the production drawings.

For Sheet Metal parts, you can check whether a part orientation is set when creating the production drawings. To do this, the new checkbox **Check part orientation for Sheet Metal parts** must be active in the Configuration Editor under **Automatic drawing derivation > Production drawings**. If the default setting is used, the checkbox is inactive.

If the checkbox is active and if bending plates without part orientation are found when creating the production drawing, a corresponding message appears:

If you choose **Yes**, the drawings will be created anyway, if you choose **No**, the drawing creation will be cancelled.

This parameter also has an effect on the **Drawing Management**.

Itemisation - Automatic conversion of the "old" itemisation with default favourites

The conversion of previous itemisation of drawings (before HiCAD 2018) to the HiCAD standard itemisation can now also be performed automatically, i.e. without displaying the itemisation dialogue. For this purpose, the checkbox **When switching to the itemisation mode 'Standard', use the stored default settings** is available in the Configuration Editor under **System settings > Itemisation > Migration**.

If the checkbox is **inactive**, then the dialogue of itemisation with options will be started, as before.

If the checkbox is **active**, then the conversion will be started directly - without displaying the dialogue of the standard itemisation. The Favourites file selected in the Configuration Editor under **System settings > Itemisation > Updating** will be used.

Major Release 2023 (V 2800)

Units of measurement

Drawing - Free choice of unit of length

The previous version HiCAD 2022 only supported drawings with the unit of length millimetre. As of HiCAD 2023, the unit of length can be selected under **System settings > Units > Unit of length** in the Configuration Editor. This means that designing in non-metric units is now also possible.

> **Please note the following:**
> The settings selected under **System settings > Units** only apply to newly created drawings. These settings cannot be changed in the drawing itself.

Please note:

All values inputs in HiCAD are evaluated in the currently set units of measurement. It is also possible to use mixed units. For example, if you have set **mm** as the unit of length of the drawing and used **1 3/4" + 12.5 mm** as the value in a function, the value will be converted to mm, i.e. the result will be 56.95 (mm). If you point to an input field in the dialogues with your cursor, the system will show in which unit the respective value is, e.g.:

In the value inputs where no formulas, fractions, variables, units and spaces are supported, e.g. in the **Part attrib-utes** dialogue, such inputs will be rejected with an error message, e.g.

Units of the model drawing

New in the **Help Topics and Information** is the **Units of the model drawing** function, which can be used to display the current settings for units, e.g. length, surface, volume etc.

Category	Unit
Length	mm
Surface	m^2
Volume	mm^3
Angle	°
Weight	kg
Density	$g * cm^{-3}$
Surface area per length	m
Mass per surface area	$kg * m^{-2}$
Moment of inertia	cm^4
Force	$kg * m * s^{-2}$
Performance	$kg * m^2 * s^{-3}$
Pressure	$kg * m^{-1} * s^{-2}$
Section modulus	cm^3
Temperature	K
Thermal conductivity	$kg * m * s^{-3} * K^{-1}$
Thermal transmittance	$kg * s^{-3} * K^{-1}$
Pitch	mm

Selecting the region in the parameter configuration

Both during installation and during subsequent parameter configuration with the **ParKonfigComp.exe** tool, you can now select the region.

Please note the following:

If you activate **North America** as the region, the system will automatically switch to imperial, i.e. non-metric units. This applies to:

- units of the drawing,
- dimensions and
- scales.

This means that settings made in the Configuration Editor may be replaced if necessary!

This applies analogously to the region **Europe**. Here, a conversion to metric units, dimensions and scales takes place.

Constraints in drawings with a unit of length different from mm

If you have selected a unit other than **millimetre mm** as the unit of length of the drawing, individual functions will not be available. These functions will be greyed out in the ribbons and menus and a corresponding message will be displayed in the tooltip.

This applies, for example, to the functions

- Information > 3-D, Further > Cylindrical pin calculation,

- Information > 3-D, Further > Radial pin calculation,

  Steel Engineering > Plate, new > Rectangular plate,
- Steel Engineering > New > Insert series, via DB document master,
- Steel Engineering > New > Insert series, via DB article master,
- Metal Engineering > Others > 2-D section of 3-D facade,
- Metal Engineering > Metal Engineering, 2-D > Sealant/Foil,
- Metal Engineering > Metal Engineering, 2-D > Sheet,
- Metal Engineering > LogiKal, 2-D > Insert facade + inserts from existing LogiKal project,
- Metal Engineering > Interfaces > Create WinIso2D part,
- Metal Engineering > Interfaces > Export to eluCad,
- Sheet Metal > New > Pipes + Vessels

as well as individual Plant Engineering and P+ID functions.

Drawing frame for North America

If **North America** is selected as the region during installation or during subsequent parameter configuration, addi-

tional drawing frames are available in the **Insert drawing frame** function.

Formulas, mixed numbers and units when entering points

In the numerical point options, you can enter the numerical values required for the point option in one step, e.g. relative coordinates. Formulas and mixed numbers are now also permitted, e. g. (a+b), 1/2, 1 3/4" + 12.5 mm. In this case, you must note the setting in the Configuration Editor under **System settings > Miscellaneous > Pocket calculator**.

- **Space**
  With this setting, the different entries must be separated by spaces. If formulas, mixed numbers or mixed units are used, they must not contain spaces. This is the ISD default setting.

- **Semicolon**
  If you want to use formulas etc. with spaces, you must select this setting. The different entries must then be separated by semicolons.

Example:

You want to use the coordinates X=(a+b)/2 y=70, z= 90 for the point option **R**, where a and b are part variables with the values a=50 and b=7.

Setting	Entry	Result	
Space	R (a+b)/2 70 90	28.5 70 90	correct
Space	R (a + b)/2 70 90	Returns 0 as X value and queries the other coordinates	false
Semicolon	R (a + b)/2;70;90	28.5 70 90	correct

Mark-up and Redlining

The **HiRedLine** tool allows you to add comments or modification suggestions to a HiCAD drawing, without changing the original file.

As of 2023, this tool is only available as part of a HiCAD/HELiOS installation. In case of a standalone installation of the HELiOS Desktop, starting it will no longer be possible.

Ribbon adjustments

The **Drawing**, **Views** and **Sketch** ribbons have been slightly modified and tightened. The most important adjustments at a glance:

Drawing

- The print functions have been combined in a pull-down menu.
- You can now send drawings via e-mail under **Save as**.
- The Variant Editor has been moved to the **Extras** menu.
- You can find the functions for the creation and updating of mounting drawings as well as the templates and dimensioning rules/settings for the drawing derivation in the pull-down menu of the drawing derivation.
- The insertion of drawing frames has been moved to the **Itemisation/Detailing** area. You can complete the title blocks using the functions of the pull-down menu.
- Showing/hiding by element type has been moved to the **Properties** area.
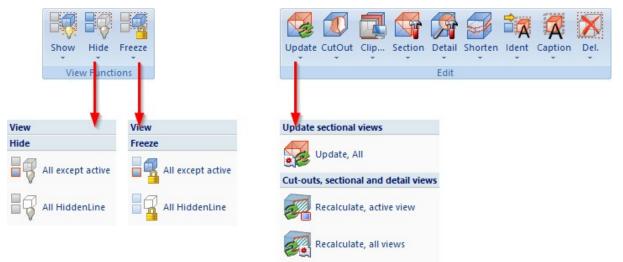
Sketch

- You can close internal corners and lengthen/shorten lines directly via the **Sketch** ribbon.
- You can directly access the functions for setting/deleting auxiliary geometry from the **Sketch** ribbon.
- The functions under Transform/Clone have been arranged differently.
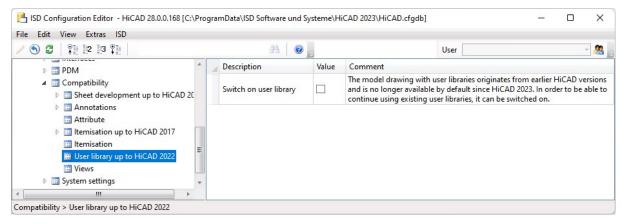
Views

- The functions for hiding and freezing have been each combined in a pull-down menu.
- The functions for updating cut-outs, sectional views and detail views have been combined in a pull-down menu.

What's new?

User libraries

As of 2023, the functions for part and macro libraries are no longer available in the **Drawing** ribbon during a new installation. Existing customers who wish to continue using their existing libraries can switch on the functionality in the Configuration Editor under **Compatibility > User library up to HiCAD 2022** by activating the corresponding checkbox.

When updating from a version prior to HiCAD 2023, the user library is still switched on for existing customers. In the case of an update from HiCAD 2023, the current setting from the Configuration Editor will be adopted.

Moving the screen display
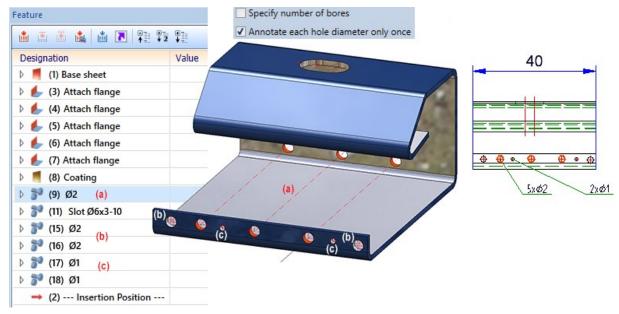
The screen display can now also be moved by holding down the right mouse button and dragging the display to the desired position. This corresponds to the behaviour of the CTRL + middle mouse button combination. If the right mouse button is simply pressed, the **3-D view** context menu will be called instead.

Derived drawings

Annotation of identical bores

You can use the **Dimension Settings** function to combine individual bores which do not belong to a bore pattern on the **Annotations** tab by activating the **Specify number of bores** checkbox. The annotation will then occur only once with specification of the number.

If only one annotation is to be set for all identical bores - regardless of whether they are individual bores or bore patterns- this can be set via the **Annotate each hole diameter only once** checkbox.

Left: Model with (a) bore pattern 3 x Ø2 as well as (b) two individual bores Ø2 and (c) two individual bores Ø1
Right: Detail of the workshop drawing

New usage for sheet metal assemblies

A new usage is available for sheet metal assemblies, i.e. assemblies with a sheet as the main part: **Sheet Metal Assembly**. For this purpose, the **Factory standards > Usage** catalogue has been extended accordingly.

For assemblies with this usage, usage-dependent configurations can be used for drawing derivation, which are managed in the ISD Configuration Editor, as for other usages. By default, the configuration **ASSEMBLY_SHEETMETAL** is predefined for assemblies with the usage **Sheet Metal Assembly**.

Production and mounting drawings - Drawing frames

As of HiCAD 2023, two additional parameters are available for production drawings in the Configuration Editor under **Automatic drawing derivation > Production drawing > Drawing frames**.

Use frame

This checkbox determines whether the frame should be offered for selection in the **Drawing derivation** dialogue or not. The ISD default setting is yes, i.e. the checkbox is active.

Detail of the Drawing derivation dialogue

Automatic determination of geometric data for the drawing frame

For each drawing frame,

- the distance of the insertion point from the centre of the frame,

- the size of the usable area (frame height / frame width) and

- the size of a blocked frame area (for the title block, BOMs or comments)

can be defined. If the blocked area is specified incorrectly or is missing, the automatic drawing derivation may cause views to collide with the title block.

If the blocked area of a frame is to be determined automatically during drawing derivation - including the width and height of the usable area - this can now be achieved by activating the **Automatic determination of geometric data for the drawing frame** checkbox. This is the ISD default setting for the drawing frames included in the delivery. If the checkbox is active, only the figure name can be specified if required. All other settings can be omitted.

> **Please note:**
> The prerequisite for this automatism is that the FGA file of the drawing frame fulfils the following criteria. This must be taken into account when using custom drawing frames:
>
> - The figure contains a point with **point number 1000**, which is located inside the frame (insertion point).
>
> - The textblock is called **SCHRIFE**.

Analogously, the parameters listed above are also available for mounting drawings under **Automatic drawing derivation > Mounting drawing > Drawing frames**.

Using frame/scale lists

During drawing derivation, there is often a desire to display everything on one sheet if possible. Sometimes it is also desired that sectional views, developments or axonometries should have different scales than the main view. Additionally, there are situations where the customer wants a certain drawing frame to be filled with views at an optimal scale. For example, a drawing frame DIN A2 should be used and the scale of the views should be adjusted accordingly. However, if the scale falls below 1:10, another drawing frame should be used.

This cannot be configured in the options of the drawing derivation. Here the new function **Frame/scale list** is the ideal solution.

Similarly to the dimensioning rules, this function can be used to define separately for each purpose, for example for assemblies, I-beams, sheets, etc., which drawing frames combines with which scales should be used. This also significantly increases performance when creating drawings, as HiCAD does not have to search through all drawing frames and scales.

Example of a frame/scale list for sheet metal parts

But what is the effect of the example shown above?

If the list is activated, then during drawing derivation, instead of the other settings for frame sizes and scales the list entries will be checked in sequence until a drawing fits on a sheet.

Thus, in our example, the following is checked for the detail drawings of sheets:

1. Does the drawing fit into the DIN A3 frame if main views and developments are shown at a scale of 1:2.5, axonometries at a scale of 1:1 and sectional views at a scale of 1:5? If yes, the drawing will be created and not checked further. The figure shows an example of the corresponding view structure.

If no, step 2 will be executed.

2. Does the drawing fit into the DIN A3 frame if main views and developments are shown at a scale of 1:5, axonometries at a scale of 1:1 and sectional views at a scale of 1:10? If yes, the drawing will be created and not checked further. If no, step 3 will be executed.

etc.

If the drawing does not fit any of the frames, a corresponding message will be displayed:

Please   **note:**
Frame/scale lists are only taken into account during drawing derivation if

- the **Use frame/scale list** checkbox is active for the respective usage and

- the **Drawing parameters from configuration** option is active in the drawing derivation dialogue.

New system attributes

In HiCAD, new system attributes are available for sheet metal parts and coated surfaces:

Attribute name	Attribute description	Type
§CW	Weight from rectangular surface of development	DOUBLE
§S2D	Rectangular surface of development	DOUBLE
§SC	Coated surface	DOUBLE
§SOC	Surface from development contour	DOUBLE

New docking window for variables

The docking window for variables docking window for variables has been completely revised for HiCAD 2023. In contrast to the old variable window, the entries no longer have to be saved or discarded using OK or Cancel. Instead, you can activate or deactivate the automatic feature recalculation. If it is activated, a recalculation takes place after every action in the variable window. An exception is the change of the comment.

2-D

Major Release 2023 (V 2800)

Modifications of the Ribbons

The Ribbon tabs **Drawing**, **Views** and **Sketch** have been slightly modified and streamlined. As a result, the 2-D functions for **Visualisation** can now be found at **Drawing > Properties > Visual.** ⏷ **> 2-D Parts: Visualisation...**.

3-D

Service Pack 2 2023 (V 2802)

Cams and cam processings

The cam creation creates cams on one part and slots (holes) in which these cams fit on another part. In doing so, you can specify how the cams and corresponding slots should look and how they should be arranged along the selected parts. The following functions are available as of HiCAD 2023 SP2:

 Cams
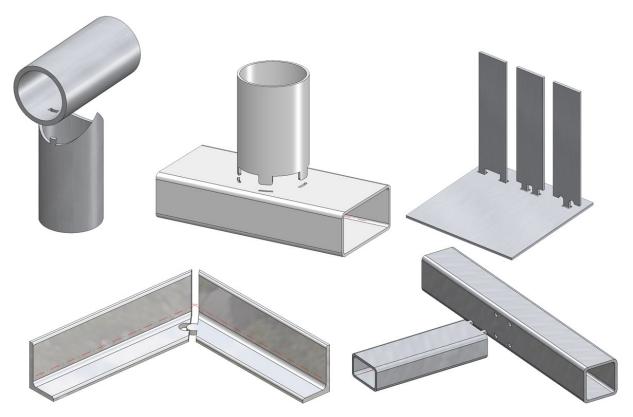
With this function, a part is provided with cams.

 Cam processing

This function creates slots or holes on one part into which the cams of another part fit.
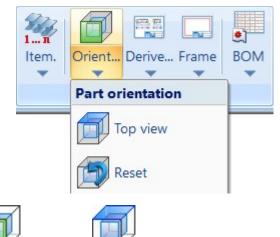
In contrast to the previous design variants to cam connections, the new functions cover significantly more practical cases and are suitable for a fully referenced working method. For example, individual cams can be placed freely here, the cams can also be distributed unevenly and they can have different edge distances on the sides. Furthermore, not only sheet metal is supported but cams can be applied to almost all types of parts - also on thin-walled solids.

The image shows different cams and the corresponding cam processings.

Part orientation and dimension alignment
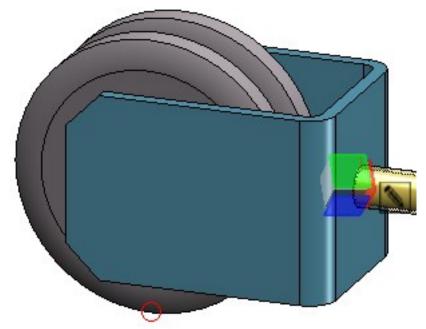
The part orientation functions have been extended. With the functions under **Drawing > Orient... > Part orientation** can now be used to set or reset the part orientation of multiple parts one after the other without having to call the function again.

After calling the **Front view** or **Top view** function, first select the part whose orientation you want to change and then the desired plane for the front view or top view. The same functions (right mouse button) are available for the determination of the plane as for the determination of processing planes. If the dimension alignment display is also active, the enveloping cube and the orientation symbol will be updated immediately. You can then change the orientation of further parts or correct the orientation directly by selecting the previously selected part again. You can terminate the function with the middle mouse button.

The corresponding functions in the context menu apply only to the active part and are automatically terminated.

In addition, the display of the part orientation has been extended. If the orientation display is active, the orientation of the part will be indicated by a space cube and an additional symbol for the source of the orientation.

Meaning of the annotations

⌐	The orientation corresponds to the part coordinate system.
⊞	The orientation results from the bearing bar direction (only for gratings DIN 24537).
↩	The orientation results from the processing direction.
✎	The orientation was set manually.
⌐ᴴ	The orientation results from the part coordinate system of the assembly main part.
⊞ᴴ	The orientation results from the bearing bar direction of the assembly main part (only for gratings DIN 24537).
↩ᴴ	The orientation results from the processing direction of the assembly main part.
✎ᴴ	The orientation of the assembly main part was set manually.

During the calculation of dimensions, the bearing bar orientation of gratings is assigned to the **Length** attribute ($03). The bearing bar orientation is ignored in the calculation of dimensions if the direction is set manually for the grating. This concerns the following cases

- The dimension direction is set.

- The part orientation is set.

- The processing direction is set.

As with the part orientation, the bearing bar orientation is also taken into account when annotating the dimension orientation.

↕⊞	The dimensions are current, the dimension orientation results from the bearing bar direction (only for gratings DIN 24537).
↕⊞ᴴ	The dimensions are current, the dimension orientation results from the bearing bar direction of the assembly main part (only for gratings DIN 24537).

Dimension calculation of beams and Steel Engineering plates

Beams

If the automatic **calculation of dimensions** is active in the Configuration Editor under **Modelling > Part properties**, then as of SP2 the dimension calculation can be restricted via the checkbox **Calculate only length for beams**. If the checkbox is active, only the length is determined by the automatic calculation of dimensions when it comes to beams. This means that any values for width and height from the catalogue are retained.

By default, the checkbox is inactive.

Steel Engineering plates

If the automatic calculation of dimensions is activated in the Configuration Editor, then length, height and width are calculated and assigned to the attributes **Length** ($03), **Width** ($02) and **Height** ($04).

If the automatic calculation of dimensions is deactivated, then length, height and width are assigned by the Steel Engineering functions. Length and width are sorted. This means that the largest value is assigned to the length.

Dimensioning

Changed default setting for dimension assignment
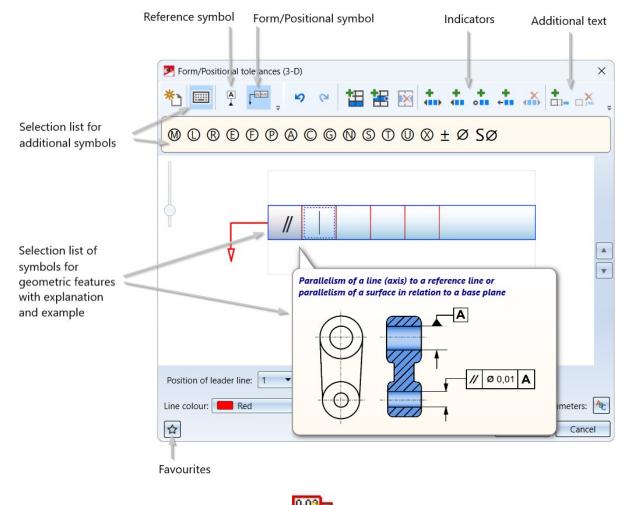
The ISD default setting for the dimension assignment has changed. As of SP2, 3-D dimensions are assigned by default to the part to which the first dimension base point belongs. The default setting can be changed in the Configuration Editor under **System settings > Annotations > Dimensioning, 3-D > Interactive dimensions**. The changed default setting applies to new installations only.

New dialogue for form/positional tolerances

The functionality for generating form and positional tolerances has been completely revised.

With the new **Form/Positional tolerances (3-D)** function, tolerances with reference symbol (DIN EN ISO 5459) as well as form/positional symbols (according to DIN EN ISO 1101 and 14405-1) can be conveniently created.

The previous functionality **Form/Positional tolerances** is still available in the menu under **3-D Dimensioning + Text > Symbols > Form/Pos**. If a drawing contains tolerances that were created with this function, the "old" dialogue will be displayed for editing these tolerances.

Variable dimensioning - Removal of structure dimensions

Previously, variable dimensioning could also be used to generate structure dimensions, although the waiting times here were significantly higher than for the corresponding functions in the **Parallel**, **Direct**, **Angle** and **Arc** function groups in the **3-D Dimensioning+Text** ribbons.

For this reason, the creation of structure dimensions with the **Variable dimensioning** function is no longer possible as of SP2.

　　　　　　　　　　　　　　　　　　　　　　　　　　　　　　　　What's new?

Extension of the dimension tables

As of SP2, additional functions are available under **3-D Dimensioning+Text > Tools > Extras** in the **Further table functions** menu:

DIN EN ISO 13920 Table 1 - Limit dimensions for linear dimensions

DIN EN ISO 13920 Table 2 - Limit dimensions for angular dimensions

DIN EN ISO 13920 Table 3 - Straightness, flatness and parallelism tolerances

Grenzabmaße für Längenmaße
EN ISO 13920

Toleranz-klasse		Nennmaßbereich(in mm)									
	2 bis 30	über 30 bis 120	über 120 bis 400	über 400 bis 1000	über 1000 bis 2000	über 2000 bis 4000	über 4000 bis 8000	über 8000 bis 12000	über 12000 bis 16000	über 16000 bis 20000	über 20000
		Grenzabmaße(in mm)									
A	±1	±1	±1	±2	±3	±4	±5	±6	±7	±8	±9
B		±2	±2	±3	±4	±6	±8	±10	±12	±14	±16
C		±3	±4	±6	±8	±11	±14	±18	±21	±24	±27
D		±4	±7	±9	±12	±16	±21	±27	±32	±36	±40

In addition, the tooltips in the menu have been revised and are now more expressive, e.g.

Views

Transfer part representation in view

In addition to the "normal part properties, parts can also be assigned to properties that apply only to a specific view. You assign these view properties of parts with the functions under **Properties > Properties in view** of the context menu for parts. In addition, individual edges of parts can be shown/hidden on a view-by-view basis using the **Views > Properties > Elements** function.

With the new **Transfer part representation in view** function, the above-mentioned view properties of a part can be transferred to other views as of SP2. The dialogue window shown in the image is displayed for this purpose. The image shows the ISD default setting.

> The function is only available in the context menu for 3-D views. Right-click on a view frame or on the name of a view in the ICN.

The dialogue depends on whether an individual view is active (1) or multiple views are selected in the ICN (2) when the function is called.

Individual view

If an individual view is active, then this view is the initial view. The name of the view will be displayed in the dialogue.

To change the initial view, click on the symbol. If the initial view is correct, select the target views, i.e. the views for which the properties of the initial view are to be applied. If an already selected target view is selected again, it will be removed from the selection list.

Multiple selection

If multiple views are selected, then these are the target views. If you want to add or remove target views, click on

. When the target views are selected, select the initial view in the drawing by clicking the view frame.

After selecting the initial and target views, select the properties to be considered. To do this, activate/deactivate the corresponding checkboxes. You can also use the following buttons for selection:

Activating all checkboxes

Deactivating all checkboxes

Activating the default settings

Only when you click **OK**, the transfer will be started and executed.

Highlighting view lists

As of SP2, disabling the view marking in the Configuration Editor only affects the active view. If a view list is active, i.e. at least one view is highlighted in blue in the ICN, then the views in this list will <u>always</u> be marked by a wider dashed frame.

This affects the parameters

- **Highlight active view in model area** and
- **Highlight active view in drawing sheets**

under **System settings > Visualisation > Views > Frame of active view** in the Configuration Editor.

View captions - Multiple selection

As of SP2, the functions

Create/edit/delete view caption(s) and

Delete view caption(s)

support multiple selection of views. This means that if multiple views are selected in the ICN, the functions act on all selected views. For example, the orientation of multiple views can be changed in one step or the caption can be hidden.
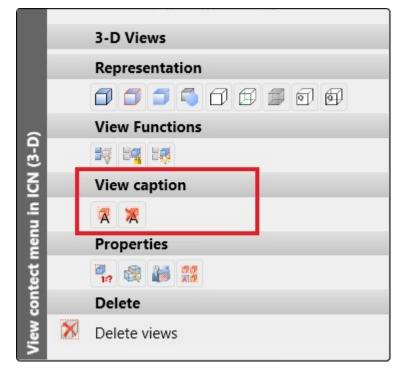
When using the **Create/edit/delete view caption(s)** function, note the following:

If multiple views are selected in the ICN when the function is called, the **View designation** is greyed out, since it generally does not make sense to give multiple views the same designation.

If none of the selected views has a caption, then all views will automatically receive a caption. The caption of the individual view can be further edited afterwards.

If at least one of the selected views already has a caption, the existing captions can be changed. Views without a caption initially remain without a caption. By activating the **Show caption** checkbox, however, captions can also be created and further edited for these views.
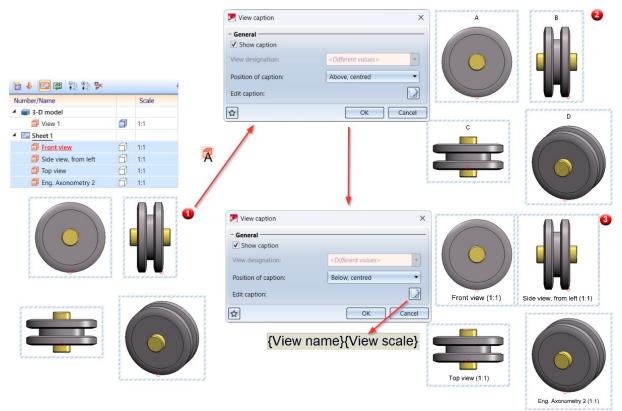
The functions are also available in the context menu when a multiple selections are made in the ICN.

Example:
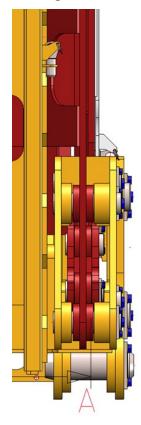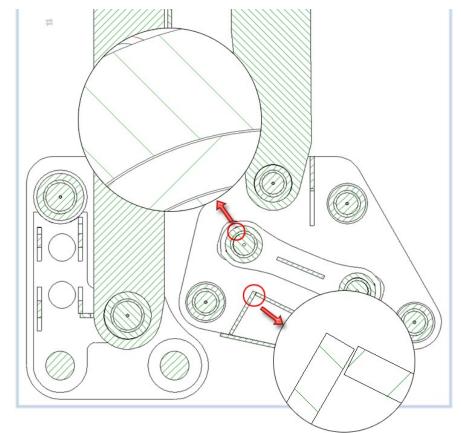
In the image 4 views - all without caption - have been marked (1). If the **Create/edit/delete view caption(s)** function is then called, all views receive a caption - here the view designation (2). If the position of the caption is now changed and edited as shown, the orientation and caption of all selected views will be changed (3).

Sectional/detail views, cut-outs - hatching of adjacent parts

Since HiCAD 2023 SP1, it has been possible to define in the **hatching settings** of the drawing how neighbouring parts are to be visually separated from each other in a sectional view. This can be done by rotating the hatching angle or by vertical mirroring. Previously, parts had to be contiguous without gaps for them to be considered adjacent. As of SP2, smaller gaps between parts are now also possible. This means that the visual borders also apply to "almost neighbours".

Sketches

Transform/clone sketch elements - isolated points

When transforming and cloning sketch elements and isolated points, isolated points were previously not considered if the elements were selected via a rectangle. As of SP2, isolated points are now also considered for this selection type. This affects the following functions in the function groups **Sketch > Transform** and **Sketch > Clone**:

Transform	Clone
Move sketch elements	Clone sketch elements, move
Move sketch elements, move+rotate	Clone sketch elements, move+rotate
Rotate sketch elements	Clone sketch elements, rotate
Scale sketch elements	Clone sketch elements, mirror
Mirror sketch elements	Clone sketch elements, scale
	Clone, copy line elements

The context menu of the line/point selection has been adjusted correspondingly.

Revision of the Move + Rotate and Rotate functions

The **Rotate sketch elements** function has been revised. Here the input of the rotation angle in the dialogue window is omitted.

As of SP2, after selecting the line element to be rotated, the rotation is determined as follows:

- **Planar sketches**

  HiCAD requests the selection of the rotation point and another point. The angle to the X-axis can then be determined dynamically with the cursor on an angle grid.
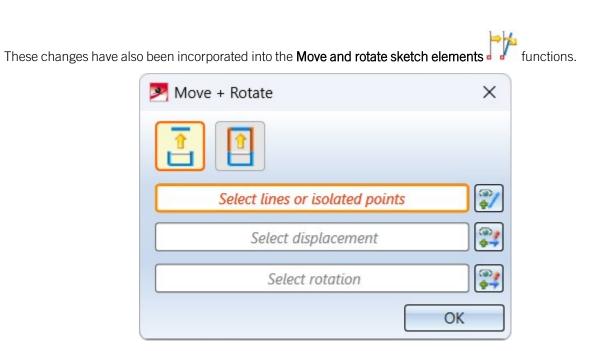
  New in the context menu is the **Angle reference** function. This function does not determine the rotation angle to the X-axis, but to a line between the rotation point and a second point.

- **3D sketches**

  After the direction has been determined, the angle on an angle grid can be determined dynamically with the cursor. If the **Angle reference** function is selected here, the perpendicular is rasterized from the selected point to the selected axis. The angle is then referenced to this line.

By tapping the cursor with the left mouse button, the rotation is applied directly.Instead of dynamically determining the angle of the rotation, it can also be entered explicitly in the HiCAD calculator. To do this, press the space bar, enter the desired angle and press **OK**. After the rotation has been executed, the dialogue window will remain open so that further lines can be rotated directly.

These changes have also been incorporated into the **Move and rotate sketch elements** functions.

Automatic scaling of planar sketches

Automatic scaling of sketches when setting HCM constraints has been optimised further. If the first length, distance or radius constraint is assigned to a sketch, it is checked whether angle constraint (also aperture angles) have already been assigned to the sketch. If so, the **3-D C-edge HCM scaling** dialogue window will be displayed.

Sketch technology - Grid angle

The ISD default setting for the grid angle has changed. As of SP2, the default setting for the grid angle in the Configuration Editor under **System settings > Identification > Grid** is 5°. Note that these grid settings apply not only to the sketch technology, but also to other functions that use distance and angle grids. The changed default setting applies to new installations only.
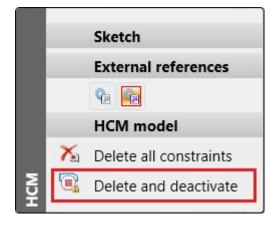
Grid		
Distance	3.333	Percentage of graphic window height
Grid angle	5 °	

Deactivate HCM for active sketch

The automatic setting of HCM conditions for sketches can now also be switched off on a sketch-by-sketch basis, i.e. for the active sketch. This can improve performance for sketches with many lines. For this purpose, the HCM settings for sketches have been extended accordingly.

Alternatively, you can also use the function in the context menu of the ICN.

If the active sketch already contains HCM constraints when you deactivate it, the following message appears:

When you click **OK**, all constraints of the sketch will be deleted.

If the HCM is disabled, the HCM functions in the **Sketch** Ribbon will be greyed out.

Automatic deactivation of the HCM

As of SP2, the HCM for sketches will be automatically deactivated if it exceeds a certain number of lines during 3-D DXF import or when converting 2-D sketches. The number of lines is determined in the Configuration Editor under **System settings > Sketch HCM**. The ISD default setting is 1000.

When importing or converting to HiCAD, the HCM functions of the **Sketch** Ribbon are then greyed out for sketches that exceed this number of lines. If the HCM is activated for such a sketch, the following message will appear:
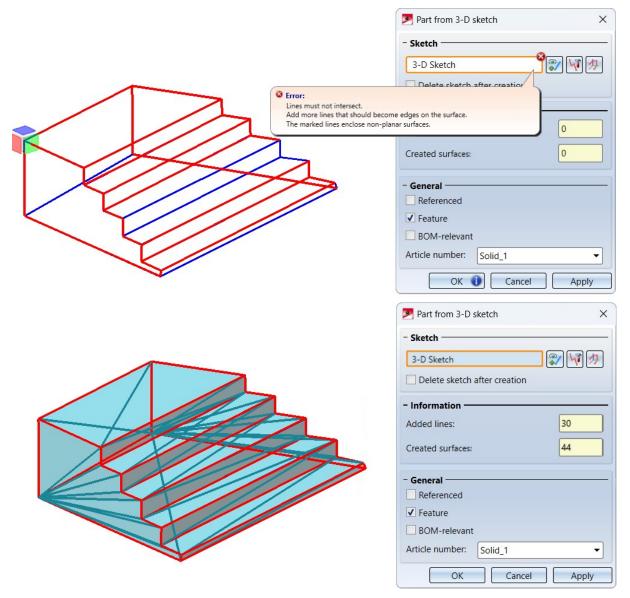
Click **Yes** to activate the HCM.

New part from 3-D sketch - optimised algorithm

The algorithm of the **New part from 3-D sketch** function has been optimised and now leads to better results and ensures smoother surfaces.

Example:
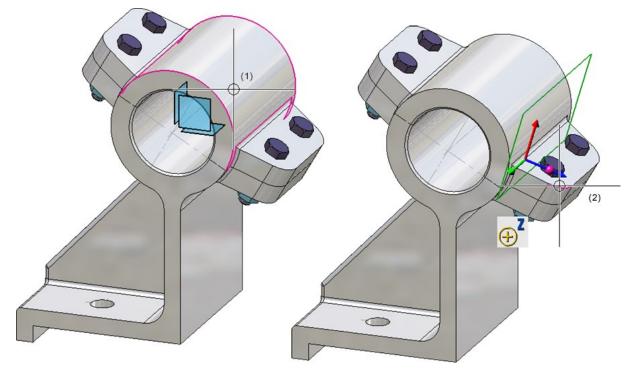
Let us consider the 3-D sketch shown, which obviously represents a staircase - albeit a somewhat skewed one. Until HiCAD 2023 SP1, it was not possible to convert this sketch into a solid, since the sketch contains lines enclosing non-planar surfaces. As of HiCAD 2023 SP2, this is now possible, since the new algorithm automatically adds the missing lines here.

Extended functionality for processing planes

The **Processing plane** function now makes it easier to create processing planes tangential to non-planar, i.e. curved, surfaces. After selecting such a surface, a preview of the processing plane will "hang" on the cursor. The position of the plane can then be explicitly defined by selecting a point, e.g. by selecting a point on an attached part or by using the 3-D point options. The point is projected onto the surface and its projection to the origin of the processing plane. The processing plane is tangential to the selected surface.

(1) Selected surface, (2) Point in the centre of the rounding

Furthermore, the context menu has been extended.

Plane perpendicular to screen plane
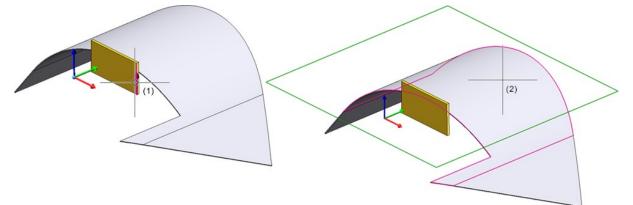
This function creates a plane perpendicular to the screen plane. The Z-direction of the plane is determined by selecting two points or by selecting a line, and depends on the order in which the points are selected or on which line end is closer to the cursor during the selection.

Tangential to surface with direction

The function replaces the previous function **Tangential to surface, parallel to plane**

With this function, you can place the processing plane tangentially to a surface. However, the direction must first be selected. This can be done by selecting two points, a straight line, a surface or a processing plane. Once the direction has been determined, select the surface to which the processing plane is to run tangentially. Only analytical surfaces are permitted.

