

# What's new?

Version 2024 News Overview

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# TOC

Discontinuations	
Basics	
Service Pack 2 2024 (V 2902)	
Start Centre	
Redesign	
Improved workflow	
Memory utilisation message	
Parameter configuration - Changed dialogue texts	21
ICN: Item text without item number	
Progress display when saving with preview for HELiOS and Viewer	
Autopilot - Identification of 2-D points	
Default setting of the context menu	
Customise quick access toolbar - Search field	24
Itemisation	25
Automatic itemisation by part group with prefix	25
Consideration of the processing direction	
Decimal places in the drawing	25
Part variables in the ICN	
Activate SpaceMouse®	
Service Pack 1 2024 (V 2901)	
Licensing	27
Changing the size of the transparent toolbar	
Part attribute masks modernised (SP1)	
Automatic itemisation - numbering according to part groups with prefix	
Frame/scale list	
HiCADGUIReset - List of documents used	
Design Checker with improved usability	
Summarising the functions for 3DFindit	
Drawing derivation - sectional view of the beam cross-section	35
SpaceMouse®	35
Major Pelease 2024 (V/ 2000)	36
Installation Noton	
Opdate derived drawing	
Drawing derivation in external drawings	
Sheet metal parts and multi-part profiles in the ICN	
Display of the number of parts of a multiple selection in the ICN	
Sorting parts in the ICN	
Design Checker - BOM-relevant parts without article master	41
Article master display via double click	
View groups when updating workshop drawings	
Support tool	

Update automatically calculated attributes when loading	
Dimensioning rules	
2 D	47
S-D.	
Cams and cam processing	
Update automatically calculated attributes before saving	
weid seam marking.	
Nove and delete inflexion points.	
Magnetic snap-in of form and positional tolerances	
Magnetic snap-in of annotation tags	
View caption - Information on the original view	
Exact representation of the view selection	
Annotation - Base point symbol	
Referencing.	
Referencing functions in the ribbon Drawing	
Show/hide elements in view extended	59
Lettering: Insertion point	61
Insertion of standard parts from a copied table	61
Service Pack 1 2024 (V/ 2001)	62
Came and cam processings	
Lettering	
Renlace standard parts	
Referencing	
Updating identical parts.	
Referenced assemblies with referenced parts	
Dimensioning and annotations.	
Adopt tolerance	
3-D annotation with HELIOS data Form and positional tolerances (3-D) - Preview window	
Edge state - preview window.	
Clean up intersections - Bore out inner corners	
Views	
Create sectional view - Extensions	
Change section path	
Fixed view point when changing scale	
Align views horizontally/vertically, via points.	
Align and distribute views flush/centred	
Time threshold for AutoQuickView (s)	
Part and dimension orientation	
Transform and clone - Move+rotate via planes	
Major Release 2024 (V 2900)	
Dimensioning and annotations	
3-D annotations - background cut-out	
New symbols in the context menus	

Form and positional tolerances Simplified orientation of the dimension figure 3-D part appotation - Insert base point	
Coating of general parts	87
Feature in part creation	88
Sketches	89
Simplified rotation of 3-D sketch elements	
Marking of sketches with deactivated HCM	
Weld seam and weld symbols	
Extended weld symbols	
Show/hide weld seams by view	
New symbols in the context menus	
Views	
Magnetic snap-in when moving	
New dialogue for sectional views	
Multiple selection of views	
Deleting dimensions in shortened views	
Temporarily deactivate view shortening	
Consideration of intrusions in Hidden Line representations	
Cams and cam processings	
Update automatically calculated attributes when loading	
Extensions in lettering	
Catalogue Editor	
Service Pack 2 2024 (V 2902)	101
Linits and unit categories	101
Profile installation - new system attributes	101
ROMA profiles	102
LIS Standard holtings	102
Flow drilling - NCX output and itemisation	103
Semi-finished products/profiles with imperial dimensions	103
Project and Folder selection	105
Threaded rods - Change of category	105
Further ALLICOBOND® colours	105
CatalogueMaker-Eactory standards	100 107
Service Pack 1 2024 (V 2901)	
Flow drillings	
Major Release 2024 (V 2900)	
Tool numbers in sheet metal processing	
User-defined columns in catalogue tables	
Displayed names of table columns adapted	
Fastenal - Threaded Rods	
Henkel Teroson foils	111
User-defined tables with Steel Engineering plates	
Point clouds	

Service Pack 2 2024 (V 2902)	
Clipping Box Manager	
Message in case of high memory utilisation	
Major Release 2024 (V 2900)	
Clipping Box Manager	
Feature Technology	
Service Pack 2 2024 (V 2902)	
New Feature functions	
Service Pack 1 2024 (V 2901)	
Naming of feature functions	
Major Release 2024 (V 2900)	119
Feature for part and sheet creation	
Create feature variant	
	100
НСМ	
Service Pack 2 2024 (V 2902)	
Groups with isolated points	
Major Release 2024 (V 2900)	
HCM update when variables are changed	
Configuration Management	
Service Pack 2 2024 (V 2902)	
Referencing - Updating identical parts	
Update automatically calculated attributes before saving	
Service Pack 1 2024 (V 2901)	
Time threshold for AutoQuickView.	
SpaceMouse®	
Referencing	
Update identical parts	
Additional column in the Dackaging dialogue	
Development attributes for Steel Engineering plates	
Drewing Management	124
Documents for general documents	
Managing general 3-D parts via part filter	
Plant Engineering	
Pipe length check no longer as macro	
Settings for down-grade symbols	
Interfaces	
Major Belease 2024 (V 2900)	126
Part properties	126
Automatically calculated attributes	
Parameter configuration	
CFGDBTool.exe	

