

# HiCAD 2025 - What's new?

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## **Discontinuations**

#### Discontinuation of Windows® 7 and Windows® 8

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

#### Discontinuation of "old" HiCAD itemization

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

#### Discontinuation of "old" OpenGL versions

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

#### **Discontinuation of old figure format (FIG)**

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

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

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

#### Discontinuation of the "old" Create detail drawing function

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

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

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

#### **Discontinuation of CADENAS PARTdataManager**

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

Discontinuation of 3-D projection grid

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

Discontinuation of the Zuken E3 interface

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

Discontinuation of the "old" Report Manager

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

Discontinuation of Part insertion v26 (PE)

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

## **Basics**

## Service Pack 2

## Freely configurable date format in PostScript file names

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

Reorganisation of the ICN

#### Properties dialogue window

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

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

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

	τ×	Properties	ф >
1	¢		¢
Item number	Comment	Designation	Value
		Part name	<different values=""></different>
	Assembly	Part type	Solid
	I - Beams	Colour	Oyster White 🔹
	I - Beams	Layer	1
	I - Beams	Material	S235JR
		Usage	TE 🗙
		Module	Steel Eng. (Beam)
		Data model	Exact
		BOM-relevant	
		Item number	
		Article number	<different values=""></different>
		Designation 1	
		Designation 2	<different values=""></different>
	Item number	Litem number Comment Assembly I - Beams I - Beams I - Beams I - Beams	Item number       Comment       Designation         Assembly       Part name         Assembly       Part type         I - Beams       Colour         I - Beams       Layer         Material       Usage         Module       Data model         BOM-relevant       Item number         Article number       Designation 1         Designation 2       Designation 2

Reworking of the view window in the ICN

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

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

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

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



Extension of the part structure toolbar

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

3-D Part structure	ф >
P / B 6 B 🖪 🏗 P	
Designation	2 Levels
DRAWING1	3 Levels
Main assembly     Assembly	4 Levels
A 1 t Assembly	5 Levels Assembly

#### Improved layout of options in the Novice Configuration

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

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#### Attribute management

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

#### **Duplicating attributes**

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

Short text	2	🛛 Long text	Ţ	ID	<u>*</u>	Data type 📃 💌	Assignment	💌 Referencing 💌	Permission	r Group	G
Beam											
Angle bottom/left - XZ		Angle bottom/left - XZ		§05	1	Floating point number	Part attribute	Standard behaviour	System	Beam	
Angle bottom/left - YZ	_	Anale bottom/left - YZ	-	§06	1	Floating point number	Part attribute	Standard behaviour	System	Beam	
Angle bottom/right - XZ		New		§07		Floating point number	Part attribute	Standard behaviour	System	Beam	
Angle bottom/right - YZ		Delete		§08	1	Floating point number	Part attribute	Standard behaviour	System	Beam	
Aperture angle				P_BW		Floating point number	Part attribute	Always transfer	System	Beam	
Commercial weight		Copy Ctrl+C		§18	1	Floating point number	Part attribute	Standard behaviour	System	Beam	
Commercial weight by length		Paste Ctrl+V		<b>§CBL</b>	1	Floating point number	Part attribute	Always transfer	System	Beam	
Commercial weight per length		n an an		§19	1	Floating point number	Part attribute	Standard behaviour	System	Beam	
Cross-section area of web in cm^2	-	Kemove all filters	2	ASTEG	1	Floating point number	Part attribute	Always transfer	System	Beam	
Curve radius about y	$\checkmark$	Group view		P_BRY	1	Floating point number	Part attribute	Always transfer	System	Beam	
		The second second		0.007	1	en las la sub-	and Lowers	1 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4	1. S.		*

#### Transfer of referencing settings to attribute management

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

#### **Attribute description**

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

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

#### **Attribute calculation**

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

#### Attribute calculation for locked assemblies

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

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

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

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

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

#### Favourites in input fields

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



### Refinement in the combination of manual and automatic itemisation

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

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

The itemisation settings are set using the dialogue window Itemisation with options. The Automatic itemisation is car-

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



#### Drawing derivation: New options for selecting parts

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

A	<b>▼</b>	Views to be created for:
A	220.00	
Or	nly new	
Se	ect individual	
Se	ection list, with sub-parts	
Se Se	lection list, without sub-parts	<u></u>
O	nly new parts with selection list (inc	duding subordinate parts)
Or	nly new parts with selection list (wi	thout subordinate parts)
1	Sheet Metal	Sheet Metal
V	General parts	General parts

#### Default setting for the drawing frame in the Configuration Editor

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

1	Description	Value	Comment
	Height	410 mm	Height of drawing frame
	Width	584 mm	Width of drawing frame
			This option is used if there is more than one view group in a drawing frame.
	Selected drawing frame (key designation)	Fit DIN frame 💙	In the case of "New sheet for each assembly" where there
	designation	Fit DIN frame	may be more than one view group in a drawing frame, the
		DIN A0	assembly settings are taken into account.
		DIN A1	
		DIN A2	
		DIN A3	
		DIN A3 SHEETS	
		DIN A4H	
		DIN A4H PLATES	
		DIN A4H BEAMS	
		DIN A4Q	]

#### Drawing tolerance

Due to rounding errors, numerical values such as point coordinates can be checked for equality up to a certain tolerance level. The default value is 0.001. This means that two points are considered identical if their distance is less

than a thousandth of a millimeter. If you change the drawing tolerance using the **Tolerance 1000** function, a warning message now appears when you load the drawing.



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



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

## Design Checker

#### **Test results**

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

	<ul> <li>Results per test</li> </ul>	<ul> <li>Results per par</li> </ul>
🕥 👠 Sheet must contain exa	ctly 1 direction symbol	
1000_001_002 {Kantblech}{}	Number of direction symbols: 2	9
1000_001_005 {Kantblech} {}	Number of direction symbols: 0	٩,
Manufacturability check	¢	
This test did not produce any	/ results	
This test did not produce any	y results	

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

- Test results		
	○ Results per test	• Results per part
1000_001_002 {Kantblech} {}		
📤 Sheet must contain exactly 1 direction symbol		
🔿 1000_001_005 {Kantblech} {} 🔍		
🔔 Sheet must contain exactly 1 direction symbol		
Test execution		
O Drawing	e dialogue) 🛛 📿	selected parts

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

#### Manufacturability check

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

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

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

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

The following new checks are available:

#### Minimum flange length

As with chamfering, the minimum length of flanges is measured directly on the outside or directly on the outer tangent. Only the outside of the cover surface is measured; chamfers may reduce this. A distinction is made between acute and obtuse angles.

#### Minimum bend radius

To avoid overloading the material, a certain minimum bend radius must be ensured.

Distance between processings

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

Minimum diameter for standard bores

This test only applies to standard bores.

Minimum Z-fold height

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

## D Please note:

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

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

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

## Optimise cursor size

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

## Service Pack 1

### Drawing derivation: New option for alignment of assemblies

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

> Drawing derivation × Drawing target To existing drawing -Externally generated drawings Leave open, switch back to original drawing Ŧ Sheet selection Alignment of assemblies New sheet • -Processing position Processing position Drawings for: Fitting position Position after part alignment All • Views to be created for: ✓ Assembly Filter Assembly Beams ✓ Beams Plates Plates ✓ Sheet Metal Sheet Metal General parts General parts Drawing parameters Settings for: From configuration Drawing sheets Configuration View groups Save Load Views Set in dialogue Sheet developments Settings file Save Load Sectional views of sheets OK Cancel

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

#### Attribute for the pre-bend length of beams

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

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

#### New options for part filters

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

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

Part type:	Assembly, Part			Change 👻	
Assembly	with main part type	is 🔹	Beam	- 🗣 🗙	÷
			General part		
Add rule:		-	Beam		4
			Steel Engineering plate		
			Sheet Metal	-	
$\times$			without main part	✓ Case	sensitive

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

Manual filter All User-defined	
Steel Engineering - Assembly	
Assembly with beam as main part Assembly with general part as main part	
Assembly with sheet as main part Assembly with plate as main part Assembly without main part	
Assembly with or without general part as main part	
Steel Engineering - Indiv. part, Facade Engineering	
Clamping strip or cover tray Foil Insulation Isolator Mullion or Transom	

1

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

#### Configuration of drawing derivation

Further settings are now available in the Configuration Editor under Automatic drawing derivation > Production draw-

ing for the automatic creation of workshop drawings using the **Drawing derivation** function.

Settings for beams

The settings Production drawing and Processing note for unprocessed beams were previously under System set-

tings > PDM > Drawing Management > Production drawing and were only relevant for the Drawing function within Drawing Management.

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

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

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

#### Settings for Steel Engineering plates

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

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

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

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

New function for creating a structure assembly

. 🍳

tions and create an assembly that is not BOM-relevant and has the **Part typeStructure assembly**.



The new functions **Create structure assembly** and **Create structure assembly as sub-part** is have been added to the **3-D Standard** > **New** > **New assembly** menu for creating structure assemblies. In addition, the new functions have been added to the **Context menu for drawings** under **New 3-D Main Part** and to the **Context menu for assemblies** under **New Part** > **Main part** or **New Part** > **Sub-part**. The functions behave like the other assembly func-

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

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

## Design Checker

#### Manufacturability check for sheets

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

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

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

Edit View Extras ISD			
S 🖉   📲 🗠 🕄 📲	AA 🛛 🞯 🕕 💂		User 🖉 🦉
HiCAD  HiCAD  A I Active configuration (Base configuration)  I Drawing  Automatic drawing derivation	Carry out manufacturability check when exporting sheets	Value	Comment
<ul> <li>Modelling</li> <li>Steel Engineering</li> </ul>	Check distance of processing from bend zone		
Metal Engineering	Take values for comparison from	Use formula from V	
<ul> <li>Profile Installation</li> <li>Blant Engineering</li> </ul>	Formula for slots running parallel	15	Example: 'L > 30mm? 5*T : 3*T' with L = leng of slot and T = sheet thickness
Sheet Metal     Bend zone tooling	Formula for rectangles running in parallel	15	Example: 'L > 30mm? 5*T : 3*T' with L = leng of rectangle and T = sheet thickness
Sheets with identical cross-sections	Formula for bores/other processing	15	Example: '5*T + 3mm' with T = sheet thickne
Manufacturability check	Distance of processings to edge		
📰 Default setting	Check distance of processing from edge	$\checkmark$	
Sheet development	Take values for comparison from	Use formula from V	
Assembling simulation	Formula for comparison value	15	Example: '5*T + 3mm' with T = sheet thickn
Analysis	Maximum sheet dimensions		
Interfaces	Check maximum sheet dimensions	✓	The values are taken from the catalogue
System settings	Collisions in developments		
Configurations	Check collisions in development	✓	
	Ignore collisions of bend zones		

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

If you want to load the values from the catalogue, you must first fill in the columns

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

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

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

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

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

P Design Checker		
- Available checks		
Manufacturability check 74	/ /	
Mirrored parts		
☆□ Sheet must contain exactly 1 direction symbol		
🕨 🗌 General		
Feature		
▶		
- Test results		
Manufacturability check		
Values for maximum dimensions are not stored. Plazes enter them in the satul		
L Collision with 'Flange'	<u></u>	
This test produced 1 results. 1 parts/assemblies were examined	<b></b>	
- Test execution		
O Description (character (character de character))	1 selected as to	
Urawing Selected parts (at dialog start)	I selected parts	
Check:		
Total progress:		
	Start Close	

Changes in user guidance

- Favourites and the meaning of the checkbox If you select I the category (e.g. General or HCM) when selecting the tests, all subordinate tests are automatically activated. If you deactivate some tests from the list, the category is marked with a square . If the test is marked with a yellow star A, it is also listed under Favourites and can be selected from there.
- Checked

parts

The checked parts are now marked in newer checks (e.g. Check distance of processing from bend zone) by activating this symbol 🙀 (next to the test result).

Custom tests in C#. Either scripts or precompiled tests can be used. Precompiled tests usually perform better.

## Major Release

## Update of the part attribute Total quantity

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

Drawings that are not saved as itemised source models include, for example, detail drawings that are created using

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

## Change of the special colours

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



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

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



#### Changes to the management of HiCAD attributes

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

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

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

dit View Extras ISD									
	A	0 0				User			-
System settings	▲ Short text	Long text 👳	ID 👱	Data type 🛛 💌	Assignment	Ŧ	Permissic 👱	Group	
Attribute management	Semi-finished product	Semi-finished produc	#ВІ	Integer	Part attribute		Internal	General	
Attributes	Document master revi	Document master rev	#DR	Integer	Part attribute		Internal	General	
Attribute calculation	Modified	Modified	#IP	Integer	Part attribute		System	General	
Attributes	Ignore for dimensions	Ignore for dimension:	#NDR	Integer	Part attribute		System	General	
Assembly HCM	Itemised source mode	Itemised source mode	#PMOD	Integer	Drawing attrib	ute	System	General	
Sketch HCM	Semi-finished product	Semi-finished produc	#RBR	Integer	Part attribute		Internal	General	
Processing plane	BOM-relevant	BOM-relevant	#SR	Integer	Part attribute		System	General	
Scales	= Designation 1	Designation 1	\$01	Text	Part attribute		System	General	
Sketches	Designation 2	Designation 2	\$02	Text	Part attribute		System	General	
📰 Units	Comment	Comment	\$03	Text	Part attribute		System	General	
Novice configuration	System notes	System notes	\$04	Text	Part attribute		System	General	
Directories	Part type	Part type	\$05	Text	Part attribute		System	General	
Load/Save	Number of section sch	Number of section sc	\$06	Text	Part attribute		Internal	General	
<ul> <li>Data save</li> <li>Identification</li> </ul>	_ New Delete	C Group view							_

Further new developments concerning attributes are:

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

%ts(atSh_\$03)	Comment	\$03	Text	Part attribute	System	General	
Multilingual	System notes	\$04	Text	Part attribute	System	General	
	Part type	\$05	Text	Part attribute	System	General	

Language	Value	
German		
English		
French		
Hungarian		
Italian		
Polish		

A list of all HiCAD attributes can be found here.

#### Differentiation when selecting parts in a rectangle

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

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

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

#### Novice Configuration - Dialogue change

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

			Novice configuration
			Settings     Apply settings from configuration database
Assembly HCM     Sketch HCM     Sketch HCM     If termisation     Processing plane     Scales     Scales     Movice configuration     Directories     Load/Save     Data save     Identification     Referencing     Annotations     Calculations     Graphic     Visualisation		Description	Control Settings     Load settings from configuration database
		Preset identification mode	- General Preset identification mode: Object
		Right-click to use active surface as plane (Element snap mode)	- Sketch and 3-D sketch
		Query plane for new sketch (only planar sketch) World CS planes for new sketch (also 3-D sketch)	✓ Right-click to use active surface as plane (Element snap mode)
		Rotate new planar sketch in model view parallel to screen	V World CS planes for new sketch (also 3-D sketch)
		Modelling	Rotate new planar sketch in model view parallel to screen: No
		Rotate model view into perspective in preview	
		Query plane when applying standard processings	Shape     Rotate model view into perspective in preview     Query plane when applying standard processings
	-		OK

#### New function in QuickAccess toolbar



function has been added to the Die QuickAccess toolbar.



#### Save referenced detail drawings as itemised source model

As of HiCAD 2025, detail drawings created via the Drawing > Save/Reference > Reference part, Save, Detail draw-

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

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



A

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

### SpaceMouse<sup>®</sup>

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

## Drawing derivation: Simplified dialogue

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

Drawing derivation	×			
(Drawing target)				
To existing drawing				
Externally generated drawings				
Leave open, switch back to original drawing				
Detail drawing for each sub-part				
Sheet selection	(Alignment of assemblies)			
New sheet	Processing position			
Drawings for:	Views to be created for:			
Al				
Assembly Filter	Assembly			
Beams	(Beams )			
✓ Plates	Plates			
✓ Sheet Metal	Sheet Metal			
General parts	General parts			
Drawing parameters	(Settings for:			

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

Drawing derivation	×			
(Drawing target)				
To existing drawing				
Externally generated drawings				
Leave open, switch back to original drawing				
Detail drawing for each sub-part				
(Sheet selection)	Alignment of assemblies			
New sheet	Processing position			
	Assembly X			
Drawings for:	With beam main part			
	With sheet main part			
Assembly Filter	✓ With plate main part ✓ General			
Beams	OK Cancel			
Plates	Plates			
Sheet Metal	(Sheet Metal			
General parts	General parts			
Drawing parameters				

# 2-D

Service Pack 2

Delete individual graphical elements

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

**Major Release** 

## Settings for 2-D standard parts and standard processings

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


# 3-D

## Service Pack 2

## Grid annotation

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

New features in the Create grid annotation function:

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

Two new options have been added:

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

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

#### New features in the Edit grid annotation

dialogue:

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

The Representation parameters . Adopt display parameters from reference annotation and Favourites

symbols are only active if at least one line is selected.

New features in the Grid annotation context menu:

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

### Optimisation of some icons in the sketch area

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



The function <b>3-D Standard &gt; Standard processings &gt; Bores, Countersinks, Threads</b>	į.	has been reworked:
New features in the dialogue:		

-

- V Perpendicular for laser cutting (i) Clearance:	1	•
- Production		
- Representation		
Preview OK	Cancel	Apply

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



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

Changes to the forms:

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

- Form		
- Туре ————		
Length:	20	•
Width:	10	-
Fillet radius:	1	-
Rotation angle:	0	•
4 🌒 (3) Rectangle 20x10		
🖌 🥅 Form	Rectangle	
E Length	20 mm	
🕅 Width	10mm	

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

New features of the grid:

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

Grid	
- Definition type	_ <i>Z1</i>
Mid-point 0	
Diameter: 50 💌	
Number: 4	11
Start angle: 0 💌	
Aperture angle: 360 💌	And the second s
- Omissions	
2 Elements	
2 3 •	
Close	

## Transfer of reference settings to attribute management

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

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

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

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

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

## Optimisations to the planning grid

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

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



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

#### Views

#### Change scale of view with multiple selection

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

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

#### Undo for changing the scale

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

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

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

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

#### Greying out of view functions

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



## Optimisation of fillet preview

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

#### Preview of KRA files

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

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

## Activate BOM-relevance

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

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

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

The default state can be restored using HicadGUIReset.exe.

## Installation depth can be selected for anchors in boltings

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

Parts Options	Settings	_
	Bolt anchor FBN II A4 V M 6 M 8 M 10 M 12	▲
	Min. anchoring 25 M 16 depth M 20	
	Max. thickness of 10 the filler plates	

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

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



## Boltings: Parts from the catalogue Purchased / Factory standard parts

From now on, you can access fasteners that were previously saved as parts in the Factory standards> Purchased /

Factory standard parts catalogue in the New bolting/riveting **W** function dialogue window.

#### Filleting and chamfering: Value input

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

## Search in the catalogue selection



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

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

Nuts	×	Find							×
•1. •1. Find ▼	Q	<b>₽</b>	mб ruc						- X
A 🛷 Fasteners	<u> </u>	Threa	ad TYPE	Designation	Size	DN [mm]	P [mm]	MATERIAL	
A 🧖 Nuts		Flat	head blind rivet nu	t, Type UC (Facto	ry standards/User-	defined fasten	ers/User-	-defined nuts,	(Filko )
	Cap nuts	M6	ST zinc-plated	M6 RUC FEFG 2.5	M6	6	1	St	
a cap nuts		M6	ST zinc-plated	M6 RUC FEFG 3.5	M6	6	1	St	
DIN 1587	<u></u>	M6	ST zinc-plated	M6 RUC FEFG 2.5	M6	6	1	St	
DIN 917		M6	ST zinc-plated	M6 RUC FEFG 3.5	M6	6	1	St	
A Castle nuts P	Castle nuts	• Flat	head blind rivet nu	t, Type UC, open	( Factory standards	/User-defined	l fastener	s/User-define	d nuts/Filke
	-	M6	ST zinc-plated	M6 RUC FEF 2.5	M6	6	1	St	
Nut with squeezing device		M6	ST zinc-plated	M6 RUC FEF 3.5	M6	6	1	St	
· ·		M6	ST zinc-plated	M6 RUC FEF 2.5	M6	6	1	St	
	Nut with squeezing device	M6	ST zinc-plated	M6 RUC FEF 3.5	M6	6	1	St	
	Various nuts	<							2)
	OK Cancel	Apply	selection directly				Ok		Cancel
P Find	x	Pind							×
	•	P1 91	ruc						- X
DIN 1597 (Fastenase Aluste Can pute )		Threa	d TYPE	Designation	Size	DN [mm]	P [mm]	MATERIAL	
(Pasteners/Wats/Capitals)		<ul> <li>Flat</li> </ul>	head blind rivet nu	t, Type UC (Facto	ry standards/User-	defined fasten	ers/User-	-defined nuts	(Filko)
(Fasteners/Nuts/Cap nuts)		M3	ST zinc-plated	M3 RUC FEFG 1.1	M3	3	0.5	St	
		M3	ST zinc-plated	M3 RUC FEFG 2.3	M3	3	0.5	St	
		M3	ST zinc-plated	M3 RUC FEFG 3	M3	3	0.5	St	
		M4	ST zinc-plated	M4 RUC FEFG 2.1	M4	4	0.7	St	
		M5	ST zinc-plated	M5 RUC FEFG 1.5	M5	5	0.8	St	
		M4	ST zinc-plated	M4 RUC FEFG 3.7	M4	4	0.7	St	
		M5	ST zinc-plated	M5 RUC FEFG 2.5	M5	5	0.8	St	
		M5	ST zinc-plated	M5 RUC FEFG 3.5	M5	5	0.8	St	
		M6	ST zinc-plated	M6 RUC FEFG 2.5	M6	6	1	St	
		M6	ST zinc-plated	M6 RUC FEFG 3.5	M6	6	1	St	
		M8	ST zinc-plated	M8 RUC FEFG 3	M8	8	1.25	St	
		M8	ST zinc-plated	M8 RUC FEFG 5	M8	8	1.25	St	
	1)	M10	ST zinc-plated	M10 RUC FEFG 4	M10	10	1.5	St	3)
	1)	M10	ST zinc-plated	M10 RUC FEFG 5.5	M10	10	1.5	St	<u> </u>
Apply selection directly	OK Cancel	Apply	selection directly				Ok		Cancel

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

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

### Divide along direction

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

Divide along direction	×
- Part	
Cuboid_0 {} {}	
- Direction	
Select direction	<b>2</b>

## Dimensioning

#### Magnetic snap-in dimensioning and annotation

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



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



Please note:

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



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

## Service Pack 1

## Further development of the new standard processing function

The dialogue window for the function 3-D Standard > Standard Processings > Bores, Countersinks, Threads

has been extended.

Standard Processings	×
- Part to be processed	9-
Sheet {Sheet Metal} {}	
- Processing plane	9-
Select processing plane	1
Reference plane for depth specifications:	
- Grid	
Individual	2
- Form	
- Туре	_
Standard:	P
Diameter: 5	
- Bore depth Depth: 20	
- End form	_
Point angle: 120	
Reference:	
- Perpendicular for laser cutting ①	<u>-</u>
	<u></u>

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

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

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

71	Grid - Definition type - Linear	
	Select	insertion point
	Mode:	Reference and 💌
	Number:	2 🔹
X1	Select st	art point or edge
	Sele	ect end point
	Start distance:	0 🗸
	End distance:	0 -
		Close

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

Since most of the functions for creating standard processings have now been integrated into the dialogue of the **Bores, Countersinks, Threads** function, the **Standard Processings** ribbon has been adapted so that the function icon

is in first place here. In the pull-down menu below, you will find the functions Copy standard processing

Bore pattern , Punch mark and Hole pattern . The tab Compatibility > Up to HiCAD 2024 has been added, where you will find the old standard processing functions.