(1) Direction, (2) Selected surface

Execution of standard processing during the drawing derivation

Standard processings that are inserted using the functions under

Bores, Threads or

Through hole, variable
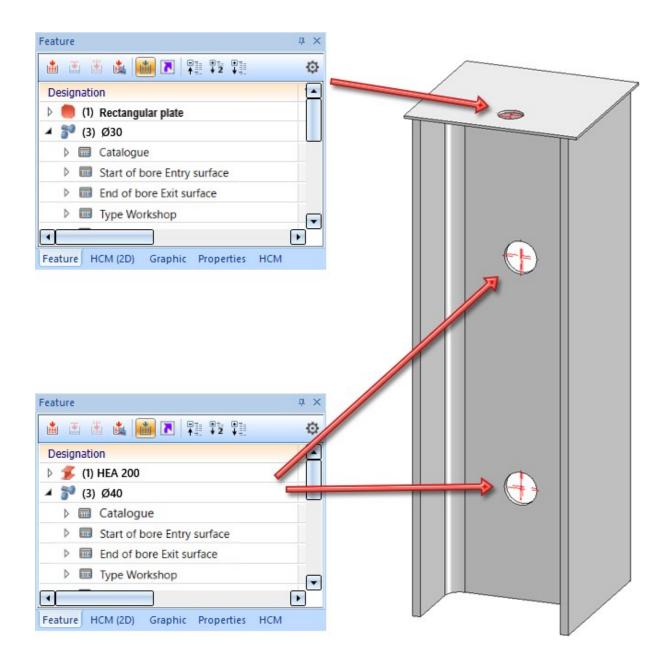
can be assigned a processing ID via the Feature log, which defined where the standard processing is to be performed - on the site or in the workshop. The **Type** parameter is available for this purpose. During the drawing derivation, no distinction was previously made between the different versions when annotating standard processings. This is possible as of SP2.

If this is desired, the following requirements must be met:

- Different annotation templates for workshop and site must be available as FTD files.

- In the Configuration Editor, under **Drawing> Annotations > Annotation template**, the checkbox **Different annotation for workshop and site** must be activated. In the lines below, the annotation templates must be specified including the path. The path can be a HiCAD drive letter or a Windows path.

If these requirements are fulfilled, then during the drawing derivation, standard processing operations are annotated differently depending on the processing ID.

Coat 3-D part

In addition to sheet metal parts and beams, the **Coating** function can now also be used to process general 3-D parts. However, it is not possible to select a different coating for the back side here.

You can find the function under **Sheet Metal > Process > Coating**.

Service Pack 1 2023 (V 2801)

Locking parts with multiple selection

Multiple selection is now also possible when locking or unlocking parts against processing.

Part and dimension orientation

Previously, the functions for determining **Part orientation** and **Dimension orientation** were only available in the context menu for 3-D parts. As of SP1, you can also find them on the **Drawing** tab under **Positioning/Detailing > Orient...**. At the same time, the function icons have been modernised.

Calculation of dimensions - Sorting the attributes length, width, height

If the automatic calculation of dimensions is active, the dimensions of assemblies and parts are automatically calculated by HiCAD and assigned to the attributes

- **Length** ($03),

- **Width** ($02) and

- **Height** ($04)

of the part / assembly. As of HiCAD 2023 SP1, it is now possible to assign the dimensions to these attributes sorted by size, i.e. the largest value is assigned to the length, the smallest to the height. To do this, the new checkbox **Sort dimensions** must be activated in the Configuration Editor under **Modelling > Part properties**. By default, the checkbox is inactive.

Example:

Case 1:

In the following example, the automatic calculation of dimensions is active, but the sorting is inactive. (1) shows the initial situation. Now the height of the extruded solid is changed in the Feature (2). This changes only the **Height** attribute.

Designation	Value
Length	220.000
Width	185.000
Height	41.000

Designation	Value
Length	220.000
Width	185.000
Height	225.000

Case 2:

Calculate dimensions	Always ∨
Sort dimensions	☑

We consider the same example, but now both the automatic calculation of dimensions and the sorting are active. (1) shows the initial situation. Now the height of the extruded solid is changed in the Feature (2). This makes the height the largest value and assigns it to the **Length** attribute. The length value from step 1 becomes the width, the width value becomes the height. This means that all attribute values change.

Switching on the sorting will only take effect after HiCAD has been restarted.

Itemisation

Transferring the part orientation during the itemisation

Parts with the same item number now have the same part orientation - analogously to the dimension orientation. In detail, the procedure is as follows:

- If a part orientation is manually assigned to a part that has not yet been itemised, this orientation will also be transferred to all existing geometrically identical parts, provided that they have not yet been itemised.

- If parts with manually assigned part orientation are present during the itemisation, it will be transferred to all parts with the same item number.

- If new parts are inserted or changed in the drawing, the system will check whether identical parts with a manually assigned orientation already exist. If so, their orientation will be transferred to the new/changed part, provided that no manual orientation has been assigned to this part. If, on the other hand, the new part has a manually set part orientation, it will be adopted for the already itemised identical part.

Itemise parts according to part filters

In the **Itemise part** function, the filters defined for automatic part annotation can now also be used for annotating templates. The dialogue window has been extended accordingly for this purpose.

By selecting this option, you can, for example, use different annotation templates for beams and sheets, sheets and plates or different beam types. The filters that can be set in the Configuration Editor correspond to the filters of the part search via the transparent HiCAD toolbar. To learn how to use these filters, refer to the section **Using part filters for automatic part annotation and itemisation**.

In connection with this extension, the setting of the filters in the Configuration Editor has been moved. As of SP1, you can find the settings under **Drawing > Annotations > Annotation template**.
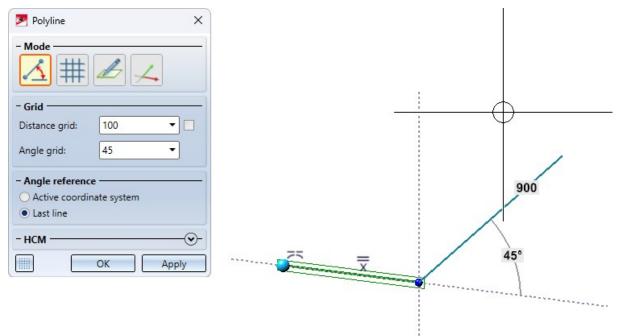
Itemise part - Increment range of numbers

The **Increment range of numbers** now takes the **By top level assemblies** itemisation mode into account. This means that if the **By top level assemblies** itemisation mode is active, only the numbers of the parts that were also itemised in this mode will be increased.
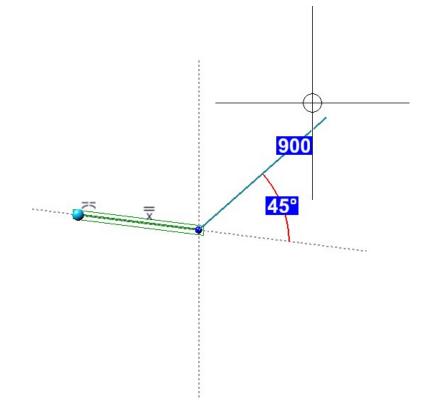
Sketches

Configuration of the graphic display

When working with sketches, especially when drawing sketch elements, various information is displayed in the model drawing, for example, the length of the line, the angle or the radius of the arc.
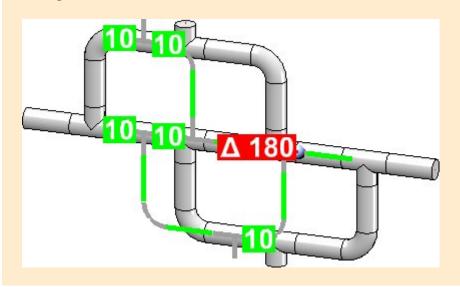
The colour display and text height of these displays can now be changed in the Configuration Editor, e.g.

You can find the settings at **System settings > Identification**.

Please note that the settings apply not only to the Sketch Technology, but also to other functions that use this type of text display. For example, the change of the display height is also considered for the **Change route** function.

Extension when moving and cloning sketch elements

The functions

 Move sketch elements and

 Clone sketch elements
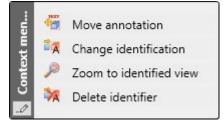
have been extended.

- If a straight line, a surface or an axis is selected to determine the displacement, the distance will now be displayed directly in the calculator and not in the dialogue window as before.

- **Undo** is possible when the dialogue window is open.

- The **Cancel** button has been removed from the dialogue windows. You now use the middle mouse button to cancel the function.

- The graphical representation of the part coordinate system is hidden during the execution of the function.

Views

Move annotation

The context menu of view markings has been extended. The **Move annotation** moves the complete view marking, i.e. symbol and text.

Detail views cuboid/sphere - Conformity with initial view

Analogously to the "normal" detail views, as of SP1 it is now also possible when creating detail views cuboid/sphere to specify whether the detail view should be associative to the initial view or not. For this purpose, the dialogue window has been extended accordingly.

Associativity to the initial view means that the detail view inherits certain changes from the source view. These are:

- cut-outs (including the creation and deletion of cut-outs),
- the section path (if the initial view is a sectional view) and
- hatchings of cut and cut-out surfaces.

Moving master views

A master view, i.e. a view that is superordinate to other views, could previously only be moved together with all linked views. In the detailing process, however, it may be necessary to move only the individual view.
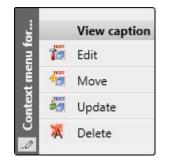
As of SP1, this is now possible. For this purpose, the **Do not move linked views** function is available in the context menu of the **Move view** function.

You activate the context menu during dragging (i.e. while the views are hanging on the cursor) by right-clicking.

Update view caption

New in the context menu for view captions is the **Update view caption** function. This function allows you to update the position and content of view captions.

The function is useful for sectional/detail views, for example. Here the captions are normally centred above the view. If annotations are then set or changed, the caption will no longer be centred. It will adjust its position only when the view is updated. Here the function can be used to restore the alignment. Additionally, the content of the caption will be updated if necessary.

Set fixed view point

New in the **Required position** menu is the function

Set fixed view point

Use this function to fix the active view at a point associative to the geometry.

> You must determine this point with a point option (Autopilot, keyboard or point option menu). Taking over the current cursor position by mouse click is not a point associative to the geometry and therefore not possible as a fixed point.

If a fixed point is already defined for the active view, it will be highlighted in colour in the drawing.
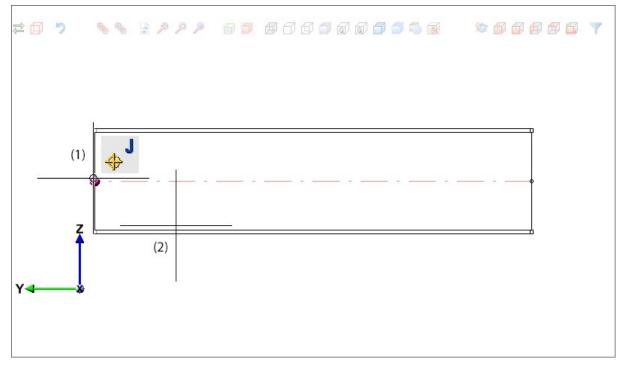
The fixed point is stored by HiCAD in associative form, its projection in 2-D natural coordinates. After changes to the part geometry, the preliminary new projection of the 3-D point is calculated and the view is shifted to that the projection falls back to the previously saved 2-D natural coordinates.

In detail, the following applies:

- If the geometry changes, HiCAD moves the view so that the specified fixed point, which may be in a different location in 3-D, is in the same location in the drawing (2-D). This means that the coordinates of the point in the cartesian XY coordinate system (natural coordinates) do not change.

- If the scale changes, the same applies as for geometry changes. However, this only applies if no fixed point is specified interactively for the scale change.

- If you move the view explicitly, for example manually via Drag&Drop or via the API, the fixed point in the drawing also moves.

- If you rotate the view dynamically around a point (mouse or space mouse), the fixed point in the drawing will shift unless the view is rotated around the fixed point. The fixed point will also shift when you rotate around an axis of the active coordinate system. For all other changes of the projection - even when you rotate around an axis of the world or image coordinate system or when you rotate incrementally - the fixed point will keep its position in the drawing.

.

Example:
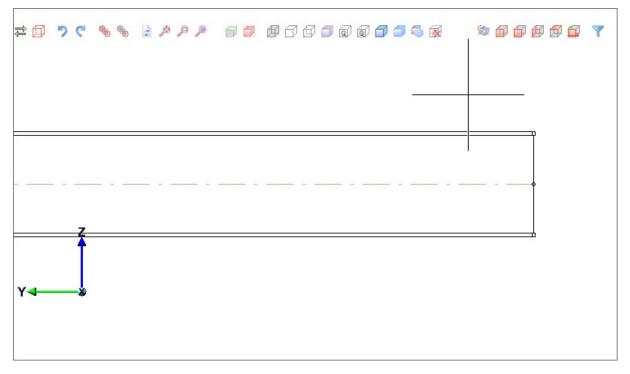
Let us consider the beam shown.

If you set the fixed view point to the start point of the beam axis (1) and select the left beam end (2) for the length change (e.g. 500) then the position of the fixed point will be retained.

Without a fixed point, the result would look like this:

The function is also available in the context menu for views.

Major Release 2023 (V 2800)

Calculation of dimensions and orientation

Until now, automatic calculation of dimensions was only possible for assemblies. As of HiCAD 2023, this feature will also be supported for 3-D parts. If the dimensions of assemblies and parts should be calculated automatically and assigned to the attributes

- **Length** ($03),
- **Width** ($02) and
- **Height** ($04)

of the part / assembly, this must be set in the Configuration Editor.

In addition, you can specify whether the dimensions should also be shown in the drawing.

The display is in the form of the so-called bounding box (1). This is the smallest cuboid that completely encloses the part. In addition, the coordinate system (2) is displayed on the bounding box. The coloured arrows of the coordinate system indicate the direction of the dimensions:
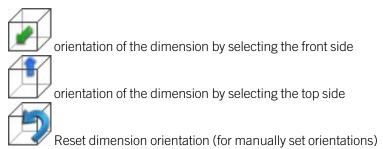
- Red: x-axis (length),
- Green: y-axis (width),
- Blue: z-axis (height).

Additionally, the bounding box is marked by a symbol (3) that indicates the up-to-dateness of the dimensions as well as the source of the orientation.

The orientation of the bounding box (dimension orientation) plays a role in the calculation of dimensions, because it determines which values are assigned to the **Length**, **Width** and **Height** attributes. The dimension orientation is determined according to the following priorities:

1. The orientation is set manually. As of HiCAD 2023, the corresponding features are available in the context menu for parts and assemblies under **Properties > Dimension orientation**:

   orientation of the dimension by selecting the front side

   orientation of the dimension by selecting the top side

   Reset dimension orientation (for manually set orientations)

2. The orientation results from the part orientation.

3. The orientation results from the processing direction.

4. The orientation results from the part coordinate system.

The following table contains an overview of the possible markings on the bounding box.

Symbol	Meaning
	The dimensions are not up-to-date. This occurs, for example, if you have selected the **Only for automatic itemization** setting under **Modelling > Part properties > Calculate dimensions**, since in this case the dimensions are only calculated / updated during the itemisation.
	The dimensions are up-to-date, the dimension orientation corresponds to the part coordinate system.
	The dimensions are up-to-date, the dimension orientation results from the processing direction.
	The dimensions are up-to-date, the dimension orientation results from the part orientation.
	The dimensions are up-to-date. The dimension orientation was set manually.
	The dimensions are up-to-date, the dimension orientation corresponds to the part coordinate system of the main part of the assembly.
	The dimensions are up-to-date, the dimension orientation results from the processing direction of the main part of the assembly.
	The dimensions are up-to-date, the dimension orientation results from the part orientation of the main part of the assembly.
	The dimensions are up-to-date. The dimension orientation of the main part of the assembly was set manually.

If parts/assemblies are to be excluded from the calculation of dimensions, this can be specified - as before - by assigning the **Ignore for dimensions** (#NDR) attribute.

Restrictions on the calculation of dimensions

Not all parts/elements are included in the calculation of dimensions.

Excluded from the calculation of dimensions	
Parts / Assemblies	▪ Sheets, if dimension attributes are filled via the development calculation. ▪ Parts and assemblies of pipeline constructions. Here the attributes are determined according to a separate logic as before. ▪ Structure assemblies The above-mentioned parts / assemblies are considered in the calculation of dimensions of superordinate parts / assemblies.
Geometrical elements	▪ Free edges, ▪ bore / bolting axes, ▪ isolated points. These elements are generally not considered.
Parts in assemblies	▪ CONTOUR parts of series beams, ▪ dummy parts, ▪ sketches and 3-D sketches, ▪ standard and purchased parts with the production type **Site assembly**, ▪ standard and purchased parts in compresses representation, ▪ standard grids, ▪ parts that are marked as not relevant for dimensions. These are parts that have been assigned the **Ignore for dimensions** (#NDR) attribute. These parts are not considered when calculating the dimensions of assemblies.

Restrictions on the dimension orientation

Changing the dimension orientation is generally not possible for parts that are locked against processing. However, they are considered when calculating the dimensions.

There is also a restriction for steel engineering plates, glasses and beams.

- **Steel engineering plates, glasses**
  The height must point in the direction of the sheet thickness.

- **Beams**
  The length must point in the direction of the beam axis, the height corresponds to the web.

Lock parts against processing

Until now, locking parts against processing was only possible for externally referenced parts. As of HiCAD 2023, this is now also supported for non-referenced and internally referenced parts. Locking against processing can be useful, for example, to protect an adjacent structure from unintentional processing.

This extension means that the functions for locking parts against processing are now available in a separate menu and not in the **Referencing functions** menu as before. You can access the function via the context menu for parts and there via the entry **Lock against processing**.

Dimension parameters

Saving the dimension parameters with the drawing

As of HiCAD 2023, the current settings of the **Set parameters for new dimensions** function are saved together with the drawing, i.e. in the corresponding SZA file, when a drawing is saved. When the drawing is loaded again, these parameters will then automatically be active.

After starting HiCAD, when creating a new drawing and when loading drawings saved with a version prior to HiCAD 2023, the parameter values set in the Configuration Editor under **Drawing > Annotations > Dimensioning, 3-D** will be active.

Selecting the bracket type for the 2nd unit of measurement

The 2nd unit of measurement was previously marked with round brackets. As of HiCAD 2023, the bracket type can now be selected:

- Round brackets,
- Square bracket or
- Without brackets.

You can change the setting on the **Dimension figure** tab of the dialogues for the units of measurement. This applies to general dimensions as well as to parameter and HCM dimensions.
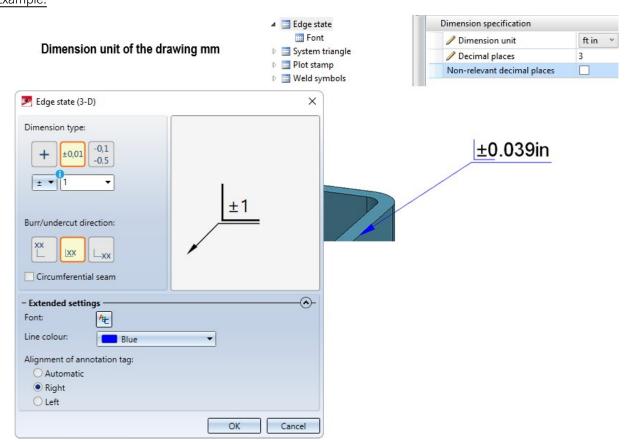
The preset can be defined in the Configuration Editor under **System settings > Annotations > Dimensioning, 3-D**.

Edge state – Dimension unit

The dimension unit selected in the Configuration Editor under **Drawing > Annotations > Edge state** is used to display the edge state. It can be selected independently of the dimension unit of the drawing.

<u>Example:</u>

The dimension unit selected for the edge state in the Configuration Editor applies to subsequently created annotations. Annotations that already exist in a drawing and that were created with different settings will not be automatically converted. When editing these annotations, the corresponding values will be marked in the dialogue.
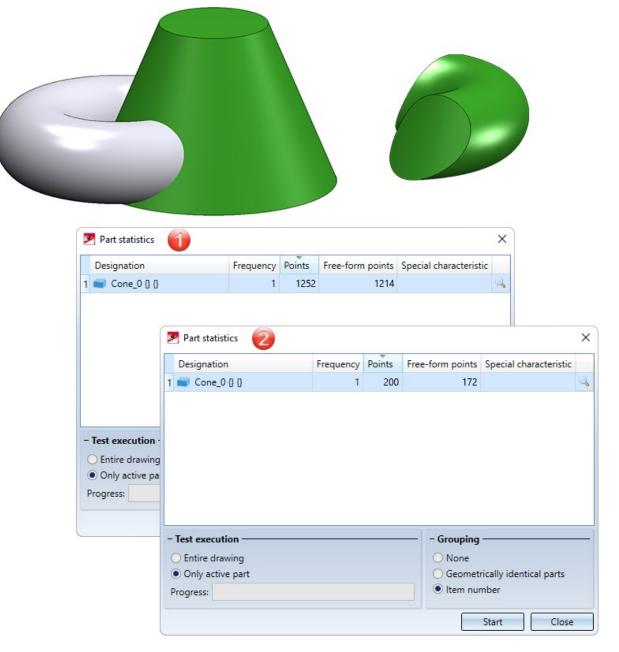
The dimension unit is not output if millimetres "mm" is selected for both the unit of length of the drawing and the dimension unit of the edge state.

Improved surface cuts between torus and cone/cylinder/torus

The algorithm for surface cuts between toruses and cones, cylinders or further toruses has been revised. This results in significantly fewer free-form points, which leads to improved performance and better quality results.

In the image, the intersection of the cone and the torus has been created. (1) shows the number of free-form points before HiCAD 2023, (2) shows the number in HiCAD 2023.

3-D grid

The dialogue window of the **3-D grid** function has been modernised.

Another new feature is that you can use formulas instead of values when specifying the distance - even in mixed units. The value will then be converted to the unit of measurement of the drawing.

Example:

Unit of measurement of the drawing: mm; Distance: 1 3/4" + 12.5 mm ➜ Result: 56.95 (mm)

Sketches

Sketch technology - Presets distance/angle grid

The value for the angle grid when **drawing planar sketches and 3-D sketches** can now be preset in the Configuration Editor under **System settings > Identification** in the **Grid** area.

Grid		
Distance	3.333	Percentage of graphic window height
Grid angle	5 °	

In this context, the preset for the distance grid has also been moved to this area.

> Please note that the specified grid settings not only apply to the sketch technology, but also to other functions that use distance and angle grids. One example is the new insertion function for beams.
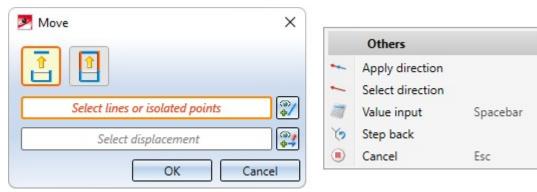
Transform sketch - Move line elements and points

The functions

Sketch > Transform > Move and

Sketch > Transform > MovePts

have been revised and optimised. The distance of the displacement is now no longer specified in the dialogue window, but can be set dynamically on a distance grid in X and Y direction. In addition, you have the option to freely select the displacement direction via a context menu.

Clone sketch - Clone, copy line elements

The function icon of the **Clone GE, Move** function has changed .

New is the function

Clone, copy line elements.

This function creates a copy of line elements/isolated points of a sketch that can be placed multiple times at different locations.
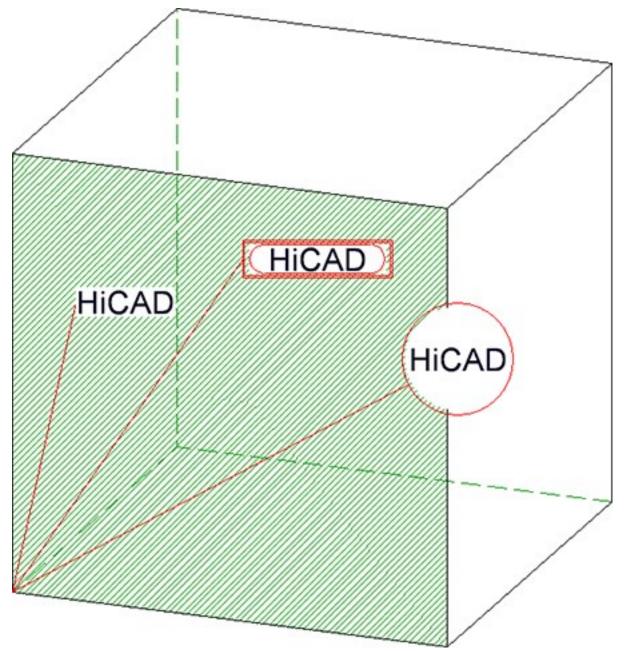
Referenced parts and itemisation

If an already existing referenced part is inserted in a drawing that is an itemised source model, the item number of the part (if present) will be invalidated and thus be shown crossed out (or marked with *). When subsequently itemised, the part receives a change marker in the ICN. This also applies to all its identical parts. Thus, further updates of the referenced part and its item number are always guaranteed.

If the current drawing is not an itemised source model, the item number of the part will be loaded/updated according to the corresponding KRA file.

Background cut-outs for annotations
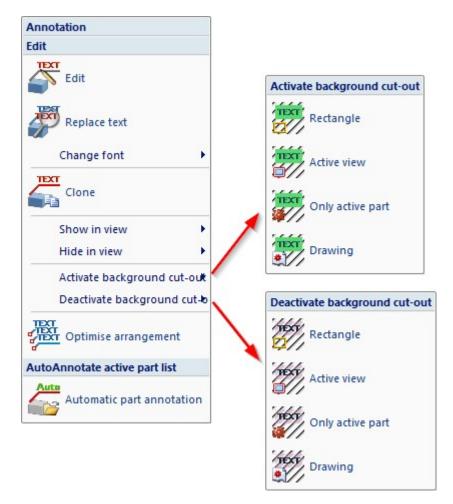
In the case of background cut-outs of annotations, the interruption area was always rectangular up to now. As of HiCAD 2023, the frame selected in the annotation settings is taken into account. This means that circular interruption areas are also possible, for example.

- Background cut-outs are possible for all annotation functions. After HiCAD has been started, the setting defined in the Configuration Editor under **System settings > Annotations > Part annotation > Interrupt background lines** will be used for new annotations. The ISD default setting is ☑ **Exclude background**. If this setting is changed during the current HiCAD session when a new annotation is created, this will apply as the new default setting for subsequently created annotations.

- For annotations from drawings created with HiCAD 2022, the setting from the Configuration Editor is used when drawing or editing for the first time. Annotations from versions prior to HiCAD 2022 must be converted for this purpose. Whether and when the conversion should take place can be defined in the Configuration Editor under **Compatibility> Annotations > Part annotation, 3-D**.

- Background cut-outs can also be set or removed in one step for multiple annotations. Corresponding functions are available under **3-D Dimensioning + Text > Text > Part annotation, with free text and leader line > Activate background cut-out** or **3-D Dimensioning + Text > Text > Part annotation, with free text and leader line > Deactivate background cut-out.**

Associative detail views

As of HiCAD 2023, when creating detail views, you can specify whether the detail view should be associative to the initial view or not. For this purpose, the **New detail view** and **Change detail view** dialogue windows have been extended accordingly.

Associativity to the initial view means that the detail view inherits certain changes from the initial view. These are:
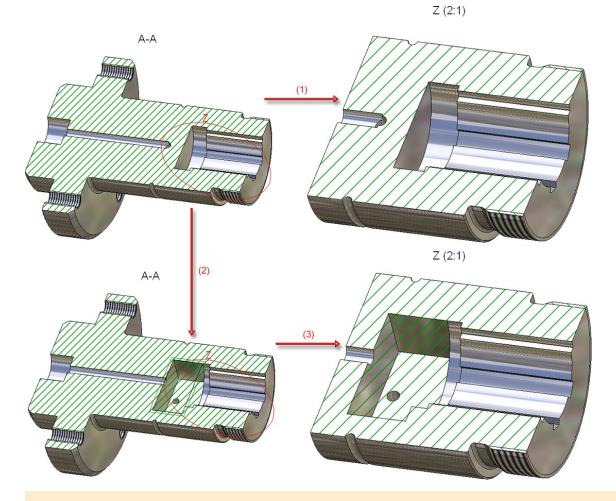
- cut-outs (including the creation and deletion of cut-outs),
- the section path (if the initial view is a sectional view) and
- hatchings of cut and cut-out surfaces.

Please note the following instructions if the checkbox is active (ISD default setting).

- Associativity must be set when defining the detail view. It cannot be set subsequently.

- Cut-outs and section paths that are inherited, i.e. originate from the initial view, cannot be edited in an associative detail view. This is only possible in the initial view. However, the detail view can also have its own cut-outs, which can then of course also be edited in the view.

- An associative detail view cannot change its initial view. Marking the detail outline is possible only in this specific initial view.

- When deleting the initial view, the associativity of the detail view is lost. However, the detail view itself does not change.

- The associativity of a detail view can be removed with the **Change detail view** function. This does not change the detail view, it just no longer follows the changes of its initial view. Cut-outs ans section paths originally "inherited" from the initial view become the detail view's "own" cut-outs and section paths.

Example:

The figure shows a section view A-A with an associative detail view Z derived from it (1). Subsequently, a cut-out has been inserted into the sectional view (2). This cut-out is then transferred to the detail view after updating (3).

> Existing detail views from older drawings (before HiCAD 2023) are non-associative and cannot be set to associative.

Catalogue Editor
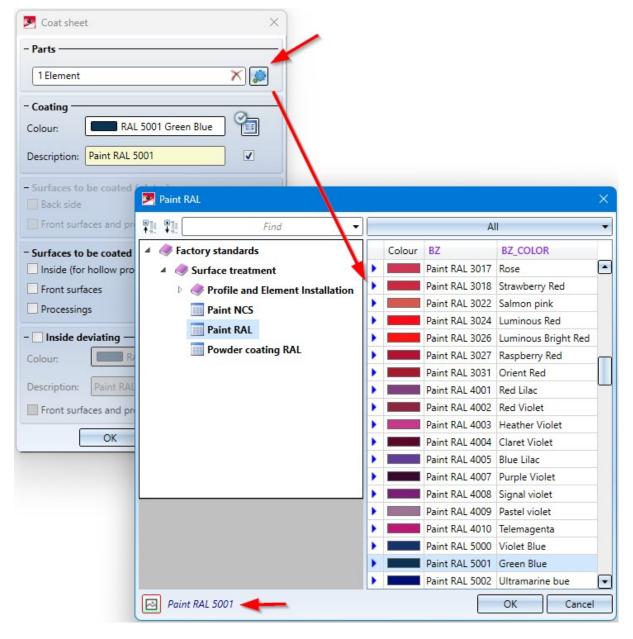
Service Pack 2 2023 (V 2802)

Paint/Powder coating RAL

The table **Paint RAL** has been extended by an additional column **BZ_COLOR** which contains the name of the respective RAL colour.

The same applies to the new table **Powder coating RAL**. Both tables can be found in the catalogue **Factory standards > Surface treatment**.

The additional column simplifies the colour selection in the catalogue for the function **Coating**. A preview of the colour and the colour name is now displayed next to the colour name during the colour selection in the catalogue. The number of the RAL colour is displayed in the lower left corner of the dialogue window.

FLUTZ profiles

The catalogue **Factory standards > Factory beams > FLUTZ profiles** has been expanded by round pipes. The corresponding table is HR-QR.

New columns for table Grid (DIN 24537)

The table **Grid (DIN 24537)** at **Semi-finished products > Gratings** has been expanded by the following columns:

- **S** Bearing bar diameter,
- **AXB** Mesh pitch and
- **R** Slip resistance.

Service Pack 1 2023 (V 2801)

Fastenal - Thread cutting screws

The catalogue **Factory standards > User-defined fasteners > User-defined bolts+screws > Fastenal > Self-drilling screws** has been extended.

The following tables are new:

- Hex-Unslotted Hex Washer Head *
- Philips Drive Pan Head *
- Philips Drive Flat Head

* The tables replace the previous tables of the same name. The new tables have been created in the unit inch.

Hexagon socket set screws with cup point
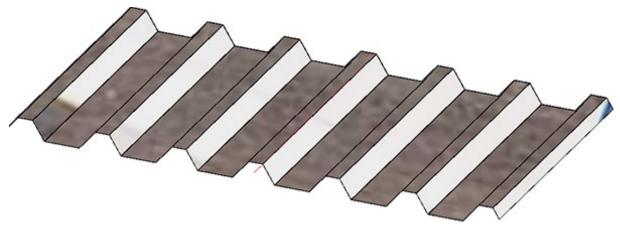
The catalogue **Factory standards** now contains new set screws according to US standard. You can find the hexagon socket set screws and cup point at **Factory standards > User-defined fasteners > User-defined bolts + screws > User-defined set screws >Hex Drive Cup Point Socket Set Screw**.

ATAS Belvedere profiles

At **Factory standards > Series > Roof Wall Facade > Room-closing profiles > ATAS**, Belvedere profiles are also available in the **Exposed Fastener** catalogue as of HiCAD 2023 SP1.

- Belvedere 6" Short Rib
- Belvedere 7.2" Rib
- Belvedere PenumWall
- Grand C
- Grand V
- Span Wall

Updated SAB profiles

The sandwich panels of SAB have been adapted to the changes of the profile manufacturer. This means that in the catalogue **Factory standards > Series > Roof Wall Facade > Room-closing profiles > SAB**, SAB's sandwich panels have been updated, new profiles have been added and old ones have been marked as deleted. This affects the following tables:

- Sandwichpanels, roof\SAB D,
- Sandwichpanels, wall\SAB W and
- Sandwichpanels, wall\SAB WB.

Hexagon steel DIN EN 10278

In the catalogue **Semi-finished products > Beams+Profiles > Hexagon steel** the table DIN 176 (standard obsolete) has been replaced by the table **DIN EN 10278 6KT**.

Major Release 2023 (V 2800)

Units and Unit categories

The Catalogue Editor now supports **Unit categories** (e.g. Length, Surface area, Volume, Weight) and **Units** (e.g. mm, m , in, ft, kg) as **Column properties**.

In the future, the units are to be set in all ISD catalogues so that when they are installed in HiCAD, it is evident in which unit the standard parts are available in the catalogue. In many catalogues, e.g. in the new US standard parts, this has already been implemented, in the other tables this will be done with the coming Service Packs.

You can also use this option in your own tables. To do this, right-click on the column header and assign the unit category and unit to the column.

New shear connector cartridges

The following, new Hilti objects have been added to the catalogue **Purchased/Factory standard parts > Shear connector cartridges**:
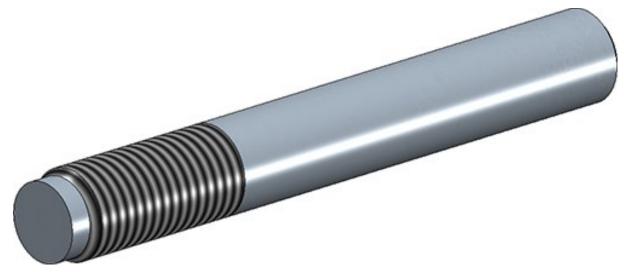
- HIT-HY 10 Plus at **Mortar > Hilti**,
- HIT-HY 200 at **Mortar > Hilti** and
- HIT-SC 16x85 at **Sieve sleeve > Hilti.**

Other US Fasteners

EJOT Bolt anchors

In the **Factory standards > User-defined fasteners > User-defined anchors** catalogue, four new tables with bolt anchors are available under **EJOT**:
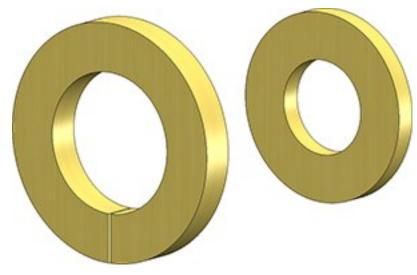
- BA-E PLUS
- BA-E_PLUS HCR
- BA-F PLUS
- BA-V PLUS

Washers

In the catalogue **Factory standards > User-defined fasteners > User-defined washers > Fastenal** two new tables are available:

- Flat Washers > General Purpose Flat Washers and
- Lock Washers > Split Lock Washers.

Bolts+Screws

In the catalogue **Factory standards > User-defined fasteners > User-defined bolts+screws > Fastenal** new tables for bolts and screws are available:

Catalogue	Table
Bolts	▪ Hex Cap Screw
Lag Screws	▪ Hex Head Lag Screw
Sheet Metal Screws	▪ Intended Hex Washer Head Slotted * ▪ Phillips Drive Flat Head ▪ Phillips Drive Pan Head
Machine Screws	▪ Phillips 100° Flat Head ▪ Phillips Drive Flat Head ▪ Phillips Drive Pan Head
Thread Cutting Screws	▪ Phillips Drive Flat Head ▪ Phillips Drive Pan Head

* replaces the previous table

Processings

The following tables have been added to the catalogue **Processings, general > Processing**:

Catalogue	Table
Bores	▪ ASTM INCH
Countersinks	▪ Countersunk Flat Head ▪ Countersunk Socked Head

Hexagon head screws

In the catalogue **Factory standards > User-defined fasteners > User-defined nuts > Fastenal** three new tables are available:

- ▪ Hex Nuts
- ▪ Hex Jam Nuts
- ▪ Machine Screw Hex Nuts

Rivets

In the **Factory standards > User-defined fasteners > User-defined rivets > Fastenal** catalogue, the previous **Rivet Dome Head** table has been replaced. The previous table has been marked as deleted.

The same applies to the table **Rivet Dome** in the catalogue **Factory standards > User-defined processings > User-defined bores for rivets**.

Groove form for easy fiX milling cutter

In the **Factory standards > Composite panels, groove form** catalogue, the **Standard groove form** table has been expanded to include a groove form for easy fiX milling cutter 135°, 3mm. This table is used, among other things, for attaching and folding flanges with milling edge zones.

US Materials

The following US materials have been added to the **Materials** catalogue.

Non-ferrous metals > Aluminium alloys	Steel > Structural steels	Fine steel > Fine steel alloys
▪ AA 6060 ▪ AA 6061 ▪ AA 3003 ▪ AA 3004 ▪ AA 3105 ▪ AA 1050A ▪ AA 5005	▪ A36 ▪ A283A ▪ A283B ▪ A283C ▪ A283D ▪ A284A ▪ A284B ▪ A284C ▪ A284D ▪ A656	▪ AISI 304 ▪ AISI 316-TI ▪ AISI 302

Sheet Metal parts according to US standard

New in the catalogue **Factory standards > Sheets** is the table **ISD US sheet**. The table contains sheet metal parts of different thicknesses (in inches) according to US standard.

ATAS room-closing elements

In the catalogue **Factory standards > Series > Roof Wall Facade > Room-closing profiles > ATAS**, new **Sandwich profiles** will be available from HiCAD 2023 onwards. The catalogue **Sandwich panels, wall** contains five multi-part profile types:

Sandwich panels

- Isoleren WL,

- Isoleren IM,

- Isoleren ML,

- Isoleren SL and

- Isoleren TE.