Intersections in the Hidden Line representation	
Part master display by double click	
Magnetic snap-in when moving views	
Negative and positive position in identical part search	
Processing external drawings	
Processing sheet metal developments in production drawings	
Generate pipe spool drawing from sheet view	
Automatic BOMs for itemised source models	
Bill of Materials / Report Manager	
Service Pack 2 2024 (V 2902)	
Text file with separator	
ERPlus interface - Excel export	
Set default configuration	
Creating an RM3 file with the HELiOS API	
Major Release 2024 (V 2900)	
General adjustments	
Empty rows in the structure list	
Round decimal places	
Header and Footer Editor.	
BOMs for HiCAD	
Cancelling the insertion of the BOM	
Adjust text lengths	
I ranster visible structure list	
Variant Editor	
Major Release 2024 (V 2900)	
Files managed by HELiOS	
Category and Unit	
Automation	
Discontinuation	
Service Pack 2 2024 (V 2902)	
API extension in 3-D	
Develop beam	
API for various View functions	
Order of drawing sheets	
Free milling for sheet development	
Major Release 2024 (V 2900)	
Cam joints	
Annotation tags	
Executing UI tasks	
Event after drawing derivation	
Form and positional tolerances	

Accessing fixed view points	145
Interfaces	
Service Pack 2 2024 (V 2902)	
FlexiCAD XML	
IFC	147
IFC default settings for import	147
Select IFC version when exporting	
NCX	
Blind hole processing	
Thread description	
Calculate cutting length of notched beams	
Flow drillings	
2-D DXF/DWG: .dat file transfer confirmation	
KISSsoft 2024	
Punch marks in ToPs-GEO export	
ERPlus interface - Excel export	150
Version number for CATIA interface omitted	
Service Pack 1 2024 (V 2901)	
Update to CADfix 13	
Opening foreign formats without dialogue	
Combining the function for 3DFindit	
3-D Import: Move to origin	
DSTV-NC	
Major Release 2024 (V 2900)	
Access to 3Dfindit	
New import formats: MicroStation and Solid Edge.	
IFC interface: Basic settings	
2D-DXF/DWG: Improved Import and Export options	
	100
Sheet Metal	
Service Pack 2 2024 (V 2902)	
Cross-breaks in developments	
Manual changes to developments	
Semi-finished product parameters	
Avoid double dimensions	
Deleting flanges and bend zones	
Free milling for sheet developments	
Service Pack 1 2024 (V 2901)	
Revised Pipes + Vessels function	
Length change by selecting front edge	
Major Release 2024 (V 2900)	
Feature for Sheet Metal part creation	
Free milling	
Sheet development	

Exclude annotations in sheet developments	172
Display of the catalogue symbols	173
Tool numbers in sheet metal processing	
Mitre with neighbouring sheets	
Apply length value	176
Extension of the coating function	177
Settings for 2-D DXF export	
Improved handling of crosshairs during bending simulations	
Transform + Clone in context menu	
Attach flange function in context menu	
Design variant SZ-20	
User-defined Steel Engineering plates in catalogue	
Steel Engineering	
Service Pack 2024 SP2 (V 2902)	
Civil Engineering - Part type catalogues	
Insert new beam - modified and extended dialogue	
Copying connections on API basis	
Railing Configurator	
Post made of round steels	
Straight cut on handrail	
Post - Sub-structure. Lateral connection with hat steels	
Continuous infills	
Staircase Configurator - Start/end platform as a step	
Redesigned Frame corner dialogue (2203)	
End plate (2102) - Base position	194
Strap joint (2310) - Chamfers of straps	195
Service Pack 2024 SP1 (\/ 2001)	106
Development attributes for Steel Engineering plates	
Development attributes for Steel Engineering plates	
Faster start	
Distance between skirting board and post	
Distance between handrail and cross-member	
Feature for mitre cuts	
Civil Engineering - Part type catalogue, 3-D	
Galvanization holes on beams	
Clearance for stiffeners	
Major Release 2024 (V 2900)	
Insert new beam	
Connections	203
New connection - Gusset plate (2510)	
Frame corners - placement points for stiffeners	
Beam to web, with 2 plates + stiffener - Filler plates	
Cross-bracing (2601) WITH TURNDUCKIE and DIADE SCREWS	
Base plate + Anchor plate (2101)	
Connections- Galvanization holes on beam	

Civil Engineering - Part type catalogue 3-D	
Unbending multi-part standard beams	
Parameter configuration	
Feature when inserting Rectangular plates	
Railing Configurator	
Post connection top, with galvanising hole	
Infill with bottom cross-member - Insert filling rods	
Usage for railing segments	
Extension of the BOM template for Steel Engineering	229
User-defined tables with Steel Engineering plates	
Drawing Management	231
Service Pack 2 2024 (V/ 2002)	201
Automatic itemisation by part groups with prefix	231
Attribute DOKLAPT (Document type) and HicadBIMConfig exe	
Service Pack 1 2024 (V 2901)	
Documents for general files.	
Managing general 3-D parts via part filter	
Simplified handling when referencing in project structures	
Major Release 2024 (V 2900)	
Remove parts from production drawings	
Processing sheet metal developments in production drawings	
Releasing assemblies with already released parts	
Automatic BOMs for itemised source models.	
Updating manually created production drawings	
Metal Engineering	
Major Release 2024 (V 2900)	
Multiple selection of items when importing from LogiKal to HiCAD	
Replacing individual profiles	
Extended transfer of LogiKal attributes	
Facade/insert with sketch: Sketch on upper glass edge	
Civil Engineering - Part type catalogue, 3-D	
Layout Planning	
Service Pack 2 2024 (V 2902)	243
New connection for ALUCOBOND SZ 20	243
Fastening for ALUCOBOND Tray panel SZ 20	244
Service Pack 1 2024 (V 2901)	245
Wall bracket usage for parts without a steel engineering axis	245
Major Release 2024 (V 2900)	210
Change of the structure in the docking window	240 2/10
Annotating installation elements	240
Profile Installation	