Powder marking lines

## 3-D Grid

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

ers	>	Planning	grid	BH7
010	-	панны	5110	

	() () ()	
X-direction		
lumber: 4	<b>-</b>	
1ode: Different ind	lividual distances 🔹	
Distance betw. bo	res Designation 🛞 Planes 🕦 📄	
3 1000	C All 🔺 📉	
4 1000	D All	
Anata	Designation: 100 -	
Angle: 0	Projection: 100	
Y-direction		+
lumber: 4		
lode: Different ind	lividual distances 🔹	
Distance betw. bo	res Designation 🛞 Planes 👔 👘	
3 1000	3 <u>All</u>	+
4 1000	4 <u>All</u>	4-A
Angle: U	Projection: 100	3-6
		2-0
Levels		1 A
Levels		B
Levels lumber: 3 fode: Different inc	lividual distances	
Levels Jumber: 3 Node: Different inc Distance betw. bo	lividual distances  res Designation	
Levels	ividual distances  res Designation 1000	
Levels	fividual distances	
Levels	ividual distances	
Levels lumber: 3 fode: Different inc Distance betw. bo 2 1000 3 1000	fividual distances	
Levels Jumber: 3 Iumber: 3 Different inc Different inc 1000 1000 General	fividual distances  res Designation 1000 2000 Projection: 100	



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

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

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

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

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



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

Revision of the grid annotation functions

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

Grid —				
1 Grid				
Views —				
1 View				
Annotatio	on text	·		
Pre	fix 🔹	Designation		ostfix 🔹
Axes —				
Direction	Annotation	Excess length, start	Excess length, end	
Х	Start 💌	50 🗸	50 👻	
Y	Start 🔹	50 🗸	50 -	
Z	Start 💌	50 👻	50 -	
eights ah	ove datum fo	r arid planes		Start 1
		r gria planes.		Start
Intorrun	t annotation	line		
	create larger	than grid		
Do not				

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

from the **Reference parameters** from the new function dialogue and therefore remove them from the pull-down menu.

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

M Edit	grid annotation				
- Grid	edges —				
1 Elen	nent			2	X
- Anno	tation text				
Anno					
	Prāfix 🔹	Designation	•	Postfix	
	Prāfix 🔻	Designation	•	Postfix	_
- Axes	Prāfix 🝷	Designation	•	Postfix	
- Axes	Prāfix •	Designation	•	Postfix	
- Axes	Prāfix •	Designation	•[	Postfix	
- Axes	Prāfix • ange excess leng ion Annotation	Designation ths Excess length, start	Excess leng	Postfix gth, end	
- Axes Chi Direct	Prāfix   ange excess leng ion Annotation Start	Designation ths Excess length, start	Excess leng	Postfix gth, end	
- Axes Cha Direct X	Prāfix • ange excess leng ion Annotation Start •	Designation ths Excess length, start 50 -	Excess leng	Postfix gth, end	
- Axes Chi Direct X Y	Prāfix • ange excess leng ion Annotation Start • Start •	Designation ths Excess length, start 50 50 50	Excess leng	Postfix	

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

#### Fif function under the tab Up to HiCAD 2024.

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

#### Delete grid annotations

.

The functions for deleting grid annotation lines have been combined into the **Delete selection of annotation lines** 

+ X function. This function allows you to select and delete individual lines manually using the left mouse button or to make multiple selections using the right mouse button to delete several lines at once. The multiple selection offers the following options:

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

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

#### New function for creating a structure assembly



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

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

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

### Remove references to an active part list

The functions Break up referencing, Individual+identical parts and Break up referencing, Active part+sub-

parts under Drawing > Save/Reference > Update referenced identical parts can be applied not only to an active part or an active assembly, but also to an active part list. To do this, select the desired parts in the ICN while hold-

ing down the CTRL key and then right-click to select the desired function for cancelling referencing.

### Annotation - base point symbol arrow

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



#### Undo when moving views

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

Selection list of the entered texts in the Text Editor

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



Moving views in alignment

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

196	Move all linked views	
980 5	Only keep alignments	
œ	Link to another view	
<u>0:0</u>	Link and align to view	

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

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

90	Move all linked views	
98 5	Only keep alignments	
<u>0:0</u>	Arrange in alignment direction	
œ	Link to another view	
<u>0:0</u>	Link and align to view	
¢ź	Delete link	

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

## Auxiliary lines for form and positional tolerances

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



## Major Release

Internal/External threads - Dialogue change

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

Marcad Thread	×
- Insertion surface	
Select insertio	on surface and thread direction
- Parameters	
Thread type:	DIN 405 -
Туре:	F
Size:	RD 8x1/10 🗸
Thread direction:	Left-handed
Length:	Full length
Start form:	Without
End form:	Without 🔻
Show thread Exact detailing (1)	
	Apply immediately
	OK Cancel Apply

## Search material

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

Find	+	All		1	₽ <u>∎</u> 10S			-
Materials	Designation	n WN	1		Material name	Designation		WM
🔺 🕔 Steel	▶ 10S10+C	1.0711	I		Structural steels			
Structural steels	10S10+N	1.0711	Search material Ctrl+F		10510+C	10510+C		10711
	▶ 10S10+SR	1.0711			10310+0	10510+0		1.0711
Free-cutting steels	15S10+C	1.0/10			10510+N	10510+N		1.0711
Case-hardening steels	15510+N	1.0710			10S10+SR	10S10+SR		1.0711
	18S10+C	1.0712			Free-cutting steels			
	18S10+N	1.0712			10510	10510		1.0711
	18S10+SR	1.0712			10510	10510		1.0700
	25CrMo4+	1.7218			TUSP620	TUSPB20		1.0722
	25CrMo4+	.C 1.7218		$\odot$	Fine steel alloy			
	26Mn5+C	1.1161			SUS310S	SUS310S		1,4845
				~	Apply selection directly		OK	Cance

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

## Create sectional view - Processing plane

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

	×
The cut cannot be created with the active processing plane as it is perpendicular to the screen plane. Use screen plane instead	id?
Yes No	

#### Example:

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



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

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



### ESC key in Process Sketch mode

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

Process sketch	×
Apply sketch	Cancel
Esc	]
P HiCAD	×
Discard all changes and	stop processing sketch?
Yes	No

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

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

📩 🖹 🖄 🎎 📠 💽 🏗 💱	! ₩	¢
Designation	Value	-
🔺 🍯 (1) Sheet from sketch		
Sketch		
Sketch Process		
Sketch Load		
Sketch Exchange		
III Hide part		
► Show part		
Semi-finished product	No	•
•		$\mathbf{P}$



😕 Sheet from sk	etch X
- Sketch	
Sketch	3 V V
✓ Delete sketch	after creation
- Sheet paramete	ers hed product
	II
Thickness:	50 💌
Fitting direction:	+Z •
- General	
Referenced	
Sachnummer:	heet 🔹 🔽

## Standard processings

#### New standard processing

The new function Bores, Countersinks, Threads

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rocessing plane	
Select processing plane  rid  ndividual  rm  pe andard: ameter: 5  re depth  Depth: 20	
rid	• ■ •
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andard: 12 (DIN EN 20273 ameter: 5 pre depth Depth: 20	
ameter: 5 ore depth Depth: 20	•
pre depth Depth: 20	
Reference:	*
Point angle: 120	-
Reference:	
Perpendicular for laser cutting ①	<u> </u>
earance:	*
oduction	
ace of production: Workshop	•
epresentation	
Show axes	
Show bores	
✓ Show thread	

าลร	been	added	to the	Standard	Processings area
ius	DCCII	auucu		Junuaru	r roccooringo arca.

Point	
Only edge	
X-direction	
Mode:	Individual dista 👻
Number:	4 🔹
Distance:	20 🗸
Insertion position:	
Y-direction	
Mode:	Individual dista 🔻
Number:	4 🔹
Distance:	20 -
Insertion position:	
	Close
///12	
	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
× 🖉 🤍	

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

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

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

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

You will find the new function in the pull-down menu of the function 3-D Standard > Standard processings >



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

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

#### Internally referenced parts in externally referenced assemblies

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

If conflicts are detected, the following dialogue is displayed:

Inconsistent interna	lly referenced parts
The internally referer possibly due to upd	nced part Cylinder_0 {} {} exists in different versions in the drawing ating externally referenced assemblies).
- Clean-up of incon	isistencies
Adopt the latest	status for all identical parts
O Make older versi	ons into different referenced parts
Accent celection	for further inconsistent parts

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

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

#### Hence the recommendation:

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

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

Inconsistent internally referenced parts
The internally referenced part IntRef01 {} {} exists in different versions in the drawing (possibly due to updating externally referenced assemblies).
- Clean-up of inconsistencies
Adopt the latest status for all identical parts
O Make older versions into different referenced partsen machen
Accept selection for further inconsistent parts
Warning: Not all identical parts of the changed part can be updated as the superordinate assemblies are locked. Caution: There are now different versions of the identical part in the drawing!
ОК

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

Edit View Extras ISD					
		A 🛛 🕐 🕕		User	
	*	Description	Value	Comment	
Attributes		Update layer	$\checkmark$	Update layer for referenced parts	
Sketch HCM		Update line type	$\checkmark$	Update line type in referenced parts	
Itemisation				At System settings / Attribute management / Attribu you can specify in the Referencing column for each attribute whether it is transferred during the synchronisation of referenced parts. If the default behaviour is selected for the attribute, the behaviour specified here applies.	
Scales		Default behaviour for transferring attributes for referenced parts	Always transfer 🗸 🗸 🗸		
Sketches	_				
Novice configuration		Automatically update referenced parts after each change	Yes ~	Updating of identical parts	
Directories		Update identical parts of referenced parts before saving		Only important if referenced parts are not to be automatically updated	
Data save Identification		Recover dimensions and weld symbols		Recovers dimensions and weld symbols the bodies of which can no longer be assigned due to the updating, moving them to the top body	
Annotations		Handle outdated internally referenced parts	Update v	How should conflicts in identical internally referenced	
Calculations Graphic Visualisation Visualisation Fature 2-D Lines Miscellaneous Standard parts and processings POM V	E	Locking against processing	Update Always adopt the latest version		
		Lock referenced parts without KRA file against processing (Repl. Manager)	Split Always make them different referenced part	ock referenced parts with no KRA file against ocessing? (Repl. Manager)	
		Lock referenced parts for other users during processing?	Query Always ask whether to update		
		Lock non-updated, referenced parts against processing			
	-	Lock referenced parts if model drawing is read-only	No	Lock referenced parts against processing if model	

The ISD default setting Query.

## Improved algorithm for insulation hatching

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

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

#### Example:

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



#### Please note:

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

## Dimensioning in loop

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

## Display hidden lines of individual parts

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

Hidden edges dashed



Hidden edges normal

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



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

# **Point clouds**

## Major Release

## Point clouds in sectional and detail views

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

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





Example of a detail view

# **Feature Technology**

Service Pack 2

## Part structure of dependent parts

Starting with HiCAD 2025 SP2, it is no longer possible to change the part structure of assemblies that have been

marked as dependent using the **3-D Standard > New > Dependent part** <sup>1</sup> function.</sup>

### Transforming sub-parts into dependent assemblies

Updating of properties				
Apply and update article master assignment				
Position of sub-parts:				
Track if position matches	•			
Do not track				
Track if position matches				
Track, no manual change possible				

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

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

## **Major Release**

#### New feature functions

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

Service Pack 2

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

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



## Service Pack 1

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

Previously, an automatic constraint search was started using the function 3-D-Standard > HCM > Dragger, Motion

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

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

> HCM > Tools > Settings

# Automation

## Discontinuation

### Discontinuation of the ISD.PDM.API

Before carrying out a HELiOS update for an older HiCAD version, please note that from HELiOS 2022 onwards, 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 version used is at least version 2502.5 or 2601.1 or newer. If you are unsure whether you are using corresponding adaptations, please talk to your administrator or contact the ISD in case of doubt.

## Service Pack 2

## Support of the new Report Manager

The new class

ISD.CAD.BOM.ReportManagerExport

is available for the Report Manager.

## Point clouds

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

## Attach bend zone without flange

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

var sheet = Context.CreatePart(new BaseSheetCreator2(100, 150, 2));

var connectingEdge = sheet.Edges.ElementAt(1);

sheet.Apply(new AttachFlange2(connectingEdge, 1, 90));

## Referencing property

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

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

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

## **Drawing Management**

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

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

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

Scene.GetDrawingManagementICNState()

### Updating drawings

The HiCAD API provides the following command for updating drawings:

Scene.ShowAttributeDialog()

## DXF/DWG export of Sheet Metal parts

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

DXFSheetDevelopment3DSettings.AllowInvalidItemNumbers

## Service Pack 1

Insert new beam

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

ISD.CAD.Steel.BeamCreator?2

Feature "Create new" for Element Installation

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

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

RecreateElemInstall.cs

## Major Release

Into assembly



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

Node.ConvertToAssembly()

Update processed bolted/riveted elements

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

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

ISD.CAD.Data.Boltings.Update()

## Delete all item numbers

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

ISD.CAD.Steel.Itemization2.DeleteData

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

Standard part - Free weld seam symbol

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

Class WeldSeamSymbolCreator

Hatching in sectional view and cut-out

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- 111	Į.

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

InsulationHatching.

# Interfaces

Service Pack 2

## IFC interface

Transfer all attributes as user-defined properties

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

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



## Export of grids

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

Increased performance and improved import

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

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

## **TREPEDIA** import interface

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

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

<li>licenses loaded!</li>	
Desic Modules       HiCAD Extension Modules       CAD Interfaces         ICAD classic       IP Design Automation       IPC Import/Export         ICAD classic       IP Design Automation       IPC Import/Export         ICAD creator       IP Design Automation       IPC Import/Export         ICAD creator       IP Design Automation       IPC Import/Export         ICAD solution       IP Beams+Profiles       IPC Import/Export         ICAD VI / AV       IP Metal Engineering       3D DXF/DWG Export         Int Engineering suite plus       IP Steel Engineering Stairs + Railings       IP AICIS Export         Int Engineering suite plus       IP PleIm Engineering / 3-D Layout Plannir       IP Parasolid Import         Interfaces       IP PleIm Engineering / 3-D Layout Plannir       IP Parasolid Export         Interfaces       IP PleIm Engineering / 3-D Layout Plannir       IP Parasolid Export         Interfaces       IP PleIm Engineering / 3-D Layout Plannir       IP Parasolid Export         Interfaces       IP PleIm Engineering / 3-D Layout Plannir       IP Parasolid Export         Interfaces       IP Plot Manager       IP CATIA Import         Itel Engineering suite premium       IP Plot Manager       IP CATIA Import         Itel Engineering suite premium       IP Plot Manager       IP LM XML Export       IP LM XML Expo	Too rt nort oort
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TREPEDIA files are imported via Drawing > Insert Part > Exp. > 3-D Import

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

* * 🖈	> Windows (C:)	> HiCAD > sys >			~ 🜮	QuickSearch (CTRL+F)	
• <b>P</b>	lantParts	Name	Size	Type	Chang		
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к <b>ч</b> р	10	Cadenas		Folder	23/		
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• 👢 p	mu_hel	CAM_Interface		Folder	23/		
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					VDAFS files	s (*.vda)	
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					ACIS files (	*.SAT)	
					Parasolid fi	les (*.x t, *.xmt txt, *.x b)	
					Creo Param	netric files (*.asm *.g *.prt)	
					NX files (* r	ort)	
					SOLIDWOR	RKS files (* sldasm * sldprt)	
					Inventor fil	os (* int * iam)	
						es ( .ipc, .idin)	
					S-D DXF/D	wG mes (".dxi, ".dwg)	
					Solid Edge	nies (par,asm,psm)	
					MicroStatic	on tiles (*.dgn)	
					JT files (*.jt)	)	
					PLM XML f	iles (*.plmxml)	
					TREPEDIA f	files (*.tpa)	
							_

# Please note:

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

## STL and OBJ import

#### **Selection of units**

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

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



The unit settings can also be saved as Favourites.

Automatic optimisation as default setting

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

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



## FlexiCAD XML: Support of further 3-D objects

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

The following new elements have been added:

•	<userwall?></userwall?>	(User-	defined		wall):
	Enables the definition of walls that measurement module. These walls which are defined by a variable nu precisely in the XML file.	have been created manua s can have complex geom Imber of corner points. Th	ally by the user and netries, such as per ne coordinates of the	do not origin Itagonal or tr ese corner p	ate from the room apezoidal shapes, pints are specified
•	<b>ObjectType?</b> Used to define subtraction bodies t	Value="Void"> hat the user can already d	<b>(Subtract</b> efine in FlexiCAD.	ion	body):
•	<b><objecttype?< b=""> Va Enables the integration of 3-D solid.</objecttype?<></b>	alue="3dObject"> s that the user has created	<b>(3-</b> in FlexiCAD.	D	solids):

<ObjectType? Value="Surface"> (Surfaces):
 Enables the definition of user-defined surfaces.

## TopsGeo Export: Bend angle

When exporting sheet developments with the **ToPsGEO interface**, the CAMInt\_HICAD\_GEO.ini file is evaluated. This file has been extended so that you can now also output border lips with a positive angle. All folds with an opening angle of  $< 30^{\circ}$  are interpreted as border lips.

#### Extension in the CAMInt\_HICAD\_GEO.ini:

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



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

## STEP format

Import and export of materials

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

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

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

#### Performance increase for exporting sheets

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

## Service Pack 1

## DXF/DWG interface

#### Spaces in layer names for DXF export

In earlier versions of HiCAD, spaces in layer names were replaced by underscores for DXF/DWG export. Since AutoCAD supports spaces in names, this restriction has been removed.

Direction of the outer contour during DXF export in sheet developments

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

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

#### **COBUS** Default

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

It replaces the old HCADACAD\_COBUS.DAT file. The configuration cannot be changed.



## **OBJ** format

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



OBJ files can also be imported via Drag & Drop.

## Navisworks: Export of GUIDs

The Navisworks export now also assigns GUIDs to parts.

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

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

## HiCAD-Trepedia interface

In order to simplify the interface between HiCAD and the Trepedia program, the **Trepedia** function has been

added to the Extras I pull-down menu in the Further functions function group of the Steel Engineering Ribbon.

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

#### KISSsoft 2024

HiCAD supports the new version of the KISSsoft plugin.

## Major Release

## DSTV-NC - Output thread as marking

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

Jutput for	- File name
Selection list	HicAD     DSTV NC Filename.ftd
All parts	HELiOS document master HELiOS article master
Consider sheet metal parts	File extension: nc 🔻
Punch mark output	- Lettering
f Beam 🔰 Plate Extended	Parts: 🗹 Beams+Profiles 🗹 Plates 🗹 Contact surfaces
Destination: Write into block BO (bore)	📝 Beam 🔰 Plate Extended
	Position: Web
	x: 300 y: 20
Powder marking line output	- Others
🝠 Beam 🔰 Plate Extended	Order number: HiCAD attribute
Destination: Write into block PU (powder mark 🔻	Drawing number: HiCAD attribute
<ul> <li>Contours, entire length of contact edge</li> </ul>	Item number: HiCAD attribute   Do not fill in
Contours with length limitation	Part number: Do not write   Do not fill in
Max. line length: 500	Write comment
Intermediate line segments	Bore:
Length: 15 Distance: 500	Rectangular hole: Write into block BO (bore) ▼
Minimal marking at 2 corners	Max. diameter of slot: 38 mm
Line length for corners: 15	Max. diameter of round hole: 38 mm
O Punch marks at corners	✓ Write mounting bores
Diameter: 3	Thread: Write 💌
	Countersink: Write
Side marking (not for SM parts) Length: 3	Outer and inner contour:
	Maximum diameter:
	Max. length of approximation edges: 300 mm
	Write value for radius 0
	Evenand outer contour values to 5 columns

An	exam	ole:
7.01	Chain	pic.

Setting	BO block				
Marking	во				
	v	728.630	69.20m	0.00	0.00
	v	437.560	74.23m	0.00	0.00
	v	104.600	76.75m	0.00	0.00
Write	BO				
	v	728.630	69.20g	12.00	0.00
	v	437.560	74.23g	12.00	0.00
	V	104.600	76.75a	12.00	0.00

## IFC export - output as surface model

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

2					[
<ul> <li>plo</li> <li>pmu</li> <li>pmu_hel</li> <li>pneu</li> <li>Printdat</li> <li>Printdat</li> <li>Printdat</li> <li>RefTeile1</li> <li>RefTeile2</li> <li>RemoteSuppor</li> <li>Resources</li> <li>Script</li> <li>Splash</li> <li>struntime</li> <li>System Image</li> <li>System Image</li> <li>Szenen</li> <li>temp</li> <li>templates</li> <li>viewer_images</li> </ul>	Name bomtemplates Cadenas CADfix CAM_Interface DxfExport env_map_panoramas filter PDF3D sym	Size	Type Folder Folder Folder Folder Folder Folder	Changed on 20/09/2024 19 20/09/2024 19 20/09/2024 19 20/09/2024 19 20/09/2024 19 20/09/2024 19 20/09/2024 19 20/09/2024 19	IFC Parts to be exported: All Schema version: IFC2X3 Model view definition: Coordination view Prefer CONTOUR part Vunite sheets Unit of length: mm V Export standard parts V Export standard parts V Transfer part structure Export displayed parts only Output identical parts as referenced parts Consider IfcSite CS Show report Output all parts as a triangular mesh Part filter for geometry merging Accuracy Mode: As in drawing Distance: 0.1 V Polygon points per 3-D quadrant: 6 V
>				>	

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

# **Sheet Metal**

Service Pack 2

## Manufacturability check

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

The following new checks are now available:

#### Minimum flange length

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

#### Minimum bend radius

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

Distance between processings

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

Minimum diameter for standard bores

This test only applies to standard bores.

Minimum Z-fold height

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

## New design variant for panels

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

## TopsGeo Export: Bend angle

When exporting sheet developments with the **ToPsGEO interface**, the CAMInt\_HICAD\_GEO.ini file is evaluated. This file has been extended so that you can now also output border lips with a positive angle. All folds with an opening angle of  $< 30^{\circ}$  are interpreted as border lips.

#### Extension in the CAMInt\_HICAD\_GEO.ini:

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



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

## Service Pack 1

Synchronize all sheet developments and generate new dimensions

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

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

Update development	×
Part selection     All	
<ul> <li>Only changed</li> <li>Selection list</li> </ul>	
- Sheet area	
O Active sheet	
Create new dimensions and a	nnotations
ОК	Cancel

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

#### Rearrange dimensions and annotations

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



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

## Colouring of standard processings in the development

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



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

## Weight attribute

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

## Removal of the technology parameters

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

### DXF export

#### Manufacturability check for Sheet Metal constructions

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

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

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

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

#### Direction of the outer contour when exporting DXF files

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

## **COBUS** Default

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

## Major Release

Segments for pipes

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

	ጆ Pipes + Vessels	×
	- Insertion point	
	Insertion point	
	Type     Type     Sheet Metal O Solid	Segment
	- Parameters	
	Height (Z):	50 🗸
	Diameter:	50 💌
	Start angle:	0 🗸
	End angle:	90 🗸
27		
V1		
×1	- Sheet parameters	<u> </u>
	Use semi-finished product	Ũ
		II
	T1 1	

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



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

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



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

# Please note:

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

### Multiple selection of Sheet Metal parts

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

## Universal attaching of flanges

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



#### The option Without flange

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



#### Clearance 1

With the new option for the bend angle **Distance to connecting flange**, you can create border lips in conjunction with the connection type **Attach flange**, and hinges in conjunction with the connection type **Without** 

flange *flange*. The width of the attachment is determined by the connecting edge or 2 points.



(1) Bend zones without flanges, Distance to connecting surface 0,5

(2) Flange attached with a distance of 0,5 to the connecting surface

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

- Z-fold
- Attach flange to surface

## Selection behaviour changed

The selection behaviour for the Attach, Z-fold, Insert bend zone, Attach flanges along sketch and Flanges along sketch, to surface functions has been adapted to user requirements. This means that after selecting a connecting

edge, for example, you do not have to select the *icon* in the dialogue to change it. You can directly select a changed connecting edge as long as you do not switch to the dialogue. This behaviour applies to edges, points, surfaces and sketches.

# **Steel Engineering**

## Service Pack 2

## New Steel Engineering connection: Bracket (2202)

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



Revision of the functions for lengthening beams

The dialogue window of the function Steel Engineering > Lengthen > Change length <sup>20</sup>/<sub>20</sub> has been revised and modernised:

- Lengthen/shorten beams X
   Beam
   Select beams
   Length specification
   Change length:
   OK
   Cancel
   Apply
- The functions **Change beam length, via new total length** *W* and **Change beam length, to point** *have* been embedded in the dialogue and can be selected in the Length specification area.
- In addition, it is possible to use the local icon to adopt a new total length from a reference beam.
- The new dialogue allows you to select several beams at once and change their length.