New usage for Sheet Metal parts

For Sheet Metal assemblies, i.e. assemblies with a Sheet Metal part as main part, a new Usage called **Sheet Metal Assembly** is now available. The catalogue **Factory standards > Usage** has been expanded accordingly.
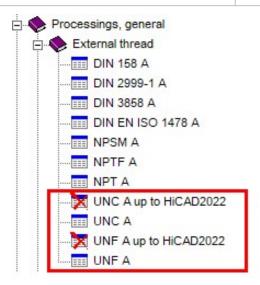
Further information can be found here.

New UNC and UNF threads

The previous tables for UNC and UNF threads have been replaced by new revised tables. The previous tables have been renamed and marked as deleted; fasteners using these threads have been adapted accordingly.

Catalogue	Table up to HiCAD 2023	Table from HiCAD 2023 onwards
Processings, general > External thread	UNC A up to HiCAD 2022	UNC A
	UNF A up to HiCAD 2022	UNF A
Processing > Thread	UNC up to HiCAD 2022	UNC
	UNF up to HiCAD 2022	UNF

The new tables have been created in the unit "inch".

Point clouds

Service Pack 2 2023 (V 2802)
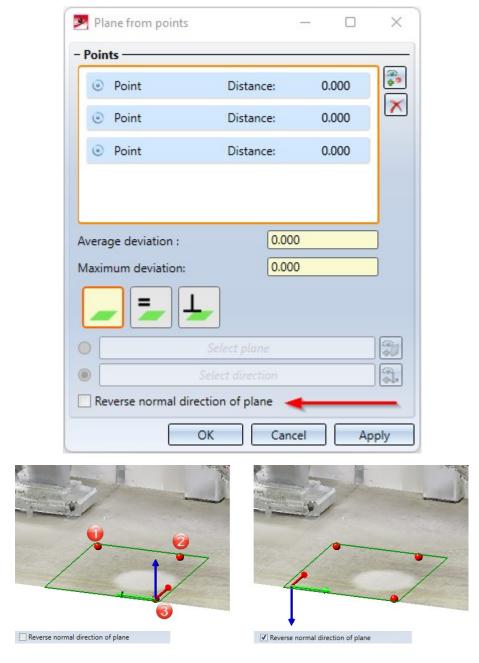
Cancelling e57 file conversions

The conversion of e57 files will be instantly cancelled, for example, if the points are so far away from the origin that display errors occur. Then start the conversion again with the option **Move point cloud into origin** or change the position of the cloud in the e57 file.

Major Release 2023 (V 2800)

Plane from points

With the function **Plane from points** the plane is now determined as with the function **Select plane from point cloud**, i.e. the Z-direction of the plane points to the viewer. In order to be able to reverse the normal direction if necessary, the **Plane from points** dialogue box has been extended by the checkbox **Reverse normal direction of plane**.

Changing the point cloud reference

If a corresponding point cloud was not found under the saved path (the point cloud reference) when opening an

SZA file and this was then changed manually, then the point cloud was provided with a change mark in the ICN, since it was not ensured that the point clouds were identical.

As of version 2023, HiCAD automatically checks whether the new path contains the same point cloud or a modified point cloud. The change mark will then only be set in the latter case.

Feature Technology

Service Pack 2 2023 (V 2802)

Indication of deviating units

If a part is inserted into a drawing created with a different unit, this will be indicated in the feature ICN by displaying the original unit.

A Sheet Metal part created in an inch drawing is inserted into a millimetre drawing.

The change of values is done with the pocket calculator in the original unit.

The depth of the sheet is changed from 4 to 6.

If, for example, you change the feature of a cone in the creation dialogue, the original unit will also be displayed here.

In dialogues, input fields containing deviating units are indicated with a blue info icon.
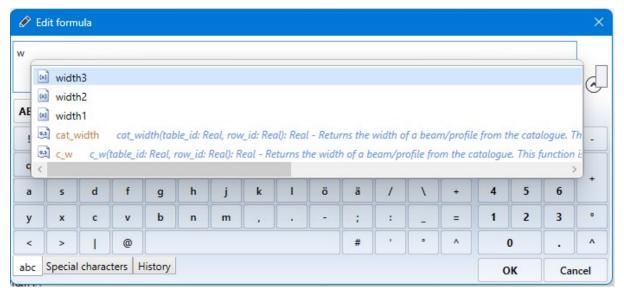
Change assembly coordinate system

Note that this may also change the geometry and position of the parts in the drawing. Referenced parts will change at all points of use.

The isolated point set with the point option **K** refers to the part coordinate system of the assembly. If you change the coordinate system, the position of the point and all elements that refer to the point will also change.

Selection of already created variables

From HiCAD SP2 onwards, the part variables that you have already created for the assembly will also be displayed when choosing the **Edit formula** function. One character is sufficient for auto-completion.

To display all formulas and variables, place the cursor in the input field and enter the key combination **CTRL + Space bar**.

Automatic recalculation after changes

If the **Automatic recalculation** is deactivated(off) in the toolbar of the feature window (ICN), from HiCAD 2023 SP2 onwards the automatic feature recalculation in the variable window (ICN) is no longer deactivated as well. The calculation can now be carried out independently of each other.

(1) Automatic recalculation off
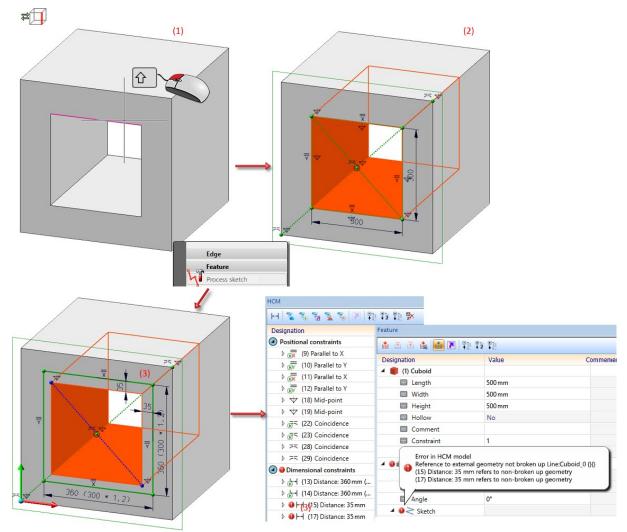
(2) Automatic feature recalculation on

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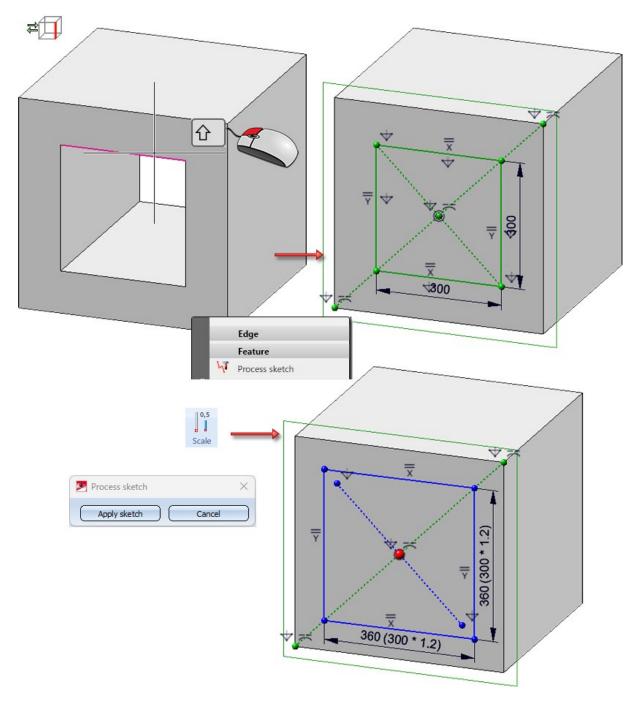
Sketch feature - Element snap mode

The user guidance for the **Process sketch** feature has been changed.

Previously, in **Element snap** mode, holding down the SHIFT key while clicking the right mouse button (1) on a sketch element highlighted the sketch and geometry in colour (2). This made the selection of sketch elements difficult in many cases. In addition, when editing the sketch, for example, HCM constraints could be set that referred to elements that did not actually exist, which could lead to feature or HCM errors. The following figure shows such an example. Here, when changing the sketch of a recess, an HCM dimension (3) was set from the subtraction contour to the sketch.

As of HiCAD 2023 SP1, the user guidance changes as follows: If a sketch element is selected in the drawing with SHIFT+ right mouse button in the **Element snap** mode and then the **Process sketch** function is called up in the context menu for edges, then only the sketch is highlighted in colour and calculated to the initial position. This avoids feature and HCM errors - as shown above - and simplifies the selection of sketch elements. This behaviour also applies when double-clicking on the **Sketch** entry in the feature as well as for all cases with which a Sketch feature can be edited.

Parameterization of string variables via formulas

Up to now, only variables of type **Number** in the variable table could be controlled via a formula. As of HiCAD 2023 SP1, **String** variables can also be parameterized.

To insert a string variable into a comment, for example, proceed as follows:

Create the **Number Variables** l, b und h for the Length, Width and Height of a cuboid (1). Then create the **String Variable Comment** (2). Enter the formula for the new variable using the **Edit formula** function from the context menu (3).
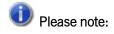
Enter the formula **real_to_string(l) + 'x' + real_to_string(b) + 'x' + real_to_string(h)**.

Comments or the **Designation** of parts can now be parameterized in the Feature.

Designation	Value
▲ 🟥 (1) Cuboid	
▦ Length	100 mm (l)
▦ Width	100 mm (b)
▦ Height	200 mm (h)
▦ Hollow	No
▦ Comment	100x100x200 (Comment)
▦ Constraint	1
▦ Visualisation	-1
➡ (2) --- Insertion Position ---	

ⓘ **Please note:**
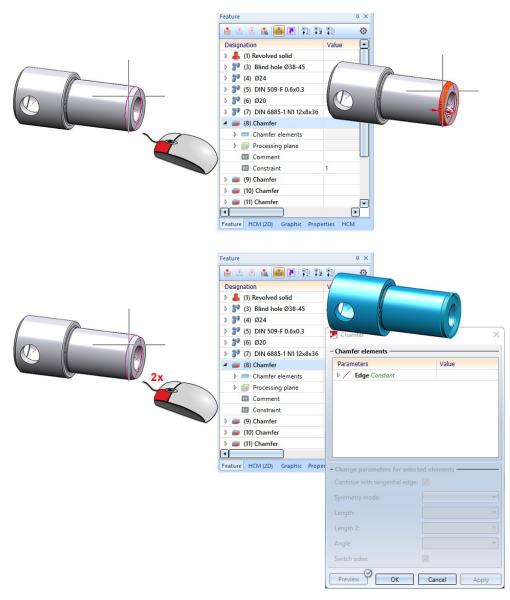
All formulas that are not used for a numerical value can only be entered using the **Edit formula** function in the context menu. This also applies to string variables. A direct input when creating the string variables with the **Value** option is not evaluated.

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Edit feature - Element snap mode

If the identification mode **Element snap** is active, then clicking with the left mouse button on a geometry will mark the associated feature in the ICN. Double-clicking this feature opens the corresponding processing dialogue:

You activate the **Element snap** mode on the transparent toolbar:

New docking window for variables

The docking window for variables has been completely revised for HiCAD 2023. In contrast to the old variable window, the entries no longer have to be saved or discarded using **OK** or **Cancel**. Instead, you can activate or deactivate

the **Automatic feature recalculation** . If it is activated, a recalculation takes place after every action in the variable window. An exception is the change of the comment.

	Part variables / View variables	With this function you switch between part variables and view variables.
		If the **Automatic feature recalculation** is active, then the views are recalculated. By including the view variables in the docking window, the function
		View variables at **Drawing > Properties > Attr...** could be removed.
	Activate/deactivate variables of active feature log	This filter restricts the view to all variables used in the feature log of the active part.
	Activate/deactivate variables of active HCM model	This filter restricts the view to the variables used in the active HCM model.
	Add variable	With this function you activate the dialogue for adding variables. The variables are added to the active part or view variables.
	Mark unused variables	All unused variables are marked. The user can delete the marked variables afterwards via the context menu.
	Variables, where-used	This function opens the **Use of variables** dialogue window, which lists for all variables of the selected part where (and whether) they are used in other places in the drawing.
	Recalculate	With this function you start the feature recalculation of the active part or the recalculation of all views.
	Activate/deactivate automatic feature recalculation	You can switch the automatic feature recalculation or the recalculation of all views on or off with this function.

Delete and Clear assignments under Parameterisation

Two functions are now available for deleting the parameterisation of a Design Variant:

- Right-click on the main entry **Parameterisation** and select **Clear**. All parameterizations of the Design Variant will then be deleted.

- Right-click a parameterisation in the sub-menu and select **Delete**: Then only this parameterization will be removed.

HCM

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Change assembly coordinate system

Like every part, an assembly also has a part coordinate system. This coordinate system is either determined automatically by HiCAD or defined by you when creating the assembly. For this purpose, a corresponding parameter is available in the Configuration Editor at **Modelling > Part creation > Assembly**.
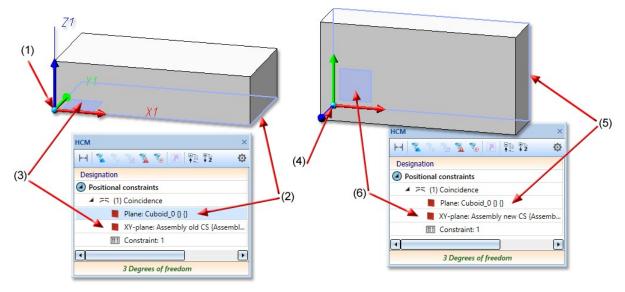
To move the part coordinate system of assemblies, the function **Change assembly coordinate system** (Drawing > Others > World CS > ...) is available.

⚠ Important:

Note that this may also change the geometry and position of the parts in the drawing. Referenced parts change at all points of use.

Example:

Coincidence (3-D Part HCM) exists between a plane of the cuboid and the XY-plane of the part coordinate system of the assembly. If you change the position of the coordinate system, the position of the cuboid will also be adjusted.

(1) Coordinate system of the assembly, (2) first plane for the coincidence is the lower surface of the cuboid, (3) second plane of the coincidence is the XY-plane of the coordinate system

(4) assembly coordinate system changed, (5) (6) coincidence rotates the plane of the cuboid back to the XY-plane of the coordinate system

Delete all part constraints in an assembly

When deleting the HCM constraints of a 3-D part, you can now also remove the constraints of its subordinate parts. You can reach the function via the pop-up (right-click after calling the **Constraints** function).

Deactivate HCM for active sketch

You can now deactivate the automatic placing of HCM constraints for the active sketch. This can improve performance for sketches consisting of many lines. Deactivating the HCM deletes all constraints and greys out the HCM functions in the **Sketch** tab.

The option can be found on the **Sketch** tab at **HCM > Tools** ⌄ **> Settings**. Alternatively, you can call up the function in the context menu of the HCM ICN with the function **Delete and deactivate** ⬚.

Automatic deactivation of the HCM

The HCM is automatically deactivated when sketches exceed a certain number of lines during 3-D DXF import or 2-D sketch conversion. You set this number in the Configuration Editor at **System settings > Sketch HCM**. The ISD default setting is 1000. If you activate the HCM constraints for such sketches, there may be considerable waiting times.

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Delete HCM constraints

The Delete functions for HCM-constraints have been merged. With the newly created function, **Delete constraint** , you delete individual HCM constraints of the active 3-D part or in the C-Edge HCM of the sketch.

When you identify a constraint, it will be deleted immediately. Use the middle mouse button to end the function. Alternatively, you can delete constraints directly in the ICN.

With a right-click you call a pop-up menu with further functions:
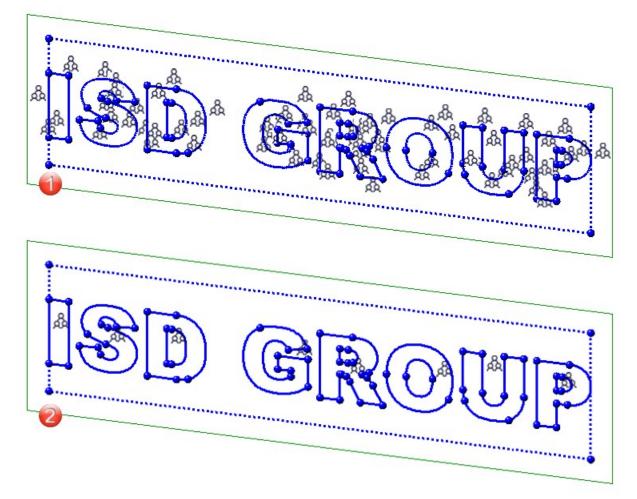
	Constraints in rectangle	With this function you delete constraints from the active 3-D part in the rectangle.
	All constraints	If you want to start again from scratch when parameterising a sketch, you can use this function to delete all HCM constraints of an (active) sketch with one mouse click.

Please note:

This function also removes the HCM constraints for all sub-assemblies of the active part.

Symbol for groups

As of HiCAD 2023 SP1, there is only one symbol for a **Group** and no longer one symbol for each line element of the group.

(1) Identification beforeHiCAD 2023 SP1

(2) Identification as of HiCAD 2023 SP1

Configuration Management

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Highlighting of list views

The deactivation of the parameters

- **Highlight active view in model area** and
- **Highlight active view in drawing sheets**

at **System settings > Visualisation > Views > Frame of active view** only affects the active view as of SP2. If a view list is active, then the views of this list are always marked by a wider dashed frame.

Automatic data save: Higher performance

The behaviour of the automatic data save with regard to the polygon model of parts can be influenced in the Configuration Editor at **System settings > Data save** via the checkbox **Quick data save**. The checkbox is active by default. This improves the performance of the automatic data save for drawings with many parts.

Default settings changed

The following ISD default settings in the Configuration Editor have changed during a new installation (no changes during updates). The settings are independent of the selection in ParConfigComp.

- **System settings > Identification > Grid angle**: The grid angle when drawing lines in sketches has been changed from 15 degrees to 5 degrees..
- **System settings > Annotations > Dimensioning, 3-D > Interactive dimensions > Dimension assignment**: 3-D dimensions are not assigned to the active part, but to the part to which the first dimension base point belongs.
- **Steel Engineering > Representation > Type of beam representation**: Beams are now created in exact representation.

Procedure for dimension calculation

Beams

In the Configuration Editor at **Modelling > Part properties** there is a new parameter **Calculate only length for beams**. This allows the automatic calculation of the beams to be limited to the length as before. Height and width are taken from the catalogue if necessary. This parameter is deactivated in the ISD default setting. The restriction only applies if the dimension calculation is activated.

Plates

If the automatic dimension calculation (**Modelliing > Part properties > Calculate dimensions**) is active, length, height and width are written with the determined values. If the automatic dimension calculation is deactivated, the previous procedure applies. Length, height and width are assigned by the steel engineering functions. Length and width are sorted (**Modelling > Part properties > Sort dimensions**.

Production drawings for mounting assemblies

In the Configuration Editor at **PDM > Drawing Management > Production drawings** the new parameter **Production drawings for mounting assemblies** is available. With this parameter, mounting assemblies (Type: Bolted assembly) can be excluded from the production drawings if required. The default setting is **Yes**.

Itemisation - small text adjustments

In the Configuration Editor, the term "Itemisation mode, Standard" at **System settings > Itemisation** and **System settings > Itemisation > Migration** has been replaced by an explanatory text.

Automatic deactivation of the HCM

The HCM is automatically deactivated if sketches exceed a certain number of lines during the 3-D DXF import or during the conversion of 2-D sketches. You set this number in the Configuration Editor at **System settings > Sketch HCM**. The ISD default setting is 1000. Please note that if you activate the HCM constraints for such sketches, then there may be considerable waiting times..

IFC attribute mapping: Display HELiOS attributes
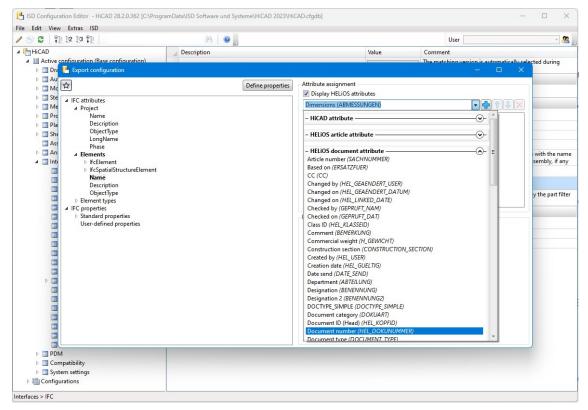
In the attribute mapping configuration for the data transformation between IFC and HiCAD, you can also export HELiOS attributes as of Service Pack 2.

In the Configuration Editor you can find a new checkbox **Display HELiOS attributes** at **Interfaces > IFC > Attribute mapping configuration > ISD defaults >** ⭐ **Define**.

If this is activated, HELiOS attributes are also available for mapping in the pull-down menu of the source attributes.

You can expand and collapse corresponding overviews of the different categories of attribute types.

Transfer of LogiKal attributes to HiCAD

In addition to the item names for profiles and glass panes, the attribute assignment between LogiKal and HiCAD has been extended by some numerical and text attributes.

Transferable numerical attributes are **[Glass] Acoustic Performance**, **[Glass] Light Transmittance**, **[Glass] Solar Factor** and **[Glass] Weight** for glass panes, as well as the **[Insertion] Handle height (from FFL)** for profile inserts.

You can assign these LogiKal attributes to corresponding, user-defined HiCAD attributes in the Configuration Editor at Interfaces > LogiKal.

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Units in the Configuration Editor

In the menu bar of the Configuration Editor, at **Extras > Units**, you can set whether the values specified there for lengths and weights are to be displayed in metric units (millimetres and kilograms) or in imperial units (inches and pounds).

Settings removed

Display of dimensional constraints of sub-sketches

To improve performance, the setting **Show dimensional constraints of sub-sketches** at **System settings > Sketch HCM** has been removed from the Configuration Editor.

OpenGL capacity

Since HiCAD no longer evaluates the setting **OpenGL display list capacity** at **System settings > Visualisation**, it has been removed from the Configuration Editor.

Hatching of adjacent parts in sectional view and cut-out

With the HiCAD function **Hatching settings** you define the delimitation of adjacent areas in sectional views and cut-outs by the hatching distance and the hatching angle. You set the default for new drawings in the Configuration Editor at **Drawing > Views**.

In order to combine the hatching settings in the Configuration Editor, the option **Scale-dependent 3-D hatching** (previously System settings > Visualisation) is now also available at **Drawing > Views**.

Dimension calculation - Sorting the attributes Length, Width, Height

From HiCAD 2023 SP1 it is possible to sort the dimension attributes (Length ($03), Width ($02) and Height ($04)) of the automatic dimension calculation by size, i.e. the largest value is assigned to the length, the smallest to the

height. To do this, the new checkbox **Sort dimensions** must be activated in the Configuration Editor at **Modelling > Part properties**. By default the checkbox is inactive.

Configuration of the graphic display

The display, e.g. when working with sketches, when lengthening sheets or filleting, can be configured. At **System settings > Identification** you can set the Text colour, Text background colour, Display height and Line colour for the preview dimension display.

Itemise parts by part filters

In the **Itemise parts** function, the filters for automatic part annotation can now be used for annotation templates. In connection with this extension, the setting of the filters in the Configuration Editor has been moved. You can find the settings from SP1 at **Drawing > Annotations > Annotation template**.

Automatic conversion of the "old" itemisation with default favourites

The conversion of earlier itemisations (before HiCAD 2018) into the HiCAD standard itemisation can now also be carried out automatically, i.e. without displaying the itemisation dialogue. For this purpose, the checkbox **When switching to the itemisation mode 'Standard', use the stored default settings** is available in the Configuration Editor at **System settings > Itemisation > Migration**.

Check Sheet Metal parts for part orientation

For **Drawing derivation** you can check whether a particular Sheet Metal part has a part orientation. To do this, open the Configuration Editor, go to **Automatic drawing derivation > Production drawing** and activate the checkbox **Check part alignment for Sheet Metal parts**.

Isometry and pipe spool drawing: Switch off symbol alignment

From HiCAD 2023 SP1 it is possible to switch off the automatic alignment of the symbolic representation of parts in the Configuration Editor at **Plant Engineering > Isometry and Pipe Spool Drawing > Align symbolic representations**.

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Free selection of length units in drawings

The previous version HiCAD 2022 only supported drawings with the unit of length millimetre. From HiCAD 2023 onwards, the unit of length can now be selected in the Configuration Editor at **System settings > Units > Unit of length**. This means that it is now also possible to design in non-metric units.

> **Please note the following:**
> The settings selected at **System settings > Units** only apply to newly created drawings. These settings cannot be changed in the drawing itself.

Sketch technology - Distance/angle grid presettings

The value for the angle grid when **drawing planar sketches and 3-D sketches** can now be preset in the Configuration Editor - at **System settings > Identification** in the **Grid** section.

In this context, the preset for the distance grid has also been moved to this area.

Please note that the specified grid settings not only apply to the sketch technology, but also to other functions that use distance and angle grids.

Formulas, mixed numbers and units in point inputs

In the numerical point options, the numerical values required for the point option can be entered directly in one step, e.g. relative coordinates. If you now enter formulas, mixed numbers and units, e.g. (a+b), 1/2, 1 3/4" + 12.5 mm, it is essential to observe the setting in the Configuration Editor at **System settings > Miscellaneous > Pocket calculator**.

- **Space**: With this setting, the various entries must be separated by spaces. If formulas, mixed numbers or mixed units are used, they must not contain spaces. This is the default setting.

- **Semicolon**: If you want to use formulas etc. with spaces, then you must select this setting. The different entries must then be separated by a semicolon **;**.

Drawing derivation

New usage for Sheet Metal assemblies

For Sheet Metal assemblies, i.e. assemblies with a Sheet Metal part as the main part, a new usage is available: Sheet Metal Assembly. The **Factory standards > Usage** catalogue has been extended accordingly. For assemblies with this usage, usage-dependent configurations can be used for the drawing derivation, which are managed in the Configuration Editor as for other usages. By default, the **ASSEMBLY_SHEETMETAL** configuration is predefined for assemblies with the usage **Sheet Metal Assembly**.

Production and mounting drawings - Drawing frames

As of HiCAD 2023, two additional parameters are available for production drawings in the Configuration Editor under **Automatic drawing derivation**> **Production drawing** > **Drawing frames**.

- Use frame

  This checkbox determines whether the frame should be offered for selection in the **Drawing derivation** dialogue or not. The ISD default setting is yes, i.e. the checkbox is active.

- Automatic determination of geometric data for the drawing frame

  If this checkbox is active, the geometric data of the frame including the usable area will be calculated automatically.
  For each drawing frame you can define
  - the distance of the insertion point from the centre of the frame,
  - the size of the usable area (frame height / frame width) and
  - the size of a blocked frame area (for the title block, BOMs or comments)

The following settings have been added to place the drawing frame exactly in the sheet:
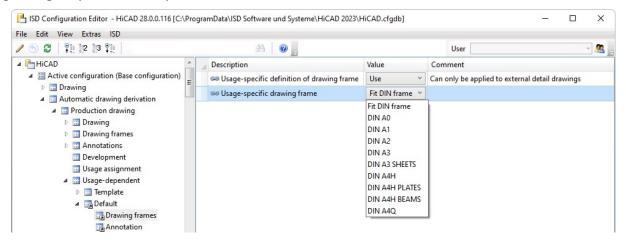
- Horizontal distance of the upper left point to the inner frame

- Vertical distance of the upper left point to the inner frame

- Total height of the figure

- Total width of the figure

These settings help position the inner frame within the outer frame.

Analogously, the parameters listed above are also available for mounting drawings under **Automatic drawing derivation**> **Mounting Drawing** > **Drawing frames**.

Show drawing frame

When selecting the drawing frame of production drawings under **Automatic drawing derivation** > **Production drawing** > **Usage-dependent** > **Template** or **Default**, the DIN name will be displayed in a ComboBox as of HiCAD 2023.

Output of PDF data in black and white

PDF files can now also be created in black and white. For this purpose, the settings in the Configuration Editor at **PDM > Drawing Management > Externalproduction documents** have been extended by the entry **Monochrome PDF data**.

Workflow when creating general documents

As with production drawings, mounting drawings and customer drawings, it is now also possible to select when general documents in DXF, DWG and PDF format are to be created. To do this, the setting in the Configuration Editor at **PDM > Drawing Management > External production documents > PDF files for general documents** has been extended by the options

- Upon Checkup and Release,
- Upon creation and update, and
- Upon Release.

Using Frame/scale lists

The lists for drawing frames and view scale that you create with the new function **Frame/scale list** are only taken into account during drawing derivation if the checkbox **Use Frame/scale list** is active for the respective usage type in the Configuration Editor at **Automatic drawing derivation > Production drawing > Usage-dependent > .... > View group** is active. In the dialogue of the drawing derivation the option **Drawing parameters: From configuration** must then be active.

If a frame/scale list has been created in HiCAD for a purpose, it is displayed in the Configuration Editor at **Frame/s-cale list** ⌐. It is not possible to change the list here.

Updating of part attributes

The settings for updating part attributes have been implemented uniformly for all affected attributes in the Configuration Editor. You can find the settings at **Modelling > Part properties**.

- ■ (1) Already existing settings
- ■ (2) Settings from the directory **System settings > Calculations > Approximation accuracy for weight calculation**
- ■ (3) Settings from **Sheet Metal > Sheet development > Transfer development dimensions to**
- ■ (4) Settings from **Steel Engineering > Weight calculations**
- ■ (5) New parameters and new attributes (§SC, §SOC, §S2D, §CW)

Dimension calculation and alignment

Until now, automatic dimension calculation was only possible for assemblies. As of HiCAD 2023, this is also supported for 3-D parts. For this purpose, the calculation must be activated in the Configuration Editor at **Modelling > Part properties > Calculate dimensions**.

If the dimension is to be displayed in the drawing in HiCAD, then activate this at **System settings > Visualisation > Show dimensions of the active 3-D part**.

If the sheet is to be calculated instead of the sheet development, then the setting **Development attributes** must be selected for **Modelling > Part properties > Transfer development dimensions to**.

Part-related status check

Since the **Part-related status check** located under **System settings > Miscellaneous** in the Configuration Editor is no longer needed, it is no longer available as of HiCAD 2023.

Edge state - Dimension unit

The dimension unit selected in the Configuration Editor at **Drawing > Annotations > Edge state** is used to display the edge state. This can be selected independently of the dimension unit of the drawing.

Selection of bracket type for the 2nd dimensioning figure

The 2nd dimensioning figure was always marked with round brackets in 3-D until now. As of HiCAD 2023, the bracket type can now be selected. The default setting can be defined in the Configuration Editor at **System settings > Annotations > Dimensioning, 3-D**.

2-D DXF/DWG import and export options

The favourite for the default setting for 2-D DXF/DWG exports, e.g. of sheet metal developments, can be set in the Configuration Editor at **Interfaces > Export > 2-D DXF/DWG**. You can create the favourites when exporting from HiCAD by editing the ISD default settings.

User libraries

The functions for part and macro libraries are no longer available in Ribbon **Drawing** for new installations from HiCAD 2023. Existing customers who want to continue using their existing libraries can switch on the functionality in the Configuration Editor at **Compatibility > User library up to HiCAD 2022** by activating the corresponding checkbox.

PDM setting removed
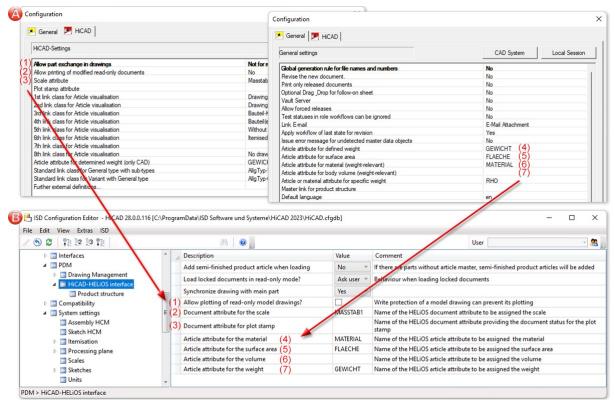
To avoid changes, the settings at **PDM > Drawing Management >**

- **Delete item numbers of project-independent parts** (Default =Yes) and
- **Itemisation of locked parts** (Default = Only project-independent part masters)

have been removed from the Configuration Editor.

HiCAD specific HELiOS settings moved to the Configuration Editor

HiCAD manages some attributes itself, which previously had to be defined in the **HELiOS Options** on the **Database** tab.

A: up to HiCAD / HELiOS 2702, B: as of HiCAD / HELiOS 2800
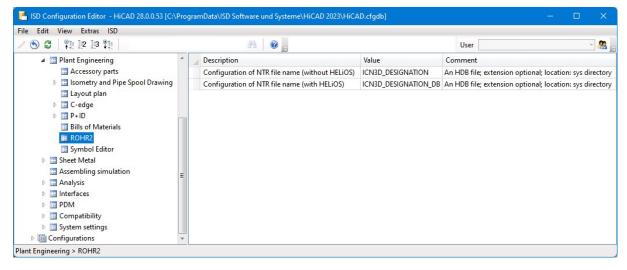
Plant Engineering - ROHR2 interface

In Plant Engineering, the part name of the pipeline was previously used as the file name when generating NTR files

via the ROHR2 interface ![icon]. As of HiCAD 2023, the file name can be configured flexibly via HDB files. For this purpose, the following parameters are available at **Anlagenbau > ROHR2** in the ISD Configuration Editor:

- Configuration of NTR file name (without HELiOS) and
- Configuration of NTR file name (with HELiOS)

The files **ICN3D_DESIGNATION** and **ICN3D_ DESIGNATION_DB** are used as ISD default settings. These are the HDB files which are also used for the display of the column **Designation** of the ICN window **3D Part Structure**. This means that the part number of the pipeline is used as the file name by default or - if this is not available - the part name.

Detailed information can be found in the section Plant Engineering > Evaluation > ROHR2 interface.

LogiKal Interface - Transfer of Text Attributes to HiCAD

When importing LogiKal items (called "Positions" in LogiKal), certain LogiKal text attributes can be specifically assigned to certain HiCAD attributes. The setting is made in the Configuration Editor at **Interfaces > LogiKal**.

Support of IFC 4 Addendum 2

HiCAD 2023 supports the IFC 4 Addendum2 data format for IFC import.

![warning icon] **Important:**

Note that as of HiCAD 2023, when importing IFC files, the IFC version specified in the Configuration Editor under **Interfaces > IFC > Interface version** is no longer evaluated. The version will automatically be detected during import. When exporting IFC files, however, the parameter will still be evaluated.

Bill of Materials / Report Manager

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Edit template for export to Excel

If you want to use a specific template for the export to Excel, activate the option **Use template**. Then determine which template you want to use. To modify the selected template in Excel, select **Edit template** ✎.

Extended Steel Engineering lists

The attribute **Coating** ($15) has been included in the Steel Engineering lists of the Excel BOM (Hicad_ Stahlbau.xlsx). If you have parts coated in HiCAD, the coating is transferred to Excel when exporting the bills of materials.

Dimension attributes Length and Width

In the Excel BOM for Steel Engineering (Hicad_ Stahlbau.xlsx), as of HiCAD 2023 SP2, the column for the Length attribute is before the column for the Width attribute.

Extended bar list

When outputting the bar list in Excel, the used length is now output in addition to the waste. For this purpose, the BOM template for Steel Engineering has been extended by the placeholders

- `!BarWaste!` - Waste

- `!BarUsed!` - Length used

Bar list

Drawing No.	ZN-232423423		Customer	ISD Software und Systeme GmbH	
Order No.	AN-651266723		Created by	Maya Mustermann	HiCAD
Order text			Created on	09.02.2022	
Designation	BN-654235456				

Fl 20x5, S235JR **12 x 6000 mm** **Waste: 9003 mm (12,5 %)** **Length used: 62997 mm**

Item	Number	Length (mm)	Cut (Web)	Cut (Flange)	Naming	Coating
4x	0 (856)	0 (856)	0 (856)	0 (856)	0 (856)	0 (856)	/////819/////
	4	856				
	4	856				
	4	856				
	4	856				
	4	856				
	4	856				
	0 (856)	0 (856)	0 (856)	0 (856)	0 (850)	0 (850)	/////830/////
	1	856				
	1	856				
	1	856				
	1	856				
	1	850	32.9°			
	1	850	32.9°			
6x	0 (849)	0 (849)	0 (849)	0 (849)	0 (849)	0 (849)	0 (849)
	6	849	32.9°			
	6	849	32.9°			
	6	849	32.9°			
	6	849	32.9°			
	6	849	32.9°			
	6	849	32.9°			
	6	849	32.9°			
	0 (849)	0 (847)	/////////////////4276/////////////////			
	1	849	32.9°			
	1	847				

Templates for DSTV and ERPlus

The new ISD Report Manager 2023 is now used for the BOM exports to Qomet and Proflex.

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New ISD Report Manager

As of Version 2023, there is a completely new, comfortable ISD Report Manager. This ISD Report Manager works with RM_SETTINGS instead of RMS files. To convert your old RMS files, the function **Convert RMS file to RM_SETTINGS** is available at **Tools**.

Further advantages:

- By importing / exporting the settings in the dialogues, you can save multiple combinations of settings and conveniently reuse them.

- The headers and footers are now configured using a graphical interface.

- A Sort sequence can be defined for all available columns.

- You can now dynamically search for attributes in the Column settings.

User interface

1. Menu bar

2. The toolbar has been retained and extended to include the settings.

3. Overview of all BOM-relevant parts in the utilized assembly.

4. Overview of all attributes of the part clicked on above.

5. The unit is automatically displayed in the Quantity list and Structure list and is configurable.

6. Status bar.

Export

A graphical interface is now available for configuring header and footer templates.

1. Function bar

2. Page parameters

3. Basic elements

4. Document info data, list of usable attributes

5. Workspace

6. Built-in elements

7. Parameters for the element Text format

8. Parameters for the element Pixel graphics

9. Status bar

The Excel export has also been revised::

- There is now a separate function for the Excel export with template or script. An Excel template or a C# script can be used here.