	Service Pack 2 2024 (V 2902)	. 251
	New attributes for sandwich panels	. 251
	Profile images in the packaging list	251
	ROMA profiles	. 252
	Service Pack 1 2024 (V 2901)	. 253
	Licensing	. 253
	Additional column in Packaging dialogue	. 254
	Rename openings	255
	Major Release 2024 (V 2900)	.256
	Preset default values for joint width towards sketch line	. 256
	Enhancements for Coating	257
	Negative and positive position and identical part search	. 259
Ρ	ant Engineering	260
	Service Pack 2 2024 (V 2902)	. 260
	SIKLA clamps	260
	Support type and support type symbols for pipe clamps	262
	DBPlantDataImport - Attribute initialisation	264
	Isometry and pipe spool drawing	266
	Recolour pipe part symbols	. 267
	Create pipe spool drawing from pipe spool drawing.	.269
	Service Pack 1 2024 (V 2901)	270
	Length of inserted pipes	270
	Knee - Horizontal bending direction (VEERING_RESTRICTION)	. 271
	Presettings for article attributes in the Configuration Editor	273
	Automatic checking of nominal diameters, pipe lengths and part structure	. 274
	Changes/enhancements for pipe part insertion	276
	Additional part information	. 276
	Undo/Redo	. 278 279
	Revised "Set all" option	281
	Line blanks acc. to DIN2626	283
	GF Piping Systems	. 285
	PROGEF	. 285
		288 
	Polvethylene pines acc. to DIN 8074	293
	Isometry and pipe spool drawing	293
	Down-grade symbol	.293
	Units in the isometry and pipe spool drawing settings	. 295
	Generate pipe book	296
	Pipeline Tools	299
	Determine volume	300
	Guideline Editor	.300
	Major Belease 2024 (V 2900)	301
		.001

Component connections with flange parameters	301
Down-grade Editor - more units	302
Isometry and pipe spool drawing	303
Generate pipe spool drawing from Sheet view	303
Changed default settings during isometry/pipe spool drawing creation	303
Part insertion	305
Insert connection nieces	305
Changed buttons	
Part data synchronisation	307
Indication of missing attributes	307
Derived variants with different attributes	308
Synchronisation with catalogue - Units and categories	308
Create pipeline - Assigning of nominal diameters	308
Files managed by HELiOS	311
Create feature variant - Units	311
PAA Editor - Units and Categories	312
Jacob Push-in pipes - Length and Weight	313
Check for invalid nominal diameter matings	314
New versions of the EN 1092-1 flanges	315
New masks for part search in Plant Engineering	316
Placeholders in imperial units	317
Notes on HELiOS Updates	319
HELIOS Desktop	320
Service Pack 2 2024 (V 2902)	320
Performance	320
Language selection in Login dialogue	321
HELIOS Options: Save settings centrally	322
Accept active project / folder	323
Structure in the project detail window	
Convert file	325
Configuration of file names when printing and converting	327
Generated files names and configuration dialogues for export	328
Export project files: Indices	329
Export folder files: Indices	330
Sheet structure. Model structure and Document reference up to dateness	331
Output product structure to Report Manager: Project context	334
Dreduct structure: Management of editing locks in the Locking Manager	225
Product structure. Management of editing locks in the Locking Manager	225
Complete desimal representation of ELOAT attributes in the product structure	
	330
Service Pack 1 2024 (V 2901)	337
Search in sub-projects/-folders	337
Automatically apply workflow of the original object when creating index or derivation	339
Deleting and renaming attribute assignments	341
Different interface, attribute mapping and import/export configurations for different HELiOS users	341

Major Release 2024 (V 2900)	
Improved user interface	
HELiOS Options: Search	
HELiOS Options: Selectable tabs	
HELiOS Options: Classification	
Improvements and extension of the HELiOS URLs	
Export file	
Export files of projects / Export files of folders	
Send file by e-mail: As Zip archive	
HELiOS Options: Attribute assignment and Export settings	
Local changes during export, printing and conversion	
Result list - Target date display for projects and articles	
User rights: Move folders	
User-defined types: Change of read permission and reference attributes	
Initialised attributes	
Workflow selection for article index	
New HiCAD Viewer format	
Italian and Polish remote maintenance	
Using the new Report Manager with HDE reports	
HELIOS in HiCAD	
Service Pack 2 2024 (V 2902)	
Model/Document structure and Sheet structure	
Document reference up-to-dateness	
Project and Folder selection in the Catalogue Editor	
Service Pack 1 2024 (V 2901)	363
Workflow selection	363
HiCAD 3-D annotation with HELiOS data	363
Major Release 2024 (V 2000)	364
Performance	364
Article master display by double-click	
HELIOS MS Office Interface	
Major Release 2024 (V 2900)	
Send file by e-mail	
HELIOS Vault Server	
Major Release 2024 (V 2900)	
Server Monitor: HELIOS Automation Service	

# **Discontinuations**

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

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

#### Discontinuation of "old" HiCAD itemization

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

#### Discontinuation of "old" OpenGL versions

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

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

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

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

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

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

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

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

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

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

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

#### Discontinuation of 3-D projection grid

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

#### Discontinuation of the Zuken E3 interface

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

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

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

#### Discontinuation of Part insertion v26 (PE)

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

# **Basics**

# Service Pack 2 2024 (V 2902)

# Start Centre

#### Redesign

The HiCAD **Start Centre** has been redesigned, although the functions have remained largely unchanged. Under **New drawing** you will now find the four functions for creating a new drawing. Under **Open drawing** you will find all the functions for opening and importing drawings and designs. The icons for **New drawing** and **Open drawing** can be executed with a double-click. These functions of **New drawing** and **Open drawing** can also be found in the **Drawing > New/Open** tab. Under **Options for new drawing**, you will find the setting options for a new drawing as usual. **Last used documents** has been added as a new function. Here you can see the last 16 open drawings as a preview and open them directly by clicking on them. The size of the preview images cannot be adjusted.