## Manufacturability check in the Design Checker

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

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

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

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

## Bar list: Non-numeric values for item texts

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

### Steel Engineering favourites in imperial units

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

## Service Pack 1

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

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



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

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

### Cross-bracing 2602

#### Extension for the turnbuckle with blade screw

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



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

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



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

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

#### **Closed turnbuckle form**

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

1st diagonal     2nd diagonal     Ist diagonal	DIN 1478 SP AE ×
Default connection         Distance from beam         ✓         I. Create welding end (L)         ✓         2. Create welding end (R)         Boltings:         ✓	Turnbuckle DIN 1478 SP AE DIN 1480 SP AE DIN 1480 SP AE (new) R
(N): St  III 1478 SP AE, St	
7	OK Cancel

## HiCAD-Trepedia interface

In order to facilitate access to the interface of HiCAD with the **Trepedia** program, the **Trepedia** function has been added to the **Steel Engineering > Further functions** function group, in the pull-down menu of the **Change** 

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

## New setting for the stairwell contour in the Staircase Configurator

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



## Attribute calculation

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

For beams and profiles there are the new attributes:

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

and for plates and sheets:

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

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

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

Here you have the option to transfer the calculation of the minimum bounding rectangle not only to the new geometry attributes (§WBL, §CBL, §SBL, §VBL or §CBA, §SBA, §VBA), but also to the standard attributes (§01, §10, §18, §20).
Due to the extensions in the Configuration Editor, the Weight calculation tab of the Steel Engineering Settings dia-

logue (Steel Engineering > Further functions > Settings (Was no longer needed and was removed. The settings on the tab only had a temporary effect until the next restart of HiCAD.

The dialogue window now looks like this:

Beams Beams	Plates	O Gratings	Glass panes
			1
🔘 Simplified	Exact	t	O Axis only
Tracing lines	S.E. axes	Axis end points	Beam annotation
	ntion for boom series	Class such als	

#### Insert new beam with processing plane suggestion

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

🖉 Beam 🛛 🕹 🗙	V1
- Cross section	
🗲 🍢 🌜 🊿	
- Catalogue beam	
HEA 400	
Point	A1 /
Select end point	45°
Rotation:	
- Fitting options	
Insertion in plane (i)	180
Distance to insertion plane: 0	
▲#₩	
- Parameters	
Length: 1200 -	

### Manual change of the article number for semi-finished products

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

In this context, an input field for manually changing the article number has been added in the General area of the

Insert new beam

dialogue, namely, for the Cross-sections Prototype beam, Beam from (multi-part) sketch,

function. If the corresponding checkbox is active, the article number will adop-

and for the **Rectangular plate** ted from the catalogue.

Referenced	✓ BOM-relevant
Article number:	HEA 200 🔻
	Apply immediately

Further options for beam insertion from sketches

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

plane And Process sketch have been added for this purpose.

🔀 Beam	×
- Cross section	
- Sketch beam Sketch Select axis point ma	nually
Delete sketch after creation	
Weight per length:	0 •
Commercial weight per length:	0 -
Surface area per length:	0 •
Material:	-

# Major Release

#### Redesigned beam insertion dialogue

The functions for inserting series beams via Steel Engineering > New have been added to the dialogue of the Insert

new beam *I* function:

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

💌 Beam	×
- Cross section	
- From HELiOS	

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



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

a pipette symbol

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



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

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

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

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

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



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

### K-bracing with I-beams

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

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

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



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



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

#### Define grid

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



### Staircase configurator - Simplified preview

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



(1) Simplified preview, (2) Result

Beam insertion - Revised dialogue

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

- Fitting	- Fitting
Select start point	Select start point
Rotation: 0 •	Rotation: 💿 💿 0 👻 🮯
- Parameters	Insertion in plane (1)
Length: 1000 -	Distance to insertion plane: 0
Fitting depth:	

### Base plate + Anchor plate (2101)

#### Threaded stud

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

- Threaded stud			_	- Threaded stud		
Semi-finished product	t: DIN 976-1-M14-A	4-70 (A4		Semi-finished product:	DIN 976-1-M14-A4-70	) (A4
Nut:	DIN 934-M10-10	(10)		Nut:	DIN 934-M10-10 (10	)
Washer:	DIN 7349-10.5-St	(St)		✓ Washer:	DIN 7349-10.5-St (St	)
Assign to assembly	y (Welded plate )			Assign to assembly	(Welded plate)	
Designation		Ite	Comment	Designation	Ite	Comment
BASE ANCHO	ORPLATE 2101			BASE_ANCHORPLAT	E_2101	
🖌 🐠 🍖 Main assem	ibly		Assembly	4 🔳 🍖 Main assembly		Assembly
Assembl	V IPE 200		Assembly	🔺 🔳 🍖 Assembly IPE 20	00	Assembly
🔳 漏 IPE 2	00		I-beam with para	🔳 🌆 IPE 200		I-beam with para
🔳 🛑 BI 16			Plate	🔳 🛑 BI 16		Plate
🔳 🛑 BI 16			Plate	🔳 🛑 BI 16		Plate
🔳 🛑 BI 16			Plate	🔳 🛑 BI 16		Plate
🔳 📒 BI 16			Plate	🔳 🛑 BI 16		Plate
💶 🛑 BI 16			Plate	🔳 🛑 BI 16		Plate
🔳 🛑 BI 16			Plate	🔳 🛑 BI 16		Plate
\rm 🎩 Weld	seam			💶 🚣 Weld seam		
🔳 📒 BI 20			Plate	🔳 🛑 BI 15		Plate
🔺 🜗 🍖 Assembl	ly Anchor plate		Assembly	assembly Anch 🔮 📲 🖌 🖌	or plate	Assembly
🕚 漏 HEB	100		I-beam with para	🜗 漏 HEB 100		I-beam with para
👂 💶 🐮 Stand	dard part group			💶 🛑 BI 10		Assembly
👂 🕕 🚼 Stand	dard part group			4 💵 🐏 Assembly Anch	or plate	I-beam with para
👂 🕕 🚼 Stand	dard part group			🕨 💶 💱 Standard pa	rt group	
🔳 🛑 BI 10			Plate	👂 💶 💱 Standard pa	rt group	
🔳 🛑 BI 20			Plate	👂 💶 💱 Standard pa	rt group	
				🔳 🛑 BI 30		Plate

#### Fixing holes

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

💌 Base	e plate +	Anchor pla	te (2101)						×
Plates	Fixing	Bore grid	Galvanization	Weld seams	Filler plate	Shear connector	Head studs	Ribs	
Boltin Cr (1) Di (2) Di (3) Di	ngs Fi reate ameter: stance () stance ()	8     ()       ():     20       ():     20	Anchor plate)						
<b>1</b>	3				<b>=</b>	<b>†↓</b> P	review	OK	Cancel

#### Weight from rectangular area of development

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

This does not apply to gratings and glass panes.

## End plate (2102) - Chamfered front plate

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

💌 End plate	e (2102)								×
Front plate	Boltings	Bore grid	Galvanization	Weld seams	Filler plate	Base			
Semi-finishe	ed produc	BI 15 ( S23	5JR )	Rota	ate (Flat steel)	)			
Vertical:	Pro	jections bey	ond beam	•		-			
(1) Heigl	ht:		150		St.	-(4)-		6	
(2) Top:			10	•		17		4	
(3) Botto	om:		10	•		- 12		<u>+</u>	
Horizontal:	Pro	jections bey	ond beam	•				(2)	
(4) Widt	h:		340				Y		
(5) Left:			10	•	1 -				
(6) Right	t:		10	•				0	
Clearance to	o beam:		0	•				Ŷ	
Inclination a	angle to b	eam, about a	axis Y: 0	<u> </u>				1	
Inclination a	angle to b	eam, about	axis Z: 0	-	<u>_</u>			- <b>*</b>	
Process co	rner: none	• • R	adius: 5	Ŧ					
Process cor Fillet corne	rner: none	e ine	255						
Chamfer co	orners								
				_					
16 ☆				<b>≠</b> †∔		Preview	OK	Cano	el

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

# **Drawing Management**

# Service Pack 1

### HELiOS settings in the Configuration Editor

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

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

# **Metal Engineering**

# Service Pack 2

## LogiKal-Import: Detail levels of cross-sections

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

💌 Fitting optio	ns X
- Level of detai	r
With all pr	ocessings
O With cuts a	and notches
O Without pr	rocessings
✓ With hardy	vare
Cross-sections:	Exact 🔹
	Exact
- Name of top	Level 1
Project nar	Level 2
O Position	Level 3
O Project nar	Level 4
O Brief descr	Level 5
[	OK Cancel

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

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

# Service Pack 1

LogiKal import, with hardware

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



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





LogiKal version 12.4. or higher is required for this.

# Major Release

### Improved algorithm for insulation hatching

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

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

# Sheet metal clamp

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

Civil Engineering functions	άx
a <b>→</b> 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1, 1,	
Steel Engineering	-
Civil Engineering, general	
Metal Engineering / Facade Engineering	
▲ 3-D Metal Engineering	
Metal Engineering beam/profile connections	
Mullion-Transom connection	
Mullion connection (1314)	
🚣 Fixing bracket (1410)	
🔔 Fixing bracket (1420)	
🚄 Fixing bracket (1430)	
Base plate/Top plate	
Sheet metal clamp	
2-D Metal Engineering	
Sheet Metal	•
Size: (Small icons)	
	<u> </u>
Processor European European Civil Equipmention for the	
Panoramas Exploded view Function search Civil Engineering functi	ons

# **Layout Planning**

# Service Pack 2

### External references for the sketch

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

😕 Element Install	ation — 🗆 🗙
- Sketch ———	
Sketch	
Use sketch fro	om drawing
Delete sketch	after creation
Туре ———	
Variant:	ALUCOBOND Tray panel SZ-20 (with accessories)
Level of detail:	Exact
Offset, active: 🕕	0 •
Offset, global: 🕕	0 •
Joint width tow	ards sketch line 🛛 📀
Global settings E	xtended settings Bracing Extensions for SZ-20 Standard
<ul> <li>Parameters</li> <li>Semi-finished pr</li> <li>Create standa</li> </ul>	oduct: ALUCOBOND 4mm I503 Champagne metallic - ALUCOBOND 4mm

# Service Pack 1

#### Window connection with additional sheet metal - Base point

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



Window connection with connecting plate

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



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

# Major Release

Transfer parameterisation



You can use the Transfer parameterisation

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



(1) Source: ALUCOBOND Tray panel SZ-20 (with accessories), with Attic, long,

(2) Target: ALUCOBOND SZ-20, Standard,

(3) Elements after transfer of Attic, long, Joint width and Offset

# **Profile Installation**

# Service Pack 2

### Annotations on exact representation

To display more than just the annotations on uncut profiles in a drawing, there is now the new function Add des-

#### ignations, cut contour view



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

Fitting point for contour

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



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

## Edge profiles for roof inclinations

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



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

# **Plant Engineering**

# Service Pack 2

Save settings for lists

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

Pipe shortening		Texto	ojects	2-D [	)rawing eler	nents	Item numbers			
Auton	natic dimensionin	g	Texts/Lines	Optim	ise positioning	C	onnections		Lists	Symbols
File										
ListDe	f.ild.xml								Oper	n )
List	output to drawi	ing		List output to	file	Save				
Inclu	ude accessory :	sets					_			
Da	lways; regardle	ss of BOM-	elevance othe	parts					Save	as
Inse	rt sub-heading	s for assem	blies							
Conten	ts							Lavout		
l ist tvn		Pipe part	ist (Drawing)				-	Fra	me	
Listip		Dino ROM	(oroning)					Ve	tical borders	
Headin	ig	Pipe DOM							incar borders	
Footer								I HOI	rizontal boro	ers
No.	Column heade	ər	Column	Attribute	Decima	al places S	Sort	Colum	nn layout	
2	Designation			POPAT/PEN			크레	Isigna	atures	•
2	Designation		2	DBAT BENE	ENNUNG)			DAut	to-fit cells	
3			4	%DBAT(NENP	WVEILE)		-		Text pos	ition
4	Qty.		5	%PART_QUA	NT	0	-		Line parar	neters
5							-		Text parar	neters
	1						<u> </u>			
	Inser	trow			Attribute					
	Delete	e row		Ed	litable column					

### Correct guidelines after creation

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

# 🕖 Please note:

Overlaps of a guideline with itself are not corrected.



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

#### Make guidelines visible after moving ~Parts and ~CEdge

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



#### Defining nominal guideline diameters that differ from the nominal pipeline diameter

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



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

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

#### Standardised context menus

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

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

Improved copying of pipe parts

Copying pipe parts with the Copy pipe parts



function can be a quick method for inserting new parts into a

pipeline. This function has therefore been added to the context menus of pipe parts. The Copy pipe parts



and the **Move pipe parts** function have been added to the context menu of multiple selections.

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



Pipe parts copied from Pipeline\_0001 (1) with guideline to Pipeline\_0002 (2) without guideline

### Straight pipes according to EN 10216

The following variants have been added:

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

#### Length allowances in Bills of Materials

Length allowances are now listed separately in the Bills of Materials for pipeline planning. The configuration files for BOMs included in the scope of delivery

- anlagenbau\_rl.rm\_settings (Used by Plant Engineering > Evaluation > Bill of Materials, for entire drawing
- anlagenbau\_szn.rm\_settings (Used by Plant Engineering> Evaluation > Bill of Materials, for active pipeline
- HiCAD\_Anlagenbau.rm\_settings (Used by Drawing > Itemisation/Detailing > Bill of Materials, for entire drawing
- HiCAD\_PipeBook.rm\_settings (Used by Isometry + Pipe Spool Drawing > Edit lists > EditPL... > Pipe book

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

Qua	analy isc Structure List												
Nur	mber Article number	Designation	Standard	Pipeline	Material	Nominal diam	Wall thickness [mn	Length [mm]	Length allowance [mm]	Total length [mm]	Angle [°]	Weight [kg]	Total weight (
1	E HEA 200		DIN 1025-3		\$235JR			1600,00			0,00	67,68	67,68
1	E HEA 200		DIN 1025-3		\$235JR			1750,00	)		0,00	74,03	74,03
1	E HEA 200		DIN 1025-3		\$235JR			3100,00	)		0,00	131,13	131,13
1	SN-027642	Flanged tube FME4 sing	EN10357	Pipeline_0001	X2CrNiMo17	300	2,60	1300,00	21,00	1321,00	0,00	27,55	27,55
1	Pipeline_0002			Pipeline_0002		200						271,30	271,30
10	SN-026643	Loose flange Type 02-32	EN 1092-1/02/32/PN 16	Pipeline_0002		200					0,00		
1	SN-027634	Flanged tube FME4 sing	EN10357	Pipeline_0002	X2CrNiMo17	200	2,60	5800,00	69,00	5869,00	0,00	82,46	82,46
1	SN-027667	Flanged tube FME doub	EN10357	Pipeline_0002	X2CrNiMo17	200	4,00	8600,00	168,00	8768,00	0,00	188,84	188,84
	2												842.00

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

## Isometry and pipe spool drawing

Merge Add/Remove elements and Update dialogues

Add/remove/update isometry el	Add/remove/update isometry elements			
	Element exists	Rebuild element		
Coordinate axes	<b>V</b>	Direction acc. to Active view		
North arrow	<b>v</b>	Direction acc. to Active view		
Dimensioning		$\checkmark$		
Rise triangles				
Auxiliary lines for arcs				
Part numbers	<b>v</b>	<b>V</b>		
Connection numbers		$\checkmark$		
Connection symbols	<b>v</b>			
Part list	<b>v</b>	<b>v</b>		
Length list	<b>v</b>	<b>v</b>		
Connection list	✓	<b>v</b>		
All		✓		

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

# Service Pack 1

## VarDbAttConfig deleted

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

### Plant Engineering Settings: Part search optimised

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

Part placing on br	anching points	Y	Fasteners	Flange co	nnection, bolted
Part insertion	Part selection	P+ID symb	ool assignment	Link to P+ID	Bills of Material
Part search	P+ID attr	ibute assignme	ent Y	Weld gap	Straight pipe
Attributes used as sea	ch criteria for the	connection of	a part.		
Part type	Connection	t][Nominal d	iam]Outer diar	meter (Wall thickness)	
Set all					
Reset all					
3-way valve					_
4-way valve					
Blank flange					
Branch		$\square$			1
D Cap					
Comer valve					
H Cross-shaped bran	nch 🔽				
Double knee					
Elbolet					
Elbow					
IIII Fastener, symmetr	ic 🔽	$\square$			
N	17	2			
When inserting on	connection, use	Pressure as sea	arch criterion.		
Also use Outer dia	meter 2 as search	criterion for pip	e clamps.		
Default setting				Connection ty	pes

From now on, the following applies:

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

### Transfer of ROHR2 standard designations to ROHR2

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

#### # Format:

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

#

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

etc.

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

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



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

#### Isometry and Pipe spool drawing

#### **Update text parameters**

After changing the text parameters in the Isometry settings, it is now easy to apply the changes to the tags. From

HiCAD 30.1, the **Complete update** function also updates the text and line parameters of the text tags.

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



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

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

Isometry: Adjusting the parameters for rise triangle hatching

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

Pipe shortening	Text object	ts	2-D Dr	awing elements	Item numbers		
Automatic dimensioning Texts/Lines C		Optimis	se positioning	Lists	Symbols		
Dimension line distances				Dimension figure height)			
1st dimension line Further	dimension lines: Ri	ise triangles:					
25 20	2	5		2,5 💌 mn	n		
Only write dimension figure	next to rise triangles						
Determine reference line ad							
Vetical institutes			(	1			
Vertical rise triangles	Line parameters		Draw triangle	lies	meters		
Uraw thangle:			Draw triangle				
Range of angles	5-	85	Range of angle	s <u>5</u> -	85		
Dimension rise angles			Dimension rise	angles			
Dimension hypotenuse			Dimension hypo	otenuse			
	0	_    "	Hatching line a	0			
ridici ing ine spacing	2	-	riatering inte s				
Triangle distance	3	┹┓╽┏	Triangle distant	ce 13	-		
Complete hatching		-	Complete hato	hing	-		
Triangle length in %	20	<b>_</b>	Triangle length	in % 20	-		
	1						
Position of dimension lines			mension types				
Always in vertical plane			Part dimensions	1			
Dimensions in all views			Dimension	ns for all parts			
Repeat dimensions			Also dime	nsion weld gaps			
Draw angular dimensions			Unimension	ns for points of action			
Angular dimensions			Dimensions for	points of action			
Invert Angular dist	tance 0,2		Dimensions for	points of action			
					)		
Default							
When a rise triangle is partially hatched, the hatched area is specified exclusively via the length of the base side of the rise triangle, and this is done proportionally in % to the length of the base side of the entire rise triangle. Before HiCAD 2025 SP1, the specification was in the length unit of the drawing.



lmportant:

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

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

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

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



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

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

#### **Isometry: Arrow functions**

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

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



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

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



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



Clicking on the desired location ends the function.

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



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

Isometry: Pipe part type headings in BOMs

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



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

Pipe shortening	Text objects	s 2-D Dr	rawing elements	ltem r	numbers
Automatic dimensioning	Texts/Lines	Optimise positioning	Connections	Lists	Symbols
			C	0	
C:\HiCAD\sys\listdef.ild.xml				Oper	1
List output to drawing		ist output to file		Save	as
Include accessory sets			00		

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

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

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

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

Both files are structured in the same way. First, headings are defined, e.g.: H2: Flanges

The heading "H2" is therefore flanges. The colon is essential to distinguish the designation of the heading from the heading itself. You can then assign the heading to pipe part types, for example, like this: 5100010;Flange;H2

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

# Major Release

### ELGEF / PROGEF - Further variants

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

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



 $^{\ast 1}$  The previous ECOFIT\_LOOSEFLANGE\_SDR11 and ECOFIT\_LOOSEFLANGE\_SDR17 variants are summarised in the ECOFIT\_LOOSEFLANGE.VAA file.

 $^{\ast 2}$  The previous variants ECOFIT\_STUB\_FLANGE\_A\_SDR17 and ECOFIT\_STUB\_FLANGE\_B\_SDR17 are summarised in the ECOFIT\_STUB\_FLANGE\_SDR17.VAA file .

# Air duct parts

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



### Pipe spool drawing - Auxiliary geometries for elbows

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

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



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



### DBPlantDataImport

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

Attribute designation	Attribute name	Selected init value	Unit
Resourcing	COMPONENT_PROCUREMENT	Purchase	
Unit of quantity	COMPONENT_QUANTITY_UNIT	Piece	
Part type	COMPONENT_TYPE	Raw-part+Plant-design	
Thickness	DICKE	10	mm
Width	BREITE	5	mm

The unit is fixed and cannot be changed.

# **Catalogue Editor**

# Service Pack 2

# Catalogue: US anchors

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

- A		(1.500 LD) N	<b>1 40</b> 17 1 546		11-merile 10	av   8									
User-defined fasteners		ID M	OE STATUS	BZ	Art. no.	SIZE	MATERIAL	OBERFL	ТҮРЕ	ICON	GEW	DN	Р	LN	TFIX_MA
Ser-defined anchors	1	1		KB1 3/8x2 1/	2221454	3/8"x2 1/2	St	zinc-plated	St zinc-plated		3/8	0.375	0.0625	2.5	2.
B • Eternit	2	2		KB1 3/8x3	223145	3/8"x3"	St	zinc-plated	St zinc-plated		3/8	0.375	0.0625	3	
🖻 🍖 Fischer	3	3		KB1 3/8x3 3/	2221456	3/8"x3 3/4	St	zinc-plated	St zinc-plated		3/8	0.375	0.0625	3.75	3.7
B Solt anchor HSA	4	4		KB1 3/8x5	223145	3/8"x5"	St	zinc-plated	St zinc-plated		3/8	0.375	0.0625	5	
Bolt anchor HSA-F HDG	5	5		KB1 1/2x3	226777	1/2"x3"	St	zinc-nlated	St zinc-plated		1/2	0.5	0.0769	3	
Bolt anchor HSA-LW, long	6	6		VP1 1/2v2 2/	2221450	1/2"-2 2/4	C+	zinc-plated	St zinc-plated		1/2	0.5	0.0769	2 75	2.75
	0	7		KD1 1/2A3 3/	2231430	1/2 3 3/4	ы сь	zinc-plated	St zinc-plated		1/2	0.5	0.0709	5.15	5.7.
Bolt anchor HST		/		KB1 1/2X4 1/	223145:	1/2 x4 1/2	St	zinc-plated	St zinc-plated		1/2	0.5	0.0769	4.5	4.:
Bolt anchor HST3	8	8	•	KB1 1/2x5 1/	2231460	1/2"x5 1/2	St	zinc-plated	St zinc-plated		1/2	0.5	0.0769	5.5	5.5
Bolt anchor HST3 BW	9	9	•	KB1 1/2x7	223146	1/2"x7"	St	zinc-plated	St zinc-plated		1/2	0.5	0.0769	7	7
Bolt anchor HST3 HICAD2025	10	10		KB1 1/2x7	3684426	1/2"x7"	St	zinc-plated	St zinc-plated		1/2	0.5	0.0769	7	7
Bolt anchor HST4	11	11		KB1 5/8x4 1/	2231462	5/8"x4 1/4	St	zinc-plated	St zinc-plated		5/8	0.625	0.0909	4.5	4.5
Bolt anchor HST4-R BW	12	12		KB1 5/8x4 3/	223146	5/8"x4 3/4	St	zinc-plated	St zinc-plated		5/8	0.625	0.0909	4.75	4.75
Bolt anchor KB1	13	13		KB1 5/8x6	2231464	5/8"x6"	St	zinc-plated	St zinc-plated		5/8	0.625	0.0909	6	e
Bolt anchor KB-TZ2	14	14	•	KB1 5/8x7	225655	5/8"x7"	St	zinc-plated	St zinc-plated		5/8	0.625	0.0909	7	ī
Ceiling anchor HK L	15	15		KB1 5/8v8 1/	223146	5/8"v8 1/2	St	zinc-plated	St zinc-plated		5/8	0.625	0.0909	85	81
Undercut anchor HDA bis HiCAD202	15	16		KD1 3/4-4 3/	222146	2/4"-4 2/4	54 C4	zine plated	Stains plated		2/4	0.025	0.0505	4.75	4.71
	10	10		KD1 3/4x4 3/	2251400	5/4 x4 5/4	SL	zinc-plated	St zinc-plated		5/4	0.75	0.1	4.75	4.75
Undercut anchor HDA-PF     Undercut anchor HDA-PR	17	1/		KB1 3/4x5 1/	223146.	3/4"x5 1/2	St	zinc-plated	St zinc-plated		3/4	0.75	0.1	5.5	5.5
	18	18	•	KB1 3/4x7	2231468	3/4"x7"	St	zinc-plated	St zinc-plated		3/4	0.75	0.1	7	7
Hollow cavity anchor HKH	19	19	•	KB1 3/4x8	2231452	3/4"x8"	St	zinc-plated	St zinc-plated		3/4	0.75	0.1	8	8
	20	20		KB1 3/4x10	2256552	3/4"x10"	St	zinc-plated	St zinc-plated		3/4	0.75	0.1	10	10
	21	21		KB1 3/4x12	223145	3/4"x12"	St	zinc-plated	St zinc-plated		3/4	0.75	0.1	12	12