- There is now the function **Create Excel document (xlsx, xls)**, which transfers the displayed lists directly to Excel without formatting. The data can be formatted with a Visual Basic Script (.vbs).

- In addition, there is a function that writes the displayed cells to an Excel XML file. The data can be formatted with a Visual Basic Script (.vbs).

Column properties

Through the new **Column settings** dialogue, you can now dynamically search for attributes.

1. Dynamic search

2. Column type

3. Visibility of columns and export of columns

4. Decimal places.

5. Identical part search and column sum

6. Category and Displayed unit are for attributes of type Integer or Double

7. The Displayed unit depends on the category. For length, units such as metres, inches, feet, etc. are available. The unit is then displayed in the column in square brackets.

8. Attribute name, Category and Displayed unit are visible when the **Extended display** is active.

9. Displays only the columns marked as visible with the **Visualisation** checkbox activated.

Head data setting

The ISD Report Manager now offers the function **Head data settings**. This allows you to set attributes centrally and comfortably for all possible exports.

Sort sequence

The sort order can now be set for all available columns. In the old Report Manager, only four columns could be set. In addition, for the column type String, one of four sorting methods can be selected for the respective column.

- Standard: Sort alphabetically

- Numerical: Sort by numbers

- Logical: As in Windows File Explorer

- Date/Time: By date and/or time

Filter settings

The data displayed in the list window of the Quantity list and Structure list can be filtered, i.e. only the rows of the list that meet certain criteria - specified in the filter definition - are displayed.

For example, you can

- output only parts with a certain item number or parts whose item number lies within a certain range,

- transfer filters via import/export and save them,

  apply the respective JavaScript methods to the data types,

- create regular expressions and

- get direct feedback on errors in the expression.

Filter criteria are defined column by column. Only the rows in the list that meet all the filter criteria specified for the

columns are then output. Columns for which a filter has been defined are marked with   .

User-definable columns

With the function **User-definable columns** you have the possibility to add additional - freely definable - columns to the BOM or to change already defined columns.

1. Create new columns by type

2. Copy and paste a column. If none is active, all are copied.

3. Filter which column types are to be displayed in the pool

4. Category and Displayed unit can be defined directly

5. Overview of all available columns, filterable by type

When editing a formula, the column name clicked on is entered there as an attribute.

Excel-BOM with pipe length list

For the new ISD Report Manager (from HiCAD 2023), an XML configuration file adapted for Plant Engineering is available with the name **HiCAD_Anlagenbau.rm_settings** . This configuration file allows the sensible creation of bills of materials (BOMs) of mixed model drawings and also contains a pipe lengths list.

The file is located in the HiCAD sys directory and facilitates in particular the use of a new BOM template for Excel export. This new template HiCAD_Anlagenbau.2800.0.xlsx can also be found in the sys directory of your installation (or as **HiCAD_Anlagenbau.EN.2800.0.xlsx** in the sub-folder **bomtemplates**).

Adapted configuration files for the new ISD Report Manager are also available for the Plant Engineering functions at **Plant Engineering > Evaluation**.

HELiOS Options: Switching between previous and new ISD Report Manager

HELiOS 2023 or HiCAD 2023 introduces a new version of the Report Manager.

When creating the BOM in HiCAD and HELiOS, you decide by selecting the configuration whether the previous Report Manager or the new ISD Report Manager opens.

- The old Report Manager uses files with the file extension **RMS** for the configuration.
- The new ISD Report Manager from version 2023 onwards uses XML configuration files with the file name extension **RM_SETTINGS**

You determine the configuration in the HELiOS Options at **Database** and then **Default configuration for product structure output**.

New BOM templates for HELiOS

There are new configuration files for creating bills of materials from the HELiOS product structure. The files (*.RM_SETTINGS) are located in the SYS directory of HiCAD.

- PRODUCT_ISD.RM_SETTINGS
- PRODUCT_ISD_HiCAD.RM_SETTINGS

Variant Editor

Service Pack 1 2023 (V 2801)

Saving variants without hollow body and solid body representation

If you have selected a plant engineering industry (Standard Plant Engineering or Air ducts), then as of SP1 (Patch 1) variants that have neither a hollow nor a solid body representation can no longer be saved. In this case an error message appears:

New part type Welding neck

A new part type called **Welding neck** is available.

You can find extensive information about this in the **Plant Engineering News**.

Major Release 2023 (V 2800)

Function call moved

The Variant Editor has been moved into the **Extras** function group on the **Drawing** Ribbon tab.
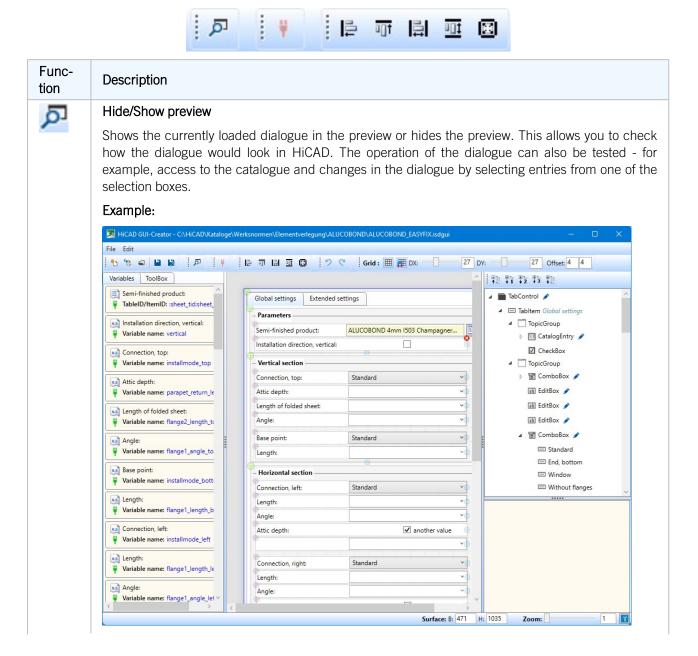
HiCAD GUI Creator

Major Release 2023 (V 2800)

Extended toolbar
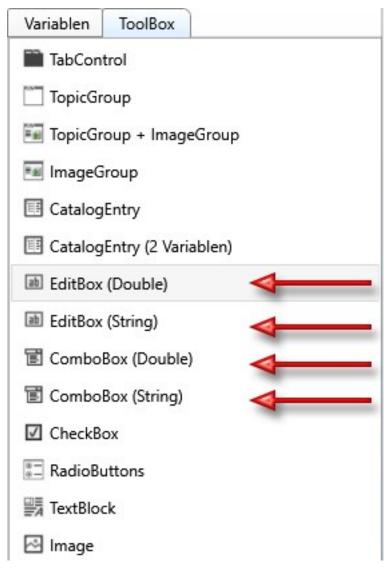
The toolbar has been extended in HiCAD 2023.

Func-tion	Description
	Hide/Show preview Shows the currently loaded dialogue in the preview or hides the preview. This allows you to check how the dialogue would look in HiCAD. The operation of the dialogue can also be tested - for example, access to the catalogue and changes in the dialogue by selecting entries from one of the selection boxes. **Example:**

Func-tion	Description
	Display of the preview

	Align, horizontal left Aligns the selected objects horizontally to the left. The last selected object is decisive for the alignment.
	Align, vertical top Aligns the selected objects vertically on top. The alignment is determined by the last selected object.
	Set equal width Sets the width of the selected objects equal. The width is determined by the last selected object.
	Set equal height Sets the height of the selected objects. The height is determined by the last selected object.
	Set equal width and height Sets the width and height of the selected objects equal. The width/height is determined by the last selected object.
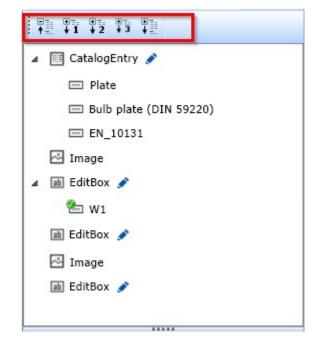
New ToolBox objects

The ToolBox objects **EditBox** and **ComboBox** now distinguish between input for numbers (Double) and text (String).

Expand/Collapse dialogue structure

The dialogue structure can now be expanded and collapsed, like the part and view structure in HiCAD.

Activity condition

In addition to the Visibility condition, an Activity condition can now be assigned to all toolbox objects. This determines when an object is active, e.g. when input should be possible in an input field.

Automation

Discontinuation

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

Service Pack 2 2023 (V 2802)

Link development

The HiCAD API provides the following command for deleting links between 2-D and 3-D developments in Sheet Metal:

- ISD.CAD.SheetMetal.SheetDevelopment.RemoveDevelopmentReference(Figure developmentFigure)

Service Pack 1 2023 (V 2801)

API

Divide along direction and Divide with sketch

The following functions are now available in the HiCAD API:

- 3-D-Standard > Process > Trim ▾ > Divide along direction (DividePartAlongDirection)

- 3-D-Standard > Process with sketch > Subt. ▾ > Divide with sketch (DividePartBySketch)

Laminated glass in the API

The following command is available in the HiCAD API for laminated glass (Steel Engineering > Plate, new > Laminated glass and Metal Engineering > New > Laminated glass:

- LaminatedGlassCreator

Determine base part of a dependent part

Using the HiCAD API (Node.GetDependentNodeSource und Node.ExchangeDependentNodeSource), you can now determine the base part for a dependent part. This is useful if the dependent and the base part are derived and the derivations are related to each other.

Set fixed view point

The following API functions are available for the new function **Set fixed view point** (Views > Transform > ReqPos ▾ >...), with which you fix the active view at a point associative to the geometry:

- View.SetFixPoint (Set fixed view point)
- View.ClearFixPoint (Delete fixed view point)

Interfaces

Service Pack 2 2023(V 2802)

Update to CADfix 12 SP2.1

With the update to CADfix 12 SP2.11 the following format versions are now available in HiCAD 2023:

- ACIS 2021 1.0.1 (V5R32)
- CATIA V5 V5-6 R2022
- DWG 2023
- DXF 2023
- Inventor 2023
- JT 11.11
- PLM XML 11.11
- CREO 9
- SOLIDWORKS 2022

2-D DXF/DWG Export: New settings dialogues

The dialogue windows of the Export options for 2-D DXF/DWG files have been extended with Service Pack 2 by several new features.

For example, the transfer of parts as blocks can now be controlled via the Export settings dialogue window. Manual configuration of the system file keyword "FIGBL" is no longer necessary.

New setting options for the transfer of layers, colours and line types into the DXF/DWG format can be found in the new menu window tabs **DXF layer assignment**, **DXF layer definition** and **Line type assignment**.

Other old configuration file keywords that have been dropped as a result are: BYLAY, LAYER, LTDEF, SONST, TXTLA.

DXF layer name	DXF colour index	Line width	DXF line type
LE_GREEN	3	Default	CONTINUOUS
LE_RED	1	Default	CONTINUOUS
LE_ALL	7	0.50	CONTINUOUS
LE_HIDDEN1	3	Default	HIDDEN
LE_HIDDEN	7	Default	CONTINUOUS
LE_MIDDLE	7	Default	CONTINUOUS
TXT_YELLOW	2	Default	CONTINUOUS
TXT_DIM49	2	Default	CONTINUOUS
TXT_ALL	7	Default	CONTINUOUS
DIM49	4	Default	CONTINUOUS
HATCH	3	Default	CONTINUOUS

Automatically generated layers ⓘ

*	7	Default	CONTINUOUS

HiCAD line type	Name of the DXF line type	Description
1:	CONTINUOUS	Solid line
2:	DOT	. . . .
3:	HIDDEN	-- -- -- -- --
4:	DASHDOT	--.--.--.--.--
5:	STRICHPUNKT2	--.--.--.--.--
6:	STRICHPUNKT3	--.--.--.--.--
7:	PHANTOM	--..--..--..--
8:	PHANTOM2	--..--..--..--
9:	PHANTOM3	--..--..--..--
10:	STR3PKT	--...--...--...
11:	2STRPKT	--.--.--.--.--
12:	GESTRICHELT2	-- -- -- -- --
13:	2STR2PKT	--..--..--..--
14:	STR3PKT2	--...--...--...
15:	2STR3PKT	--...--...--...
16:	STRKSTR	-- - -- - -- -
17:	STRKSTR2	-- - -- - -- -
18:	STRKSTR3	-- - -- - -- -
19:	PUNKT2	. . . .
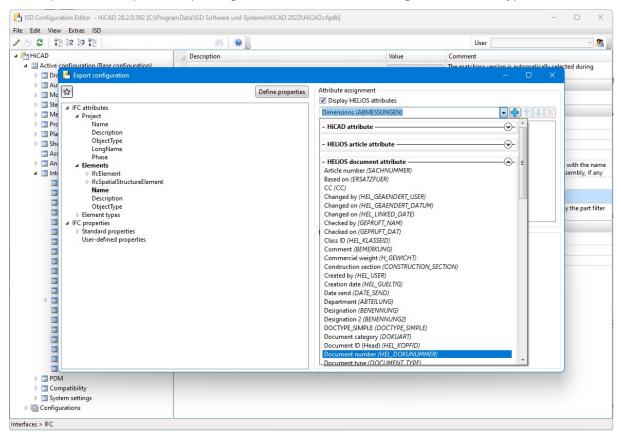
Export settings 2-D DXF/DWG

General | DXF layer assignment | DXF layer definition | Line type assignment | Colour assignment | Compatibility

OK   Cancel

IFC attribute mapping: Display HELiOS attributes

In the attribute mapping configuration for the data transformation between IFC and HiCAD, you can also export HELiOS attributes as of Service Pack 2.

In the Configuration Editor you can find a new checkbox **Display HELiOS attributes** at **Interfaces > IFC > Attribute mapping configuration > ISD defaults >** ⭐ **Define**.

If this is activated, HELiOS attributes are also available for mapping in the pull-down menu of the source attributes.

You can expand and collapse corresponding overviews of the different categories of attribute types.

The target attribute determines which HELiOS attributes are offered to you: For a mapping with the IFC attribute **Elements > Name**, for example, the attributes of the types **HELiOS article attribute** and **HELiOS document attribute** are available; if the selected IFC target attribute is, for example, **Project > Name**, then **Project attributes** and **Folder attributes** are displayed for possible mapping.

HELiOS attributes of all four object types are available for other **IFC properties**.

Service Pack 1 2023 (V 2801)

2-D DXF/ DWG: New settings dialogues for import and export

The import and export options for 2-D DXF/DWG files have been extended by dialogue windows with the help of which you can conveniently set, for example, the processing of colours and layers, the conversion of units of measurement or the target version of the DXF format.

Please note the following in this context:

In earlier HiCAD versions the header variable $INSUNITS of DXF files was not evaluated and thus the imported data was not scaled. Using the import setting option for the **Unit of length**, the variable is now evaluated, converted into the current HiCAD drawing unit and scaled accordingly if necessary.

Furthermore, by providing the new dialogue options, some codes that were manually edited in the corresponding configuration files in earlier HiCAD versions are omitted:

- When importing, the following codes are omitted: INSUNI, BEMAS, LAYFI, LAYNR, LAYNA, COLBE, COLSR, COLOUR, BLKBE, BLKSR, SPFIG, PROXY, TSCHO, TLEER.
- When exporting, the following codes are omitted: ACDXF, INSUNI, BEMAS, NKST, COLOR.

DSTV-NC: Threads and countersinks

The **Others** area of the DSTV-NC format export dialogue has been extended by the options to suppress **Threads** and **Countersinks** or to write only a core drilling instead.

By default, threads and countersinks are exported (setting selection is set to **Write** in each case).

Major Release 2023 (V 2800)
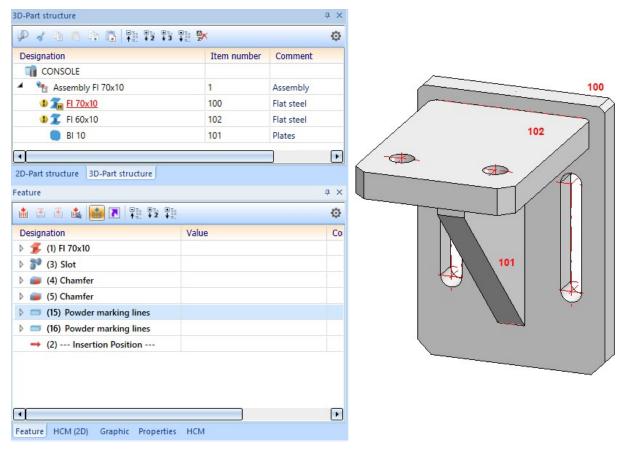
DSTV-NC - Combining powder marking lines

For the DSTV NC export, it is now possible to specify on the **Extended** tab whether powder marking lines are to be listed individually in the NC file or whether the powder marking lines are to be combined into one block per marking line causing part.

Example:

In the image below, Parts 101 and 102 cause powder marking lines on Part 100.

Excerpt from the DSTV NC file of Part 100.

IFC import

Support of IFC 4 Addendum 2

HiCAD 2023 supports the IFC 4 Addendum2 data format for IFC import.

> Note that as of HiCAD 2023, when importing IFC files, the IFC version specified in Configuration Editor at **Interfaces > IFC** is no longer evaluated. The version is automatically detected during import. However, when exporting IFC files, the parameter will still be evaluated.

Set BOM-relevance

For **IFC import**, an additional option is now available for determining the BOM-relevance (HiCAD attribute **#SR**). For this purpose, the previous checkbox in the dialogue window has been replaced by a selection box.

The following options are available for selection:

Delete	The BOM-relevance of the imported parts is set to **No**, i.e. the imported parts are not BOM-relevant.
Set	The BOM-relevance is assigned to the imported parts.
From file	The BOM-relevance is determined by the imported file or by the attribute mapping defined in the Configuration Editor for the import.

Navisworks export

When inserting a NWC file created by HiCAD into the Navisworks Manager, attributes exported by HiCAD are also displayed there. Which HiCAD attributes are transferred and how can be configured via a system file.

The group name of the HiCAD attributes displayed in Navisworks can also be changed if required.

NCX-Export - Rotate beams and profiles

HiCAD offers the possibility to rotate beams and profiles during NCX export - from HiCAD 2023 onwards also by 90° and 270°.

To use this option, the files

- BRW_3DTEIL_PROFIL_H.HDX und
- BRW_3DTEIL_PROFIL.HDX

in the HiC sys directory must be manually supplemented by the following line:
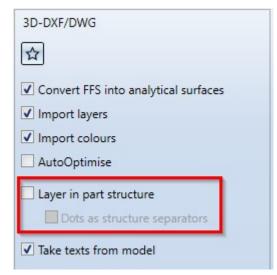
`<H>::TEXT=""::ATTR="%NCROT"::TYP="COMBOBOX"::EDIT="NO":::E_WIDTH="401"::E_LIMIT-T="130"::FILE="..\sys\H_NCRotation.txt"::TL_WIDTH="130"`

This extends the **Part attributes** dialogue for beams and profiles by a selection box for the setting **NCX beam rotation**.

3D DXF/DWG-Import Transfer of layers to 3-D data structure

Until now, when importing 3-D DXF/DWG files created with flexijet, the part structure (layer) was transferred in the form of layers. This meant that the complete flexijet model was transferred to HiCAD as one part. From HiCAD 2023 it is possible to transfer these layers also as 3-D parts into the HiCAD part structure. For this purpose, the dialogue window has been extended by corresponding options.

Layer in part structure

If this checkbox is activated, an attempt is made to transfer the layers from the DXF file into the HiCADAD part structure. For this purpose, the points, lines and parts, which are assigned to certain layers, are collected in parts, which are named according to these layers. Activating the checkbox can be useful, for example, when importing files created with flexijet.

Dots as structure separators

This option is only available if the checkbox **Layer in part structure** is activated. If the checkbox **Dots as structure separators** is active, it will be checked whether a highly detailed part structure is given by the layer names, namely by dots as separators. Components of a layer name preceding a dot are then created in HiCAD as an assembly, the associated remainders of the name then result in the parts lying under the assembly.

2-D DXF/DWG import and-export: New dialogues

In order to be able to edit the import and export settings of 2-D DXF/DWG files more comfortably, new configuration dialogues have been integrated into the HiCAD interface.

These replace the manual editing of system files (acadhcad.dat, hcadacad.dat), which are thus no longer necessary.

However, settings from older configuration files can still be read in and taken into account accordingly.

Sheet Metal

Service Pack 2 2023 (V 2802)

Coat parts

Using the optimised **Coating** function in Ribbon **Sheet Metal** and **Steel Engineering**, you define the coating description, the coating colour (RAL, NCS and system construction colour) and the coating type (outside, back-/inside and face) of sheets and plates as well as beams and profiles and 3-D parts. The **Extend coating** function, with which you transfer the coating to the new or processed elements, can now also be applied to profiles.

The coating description and the coating type are transferred to the part attributes and can thus also be made available to the bill of materials.

After selecting the function, a preview of the coating is displayed for the active part (in shaded mode). You can now select other parts and apply the coating with **OK**.

An arrow is visible in the middle of sheets and hollow profiles after the selection. Use this to exchange the outside with the back of sheets or the inside with hollow profiles. For non-hollow profiles, the specification for the inside has no meaning.
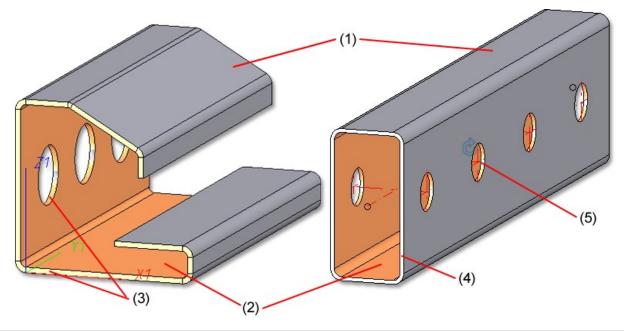
If you have already assigned an outer side to the sheet or plate with the function **Determine visible side**, this will be taken over as the outer side when coating.

You can select a RAL or NCS colour from the catalogue or a system colour or drawing colour as the coating colour. If you have selected a catalogue colour, you can also adopt the description from the catalogue. The description is transferred to the part attributes.

Surfaces to be coated

For sheets, you can coat the end faces and the processings either in the colour of the outer side or the back side. For profiles, you can set the coating of front surfaces and processings separately. These settings are transferred to the part attributes.

If **Back side deviating** is deactivated, the second colour selection is greyed out and both sides will get the same colour.

(1) Coating (outer side), (2) Back side/Inner side deviating, (3) Coat front surfaces and processings like back side of sheet, (4) Do not coat front surfaces for hollow profiles, (5) Coat processings like inner side (for hollow profiles)

Attach flanges to surface

The **Attach flanges to surface** function has a new option for corner processing.
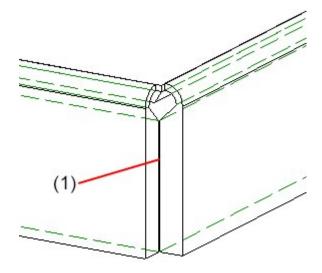
This option allows you to close the corners of sheets with milling edge zones , taking into account the **Joint**, i.e. **Inner edge, flush** or **Sheet edge as milling edge**.

Joint: **Inner edge, flush**
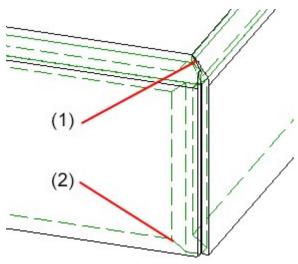
With the joint you determine whether the inner flange edges are shortened or lengthened flush.

(1) Clearance

Joint: **Sheet edge as milling edge**

For composite panels you can finish the flanges with a milling edge zone. For the milling edges, select the appropriate milling tool from the **Groove form** option in the **Factory standards > Composite panels, groove form catalogue**.

(1) Milling edge closed
(2) Groove form V 90° for the flanges from the catalogue

Copy flange or bend zone

If you copy the flange or bend zone of a Sheet Metal part using the **Copy to HiCAD Clipboard** function (in the ICN toolbar or context menu), the entire Sheet Metal part is always copied if the feature is active.

You can then paste the Sheet Metal part as a new main or sub part into the current drawing or into a new drawing using the **Paste from HiCAD Clipboard** function.

Clone/Transform multiple selections

If you select flanges and bend zones in multiple selections, as of HiCAD 2023 SP2 the complete Sheet Metal part is always taken for transforming and/or cloning. Previously, an error message appeared and the function was cancelled.

Please note:

For flanges and bend zones without feature, **Transform** and **Clone** operations applied to individual parts are still allowed.

New setting options for DXF/DWG export

The dialogue for setting the DXF/DWG export options has been further modernised.

- On the **General** tab the option **Create blocks** has been added. This replaces the previous keyword FIGBL.

- Use the table on the **DXF layer assignment** tab to assign HiCAD elements to corresponding DXF layers. This table replaces the previous keywords LAYER, TXTLA and SONST.

- In addition, LAYER * and SONST * have been replaced by the checkbox below: Here you can choose for wHiCADh elements DXF layers should be generated (and assigned accordingly) according to the HiCAD layer of the elements.

- The properties of DXF layers can be configured on the **DXF layer definition** tab. This replaces the previous keyword BYLAY.

- On the **Line type assignment** tab, the HiCAD line types can be assigned DXF line types with name and description. This replaces the previous keyword LTDEF.

For the two dimensioning rules, an isolated point was added to the agraffe in the bore pattern. This point serves as a dimensioning point and has the designation **Dim**.

Service Pack 1 2023 (V 2801)

Sheets form bend zones

In HiCAD you can now create and edit Sheet Metal parts consisting of bend zones only. For creation, the **Sheet along sketch** function is available or you import a bend zone via the STEP interface. You can then perform a bending simulation, development or a sheet thickness calculation, for example.

If you have created a Sheet Metal part with the **Sheet along sketch** function from bend zones, then you can edit the sketch in the feature or change the bend zones via the dialogue. If the sketch contains arcs that do not merge tangentially, a bend zone will beinserted.
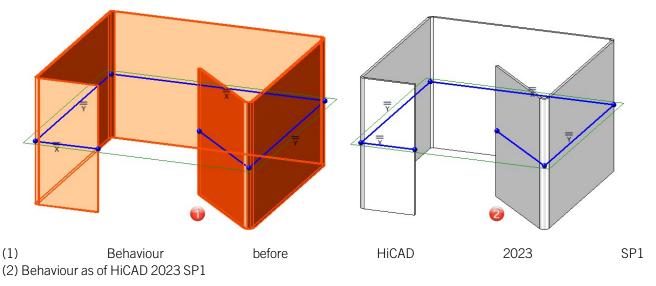
(1) Sketch, (2) Arcs without tangential transition, (3) Bend zones inserted

Sketch feature - Element snap mode

In HiCAD Sheet Metal, sketches are often used when creating and processing sheets. You can process these sketches via the feature log. The user guidance for the **Process sketch** feature has been changed.
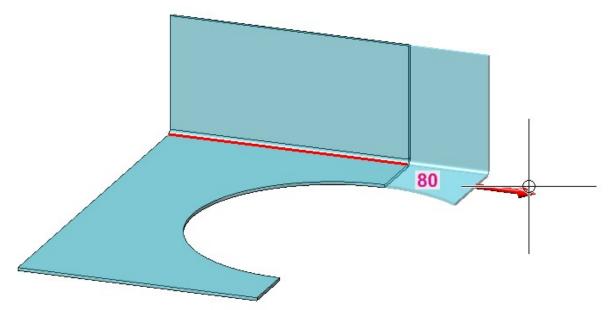
Previously, in **Element snap** mode, holding down the SHIFT key while right-clicking on a sketch element highlighted the sketch and geometry in colour. This made the selection of sketch elements difficult in many cases and possibly led to errors in HCM constraints.

As of HiCAD 2023 SP1, a sketch element is selected in the drawing in **Element snap** mode via SHIFT + right click and then the **Process sketch** function is called in the context menu for edges. Afterwards, only the sketch is marked in colour and the feature is calculated up to the initial position.
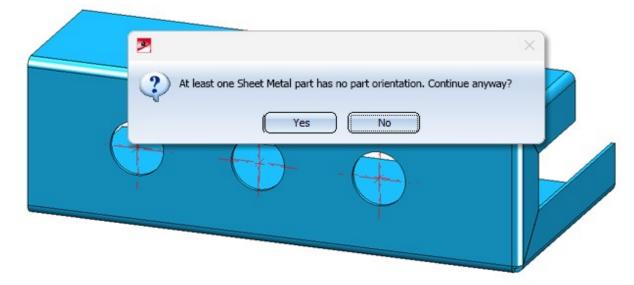
(1) Behaviour before HiCAD 2023 SP1
(2) Behaviour as of HiCAD 2023 SP1

Configuration of the graphic display

The display, e.g. when working with sketches, when lengthening sheets or filleting, can be configured. At **System settings > Identification** you can set the Text colour, Text background colour, Display height and Line colour for the preview dimension display.

Check part orientation of Sheet Metal parts

For **Drawing derivation** you can check whether a particular Sheet Metal part has a part orientation. To do this, open the Configuration Editor, go to **Automatic drawing derivation > Production drawing** and activate the checkbox **Check part alignment for Sheet Metal parts**.

If you then have not defined a part orientation, a message will be displayed during drawing derivation.

If you choose **Yes**, the drawings will be created. If you choose **No**, you cancel the drawing derivation.

With the Part orientation function you determine the front view or top view of the sheet. You define the part orientation with the functions Orient... (Drawing > Positioning/Detailing) or via the context menu of the Sheet Metal part (Properties).

 Please note:
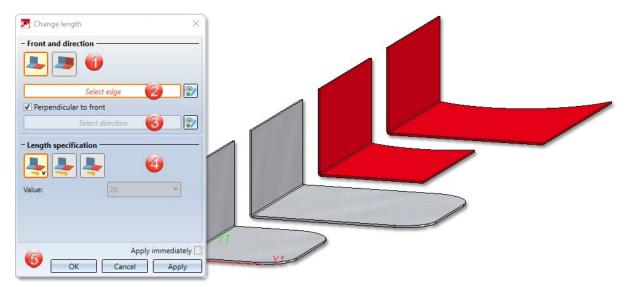
When you create a sheet using the Sheet along sketch function, both the part orientation and the dimension orientation are automatically set to the front view of the sheet.

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Change length

On the **Sheet Metal** tab, the lengthening of sheets in the same plane is summarised in the **Change length** dialogue. With the dialogue you can lengthen flanges and bend zones as you like or shorten them by a negative value or by selecting the direction. You can also change the length in the bending simulation.

Select via the icon whether you want to lengthen a flange or a bend zone, a milling edge zone or the entire sheet with bend zones and flanges.

	Only selected surface	If you have activated this icon, you can change the length of a flange or a bend zone or a milling edge zone.
	All surfaces in same plane	Bend zones and flanges that are in the same plane are lengthened in one step.
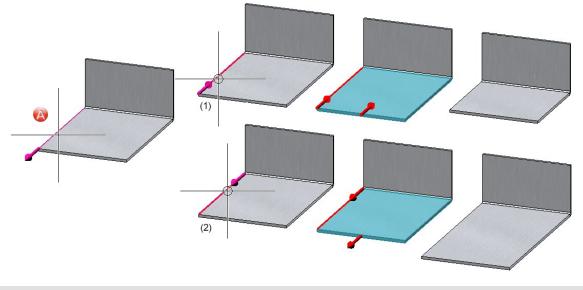
By identifying the edge, a top surface, a front surface and the front edge in between are now clearly determined. If you activate the option **Perpendicular to front**, the direction is automatically determined and displayed on the flange.
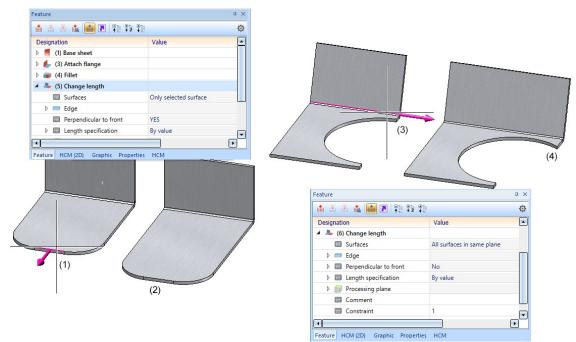
(A and B) The identification point of the edge determines the front side.

(1), (2) and (3), (4) When selecting the direction, an arrow is displayed for clarification.

You have three options for length specification:

- **By value**: This option lengthens a flange/bend zone or a complete sheet by the entered value, depending on the previous selection.

- **Total length**: Lengthens a flange/bend zone or sheet by entering the new total length. The total length refers either to the flange/bend zone or to the complete sheet.

- **To point**: Lengthens a flange/bend zone or a sheet in the selected direction over a point and a distance to the point.

(1) and (2) Lengthening of the surface by selecting the front edge (right-click))

(3) and (4) Lengthening of the sheet perpendicular to the front by selecting the longitudinal edge.

Develop surfaces (analytical)

The completely revised **Develop surfaces** function analytically develops the surface of a 3-D part (e.g. truncated cone or free-form surface). Either a sheet thickness (offset) is queried for the depth of cut or, for example, the thickness is adopted for hollow bodies.

You can derive several 3D developments with different parameter settings from one 3-D part. The parameter settings are saved with the development. A new view is created for each development. The view and the 3-D part are linked. If the sheet metal part is deleted, the view with the development also disappears.
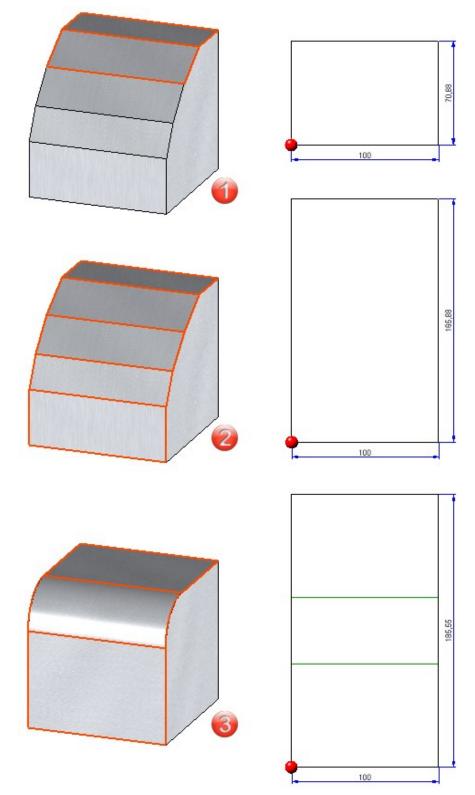
In the ICN, the development appears below the 3D part and can contain sketches as a secondary part for later processing with 3-D functions. If the development is active, the 3-D part is greyed out.

You have the option of defining the development parameters when creating the development. You can access the functions for changing the parameters by right-clicking on the development.

After activating the function, the following dialogue appears.

The 3 options **Individual**, **Tangential** and **Tangential with angle** are available for the selection of the surfaces to be developed.
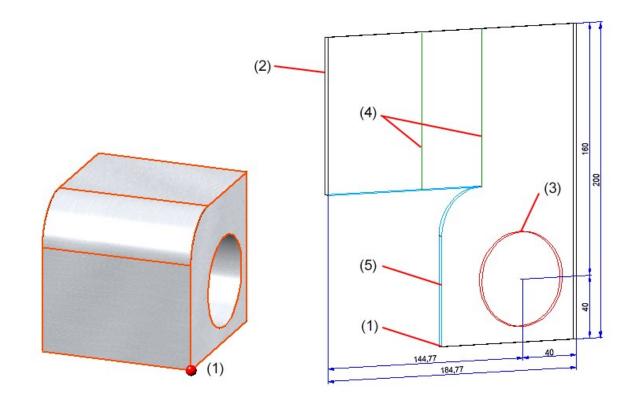
(1) **Individual**: Here you select the surfaces to be developed individually.

(2) **Tangential with angle**: Adjacent surfaces and tangential surfaces are automatically recognised taking into account an angle.

(3) **Tangential**: Tangentially adjacent surfaces are recognized automatically.

With the help of the Annotation Editor, you can define additional texts, e.g. for production, or load the annotation via the Favourites. The attributes that can be evaluated are displayed in the Annotation Editor. These can be supplemented with your own notes.

In the Extended settings, the line colour, line type and the layer of the various edges are set. For example, you can set the colour, line type and layer in the development for crosshairs.

(1) Tear-open point, (2) Sheet development edges, (3) Inner contours, (4) Bend zome, (5) Tear-open edges

Connecting sheet

With the modernised **Connecting sheet** function you can connect two composite edges (guidelines), from a 3-D sketch or a planar sketch, with a sheet. Depending on the selected function, the sheet is created as a main or sub-part. As soon as your entries are sufficient, a preview is displayed. The feature log is available for changes.

Connecting sheets can be derived from planar sketches and 3-D sketches. In the 3-D sketches, both composite edges can be in one 3-D sketch if the icon is active. If the composite edges are in two 3-D sketches or if you are working with plane sketches, activate the following icon . Make sure that there is only one composite edge in the sketch.

If you choose a semi-finished product from the Catalogue Editor , the sheet thickness is not requested. The sheet thickness and, if applicable, the **Article number** are then taken over from the Catalogue Editor for the sheet. In the **General** area you can also change the **Article number** for the selected semi-finished product. If you also want to take over the bend radius from the Catalogue Editor, activate the checkbox next to the **Bend radius** input field. Proceed in the same way with the **Allowance method**.

Under **Offset** you select to which side the sheet thickness (material thickness) is to be removed.

The following options are available for the processing of bend zones:

	No special pro-cessing	Bend zones that meet in a corner of the sheet are shortened to avoid overlapping.
	Flush	If several bend zones converge in one corner, the sheet is widened in the corner to make room for the bend zones. The front surfaces of the bend zones are flat surfaces. They are not adjusted to the adjacent flanges..
	Flush and corrected	If several bend zones converge in a corner, the sheet is widened in the corner. The front surfaces of the bend zones are adjusted to the adjacent flanges.

The arcs of the guidelines are approximated synchronously by polygon lines so that two lines lie in one plane and thus define a flange. You define the accuracy of the resulting sheet metal flanges in the **Accuracy criterion** area.

Inputs that cannot be evaluated are marked with the ✖ symbol.