I mento otare center					
New drawing		Open drawin	ng		
Options for new drawing -					-0
Main assembly	1:1		•		
Start mode:	(no	one)			•
View projection:					
View representation:	Æ		9 🗇 🗗 🗗		
Processing planes:	<b>v</b> >	(Y-plane	X7-plane	YZ-plane	
Last used documents					
Last used documents —				ŧ	
Last used documents	3DSKIZZE_SC		VARIABLE FÜLLSTÄ	TEST_EXPLO	
Last used documents	3DSKIZZE_SC		VARIABLE FÜLLSTÄ	TEST_EXPLO	

### Improved workflow

The **Start Centre** has been given its own function in the **New/Open** function group. For this reason, the **Do not show Start Centre** checkbox has been removed from the Start Centre. The Start Centre is now only displayed automatically when HiCAD is started. After that, it is only displayed if you use the **Start Centre** function.

	Drawin	g					_			
Start.	New	Docu	Article	Open	Docu	Article	Save	Save as*	Close	Print
		1	New/Oper	1				Save	/Print	

# Memory utilisation message

While working in HiCAD, you will receive a message if the memory is fully utilised. In this way, you can determine for yourself whether you need more memory to work efficiently.

Warning /ery high memory utilisatio f there is not enough worki processes in HiCAD are exec to that very long waiting tin n extreme cases, the progra	n! ng memory available, cuted much more slowly, nes can occur. amme may crash.	8
Details		
PC:		
Name	Value	
Total of physical memory	31,82 GB	
PC memory in use	96%	
Available PC memory	1,21 GB	
Total of PC paging file	58,39 GB	
Available in PC paging file	12,15 GB	
Current directory	C:\HiCAD2024\exe	
Free memory space C:\	408,61 GB	
	Not responding	
HiCAD:		
Name	Value	
Total of physical memory	31,82 GB	
PC memory in use	96%	
Available PC memory	1,21 GB	
Total of PC paging file	58,39 GB	
Available in PC paging file	12,15 GB	
Current directory	C:\HiCAD2024\exe	
Free memory space C:\	408,61 GB	
	Not responding	
] No more warnings in this	HiCAD session	

In particular, the message is also displayed if the conversion of point clouds leads to high memory utilisation.

If the message should no longer be displayed during the current HiCAD session or the warning mechanism should be completely deactivated, activate the corresponding checkbox. To reactivate the warning mechanism or to display the message again in the current session, use the **Process info** function under **Help Top-ics and Information** 

If the window should not always be displayed in the foreground, deactivate the **Always display dialogue in the foreground** checkbox.

# Parameter configuration - Changed dialogue texts

With the **ParKonfigComp.exe** tool, you can choose whether you want to use the ISD standard configurations or your own configurations when working with HiCAD. In particular, you can specify here in which or for which region - North America or Europe - you are working. The choice of region affects the units of measurement for the drawing, scales and drawing frames.

The texts in the dialogue window have been changed to make it easier to understand the meaning of switching.

Default template		<ul> <li>Regional setting</li> <li>Metrical</li> <li>Imperial</li> </ul>	
Module-		Font	
<ul> <li>Mechanical Engineer</li> <li>Steel/Metal Engineer</li> <li>Plant Engineering</li> </ul>	ing	<ul> <li>HiCAD</li> <li>Arial</li> <li>ISOCPEUR</li> </ul>	
<ul> <li>Attribute configuration a</li> <li>HiCAD / HELiOS Default</li> <li>Novice configuration</li> </ul>	according to DSTV ult Solution	Drawing Management	
User-defined template			

### ICN: Item text without item number

In the ICN settings, you can specify whether you only want to display the item number, the item text or both. Invalid item texts - like invalid item numbers - are crossed out.

- 🗸 Multi-column structure ———
Item number column
Show item number
Show item text
Invalid item numbers
O Mark with asterisk
<ul> <li>Distinguish by colours</li> </ul>
Blau 👻
<b>123</b> <i>123</i> <del>(123)</del>

If only the item texts are displayed, the display can also be sorted alphabetically by clicking on the column header.

### Progress display when saving with preview for HELiOS and Viewer

If the **Complete** option is selected for the **Save as** under **Scope of Preview for HELiOS and Viewer**, all sheets of the drawing can also be displayed in HELiOS. Depending on the drawing, this can make the saving process more time-consuming.

As of SP2, the HiCAD status bar next to the progress bar will explicitly indicate when the preview calculation is running.

- Scope of preview for HELiOS and Viewer
O Active sheet
• Complete

Saving drawing (Calculating preview..)

This also applies to saving in SZX format (viewer) and the Save with preview, All sheets unction.

## Autopilot - Identification of 2-D points

If a 2-D point is captured in a 3-D selection, for example when working with a sketch, and converted into a 3-D point, the Autopilot displays an additional 2-D symbol at the cursor from SP2 onwards.



# Default setting of the context menu

The default settings of the context menu were changed the first time HiCAD was started. The sliders for the maximum size of the icons and the maximum number of icons per row have been set to the maximum. The slider for the minimum size of the icons is determined automatically based on the screen resolution and its scaling.



# Customise quick access toolbar - Search field

HiCAD offers a very extensive range of functions. If you are looking for a specific function when customising the quick access toolbar or creating new ribbon bars, but no longer know exactly on which ribbon the function can be found, the search field can help from SP2 onwards. If you enter a search term here, HiCAD lists all functions whose name contains the specified search term. The search is carried out in all ribbon bars and in the transparent toolbar.