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

eel Engineering 🔹 🛃	• •	1 1	<b>₩</b> #  Ω Ω	≥  <b>*e *e *e </b> ⊺e	h e	8			
E Sactory standards		ID M	OC STATUS	Designation	Size	DN	DB	TOL	8
User-defined building materials	1 1	1	•	Ø 0.25	1/4	0.25	0.25		
User-defined bores	2	2		Ø 0.375	3/8	0.375	0.375		
	3	3		Ø 0.5	1/2	0.5	0.5		
	4	4		Ø 0.625	5/8	0.625	0.625		
🗉 🔶 SFS	5	5		Ø 0.75	3/4	0.75	0.75		
<ul> <li>User-defined bores for blind rivet nuts</li> <li>User-defined bores for insert nuts</li> </ul>									
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### Gratings (Steel Bar Grating)

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

	1 25 VB		1 100	I to to to to Do Do 100							
ineering 🖉 🛃			1 H   52 52		3						
Processings, general Semi-finished products		ID NO	STATUS	BZ	SIZE	MATERIAL	OBERFL	TYPE	DSTV	No. of Bearing Bars	
Plates	1	1		TTW4-ADA T X1/6	1 X1/8	ASTM ATOTICS Type B		ASTM ATOTICS Type B		2	
Gratings	2	2	•	11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		2	
Standard Duty	3	3		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		2	
B ♦ Imperial	4	4		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		2	
11W4-ADA	5	5		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		2	
19W4-SD	6	6		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		3	
B Metric	7	7		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		3	
24W102-Close-Mesh	8	8		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		3	
30W102-SD	9	9		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		3	
B • Imperial	10	10	•	11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		3	
15HW4-HD Close-Mesh	11	11		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		4	
30HW4-Wide-Gap	12	12		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		4	
38HW4-Extra-Wide Gap	13	13	•	11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		4	
24W102-HD Close-Mesh	14	14	•	11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		4	
	15	15		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		4	
60HW102-Extra-Wide Gap	16	16		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		5	
Grid (DIN 24537)	17	17		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		5	
Wood	18	18		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		5	
Cold rolled sections	19	19	•	11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		5	
	20	20		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		5	
	21	21		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		6	
	22	22		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		6	
	23	23		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		6	
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	24	24		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		6	
	25	25		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		6	
	26	26		11W4-ADA 1"x1/8"	1"x1/8"	ASTM A1011CS Type B		ASTM A1011CS Type B		7	
										_	

### Grating steps (Steel Stair Treads)

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

eel Engineering	•	A C	) 🕲 🗈 To I	H #   Q Q   10 10		@ <b>T</b>					
B Factory standards		ID	NOT STATUS	BZ	SIZE	MATERIAL	OBERFL	ТҮРЕ	DSTV No. of Bearing Bars	Н	Α
Fasteners	1	25		11W4 1"x3/16"	1"x3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	9	1	0.6
Plant Engineering	2	31		11W4 1"x3/16"	1"x3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	9	1	0.6
Processings, general	1 3	26		11W4 1"x3/16"	1"x3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	11	1	0.8
Semi-finished products     Sheet tools	4	32		11W4 1"x3/16"	1"x3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	11	1	0.8
Steel Engineering	5	27		11W4 1"x3/16"	1"x3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	13	1	0.7
Steel Engineering Standard Parts	6	22		11W/4 1"v2/16"	1"v3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W/A	12	1	0
DIN 24531	7	20		11114 15-2/10	11,0/10	ASTM ATOTICS Type D		ACTM A1011CS Type D (11W4	15		0.
🗉 🔶 Vulcraft		28		11004 1 x3/16	1 x3/10	ASTM ATUTICS Type B		ASTM ATUTICS Type B (TTW4	15	- 1	0.
Imperial Metric	8	34	•	11W4 1"x3/16"	1"x3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	15	1	0.
B 🔶 Elbow	9	29	•	11W4 1"x3/16"	1"x3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	16	1	0
Tensioning element	10	35		11W4 1"x3/16"	1"x3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	16	1	0
System settings	11	30	•	11W4 1"x3/16"	1"x3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	18	1	0
Textures and Colours	12	36		11W4 1"x3/16"	1"x3/16"	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	18	1	0.
	13	13		11W4 1 1/4"x3/1	1 1/4"x3/16	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	9	1.25	0
	14	19		11W4 1 1/4"x3/1	1 1/4"x3/16	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	9	1.25	0.
	15	14		11W4 1 1/4"x3/1	1 1/4"x3/16	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	11	1.25	0
	16	20		11W4 1 1/4"x3/1	1 1/4"x3/16	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	11	1.25	0.
	17	15		11W4 1 1/4"x3/1	1 1/4"x3/16	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	13	1.25	0.
	18	21	•	11W4 1 1/4"x3/1	1 1/4"x3/16	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	13	1.25	0.
	19	16	•	11W4 1 1/4"x3/1	1 1/4"x3/16	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	15	1.25	0.
	20	22		11W4 1 1/4"x3/1	1 1/4"x3/16	ASTM A1011CS Type B		ASTM A1011CS Type B (11W4	15	1.25	0.

### Würth fixing anchor added

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

an tien eguas fieldos setangs	- 4	•	1 B 🔥 🖬 1	si⊇ @ 10 10 10 10 10 10 10	8									
Factory standards	- P1	ID	AOT STATUS	BZ	Art. no.	SIZE	MATERIAL	OBERFL	TYPF	ICON	DN	Р	LN	TFIX MAX
User-defined building materials	1	1		W-FAZ/S-M8-11/65	592825801	M8	St	zinc-plated	St zinc-plated		8	1.25	65	11
User-defined processings	1 2	2	•	W-FAZ/S-M8-10-21/75	592820801	M8	St	zinc-plated	St zinc-plated		8	1.25	75	21
<ul> <li>Set settings</li> <li>User-defined moulding tools</li> </ul>	2	3		W-FA7/S-M8-15-26/80	592820801	MR	St.	zinc-plated	St zinc-plated		8	1.25	80	26
User-defined dished ends				W FAZ/S MR 20 41/05	502020000	MO	G.	aine plated	Chains plated		0	1.25	00	41
User-defined semi-finished products      Second semi-finished products      Second secon	4	4		W-FAZ/S-IVI8-50-41/95	592820805	NIO	SL	zinc-plated	St zinc-plated		0	1.25	95	41
Vser-defined fasteners	5	5		W-FAZ/S-M8-50-61/115	592820805	M8	St	zinc-plated	St zinc-plated		8	1.25	115	61
User-defined anchors	6	6		W-FAZ/S-M8-100-111/16	592820810	M8	St	zinc-plated	St zinc-plated		8	1.25	165	111
Elemit	7	7		W-FAZ/S-M10-10/70	592825101	M10	St	zinc-plated	St zinc-plated		10	1.5	70	10
🗄 🔶 Fischer	8	8		W-FAZ/S-M10-20/80	592825102	M10	St	zinc-plated	St zinc-plated		10	1.5	80	20
B      Cemplates	9	9	•	W-FAZ/S-M10-10-30/90	592821001	M10	St	zinc-plated	St zinc-plated		10	1.5	90	30
8 - S WURTH	10	10		W-FAZ/S-M10-15-35/95	592821001	M10	St	zinc-plated	St zinc-plated		10	1.5	95	35
Fixing anchor W-FAZ/S	11	11		W-FAZ/S-M10-20-40/100	592821002	M10	St	zinc-plated	St zinc-plated		10	1.5	100	40
Fixing anchor W-FAZ/HCR	12	12		W-FAZ/S-M10-30-50/110	592821003	M10	St	zinc-plated	St zinc-plated		10	1.5	110	50
Hook bolt	13	13		W-FAZ/S-M10-50-70/130	592821005	M10	St	zinc-plated	St zinc-plated		10	1.5	130	70
B Vser-defined studs	14	14		W-FA7/S-M10-75-95/15	592821007	M10	Sł.	zinc-plated	St zinc-plated		10	15	155	95
• User-defined sockets	14	15		W FAZ/S M10 100 120/	502021007	MIO	5t	zine plated	St zine plated		10	1.5	190	120
Wer-defined nuts     User-defined rivets	15	15		W-FAZ/S-M10-100-120/	592821010	MIU	St	zinc-plated	St zinc-plated		10	1.5	180	120
	16	16		W-FAZ/S-M10-150/230	090452100	M10	St	zinc-plated	St zinc-plated		10	1.5	230	230
	17	17	<u> </u>	W-FAZ/S-M12-10/85	592825201	M12	St	zinc-plated	St zinc-plated		12	1.75	85	10
	18	18	•	W-FAZ/S-M12-20/95	592825202	M12	St	zinc-plated	St zinc-plated		12	1.75	95	20
	19	19		W-FAZ/S-M12-15-35/110	592821201	M12	St	zinc-plated	St zinc-plated		12	1.75	110	35
	20	20		W-FAZ/S-M12-20-40/115	592821202	M12	St	zinc-plated	St zinc-plated		12	1.75	115	40
	21	21	•	W-FAZ/S-M12-30-50/12	592821203	M12	St	zinc-plated	St zinc-plated		12	1.75	125	50
	22	22		W-FAZ/S-M12-50-70/14	592821205	M12	St	zinc-plated	St zinc-plated		12	1.75	145	70
	23	23		W-FAZ/S-M12-65-85/160	592821206	M12	St	zinc-plated	St zinc-plated		12	1.75	160	85
	24	24		W-FAZ/S-M12-85-105/18	592821208	M12	St	zinc-plated	St zinc-plated		12	1.75	180	105
	25	25		W EAT/E M12 10E 12E/	502021210	M12	6	ning plated	Chains alated		12	1 75	200	125

### Hilti anchor added

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

	-	@ @		Lo olte te te Telli	b el A	9							
Eactory standards	A		AOT STATUS	Designation	Circo	MATERIAL	OPEREI	TVDE	ICON	DN	D	IN	
Ser-defined building materials	1	18	NOL STATUS	HST M8x75/10	M8	St	OBERFL	St	ICON	8	125	75	10
Ser-defined processings				UST M0-05 (20	MO	Ch.		9		0	1.25	05	20
Ser settings     Ser-defined moulding tools	2	22		HST M6x95/50	IVIO	SL		SL		0	1.25	95	50
User-defined dished ends	3	23	•	HST M8x115/50	M8	St		st	_	8	1.25	115	50
User-defined semi-finished products	4	19		HST M10x90/10	M10	St		St		10	1.5	90	10
Ser-defined textures      Ser-defined fasteners	5	24		HST M10x100/20	M10	St		St		10	1.5	100	20
B-  User-defined anchors	6	25		HST M10x110/30	M10	St		St		10	1.5	110	30
B Store	7	26		HST M10x130/50	M10	St		St		10	1.5	130	50
B Sischer	8	27		HST M10x160/80	M10	St		St		10	1.5	160	80
B 🍖 Hilti	9	28		HST M10x200/120	M10	St		St		10	1.5	200	120
Bolt anchor HSA	10	43		HST M12x105/10	M12	St		St		12	1.75	105	10
Bolt anchor HSA-LW, long	11	20		HST M12x115/20	M12	St		St		12	1.75	115	20
Bolt anchor HSA-R	12	29		HST M12x145/50	M12	St		St		12	1.75	145	50
Bolt anchor HST	13	30		HST M12x185/90	M12	St		St		12	1.75	185	90
Bolt anchor HST3	14	22		UST M12x215/120	M12	C4		C+		12	1.75	215	120
	14	33		HST M12x215/120	MIZ	St.		St.		12	1.75	215	140
Bolt anchor HST3 HICAD:	15	31		HST M12x255/140	MIZ	St		St		12	1.75	255	140
	16	32		HST M12x255/160	MIZ	St		St		12	1.75	255	160
	17	34		HST M12x295/200	M12	St		St	_	12	1.75	295	200
	18	44		HST M16x130/15	M16	St		St		16	2	130	15
	19	1	<u>*</u>	HST M16x140/25	M16	St		St		16	2	140	25
	20	2		HST M16x165/50	M16	St		St		16	2	165	50
	21	3		HST M16x215/100	M16	St		St		16	2	215	100
	22	4	•	HST M16x255/140	M16	St		St		16	2	255	140
	23	5		HST M16x295/180	M16	St		St		16	2	295	180
	24	6		HST M20x170/30	M20	St		St		20	2.5	170	30
	25	7		HST M20X200/60	M20	St		St		20	2.5	200	60

### EJOT screws added

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

	1	<b>A A B</b>			b 8 /5	9							
<u>•</u>	A						ontori	7,05	ICON	014	0		TEN LAN
Factory standards     Sectory standards     Sectory standards	1	10	OL STATUS	Designation	Size	MATERIAL C+	OBERFL	IYPE C+	ICON	DN	1 25	LIN 75	IFIX_MAX
User-defined processings	-	10			NIO	St		SL		0	1.25	75	10
	2	22		HST M8x95/30	M8	St		St		8	1.25	95	30
	3	23	•	HST M8x115/50	M8	St		St	_	8	1.25	115	50
User-defined semi-finished products	4	19		HST M10x90/10	M10	St		St		10	1.5	90	10
Vser-defined textures	5	24		HST M10x100/20	M10	St		St		10	1.5	100	20
• User-defined anchors	6	25	•	HST M10x110/30	M10	St		St		10	1.5	110	30
⊞ – S EJOT ⊞ – S Eternit	7	26	•	HST M10x130/50	M10	St		St		10	1.5	130	50
🗉 🔶 Fischer	8	27		HST M10x160/80	M10	St		St		10	1.5	160	80
B - Selt anchor HSA	9	28	•	HST M10x200/120	M10	St		St		10	1.5	200	120
Bolt anchor HSA-F HDG	10	43		HST M12x105/10	M12	St		St		12	1.75	105	10
Bolt anchor HSA-LW, long	11	20		HST M12x115/20	M12	St		St		12	1.75	115	20
Bolt anchor HSA-R2 SS	12	29		HST M12x145/50	M12	St		St		12	1.75	145	50
Bolt anchor HST	13	30		HST M12x185/90	M12	St		St		12	1.75	185	90
Bolt anchor HST3	14	33		HST M12x215/120	M12	St		St		12	1.75	215	120
Bolt anchor HST3 BW	15	31		HST M12x235/140	M12	St		St		12	1 75	235	140
Bolt anchor HST4-R	16	32	-	HST M12x255/160	M12	St		St St	-	12	1.75	255	160
	= 17	34		HST M12v295/200	M12	St		St.		12	1.75	295	200
	10	44		HST M16+120/15	MIE	St.		St.		16	2	120	15
	10	44		HST M16x130/15	MIG	э. С		51		10	2	140	15
	19	1	1.1	HST MI10x140/25	MIIO	SL		SL		10	2	140	25
	20	2		HST M16x165/50	M16	St		St		16	2	165	50
	21	3	•	HST M16x215/100	M16	St		St		16	2	215	100
	22	4	•	HST M16x255/140	M16	St		St		16	2	255	140
	23	5	•	HST M16x295/180	M16	St		St	_	16	2	295	180
	24	6	•	HST M20x170/30	M20	St		St		20	2.5	170	30
	25	7		HST M20X200/60	M20	St		St		20	2.5	200	60

#### Blind rivet added to catalogue

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

el Engineering	- A .	1 R To 1	# 20	2 to to to to 10 10 10 10 10	8									
🗉 🔶 Nuts		ID 10	STATUS	Designation	Size	MATERIAL	OBERFL	TYPE	DN	LN	D2	K	SMIN	SMAX
B- Ivets	1	1		ISO 16585-3.2x6-A2/SS	3.2x6	A2		A2/SSt	3.2	6	6.7	1.3	0.5	1.5
DIN 124 A	2	2	•	ISO 16585-3.2x8-A2/SS	3.2x8	A2		A2/SSt	3.2	8	6.7	1.3	1.5	з
	3	3		ISO 16585-3.2x10-A2/S	3.2x10	A2		A2/SSt	3.2	10	6.7	1.3	3	5
DIN 660 A		4		ISO 16585-3 2x12-A2/S	3.2x12	A2		A2/SSt	32	12	67	13	5	65
	5	5		ISO 16585-3 2x14-42/S	3 2×14	A2		A2/55t	3.2	14	67	13	6.5	8
DIN 674 A	5	6		150 16505 3.2.14 42/5	446	A2		12/55	J.2	6	0.7	1.5	0.5	15
DIN 7337 B	0	0		150 10303-4x0-A2/35L	4x0	AZ		A2/55	4	0	0.4	1.7	0.5	1.5
		/	•	ISO 16585-4x8-A2/SSt	4x8	AZ		A2/SSt	4	8	8.4	1.7	1.5	3
DIN EN ISO 15975	8	8	•	ISO 16585-4x10-A2/SSt	4x10	A2		A2/SSt	4	10	8.4	1.7	3	5
	9	9	•	ISO 16585-4x12-A2/SSt	4x12	A2		A2/SSt	4	12	8.4	1.7	5	6.5
DIN EN ISO 15978	10	10	•	ISO 16585-4x14-A2/SSt	4x14	A2		A2/SSt	4	14	8.4	1.7	6.5	8
DIN EN ISO 15980	1 11	11		ISO 16585-4x16-A2/SSt	4x16	A2		A2/SSt	4	16	8.4	1.7	8	11
	12	12		ISO 16585-4.8x8-A2/SS	4.8x8	A2		A2/SSt	4.8	8	10.1	2	0.5	3
DIN EN ISO 15982	13	13		ISO 16585-4.8x10-A2/S	4.8x10	A2		A2/SSt	4.8	10	10.1	2	3	5
DIN EN ISO 15984	14	14		ISO 16585-4.8x12-A2/S	4.8x12	A2		A2/SSt	4.8	12	10.1	2	5	6.5
DIN EN ISO 16582	15	15	•	ISO 16585-4.8x16-A2/S	4.8x16	A2		A2/SSt	4.8	16	10.1	2	6.5	9
	16	16		ISO 16585-4 8v20-42/S	4.8v20	Δ2		A2/55t	4.8	20	10.1	2	9	12
DIN EN ISO 16585	10	17		ISO 16505 4.0x20 A2/5	6 4.12	A2		A2/55	6.4	12	12.4	27	15	65
	17	17		150 10505-0.4x12-A2/5	0.4112	A2		A2/55L	0.4	12	13.4	2.1	1.5	0.5
	18	18	•	ISO 16585-6.4x16-A2/S	6.4x16	AZ		A2/55t	6.4	16	13.4	2.7	6.5	8
	19	19	•	ISO 16585-6.4x20-A2/S	6.4x20	AZ		A2/SSt	6.4	20	13.4	2.7	8	12

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

If you have saved fasteners in the Purchased/Factory standard parts catalogue, you can access this catalogue in the

**New bolting/riveting** function dialogue window and insert the corresponding elements into the drawing as bolting.

#### Templates for anchors

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

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

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

#### Corrections in the Hilti anchor catalogue

In the Factory standards > User-defined fasteners > User-defined anchors > Hilti > HST 3 bolt anchors catalogue, the same designations were used for anchors with different thread lengths. When inserting anchors using the func-

tion **3-D Standard > Standard Parts > New bolting/riveting** , this meant that not all available anchors were available for selection. The duplicate table entries have been removed and the bolting dialogue for anchors has been expanded so that the selection of an anchor can be specified using the options **Anchoring depth** and **Thickness of filler plates**.

#### EJOT tapping screws without sealing washer

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

•	M 🗢 🕘 🕲		a # 12.2211			<i>c</i> :		00505	THOS	70.00				
User-defined fasteners      User-defined anchors		ID N	NOC STATUS	Designation	ARTICLE	Size	MATERIAL	OBERFL	TYPE	TYP	Thread	DN	P	LN
B User-defined studs	1	38		JT3-6-5,5x25	738030630	ST5.5X25	AZ		AZ		515.5	5.5	1.8	-
Over-defined dowels      User-defined sockets	2	39		J13-6-5,5x30	/38030/30	S15.5x30	A2		A2	1	515.5	5.5	1.8	
• User-defined nuts	3	40	•	J13-6-5,5x35	/38030830	\$15.5x35	A2		A2	1	\$15.5	5.5	1.8	
User-defined rivets	4	41	•	JT3-6-5,5x50	738030930	ST5.5x50	A2		A2	1	ST5.5	5.5	1.8	
Ser-defined elbows	5	42	•	JT3-6-5,5x70	738038330	ST5.5x70	A2		A2	1	ST5.5	5.5	1.8	
ALUCOBOND	6	43		JT3-6-5,5x90	738038430	ST5.5x90	A2		A2	1	ST5.5	5.5	1.8	
User-defined clinch studs     Elot	7	44		JT3-6-5,5x110	738038530	ST5.5x110	A2		A2	1	ST5.5	5.5	1.8	1
JA3-LT	8	45	•	JT3-6-5,5x130	738038630	ST5.5x130	A2		A2	1	ST5.5	5.5	1.8	1
JF3	9	46	•	JT3-6-5,5x150	738054830	ST5.5x150	A2		A2	1	ST5.5	5.5	1.8	1
JF6	10	47	•	JT3-6-5,5x170	738054930	ST5.5x170	A2		A2	1	ST5.5	5.5	1.8	1
	11	48	•	JT3-6-5,5x190	738055030	ST5.5x190	A2		A2	1	ST5.5	5.5	1.8	1
	12	1	•	JT3-6-5,5x25-E14	357220936	ST5.5x25	A2		A2-E14	1	ST5.5	5.5	1.8	
JT3-6	13	2	•	JT3-6-5,5x30-E14	357237736	ST5.5x30	A2		A2-E14	1	ST5.5	5.5	1.8	
JT3-LT	14	3	•	JT3-6-5,5x35-E14	357257736	ST5.5x35	A2		A2-E14	1	ST5.5	5.5	1.8	
	15	4		JT3-6-5,5x25-E16	357221136	ST5.5x25	A2		A2-E16	1	ST5.5	5.5	1.8	
	16	5		JT3-6-5.5x30-E16	357231136	ST5.5x30	A2		A2-E16	1	ST5.5	5.5	1.8	
	17	6		IT3-6-5.5x35-F16	357251136	ST5.5x35	A2		A2-F16	1	ST5.5	5.5	1.8	
	10	7		IT3-6-5 5x50-E16	259291126	ST5 5×50	A2		A2-E16	1	STS 5	5.5	1.9	
- AMP	10	0		IT2 6 5 5 70 516	250201126	ST5.5x50	12		A2 E16	1	CTE E	5.5	1.0	
	19	0		IT2 6 5 5:00 516	250611130	ST5.5x70	12		A2 E10		CTE E	5.5	1.0	-
	20	9		J13-6-5,5X90-E16	359611136	S15.5X90	AZ		A2-E16	1	515.5	5.5	1.8	

# Service Pack 1

# Spring steels in the catalogue

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

CATEditor - [ Catalogues\Materials\Steel\Sprin	ng steels ]	[ C:\H	HiCAD\Kata	loge ]	[ Version:	30.1.0.220 ]								-	o x
File Edit View Extras HELiOS Settings ?	ISD			_											
•	M 🛞	<u>ا (</u>			tt [Ω 9	2 <b>* 0 *0 *</b> 0		<b>a ?</b>							_
Catalogues			ID	MOD	STATUS	Designation	MATERIAL	BBZ	Designation as of 2004	WN	Density	RM	RE	SRAF	COLOR
Air ducts		1	1		•	38Si7	38Si7		38Si7	1.5023	7.85	1600	1150	-1	-1
Bearings		2	2			51CrV4	51CrV4		51CrV4	1.8159	7.85	1650	1200	-1	-1
Dished ends		3	3		•	52CrMo V4	52CrMo V4		52CrMo V4	1.7701	7.68	1750	1300	-1	-1
Factory standards		4	4			61SiCr7	61SiCr7		61SiCr7	1,7108	7,43	1850	1400	-1	-1
E Steners		5	5			C675	C675		C675	1 1231	7.85	1140	510	-1	-1
Aterials		6	6			V10CrNi18-9	X10CrNi18-9		V10CrNi18-8	1.4210	7.0	750	105	-1	-1
E Steel		0				XIOCITATIO-0	XIOCINITO-0		XIOCINITO-0	1.4510	1.5	150	135		
		_	_											_	
Stell for surface instreming     Underched and tempered steels     ♦ Cast iron     ♦ Cast iron     ♦ Non-ferrous metals     ♦ Pinasics     ♥ Pinasics     ♥ Ø Glass															
sarthy		_										6	14-17-11	(1058	