Once you have entered all the required data, the connecting sheet can be inserted as shown in the preview. If you select **Apply** or press the middle mouse button, the flange is inserted, but the dialogue window remains open - in contrast to **OK**. This way you can change the data and assign it to other sketches with **Apply**.

(1) First and second guideline in a 3-D sketch

(2) Connecting sheet

(3) Accuracy criterion: Angle 5

New attributes for calculating the surface area of sheet metal parts

In HiCAD, new system attributes are available for sheet metal parts and coated surfaces:

§SOC Calculate surface area from the development contour, internal exclusions and processings are ignored

§S2D Calculate rectangular surface area of the development from length * width

§CW Calculate weight from the rectangular surface area of the development

§SC Calculate the coated surface area for coated sheet metal parts

In the Configuration Editor, at **Modelling > Part properties**, you set the system attributes for when the calculation should take place, e.g. for the creation of bills of materials.

The following three procedures are available for selection:

- Do not auto-calculate

- Only for automatic itemization

- Always

2-D DXF/DWG export with new settings

In order to be able to edit the settings more comfortably when exporting developments as 2-D DXF/DWG files, new functions have been integrated.

These replace the manual editing of system files (acadhcad.dat, hcadacad.dat), which are thus no longer required in this form.

However, settings from older configuration files can still be read in and taken into account accordingly.

You can load the ISD defaults with this icon , customise them with the **Edit settings** function and then save them as Favourites under a new name. .

Planar symbol for all processings in the development

For developments, symbols can be output for processings (bore patterns, punching tools, ...). To do this, the symbol must be entered in the **TOPSYMBOL** and **BOTTOMSYMBOL** columns in the Catalogue Editor.
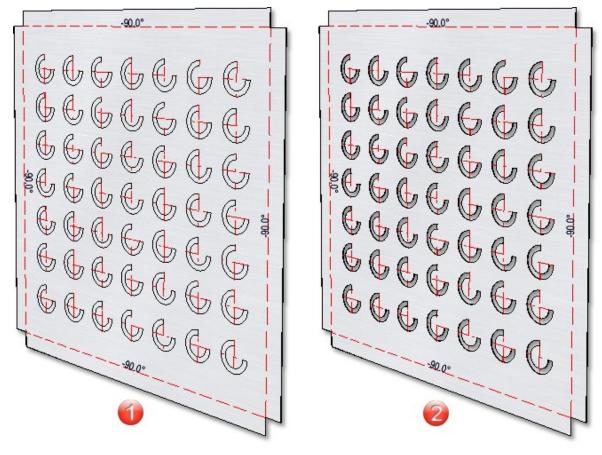
	ID	MOD	STATUS	BZ	NAME	TYP	ICON	R1	R2	TOPSYMBOL	BOTTOMSYMBOL
1	1		▶	R1-10, R2-5	MUSTERBOHRUNGEN\001_0001.DCF	1	001_0001.jpg	10	5	MUSTERBOHRUNGEN\001_0001.DCF	MUSTERBOHRUNGEN\001_0001.DCF
2	2		▶	R1-20, R2-10	MUSTERBOHRUNGEN\001_0001.DCF	0	001_0001.jpg	20	10	MUSTERBOHRUNGEN\001_0001.DCF	MUSTERBOHRUNGEN\001_0001.DCF
3	3		▶	R1-30, R2-25	MUSTERBOHRUNGEN\001_0001.DCF	0	001_0001.jpg	30	25	MUSTERBOHRUNGEN\001_0001.DCF	MUSTERBOHRUNGEN\001_0001.DCF
4	4		▶	R1-40, R2-30	MUSTERBOHRUNGEN\001_0001.DCF	0	001_0001.jpg	40	30	MUSTERBOHRUNGEN\001_0001.DCF	MUSTERBOHRUNGEN\001_0001.DCF
5	5		▶	R1-45, R2-30	MUSTERBOHRUNGEN\001_0001.DCF	0	001_0001.jpg	45	30	MUSTERBOHRUNGEN\001_0001.DCF	MUSTERBOHRUNGEN\001_0001.DCF

You can also extend tables (e.g. countersinks) with these columns yourself and thus output symbols during development. To do this, add the **TOPSYMBOL** and **BOTTOMSYMBOL** columns to the table in theCatalogue Editor and then enter a relative path to the symbol file. This can be 2-D/3-D parts (FGA, KRA) or HCM models (DCF).

In the development, select the option **Planar symbol from catalogue** on the **Extended representation** tab in the **Extended settings** dialogue.

🛈 **Please note:**

To read out the symbol in the development, processing must be done in the sheet metal flange..

(1) The planar symbol from the catalog is displayed.

(2) Example of a bore pattern displayed in the development
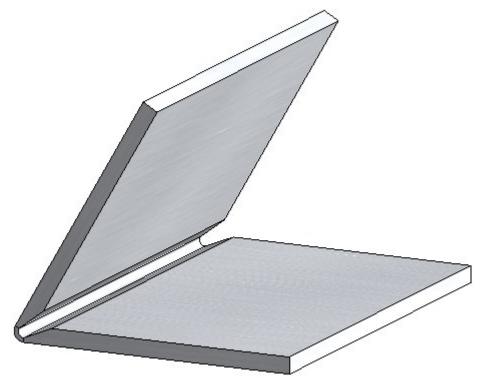
Part orientation in workshop drawings for New sheet along sketch function

Up to now, with the **New sheet along sketch** function, the part orientation of the front view was placed on the cross-section surface. However, this is not useful for the workshop drawing, as the front view is evaluated for the drawings. Therefore, the front view is now placed on the sheet so that the drawings can be evaluated better.

When creating a sheet along a sketch, both the part orientation and the dimension direction are set.

New groove form for milling edge zones

In the catalogue **Factory standards > Composite panels, groove form**, the table **Standard groove form** has been extended by an entry for easy fiX side milling cutters **V 135°, 3 mm**. This groove form can be selected when attaching flanges.

Drawing derivation: New Usage for Sheet Metal assemblies

For Sheet Metal assemblies, i.e. assemblies with a Sheet Metal part as the main part, a new usage is available: Sheet Metal Assembly. The catalogue **Factory standards > Usage** has been extended accordingly.

For assemblies with this usage, usage-dependent configurations can be used for the drawing derivation, which are managed in the Configuration Editor as with other usages. For assemblies with the usage **Sheet Metal Assembly** the configuration **ASSEMBLY_SHEETMETAL** is predefined.

Steel Engineering

Service Pack 2 2023 (V 2802)

Insert new beam

The function for inserting beams and profiles has been redesigned and significantly expanded with SP2. With the new **Insert new beam** ![beam icon] function, not only 3-D standard beams and profiles but also multi-part standard beams, prototype beams and elongated plates can now be inserted. The selection is made in the following catalogues:

- **Semi-finished products > Plates**,
- **Semi-finished products > Cold rolled sections**,
- **Semi-finished products > Beams + Profiles**,
- **Factory standards > Multi-part standard beams** and
- **Factory standards > Prototype beams**.

Once you have now entered the required parameters, a preview of the beam is displayed. Another new feature is that a fitting point can now be selected to simplify insertion.

The previous functions for beam insertion (before HiCAD 2023), which are now integrated into the new functionality, are still available in the menu under **Up to HiCAD 2022**.

Coat beams and profiles

In addition to sheets and plates, as well as general 3-D parts, beams and profiles can now be processed with the

Coating [icon] function.

You can find the function at **Steel Engineering > Further functions > Coating**.
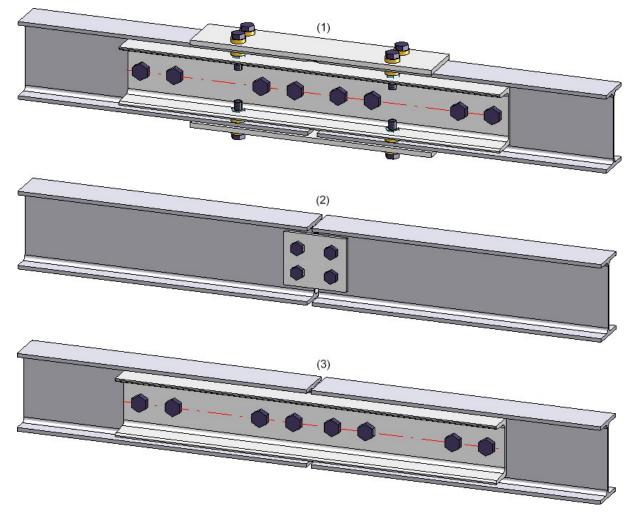
Purlin joint according to DAST PM/PQ (2415)

The previous design variants for purlin joints,

- Purlin joint acc. to DAST PM (405) and

- Purlin joint acc. to DAST PQ (415)

have been replaced with SP2 by the new Design Variant **Purlin joint acc. to DAST PM/PQ (2415)**. This variant connects two beams by beams of a rigid joint. Allowed are I- and U-beams, which have to be aligned accordingly. The insertion can be carried out can be user-defined or - with corresponding beams - based on a table according to DAST PM or PQ.

In the user-defined configuration, the joint can be created with one strap each on the upper and lower flange as well as on the web, whereby all straps as well as the corresponding boltings and hole patterns can be set separately. U-beams or flat steels are possible as straps. The connection can be created flush or with clearance.

(1) User-defined, (2) DAST PQ, (3) DAST PM

Purlin joint (2415)

DAST

1st beam

U 400

2nd beam

U 400

Beams+Profiles | Straps | Boltings | Geometry | Bore grid

Preview | OK | Cancel

Automatic dimension calculation for beams and plates

Beams

If the automatic dimension calculation is active in configuration management, the calculation for beams can now be restricted so that only the length is determined by the automatic dimension calculation and any values for width and height transferred from the catalogue are retained.

Plates

If the automatic dimension calculation is activated in the Configuration Editor, length, height and width are calculated and assigned to the attributes **Length** ($03), **Width** ($02) and **Height** ($04). Length and width are sorted. That is, the largest value is assigned to the length..

Steel Engineering plates - Development length and Development width

For Steel Engineering plates, the Length, Width and Height are now additionally assigned to the attributes

- **Development length** (§L2D),
- **Development width** (§B2D) and
- **Sheet thickness** (§T2D)

In this way, uniform bills of materials can be created for Sheet Metal parts and Steel Engineering plates..

Unlike Sheet Metal parts, the values are calculated automatically, i.e. the settings for Sheet Metal parts in the Configuration Editor at **Modelling > Part properties** do not apply to Steel Engineering plates!

For drawings saved before HiCAD 2023 SP2 (V 2802), a feature recalculation must be performed in order for the values to be calculated.

Laminated glass

Laminated glass - Definition of the article number

For the individual layer structure of laminated glass, various options are now available for defining the article number:

- **Manual**
  The article number can be assigned as desired.

- **Automatic**
  The article number is automatically generated based on the layer structure by combining the designations of the individual layers separated by a separator. The default separator is "/". However, this can be changed in the Configuration Editor at **Steel Engineering > Products > Laminated glass pane > Separator for auto-generated article number of a laminated glass pane**.
  Example: ESG 8/RV ISD-08/ESG 6

- **Automatic, with total thickness**
  Here, too, the article number is generated according to the layer structure as with the option **Automatic**, but here the total thickness is also output.
  Example: VSG 16,2 mm (ESG 8/Folie 0.2/TVG 8)

Additional layer thicknesses

From SP2 onwards, the **Laminated glass** function makes additional film thicknesses available when installing individual glass panes with foil layers - 0.38, 0.76 and 1.52, each in matt and clear version.

Changed default setting for beam and profile representation

The default setting for the type of beam and profile representation has changed. Beams and profiles are represented exactly by default from SP2 onwards. The default setting can be changed in the Configuration Editor at **Steel Engineering > Representation > Type of beam representation**. The changed default applies to new installations only.

Rectangular plate - New function

For the insertion of rectangular plates the new function **Rectangular plate** ▢ is now available at **Steel Engineering > Plate, new**. The previous function **Standard** has been moved to the sub-menu.

The new function can be used to install plates from the HiCAD catalogue **Semi-finished products > Plates**. The function basically corresponds to the Sheet Metal function **Create base sheet**.

- Semi-finished product selection,
- Insertion via points or value input,
- Referenced saving,
- Feature creation,
- Integrated preview function.

Enhanced bar list

As of SP2, the Excel bar list displays not only the waste but also the actual length used. The BOM template for Steel Engineering has been adapted accordingly. (see also: Report Manager - What's new?).
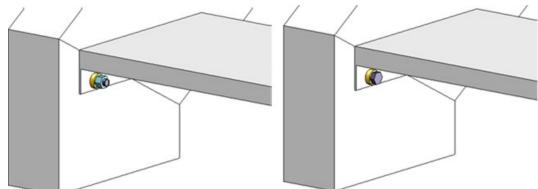
Enhanced Staircase Configurator

- Stairs can now also be created without platforms. For this purpose, the new variant **Do not create** is available on the **Platforms** tab.

- If **Standardized grating step** is selected as the variant on the **Steps** tab, the direction of the bolted connection can now be reversed. To do this, activate the new **Bolts from inside** checkbox.

Left: Bolts from outside, Right: Bolts from inside

Gratings - Dimension calculations

In the dimension calculation, the bearing bar direction of gratings is assigned to the attribute **Length** ($03). The bearing bar direction is ignored in the dimension calculation if the direction is set manually for the grating. This concerns the following cases:

- The dimension orientation is set.
- The part orientation is set.
- The processing direction is set.

The bearing bar direction is also taken into account for the part and dimension orientation.
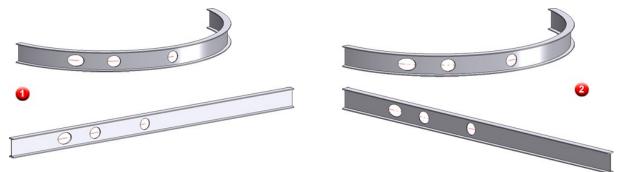
	The part orientation results from the bearing bar direction (only for Grating steps DIN 24537).
	The part orientation results from the bearing bar direction of the assembly main part (only for Gratings steps DIN 24537).
	The dimensions are up to date, the dimension orientation results from the bearing bar direction (only for Gratings steps DIN 24537).
	The dimensions are up to date, the dimension orientation results from the bearing bar direction of the assembly main part (only for Grating steps DIN 24537).

Unbend beams and profiles

Multi-part beams and profiles consisting of straight and circular curved sections can also be unbent. However, the sections must merge tangentially.

With HiCAD 2023 SP2 (V 2802) the geometric algorithm of the unbending calculation has changed. Therefore, there may be changes in the result. In particular, beams, which could also be unbent before, lie differently in space (see image). However, beams already unbent with a version before HiCAD 2023 SP2 will not be changed.
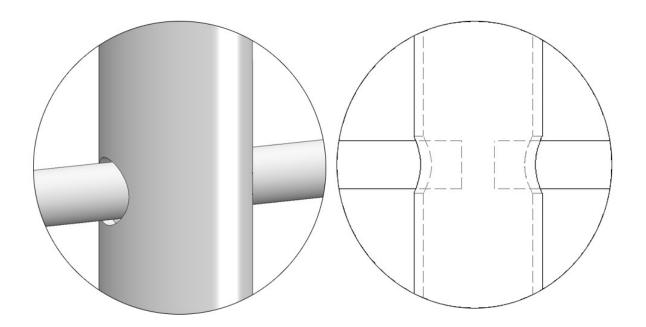
(1) Unbending with HiCAD 2022, (2) Unbending with HiCAD 2023 SP2

Railing Configurator - New variant for knee rails

Knee rails of a railing can now also be inserted into the post. For this purpose, the new variant **Insertion into post** is available on the **Infill** tab for infills with knee rails.
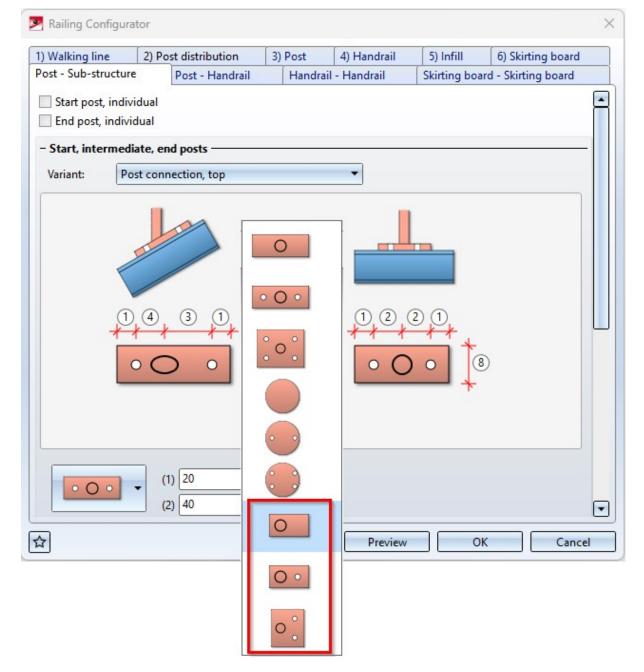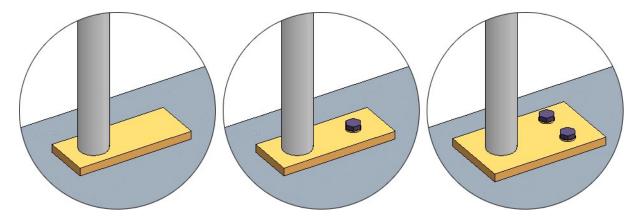
Service Pack 1 2023 (V 2801)

Railing Configurator - new base plate types

In the Railing Configurator, new base plate types are available on the **Post - Sub-structure** tab:

- One-sided, no bore,
- One-sided, 1 bore,
- One-sided, 2 bores in transverse direction

(1) One-sided, no bore; (2) One-sided, 1 bore; (3) One-sided, 2 bores in transverse direction
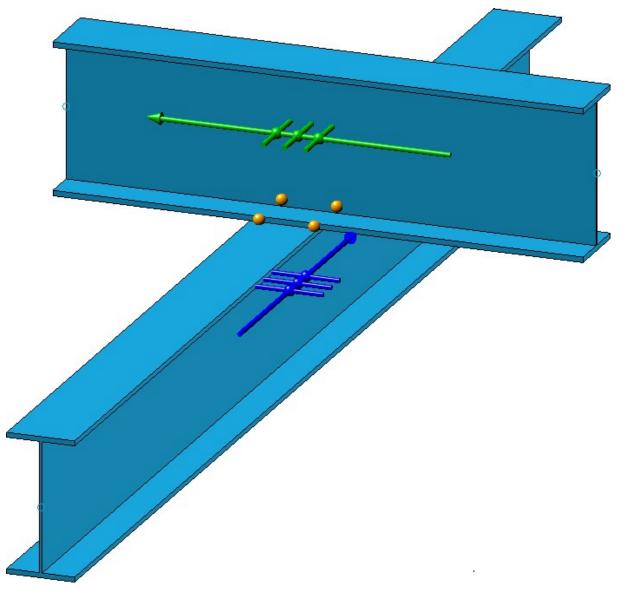
Purlin joints 3204 and 3206

The Purlin joints 1204 and 1206 have been modernised and adapted to the HiCAD standard. In this context, the joint numbers have been changed to 3204 and 3206.

For example, the current settings are now visualised in the model drawing during input, e.g. the insertion direction, stiffeners and boltings.

In addition, there are now - compared to the previous purlin joints - more setting options available. Examples are the offset of the purlin to the beam and the usages for the assemblies, etc.

Cross-bracings 2601 / 2602 - Distance of the gusset plates from the flange

From HiCAD 2023 SP1 onwards, Cross-bracings 2601 and 2602 can also be installed with indication of the distance of the gusset plates from the front or back flange. This is used, for example, in hall construction. There it can happen that a cross-bracing is to be installed between two differently sized beams whose upper flanges are on the same level.
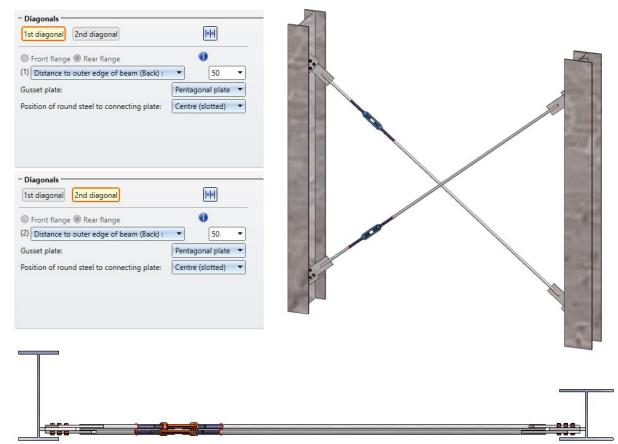
For this purpose, the **Fixing** tab has been extended accordingly.

In this case, however, the gusset plates cannot be fixed to the flange, i.e. the corresponding checkbox on the **Connecting points** tab must not be active.

Example:

The Cross-bracing 2601 shown was selected with the following settings for the 1st and 2nd diagonal:

Major Release 2023 (V 2800)

Dialogue windows for connections

Opening the dialogues of Steel Engineering connections and scrolling through the dialogues has been accelerated.

Circumferential seams for stiffeners

For stiffeners, circumferential seams are often used in practice, which also makes drawing in particular much easier. For this reason, the corresponding tabs of the following design variants have been extended:

Design variant	Tab
Stiffener (2401)	▪ Others
Strap joint (2310)	▪ Weld seams
Beam to web, with 2 plates + stiffener (1211)	▪ Weld seams- Front plate ▪ Weld seams- Stiffener
Column connection, Frame corner (2203)	▪ Weld seams- Front plate ▪ Weld seams- Tension plate / Top plate
Column connection, Frame corner (2204)	▪ Weld seams- Front plate ▪ Weld seams- Tension plate / Top plate

Please note that circumferential seams cannot be applied to haunches and ribs.

Beam to web, with 2 plates and stiffener - Optional 2nd stiffener

With the **Beam to web, with 2 plates and stiffener (1211)** connection, a second stiffener can now be optionally installed - aligned with the first stiffener.

Bar Configurator - Bars on free sketch

With the **Bar Configurator**, the area between the edges of a "free" sketch can now also be filled with bars. Until now, only square sketches were allowed here.
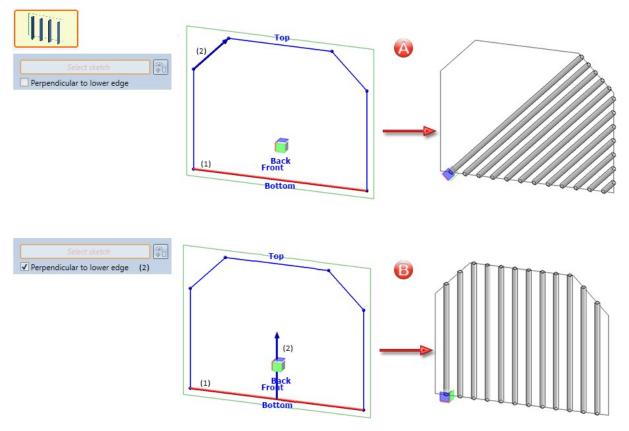
After selecting the sketch, you determine the lower edge of the installation area and then the direction of the installation or you select the installation perpendicular to the lower edge.

The installation of the bars is always determined by the sketch line defined as the **Bottom** (see A in the example), so the entire sketch is not necessarily filled with bars.

Example

The area between the six edges of the sketch shown is to be filled with bars..
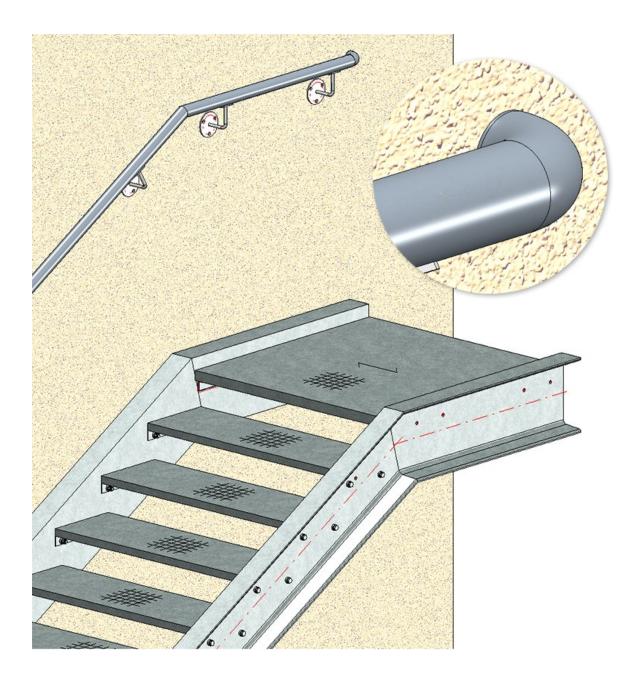
A: (1) lower edge, (2) direction / B: (1) lower edge, (2) perpendicular to lower edge

Railing Configurator - Elbow at hand rail end

On the **Handrail** tab of the Railing Configurator, elbows with/without end cap or steel plates can now be placed at handrail terminations.

3-D grid

As with the 3-D function of the same name, the dialogue window of the **3-D grid** function has been revised and modernised.

Also new is that you can use formulas instead of values for the distance specification - also in mixed units. The value is then converted into the unit of measurement of the model drawing.

Example:
Unit of measurement of drawing: mm, Distance: 1 3/4" + 12.5 mm. Result: 56.5 (mm).

Rectangular plates - Revised dialogue

The dialogue window of the function **Rectangular plates** has been revised and modernised.

Settings for weight calculation

In the Configuration Editor, the settings for weight calculation have been moved. You can now find the parameters at **Modelling > Part properties**.

Drawing Management

Service Pack 2 2023 (V 2802)

Updating mounting drawings

When updating mounting drawings created with the setting
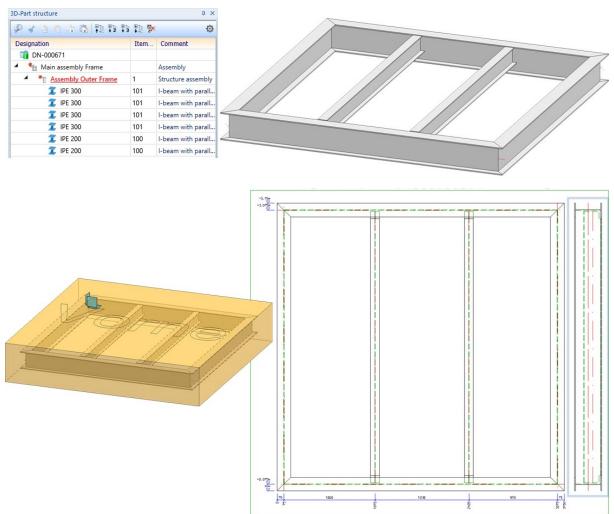
 New view group via active assembly(-ies),
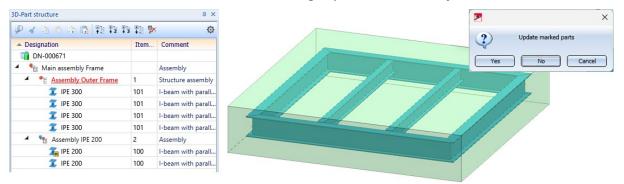
from SP2 onwards only the parts of the view group belonging to the selected assemblies are taken into account and not all parts lying within the bounding box.

Example:
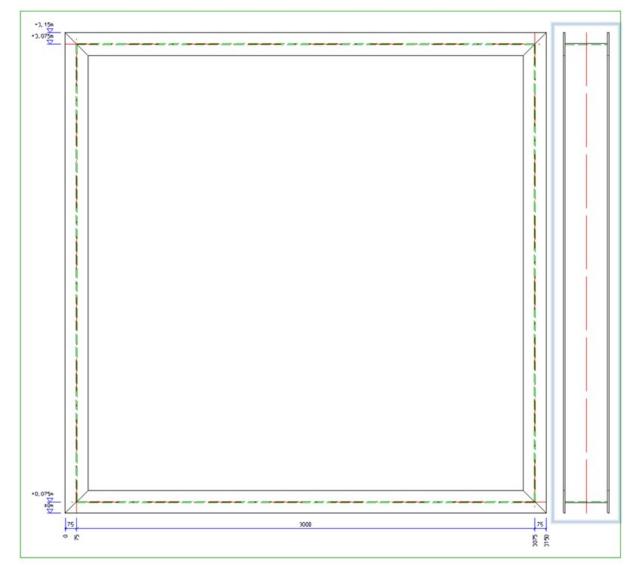
A mounting drawing with the setting **New view group via active assembly(-ies)** is to be created for the **Assembly Outer frame** shown below.

Now the two cross-beams (IPE200) are combined into one assembly, which is moved up one level in the hierarchy. This means that both assemblies are now on the same level. Next, the drawing is itemised and saved. Then the mounting drawing is updated with the function **New view group via active assembly(-ies)**.

Although the **IPE200** assembly is in the bounding box of the view group, it is now not included in the mounting drawing because it does not belong to the **Assembly Outer frame**.

Production drawings for mounting assemblies

In the Configuration Editor at **PDM > Drawing Management > Production drawings** the new parameter **Production drawings for mounting assemblies** is available. With this parameter, mounting assemblies (Type: Bolted assembly) can be excluded from the production drawings if required. The default setting is **Yes**.

If **No** is selected here, no production drawing is created for the mounting assembly. However, the drawings for the individual parts of the assembly are created.

Service Pack 1 2023 (V 2801)

Derive parts and assemblies

The following new Derive functions are now available:

Derive article masters and linked drawings of the active assembly, incl. sub-parts

With this function, a new assembly is derived (copied) from the active assembly - also possible with structure assemblies - including all sub-parts and all links. This means that article masters, indices, item numbers and all linked drawings are also copied.

Derive article master selected via HELiOS, incl. sub-parts and linked drawings
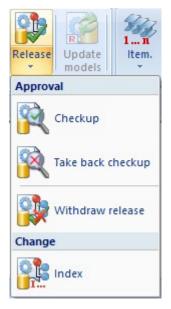
This function works essentially like the function above. Here, however, the active assembly is not derived, but the part or assembly to be derived is selected by selecting the corresponding article in HELiOS.

Major Release 2023 (V 2800)

Check, release, index models

To change the workflow status and to index models (.SZA files) that have not been automatically created by Drawing Management, it was previously necessary to switch to the HELiOS Desktop and call the corresponding function there via the context menu for articles/documents.

As of HiCAD 2023, these options are now available directly in the **Drawing Management** Ribbon via the **Release** function in the **Model** function group.

Output of PDF data in black and white

PDF files can now also be created in black and white. For this purpose, the settings in the Configuration Editor at **PDM > Drawing Management > External production documents** have been extended accordingly.

Workflow when creating general documents

As with production drawings, mounting drawings and customer drawings, it is now also possible to select when general documents in DXF, DWG and PDF format are to be created. For this purpose, the settings in the Configuration

Editor at **PDM > Drawing Management > External production documents** have been extended accordingly.

Metal Engineering

Service Pack 2 2023 (V 2802)

Extended transfer of LogiKal attributes

The HiCAD part attribute **LogiKal item name ($LPN)** was already assigned automatically in earlier HiCAD versions for glass panes, beams and profiles imported from LogiKal.

As of Service Pack 2 of HiCAD 2023, this attribute assignment is available individually for glass panes, beams and profiles.

Since this attribute is composed of several LogiKal attributes that are different for glass panes, beams and profiles, the item name can now also be transferred to other HiCAD attributes if required.

In addition, the attribute mapping between HiCAD and LogiKal has been extended by some more number and text attributes:

5 new, transferable numerical attributes are **[Glass] Acoustic Performance**, **[Glass] Light Transmittance**, **[Glass] Solar Factor** and **[Glass] Weight** for glass panes, as well as the **[Insertion] Handle height (from FFL)** for profile inserts.

These LogiKal attributes can be assigned to corresponding user-defined HiCAD attributes via the **Configuration Editor**.

New transferable text attributes for **Hardware** parts of profile inserts are also compiled from several LogiKal attributes: If **Hardware** parts exist, a total of ten attributes each contain a composite description text of the **Hardware** part of an **Insertion** part.

Description	Value
General	
Automatically update LogiKal items in HiCAD	Do not update automatically
Attribute assignment during import	
✎ [Position] Description	Designation 1
[Position] Brief Description	(none)
[Position] Position Number	(none)
[Position] Rule Package	(none)
[Position] Subsystem	(none)
[Glass] Description 1	(none)
[Glass] Description 2	(none)
[Glass] Acoustic Performance	(none)
[Glass] Light Transmittance	(none)
[Glass] Solar Factor	(none)
[Glass] Weight	(none)
🔒 [Glass] U-value	Heat transfer coefficient (U-value)
[Glass] Item name	LogiKal item name
[Insertion] Type	(none)
[Insertion] Opening direction	(none)
[Insertion] Handle height from FFL	(none)
[Insertion] Hardware Package	(none)
✎ [Insertion] Hardware 1	Designation 1
[Insertion] Hardware 2	(none)
[Insertion] Hardware 3	(none)
[Insertion] Hardware 4	(none)
[Insertion] Hardware 5	(none)
[Insertion] Hardware 6	(none)
[Insertion] Hardware 7	(none)
[Insertion] Hardware 8	(none)
[Insertion] Hardware 9	(none)
[Insertion] Hardware 10	(none)
[Beam/Profile] Item name	LogiKal item name

Importing all items of a LogiKal project

From SP2 onwards another import function is available:

 All facades+insert elements

With this function all items of a LogiKal project can be imported into HiCAD in one step.

After calling up the function, the LogiKal Project Management dialogue is displayed as with the single import.

Select the desired project and click on **Select Project**.

The **Fitting options** dialogue window is then displayed.
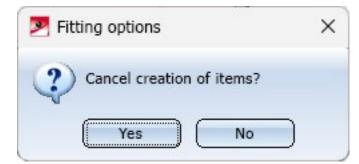
Select the desired options and then **OK**.

The LogiKal data for all items of the project are now prepared internally. This may take a moment, depending on the size and number of items of the project.

The import will then be started. A progress bar is displayed during the import. The imported items will be automatically placed in the origin of the drawing.

The import can be cancelled by pressing the ESC key. The item import currently in progress will be completed before the prompt shown appears:

If you click **Yes**, the import will be cancelled.

If errors occurred during the import or if an item cannot be imported at all, an error log will be displayed:

Major Release 2023 (V 2800)

Omission of the LogiKal login window

Until now, a login window was displayed the first time a LogiKal function was called up:

This window is no longer needed and will therefore be omitted from HiCAD 2023.

Configuration of the name of LogiKal items in HiCAD

Up to now, when importing LogiKal items into HiCAD, the fixed setting **Project name + Position** was always used as the name of the top assembly, both separated by commas and with the character # in front of the number of the LogiKal item (e.g. Rounded glass, #001).

As of HiCAD 2023, only the item, the project name or the short description can be used as the name of the assembly. The Fitting options dialogue window has been extended accordingly.

Please note:

- The setting for the level of detail and for the name of the top assembly Detail are defined once when importing a LogiKal item. HiCAD remembers this setting for the built-in LogiKal item. A subsequent change is not possible.

- When updating a LogiKal item imported into HiCAD, the name is also updated according to the selected name setting.

- LogiKal items imported into a model drawing with a version before HiCAD 2023 do not have a name setting. Here, the setting **Project name + Position** is used as the name of the assembly as before (before HiCAD 2023).

LogiKal Interface – Transfer of Text Attributes to HiCAD

When importing LogiKal items (which are called "positions" in LogiKal), certain LogiKal text attributes can be specifically assigned to certain HiCAD attributes. The setting is made in the Configuration Editor at **Interfaces > LogiKal**.

This is possible for the following LogiKal attributes:

Attribute	Attribute type	Example / Explanations
Description	Position attribute	e.g. window element
Description short	Position attribute	is defined by the user
Position number	Position attribute	e.g. 001
Rule package	Position attribute	e.g. RC3
Subsystem	Position attribute	e.g. BL
Description 1	Glass attribute	is defined by the user for project glasses
Description 2	Glass attribute	is defined by the user for project glasses
U-value	Glass attribute	The U-value is transferred to the HiCAD attribute §26=Heat transfer coefficient (U-value). This setting is fixed and therefore locked.
Type	Insertion attribute	e.g. fixed field

Attribute	Attribute type	Example / Explanations
Openiing direction	Insertion attribute	e.g. L or R
Hardware package	Insertion attribute	WIC 40001 REBATE GEAR 160 KG FITTING CONCEALED

In order to assign one of the listed attributes to a HiCAD attribute, simply select the desired HiCAD attribute in the selection box in the corresponding line, e.g.

In addition to the attributes listed above, the following HiCAD attributes are automatically assigned:

- The **LogiKal Position name** is automatically assigned to the HiCAD attribute **$LPN**.

- For the top assembly of the LogiKal item the HiCAD attribute **Article number $BB** is determined by the settings in the Fitting options dialogue when importing LogiKal positions (name of the top assembly).

If one of the LogiKal item attributes listed in the table above is assigned to the HiCAD attribute $BB, then this assignment has priority over the name setting in the fitting options.

If further LogiKal attributes are to be transferred, the use of customer-specific system attributes may also be useful. You can find out more about this in the Help of the Catalogue Editor in the topic System Attributes and System Colours.

Please note:

■ The attribute assignment settings are set once when importing a LogiKal item. HiCAD remembers this setting for the installed LogiKal position. A subsequent change is not possible.

■ When updating a LogiKal position imported into HiCAD the attributes are also updated according to the noted attribute assignment.

■ For LogiKal positions imported into a model drawing with a version before HiCAD 2023, no attribute assignment is used.

Element Installation

Service Pack 2 2023 (V 2802)

Feature - Enhanced Create new function

When editing an element installation, no complete recalculation is performed, but only the installation elements selected for editing are handled. If, for example, you have edited parts of an installation element with functions outside the element installation, e.g. moved, deleted or drilled, <u>and</u> this installation element is <u>not</u> changed by selecting another variant or changing the parameters, then these parts remain moved, deleted, drilled, etc.

This behaviour can be changed afterwards with the function **Create new** in the context menu of the feature **Element installation**. Previously, dimensions and other edits, e.g. drilling, to parts of the installation elements were lost. As of SP2, these are retained.