Customize UI			-		×
dimension para v 🕅					
Scientification		QuickAccess Toolbar		v	
Ribbon: 3-D Dimensioning + Text     Group: Tools     Dimensioning + Text     Group: Tools     Dimensioning + Text     Group: Edit     Dimensioning + Text     Group: Edit     Dimensioning settings (2-D)     Change dimension parameters, structure (2-D)     Group: Edit     Change dimension parameters, structure (2-D)     Group: Edit     Change dimension parameters, contour (2-D)     Group: Edit     Change dimension parameters, the reference dim. (2-D/3-D)     Group: Edit     Change dimension parameters, Like reference parameters, Individu     Group: Edit     Change dimension parameters, Like reference parameters, Structur     Change dimension parameters, Like reference parameters, Structur     All shortcuts	•	Pop-up menu: Open drawing Pop-up menu: Save drawing as Pop-up menu: Save drawing as Separator>			
Show QuickAccess Toolbar above Ribbon		0	<	Cance	el

#### Itemisation

#### Automatic itemisation by part group with prefix

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

Until now, automatic itemisation by part group with prefix was not supported by the drawing management. This is now possible as of SP2.

#### Consideration of the processing direction

Previously, the processing direction was only taken into account as a criterion for identical part search for sheet metal plates and sheet steel. As of SP2, this now also applies to all other part types. Please note the following:

- The direction symbols here only ever describe the processing direction for one facet. This means that, unlike with edge plates, the processing direction is not continued across facet boundaries.
- For plane facets, only the direction, but not the position of the symbol plays a role. For non-plane facets, the position and direction of the symbol are compared.
- A distinction is always made between one-sided and two-sided direction symbols.

#### Decimal places in the drawing

The number of decimal places displayed now depends on the settings you make with the **Decimal places** 

function or on the value set for decimal places (::NKS="") in the HDX files of the attribute masks.

The default setting for new drawings is made in the Configuration Editor (ISDconfigeditor.exe) under ... > System settings > Miscellaneous with the parameter Decimal places, Coordinates or Decimal places, Angles.

# Part variables in the ICN

If the part type or the part type and the item number are displayed in the **3D-Part structure** docking window of the ICN, a corresponding display is also shown in the docking window of the **Variables** from HiCAD SP2.



# Activate SpaceMouse®

When editing 3-D models, you can also use a SpaceMouse<sup>®</sup>. As of SP2, this no longer needs to be activated in HiCAD under **Settings**  $\mathbb{Z}^{\ast}$ .

# Service Pack 1 2024 (V 2901)

# Licensing

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

# Changing the size of the transparent toolbar

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



## Part attribute masks modernised (SP1)

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

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

# Automatic itemisation - numbering according to part groups with prefix



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

7		Jutes			(i)
	Assembly		Item prefix:	BG	
2	Number assignment:	Rule-compliant	Start value:	1	
~	Identical part search:	Yes	Increment:	1	
	Sheet Metal		Item prefix:	В	
•	Number assignment:	Rule-compliant	Start value:	100	
	Identical part search:	Yes	Increment:	1	
_	Standard part				
•)	Number assignment:	Rule-compliant	Start value:	10000	
	Identical part search:	Yes	Increment:	1	
_	General part				
*)	Number assignment:	Rule-compliant	Start value:	1000	
	Identical part search:	Yes	Increment:	1	
	Weld seam				
2	Number assignment:	Rule-compliant	Start value:	20000	
	Identical part search:	No	Increment:	1	
~	Beam		Item prefix:	Р	
•	Number assignment:	Rule-compliant	Start value:	100	
	Identical part search:	Yes	Increment:	1	
_	Sheet		Item prefix:	В	
*)	Number assignment:	Rule-compliant	Start value:	100	
	Identical part search:	Yes	Increment:	1	
~	Elongated plate		Item prefix:	B	
*	Number assignment:	Rule-compliant	Start value:	100	
	Identical part search:	Yes	Increment:	1	
	Others		<b>.</b>	50000	
2	Number assignment:	Kule-compliant	Start value:	50000	
	Identical part search:	Yes	Increment:	1	

Item number with prefix is permanently entered in the Item text field. This configuration can be changed by

clicking on . As with "normal" item texts, you can change the order of the components or add further components.

$\circ$			
Definition:	Assembly		
Number assignment:	Rule compliant (automatic / manual) 🔹		
Identical part search:	Yes 🔻	P Item text	×
Item prefix:	A •	- General -	
Start value formula:	Value 🔻	Displayed name: Item number with prefix	
Value:	<b>1</b> ▼	- Item text definition	
Increment:	<b>1</b> ▼	Add component:	
Item text:	Item number with prefix	{Item prefix} {Item number}	
			1
		✓     OK     Cancel	

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



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

#### Example

The drawing shown should be itemised with a prefix.



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



The following figure shows the result of the itemisation.

P 🖌 🖻 🖪 🖪 🖪 📲 🃭 📲 🗫		¢
Designation	Item number	Comment
772634949		
Main assembly		Assembly
Assembly Pipe DIN 2395-A-120x60x4	1 BG1	Assembly
TH Pipe DIN 2395-A-120x60x4	1 <i>P1</i>	Shaped pipes
I Pipe DIN 2395-A-120x60x4	2 <i>P2</i>	Shaped pipes
🗾 Pipe DIN 2395-A-120x60x4	2 <i>P2</i>	Shaped pipes
Assembly Pipe DIN 2395-A-120x60x4	1 BG1	Assembly
The Pipe DIN 2395-A-120x60x4	1 <i>P1</i>	Shaped pipes
🗾 Pipe DIN 2395-A-120x60x4	2 P2	Shaped pipes
1 Pipe DIN 2395-A-120x60x4	2 <i>P2</i>	Shaped pipes
Assembly Pipe DIN 2395-A-120x60x4	2 BG2	Assembly
🗾 Pipe DIN 2395-A-120x60x4	3 <i>P3</i>	Shaped pipes
The Pipe DIN 2395-A-120x60x4	3 <i>P3</i>	Shaped pipes
I Pipe DIN 2395-A-120x60x4	4 <i>P</i> 4	Shaped pipes
I Pipe DIN 2395-A-120x60x4	4 <i>P</i> 4	Shaped pipes
Assembly Environment	3 BG3	Assembly
Þ 🦕 BI 3	1 <i>B1</i>	Foundation plate
Floor	1 A1	
i Wall	2 A2	Sheet Metal