# Thin sheets in the catalogue

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

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Factory standards		ID	MOD	STATUS	Designation	Size	MATERIAL	OBERFL	TYPE	DSTV	τ	BRAD	APROC_OLD	
User-defined building materials		1 1		•	Thin sheet 0.3mm	0.3	DC01		DC01		0.3	0.3	R:DIN6935.ABW	
User-defined processings		2 2			Thin sheet 0.4mm	0.4	DC01		DC01		0.4	0.4	R:DIN6935.ABW	
User-defined moulding tools		3 3			Thin sheet 0.5mm	0.5	DC01		DC01		0.5	0.5	R:DIN6935.ABW	
User-defined dished ends		4 4		•	Thin sheet 0.63mm	0.63	DC01		DC01		0.63	0.63	R:DIN6935.ABW	-
User-defined semi-finished products		5 5			Thin sheet 0.75mm	0.75	DC01		DC01		0.75	0.75	R:DIN6935.ABW	-
Ner-defined textures		6 6			Thin sheet 0.88mm	0.88	DC01		DC01		0.88	0.88	R-DIN6935 ABW	-
User-defined fasteners		7 7			Thin sheet 1.00mm	1.00	DC01		DC01	-	1	1	R-DIN6935 ABW	-
User-defined materials		0 9		-	Thin sheet 1.25mm	1.00	DC01		DC01		1.25	1 25	P-DIN6935 ABW	-
Series		0 0			This cheet 1 50mm	1.50	DC01		DC01	1 1	1.5	1.5	P.DIN6025 ARW	-
Bend zone tooling		9 9		- (	Thin cheet 1.75mm	1.50	DC01	-	DC01		1.5	1.7	P.DIN6025 ADM	-
Blechfertigung		1 11		-	This sheet 1.79mm	2.00	DC01		DC01	-	1.75	1./3	D.DINGODE ADIA	-
Railing		1 11			This is a 2.00mm	2.00	DCUI		DC01	-	2 22	2	N:DIN0935.ABW	-
Glass dimensionings		2 12			Thin sneet 2.25mm	2.20	DCUI		DCUI	-	2.25	2.25	K:DIN0935.ABW	-
Glass panes		3 13			Thin sheet 2.50mm	2.50	DC01		DC01		2.5	2.5	R:DIN6935.ABW	-
Sheets		4 14		•	Thin sheet 2.75mm	2.75	DC01		DC01	-	2.75	2.75	R:DIN6935.ABW	-
ALUCOBOND	1	5 15		•	Thin sheet 2.99mm	2.99	DC01		DC01		2.99	2.99	R:DIN6935.ABW	_
ALUCOBOND_US	1	6 16		•	Thin sheet 0.3mm	0.3	DC03		DC03		0.3	0.3	R:DIN6935.ABW	
🗈 🍉 Eternit	1	7 17		•	Thin sheet 0.4mm	0.4	DC03		DC03		0.4	0.4	R:DIN6935.ABW	
Aluminium sheet/plate	1	8 18		•	Thin sheet 0.5mm	0.5	DC03		DC03		0.5	0.5	R:DIN6935.ABW	
This sheet	1	9 19		•	Thin sheet 0.63mm	0.63	DC03		DC03		0.63	0.63	R:DIN6935.ABW	
ISD sheet	2	20		•	Thin sheet 0.75mm	0.75	DC03		DC03		0.75	0.75	R:DIN6935.ABW	
ISD US sheet	2	1 21		•	Thin sheet 0.88mm	0.88	DC03		DC03		0.88	0.88	R:DIN6935.ABW	
🔀 ISD US sheet	2	2 22		•	Thin sheet 1.00mm	1.00	DC03		DC03		1	1	R:DIN6935.ABW	
Steel sheet/plate	2	3 23		•	Thin sheet 1.25mm	1.25	DC03		DC03		1.25	1.25	R:DIN6935.ABW	
Bulb plate	2	4 24		•	Thin sheet 1.50mm	1.50	DC03		DC03		1.5	1.5	R:DIN6935.ABW	
Hole patterns	2	5 25		•	Thin sheet 1.75mm	1.75	DC03		DC03		1.75	1.75	R:DIN6935.ABW	
¥	_ 2	6 26			Thin sheet 2.00mm	2.00	DC03		DC03		2	2	R:DIN6935.ABW	
	2	7 27			Thin sheet 2.25mm	2.25	DC03		DC03		2.25	2.25	R:DIN6935.ABW	-
	2	8 28			Thin sheet 2.50mm	2.50	DC03		DC03		2.5	2.5	R:DIN6935.ABW	-
1		9 29			Thin sheet 2.75mm	2.75	DC03		DC03		2,75	2,75	R:DIN6935.ABW	-
		0 30			Thin sheet 2.99mm	2.99	DC03		DC03		2,99	2,99	R:DIN6935.ABW	-
		1 31	-		Thin sheet 0.3mm	0.3	DC04		DC04		0.3	0.3	R-DIN6935 ABW	-
		2 22			This sheet 0.4m	0.4	0004		DCOA		0.5	0.5	D-DINIGODE ADIA	+

#### ALUCOBOND SFS screw

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



#### Additional thicknesses for sheets

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

#### Query for material sync

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



# Washer DIN 9021

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

Edit view Extras filelios settings i ise	000					0								
<i>₽</i> 4	ि	19 E Lo E	111 12:			8								
Fasteners		ID MO	STATUS	Designation	Size	MATERIAL	OBERFL	TYPE	HKL	DN	D1	D2	S	Weig
General geometry	1	1	•	DIN 9021-2.7-A2-50	2,5	A2-50		A2-50	140 HV	2.5	2.7	8	0.8	0.0
E Spring	2	2	•	DIN 9021-3.2-A2-50	3	A2-50		A2-50	140 HV	3	3.2	9	0.8	0.00
Spring connector	3	3	•	DIN 9021-3.7-A2-50	3.5	A2-50		A2-50	140 HV	3.5	3.7	11	0.8	0.0
🗉 🔶 Nuts	4	4	•	DIN 9021-4.3-A2-50	4	A2-50		A2-50	140 HV	4	4.3	12	1	0.0
Rivets	5	5	•	DIN 9021-5.3-A2-50	5	A2-50		A2-50	140 HV	5	5.3	15	1.2	0.
🕀 🎨 Feather keys	6	6		DIN 9021-6.4-A2-50	6	A2-50		A2-50	140 HV	6	6.4	18	1.6	0
Shim rings	7	7		DIN 9021-7.4-A2-50	7	A2-50		A2-50	140 HV	7	7.4	22	2	0
Bolts+Screws				DIN 9021-8 4-02-50	8	A2-50		A2-50	140 HV		8.4	24	2	0
Adjusting rings		0		DIN 9021 10 5 42 50	10	A2 50		A2 50	140 LIV	10	10.5	20	26	
🗄 🏀 Pins	9	9		DIN 9021-10.3-A2-50	10	A2-30		A2-50	140 100	10	10.5	30	2.3	
🖻 🔷 Washers	10	10		DIN 9021-13-A2-50	12	A2-50		A2-50	140 HV	12	15	5/	3	
Ceramic rings for welded studs	11	11	•	DIN 9021-15-A2-50	14	A2-50		A2-50	140 HV	14	15	44	3	
Clamping plates	12	12	•	DIN 9021-17-A2-50	16	A2-50		A2-50	140 HV	16	17	50	3	
Washers	13	13		DIN 9021-20-A2-50	18	A2-50		A2-50	140 HV	18	20	56	4	
DIN EN ISO 7090	14	14	•	DIN 9021-22-A2-50	20	A2-50		A2-50	140 HV	20	22	60	4	
DIN EN 14399-7	15	15	•	DIN 9021-26-A2-50	24	A2-50		A2-50	140 HV	24	26	72	5	
DIN 9021	16	16	•	DIN 9021-33-A2-50	30	A2-50		A2-50	140 HV	30	33	92	6	
DIN 7349	17	17	•	DIN 9021-39-A2-50	36	A2-50		A2-50	140 HV	36	39	110	8	
	18	18	•	DIN 9021-2.7-A2-70	2,5	A2-70		A2-70	140 HV	2.5	2.7	8	0.8	0
	19	19	•	DIN 9021-3.2-A2-70	3	A2-70		A2-70	140 HV	3	3.2	9	0.8	0.0
	20	20	•	DIN 9021-3.7-A2-70	3.5	A2-70		A2-70	140 HV	3.5	3.7	11	0.8	0.0
	21	21		DIN 9021-4.3-A2-70	4	A2-70		A2-70	140 HV	4	4.3	12	1	0.0
	22	22	1	DIN 9021-5-3-A2-70	5	A2-70		A2-70	140 HV	5	5.3	15	1.2	0
	22	23		DIN 9021-64-42-70	6	A2-70		A2.70	140 HV	6	6.4	18	16	0
	23	23		DIN 9021-74-62-70	7	A2-70		A2-70	140 LIV	7	7.4	22	1.0	0
	24	24		Diry 5021-7.4-A2-70	-	M2-10		M2-10	140110	/	7.4	22	2	0.

# Major Release

### Create tables in catalogues

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



### Attribute management now in the Configuration Editor

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

### Column comments for countersinks

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

	ID	MOD	STATUS	Designation	Size	TYPE	DN	DB	DS	Т	ALFA	TOL	WTOL+-	TOPSYMBOL
1	1			ISO 15065-1.6	1.6	М	1.6	1.8	3.6	0.95	Count	ersink angle		
2	2	2		ISO 15065-2	2	M	2	2.4	4.4	1.05	Count	craink ungit		
3	3		•	ISO 15065-2.5	2.5	М	2.5	2.9	5.5	1.35	#Date	ntyp: Fließk	ommazahl (I	D ONLY)
4	4			ISO 15065-3	3	М	3	3.4	6.3	1.55	#Wert	eeinheit: °	(ISD ONLY)	
5	5			ISO 15065-3.5	3.5	М	3.5	3.9	8.2	2.25	90	H 13	1°	
6	6	;	•	ISO 15065-4	4	M	4	4.5	9.4	2.55	90	H 13	1°	
7	7		•	ISO 15065-5	5	M	5	5.5	10.4	2.58	90	H 13	1°	
8	9			ISO 15065-6	6	M	6	6.6	12.6	3.13	90	H 13	1°	
9	10	)	•	ISO 15065-8	8	М	8	9	17.3	4.28	90	H 13	1°	
10	11			ISO 15065-10	10	М	10	11	20	4.65	90	H 13	1°	
11	12		•	ISO 15065-2	2.2	ST	2.2	2.4	4.4	1.05	90	H 13	1°	
12	13			ISO 15065-3	2.9	ST	2.9	3.4	6.3	1.55	90	H 13	1°	

### Wedge lock washers added to the catalogue

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

	-	1.00	I													
Vser-defined elbows		ID	MOD	STATUS	Designation	Size	MATERIAL	OBERFL	TYPE	DN	D1	D2	D3	D4	S	ALFA
User-defined poits+screws	1	1	1	•	M2.5	M2.5	A1		A1	2.5	2.7	9	6.3	5.7	1.6	90
E- User-defined pins	2	2	2	•	M2.6	M2.6	A1		A1	2.6	2.7	9	6.3	5.7	1.6	90
E 🔖 User-defined washers	3	3	3	•	M3	M3	A1		A1	3	3.2	10	7.3	6.5	1.8	90
🕀 🎨 Ejot	4	4	4	•	M4	M4	A1		A1	4	4.3	14	9.5	8.6	2.3	90
Eternit	5	5	5	•	M5	M5	A1		A1	5	5.3	16	11.5	10.4	2.8	90
E Fastenal	6	6	5	•	M6	M6	A1		A1	6	6.4	18	13.5	12.4	3.3	90
	7	7	7	•	M8	M8	A1		A1	8	8.4	25	17.5	16.4	4.8	90
E Stindapter	8	8	3	•	M10	M10	A1		A1	10	10.5	30	22.5	21	6.5	90
Wedge lock washers (imperia)     SFs     Wedge lock washers (imperia)     SFs     Washer for countersunk head screws     User-defined materials																

### Round steel added for the US market

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

.ock 🚬	<b>#</b>			3 H	tt 🖸 🤉	⊇ <b>*0 *0 *</b> (		6 6 ?							
User-defined moulding tools			ID	MOD	STATUS	Designation	Size	MATERIAL	OBERFL	TYPE	D	Weight	MANTELFL	BEZ	DS
User-defined dished ends	F	1			•	RB 1/4"	RB 1/4"	A36		A36	0.25	0.2	0.07		
User-defined semi-finished products	. 1	2	2	2		RB 3/8"	RB 3/8"	A36		A36	0.375	0.4	0.1		
User-defined profiles		3		3		RB 1/2"	RB 1/2"	A36		A36	0.5	0.7	0.13		
Flat steels		4	4	1		RB 5/8"	RB 5/8"	A36		A36	0.625	1	0.16		
Round steel		5		5		RB 3/4"	RB 3/4"	A36		A36	0.75	1.5	0.2		
RB (imperial)		6	(	5		RB 7/8"	RB 7/8"	A36		A36	0.875	2	0.23		
User-defined textures	-	7		7		RB 1"	RB 1"	A36		A36	1	2.7	0.26		
User-defined materials	-	8	8	2	-	RB 1 1/8"	RB 1 1/8"	A36		A36	1,125	3.4	0.29		
User-defined cylinders	ŀ	0			-	RB 1 1/4"	RB 1 1/4"	436		A36	1.25	42	0.33		
E Series	-	10	10			PR 1 2/9"	PR 1 2/9"	A26		A26	1 275	5.1	0.35		
🗉 🎨 Bend zone tooling	-	11	1		-	PD 1 1/2"	PD 1 1/2"	A30		A36	1.575	5.1	0.30		
Sheet dimensions	-	12	1		-	DD 1 5/0"	DD 1 5/0"	A30		A30	1.5	71	0.59		
Railing     Glass dimensionings	-	12	14	-	-	RB 1 3/8	RB 1 3/8	A30		A30	1.020	7.1	0.45		
Glass panes	-	13	1.		-	KB 1 3/4	KB 1 3/4	A30		A30	1./3	8.2	0.40		
		14	14	+		KB 1 7/8	KB 1 //8"	A30		A30	1.8/5	9.4	0.49		
-	ŀ	15	1:		•	RB 2"	RB 2"	A36		A36	2	10.7	0.52		
	-	16	16	5		RB 2 1/8"	RB 2 1/8"	A36		A36	2.125	12.1	0.56		
	-	17	17	7	•	RB 2 1/4"	RB 2 1/4"	A36		A36	2.25	13.5	0.59		
		18	18	3	•	RB 2 3/8"	RB 2 3/8"	A36		A36	2.375	15.1	0.62		
		19	19	)	•	RB 2 1/2"	RB 2 1/2"	A36		A36	2.5	16.7	0.65		
		20	20			RB 2 5/8"	RB 2 5/8"	A36		A36	2.625	18.4	0.69		
		21	21		•	RB 2 3/4"	RB 2 3/4"	A36		A36	2.75	20.2	0.72		

### Fischer High performance anchor FH II with countersunk head

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

ctory standards		ID	MOD	STATUS	Designation	Size	MATERIAL	OBERFL	TYPE	ICON	DN	P	LN	TFIX_MAX	EINB
User-defined building materials	1	19	2	•	FH II 10/10 S	FH II 10/10 S		galvanized	A4 galvanized	510923	6	1	69	10	
User-defined processings	2	20			FH II 10/25 S	FH II 10/25 S		galvanized	A4 galvanized	510924	6	1	84	25	
User settings	3	2.			EH II 12/10 S	EH II 12/10 S		galvanized	A4 galvanized	510925	9	1.25	90	10	
User-defined dished ends	4	21	,	-	FH II 12/25 S	FH II 12/25 S		galvanized	A4 galvanized	510926	9	1.25	105	25	
User-defined semi-finished products	-	21	2		EH II 15/10 S	EH II 15/10 S		galvanized	A4 galvanized	510027	10	1.5	107	10	
User-defined textures	-	2			FILL 15/10 5	FILL 15/10 5		gaivanized	A4 galvanized	510020	10	1.5	107	25	
User-defined fasteners	0	2.			FH11 13/25 5	FH II 10/25 S		gaivanizeu	A4 galvanizeu	510920	10	1.5	122	25	
User-defined anchors	-	2			FH II 18/25 S	FH II 18/25 5		galvanized	A4 gaivanized	210353	12	1.75	155	25	
E- EJOT	8	20	2		FH II 24/25 S	FH II 24/25 S		galvanized	A4 galvanized	502/11	16	2	160	25	
E Scher	9			•	FH II 10/10 S	FH II 10/10 S		galvanized	St galvanized	503133	6	1	70	10	
Anchor rod RG M	10		2		FH II 10/25 S	FH II 10/25 S		galvanized	St galvanized	503134	6	1	85	25	
Bolt anchor FAZ II Bolt anchor FAZ II-GS Bolt anchor FBN II Bolt anchor FBN II Bolt anchor FBN II GS	11	-	3	•	FH II 10/50 S	FH II 10/50 S		galvanized	St galvanized	503135	6	1	110	50	
	12	4	1	•	FH II 12/10 S	FH II 12/10 S		galvanized	St galvanized	044884	9	1.25	90	10	
	13		5	•	FH II 12/25 S	FH II 12/25 S		galvanized	St galvanized	044885	9	1.25	105	25	
Bolt anchor FBN II GS	14	6	5	•	FH II 12/50 S	FH II 12/50 S		galvanized	St galvanized	044886	9	1.25	130	50	
High performance anchor EH II B	15	7	7	•	FH II 15/10 S	FH II 15/10 S		galvanized	St galvanized	044887	10	1.5	106	10	
High performance anchor FH II-H	16	8	3	•	FH II 15/25 S	FH II 15/25 S		galvanized	St galvanized	044888	10	1.5	121	25	
High performance anchor FH II-S	17	9	9		FH II 15/50 S	FH II 15/50 S		galvanized	St galvanized	044889	10	1.5	146	50	
High performance anchor FH II-SK	18	10	)		FH II 18/10 S	FH II 18/10 S		galvanized	St galvanized	046847	12	1.75	118	10	
🗄 🚸 HIB	19	1	1		FH II 18/25 S	FH II 18/25 S		galvanized	St galvanized	044894	12	1.75	132	25	
	- 20	12	2		FH II 18/50 S	FH II 18/50 S		galvanized	St galvanized	044896	12	1.75	157		
	21	13	3		FH II 24/25 S	FH    24/25 S		galvanized	St galvanized	044898	16	2	/		
	22	14	1		FH II 24/50 S	FH II 24/50 S		galvanized	St galvanized	044900	16	2			
	22	14	5		FH II 28/30 S	FH II 28/30 S		galvanized	St galvanized	044901	20				
	24	16			FH II 28/60 S	FH II 28/60 S		galvanized	St galvanized	044902	20			50	
	24	1	,	- (	EL II 22/20 C	EL II 22/20 C		galuanized	St galvanized	044002				20	
	25	1.			FH II 52/50 5	FH II 52/50 5		gaivanized	St galvanized	044905			210	50	
	20	18	s I	•	FH II 32/00 S	FH II 32/00 S		galvanized	st gaivanized	044904			245	ou	
	II —	_												6	
											J[0 ][20	11:4:	0:20		a

#### Material synchronisation

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

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

CATEditor - [ Catalogues ]	[C:\HiCAD\Kataloge] [Version: 3	30.0.0.1	51]
File Edit View Extras H	ELiOS Settings ? ISD		
⊡12 Catalogues ⊕ ♦ Air ducts	Connect upon program launch Login Remove assignment to HELiOS		
	Project	>	
	Folder	>	dards
Fasteners     Materials	Material synchronisation		
<ul> <li>Plant Engineering</li> <li>Processings, general</li> </ul>	Pla Pro Ser	nt Engin cessing mi-finishe	eering s, general ed products

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

0

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

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

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

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

# **Bill of Materials / Report Manager**

#### Discontinuation of the "old" Report Manager

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

### Service Pack 2

#### New attributes for the Report Manager

The new attributes for Steel Engineering beams and profiles and Steel Engineering plates have been integrated into the Report Manager export files**rm\_h\_exportpart.HDB** and **rm\_db\_exportpart.HDB**.

Attributes for Steel Engineering beams and profiles:

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

and Steel Engineering plates:

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

#### Activate BOM-relevance

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

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

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

The default settings can be restored using HicadGUIReset.exe.

#### Bar list: Non-numeric values for item texts

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

1	Α	В	С	D	E	F	G
1	Bar	list					
2 2 D	rawing N	lo			Customer		
4 0	order No.				Created by		
5 0	order tex	t i			Created on		HLEAD
6 D	esignati	on					
12 F	RR 12	0x60x4,	S235JRH		2 x 6000 mm	Waste: 5680 mm (47,33 %)	
13	Item	Qty.	Length (mm)	Cut (Web)	Cut (Flange)	Designation	Coating
14							
15			102 (1700)		102 (1700)	103 (790)	103 (790) 100 (670) 2/308 2/
16	102	2	1700		45° 🗆 🖾 45	•	

# Service Pack 1

# Configuration of product structure

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

## **Major Release**

### Coloured HTML tables

#### Column colour

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

Text parameters		
Font:	Arial Regular	*
	Bold Italic	
Text height:	12 *	
Colours:	Column header 🔗	A
	Odd rows	A
	Even rows 🔗	A

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

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

#### Print coloured BOMs

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

#### Units in Lists of sawn beams

The displayed units (mm, inch) of the values (e.g. length) are taken from the List of sawn beams as well as the other lists from the Report Manager. If you now want to output inches instead of millimetres, use the **Column settings** 

function to set the display unit for the length to inches for the Quantity list. The column header may then need to be adjusted in the Excel template. To do this, open the XLSX file and place a placeholder in the column header. If you then change the unit in the Report Manager, the column headers in the column list will also be changed.

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

- BOMTemplates/HiCAD-DB\_Anlagenbau.EN.30.0.xlsx
- BOMTemplates/HiCAD-DB\_Mechanik.EN.30.0.xlsx
- BOMTemplates/HiCAD-DB\_Stahlbau\_BIM.EN.30.0.xlsx
- BOMTemplates/HiCAD\_Anlagenbau.EN.30.0.xlsx

- BOMTemplates/HiCAD\_Blech.EN.30.0.xlsx
- BOMTemplates/HiCAD\_Mechanik.EN.30.0.xlsx
- BOMTemplates/HiCAD\_Metallbau.EN.30.0.xlsx
- BOMTemplates/HiCAD\_PipeBook.EN.30.0.xlsx
- BOMTemplates/HiCAD\_Stahlbau.EN.30.0.xlsx
- BOMTemplates/Proflex.EN.30.0.xlsx

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

### Sheet Metal parts with development

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

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

Template HiCAD\_Stahlbau.EN.30.0.xlsx, Settings sheet

	A B	с	D	E	F	G	н	1	J K
1	Sheet M	letal							
2	Drawing No	021135D	Customer	ISD Software und Systeme G	imbH				
3	Order No	200 003 001	Created by	Maya Mustermann					
	Order text	002 0099 111	Created on	04.07-2024				1	
5	Naming	200 003 001 111	oreated on	04.07 2024				- <b>n</b>	CTU -
7	- tuning	200_000_001_111							
8				Item	1		Surf. (m2)	0.02	
9				Designation	BL 0.75 001		Weight (kg)	0.02	
10				Material	A199.0		Total weight	0.02	
11				Designation			0		
12		N N N N		Coating					
13		$\oplus$		Development length (mm)	112				
14		120-		Development width (mm)	106				
15		111,63		Thickness (mm)	1				
16									
17									
18				Item	2		Surf. (m2)	0,02	
19				Designation	BI 0.75 002		Weight (kg)	0,02	
20				Material	A199,0		Total weight	0,02	
21				Designation					
22		a ⊕ ⊕ [£]		Coating					
23		20*		Development length (mm)	112				
24				Development width (mm)	91				
25		90,93		Thickness (mm)	1				
26									
27									
28				Item	3		Surf. (m2)	0,03	
29				Designation	BI 0.75 003		Weight (kg)	0,03	
30				Material	Al99,0		Total weight	0,03	
31		0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0		Designation					
32		140,6		Coating					
33		Φ Φ		Development length (mm)	141				
34				Development width (mm)	100				
35		100		Thickness (mm)	1				
36									

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

### Quantity list for profile installation

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

1	A	В	C	D	E	F	G	Н	Ĵ.	J	K	L
1	Qua	ntit	y list, Profile ins	stallati	on							
2	Drawing N	No.	123091827	Customer	ISD Software	und Systeme GmbH	1				1	
4	Order No.		1000_0001	Created by	Maya						10	
5	Order tex	t	Hall	Created on	10.04.							
6	Naming		H_298_B									
7												
8	Item	Qty.	Designa	ation			Nami	ng	Coating		Weight	Total weight
9	500	2	ONDAFIBRE 3003 B (150 mm) liniert/liniert	CES 50 C					Paint RAL 1000 / Paint RAL 6000		22,40	44,80
10	501	(	ONDAFIBRE 3003 B (150 mm) liniert/liniert	CES 50 C					Paint RAL 1000 / Paint RAL 6000		67,20	403,20
11	502	Ę	5 Frequenze 18/76 T (1.00 mm) St						Paint RAL 1000 / Paint RAL 2002		19,23	96,16
12	503	6	6 Hacierco 39/333 (1.25 mm) St		17				Paint RAL 1000 / Paint RAL 2005		36,25	217,50
13		19	9									

# **Configuration Management**

Service Pack 2

Attribute management

#### **Attribute calculation**

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

#### Attributes and referencing

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

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

#### Attribute description as tooltip

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

#### Duplicating attributes in attribute management

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

#### Attribute calculation upon change

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

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

#### Manufacturability check

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

The following new checks are now available:

#### Minimum flange length

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

#### Minimum bend radius

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

#### Distance between processings

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

#### Minimum diameter for standard bores

This test only applies to standard bores.