The same applies to sub-structures.

You can find an example here.

Major Release 2023 (V 2800)
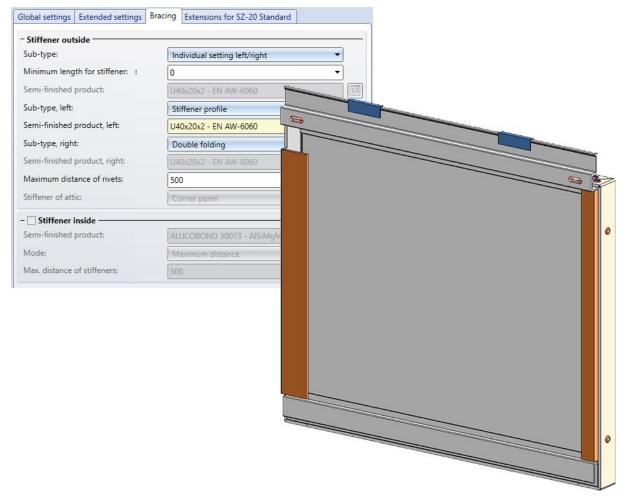
ALUCOBOND® SZ 20 tray panel (with accessories)

New tab

When installing ALUCOBOND® SZ 20 tray panels (with accessories), the settings for stiffener profiles are now available on a separate **Bracing** tab.

Bracing - Individual setting left/right

For the stiffeners, the settings for the left and right sides can be defined separately. To do this, select the new **Individual setting left/right** option for **Sub-type**.

Maximum spacing of screws for sub-structure

For ALUCOBOND® SZ 20 tray panels, the maximum spacing of the screws for the sub-structure can now be defined. The **Extended settings** tab has been adapted for this purpose.

ALUCOBOND® SZ 20 Polygon (3-8) - Stiffener profiles

When installing panels of type ALUCOBOND® SZ 20 Polygon (3-8), further options are available for the edge configuration:
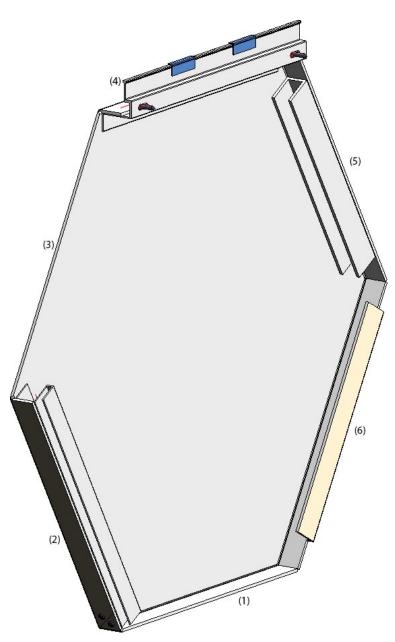
- **Stiffener profile**
  Stiffener profiles are always riveted to the sheet. You define the profile type and the maximum distance of the rivets on the **Extended settings** tab.

- **Double folding**
  Instead of inserting stiffener profiles on the edge, the material can also be double folded.

(1) Standard, (2) 44024, (3) Without flanges, (4) 44025, (5) Stiffener profile, (6) Double folding

Design variants - SZ20 Base point with projection

The dialogue window now has a **Preview** button, so that the result can be visualised and checked graphically on the basis of the current entries before applying any changes.

Revised Wall bracket ALUCOBOND U

The **ALUCOBOND U-bracket** table has been replaced by the **Systea U-bracket** table. These wall brackets consist of two parts:
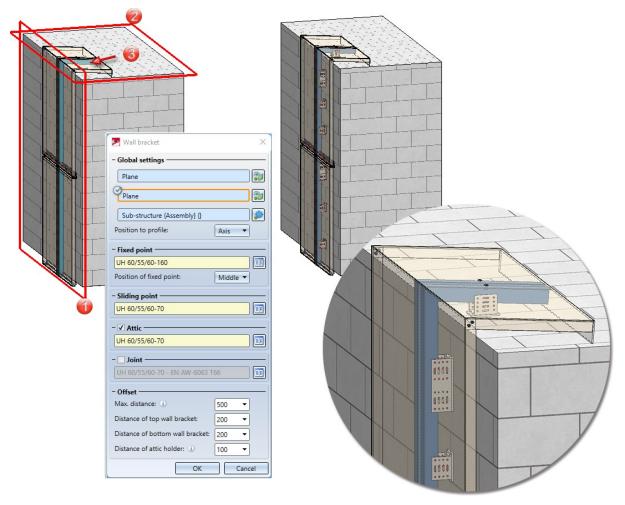
- Wall consoles (Systea)
- Thermostop (Systea)

As an example, the following brackets are provided:

- Fixed point:
  - UH 60/55/60-160
  - Thermostop 40/6-160 mm
- Sliding point :
  - UH 60/55/60-70
  - Thermostop 40/5-85 mm

The following image shows a wall with an ALUCOBOND SZ20 element installation with an attic and a sub-structure. The wall brackets have been installed with the settings shown.

Left: (1) Wall plane, (2) Roof plane, (3) Sub-structure/ Right: Result

The wall brackets are installed as a structure assembly and combined in an assembly called **Fixing**.

Designation	Item...	Comment
WALL_BRACKET_SYSTEA		
wall		
▷ Sub-structure		Assembly
◢ Element installation		Assembly
▷ ALUCOBOND SZ 20 K...		Structure assembly
▷ ALUCOBOND SZ 20 K...		Structure assembly
◢ Fixing		Assembly
◢ U-bracket 60/55/60-160		Structure assembly
UH 60/55/60-160		Systea
Thermostop 40/6-...		Thermostop
▷ U-bracket 60/55/60-160		Structure assembly
▷ U-bracket 60/55/60-70		Structure assembly
▷ U-bracket 60/55/60-70		Structure assembly
▷ U-bracket 60/55/60-70		Structure assembly
▷ U-bracket 60/55/60-70		Structure assembly
▷ U-bracket 60/55/60-70		Structure assembly

2D-Part structure | 3D-Part structure

ALUCOBOND © easy fiX

When installing ALUCOBOND easy fiX profiles, you now have the option of selecting a **Mode** for their insertion:
Maximum distance
In this mode, the number of profiles created is calculated using the value specified here.

- **Number**
  If this mode is selected, the value specified here determines how many profiles will be created. This value only applies to profiles that are not continuous.

What's new?

Profile Installation

Service Pack 1 2023 (V 2801)

Part filters for sub-parts

With the **Find** ![funnel icon] function in the transparent toolbar, the sub-parts of the profile installation can also be easily selected from SP1 onwards. This concerns the following parts:

- Contour, cut,
- Contour, uncut,
- Sandwich panel, outside sheet,
- Sandwich panel, inside sheet,
- Insulation and
- Others, e.g. sealing chords.

To filter these parts, the rule **Profile sub-part type (Profile Installation)** is available under **Properties**.

If you select this rule (is / is not), you can determine the sub-part type to be filtered.

Example:

Let's take a look at the depicted part structure of a 3-D drawing. Only the first assembly consists of assemblies created by the profile installation. However, both assemblies contain outside and inside sheets with the same name.

Now we define the following part filter:

If this filter is executed, then only the **Sandwich panel, outside sheet** that was created by the profile installation will be selected.

Designation	Item...	Comment
40398_TEST		
▲ Contains such parts		Assembly
▲ BRUCHA Panel FP 60 Profi...		Sandwich panel
Insulation		Insulation
Sealing chord		
Outside sheet		Sandwich panel, outside sheet
Inside sheet		Sandwich panel, inside sheet
CONTOUR		
CONTOUR		
▲ FALK 1000/35 TR (0.55 m...		Trapezoidal profile
FALK 1000 TR		
CONTOUR		
CONTOUR		
▲ Does not contain such parts		Assembly
▲ Ondatherm 2000 B (140...		Sandwich panel
Insulation		Insulation
Sealing chord		
Outside sheet		Sandwich panel, outside sheet
Inside sheet		Sandwich panel, inside sheet
Z1 Bargeboard profile 300...		

Coating
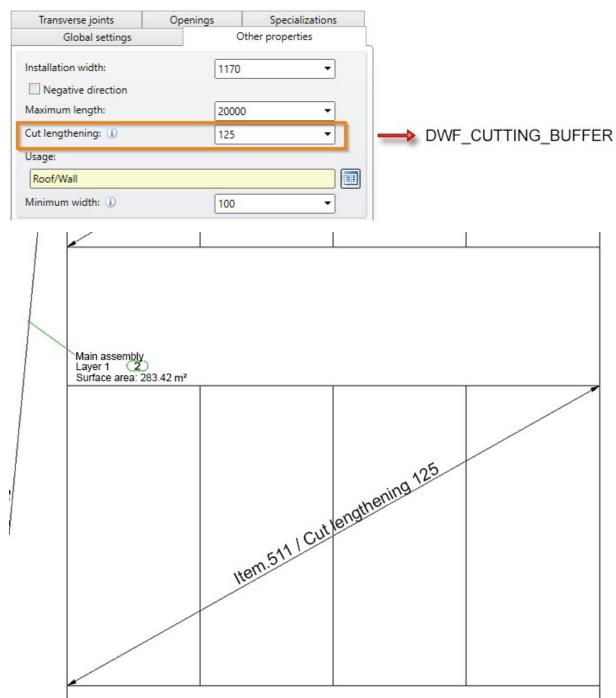
Up to now, the inside and outside colour could be selected from the **Textures and Colours** catalogue in the profile installation dialogue. As of HiCAD 2023 SP1, the coating type is selected from the catalogue **Factory standards > Surface treatment** instead. This applies to the **Other properties** and Specializations tabs.

Major Release 2023 (V 2800)

Cut lengthening - new part attribute

The **Cut lengthening**, i.e. the extension of diagonally cut installation elements, is now assigned to each installation element as the attribute **DWF_CUTTING_BUFFER**. In this way, the value can also be output in annotation blocks, for example..

Changing the property of individual profiled sheets

HiCAD 2023 also offers the possibility to change the properties of individual profiled sheets. For this purpose, the profiled sheet installation dialogue has been extended by a new **Specializations** tab.

With the options of this tab, you have the possibility to assign a profile type to individual or several profiled sheets that differs from the basic settings of the installation surface. In addition, the selection of colour and interior colour is possible.

The function also supports multiple selection.
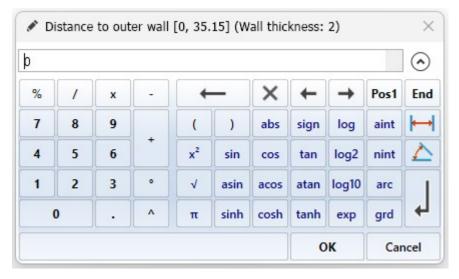
Plant Engineering

Service Pack 2 2023 (V 2802)
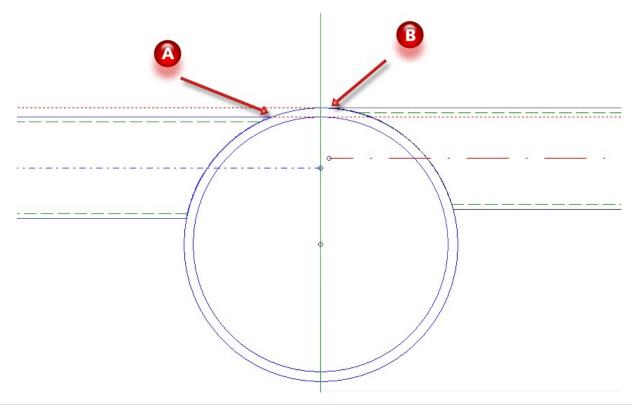
Eccentrically inserted pipes

The insertion of a pipe is often done by placing both pipes on the ground and then cutting/welding them together. Accordingly, the inserted pipe must be able to have a distance 0 from the outer wall of the larger pipe. However, the previous implementation in HiC 2023 (V 2800 and 2801) forces a distance of at least the wall thickness of the larger pipe.

For this particular use case, the function **Align component connection tangentially** has been adapted in SP2. Instead of the internal distance to the outer wall, the distance to the outer wall must now be specified.

In addition to the sensible input range, the wall thickness of the larger pipe is now displayed in the title line of the input window. So if you want to emulate the previous behaviour (A) and assign an internal distance of 0 to the inserted pipe, enter the wall thickness as the distance.

(A) up to HiC 2023 SP1: Outer diameter of inserted pipe tangential to inner diameter of main pipe

(B) from HiC 2023 SP2 onwards: Outer diameter of inserted pipe tangential to outer diameter of main pipe, constructive distance 0.1 mm

ℹ️ **Please note:**

To avoid unaesthetic protrusions on the inserted tubes, the function ensures a minimum constructive distance of 0.1mm.

Pipe parts: Prevent seals

The insertion of seals and gaskets can be prevented via the article master of a corresponding part. To do this, set the attribute value for **Seal** in the article master to the value **-** (minus). This ignores seal settings at all connections of the part, i.e. no seals are set. This also applies when other parts are connected to this part.

Example:

Nozzles for DIN EN 1092-1

With SP2, new variants are available for nozzles:

- STUTZEN_FL_PN10.vaa
- STUTZEN_FL_PN16.vaa
- STUTZEN_FL_PN25.vaa
- STUTZEN_FL_PN40.vaa

These are revisions of the variant STUTZEN_FL1.vaa, which still refers to the old standard DIN 2630.

The following seal variants have been added to the parts inventory to match the new nozzles:
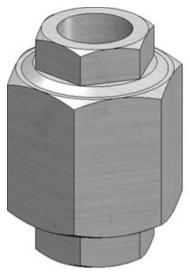
- DIN1514-1-PN10.vaa
- DIN1514-1-PN16.vaa
- DIN1514-1-PN25.vaa
- DIN1514-1-PN40.vaa
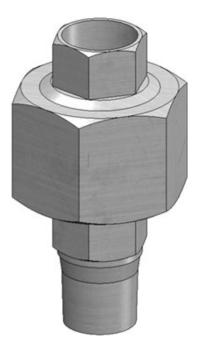
These variants are summarised in the list file nozzles_1092-1.lst.

New malleable cast iron fittings

The stock of Georg Fischer malleable cast iron fittings according to standard EN 10242 has been extended by two further variants:

- GF10242_S330.VAA und
- GF10242_S331.VAA.

Welding neck flanges acc. to EN1092-1-11-C/D

The parts inventory for Plant Engineering has been expanded with SP2 to include the following variants:
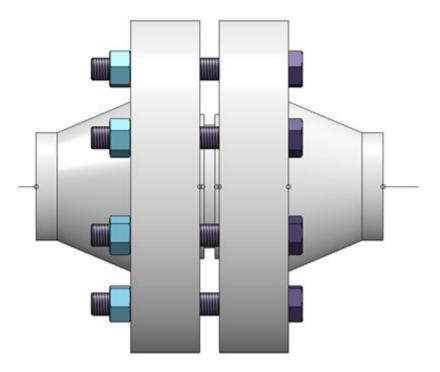
Flanges with notch	Flanges with groove
EN1092-1-11-C-PN10.vaa	EN1092-1-11-D-PN10.vaa
EN1092-1-11-C-PN16.vaa	EN1092-1-11-D-PN16.vaa
EN1092-1-11-C-PN25.vaa	EN1092-1-11-D-PN25.vaa
EN1092-1-11-C-PN40.vaa	EN1092-1-11-D-PN40.vaa
EN1092-1-11-C-PN63.vaa	EN1092-1-11-D-PN63.vaa
EN1092-1-11-C-PN100.vaa	EN1092-1-11-D-PN100.vaa
EN1092-1-11-C-PN160.vaa	EN1092-1-11-D-PN160.vaa
EN1092-1-11-C-PN250.vaa	EN1092-1-11-D-PN250.vaa
EN1092-1-11-C-PN320.vaa	EN1092-1-11-D-PN320.vaa
EN1092-1-11-C-PN400.vaa	EN1092-1-11-D-PN400.vaa

Also new are the following variants:

EN1092-1_Insertring.vaa   Insert ring to connect two grooved flanges to each other.

The insert ring is classified as a valve and behaves like an intermediate flange valve when bolted together.
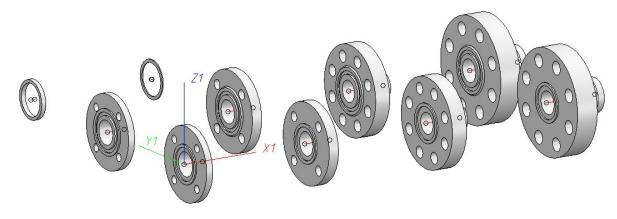
Seals

DIN2691_grooved_gasket.vaa

EN1541-1_FLAT_GASKET.VAA

EN1541-2_SPIRAL_GASKET.VAA   The thickness of the gasket represents the thickness after installation.

Flanges up to and including PN63 have a seal according to EN1541-1 in their attribute DICHTUNG (Seal). The flanges from PN100 onwards, on the other hand, have the seal according to EN1541-2.

All new variants are summarized in the list file **EN1092-1-11-C+D.lst**.
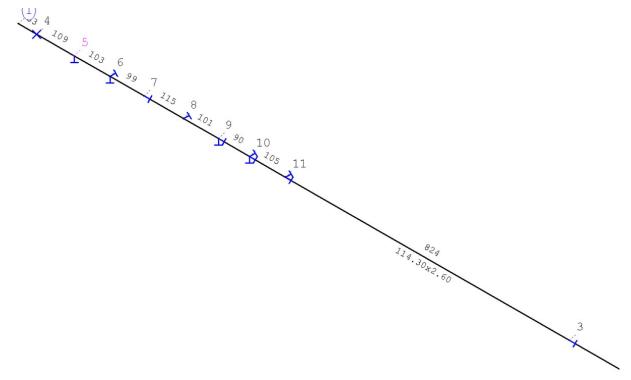
New part attribute "Pipe clamp support type"

As of SP2, the attribute **Pipe clamp support type** - attribute name PIPE_SUPPORT_TYPE - can be assigned to pipe clamps in the part attributes.

By assigning this attribute, different pipe clamp types can be displayed in ROHR2.

BOM tables - Line breaks

Three BOM tables can be generated automatically in isometries and pipe spool drawings.

- Pipe BOM,
- Pipe length list and
- Weld seam list.

The cells of these tables are filled from the attributes of the parts or by fixed default values. You can define the width of such a cell in the Configuration Editor.

On the other hand, this can also be done directly in the isometry or pipe spool drawing settings if you deactivate the **Column layout** option.

Previously, texts that were too long for a table cell were automatically shortened by HiCAD. The shortening was marked by an ellipsis (...). As of HiCAD 2023 SP2, texts that are too long are wrapped.

(1) before HiCAD 2023 SP1, (2) as of HiCAD 2023 SP2

However, the automatic line break is only applied to BOMs that appear on the drawing. BOMs that are saved in files contain the full text without shortening and without additional line breaks.

If you edit the BOM manually, no line breaks are inserted. If the lines become too long, the text is shortened. When manually editing with the **Edit list** function, you have the option of forcing a line break with <CTRL>+<INPUT>.

Part insertion - Check pipe class for reference parts
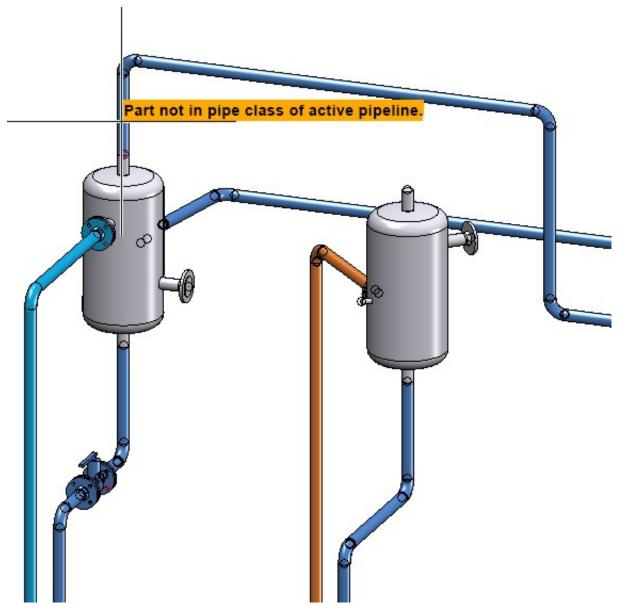
If a pipeline with a pipe class is active when selecting the **Part insertion** function, then the functions

- **Select reference part** and
- **Select reference part depending on part type**

a message is displayed if the pipe class of the selected reference part does not match the pipe class of the pipeline.

Part not in pipe class of active pipeline.

However, the part can still be selected as a reference part.

Isometry - Set reference coordinate system per pipeline

Normally the world coordinate system of the layout plan is used as the reference coordinate system for determining coordinates in the isometries and pipe spool drawings. However, you can give pipelines a reference coordinate system that is taken into account when generating isometries and pipe spool drawings, e.g. when displaying coordinates in annotation tags.

To do this, use the **Set reference CS for isometry** function (**Plant Engineering > Isometry/Pipe Spool Drawing > Ref...**).

Previously, a reference coordinate system set with this function was valid for all pipelines in the drawing. It was not possible to restrict this to individual pipelines.

For SP2 this function has been adapted so that it sets the reference coordinate system of individual pipelines.

The prerequisite is that a local coordinate system exists in the layout plan. For this purpose you can use, for example, the function **Plant Engineering > Settings > Settings** ▾ **> Set on connection** . If no local coordinate system is defined, a corresponding message appears:
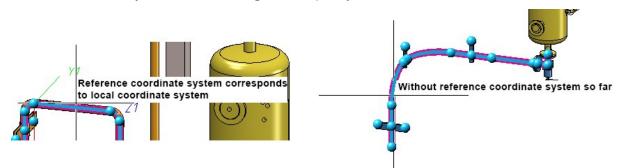
If a local coordinate system exists, you are prompted to select a pipeline when you start the **Ref...** function. you will be asked to select a pipeline.

As soon as the graphic cursor is moved over a pipeline, it is displayed directly at the cursor whether the pipeline already has a reference coordinate system.

One of the three following messages can be displayed:

- Without reference coordinate system so far
  By clicking on the pipeline the local coordinate system is taken over as the reference coordinate system of the pipeline.

- Reference coordinate system corresponds to local coordinate system

- Reference coordinate system deviates from local coordinate system
  The local coordinate system has been changed subsequently.

In the dialogue of the isometry as well as the pipe spool drawing generation it is marked by the symbol whether a pipeline has a reference coordinate system.
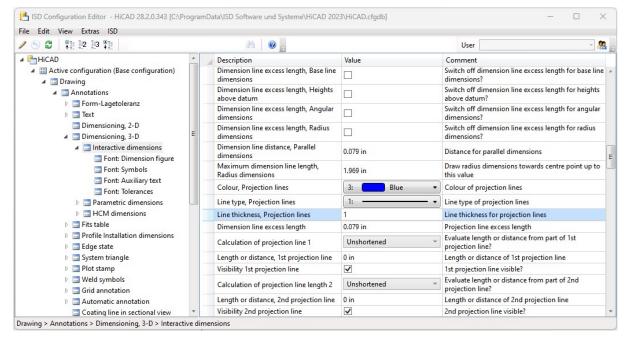
Likewise, the function **Delete reference CS for isometry** has been adjusted, i.e. the function deletes the reference coordinate systems of individual pipelines. As soon as the graphic cursor is moved over a pipeline, you are informed directly at the cursor whether the pipeline already has a reference coordinate system. The messages are the same as when setting a reference coordinate system.

Display of projection lines in isometry and pipe spool drawing

To display the projection lines in the isometry and in the pipe spool drawing, use the settings in the Configuration Editor at **Drawing > Annotations > Dimensioning, 3-D > Interactive dimensions**.

PartDataAutoSync / DBPlantDataImport

PartDataAutoSync - Deleting articles
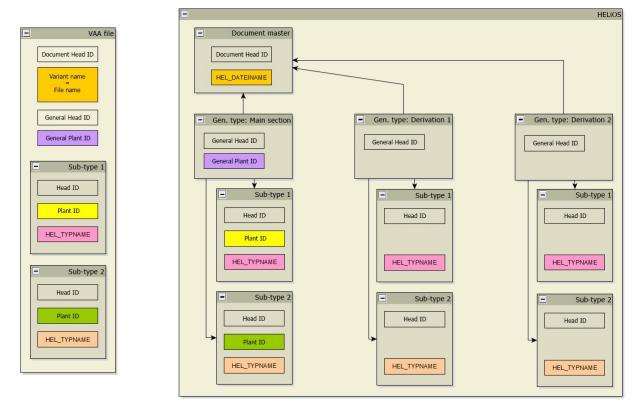
The tool **PartDataAutoSync** deletes articles for which there is no sub-type in the variant file. If the deletion fails, the message *Data AutoSync incomplete* is displayed directly in the dialogue. A list of the articles for which deletion failed appears in the log, e.g.:

Unsynchronized articles:
Deletion of article failed: HELIOS:\\B00XIKCF0R3DD0000040F0_0

Check of Consistency Between Archive File and HELiOS Article

In both **PartDataAustoSync** and **DBPlantDataImport** you can now get an overview of how the sub-types of an archive file (i.e. VAA or PAA) are displayed as linked articles in HELiOS.

For a concrete variant, there is usually a linkage structure that looks something like this:

So in the VAA file several sub-types are defined, which are identified by an article type, which in turn is mapped in HELiOS to the attribute **HEL_TYPNAME**.

When the VAA file is transferred to HELiOS, the following are created:

- a document master,
- an article master for the general type and
- one article master for each sub-type.

These articles together with the general type form the **main section** and are connected to each other by links. The articles are characterised by the fact that identifiers are stored for them, which can be found again in the VAA file. These are the **Plant IDs**.

In addition, further sub-sections may have been created with the HELiOS function Derive variant ![icon] .. These are also linked to the document master, but have no unique identifiers. Instead, these items can be assigned to the sub-types via the item type.

To get an overview of the current state of this linkage structure before matching a variant, use the function **Consistency check File <-> HELiOS**. This function is available in the part data sync in the context menu of the file selection.

After executing the function, the file **compare_file_to_HELiOS_protocol.txt.** can be found in the HiCAD Temp directory. In addition, the Notepad opens automatic and displays the contents of the file. The content looks something like this:

```
09.01.2023 09:14:12:
File: A2RF1_100.vaa


General type: HELIOS:\\B00A137W6LK2UT00008TXJ_0
Status: Sub-section derived via HELiOS
Article types of sub-types for which no article exists:
300
Articles without article type: none
Articles with article type that does not exist in the archive:
HELIOS:\\B00A137W6LK2UT00008TXK_0
HELIOS:\\B00A137W6LK2UT00008TXN_0
Articles that could not be accessed: none


General type: HELIOS:\\B00A137W6LK2UT00008TXX_0
Status: Sub-section derived via HELiOS
Article types of sub-types for which no article exists:
300
Articles without article type:
HELIOS:\\B00A137W6LK2UT00003NR8_0
Articles with article type that does not exist in the archive:
HELIOS:\\B00A137W6LK2UT00008TXY_0
HELIOS:\\B00A137W6LK2UT00008TY1_0
Articles that could not be accessed: none


General type: HELIOS:\\B00DH4XULBEB5J000004FY_0
Status: Error: Main section with additional articles
Article types of sub-types for which no article exists:
300
Articles without article type: none
Articles with article type that does not exist in the archive:
HELIOS:\\B00DH4XULBEB5J000004G0_0
HELIOS:\\B00DH4XULBEB5J000004G3_0
Articles that could not be accessed: none
```

At the top of the file, the date and time of the check is shown, followed by the VAA file that was checked. Then follows one paragraph for each derived section. So in the above example there are three derivation sections. For each section, the article of the general type is first output as a HELiOS link. This is followed by the status of the section, i.e. main or sub-section.

Next, it is listed whether there are deviations from the ideal state in this section, such as:

- Article types of sub-types for which no article exists.
  This is OK and occurs if there are additional sub-types in the VAA that have not yet been transferred to HELiOS.

- Articles without article type.
  Articles that do not have an article type are linked to the general type. Either this has been deleted or an invalid link has been created. In either case, this is an error.

- Articles with article type that does not exist in the archive.
  Usually occurs when a sub-type has been deleted from the archive. This is not an error. However, it can also occur if a link has been created to an article that does not come from the VAA file but has an article type. This would be an error.

- Articles that could not be accessed.
  Errors when accessing the article, this first says nothing about the link structure.

Part data sync - Restoring the main section for derivation

It can happen that the main section of a variant is no longer in the database, usually because articles were accidentally deleted.

Until now, the only way to restore this main section was to remove the variant from the database and match it again. This is particularly annoying if further derivations have already been created for the variant via the HELiOS function
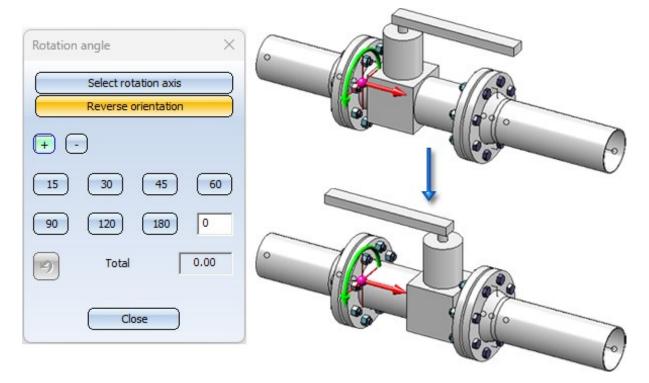
Derive variant , because these are also removed in the process and have to be created again.

As of SP2, the part data sync restores the main section if it could not be found.

Service Pack 1 2023 (V 2801)

Rotate part: Reverse orientation

The function **Rotate part** has been extended. You have now the option to reverse the orientation of a part.

However, this is only possible for parts that have two connections and could be placed on a straight piece of guideline, i.e. parts for which a rotation cannot lead to inconsistencies with the possibly underlying guideline.

Check article numbers of pipelines

As of SP1, HiCAD 2023 offers the possibility to check whether all pipelines in your drawing have an article number and whether this number is unique.

For this purpose, the function

 Check article numbers of pipelines

is now available at **Plant Engineering > Pipeline Tools > Coll...** ▾ **> ...**. After the check has taken place, a dialogue is displayed with information about the status of the pipelines. By clicking on an article number, the corresponding pipeline can be selected directly.
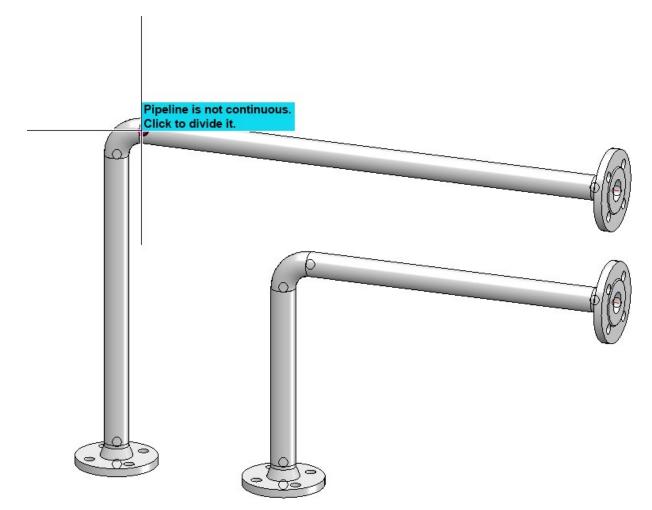
The result list can be output as a CSV file.

Divide pipeline into connection components
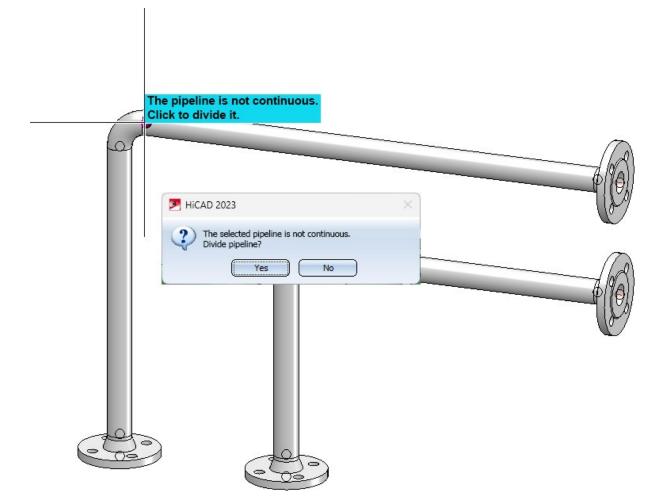
Sometimes it can accidentally happen that one pipeline has been constructed which should actually consist of several pipelines. That is, the pipeline consists of several so-called connection components. The following image shows such an example:
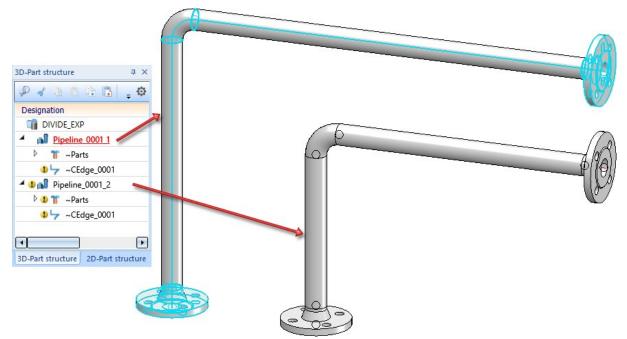
The **Divide pipeline** function has not supported the division of such a pipeline so far. As of HiCAD 2023 SP1, the system checks whether the pipeline to be divided consists of several connection components and indicates this - if so - at the cursor by a corresponding message.

Pipeline is not continuous.
Click to divide it.

If the pipeline is to be divided, it is sufficient to click on the pipeline and to answer the subsequent query with **Yes**.

The pipeline is not continuous.
Click to divide it.

HiCAD 2023

The selected pipeline is not continuous.
Divide pipeline?

Yes   No

Each connection component will then become a separate pipeline.

3D-Part structure

Designation

DIVIDE_EXP

Pipeline_0001_1
  ~Parts
  ~CEdge_0001
Pipeline_0001_2
  ~Parts
  ~CEdge_0001

3D-Part structure   2D-Part structure
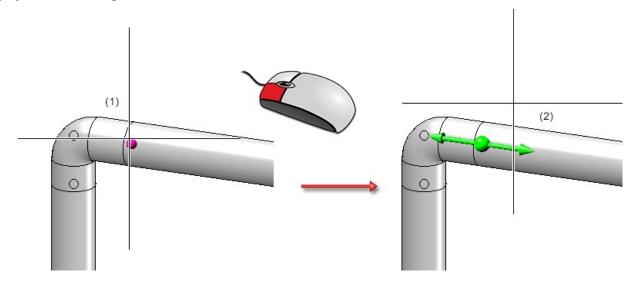
Set weld seam gaps manually

At **Plant Engineering > Pipeline Tools > Change** ▾ **> ...**you can find the new **Add/remove gap** function. With this function weld seam gaps can be added to or removed from a pipeline. The behaviour of the function depends on whether the Consider weld seam gaps checkbox is active in the Plant Engineering settings on the weld seam gaps tab.

Consider weld seam gaps is activated

After calling the function, HiCAD prompts you to select a connection. If you move the cursor near a weld seam, it will be highlighted (1). To select the connection for editing, press the left mouse button. HiCAD then prompts you to determine the way in which a gap is to be added or removed. For this purpose, selectable graphic elements are displayed in the drawing (2).
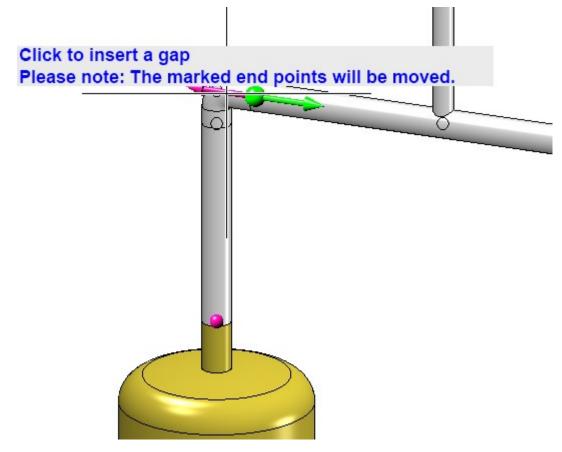
In the example shown you can make the connection

- on the left side,
- centrally, or
- on the right side.
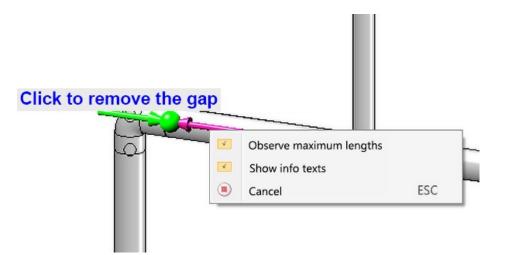
These options are pointed out to you when you move the cursor over the respective graphic element.

If end points of the pipeline are moved by adding or removing a gap, this is indicated. In the example shown above (left) this hint appears because the pipeline is connected to a vessel.
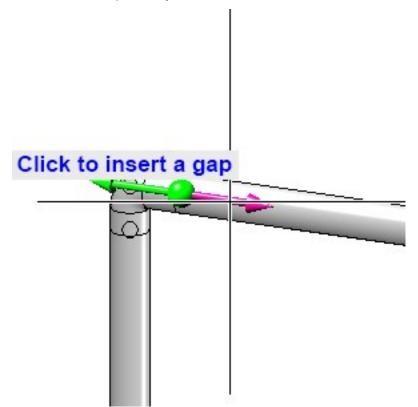
With a right-click a context menu can be activated, which can also be used to deactivate the display of the information texts.
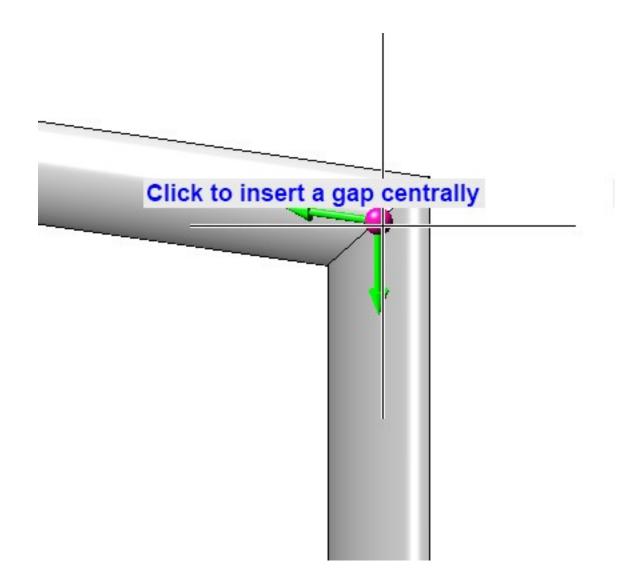
In addition, the consideration of maximum lengths can also be deactivated here, but this should be unnecessary as a rule.