Frame/scale list



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

Settings for drawing sheets			×						
(Offset)	Horizontal	Vertical							
Between view groups	10	10		Cattions for view and the					~
				Setungs for view groups					^
Frame:	Width	Height		(Minimum distances)	Hasiaaatal	Vertical	Bill of Materials (BOM)		
Frame/scale list	287	200		Between Views	1	1	Insert BOM	Quantity list in drawing frame if there is or	nly or 🔻
✓ Insert drawing frame							Name of settings file	HiCAD_Stahlbau	-
Complete title block				Scale All views	1:10		Position in view group	Bottom left	•
Adjust scale to frame size		-		Automatically adjust axonom	etric views		Position in drawing frame	Bottom left	•
Rearrange Views				Frame/scale list	٦ C		[		
Rearrange view groups	ОК	Cancel		Visualisation	<u> </u>		Insert frame	Insert, if there are multiple view groups	
				Standard views	Hidden Line,	dashed 💌	(Continenal viewa)		
				Sectional views	Hidden Line	•	Attached parts for sectional v	iew creation:	
				Axonometric views	Hidden Line,	dashed 🗾	Sheets longitudinal or diag	onal to main part axis	
				Shortened view			Plates longitudinal to main	part axis	
				Shorten axonometric views			Profiles perpendicular to m	ain part axis	
				Rearrange Views			Views	s to be created	
				Rearrange view groups				OK Cancel	

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

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

# HiCADGUIReset - List of documents used

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

P HiCAD-GUIReset	×
Restore delivery state of GUI	) 💾 🖆
Reset button ID's	]
Save or Load without list of last	t used files 🗹
HiCAD 2024 (C:\HiCAD\exe\hicad.exe)	

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

# Design Checker with improved usability

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

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

Available checks		_
	bom 👻 🤇	5
General		
✓ BOM-relevant pa	arts without item number	
Non-BOM-relev	ant parts with HELiOS article master	
BOM-relevant page	arts without HELiOS article master	
Steel Engineering		
Non-BOM-relev	ant main parts	
Non-BOM-relev	ant assemblies with main parts	
Assemblies bene	eath BOM-relevant assemblies	
✓ BOM-relevant as	ssemblies without main parts	
BOM-relevant p	arts (except for standard parts) without article master	

# Summarising the functions for 3DFindit

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

# Drawing derivation - sectional view of the beam cross-section

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



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

## SpaceMouse®

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

# Major Release 2024 (V 2900)

### **Installation Notes**

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

NEW		+	1	$\bigcirc$	$\odot$	
	Installation	Notes				

## Update derived drawing

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

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

Update derived drawings	×
Recreate with settings from cor	figuration
(Sheet selection)	Part selection
	Jonly changed
Update only BOM and title block	
Rearrange Views	
Rearrange view groups	
AutoSync sectional views	
Rearrange dimensions and anno	otations
	OK Cancel

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

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

1.
## Drawing derivation in external drawings

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

With the ISD default setting, this checkbox is inactive.

#### Access to 3Dfindit

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

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

3Dfindit (Basis)

The free version offers you access to the available 3D catalogues that have a download contract with CADENAS (as of 06/2023, there are approx. 900 manufacturers).

3Dfindit

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

The previous functions

- parts4cad (Basis),
- parts4cad and
- bimcatalogs

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

## Sheet metal parts and multi-part profiles in the ICN

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

– 🗹 Multi-column structure –	
Item number column	
Show item text	
Invalid item numbers	
<ul> <li>Mark with asterisk</li> </ul>	
<ul> <li>Distinguish by colours</li> </ul>	
Blue	<b>•</b>
<b>123</b> 123 (123)	
User-defined columns	
Totalquantity	
🗌 Weight	
Designation 2	
HELIOS	
Display attributes	
Extended	
Show feature error.	
Show Assembly (Part) HCM errors	When activating a flange, a bend zone or a profile of a profile group in the graphi
Show Sketch HCM errors	, the Sheet Metal or profile main part is expanded in the ICN.
Show grid lines	
Expand Sheet Metal parts and profile	grou
Clara	

With the ISD default setting, the checkbox is inactive, i.e. the structure of the sheet metal part or profile is not

expanded automatically, but only up to the corresponding main part. This is indicated in the ICN by the symbol.