#### Minimum Z-fold height

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

### Default setting for drawing frame

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

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



### Design tolerance

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

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

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S S   12 13 11	A 0 0		User	- 🧟
▲ Compatibility	Description	Value	Comment	
Breet development up to HiCAD     Annotations     Attribute calculation	Check design tolerance	•	A tolerance setting that deviates from the standar Output a warning message if the design tolerance case of a strong deviation from the standard, a w	d can lead to serious errors. e deviates from the standard. In arning is issued in any case.
<ul> <li>HiCAD-HELiOS interface</li> <li>Itemisation up to HiCAD 2017</li> <li>Itemisation</li> <li>Standard parts and processings u</li> <li>Warnings</li> <li>User library up to HiCAD 2022</li> <li>Views</li> <li>Automatic drawing derivation</li> <li>System settings</li> </ul>				

## Service Pack 1

### Manufacturability of Sheet Metal parts

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

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

and the new checks

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

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

ISD Configuration Editor - HiCAD 30.1.0.255 [C:\Program	Data\ISD Software und Systeme\HiCAD 2025\HiCAD.cfgdb]		- 0	×
File Edit View Extras ISD				
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S     C     Pill     Pill	Carry out manufacturability check when exporting sheets     Description     Carry out manufacturability check when exporting sheets     Distance of processings to bend zone     Check distance of processing from bend zone     Take values for comparison from     Formula for slots running in parallel     Formula for tobres/other processing     Distance of processings to edge     Check distance of processing from edge     Take values for comparison from     Formula for comparison row     Formula for comparison row     Formula for comparison value     Maximum sheet dimensions     Check maximum sheet dimensions	Value	User Comment Example: 'L > 30mm? 5'T: 3'T' with L = lengt and T = sheet thickness Example: 'L > 30mm? 5'T: 3'T' with L = lengt rectangle and T = sheet thickness Example: '5'T + 3mm' with T = sheet thicknes Example: '5'T + 3mm' with T = sheet thicknes The values are taken from the catalogue	h of slot th of
B Interfaces     B Compatibility     J System settings     Moniformations	Collisions in developments Check collisions in development Ignore collisions of bend zones	<ul> <li>✓</li> <li>✓</li> </ul>		
Sincer metal 2 manufacturability check				9

### Production drawing / processing notes for unprocessed beams and profiles

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

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

ile Edit View Extras ISD			
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HiCAD 🔶	Description	Value	Comment
<ul> <li>Active configuration (Base configuration)</li> <li>Drawing</li> </ul>	Detail drawings for beams/profiles		Create view groups for beams/ profiles
Automatic drawing derivation	Create detail drawings of SE plates		Create view groups for steel plates
Production drawing	Detail drawings for sheets		Create view groups for metal sheets
Drawing	Detail drawings for general parts		Create view groups for general parts
Drawing frames	Production drawing for unprocessed beams	Yes	Create production drawings for unprocessed beams?
<ul> <li>Annotations</li> <li>Development</li> <li>Usage assignment</li> </ul>	Processing note for unprocessed beams	Collection	Unprocessed beams with processing note from the list (HiCAD attribute SBHW) obtain a production drawing anyway
▷ Isage-dependent ■ Mounting drawing	Sectional views for full stiffeners	Yes	Create sectional views for full stiffeners?
<ul> <li>Modelling</li> <li>Steel Engineering</li> </ul>	Top view for steel engineering plates dependent on powder marking lines and lettering	No	Should the top view be determined by the number of powder marking lines and lettering?
Image: Metal Engineering     Image: Profile Installation	Drawing target (Sheet area)	On new sheet	Sheet area in which the drawings are generated.
Image: Plant Engineering     Sheet Metal	Drawing target (entire drawing)	Existing drawing	Drawing file in which the drawings are generated
<ul> <li>Assembling simulation</li> <li>Analysis</li> </ul>	Remove BOM-relevance of parts without detail drawing in external model drawing		Remove BOM-relevance of parts without detail drawing in external model drawing
<ul> <li>Interfaces</li> <li>Compatibility</li> </ul>	Leave external drawing open	Leave drawing open, switch to original drawing	Behaviour when creating external drawings
System settings	Also detail drawings for sub-parts		Also detail drawings for sub-parts
## Setting for Steel Engineering plates in drawing derivation

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

Drawing derivation	×		
(Drawing target)			
To existing drawing	)		
Externally generated drawings			
Leave open, switch back to original d	Irawing 💌		
Detail drawing for each sub-part			
(Sheet selection)	Alignment of assemblies	Views for plates	×
New sheet	Processing position		
Drawings for	·	View group	
		AXO landscape	Arrange at bottom 👻
	Views to be created for:	AXO portrait	Arrange at left side 👻
Assembly (Filter	Assembly	✓ Top View	Bottom view
🔲 Beams	Beams	Erent View	Rock view
V Plates	Plates	From view	Dack view
V Sheet Metal	Sheet Metal	View from left	View from right
General parts	General parts	Development	Arrange at bottom 💌
(Drawing parameters)	Settings for:	Sectional views	Aligned (default)
	Drawing sheets	Minimum number of sectional vie	ews per page
Save Load	View groups		
Set in dialogue	Views		
Settings file	Sheet developments		OK Cancel
Save Load	Sectional views of sheets		
	OK Cancel		

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

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

# Attributes

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

For beams and profiles there are the new attributes:

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

and for plates and sheets:

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

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

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

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

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

Edit View Extras ISD							
) 🕄 🕴 12 13 💱	A 0 0 -		User 🖉 🧟				
<ul> <li>System settings</li> <li>Gamma System anagement</li> </ul>	Description     Calculate rectangular surface area of     development	Value Always ~	Comment Assigns value to attribute \$52D, Calculated from length * width of development				
Attributes	Calculate weight from rectangular surface area of development	Always ~	Only for Sheet Metal parts. Assigns attribute SCW.				
Assembly HCM Sketch HCM	Calculate dimensions of development	Always ~	Assigns value to attribute §L2D, §B2D, §T2D				
	Transfer development dimensions to	Development attributes and Dim 👻					
Processing plane	Steel Engineering beams						
<ul> <li>Frocessing plane</li> <li>Scales</li> <li>Sketches</li> <li>Units</li> <li>Novice configuration</li> </ul>	Transfer geometry attributes by length	Geometry attributes by length 🛛 👻					
	E Consider roll and bend radii	Geometry attributes by length Geometry attributes from rectangle area (occupies \$WBL, \$CBL, \$SBL, \$VBL) Geometry attributes by length and standard attributes					
Directories	Geometry attributes from rectangle area (occupies \$WBL, \$CBL, \$SBL, \$VBL) and standard attributes (\$01, \$18, Plates						
Data save	Geometry attributes from rectangle area transferred to	Geometry attributes from rectan ~					
Identification Referencing	Consider recessed corners		Consider recessed corners of the sheet blank for rectangle with minimum blank size				
Annotations	Min. surf. area [m <sup>2</sup> ]	0.2	Minimum surface area in [m <sup>2</sup> ] of recessed				

Settings for beams and profiles

Edit View Extras ISD							
) 😂   🦹 12 13 📲		A 0 D		User 🗸 🖉			
<ul> <li>System settings</li> <li>Attribute management</li> </ul>	*	Description Calculate rectangular surface area of development	Value Always	Comment Assigns value to attribute 352D. Calculated from length * width of development			
Attributes Attribute calculation		Calculate weight from rectangular surface area of development	Always ~	Only for Sheet Metal parts. Assigns attribute §CW.			
Assembly HCM	E	/ Calculate dimensions of development	Always ~	Assigns value to attribute §L2D, §B2D, §T2D			
Sketch HCM		Transfer development dimensions to	Development attributes and Dim 👻				
Processing plane		Steel Engineering beams					
Scales					Transfer geometry attributes by length to	Geometry attributes by length ~	
<ul> <li>Dits</li> <li>Novice configuration</li> </ul>		Consider roll and bend radii		Consider roll radii and bend radii of simplified standard beams as allowances for exact weight calculations			
III Directories		Plates					
I Load/Save		Geometry attributes from rectangle	Geometry attributes from rectan 👻				
<ul> <li>Bata save</li> <li>Identification</li> <li>Referencing</li> </ul>		Consider recessed corners	Geometry attributes from rectangular area Geometry attributes from rectangle area (occupies \$CW, \$CBA, \$SBA, \$VBA)				
Annotations	-	Geometry attributes from rectangular area and standard attributes     Geometry attributes from rectangle area (occupies SCW, SCBA, SSBA, SVBA) and standar     Geometry attributes from rectangle area (occupies SCW, SCBA, SSBA, SVBA) and standar					

#### Settings for plates and sheets

Due to the extensions in the Configuration Editor, the Weight calculation tab of the Steel Engineering Settings dia-

logue (Steel Engineering > Further functions > Settings) was no longer needed and was removed. The settings on the tab only had a temporary effect until the next restart of HiCAD.

The dialogue window now looks like this:

Beams eams	Plates	Gratings	🔘 Glass panes
			1
			1
Simplified	Exact	t	Axis only
Tracing lines	S.E. axes	Axis end points	Beam annotation
Contour represent	tation for heam series	Glass symbols	

# HELiOS Settings in the Configuration Editor

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

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

# New function for creating a structure assembly

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

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

## P+ID

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

# Major Release

## Attribute management

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

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

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

### Settings for standard parts and processing

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

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

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

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

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

## Novice configuration - Dialogue change

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

# Activate frame / scale list

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

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



## Save referenced detail drawings as itemised source models

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

## Internally referenced parts in externally referenced assemblies

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

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

# **Notes on HELiOS Updates**

#### Microsoft SQL Server

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

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

#### **HELiOS Workspaces: Conversion of the system directories**

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

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

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

#### Notes on update installations

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

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

#### Data backup before updating

Make sure that a data backup was made before carrying out the update of your HELiOS database. For the backup, either use the HELiOS Database Creator (further information can be found in the Installation Notes) or your SQL Server Application.

In case of any questions, or if you need any help with regard to your customized system architecture , contact the ISD Hotline.

#### Log file for update

If any conflicts occur during the update, these will be recorded in the log file **HeliosDbUpdate.txt** (in the system path **%appdata%\ISD Software und Systeme\HeliosDbUpdate**\).

Have this file ready when contacting the ISD Hotline in case of an unsuccessful update, so that they can help you solve the problem and make a successful update.

#### New mask format

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

#### MultiCAD interfaces

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

In this case, please contact the Consulting department of the ISD Group.

# **HELiOS Desktop**

Service Pack 2

**Result lists** 

Sort

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

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

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

Sele	cted attribu	tes			
📑 List		🗟 Context menu	Report		
1	<b>I</b>				
Туре	Object type	Designation	Sort sequence	Format	
	0	Changed on	Ascending		
	0	Article number	Ascending	Text	1
	0	Index	Ascending	Text	0

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

Select	ed attri	butes			
📑 List		🛛 🕅 Context menu	Report		
Type C	)	ne Designation	Sort sequence	Format	
	0	Changed on	Ascending		
	0	Article number	▼ Descending	Text	1
a	0	Index	Ascending	Text	1

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

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

Part type	Designatio	Designation		
Assemb	Change title			
Assemb	Horizontal alignment			
Assemb	Representation type 🕨			
Assemb				
Others	Save			
Part	Reset			
Part	Configure			

#### Sorting arrow

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



#### Subsequent sorting

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

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

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

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

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

#### **VARCHAR** attributes as Text or Numbers

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

🗐 List		Context m	enu   📔 Rej	port #1 Sort		
Type C	bject ty	peDesignation		Sort sequence	Format	
	1	Item number		Ascending	Text	1
		Int o	erpret attribu Text Number	Edit format ute content as:		

	1 Item	💱 Qty.	18 CAD	li 🔾 Article number	010	(0)	Designation
	1	1	2	SN-025891	•	1	front side plate r
₿.	11	2		SN-025903	•	1	Shock Bushing
	12	1		SN-025894	•	1	Front Shock Brace
₿.	13	2		SN-025901	•	1	Front Lower Arm
	14	1		SN-025895	•	1	Front Lower Arm Brace
₿.	15	4		SN-025942	•	1	ANSI B18.22.1 - No. 5 - narrow - Type
₿.	16	8		SN-025917	•	1	E-Clip
₿.	17	2		SN-025943	•	1	ISO 4035 - M3ISO
₿.	18	6		SN-025944	•	1	ISO 4762 - M3 x 12ISO
₿.	19	2		SN-025945	•	1	ISO 4762 - M3 x 20ISO
	2	1		SN-025893	•	1	Front Upper Arm Mount
₿.	20	2		SN-025905	•	1	Shock Absorber Front

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

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

	1 Item	1 Qty.	1 CAD I	i 🔾 Article number	QI	Q	OV	C Designation
_	1	1	2	SN-025891		•	1	front side plate r
	2	1		SN-025893		•	1	Front Upper Arm Mount
	3	1		SN-025923		۰	1	Front Hub Carrier L
	4	1	2	SN-025939		•	1	Front Hub Carrier R
	5	1		SN-025892		•	1	front side plate l
1×+	6	2		SN-025896		•	1	Front Lower Hinge Pin
¥.	7	2	2	SN-025897		•	1	Front Upper Hinge Pin
1	8	2	2	SN-025904		•	1	Front Suspension Adjustment Clip
1×	9	2	2	SN-025898		•	1	front Upper arm
1×+	11	2	2	SN-025903		•	1	Shock Bushing
	12	1		SN-025894		•	1	Front Shock Brace
1×+	13	2		SN-025901		•	1	Front Lower Arm
	14	1		SN-025895		•	1	Front Lower Arm Brace

Utilized articles: Sequence in the product structure

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

O G Volume [mm <sup>3</sup> ]     VOLUMEN  Attribute description:	
Attribute description:	
sequence in which the article appears for the first time when running through the product struc	ture

#### **Hide internal attributes**

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

Edit attribute configuration							
- Available attributes	]	- Selected attributes					
Filter: Wei	<ul> <li>Hide internal attributes</li> </ul>	Elist Context menu Report Sort					
📰 🖳 🏣 Type Designation	Attribute name						
OOO Weight [kg]	GEWICHT	Open, read-only					

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

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

gs						
directory:						
			遷 Edit att	ribute co	onfiguration	
– Available attrik	butes				- Selected attribu	utes
Filter:		-	🗹 Hide internal attribute	s	🚍 List	Context
Document		2			Pocult	lict dicolar
🗐 💽 Type De	signation	Attribute name		1	Nesuit	inst dispidy.
🖸 🗹 🏐 Op	oen, read-only	UI_OpenItemRead	lOnly		/ Type Designa	ition
🔽 🖉 🇐 Do	ocument number with icon	VA_DocumentNur	mberWithIcon		Open, re	ead-only
🗹 🗹 💷 Inc	dex	HEL_INDEX			Docume	ent number w
🔽 🗹 🇐 File	e size	VA_DocumentFileS	Size		U Index	
🔽 🗹 🎯 Ge	nerated file name	UI_ExportFileName	e		File size	
🔽 🖉 🇐 File	e extension	VA_DocumentFile	Extension		Generation Generation	ed file name
🔽 🗹 🏐 Art	ticle	UI_DocumentLinke	edArticles		File exte	nsion
🔽 🗹 🇐 Pro	oject	UI_DocumentProje	ects		Article	
🔽 🗹 🏐 Fo	lder	UI_DocumentFold	lers	-	Project	
Ad	lopt original file name	HEL_PLAINFILENA	ME		Folder	
Art 🗐 🗆	ticle number	SACHNUMMER				
	sed on	ERSATZFUER				
		CC				
🗆 🗆 🗐 Ch	anged by	HEL_GEAENDERT_	USER			
🗆 🗆 💷 Ch	anged on	HEL_GEAENDERT_	DATUM			
					111	

# Please note:

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

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

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

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

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

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

K HELiOS Options					
General	Select the components that you want to export:				
Display Result lists	General and combined search Settings for the search of records				
Project and Folder structures Product structures Product structures Print / Convert	Settings for full-text search				
Workflow	Display      Settings for window display     Result lists				
Cog	Settings for result lists     Project and Folder structures     Project and Folder structure settings				
Plugins Database	Product structures Settings for product structures Print / Convert Settings for printing				

## Partially activated checkboxes

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

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

😽 ອ Standard	5			
Derive and link	Number	Wor	Locked by	
8	DN-000245	1		
<b>7</b>	A	1		
The sta	te 'PartlyMarked' mean	is that th	e element will	only be li
			1	



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

### Default of project and folder change behaviour

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

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

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

- Combined search	
Project change behaviour:	
O Same project in all masks	
O Different projects in all masks	
<ul> <li>Always project-independent in non-active mask</li> </ul>	
Folder change behaviour:	
○ Same folder in all masks	
O Different folders in all masks	
<ul> <li>Always folder-independent in non-active mask</li> </ul>	

# Print (Spooler)

Bundling

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

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

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

Assign automatically     Select manually    Select manually     General    Copies: 1    Colour:     Q    Output-date-time:    Immediately   Options:    Bundling:     One on all    Document (index-independent)   Priority:    1     Print     Pocument number vind     Wo Designation    Article     Project     Folder     Q     Print      Document number vind     Wo Designation    Article     Project     Folder     Q     Print      Document number vind     Wo Designation    Article     Project     Folder     Q      Print      Document number vind     Wo Designation    Article     Project     Folder      Q      Print      Document number vind      Wo Designation      Assembly drawing      SN-000005      Q      Q      Q      Q </th <th>Printer</th> <th>- Page setup -</th> <th></th> <th></th>	Printer	- Page setup -		
Select manually   Select manually   Select manually   Adobe PDF <th>Assign automatically</th> <th>Paper format:</th> <th>Automatic</th> <th></th>	Assign automatically	Paper format:	Automatic	
Adobe PDF   Postprocessing: None   General   Copies:   1   Colour:   Image:   Orientation:   Image:	Select manually	Paper tray:	Automatically Select	
General   Copies:   1   Colour:   Image:   Output-date-time:   Immediately   Options:   Bundling:   Immediately   Options:   Immediately   Immediately   Options:   Immediately   Immediately </th <td>Adobe PDF</td> <td>Postprocessing</td> <td>None</td> <td></td>	Adobe PDF	Postprocessing	None	
Copies: 1 Colour: Cour: Courtait Count C	General	Scale:	Automatic	
Colour:  Col	Copies: 1		O Scale: 100 <sup>●</sup> %	
Conduit   Orientation:    Output-date-time:     Immediately    Options:     Bundling:     One    All     Output-date-time:     Immediately    Options:           Bundling:      One    All     Output-date-time:           Bundling:      One     All      Output-date-time:     Display="block">Immediately     Options:      Bundling:      One     All      Output-date-time:     Bundling:     Display="block">Immediately     Options:     Bundling:     Priority:     1      Output-date-time:     Display="block">Immediately     Options:      Priority:      1      Priority:      1      Priority:     Priot:      Priot:     Priot:            Priot:      Priot:      Priot:      Priot:      Priot:      Priot:<	Colours 2		Scale line widths	
Output-date-time: Immediately   Immediately Options:     Bundling: Immediately   Immediately <td< th=""><th></th><th></th><th></th><th></th></td<>				
Output-date-time: Immediately   Bundling: None   All   Priority: 1   Immediately			Page margins	
Bundling:  None All Document (index-independent)  Priority:	Output-date-time: 💿 Immediately 🔘 Options:		Banner	
Image: Standard         Standard         Project         Folder           Print         Image: Polynonononononononononononononononononon	Priority: 1 🔴	User-defined	i	
Print         Image: Second system         Document number vind         Wo Designation         Article         Project         Folder           Image: Very Second system         Very Designation         Article         Project         Folder         Folder           Image: Very Second system         Very Designation         Assembly drawing         SN-00003         PN-01-06-K         Image: Very Second system         PN-01-06-K         Image: Very Second system         PN-01-06-K         Image: Very Second system         Image: Very Second system         PN-01-06-K         Image: Very Second system         Image: Very Second system         PN-01-06-K         Image: Very Second system         Image: Very Second system         Image: Very Second system         Image: Very Second system         PN-01-06-K         Image: Very Second system         Image: Ver	Priority: 1 💌	User-defined     All     As last savee     Include mode	d lel area	
☑         ☑	Priority: 1 👻	User-defined     All     As last saved     Include mode	i i lel area	
☑         ☑	Priority: 1 + + + + + + + + + + + + + + + + + +	User-defined     All     As last saved     Include mod	j j lel area Project Fo	lder
✓ Image PR-000007 Image PR-01-06-K Image PR-01-06-K Image PR-01-06-K Image PR-01-06-K	Priority: 1 + Compared to the second	User-defined     All     As last saved     Include mod	I I Iel area Project PN-01-06-K I	lder
	Priority: 1 Standard rint Document number vInd Wo Designation Article Document number vInd Assembly drawing SN-000 Assembly drawing SN-000 Assembly drawing SN-000	OUD3     OUD3	Project Fo PN-01-06-K	lder
	Priority: 1 Standard rint Document number vInd Wo Designation Article Document number vInd Assembly drawing SN-000 Assembly drawing SN-000 Assembly drawing SN-000 Assembly drawing SN-000 Assembly drawing SN-000 Assembly drawing SN-000 Assembly drawing SN-000	Outright Street and Street a	Project Fo Project Fo PN-01-06-K Internet PN-01-06-K Internet PN-01-	lder

Job overview

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

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

4 👼	Standard	•				
Status	Document numb	Article number	Designation	Туре	Creation	Sched
Compl	et 2000-000005	SN-000003	Housing assembly	Print job	07.05.2025 09:05:01	07.05.2
Compl	et PDN-000005	SN-000003	Housing assembly	Print job	07.05.2025 09:05:01	07.05.2

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

📑 List	📑 Context menu	41 Sort	
Type O	niect type Designation	Sort sequence	Format
Туре О	bject type Designation	Sort sequence	Format

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

٠		General
	1	Edit priority
		Remove all completed jobs
		Pause all jobs
		Continue all paused jobs
ą		Repeat all failed jobs
_0		Remove all failed jobs

Output different formats of a print job on different printers

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

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

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

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

Jobs —			-						
<b>∻</b> (	(D) (D) (B) (D)	Standard	•						
Status	Document numbe	Article number	Designation	Туре	Creation	Scheduled time	Completed	User	Output
√Compl	et 🔊 DN-000005	SN-000003	Housing assembly	Print job	07.05.2025 09:05:01	07.05.2025 09:05:01	07.05.2025 09:05:34	Administrator	Printer10
✓Compl	et 🐊 DN-000005	SN-000003	Housing assembly	Print job	07.05.2025 09:05:01	07.05.2025 09:05:01	07.05.2025 09:05:47	Administrator	Printer12

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

Context menu in the Value column of the Properties window

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

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



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

Properties X Grap	ohic	Create index (with file selection)
Description	Valu	🕜 Create derivation
Number	9	Screate derivation, with link
Index		Create follow-on sheet
Workflow status	•••	Create follow-on sheet, with link
Designation	Dra	Se Derive variant
Document type	HiC	Create variant index
Creation date	02.1	Import file
Created by	Kon	import me
File changed on	24.1	Output
Locked by		Show Version Management
Locked on		Export file
		Pack & Go

# Virtual document attribute: File extension

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

- Av	ailat	ole a	ttrib	utes					- Sel	ecte	d attributes —	TA
Filter	-						· ·		~~ [		ontext menu	Report
	Docu	mer	nt						1	I	Result list displa	y: Document
			Туре	Designation		Attribute name			1	Type	Designation	
				Document ID (Head	)	HEL_KOPFID	*			(B)	Open read only	
				Document number		HEL_DOKUNUMMER			H	(8)	Open + Edit	
			۲	Document number with ico Document type		VA_DocumentNumberWithI DOCUMENT_TYPE				184	Document number with ico	
											Index	1 with icon
				Drawing origin		ZNRURSPRUNG			H	180	Index up to dater	000
				Drawing type		ZEICHNART			H	9	Index up-to-dater	less
				DV STR Update		HEL_DV_STR_UPDATE	_		F		Designation	
				E-mail ID		MAILID			H		Creation data	
				File changed by		HEL_FILE_GEAENDERT_US	ER		H	9	Creation date	
				File changed on		HEL_FILE_GEAENDERT_D	ATL		H	9	Cleated by	
~	$\overline{\checkmark}$		٢	File extension		VA_DocumentFileExtensio	on		H		File changed on	
				File name		HEL_FILENAME			H		Locked by	
				File name for HiCAD	)	HEL_DATEINAME			H	100	Cile extension	
			٢	File size		VA_DocumentFileSize			-	4	File extension	
				File status		HEL_STATUS	*					
			Do	ocuments ×								
			4	🖣 📑 🍓 Sta	ndard	• 7 9	ĸ				•	
				Number	Ind ●	Designation	Docum	nent typ	e	F	ile extension	
			2	😂 🛞 DN-000026	•	Gear wheel geometry 2-	HICAD	Drawin	g	.5	sza	
				🤛 👰 DN-000033	•	Production drawing	HICAD	Drawin	g	.5	sza	
			-	😂 🔊 DN-000034	•	3-D model	HICAD	Part/Va	riant	ł.	kra	
			2	CDN-000035	•	3-D model	HICAD	Part/Va	riant	4	cra	

Production drawing
 HiCAD Drawing

Office Document

Office Document

Office Document

Word Document

Word Document

.sza

.docx

.docx

.docx

-

-

> PDN-000036

📂 🔤 DN-000242

DN-000243

🕫 😇 DN-000244 🛛 🔵 Word Document

## Plugin management in the HELiOS Options

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

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

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

	0	Info	
Do you w	vant to select dep	pendent files f	or the plugin?
		No	Yes

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

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

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

Loading behaviour		Loaded	Signed	Name
Load	•		•	ExamplePlugin1

You can also influence the Loading behaviour of plugins there.