After you have edited a gap (or pressed ESC), you will be asked to select another connection. In the example, we select the same connection that we have previously edited.
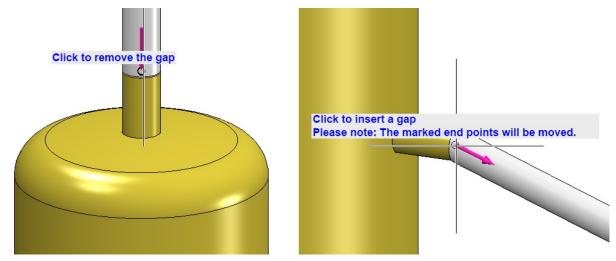
From the direction of the arrows you can see directly whether a gap is inserted or closed.

At mitres, the graphic elements are drawn according to the direction of the pipes involved.

Click to insert a gap centrally
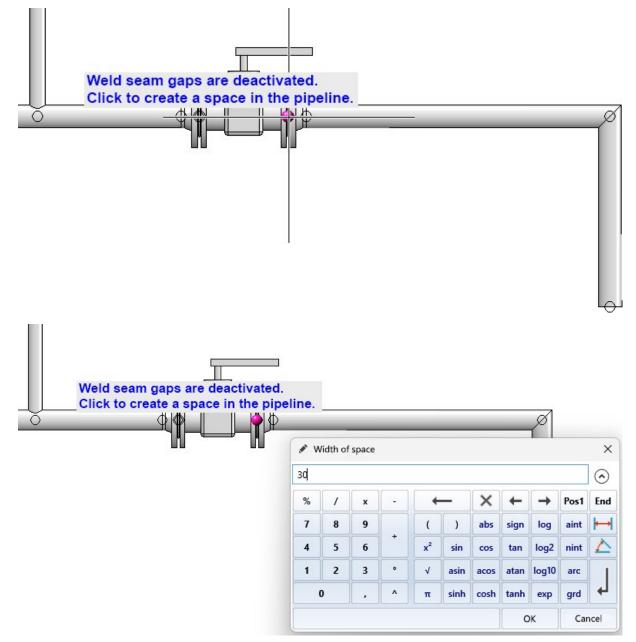
The position of components such as vessels and pumps is not changed by the function. Accordingly, only one possible action is offered here.

Click to remove the gap

Click to insert a gap
Please note: The marked end points will be moved.
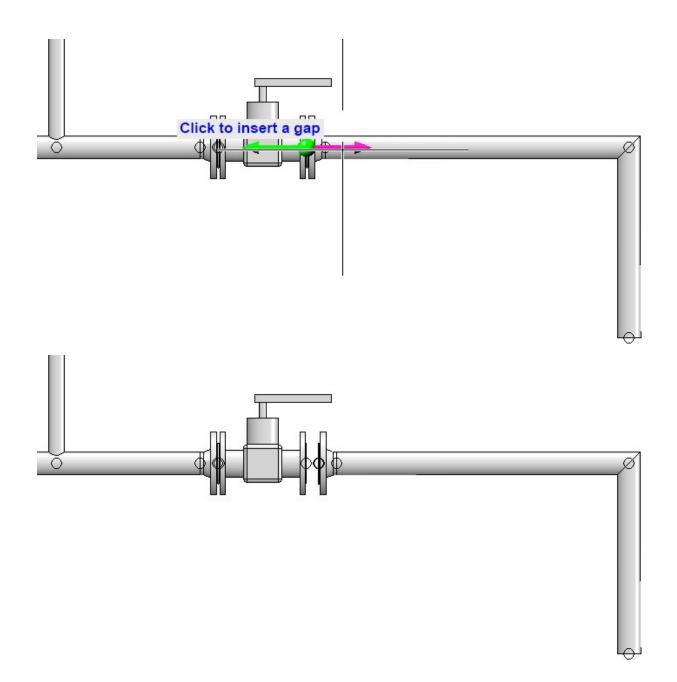
Consider weld seam gaps is deactivated

☐ Consider weld seam gap

If the Consider weld seam gaps option is deactivated in the Plant Engineering settings, then you still remove existing gaps. Instead of inserting a gap, however, you are offered to create a space in the pipeline. You are no longer limited to welded joints. You define the width of the space by entering a value in the calculator.

Click to insert a gap
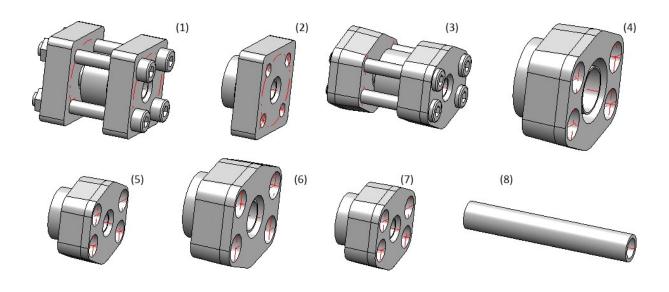
> Please note that such a space cannot be closed again with this function, as it interrupts the pipeline connection and - unlike a weld seam gap - does not establish a relationship between the parts of the pipeline. To close the space, use the function **Close spaces**. Or use the function **Change route** to drag the now free end point to another position.

VOSS-ZAKO flared flanges

The part inventory for Plant Engineering has been extended by ZAKO flared flanges from the manufacturer VOSS. The following variants have been added:

1	Square pipe flange connection - high-pressure series	Z-TTO_Q-FLANSCHVERBINDUNG_HD.VAA
2	Square flange connection - high pressure series	Z-TPO_Q-NUTFLANSCH_HD.VAA
3	Pipe-flange connection SAE - standard series	Z-TTO_SAE-FLANSCHVERBINDUNG_HD.VAA
4	Connecting flange connection SAE - high pressure series with reducer	Z-TPO_SAE-NUTFLANSCH-HD-RED.VAA
5	Connecting flange connection SAE - high pressure series	Z-TPO_SAE-NUTFLANSCH-HD.VAA
6	Connecting flange connection SAE - standard series with reducer	Z-TPO_SAE-NUTFLANSCH-RED.VAA
7	Connecting flange connection SAE - standard series	Z-TPO_SAE-NUTFLANSCH.VAA
8	Straight pipe according to EN10305, the sub-types of which are matched to the new VOSS-ZAKO variants.	Z-TPO_PIPE_10305.VAA

The nominal diameters in the manufacturer's specifications have been adapted to the usual nominal diameters.

Welding necks and handling of loose flanges

New part type and variants

In pipeline planning there is the situation that loose flanges can be inserted once as flanges and once as fasteners. This leads to problems, especially in interaction with HELiOS, because in the worst case two articles have to be created for one part, one as a flange and one as a fastener.

Therefore, existing loose flanges N2642_LF_R1.VAA and N2642_LF_R2.VAA classified as fasteners are additionally offered classified as flanges as of SP1.

Series	Variant *	Standard designation
1	N2642_LFC_R1.VAA	N2642 R1
2	N2642_LFC_R2.VAA	N2642 R2

These loose flanges were previously used as fasteners for welding necks, which were previously designed as flanges.

As of HiCAD 2023 SP1, there are now welding necks as a new part type. On the one hand, these are designed for interaction with the regular loose flanges. On the other hand, they support functionalities that are otherwise found in flanges. One example is the automatic setting at free ends and the automatic counterflanging.

The part families **KM10357_WN_DIN.VAA** and **ROFI10357_WN_ISO.VAA** (previously classified as flanges) are available classified as welding necks:

Variant *	Standard designation	
KM10357_WNC_DIN.VAA	Kieselmann 10357	
ROFI10357_ WNC_ ISO.VAA	RO-FI 10357	contains all sub-types of the modelled welding neck
ROFI10357_ WNC_ ISO_ R1.VAA	RO- FI 10357 R1	each contains a share of the sub-types from ROFI10357_ WNC_ISO.VAA
ROFI10357_ WNC_ ISO_ R2.VAA	RO- FI 10357 R2	

* The C in the file name of the adapted variants stands for collar (as short form of weld neck collar). The C in the file name of the adapted parts therefore has an additional **C** in the name (for collar as short form of weld neck collar).
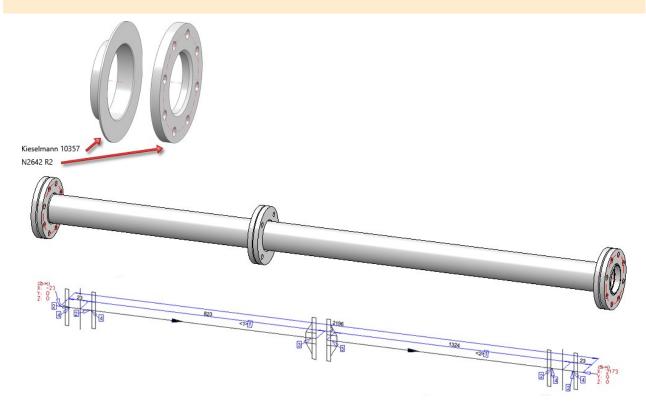
The loose flanges of Series 1 fit the variant ROFI10357_WNC_ISO_R1.VAA. The loose flanges of Series 2 fit the variants ROFI10357_WNC_ISO_R2.VAA and KM10357_WNC_DIN.VAA.

Connection type attribute

Welding necks can provide loose flanges for their connection type. As there is no part type loose flange, it was defined for welding necks that a loose flange is meant if the part type ID for flanges is used in the connection type, i.e. 5100010. If welding necks are to be inserted with the loose flange from the connection attributes, the **AutoFit connecting parts** option must be activated on the **Part selection** tab of the **Plant Engineering Settings** dialogue window.

> To model your own welding neck, please read the information in the topics Part Type: Welding Neck or Variant for Part Type: Welding Neck.

Standards in the connection type attribute
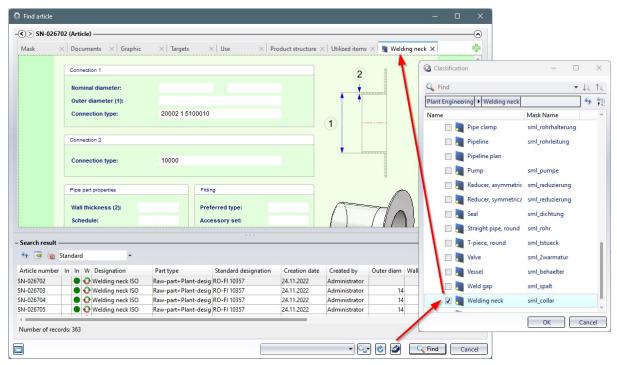
As of HiCAD 2023 SP1, a part type ID can be used in the connection type definition without additionally specifying the standard. For welding neck flanges, you can therefore specify that you want to automatically install a loose flange without specifying the standard. If the loose flange is not unique, you will be offered a corresponding search mask (catalogue or HELiOS). However, if you use HELiOS, you must first update HELiOS for Plant Engineering as described in the next point. If you use the catalogue for part selection, you may have to use the tool **vartocat.exe** in the HiCAD exe directory to be able to use the new part type.

New part type "Welding neck"

In HiCAD 2023 SP1, a new part type for Plant Engineering and thus a new article class is available in the form of the **Welding neck**.

If you use HELiOS, you must inform HELiOS of the new classification for welding neck flanges. This is done via the programme **DBPlantDataImport.exe**, which you will find in the HiCAD exe directory. There you click on the button **Update HELiOS for Plant Engineering**. Afterwards you can use the HELiOS search mask in your HELiOS Desktop.

Function adjustments

Due to the new part type **Welding neck**, the functions listed below have been adapted accordingly.

- **Part insertion**
  The new Part insertion function offers for welding neck also the options **AutoFlange** and **In all similar places**.

- **Pipe parts V26** and **Part exchange**
  In the dialogues of the old Part insertion and Part exchange functions the welding neck can now also be found.

- **AutoPlace parts on guidelines**
  AutoPlacing of parts on guidelines supports welding necks with the new option **Welding necks on free ends**.

If this option is activated, then welding necks are placed on all free guideline ends, as for example with the option **Flanges on free ends**. If the option **Counterflanges** has been selected, welding necks that already exist individually will be counterplaced. If the **AutoFlange...** option is also activated on the **Part selection** tab of the Plant Engineering Settings dialogue window, welding necks are also automatically placed against the newly placed ones.

Loose flanges in the connection type attributes of straight pipes

As with welding necks, you can also define loose flanges in the connection type attributes for straight pipes. This is especially helpful for pipes with (one-sided) welding necks. This allows you to explicitly define the pipe ends where the loose flanges can be installed.

By eliminating the need to assign a standard in the connection type attribute, you also do not have to specify in the VAA file which loose flange you want to insert.
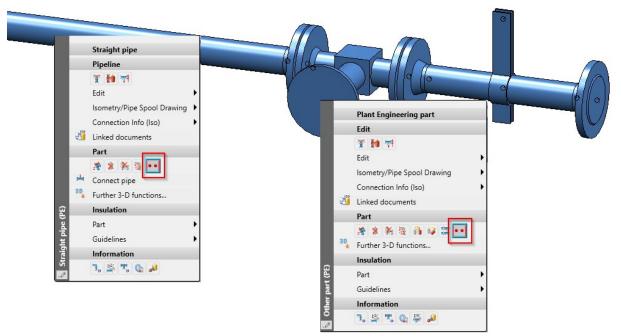
For this purpose the following flared pipe have been adjusted accordingly:

- EN10357_PIPE_BEADED_BOTH_PN10_FME4.VAA

- EN10357_PIPE_BEADED_BOTH_PN10_FME8.VAA

- EN10357_PIPE_BEADED_SINGLE_PN10_FME4.VAA

- EN10357_PIPE_BEADED_SINGLE_PN10_FME8.VAA

Align component connection tangentially, via context menu

The function Align connection tangentially was previously only available in the context menu of a component connection. These are sometimes difficult to identify, especially in drawings with a lot of parts..

To facilitate the use of this function, from HiCAD 2023 SP1 the function is also available in the context menu for pipe parts.
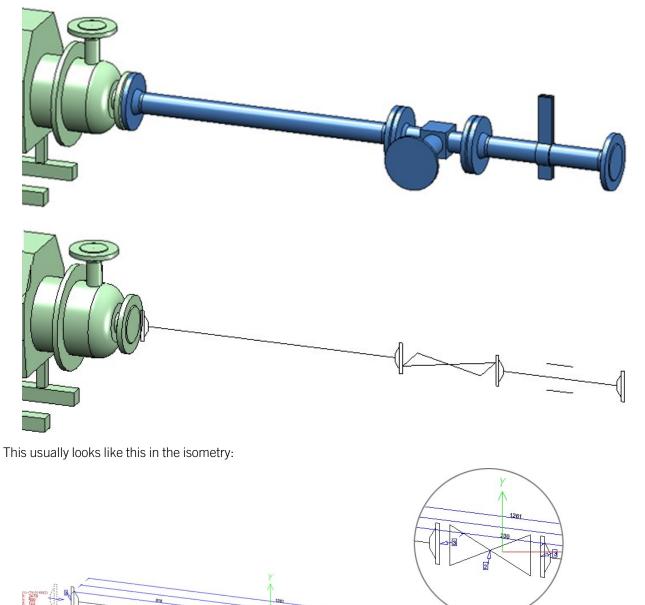
In this way, the selection is possible even if the component connection is hidden by other parts.

Isometry and Pipe spool drawing

Switch off symbol alignment

When creating an isometry or pipe spool drawing, HiCAD automatically aligns the symbolic representations of the parts.

Here is the example of the symbol of a valve installed between two flanges:

This usually looks like this in the isometry:
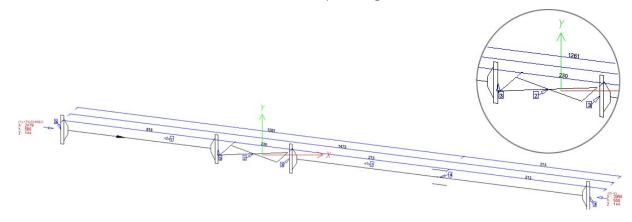
Until now, there was no way to bypass this automatism. As of HiCAD 2023 SP1, it is possible to disable this automatism in the Configuration Editor at **Plant Engineering > Isometry and Pipe Spool Drawing > Align symbolic representations**.

If the checkbox is deactivated, the result in the above example changes as follows:

As a rule, you should leave the automatic symbol alignment switched on (ISD default setting).Exceptions can arise in individual cases, for example, if the symbol of a valve is to reflect the angle of its installation. In this case, you must manually adjust the symbol alignment in the layout plan. Please note that symbols are preferably rotated in groups in layout plan as well. So you may have to temporarily lock parts against editing, as shown in the following image for the two flanges in the ICN.

Connection item numbers at the end of/within pipelines

For connection item numbers it is now possible to determine whether they are to be assigned at pipeline ends or inside a pipeline. For this purpose the tab **Connections** in the isometry and pipe spool drawing settings has been extended..

> **Please note:**
> HiCAD tries to keep item numbers once they have been assigned. If you change one of these options, it is usually necessary to call the function **Renew connection item numbers** (Plant Engineering > Isometry / Pipe Spool Drawing > Spool) to get the desired result.

Update north arrow/crosshairs

From SP1 onwards, it is possible to update the alignment of the north arrow and crosshairs on the sheet with the function **Update all**. The coordinate system of the active view is used for alignment. This is especially important in pipe spool drawing, where there is often more than one view.
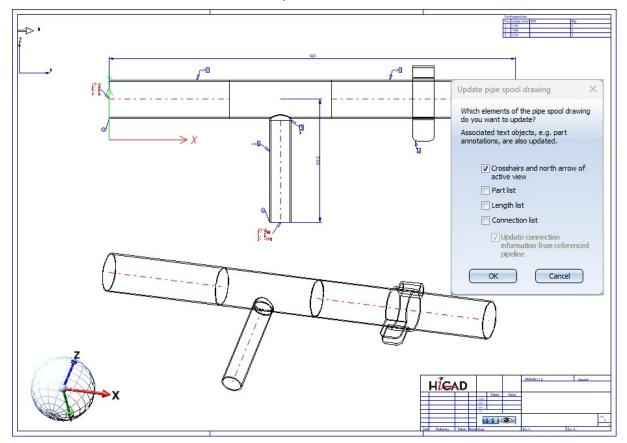
The dialogues have been adapted accordingly:
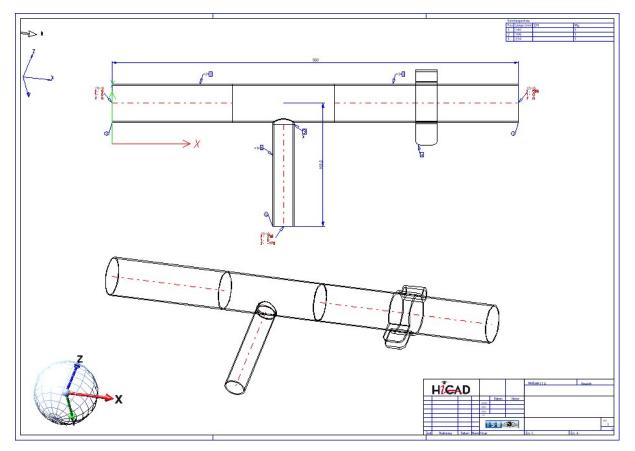
For the pipe spool drawing, crosshairs and north arrow have been added to the dialogue of the function **Add/Re-move pipe spool drawing elements**:

In the following example the alignment has been changed manually in the lower view of the pipe plan. To reflect this in the crosshairs and the north arrow, the function **Update** is used.

Since only one crosshairs and one north arrow are created, if there are several views you may have to adjust both figures on the sheet using the function **2-D Part > Transform > Move** ▾ **> Move part, free**.

Major Release 2023 (V 2800)

New parts - Y-pipes

The parts inventory for Plant Engineering has been expanded by two variants for the part type **Y-pipe**:

- **Y_PIPE_WELDED_60.VAA** with an opening angle of 60° and
- **Y_PIPE_WELDED_90.VAA** with an opening angle of 90°.

The geometry of these variants - in terms of outer diameter, wall thickness and the length of the "legs" - is based on the Y-piece YS.S from the company HECO®.

ROHR2 interface

Loose flanges

The program system ROHR2 does not allow overlapping pipe parts since version 33. Furthermore, parts cannot be transferred in such a way that they are only assigned to a connection point for representation. For this reason HiCAD 2023 does not transfer loose flanges to ROHR2 any more.

Naming of NTR files

When creating NTR files via the ROHR2 interface ![R2 icon], the part name of the pipeline was used as file name until now. From HiCAD 2023 onwards the name can be configured flexibly via HDB files.

For this purpose the following parameters are available in the Configuration Editor at **Plant Engineering > ROHR2**.

- Configuration of NTR file name (without HELiOS) and
- Configuration of NTR file name (with HELiOS)

The files **ICN3D_DESIGNATION** and **ICN3D_ DESIGNATION_DB** are used as ISD default settings. These are the HDB files which are also used for the display of the column **Designation** of the ICN window 3D Part Structure. This means that the part number of the pipeline is used as the file name by default or - if this is not available - the part name.

If you want to use a different file name, you can adapt the files accordingly or use your own files. For more information, see the topic HDX and HDB Files.

> **When changing the ICN3D_DESIGNATION and ICN3D_DESIGNATION_DB files, please note that these changes also affect the ICN display!**
>
> **Alternatively, you can create your own HDB files and set them in the Configuration Editor. The files must have the extension .HDB and be located in the HiCAD sys directory.**

Example:

You want to use Designation 1 (Attribute $01) or Designation 2 (Attribute $02) of the pipeline as the names of the NTR files, but do not want to change the ICN display. Then you could create copies of the above files, e.g. ICN_NTR_PLANT and ICN_NTR_PLANT_DB, adapt them accordingly and then set them in the Configuration Editor:

ICN_NTR_PLANT.HDB	ICN_NTR_PLANT_DB.HDB
HDB # Configuration file for the output of any number of # HiCAD part attributes instead of the part name in the 3-D browser. # TEXT - Commentary (will not be displayed) # ATTR - Attribute to be displayed # TYP - STRING(default), DOUBLE # NKS - Decimal places # <H> - HiCAD part attribute # <E> - Features # _BEGIN_OR_ _END_OR_ - "ODER" link _BEGIN_OR_ <H>::TEXT=""::ATTR="$01"::TYP="WCHAR" <H>::TEXT=""::ATTR="$02"::TYP="WCHAR" <E>::TEXT-T="Teilename"::ATTR="NAME"::TYP="STRING" _END_OR_	HDB # Configuration file for the output of any number of # HiCAD part attributes instead of the part name in the 3-D browser. # TEXT - Commentary (will not be displayed) # ATTR - Attribute to be displayed # TYP - STRING(default), DOUBLE # NKS - Decimal places # <H> - HiCAD part attribute # <E> - Features # _BEGIN_OR_ _END_OR_ - "ODER" link _BEGIN_OR_ <T>::TEXT-T="Normbezeichnung"::ATTR="NORMBEZEICHNUNG" <T>::TEXT="Sachnummer"::ATTR="HEL_SACHNUMMER" <D>::TEXT="Dokumentnr."::ATTR="HEL_DOKUNUMMER" <H>::TEXT=""::ATTR="$01"::TYP="WCHAR" <H>::TEXT=""::ATTR="$02"::TYP="WCHAR" <E>::TEXT-T="Teilename"::ATTR="NAME"::TYP="STRING" _END_OR_

Load meta data

In the **Part attributes** dialogue window of a pipeline you can specify the

- Design pressure,
- Working pressure,
- Design temperature,
- Operating temperature and
- Density of medium

These attributes will be transferred during export via the pipeline interface.

The catalogue **System settings > System attributes** has been extended accordingly for this purpose.

New Part insertion function

Straight pipes
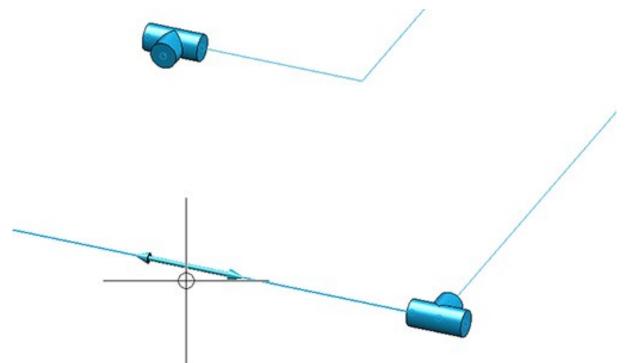
When inserting straight pipes with the **Pipe part** function , from HiCAD 2023 only the option **Place pipeline on edge** is available in the context menu. This function has been extended and now places bent pipelines on bent guidelines:
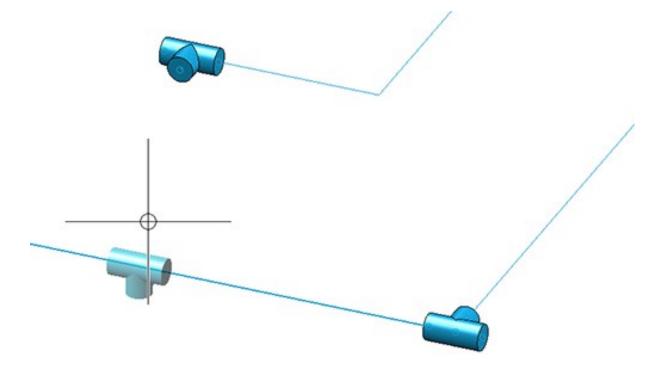
Freely placeable parts

From HiCAD 2023 onwards, for parts that can be placed freely (e.g. T-pieces or branches), selecting an edge does not directly lead to the part being dropped. When you move the cursor near an edge, a preview is first displayed as follows:
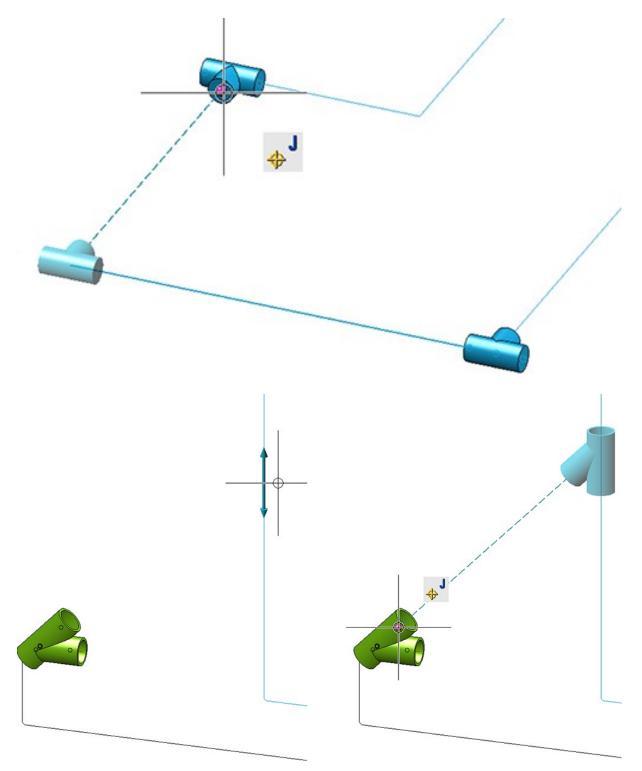
Now you have two options:

1. You click on the edge. Then a preview of the part is displayed as usual.

This part can now be flexibly aligned along the edges, for example at other connections. Only when the placement or connection point is selected does the insertion take place.

Alignment to other connections is also indicated graphically. If a guideline edge is selected and then another connection, the part will be aligned to that connection. This means that if the part can be rotated around the guideline edge so that the connection of the part and the selected connection are aligned, this rotation is suggested to you. A dashed line shows you that the connections are aligned with each other.

Otherwise, the next point on the guideline edge is used for placement.

2. If the part is to be attached directly to another connection, select the desired connection or point <u>without clicking the guideline edge first</u>. The part will then be placed directly.

Immediate insertion in all similar places
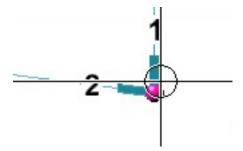
If the checkbox **Insertion in all suitable places** is active in the dialogue on the **General** tab, then it is now also possible to specify whether the part is to be inserted automatically in all suitable places. A corresponding checkbox has been added to the dialogue.

This is possible for the following part types:

- Straight pipe,
- Elbow,
- T-piece,
- Knee,
- Branch and
- Flange.
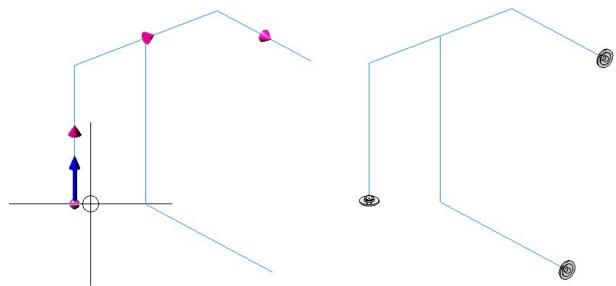
First select an appropriate position, e.g. for an elbow.
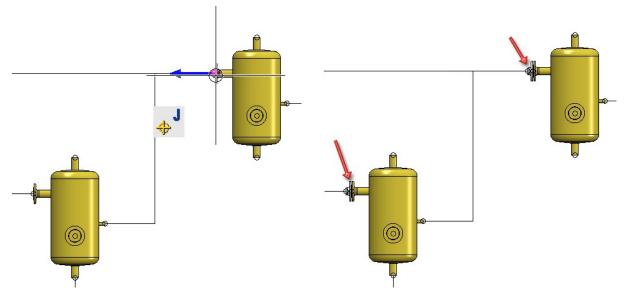
The subsequently selected part is then installed at all suitable points (also applies to knees, T-piece and branches). For straight pipes, it is sufficient to select a point on the desired pipeline.

Flanges are a special case. Here there are two possible applications.

1. The insertion position is the free end of a guideline.
   Then the selected flange is placed at all free ends of the pipeline.
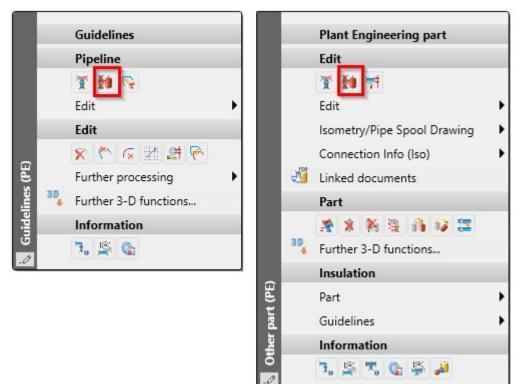
2. The insertion position is the connecting point of a flange.
   Then the same flange is installed as a counterflange at all points of the pipeline that have the same insertion situation.

Changed context menus
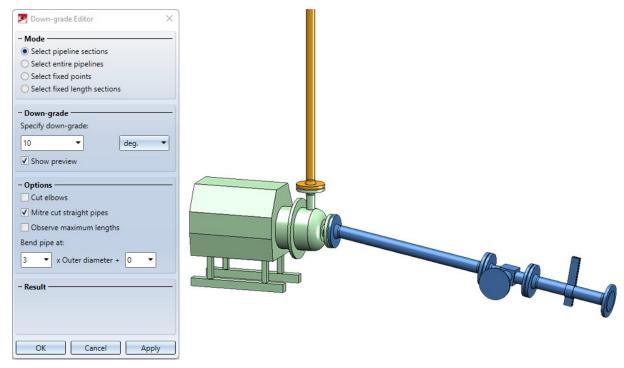
The function **Pipe parts** is now also available in the context menus for guidelines and Plant Engineering parts.
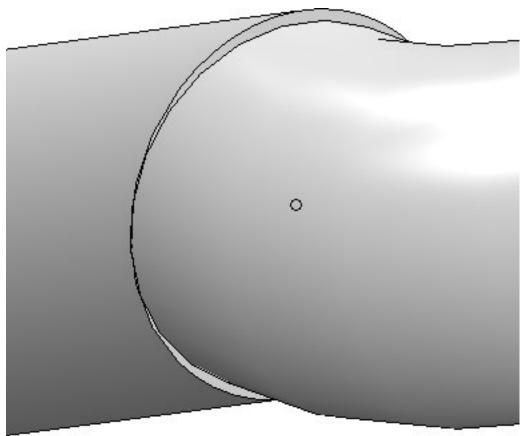
Down-grade Editor - Mitre cut straight pipes

The **Down-grade Editor** has been extended by the option **Mitre cut straight pipes**. If this option is selected, straight, divisible pipes are mitred in order to place them in the down-grade.

Note that the angle for the down-grade should not be too large in this case, as the mitre necessarily causes the pipe cross-section to deviate from the circular shape and thus no longer matches exactly the connection of the following pipe.

The angle for the mitre cut should not be too large, as otherwise pipe protrusions may result from the cut.

To place a pipe section in the down-grade, the Down-grade Editor now proceeds as follows:

1. For pipes that already have a mitre (e.g. because it was set by the **Edit arcs** function), the mitre angle is changed.

2. If the option **Cut elbows** is active, the angles of elbows are adjusted.

3. If the option **Mitre cut straight pipes** is active, straight pipes are mitred.

4. If none of the options apply, straight pipes are bent.

Copy, paste, mirror and clone pipelines

It is no longer absolutely necessary to copy pipelines from the pipeline planning via the corresponding function. The function takes care to keep the part name of the pipeline unique because the generation of isometry and pipe spool drawing require this.

- To copy and paste pipelines, you can also use the corresponding clipboard functions in the ICN:
  - Copy to HiCAD clipboard
  - Paste from HiCAD clipboard

When pasting from the clipboard, HiCAD takes care not to insert a pipeline below another pipeline.
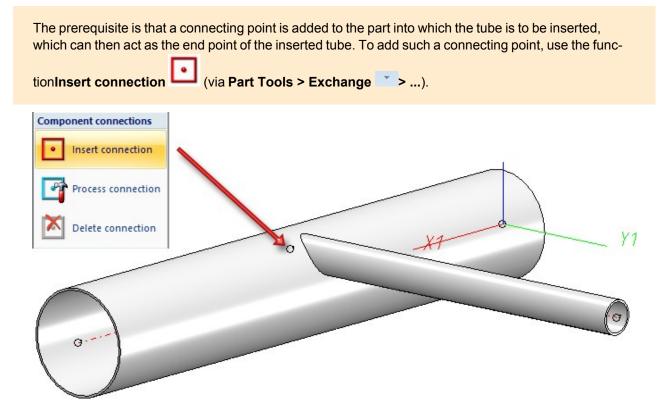
- Pipelines can also be mirrored and cloned using the functions on the **3-D Standard** Ribbon.

Eccentriccally inserted pipes
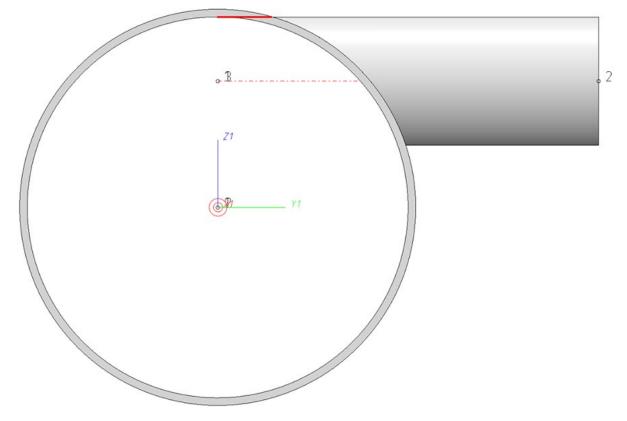
Up to now, straight pipes could only be inserted into other parts in such a way that the end point of the inserted pipe ends on the centre line of the other part. As of HiCAD 2023, this restriction no longer exists.

> The prerequisite is that a connecting point is added to the part into which the tube is to be inserted, which can then act as the end point of the inserted tube. To add such a connecting point, use the function **Insert connection** (via **Part Tools > Exchange** ▾ **> ...**).

The insertion can then be carried out with the function **Plant Engineering > New > PipePrt** ⏷ **> Pipe parts v26** or **Plant Engineering > Guideline Tools > AutoPlace parts on guidelines** .

Common use cases are **top-of-pipe** or **bottom-of-pipe** insertion. The figure above shows the **top-of-pipe** insertion.

In both cases, the inserted pipe leads tangentially out of the thicker pipe, with the pipe placed as close to the edge of the thicker pipe as its wall thickness allows. In the following figure, this is indicated by the red line:
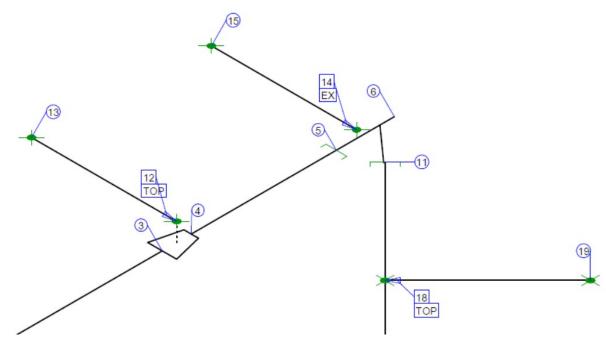
When setting the connection point freely, it is rather cumbersome to enter this point correctly manually. For this purpose, the outer diameter and wall thickness of the thicker pipe and the outer diameter of the pipe to be inserted must be calculated with each other. For this purpose, the **Align connection tangentially** function is available in the context menu for connections:

Extensive information and an example can be found ion the topic Eccentrically Inserted Pipes.