#### Example:

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



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

	3D-Part structure		φ×
	P / D B B B 🖪 🏗 💱	<b>1</b> 3 <b>1</b> 1	<b>∯× ©</b>
$\square$ Expand Sheet Metal parts and profile grou $\Omega$	Designation	Ite	Comment
	ICN_SHEETS		
	Main assembly		Assembly
	🕨 🦕 Sheet 4 mm	5001	Sheet Metal
	🕥 🆕 <u>Sheet 2 mm</u>	5000	Sheet Metal
	Assembly FI 70x10	1	Assembly
	L		
	Selected element	ts: 0	
	2D-Part structure 3D-Part structure		
	3D-Part structure		ąх
	P / D D D D B P P	<b>1</b> 3 <b>1</b> 1	<b>≅× </b> ©
	Designation	Ite	Comment
	ICN_SHEETS		
	Main assembly		Assembly
	👂 🦕 Sheet 4 mm	5001	Sheet Metal
	🖌 🦕 Sheet 2 mm	5000	Sheet Metal
	🗢 Sheet flange		
	🥔 Sheet flange		
🗹 Expand Sheet Metal parts and profile $\operatorname{grou}_{\mathrm{P}}$	🖢 Bend zone		
	Assembly FI 70x10	1	Assembly
	1		
	Selected elemen	ts: 0	

2D-Part structure 3D-Part structure

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



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

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

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

3D-Part structure		‡×
P / B B B B 🕄 🏗 🤧	₽3 ₽1	<b>⊵× </b> ⊉
Designation	Ite	Comment
PROJECT_2320		
Assembly		Assembly
Assembly		Assembly
🌋 HEA 200		I - Beams
🛑 BI 1		Plates
T HEA 200		I - Beams
		_
<u> </u>		L L
Selected elements	: 3	
2D-Part structure 3D-Part structure		

#### Sorting parts in the ICN

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

#### Design Checker - BOM-relevant parts without article master

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

Besign Checker	×	S Design Checker
- Available checks		- Available checks
General     Empty / hidden views     Dummy parts     Mirrored parts     BOM-relevant parts with 0ut item number     Non-BOM-relevant parts with HELiOS article master     V BOM-relevant parts without HELiOS article master		Steel Engineering     Invalid item numbers     Mountability check for boltings     Non-BOM-relevant main parts     Non-BOM-relevant assemblies with main parts     Assemblies beneath BOM-relevant assemblies     St parts below SE parts
Parts without article number Reduced loaded parts	•	BOM-relevant assemblies without main parts     BOM-relevant parts (except for standard parts) without article master     Maximum length of beams     Maximum weight of beams     Maximum dimensions and weight of assemblies

### Article master display via double click

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

		AA 🛛 🞯 💂	User	-
▲ System settings	*	Description	Value	Comment
Assembly HCM		DB project	From document management	Database project
<ul> <li>Sketch HCM</li> <li>Itemisation</li> </ul>		🥒 Check default DB links	Check, with query before correction	Check database connection (and correct if required)
Processing plane		Use HELiOS/HiCAD Default Solution		
<ul> <li>Scales</li> <li>Sketches</li> <li>Units</li> </ul>		Handling of article master	Query	Handling of manually assigned article master during exchanging of sen finished products
Start configuration Directories Load/Save Data save Data save Data save		Taking over of semi-finished product attributes	Collection	Which semi-finished product attributes are to b taken over to manual arti- masters? (Format : Semi- finished product attribute;Article attribute)
<ul> <li>Referencing</li> <li>Annotations</li> <li>Calculations</li> </ul>		Attributes for BOM via product structure	Only selected attributes	Selection of attributes to b transferred for BOM transf via product structure
Graphic	ш	Transfer part attributes to HELiOS		Transfer part attributes of changed parts to article master when saving
iii Feature iii 2-D Lines iii Miscellaneous		Transfer product structure to HELiOS		Transfer product structure changed parts when savin 1 level
Standard Parts		When double-clicking on a part, display article master instead of part attribute mask		

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

# View groups when updating workshop drawings

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

- Rearrange Views and
- Rearrange view groups.

Update derived drawings	×
Recreate with settings from configuration	
Sheet selection All Only changed	•
Update only BOM and title block	
Rearrange view groups	
AutoSync sectional views	
Rearrange dimensions and annotations	ancel

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

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

1n Item. v	Derive Frame BOM Objec	t C					
Itemisa	Drawing	te					
	Drawing derivation						
	Update						
	Change settings						
	Active drawing sheet						
	Active view group						
	Active view						
	Mounting drawing						
	Mounting drawing						
	Update mounting drawing						
	Settings						
	Frame/scale list						
	Templates	•					
	Dimensioning rules						
	Compatibility						
	Up to HiCAD 2021						

# Support tool

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

#### Update automatically calculated attributes when loading

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

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

0 11 12 13 11	A 0		User		
HiCAD	Description	Value	Comment		
Active configuration (Base co Configuration (Base co Configuration (Base co Configuration (Base configuration) Configuration (Base configuration) Configuration Configura	Update automatically calculated attributes when loading	No ~	Update automatically calculated attributes whose calculation is set to 'Always', Attributes are calculated at the latest when itemising or manu- updating.		
Modelling	Calculate weight	Manual / When itemising Y	Assigns value to attribute §01		
Grid	Approximation accuracy for weight calculation	Do not auto-calculate	0 (imprecise but fast) - 100 (precise but slower)		
Miscellaneous	Calculate surface area	Manual / When itemising	Assigns value to attribute §10		
Part creation	/ Calculate coated surface	Always	Assigns value to attribute §SC		
Part properties	Transfer coated surface to	Coated surface ~			
Weld seams	Calculate volume	Manual / When itemising ~	Assigns value to attribute §20		
Steel Engineering	Calculate quantity in assembly	Do not auto-calculate ~	Assigns value to attribute %13		
Metal Engineering     Brofile Installation	Calculate total quantity		Validly itemized parts are counted. Switching of calculation deletes the total number.		
Plant Engineering     Sheet Metal	Material	<no catalogue="" entry="" td="" 🕅<="" 🛛=""><td>Material from catalogue</td></no>	Material from catalogue		
Assembling simulation	Dimension calculation				
<ul> <li>Analysis</li> <li>Interfaces</li> <li>PDM</li> <li>Compatibility</li> <li>System settings</li> <li>Configurations</li> </ul>	Calculate dimensions	Do not auto-calculate v	Determine the dimension for parts and assembl from the geometry. Assigns the attribute Lengtl (503), Width (502) and Height (504). For certain p types, dimension attributes are set even if the calculation from geometry is switched off here.		
	Sort dimensions		When calculating dimensions, write the dimens in descending order of size on the attributes Ler (\$03), Width (\$02) and Height (\$04).		
	Calculate only length for beams		Restriction only applies if dimension calculation activated. If this setting has been chosen, the he and width are not used for beams and any other		

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

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



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

The ISD default setting is No.