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



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

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

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

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

## Activate functionality for HiCAD Plant Engineering

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

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

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

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

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



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

## **HELiOS Mail Proxy**

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

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

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

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

Asynchronous execution of viewers

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

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

### Improvements and performance increases

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

This includes the error output process during update installations.

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

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

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

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

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

Update		Software-Version:
		331
		Database-Version:
	1	330
	Lindata databasa	Database ID:
	Update database	B00CG5Y719S30P
	HeliosDbUpdate	×
	Actions are en These are no Please contac	ntered in the action list when deleting objects. longer executed with the current version. t ISD Consulting.
		ОК
		database

# Service Pack 1

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

The context menus for projects and folders, as well as the Ribbon menus for the project and folder detail masks,

now include the new functions Copy HELiOS URL (Project Explorer) and Copy HELiOS URL (Folder Explorer)

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



		嬣 AN-100/06 (F	older)			- 0	×
General	Assignments	put / Output				* 🖻	$\bigcirc$
Enter Derive, v Derive, v	with content w/o content put	•   ut					
Mask	X Object	t files Print documents older of folder	Call report Output	Copy HELIOS URL (	ppy HELiOS URL Folder Explorer)	Show ID	ŀ
	Basic information						
	Folder name:	AN-100/06		1.5.1			
	Assignment:						
	Designation:	Slip-on gear					
	Comment:	Modul=2; z=22/36					
	Extended informatio	n					
	Creation date:	03.10.2006					
	Created by:	Administrator					

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

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

# Export file and Pack & Go

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

The additional settings dialogue has been removed.

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

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

Settings								
Export directory:								
	Conv to individua	I nath O Keen folder struct	ture					
File name:	copy to manuada	path O keep tolder struct	uic					
Attribute assignm	entr Standard	•	Destauri	11 J	0 (+2	105		
	ents standard	<u> </u>	Replace inv	alid characters with unde	rscore (V:"(<	>  ).		
O Individual specific	ation							
As saved								
Options:								
As ZIP archive	IP archive name							.zip
<ul> <li>With export inform</li> <li>With export and and</li> </ul>	nation •							
<ul> <li>With export inform</li> <li>With export inform</li> <li>Standard</li> <li>Document number w</li> </ul>	nation • Ind File size	Generated file name	Extensio	rArticle		Project		Folder
With export inforr  With export inforr  Standard  Document number w  DN-000001	Ind File size	Generated file name	Extensio	r Article SN-00001	<u> </u>	Project PN-01-06-K	8	Folder
With export inforr Standard Document number w Document number w Document number w	Ind File size 2355,48 KB 1172,04 KB	Generated file name DN-000001 DN-000002	Extensio sza kra	r Article SN-000001 SN-000002	<u></u>	Project PN-01-06-K PN-01-06-K	<u>×</u>	Folder
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With export inforr           With export inforr           Standard           Document number w           Dolocon           DN-000001           DN-00002           DN-00003           DN-00004           DN-00005	Ind File size 2355,48 KB 1172,04 KB 1386,29 KB 507,02 KB 626,56 KB	Generated file name           DN-000001           DN-000002           DN-000003           DN-000004	Extension sza kra sza kra sza	Article SN-000001 SN-000002 SN-000002 SN-000003 SN-000003	© © © ©	Project PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K		Folder
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With export inforr           With export inforr           Standard           Document number w           DOL00001           DOL00001           DN-00002           DN-00003           DN-00004           DN-00005           DN-00006           DN-00006	Ind File size 2355,48 KB 1172,04 KB 1386,29 KB 507,02 KB 626,56 KB 614,08 KB 501,75 KB	Generated file name           DN-000001           DN-000002           DN-000003           DN-000004           DN-000005           DN-000006	Extension sza kra sza kra sza sza sza sza	Article SN-000001 SN-000002 SN-000002 SN-000003 SN-000003 SN-000004 SN-000005		Project PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K		Folder
With export inforr           With export inforr           Standard           Document number w           PDN-00001           PDN-00002           PDN-00003           PDN-00004           PDN-00005           PDN-00006           PDN-00007           PDN-00008	Ind File size 2355,48 KB 1172,04 KB 1386,29 KB 507,02 KB 626,56 KB 614,08 KB 501,75 KB 162,15 KB	Generated file name           DN-000001           DN-000002           DN-000003           DN-000004           DN-000005           DN-000006           DN-000007	Extensic sza kra sza kra sza sza sza sza kra	Article SN-000001 SN-000002 SN-000002 SN-000003 SN-000003 SN-000004 SN-000005 SN-000006		Project PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K		Folder
With export inforr           With export inforr           Standard           Document number w           PDN-00001           PDN-00002           PDN-00003           PDN-00004           PDN-00005           PDN-00006           PDN-00007           PDN-00008           PDN-00009	Ind File size 2355,48 KB 1172,04 KB 1386,29 KB 507,02 KB 626,56 KB 614,08 KB 501,75 KB 162,15 KB 482,53 KB	Generated file name           DN-000001           DN-000002           DN-000003           DN-000004           DN-000005           DN-000006           DN-000007           DN-000008	Extensic sza kra sza kra sza sza sza sza kra sza	Article SN-000001 SN-000002 SN-000002 SN-000003 SN-000003 SN-000004 SN-000005 SN-000006		Project PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K		Folder

Print (Spooler)

## Automatic selection of paper format

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

				– Page setup			
O Assign automatically			÷	Paper format: Au	tomatic		2
Select manually				Paper tray: Au	to source selection		
Image: Antiperiod A	10		*	Postprocessing: N	ne		
				reseprocessing.			
General				Scale: O Automatic			
Copies: 1			0	Scale: 100 📮 %			
Colour: 🗹				Scale line widths			
Orientation: 💿 Automatic (	) Portrait () Landscape				Page m	nargins	
Output-date-time: 💿 Immediately	O Options:				Bi	anner	
Priority: 1					rea		
🐓 蔢 🍓 Standard							
🐓 🎑 🍇 Standard int 🛛 🕞 Document numb	r w Ind  WoDesignation	Article	Project		Folder		Sheet area
int 🔤 🍉 Standard	<ul> <li>w Ind ● Wo Designation</li> <li>Oraught</li> </ul>	Article SN-000001	Project	8	Folder AN-100/06	<u>()</u>	Sheet area
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Standard           int         Image: Document number of the product	r w Ind Wo Designation Draught Assembly drawing Assembly drawing	Article SN-000001 SN-000002 SN-000003	Project Project PN-01-06-K N-01-06-K N-01-06-K	8 8 8	Folder AN-100/06 	<u>()</u>	Sheet area Sheet 1 8 Sheet 1 8 Sheet 1 8
Standard           fint         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constraint of the standard           Image: Constraint of the standard         Image: Constandard </td <td>r w Ind W C Designation C Draught Assembly drawing Assembly drawing Assembly drawing</td> <td>Article SN-000001 SN-000002 SN-000003 SN-000004</td> <td>Project Project PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K</td> <td>ହ ହ ହ ହ</td> <td>Folder  AN-100/06 </td> <td>8</td> <td>Sheet area Sheet 1 8 Sheet 1 8 Sheet 1 8 Sheet 1 8</td>	r w Ind W C Designation C Draught Assembly drawing Assembly drawing Assembly drawing	Article SN-000001 SN-000002 SN-000003 SN-000004	Project Project PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K PN-01-06-K	ହ ହ ହ ହ	Folder AN-100/06 	8	Sheet area Sheet 1 8 Sheet 1 8 Sheet 1 8 Sheet 1 8

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

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

uments function is used.

You can choose between three options:

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

📻 General 🔍 Find	Settings for Print / Convert					
🗊 Input	- Print documents (Spooler)					
Display	Pre-definition of sheet areas to be printed:					
Result lists	<ul> <li>User-defined</li></ul>					
Project and Folder structures	Documents linked to an article that are offered for printing:					
😤 Product structures	○ All ○ CAD drawings only ④ Filter: CAD Document search_gl	• • •				
Print / Convert	Link classes that are taken into account for determining the article conte	xt when printing documents:				
Import / Export	🗆 Link class 📵					
Workflow	AllgTyp-Geometrie					
Log	Bauteil(e)-Konstruktion Bauteil-Konstruktion	_				
e-mail	Bauteil-Teilegeometrie					
Preview	Bill of Materials (BOM)	*				
Database	Final check for linked articles:  Must be in the same selected project as the document to be printe	d				
Document type	Must be in the same selected folder as the document to be printed	1				
😕 HELiOS Desktop	Must have been selected if available					

#### **K HELiOS Options**

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

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

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

Link class 📵	
AllgTyp-Geometrie	A
Bauteil(e)-Konstruktion	
Bauteil-Konstruktion	
Bauteil-Teilegeometrie	
Documentation	-

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

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

				Link classes			×
🖘 🗟 🍇 Standard	•						
Name	Source class	Cardinality	Target class	Description	Release-relevance	Automation, source	Automation
3DAssy-PID	Dokumentversion	3	Dokumentversion	Linkage of assembly (3D referenced part) and P&ID chart	0	1	1
3DPlant-PID	Dokumentversion	3	Dokumentversion	Linkage of Plant Engineering drawing and P&ID chart	0	1	1
AllgTyp-Geometrie	Bauteilversion	4	Dokumentversion	Allgemeiner Typ mit VAA-Dokument	0	0	2
AllgTyp-Variante	Bauteilversion	2	Bauteilversion	General type with sub-types	0	2	1
AnlBauteil-ZubSatzN	Bauteilversion	3	Bauteilversion	Linkage of part and accessory set	0	0	0
Bauteil(e)-Konstruktion	Bauteilversion	3	Dokumentversion	Model/Drawing with several single parts	0	1	1
Bauteil-Konstruktion	Bauteilversion	3	Dokumentversion	Model, Part/assembly drawing	2	1	1
Bauteil-Teilegeometrie	Bauteilversion	3	Dokumentversion	3-D body, 2-D figure	2	1	1
Documentation	Bauteil	3	Dokument	General version-independent supplementary documentation for one or several part(s)	0	0	0
Document-Document	Dokumentversion	3	Dokumentversion	Link between two documents	0	1	1
E-Mail Attachment	Dokumentversion	3	Dokumentversion	E-Mail Attachment	0	1	1
Markup	Dokumentversion	3	Dokumentversion	Links an original document to a document with notes information (markup/redline)	0	0	0
Markup abgelehnt	Dokumentversion	3	Dokumentversion	Verknüpft ein Originaldokument mit einem Dokument mit Notizinformationen (Markup/Redlin	0	0	0
Notizdokument	Dokumentversion	3	Dokumentversion	Verknüpft ein Originaldokument mit einem Dokument mit Notizinformationen (Markup/Redlin	0	0	0
Notizdokument abgelehnt	Dokumentversion	3	Dokumentversion	Verknüpft ein Originaldokument mit einem Dokument mit Notizinformationen (Markup/Redlin	0	0	0
Notizdokument angenommen	Dokumentversion	3	Dokumentversion	Verknüpft ein Originaldokument mit einem Dokument mit Notizinformationen (Markup/Redlin	0	0	0
Part-Document	Bauteilversion	3	Dokumentversion	Link between a part and a document	0	1	1
Pipeline-Isometry	Bauteilversion	3	Dokumentversion	Linkage of referenced part of pipeline and isometry document	0	1	1
Pipeline-Layoutplan	Bauteilversion	3	Dokumentversion	Linkage of referenced part of pipeline and Plant Engineering drawing	0	0	1
Pipeline-Spool	Bauteilversion	3	Dokumentversion	Linkage of referenced part of pipeline and spool document	0	1	1
Teilegeometrie-Konstruktion	Dokumentversion	3	Dokumentversion	3-D body, 2-D figure, linked to Model/Drawing	1	0	1
							•
Rows: 21							

#### Improved user guidance

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

# HiCAD Viewer: Project and folder context

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



# HELiOS Options: Attribute assignments for Cadmium Cloud

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

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

	- create new attribute assignment	ient for a model proper	rty		
- Source					
	Converter:	•			
Value		Converter			
0	et tree (DOCUMENT TVDE Decument)				
Docume	ent type (DOCOMENT_TYPE, Document)				
J Docume	nt type (DOCOMENT_TYPE, Document)				
U Docume	nt type (DOCOMENT_TTPE, Document)				
J Docume	nt type (DOCOMENT_TTPE, Document)				
Jocume	nt type (DOCOMENT_TTPE, Document)				
Docume	nt type (DOCOMENT_TTPE, Document)				
Docume     Docume     Docume	nt type (DOCOMENT_117E, Document)				
Docume     Docume     Destination     Property:	Description				
Docume     Docume     Property:	n Description	· · · · · · · · · · · · · · · · · · ·			
Docume     Docume     Destination     Property:     Constraint	n Description	· · · · · · · · · · · · · · · · · · ·			

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

id 🛧	Disolay name	Description	Date	GUID	File	Upload	Creator
IG 1	Display hame	Description	Date	0010	Size	state	Creator

# Performance optimisation of the PDF preview

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

# System link

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

A new column has been added to the corresponding overviews:

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

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

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

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

Document			nt	🕥 Article	<b>4</b> ▶ Link	💫 Article/document li		
🗄 😹 🔚 Type		Туре	Designation		Attribute name			
•			3	Name of link class		VA_LinkldWithlcon		
				Automation, sour	ce	HEL_QAENDERUNG		
				Automation, targe	t	HEL_ZAENDER	JNG	
				Cardinality		HEL_KARDINAL	ITAET	
				Description		HEL_BESCHREI	BUNG	
				Ignore write prote	ction, source	HEL_SG_ANLEG	GEN_Q	
				lgnore write-prote	ction, target	HEL_SG_ANLEGEN_Z		
				List-relevance		HEL_DISPLAY		
				Name		HEL_RELID		
				Release-relevance		HEL_FREIGABE		
				Source class		HEL_QUELLKLASSE		
			۲	System link		VA_SystemLink	Class	
				Target class		HEL_ZIELKLASS	E	
Attr	ibute	e des	cripti	on:				
nfo can	not b	tion of	on wi ange	d or deleted.	s is predefined	by the system a	and therefore	

# **HELiOS Options: Product structures**

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

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

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

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

Example:

	HELiOS Desktop
۹	HELiOS 2025 Version 30.1.0.265
	64Bit Version
Datab ODBC Server Datab	ase ID: B00K71YIVXBQ2M data source: bauteil_en_3000 : DEDTM066\SQLEXPRESS ase: bauteil_en_3000
© 199	-2025 ISD Software und Systeme GmbH
This p Softwa	ogram is part of the software family by the IS re und Systeme GmbH. All rights reserved.
This is unlaw subjec	a copyrighted product. Illegal copying or ul use is a criminal offence. Violations are t to criminal prosecution and to claims for jes.
damag	

# Locking Manager: Unlocking independent of the host computer

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

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

# Errors of last HELiOS update

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



Automatic saving of dumps when HELiOS crashes

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

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

# Major Release

## Improvements of the UI and user guidance

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

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

	3/ HELIOS Options
General	Settings for the HELiOS Desktop
Display Result lists	Settings for the Workspace     Automatically end editing of documents without local changes after closing the application     Remove documents that are currently not being edited from Workspace after closing the application     Remove locally changed files
Product structures Print / Convert Print / Export	Attribute assignments      Transfer to HELIOS: Attribute import      Product Explorer      Astroportically above last and tage after UTUIOS start
Workflow Control Contr	
Plugins Database Document type	
HELiOS Desktop	
Manage •	Apply Cancel OK

## Revised preview and viewer integrations

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

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

		🔀 HELIC	OS Options			
🚰 General	Settin	igs for Preview				
Tinout	- Show previ					
an put	Always im	modiatoly (may take)	onger)			
Display	Aiways in	mediately (may take	onger)			
Z Result lists	Up to a file	e size of U M	B immediately, othe	erwise only by clicking	g in the preview area	3
Project and Folder structures	O Only by cli	icking in the preview	area			
Product structures	- Viewer assi	ignments				
Print / Convert	There are diffe locations liste	erent places in HELiO d below. For each sp	S where a preview of ecified file extension	an be displayed. We n, each location can b	basically differentiat e configured separa	te between the 4 ately
🕌 Import / Export	Docume     the article	nt mask: Integrated p display.	preview in the Mask	tab of the documen	t display, as well as i	in the Document tab o
Workflow	Graphic	tab: Preview in the Gr	aphic tab in the do	cument and article di	splav.	
1	• Graphic	window: Proviou in d	ockable graphic wir	adow of the main wir	adow.	
Log	Giapriic	window. Fleview in d	· · · · · · · · ·	luow of the main wir	idow.	
E-mail	Explorer:	Preview in the Graph	lic tab of the variou	s explorers (e.g. Proje	ect Explorer).	
Deres ince	ı   🗣 📉 🖒					
	File extensio	n Standard	Document mask	Graphic tab	Graphic window	Explorer
Plugins	PDE	Mozilla PDE view (i	Standard	Mozilla PDE view	Mozilla PDE view	Standard
Database	574	HiCAD Viewer (ima	Standard	HiCAD Viewer	HiCAD Viewer	Standard
	KRA	HiCAD Viewer (ima	Standard	HiCAD Viewer	HiCAD Viewer	Standard
Document type	IAM	Preview graphic (th	Standard	Preview graphic (H)	Preview graphic (H	Standard
HELiOS Desktop	IPT	Preview graphic (th	Standard	Preview graphic (H)	Preview graphic (H	Standard
	.IDW	Preview graphic (th	Standard	Preview graphic (H	Preview graphic (H	Standard
	TIF	Windows Explorer	Standard	Image viewer	Image viewer	Standard
	TIFF	Windows Explorer	Standard	Image viewer	Image viewer	Standard
	JPG	Windows Explorer	Standard	Image viewer	Image viewer	Standard
	JPEG	Windows Explorer	Standard	Image viewer	Image viewer	Standard
	.PNG	Windows Explorer	Standard	Image viewer	Image viewer	Standard
	- Cadmium (	loud		* * *		
	Cadimum C					
	Server addres	s: https://				
	Login:					
	Password:					
	File formats:					
	File extensio	n				
	- Attribute a	ssignments for Cad	mium Cloud			
	Export from	HELIOS: Attribu	ute export			
Manage *	7				Apply	Cancel OK

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

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

HELiOS generally distinguishes between integrated and external viewers:

 Integrated viewers are viewers that HELiOS provides automatically. A separate installation by the user is not necessary. Examples include the **Image Viewer** for Bitmap formats, **Mozilla Pdf** for previewing PDF files or the **xbim toolkit** for IFC files.

Changes to the configuration of the viewer assignments have a direct effect for integrated viewers (i.e. without restarting HELiOS).