In addition, the display of connections in the isometry and in the pipe spool drawing has been extended. The additional texts TOP, BOP and EX are now displayed at the connection numbers of inserted eccentric pipes. This is the default setting for connections which are inserted:

- **top-of-pipe** (TOP),

- **bottom-of-pipe** (BOP) or

- **otherwise eccentric (EX)**.
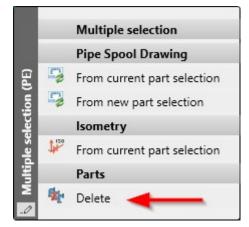
You can customize this representation by modifying the corresponding **FTD files**:

- ISD_inserted_top.ftd,

- ISD_inserted_bop.ftd and

- ISD_inserted_ex.ftd.

Especially if you want to have the concrete coordinates displayed at such points, these three text objects have to be adapted.

Delete multiple selection of parts

If several parts are selected in the drawing, then these parts can now also be deleted via the context menu of the multiple selection.

As described above, a distinction is made between parts that have been placed on a guideline and parts that have been placed in the drawing without a guideline.
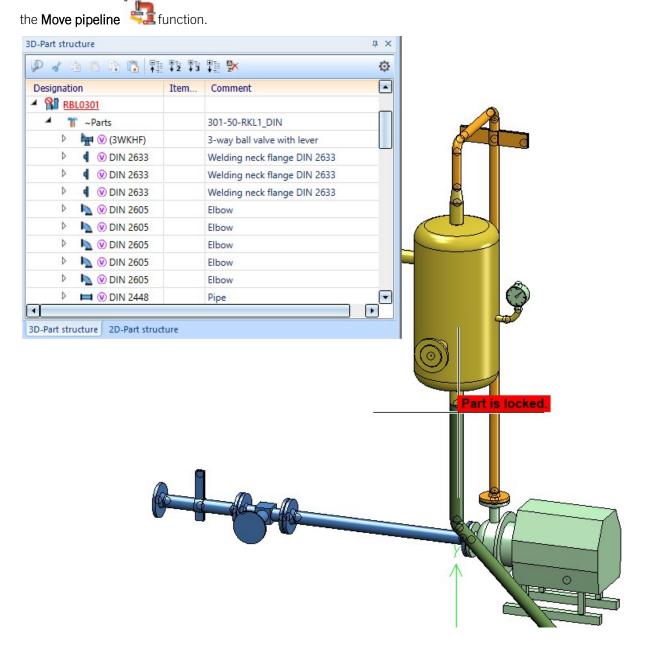
Alternatively, you can also use the DEL key to delete parts in the ICN.

Lock parts

HiCAD offers the possibility to lock parts for other users; for example to protect it from accidental changing of parts.

These parts are marked with a corresponding symbol 🚫 in the ICN.

For some Plant Engineering functions, a note is displayed in such cases to indicate the lock. The form of the message depends on the respective function.
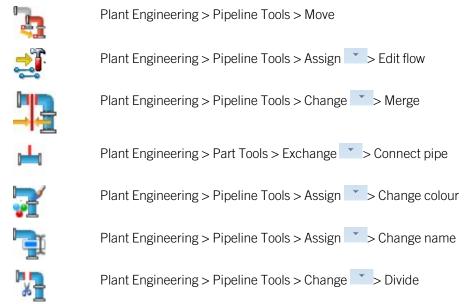
- In some cases, the message is displayed directly during the selection of a pipeline, for example, in the case of

  the **Move pipeline** function.
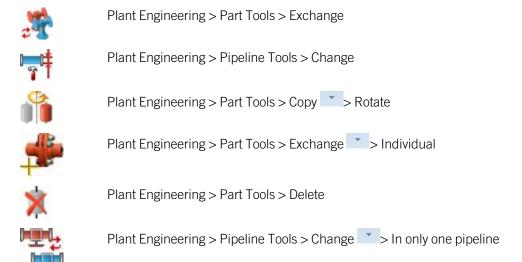
- In other cases, a corresponding message window is displayed, e.g.:

If a pipeline is locked, the following functions display a corresponding hint directly during the selection (some only if they are not called via context menu):

	Plant Engineering > Pipeline Tools > Move
	Plant Engineering > Pipeline Tools > Assign ▼ > Edit flow
	Plant Engineering > Pipeline Tools > Change ▼ > Merge
	Plant Engineering > Part Tools > Exchange ▼ > Connect pipe
	Plant Engineering > Pipeline Tools > Assign ▼ > Change colour
	Plant Engineering > Pipeline Tools > Assign ▼ > Change name
	Plant Engineering > Pipeline Tools > Change ▼ > Divide

The message window appears when choosing the following functions:

	Plant Engineering > Part Tools > Exchange
	Plant Engineering > Pipeline Tools > Change
	Plant Engineering > Part Tools > Copy ▼ > Rotate
	Plant Engineering > Part Tools > Exchange ▼ > Individual
	Plant Engineering > Part Tools > Delete
	Plant Engineering > Pipeline Tools > Change ▼ > In only one pipeline

The function Plant Engineering > Pipeline Tools > Change ⌄ > In all pipelines ⚙️🖥️ handles locks by applying the desired change to all non-locked pipelines. The locked, i.e. not modified, pipelines are named in a message:

Bills of Materials (new ISD Report Manager)

Excel-BOM with pipe length list

For the new ISD Report Manager (from HiCAD 2023), an XML configuration file adapted for Plant Engineering is available with the name **HiCAD_Anlagenbau.rm_settings** . This configuration file allows the sensible creation of bills of materials (BOMs) of mixed model drawings.
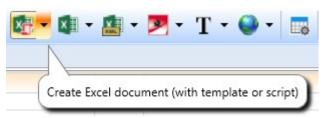
The file is located in the HiCAD sys directory and facilitates in particular the use of a new BOM template for Excel export. This new template HiCAD_Anlagenbau.2800.0.xlsx can also be found in the sys directory of your installation (or as **HiCAD_Anlagenbau.EN.2800.0.xlsx** in the sub-folder **bomtemplates**).

Proceed as follows:

- To create a BOM, choose the function **Drawing > Itemization/Detailing > BOM**.

- In the dialogue window, select **HiCAD_Anlagenbau** for **Configuration** in the **XML Format** section.

- When the BOM is displayed in the ISD Report Manager, select the **Create Excel document (with template or script** function there.
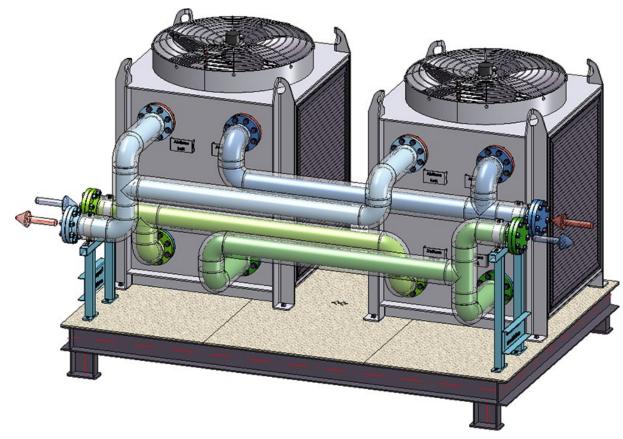
- After selecting a file name, Excel will open.

In the opened file you will now find the following BOMs (as Excel tables):

Table	Content
Structure list	Displays all individual parts and assemblies of a product on the basis of the HiCAD part structure, i.e. a summary of all production stages with logical relationships
Quantity list	Displays the quantity per item, i.e. without further summary
Quantity sum list	Shows the quantity per article, where length, area and weight are summed up
Profile totals list	A grouped layout of the quantity list for articles with length
Components list	A grouped layout of the quantity list without even pipes or articles without length
Bar list	Similar to the profile totals list, but grouped by delivery length with remainder
Bar - Summary	Basically the table headers from the bar list, so a summary of the bar list

Example:

280X_PE_EXP

FILE　HOME　INSERT　PAGE LAYOUT　FORMULAS　DATA　REVIEW　VIEW　ACROBAT

F81

	A	B	C	D	E	F	G
29	SN-025052, 100, 3.6,				4 x 6000 mm	Waste: 4732 mm (19.72 %)	
30	Item	Number	Length (mm)	Cut (Web)	Cut (Flange)	Designation	
31							
32				16　16　18　18	0　0　0　0　0	0　0　0　0　0　0	0 (372)　0 (372)　0 (372)　0 (372)
33	13	4	26			Pipe	
34	23	2	39			Pipe	
35		6	42			Pipe	
36	15	2	69			Pipe	
37	12	2	72			Pipe	
38		4	80			Pipe	
39	19	2	89			Pipe	
40	16	2	186			Pipe	
41	18	2	189			Pipe	
42		12	192			Pipe	
43		4	372			Pipe	
44	17 (376)　17 (376)　17 (376)　17 (376)		0 (392)　0 (392)　0 (392)　0 (392)		0 (392)　0 (392)　0 (392)　0 (392)	0 (729)　553	
45	17	4	376			Pipe	
46		8	392			Pipe	
47		1	729			Pipe	
48	0 (729)　0 (729)		0 (729)		14 (1489)	14 (1489)　793	
49		3	729			Pipe	
50	14	2	1489			Pipe	
51	14 (1489)			14 (1489)		2993	
52	14	2	1489			Pipe	
53							
54	U 160, NaN, NaN, 1.0038				1 x 6000 mm	Waste: 2530 mm (42.17 %)	
55	Item	Number	Length (mm)	Cut (Web)	Cut (Flange)	Designation	
56							
57	0 (2870)				0　0　0　0	2488	
58		1	2870				
59		4	150				
60							
61	TN-06815, 100, 25,				-4 x Fixed length		
62	Item	Number	Length (mm)	Cut (Web)	Cut (Flange)	Designation	
63			28			Pipe trace heating with insulation	
64		-2	41			Pipe trace heating with insulation	
65		-2	44			Pipe trace heating with insulation	
66		-2	71			Pipe trace heating with insulation	
67		-2	74			Pipe trace heating with insulation	
68		-2	91			Pipe trace heating with insulation	
69			106			Pipe trace heating with insulation	
70		10	162			Pipe trace heating with insulation	
71		-2	188			Pipe trace heating with insulation	
72		-2	191			Pipe trace heating with insulation	
73			212			Pipe trace heating with insulation	
74			378			Pipe trace heating with insulation	
75			1491			Pipe trace heating with insulation	

◄　►　...　Quantity list　Quantity sum list　Profile totals list　Bolt|screw list　Components list　**Bar list**　Bar - Summary　⊕

READY

BOM-creation with Plant Engineering functions

Adapted configuration files for the new ISD Report Manager are also available for the Plant Engineering functions at **Plant Engineering > Evaluation**:

- Entire drawing: **anlagenbau_szn.rms_settings**
- Active pipeline: **anlagenbau_rl.rms_settings**
- Total length list: **anlagenbau_geslen.rms_settings**

These are preset in the Plant Engineering Settings, on the **Bills of Materials** tab.

Furthermore, as of HiCAD 2023, only two HDB files control which attributes are transmitted to the ISD Report Manager:

- rm_anl_exportszn.HDB (für Konstruktionen) und
- rm_anl_exportpart.HDB (für Teile)

These contain HiCAD part attributes only. If you need additional attributes of the HELiOS PDM system, the prepared files

- rm_anl_exportszn_db.HDB und
- rm_anl_exportpart_db.HDB

serve as templates. You will find the files in the HiCAD sys directory. In the simplest case, it is sufficient to rename them accordingly:

- rm_anl_exportszn_db.HDB ➡ rm_anl_exportszn.HDB
- rm_anl_exportpart_db.HDB ➡ rm_anl_exportpart.HDB
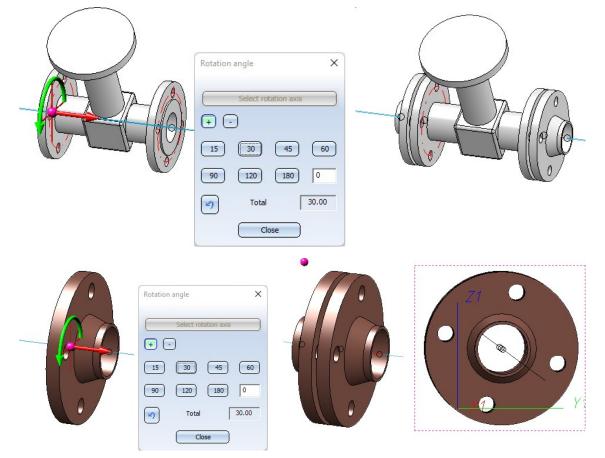
> **Total length attribute**
>
> Prior to HiCAD 2023, when creating the total length list using the **Total length list for straight pipes** func-tion, the total length was assigned to the **Quantity1** attribute (§11). This attribute did not take into account any length unit..
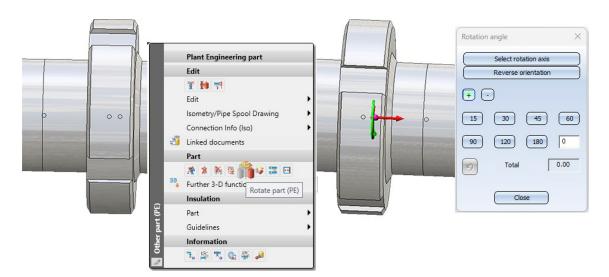>
> As of HiCAD 2023, when creating the total length list with an rm_settings configuration file of the ISD Report Manager, the total length will be assigned to the attribute **Total length** (§22). Thus, the total length is also available in the ISD Report Manager as a length with a suitable unit..

Rotate flange connections

When installing parts with sub-part flanges, e.g. valves, if you rotate the part after insertion, these flanges will now be rotated as well. The same applies if the **AutoFlange** option is activated.

This functionality is only offered by the new **Part insertion** function, via the old one (V26) only the valve or similar is rotated as before. If the parts are to be rotated together, either the new **Part insertion** function is to be used or one places the parts first without rotation and then uses the function **Rotate part**.

HiCAD - HELiOS Head ID

As of HiCAD 2023, it is no longer possible to specify the Head IDs of articles and documents in HELiOS. This ID was previously used by the catalogue editor and the DBPlantDataImport/PartDataAutoSync tools to find articles again.

Now, unique IDs are assigned by the variant editor itself and transferred to HELiOS during matching. These IDs are directly visible in HELiOS, as they are not a regular article attribute.

The new IDs are only directly visible in a few places in HiCAD:

- When matching with the catalogue via the tool VarToCat, an additional table column PLANTID is inserted.
- In P+ID, the new ID becomes visible in the attribute **Database article ID** when assigning article data.
- After a part has been installed, the PLANTID is also found in its part attributes.

What's new?

Notes on HELiOS Updates

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

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

- **Data backup before updating**

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

- **Log file for update**

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

- **New mask format**

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

- **MultiCAD interfaces**

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

HELiOS Desktop

Service Pack 2 2023 (V 2802)

Utilized articles

In the context of the document detail window you will find the new result list tab **Utilized articles**.

This shows you the article contexts stored in the model structure for the document.

In contrast to the model structure, this result list can be sorted and filtered more easily and articles can also be displayed consolidated, i.e. they only appear once in the result list, as is not possible in the context of the model structure due to its model tree and instance approach.

The articles listed there may deviate from the bill of materials (product structure) if, for example, you are working with assemblies that are not BOM-relevant.

For use cases such as the release of an assembly structure, the display of used articles based on the model structure, in the results list of the document context can therefore be very helpful.
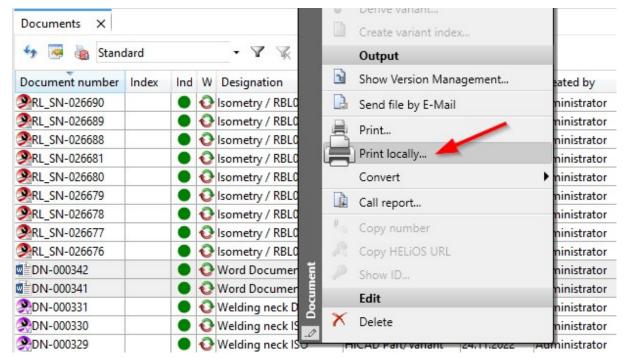
Print locally

In the context menu of documents or in the Output tab of the document detail window you will find the new function

Print locally.

With this function you can print documents such as Office or PDF files from the HELiOS Desktop via the Windows standard printer.

The function can also be used for a multiple selection of documents. In this way, you can print documents from a result list in the present order without having to select them individually for printing via the HELiOS Spooler.

Project and folder change behaviour in the Combined search

In older HELiOS versions, when working with the **Combined search**, you could set the behaviour of how selected projects or folders are handled when the selection is changed: Should the change of a project or folder in a search mask also affect the other masks of the Combined search, should projects or folders be set individually on each mask, or should inactive search masks be set independently of projects or folders?

Old procedure: Combined search > Project change behaviour

With the update to HELiOS 2023 Service Pack 2, you will find the settings for this at **HELiOS Options > General**, under **General Settings > Find**.

This has the advantage, among other things, that this option can then also be exported / imported and made available as uniform behaviour on other computers.

Edit workflow status: Improved dialogue

In the **Edit workflow status** dialogue window you can either make changes by clicking **OK**, which will close the window, or by clicking **Apply**, which will keep the dialogue window open, so that you can also set the next Workflow step directly, if desired.

Attribute mappings for document input

The attribute mappings for file import, which you can configureat **HELiOS Options > General > HELiOS Desktop**, also apply to document input via **Input > Document** and for template files, and can be selected accordingly in a pull-down menu of the Enter document dialogue.

Import

The re-import of files via **Input > Serial input > Import** has been adapted to reflect possible error sources more transparently:

If files contained in the selected XML file are not found in HELiOS during the import, you will receive a corresponding error message before the remaining existing files are offered for re-import.

If the target document is no longer contained in the database, you will also receive a corresponding error message when attempting a re-import:

Central configuration for date fields and time zone

In addition to the possibility to assign an **Own date/time format** or the **Database time format** to result list areas, the possibility to further influence the default formatting of date values in HELiOS masks has been created.

For this purpose, you will find in the **HELiOS Options** under **General > Units and formatting** the new configuration options for **Standard formatting of date with time, if applicable** and **Standard formatting of date without time**:

The selection lists are preset with a number of frequently used formattings, which you can select accordingly. A preview of the selected formatting is displayed in the field to the right of the selection/input field.

You can also make your own entries for the date/time formatting in the field, as long as you use a correct syntax in line with the date and time setting in result lists.

Changing the formatting will then have an effect in two places:

- The display of date fields in masks
- The display of date/time fields in result lists where the **Date/time display** is set to **Standard**.

The new setting **Standard** is also the new default for date/time fields in result lists, which applies to new installations of HELiOS.

The desired **Time zone** (in which date/time display takes place) can be selected here via a combined list if required.

The default time zone is the local time set in your operating system.

If the set time zone supports a distinction between winter and summer time, this is automatically taken into account. (I.e. in Germany UTC+1 (winter time) would automatically change to UTC+2 (summer time) on the corresponding date).

Performance

Especially in the area of the product structure tree and when generating HDE reports using many multilingual attributes, a significant performance increase of the HELiOS Database could be achieved with Service Pack 2.

Service Pack 1 2023 (V 2801)

Optimised search result lists

In order to increase performance, only objects for which the logged-in user has corresponding read rights are displayed in result lists of HELiOS objects (articles, documents, projects, folders, pipe classes and user-defined objects).

The sorting of result lists has been accelerated many times over due to a smaller number of internal queries, which also improves the creation of result lists in particular, in which case the database has a somewhat higher latency.

Earlier database options, with which the display of objects without access rights could be set, are thus no longer applicable.

Errors during printing or export

Errors that prevent documents from being printed or exported are displayed in the corresponding dialogue below the document result list when you click **OK**.

Double-clicking on an error line also marks the corresponding document in the result list above.

Example: If the dialogue returns the error description "File for document is not in the Vault", it is recommended to load the document once into the workspace.

Context menu entry for saving to HELiOS

After an installation of HELiOS, the Windows context menu has been extended by default by the menu item **Save to HELiOS**.

This makes it possible to save documents in the HELiOS database in connection with **HELiOSScan** in this simple way if the automatic transfer did not take place (e.g. by cancelling the document dialogue or due to an error).

Under certain circumstances this feature is not desired. Therefore, as of version 2801 of HELiOS, it is no longer auto-matically set up during a new installation. If the context menu entry is to be created in the process, you can activate the checkbox **Create context menu entry in Windows Explorer for saving files in HELiOS** in the **General** tab of the **Extended settings** installation step dialogue.

For update installations of older versions, the option is active and can be deactivated at the same point if desired.

New data type for List attributes

The new data type "LIST" is available for list attributes in the **Description** of attributes with the help of the Attribute Editor.

Under **Selection list** you can differentiate between a **Predefined** list and a free list (Option: **None**).

The separate area in the attribute editor, in which list attributes were displayed separately in earlier versions, is no longer necessary.

Installation DVD: Changed path of the default database

When importing the HELiOS standard database with the **HELiOS Database Creator**, please note that its path on the installation DVD has changed.

As of HELiOS 2023 Service Pack 1, you will find the database there at ...\Databases\en.

Major Release 2023 (V 2800)

Units in the context menu
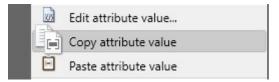
When right-clicking in result list columns of (float) attributes with unit category, the context menu shows you the corresponding unit, with designation and abbreviation.

Copy and paste attribute values with unit category

The behavior of copying and pasting attributes with unit category using the **Copy attribute value** and **Paste attribute value** context menu functions has been further improved.

Please read the corresponding notes.

Export and import files

Both the export of HELiOS documents and the import of files to HELiOS have been revised.

In the export dialogue you have e.g. extended setting and attribute assignment options for the generation of file names.

In the extended export dialogue you can de-/reselect linked objects and their contexts if necessary.

The file import, in turn, has been adapted to the common use case of re-importing exported HELiOS documents. The simple selection of an XML file belonging to the export offers all documents in a clearly arranged import dialogue.

The previously necessary editing of the system file hel_fileimport.ini is no longer necessary. However, configurations of older HELiOS versions stored there are still taken into account during the export.

Please also note the changes that result from this at **HELiOS Options > Import/Export**.

Print documents: File name configuration

As with the new options for attribute mappings when generating file names for exporting files, you can also configure the output file name by means of an attribute mapping when printing neutral formats in files (e.g. when outputting PDFs via the HELiOS Spooler).

The contents of the hel_fileprint .ini system file previously used for this purpose are copied to the attribute mappings when the file is first called. After that, the old ini file is no longer needed, but it is not automatically deleted from the system.

Index up-to-dateness

In result lists, the column **Index up-to-dateness** indicates by a symbol whether it is the most recent version index of a HELiOS object.

Most recent versions of objects are indicated by a green dot, old versions by an orange dot.

HELiOS Options: Default configuration for product structure output

From HELiOS 2023 and/or HiCAD 2023 onwards, a new version of the **ISD Report Manager** will be introduced.

The old version of the Report Manager will initially remain installed in the system in parallel.

You can control which Report Manager version is used for the product structure output in the **Database** Options. In the pull-down menu of the **Default configuration for product structure output** option, you can select which RMS or RM_SETTINGS file is to be used for the output of the product structure with the Report Manager.

In the default delivery database (i.e. for new installations of HELiOS 2023) the template PRODUCT_ISD.RM_SETTINGS is set. This new template file calls the new ISD Report Manager during data transfer.

However, existing settings of existing databases will not be overwritten by an update installation. A parts list template (of file type .RMS) previously defined as default configuration is retained and still uses the old Report Manager.

In this situation, it is recommended to switch to the new Report Manager by using a template file of the type .RM_SETTINGS.

Please note:

For printing result lists, the new ISD Report Manager is already used.

Detailed information on the new ISD Report Manager can be found here.

Mark-up and Redlining

Note that when working with the extended markup and redlining functionality of HiCAD and HELiOS, the redlining tool **HiRedLine** is only installed by a combined HiCAD/HELiOS installation. In a standalone installation of HELiOS Desktop the tool is not available.

A failed call will return a corresponding system message.

HELiOS in HiCAD

Service Pack 2 2023 (V 2802)

Automatic removal of documents from the HELiOS workspace

When HiCAD is closed, all documents not being processed are removed from the HELiOS workspace.

This also applies to neutral formats (e.g. PDF files) that were loaded into HiCAD via a HELiOS context menu.

Files with local changes (which have not yet been transferred to the HELiOS database) are not automatically removed.
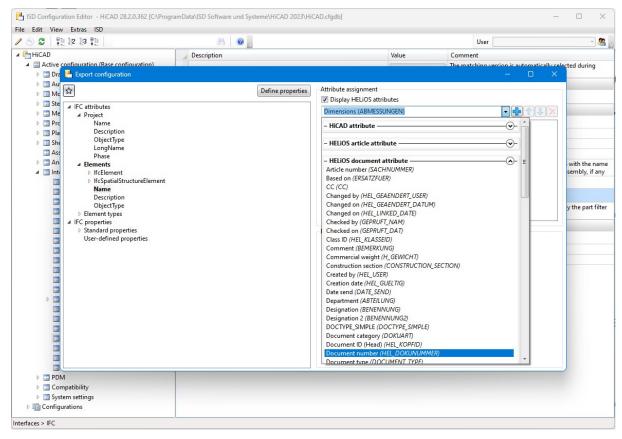
Message if HiCAD Integration module is missing

If, when starting HiCAD, a HiCAD basic module other than "HiCAD VI/AV" and a HELiOS basic module other than "HELiOS Viewer" is selected during licence selection, while no licence for the "HiCAD Integration Module" can be found in the system, then only viewing functionalities are available in HELiOS, regardless of the selected basic module. You will receive a corresponding message.

IFC interface: Show HELiOS attributes

In the attribute mapping configuration for the data transformation between IFC and HiCAD, you can also export HELiOS attributes from Service Pack 2.

In the Configuration Editor at **Interfaces > IFC > Default options for export > Attribute mapping configuration >** ☆ **Define** you will find the new checkbox **Display HELiOS attributes**.

If this is activated, HELiOS attributes are also available for mapping in the pull-down menu of the source attributes.
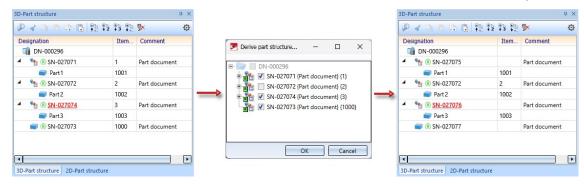
For further information, please refer to the HiCAD Help for **Interfaces**.

Service Pack 1 2023 (V 2801)

Deleting the item numbers when deriving

If you use HiCAD and HELiOS in the context of a project-specific working method in which the referencing option **For main parts and sub-parts** is set in the **Configuration Editor** at **System settings > Referencing > Synchronize item numbers/part attributes** when updating file, item numbers for derived parts are automatically deleted when the part structure is derived via the CAD structure or when a derivation is created via the HELiOS Desktop.
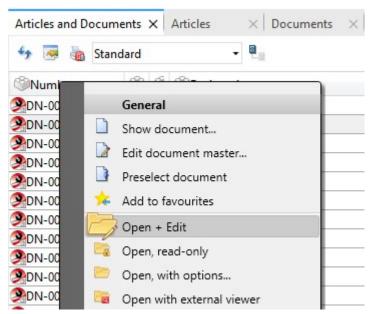
Open STEP file via HELiOS in HiCAD

When opening an object for editing in HELiOS with the **Open+Edit** function, it is intended that documents are opened for editing in the corresponding application. For example, .SZA files are opened in HiCAD.

This also applies to other formats such as STEP files (.stp) if they have been assigned HiCAD as the opening application via the Windows properties.

Under such conditions, if a STEP file is opened from HELiOS for editing, it is loaded for editing in HiCAD if HiCAD is already running, or HiCAD is started for editing the file if necessary.

Of course, opening a STEP file in HiCAD also works outside the HELiOS context by double-clicking in Windows Explorer.

New article class for part type Welding neck

If you use HELiOS in connection with the HiCAD module Plant Engineering, a new HELiOS article class is available from Version 2023 SP1 onwards with the new part type **Welding neck**.

You must inform HELiOS of the new classification for welding neck flanges. This is done via the programme **DBPlantDataImport.exe**, which you will find in the HiCAD exe directory. There you click on the button **Update HELiOS for Plant Engineering**.

Afterwards you can use the HELiOS search mask in your HELiOS Desktop.

Major Release 2023 (V 2800)

Transfer of the HiCAD sheet structure

When saving SZA files to HELiOS, the HiCAD sheet structure is also transferred to HELiOS. This contains the number of sheets and the **Type**, **Sheet number** and **Sheet name** for each sheet.

The sheet structure of SZA documents can then be viewed in the document master on the tab **Model structure**:
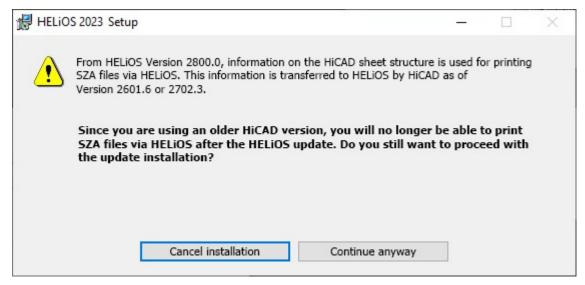

When printing documents via HELiOS, this sheet structure is also displayed accordingly for the selection of the sheet area to be printed:

⚠ Please note that when updating to HELiOS 2023 (V. 2800) in combination with a HiCAD version older than HiCAD 2023 (V 2800), printing of SZA files via HELiOS is no longer possible due to this change.

In this case, you will receive a corresponding message during the installation:
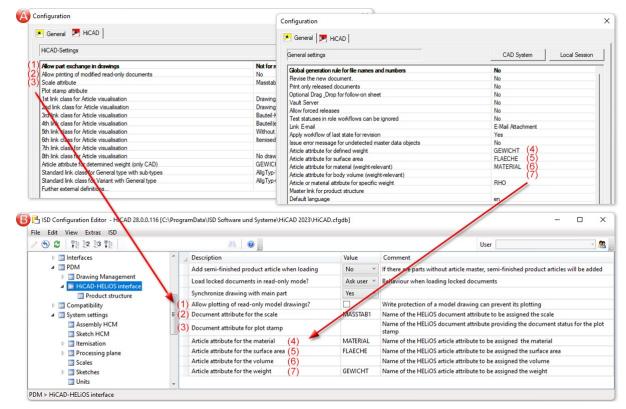

A migration of inventory data can then be automatically carried out when HiCAD is started with HELiOS.

This applies to the following HiCAD Versions:

- HiCAD 2021: As of Version 2601.6
- HiCAD 2022: As of Version 2702.3
- HiCAD 2023: As of Version 2800.0

HiCAD-specific HELiOS settings moved to Configuration Editor

As of version 2800, HiCAD manages some attributes that previously had to be defined in the HELiOS settings.

A: up to HiCAD / HELiOS 2702 (HELiOS Options > Datenbank), B: from HiCAD / HELiOS 2800 onwards (ISD Configuration Editor: PDM > HiCAD-HELiOS interface).

When updating the database scheme you will receive the following message:

New ISD Report Manager

As of HELiOS 2023 / HiCAD 2023, a new version of the ISD Report Manager will be introduced for the issuance of bills of materials, which is much simpler and more convenient, especially when configuring BOM templates.

> Detailed information can be found in the Online Help chapter for the ISD Report Manager.

HiCAD - HELiOS Head ID

As of HiCAD 2023, it is no longer possible to specify the Head IDs of articles and documents in HELiOS. This ID was previously used by the catalogue editor and the DBPlantDataImport/PartDataAutoSync tools to find articles again.

Now, unique IDs are assigned by the variant editor itself and transferred to HELiOS during matching. These IDs are directly visible in HELiOS, as they are not a regular article attribute.

The new IDs are only directly visible in a few places in HiCAD:

- When matching with the catalogue via the tool VarToCat, an additional table column PLANTID is inserted.

- In P+ID, the new ID becomes visible in the attribute **Database article ID** when assigning article data.

- After a part has been installed, the PLANTID is also found in its part attributes.

HELiOS Spooler

Service Pack 2 2023 (V 2802)

Performance

With Service Pack 2 a further performance increase of the HELiOS Spooler could be achieved.

The optimization becomes especially obvious when performing and displaying a very high number of active print/-convert jobs.

Result list attribute to mark documents being updated by Spooler

In the user interface configuration of HELiOS document result list you will find the new attribute **Update status** (VA_DocumentPendingUpdateJob).

If the column has been added to result lists, it marks there, if applicable, documents that are currently being updated by the Spooler and therefore cannot be currently displayed with a ↻ symbol.

PDF printing via Windows

As of Service Pack 2, the Spooler supports the possibility to use the Windows functionality **Microsoft Print to PDF** for printing to PDF.

This output may be faster and an installation and setup of the ISD FilePrinter PS may not be necessary.

Service Pack 1 2023 (V 2801)

Performance

By loading the document structure directly on the Spooler Server instead of from the Client side, a significant increase in performance could be achieved during printing/conversion.

Major Release 2023 (V 2800)

Support of RM3/ RMA files discontinued

Note that printing/converting RM3/ RMA files is no longer supported by the HELiOS Spooler as of Version 2800.

Server Monitor

With the help of the **Server Monitor** tool, you can monitor the runnability of HELiOS server applications such as the **Vault Server** or the **HELiOS Spooler** and make any necessary adjustments in the event of problems.

HELiOS Task Starter

The new **HELiOS Task Starter** tool can be used for automatically re-starting the HELiOS Spooler after it has been closed, or after a possible crash.

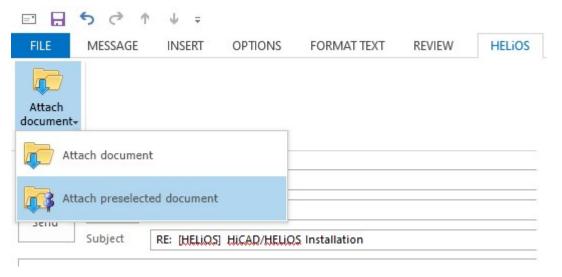# HELiOS MS Office Interface

Service Pack 2 (V. 2802)

Combined search

Calling the functions **Open + Edit** or **Open, read-only** from the menu of a HELiOS-Office interface and **Attach document** in an active Outlook mail opens the **Combined search** mask search of HELiOS as of Service Pack 2.

Major Release 2023 (V 2800)

Outlook: Attach preselected document

In addition to inserting E-mail attachments via the HELiOS function **Find document**, you can also attach a document

that you have previously marked for this purpose via the context menu function **Preselect document**, e.g. in a

result list of the HELiOS Desktop, by simply clicking on **Attach preselected document** in the context of an active E-mail.

HELiOS Vault Server

Service Pack 2 2023 (V 2802)

Server Monitor: Communication port for article synchronisation during installation

The **Port** via which the Server Monitor communicates with **HELiOS Article Synchronisation** with ECAD/CAE applications such as EPLAN or Zuken E3 can be defined not only in the settings of the Server Monitor tool, but also during the installation process of the part synchronisation.

The default port number is still 35727.

Service Pack 1 2023 (V 2801)

Server Monitor: HELiOS Article Synchronization

The **Server Monitor** has been extended to include the option of monitoring the HELiOS Article Synchronization service for HELiOS interfaces to ECAD/CAE applications such as EPLAN or Zuken E3.

Major Release 2023 (V 2800)

Server Monitor

With the help of the **Server Monitor** tool, you can monitor the runnability of HELiOS server applications such as the **Vault Server** or the **HELiOS Spooler** and make any necessary adjustments in the event of problems.

What's new?

Legal notes

Germany

Headquarter Dortmund
ISD Software und Systeme GmbH
Hauert 4
D-44227 Dortmund
Tel. +49 231 9793-0
info@isdgroup.de

Sales office Berlin
ISD Software und Systeme GmbH
Paradiesstraße 208a
D-12526 Berlin
Tel. +49 30 634178-0
berlin@isdgroup.de

Sales office Hamburg
ISD Software und Systeme GmbH
Strawinskystraße 2
D-25337 Elmshorn
Tel. +49 4121 740980
hamburg@isdgroup.de

Sales office Hanover
ISD Software und Systeme GmbH
Hamburger Allee 24
D-30161 Hanover
Tel. +49 511 616803-40
hannover@isdgroup.de

Sales office Nuremberg
ISD Software und Systeme GmbH
Nordostpark 7
D-90411 Nuremberg
Tel. +49 911 95173-0
nuernberg@isdgroup.de

Sales office Ulm
ISD Software und Systeme GmbH
Wilhelmstraße 25
D-89073 Ulm
Tel. +49 731 96855-0
ulm@isdgroup.de

International

ISD Austria
ISD Software und Systeme GmbH
Hafenstraße 47-51
A-4020 Linz
Tel. +43 732 21 04 22-0
info@isdgroup.at

ISD Benelux - Hertogenbosch
ISD Benelux B.V.
Het Zuiderkruis 33
NL-5215 MV 's-Hertogenbosch
Tel. +31 73 6153-888
info@isdgroup.nl

ISD Benelux - Zwolle
ISD Benelux B.V.
Grote Voort 293A
NL-8041 BL Zwolle
Tel. +31 73 6153-888
info@isdgroup.nl

ISD France
ISD Group France SAS
10 -12 Boulevard Vivier Merle
F-69393 Lyon
Tel. +33 6 73 72 04 67
info@isdgroup.fr

ISD Switzerland
ISD Software und Systeme AG
Rosenweg 2
CH-4500 Solothurn
Tel. +41 32 624 13-40
info@isdgroup.ch

ISD Switzerland
ISD Software und Systeme AG
Rte du Jura 37 A, 4. Étage
CH-1700 Fribourg
Tel. +41 79 803 51 51
info@isdgroup.ch

ISD USA - North Carolina
ISD Group USA Inc.
20808 N Main Street, Suite 101
USA-Cornelius NC 28031
Tel. +1 770 349 6321
info@isdgroup.us

ISD USA - Georgia
ISD Group USA Inc.
5126 South Royal Atlanta Drive
USA-Tucker GA 30084
Tel. +1 770 349 6321
info@isdgroup.us

www.isdgroup.com