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

.IPN	Windows Explorer
.SZA	HiCAD Viewer (image display) 👻
.KRA	
.VAA	HiCAD Viewer (image display)
.PAA	HiCAD Viewer Windows Explorer

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

Clicking on the **1** icon in the **Viewer assignments** area of the dialogue window displays a list of all viewers supported by HELiOS with further information:

External vie	wers supported by HELiOS					
Availability	Name	Version	Minimum version	Available file extensions		
•	DWG TrueView	24.2.50.0	24.2.192.0	.dwg, .dxf		
•	eDrawings		31.5.0.33	.3dxml, asm, asmdot, .cal, .ct1, .dlv, .drwdot, .dwg, .dxf, .easm, .easmx, .edrw, .edrwx, .eprt, .eprtx, .exp, .hmf, .hsf, .iam, .ifc, .iges, .igs, .ipt, .jt, .model, .neu, .obj, .par, .prt, .prtdot, .psm, .pwd, .sab, .sat, .session, .sldasm, .slddrw, .sldprt, .step, .stl, .stp, .stpz, .x_b, .x_t, .xas, .xmt, .xmt_txt, .xpr		
	HiCAD Viewer	30.0.0.109	29.0.2.248	.fga, .kra, .paa, .sza, .vaa		
•	HiCAD Viewer (image display)	30.0.0.109	29.0.2.248	.fga, .kra, .paa, .sza, .vaa		
•	Kisters 3D Viewstation		2024.3.317.0	.3dm, .3ds, .3dvs, .3dxml, .3mf, .arc, .asm, .bmp, .catdrawing, .catpart, .catproduct, .catshape, .cgm, .cgr, .cpixml, .dae, .dft, .dgn, .dlv, .doc, .docx, .drw, .dwf, .dwfx, .dwg, .dxf, .exp, .fbx, .ger, .gif, .glb, .gltf, .iam, .ifc, .iges, .igs, .ipt, .jp2, .jpeg, .jpg, .jt, .mf1, .mil, .model, .neu, .nwd, .obj, .par, .pdf, .pkg, .plmxml, .plt, .png, .ppt, .pptx, .prc, .prt, .psm, .ptr, .pwd, .ifa, .rvt, .sat .sat, .session, .sldasm, .slddrw, .sldprt, .step, .stl, .stpx, .stpxz, .stpz, .svg .tif, .tiff, .u3d, .unv, .vda, .vrml, .vsxml, .webp, .wrl, .x_b, .x_t, .xas, .xls, .xlsx, .xmt, xmt_txt, .xpr		
	Office (Excel)	16.0.17830.20138	15.0.0.0	.xla, .xlam, .xls, .xlsb, .xlsm, .xlsx, .xlsxm, .xlt, .xltm, .xltx, .xltxm		
•	Office (Outlook)	16.0.17830.20138	15.0.0.0	.msg		
٠	Office (PowerPoint)	16.0.17830.20138	15.0.0.0	.pot, .potm, .potx, .potxm, .pps, .ppsm, .ppsx, .ppsxm, .ppt, .pptx, .pptxm, pptm		
•	Office (Word)		15.0.0.0	.doc, .docm, .docx, .docxm, .dot, .dotm, .dotx, .dotxm		
Viewers sup	plied with HELiOS	Version		Available file extensions		
		Version		hmp aif ico ipe ipea ipa ixr ppa tif tiff		
Mozilla PDF p	review graphic	4.3.136.0				
Mozilla PDF view		4.3.136.0		.pdf		
Preview graphic (HQ)				.dwgiamidwipnipt		
Preview graphic (thumbnail)				.dwg, .iam, .idw, .ipn, .ipt		
Video plaver		8.12.0		.mp3, .mp4, .wav, .webm		
Video player	Web browser			.htm, .html, .txt, .url		
Video player Web browser	Windows Explorer			*		
Video player Web browser Windows Expl	orer	1110		.ifcifcxmlifczipxbim, .xbimf		

The new configuration of the preview is saved user-specifically. (In older HELiOS versions, it was identical for all logged-in users).
The **Preview** settings can also be managed via the export/import mechanism of the UI settings in HELiOS:

HELIOS Options
Special settings from the HELiOS options or various areas
General 🚨 General Settings
Settings for the search of records
Input of data records
Settings for window display
Settings for result lists
Project and Folder structures Project and Folder structure settings
Settings for printing
Settings for import and export
Vorkflow 🚨
Settings of directories for logs
Settings for preview
Viewer assignments and general settings for preview
Cadmium Cloud 🗐 Settings for preview with Cadmium Cloud
HELIOS Desktop 💩 Settings for the HELiOS Desktop

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



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

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

## Export and import of UI settings

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

HELIOS Options
Special settings from the HELiOS options or various areas
👦 🚛 General 🚨
General Settings
Find 🚨
Settings for the search of records
👝 📑 Input 🚨
Input of data records
👝 🕅 Display 🔱
Settings for window display
👝 🧰 Result lists 🤱
Settings for result lists
👝 💻 Project and Folder structures 🔱
Project and Folder structure settings
👝 🏹 Print / Convert 🚨
Settings for printing
👝 д 🐙 Import / Export 🔱
Settings for import and export
👝 📣 🔲 Workflow 🚨
Settings for workflows
👝 🚎 Log 🚨
Settings of directories for logs

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

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

는 Import		
Select the components that you want to import:		
Tab layout and positions The tab layouts and window positions (for e.g. Project Explorer, Favourites list, etc.) in the HELiOS Desktop r	main window.	Î
Tab layout 🚨 The tab divisions within the windows (e.g. tabs in the detail dialogue, document tabs in the Project Explorer	r, etc.).	- 1
Masks II All adapted masks		
Result list templates and contained result list representations for all result lists		
Attribute assignments I Attribute assignments for import/export		
QuickAccess Toolbar 🚨 Commands of the Quick Access toolbar in the HELiOS Desktop		
Contents of the Favourites result list in the HELiOS Desktop		
Customizations of the context menus differing from defaults		
HELIOS Options Special settings from the HELIOS options or various areas		
General 🚨 General Settings		
Settings for the search of records		- I
Input 🚨 Input of data records		
Settings for window display		
Settings for result lists		
Project and Folder structures a Project and Folder structure settings		
Settings for printing		
Settings for import and export		
Vorkflow 🚨 Settings for workflows		
✓ Image: Settings of directories for logs		*
	Cancel	ОК

#### Messages in case of non-printable documents

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

Print / Convert	Documents linked to an article that are offered for printing:							
🚰 Import / Export	O All ○ CAD drawings only ○ Filter:							
Workflow	Link classes that are taken into account for determining the art	ticle context when printing documents:						
Jan Log	Link class 📵							
F-mail	AllgTyp-Geometrie	-						
	Bauteil(e)-Konstruktion							
Preview	Bauteil-Konstruktion							
Database	Bauteil-Teilegeometrie							
	Documentation	*						
Document type	Final check for linked articles:							
HELiOS Desktop	Must be in the same selected project as the document to be printed							
	Must be in the same selected folder as the document to	be printed						
	Must have been selected if available							
	Messages for non-printable documents:							
	Show info message if linked documents are to be printed printing	d for which no driver is configured for						
	Show info message if documents of a project or folder a configured for printing	re to be printed for which no driver is						
	Notes document:							

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

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





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

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

	Warning		
Not all selected documents can be printed a file extensions: .KRA. Do you want to continue?	as no driver is configur	ed for printing for t	he following
		No	Yes

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

#### Temporary editing of utilized items

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

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

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



## Product structure: Reserve 1-level assemblies for editing

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

Project Explorer Folder Explorer Produ	uct Explorer × Article Class Explorer Docum
🔍 Find	<ul> <li>↓a ↑a</li> </ul>
	<b>↔</b> ₽.
✓	- marce Data durat
> 🕥 1, 1, SN-000005, , Pinion assen	General
> 🕥 2, 1, SN-000004, , Gear wheel a	Add top article (to window)
> 🐼 3, 1, SN-000003, , Housing ass	Remove top article (from window)
> 🕥 4, 1, SN-000002, , Clamping el	Reserve item
🕥 5, 1, SN-000015, , Sealing wash	Apply item edits
🔯 6, 1, SN-000016, , Sealing wash 🐘	Cancel item editing
😳 99, 1, SN-000017, , Lubricant R 🛛 🐌	Show article
	Edit article master
*	Add to favourites
	Input
1	Enter item (via Find article)
1	Enter item (via Enter article)
<b>B</b> LE	Derive item

Please note in this context:

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



Editing is possible in all dialogue windows with this marking.

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

	- □ >				
Show all locked	objects	5 💿	Show	v only my locked objects	
Articles $\times$	Docu	ment	ts	$\times$   Folders $\times$   Projects $\times$ 1-level asse	mblies 🗙 🛛 📢
✤	lected	1-le	vel a	sembly(-ies) 🐺 🌆 Standard 🔹	6 M
Article number	Inc	Inc	Wc	Designation	Reservation time
SN-000001		16.09.2024 13:38:3			
Rows: 0					
				Unlock all objects	

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

- Av	ailabl	le att	ribute	5		-
Filter	-					•
0	Articl	e				
			Туре	Designation	Attribute name	Γ
ō	0		۲	Workflow status	VA_ObjectWorkflowStatus	^
			۲	Target date	VA_ObjectWorkflowTargetDate	
			۲	Article number with icon	VA_PartNumber	
			۲	Workflow status	VA_PartReleaseStatus	
<b>V</b>	<		۲	Reservation time	VA_ReservedProductsReserveTime	
<b>V</b>	•		۲	Username	VA_ReservedProductsUserName	
				Volume [mm³]	VOLUMEN	
				Preferred type	VORZUGSTYP	

### HELiOS-URLs

**Opening and activating Project and Folder Explorers** 

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

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

The new partial commands in a URL row are:

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

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

#### Show newest index

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

Example of a link:

#### helios://show-article?attributes=%7bHEL\_SACHNUMMER:SN-000016%7d&indexStrategy=NewestRevision

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

#### Change pipe class assignment

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

unction selection:			Access rights granted for following f	functions:
Function			Function	
change documents change folders change general document change material change parts list change pipe class Change pipe class assignment change projects create customers create design release create documents	I	-> <-	change parts create parts delete parts view parts	

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

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



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

#### Note: Adding the KRA version number for update installations

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

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

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

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

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

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

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

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

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

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

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

(\*plus Location ID and User ID).

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

# **HELIOS in HiCAD**

# Service Pack 2

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

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

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

Document	0	Cocument reference struct	<b>,</b> Insl
✓ DN-013498		Not assigned	Change title
<ul> <li>DN-013501</li> </ul>	0	🚰 Outdated	Horizontal alignment
<ul> <li>DN-013500</li> </ul>	0	🚰 Outdated	✓ With text
DN-013499	0	🚰 Outdated	Representation type
<ul> <li>DN-013503</li> </ul>	0	Current	Save
DN-013502	0	Current	Reset

#### Performance increase for derivations

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

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

## HELiOS user in HiCAD window caption

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



# Service Pack 1

# HELiOS Ribbon with new functions

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

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

											HELIOS
New Activate Independent Display Ex	Plorer New Activate Independent Display Explore	r Docu Project Workfl	low Article Pro	ject Workflow	Assign U	pdate Docu Project Work	flow Article Assign Project V	Vorkflow Assign Prod	Drawing On	Off Login Net	w Link Check
Project context	Folder context	HEDO	S PDM functions	for active draw	ing		HELIOS PDM functions for	active part	Man	agement	Others
	Project Explorer – – – – – –										- 🗆 ×
Q	+ ↓a, ↑a,	Articles 3	×								-
PN-01-06-K	<b>∲ ₽</b>	🤸 🖳 👼	🚡 Standa	rd		• 7 %					
Name	Designation	Article number	Ind We	Designat	ion	F	Part type	Designation St	andard designatio	Creation date	Created I
PN-01-06-K	Construction Documents	SN-026018					art	*		11 12 2024	Adminis +
		SN-000001		Slin-on c	ear mech	anism F	Product			02 10 2006	Konstruk
		SN-00002		Clampin	a element	t 4	Assembly			02.10.2006	Konstruk
		SN-000003	00	Housing	assembly	, k	Assembly			02.10.2006	Konstruk
		SN-000004	• 0	Gear whe	el assemi	bly A	Assembly			02.10.2006	Konstruk
		SN-000005	• 0	Pinion as	sembly	4	Assembly			02.10.2006	Konstruk
		SN-000006	• 0	Gearbox	housing	F	Part			02.10.2006	Konstruk
		SN-000007	• 0	Gearbox	cover	F	Part			02.10.2006	Konstruk
		SN-000008	• •	Gear whe	eel	F	Part			03.10.2006	Konstruk
		SN-000009	• •	Pinion		F	Part			03.10.2006	Konstruk
		SN-000010	• 0	Guide sle	eve	F	Part			03.10.2006	Konstruk 👻
+											
Properties X Graphic	×	Rows: 18 Displa	ay: All states	5							
Description	Value	Documents 2	×								
Number	🔋 PN-01-06-К	4y 🖳 🥃	Standa	rd		• 7 %					
Designation	Construction Documents		-				10 0.00				
Project type	Order	Document num	iber wIndex	• Ir W	orkflow s	Designation	Document type	Creation dat	e Created by	File changed o	n l
Customer name	ISD Software und Systeme GmbH	2000008 ND		•	0	3-D model	HiCAD Part/Varia	nt 02.10.2006	Konstrukteur1	02.10.2006 17:	19:59
Customer number	1000	2DN-000011		•	Ð	Production drawing	HiCAD Drawing	03.10.2006	Konstrukteur1	25.10.2006 09:4	48:15
Responsible	Buchmann	2DN-000005		•	0	Assembly drawing	HiCAD Drawing	02.10.2006	Konstrukteur1	24.10.2006 17:	56:02
		2DN-000023		•	0	Production drawing	HiCAD Drawing	03.10.2006	Konstrukteur1	24.10.2006 17:5	56:36 / U
		2DN-000010		•	0	3-D model	HiCAD Part/Varia	nt 02.10.2006	Konstrukteur1	02.10.2006 17:2	23:18
		2DN-000237		•	0	*	HiCAD Drawing	11.12.2024	Administrator	11.12.2024 10:0	)6:44
		20N-000025			0	Production drawing	HiCAD Drawing	03.10.2006	Konstrukteur1	24.10.2006 17:	36:38
		20N-000013			0	Production drawing	HiCAD Drawing	03.10.2006	Konstrukteur1	24.10.2006 17:	56:21
		ZDN-000014		-	0	3-D model	HICAD Part/Varia	nt 03.10.2006	Konstrukteur1	03.10.2006 13:	32:58
		2DN-000019		-	0	Production drawing	HICAD Drawing	03.10.2006	Konstrukteur1	24.10.2006 17:	30:29
		4		1-1	••	s-u model	Hit AD Part/Varia	nt 118 10 2005	Konstrukteur	18 10 2006 11-	ia:26
[ <u> </u>		Rows: 31 Displa	ay: All state:	5							

#### Recursive visualisation of outdated documents in the document structure

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

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

٢	Document reference up-to-dateness	UI_ReferencelsOutdated
۲	Document reference structure up-to-dateness	UI_ReferenceTreelsOutdated

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

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

Model structure × Mask	$\times  $	Exte	nde	d m	nask	×	Ta	argets × Sources
🔸 📲 🐂 😹 Standard			•					
Document	0	Ł		B	Ind	•	Wo	Designation
✓ DN-013498			Pa.	B		•	0	Draft
<ul> <li>DN-013501</li> </ul>	0	f		B		•	0	Part document
<ul> <li>DN-013500</li> </ul>	0	f		B		•	0	Part document
DN-013499	0	f		B		•	0	Part document
<ul> <li>DN-013503</li> </ul>	0	~		B		•	0	Part document
DN-013502	0	~	2	B		•	0	Part document

You can also switch to a display With text:

Document	0	Document reference struct	• • • • Insli
✓ DN-013498		Not assigned	Change title
✓ DN-013501	0	Outdated	Horizontal alignment
<ul> <li>DN-013500</li> </ul>	0 1	Outdated	✓ With text
DN-013499	0 1	Outdated	Representation type
<ul> <li>DN-013503</li> </ul>	0	Current	Save
DN-013502	0	Current	Reset

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

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

### Configuration Editor: HiCAD-HELiOS interface

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

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

Edit View Extras 155			
) 🖉  📲 🗠 🖓 📲	A 0 0		User 🖉
HiCAD	Description	Value	Comment
Active configuration (Base configuration	Load locked documents in read-only mode?	Ask user Y	Behaviour when loading locked documents
<ul> <li>Image: Image of the second seco</li></ul>	Add semi-finished product article when loading	No ~	If there are parts without article master, semi finished product articles will be added
Modelling	Synchronize drawing with main part	Yes v	
<ul> <li>Image: Steel Engineering</li> <li>Image: Image: Image:</li></ul>	Prevent plotting of modified read-only drawings?		If a drawing has been edited but is read-only plotting of the edited drawing will be preven
Profile Installation	DB project	From document management V	Database project
<ul> <li>Plant Engineering</li> <li>Sheet Metal</li> </ul>	/ Check default DB links	Check with query before correction ~	Check database connection (and correct if required)
Assembling simulation	Use HELiOS/HiCAD Default Solution	$\checkmark$	Combined search via articles and document
<ul> <li>Analysis</li> <li>Interfaces</li> </ul>	Handling of article master	Query ~	Handling of manually assigned article maste during exchanging of semi-finished product
Compatibility     System settings	Update part annotations with HELiOS attributes?		Part annotations with HELiOS attributes sho display the current HELiOS status after loadi model drawing.
	Attribute display when double-clicking on part	Part attributes ~	
Sketch HCM  Itemisation	Article attribute for the weight	GEWICHT	Name of the HELiOS article attribute to be assigned the weight
Processing plane Scales	Attributes for BOM via product structure	Only selected attributes ~	Selection of attributes to be transferred for transfer via product structure
Sketches	Document and article attributes		
<ul> <li>Units</li> <li>Novice configuration</li> </ul>	Taking over of semi-finished product attributes	Collection	Which semi-finished product attributes are taken over to manual article masters ? (Form Semi-finished product attribute;Article attril
Load/Save	Transfer part attributes to HELIOS		Transfer part attributes of changed parts to article master when saving
<ul> <li>Data save</li> <li>Identification</li> </ul>	Document attribute for the scale	MASSTAB1	Name of the HELiOS document attribute to assigned the scale
Referencing Annotations Calculations	Document attribute for plot stamp		Name of the HELiOS document attribute providing the document status for the plot stamp
Graphic	Article attribute for the material	MATERIAL	Name of the HELiOS article attribute to be assigned the material
<ul> <li>Visualisation</li> <li>Feature</li> </ul>	Article attribute for the surface area	FLAECHE	Name of the HELiOS article attribute to be assigned the surface area
III 2-D Lines III Miscellaneous	Article attribute for the volume		Name of the HELiOS article attribute to be assigned the volume
Standard parts and processings	Product structure		
<ul> <li>DDM</li> <li>Drawing Management</li> <li>HiCAD-HELIOS interface</li> <li>Configurations</li> </ul>	Transfer product structure when saving drawing		Transfer the product structure of the main assembly to HELiOS when saving the drawi This option leads to increased waiting time when saving. It is recommended to transfer product structure when saving parts.
	Transfer product structure when saving parts		Transfer the product structure of parts to H when saving.
	AutoSave corrected parts	Ask user	
	Semi-finished product article as sub-item		When transferring the product structure, ad

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

			User	- 🧖
HiCAD	Description	Value	Comment	
<ul> <li>Active configuration (Base configuration</li> <li>Drawing</li> </ul>	Allow loading/saving with interrupted HELiOS connection		If there is no HELiOS connection, a documents is not permitted by def	ccess to HELiOS-manage fault!
Automatic drawing derivation	Transfer product structure attributes to part attributes when updating HELiOS attributes		Transfer product structure attribute updating HELiOS attributes	es to part attributes when
<ul> <li>Steel Engineering</li> <li>Metal Engineering</li> <li>Profile Installation</li> <li>Plant Engineering</li> <li>Sheet Metal</li> <li>Assembling simulation</li> <li>Analysis</li> <li>Interfaces</li> <li>Compatibility</li> <li>Sheet development up to HiCAD</li> <li>Annotations</li> <li>HiCAD-HELIOS interface</li> <li>Itemisation</li> <li>Standard parts and processings u</li> <li>User library up to HiCAD 2022</li> <li>Views</li> <li>System settings</li> <li>Configurations</li> </ul>				

## Major Release

#### Editing rights in the user interface

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

Hicad-Helios	×
- HELiOS links	
O Automatic check, with warning	
$\textcircled{\bullet}$ Automatic check + correction, with query	
<ul> <li>Automatic check + correction, without query</li> </ul>	
<ul> <li>Do not check</li> </ul>	
Check link of 3-D main part/main assy. to drawing	
- Consider in GUI	
With a slow connection to HELiOS, this option can be us	eful to reduce waiting times Apply Cancel

It is not activated by default.

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



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

Merging it with other options changes the name.

## Document number of drawing

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





#### Load/Save behaviour when the HELiOS connection is interrupted

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

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

# **HELiOS Spooler**

Service Pack 2

## UI improvements and target directories

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

4 MIS						
	(iii) (X) (iii) 💌	Standard	•			
tatus	Document number w	Туре	Creation	Scheduled time	Completed	User
Completed	DN-000246	Print job	14.05.2025 11:04:26	14.05.2025 11:04:26	14.05.2025 11:04:29	Administrator
Completed	PDN-000003	Print job	21.05.2025 12:34:00	21.05.2025 12:34:00	21.05.2025 12:34:05	Administrator
Completed	2DN-000005	Print job	21.05.2025 12:34:00	21.05.2025 12:34:00	21.05.2025 12:34:06	Administrator
Completed	PDN-000007	Print job	21.05.2025 12:34:00	21.05.2025 12:34:00	21.05.2025 12:34:08	Administrator
Rows: 4						

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

types	Set target directorie	IS	
r assignments	- Print		
directories	Default directory:		
ocessings			
	Printer	Directory	Postprocessing
	- Conversion		× 🚯
	- Conversion Default directory: D:\Temp	)	-
	- Conversion Default directory: D:\Temp		•
	<ul> <li>Conversion</li> <li>Default directory: D:\Temp</li> <li></li></ul>	Directory	-
	<ul> <li>Conversion</li> <li>Default directory: D:\Temp</li> <li>Temp</li> <li>Temp</li> <li>Temp</li> <li>Temp</li> <li>Temp</li> </ul>	Directory D:\Temp	•
	- Conversion Default directory: DATemp Tormat 3DVS ACIS ACIS	Directory D:\Temp D:\Temp	-
	Conversion Default directory: D∆Temp	Directory D:\Temp D:\Temp D:\Temp D:\Temp	
	<ul> <li>Conversion</li> <li>Default directory: D:\Temp</li> <li>Temp</li> <li>Temp<!--</td--><td>Directory D:\Temp D:\Temp D:\Temp D:\Temp D:\Temp</td><td></td></li></ul>	Directory D:\Temp D:\Temp D:\Temp D:\Temp D:\Temp	

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

# **HELiOS MS Office Interface**

## Service Pack 1

Revised Office interface with an extended range of functions and options

#### **HELiOS-Ribbon**

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



#### New in HELiOS

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

Create	Article number	Document number	WoArti	Docu	iment							
🖑 📄 Staff absence plan		DN-000340										
					Basic information							
					Document number	DN-000340		Sheet:				
					Project number:	Project-indepe	endent 🛄 🖪	Index:		1.2.1		
					Folder number:	Folder-indepe	ndent 🛄 📁					
					Designation:		•	Release:	In P	rogress		
								Document	type: Offic	e Document		-
						Datum:	Name:		_			
					Created:	15.01.2025	Administrator	Scale:				
					Checked:			Format:				5
		_	) b									
Rows: 1 Number of selected	d rows: 1			4 =								_
- General Settings		- < > Graphic								1		
🗹 When selecting documer	nt also automatically select article	A .	В	С	D E	F	G H	1	J	К	L	M
– Input settings	]	2										
Import configuration:	Standard 👻	3										
Project:	Project-independent 🛄 関	4										
Folder:	Folder-independent 🛄 길	6										
Article Workflow:	part (R) 🔹	7										
Document Workflow:	general document (R) 🔹	8										
Select link class:	- -	10										
Recognition of documents:	• OFF	11										
-	O By file name	12										
	O By original path	13										
		14										
		15										
		16										

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

#### **Options**

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

	3/2 HELiOS Options
📰 General 🔍 Find	Settings for the HELiOS-Outlook interface
Input Display Project and Folder structures Product structures Print / Convert Print / Convert Print / Convert Vorkflow Cog Cog Preview Database Document type Corr Outlook	- Settings for the Workspace   Automatically end editing of documents without local changes after closing the application   Remove documents that are currently not being edited from Workspace after closing the application   Remove locally changed files   - Attribute assignments   Transfer to HELiOS:   Attribute import   - Login settings
Manage •	Apply Cancel OK

#### **Attribute assignments**

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

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

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

	<sub> K</sub> Attribute assignment	ts for import to HELiOS	
	- 🕂 🕈 🗙 👘	File: Standard	- 🛃
	•		
ource	Destination	Constraint	Converter
ile extension, with 2 case differenti	ations Document type (DOCUMEN	T_TYPE, Document)	
verter Conditions			Apply Cancel
- <b>Source</b> Case differentiation	Cdit attribut	e assignment	
File property:	File extension		$\times \mathbb{Z}$
Property value	Attribute value		
.MSG	Fixed value: E-Mail (de), E-M	Mail (en), E-mail (fr), E-mail (it)	, E-Mail (pl)
	Fixed value: Sonstige (de), (	Others (en), Autres (fr), Varie (it	), Inne (pl)
- Destination Attribute: Docur	nent type (DOCUMENT_TYPE, [	ocument) Select.	
– Constraint – – – – – – – – – – – – – – – – – – –		•	

E-mail pr	operties can	be mapped with	n article and	document attributes.
-----------	--------------	----------------	---------------	----------------------

	zal		1711 BUELVEND
		💥 Attribute assignments for impo	rt to HELiOS
E-mail proper	ties 🔹 🔮		File: Standard
	•		
Source		Destination	Constraint
File extension,	with 2 case differentiations	Document type (DOCUMENT_TYPE, Document type)	cument)
- <b>Source</b> Property:			
– <b>Source</b> Property: Area:	всс	-	
- <b>Source</b> Property: Area:	BCC Categories	-	
- Source - Property: Area: - Destination	BCC Categories CC	-	
- Source Property: Area: - Destination Attribute:	BCC Categories CC Creation time E-mail address of the sender	Select	
- Source Property: Area: - Destination Attribute:	BCC Categories CC Creation time E-mail address of the sender E-mail ID	Select	
- Source - Property: Area: - Destination Attribute: - Constraint -	BCC Categories CC Creation time E-mail address of the sender E-mail ID Name of sender	Select	
- Source — Property: Area: - Destination Attribute: - Constraint – Name:	BCC Categories CC Creation time E-mail address of the sender E-mail ID Name of sender Received on	Select	
- Source Property: Area: - Destination Attribute: - Constraint - Name:	BCC Categories CC Creation time E-mail address of the sender E-mail ID Name of sender Received on Receiver	Select	
- Source - Property: Area: - Destination Attribute: - Constraint - Name: - Converter -	BCC Categories CC Creation time E-mail address of the sender E-mail ID Name of sender Received on Receiver Subject	Select	

# HeliosCouplings

An adjustment of the HeliosCouplings tool indicates which installed Office versions may not be supported by HELiOS and subtracts them from the selection options.

# Major Release

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

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

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

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

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

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

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

(\*plus Location ID and User ID).

The version-dependent workspace databases are stored at programdata%\ISD Software und Systeme\HELiOS

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HiCAD 30.2

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