## **Dimensioning rules**

#### Position of sub-parts in sectional view

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

- 8: ATTACHING\_PARTS,
- 10: ATTACHING\_SHEETS and
- 57: ATTACHING\_SHEETS\_SEPARATELY

have been updated accordingly.



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

		Usage:ASSEMB	ABLY_COLUMN 🥃 🔜	
<ul> <li>57: Attached plates, one</li> </ul>	chain dimension for each plate			
ID:	57: Attached plates, one chain dimension for each plate	~ All	General	×
Type of dimension:	Chain dimension			
Position of dimension chain:	Тор		Ceneral	_
Dimension chain reference:	Outer sheet/plate edges		Dimensioning Settings	4
Direction of dimension chain:	Parallel to sheet/plate X-axis		- Dimensioning Settings	-
<ul> <li>57: Attached plates, one</li> </ul>	chain dimension for each plate		Chains of dimensions         Deam axis in web, regarding:         Upper + Lower edge           Bores/Boltings         beam axis in flange, regarding:         Upper + Lower edge	•
ID:	57: Attached plates, one chain dimension for each plate	~ All	Annotations Perpendicular to beam ax	is 🔻
Type of dimension:	Chain dimension		Dimension type Do not annotate	-
Position of dimension chain:	Right		General beam axis with sub-parts	_
Dimension chain reference:	Outer sheet/plate edges		lar to beam axis with sub-parts	
Direction of dimension chain:	Parallel to sheet/plate X-axis		OK Cancel	
			Separate dimension chains according to part type for fitting situation of sub-parts	
			Ignore hidden parts and processings	
			Chains of dimensions for sub-parts: Start- and End point	-
			Suppress 90° angles of bend zones in sectional views of folded sheets	
			ОК Са	ncel

#### Usage for railing segments

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

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

# 3-D

# Service Pack 2 2024 (V 2902)

# Cams and cam processing

The previous functions Cams and Cam processing have been into one function Cams

Cams	×	Cams	×
Cams Cam process	ing	Cams Cam processing	
- Edges		- Part to be processed	& -
		Select part	
		- Part with cams	
Continue tangentially	Select edge	Select part	
- Cam shape		- Properties	
		Clearance in longitudinal direction:	0.1 🔹
		Clearance in transverse direction:	0.1 💌
- Parameters		✓ Perpendicular to surface	
Length:	6 🔹	- Corner processing	
Height:	13 🔹	Form:	Drilled
Diameter:	10 👻	Diameter:	1 •
Cam end:	Without processings 💌	Depth:	1 🔻
Length:	1	Move mode:	No move 🔻
Corner processing:	Drilled		No move
Diameter:	1 •		Symmetrical
Depth:	1		Tangential longitudinal direction
Move mode:	No move 🔻		Tangential transverse direction
- Distribution	No move		
Pattern:	Symmetrical		
Sele	Tangential longitudinal direction	n	
Number:	Tangential transverse direction		
Start distance:			
End distance:	0 -		
		☆	
OI	K Cancel Apply	ОК	Cancel Apply

#### Please note:

Cams and cam processings can be created in one dialogue. However, subsequent processing is only possible separately via the respective feature.

- The settings can be saved as **Favourites** for later reuse
- The order of the cam shapes has been changed.
- The **Round** cam shape has been replaced by the **Puzzle piece** cam shape.

- Cam shape		
- Paramete Length:	6 •	
Height:	13 🔹	
Diameter:	10 🔹	
Cam end:	Filleted 🔹	
Radius:	1 •	
Corner processing:	Drilled	
Diameter:	2 🔹	

- If Drilled is selected as the corner processing when creating the cam, a move mode can be selected for the cam. This is independent of the selected cam shape. The following options are available:
  - No move (1)

The bore is created without moving. The centre of the bore lies at the intersection of the corresponding edges.

• Symmetrical (2)

The bore is moved so that it lies symmetrically between the selected edge and the left/right side of the cam.

- Tangential longitudinal direction (3)
   The bore is moved in the longitudinal direction, i.e. in the direction of the selected edge, so that it is tangential to the left/right side of the cam.
- Tangential transverse direction (4)

The bore is moved in the transverse direction, i.e. in the direction of the cam sides, so that it is tangential to the selected edge.



• The move mode can also be selected for the **Drilled** corner processing when processing cams. The previous settings **Move in longitudinal direction** and **Move in transverse direction**.

The move modes can be used, for example, to ensure that bores are not visible after assembly.



## Update automatically calculated attributes before saving

For the calculation of attributes that are set to **Manual / When itemising** in Configuration Editor, you can specify whether or not these calculations should be performed automatically before referenced parts and assemblies are saved. The setting is made under **Modelling > Part properties** with the parameter **Update automatically calculated attributes before saving**. The **Weight** (§01), the **Surface** (§10 and §SC) and the **Volume** (§20) are updated. The attributes **Total quantity** (%06) and **Qty. in assembly** (%13) are not updated.

ISD Configuration Editor - HiCAD 29.2.0.403 [C:\Pro	gramData\ISD Software und Systeme\HiCAD 2024\HiCAD	.cfgdb]	– 🗆 X
File Edit View Extras ISD			
/ 🕥 😂 🕴 🔢 🕮	A 🛛 💿 🖕		User 📃 🦉
⊿ 💾 HiCAD 🔄	Description	Value	Comment
Active configuration (Base configuration)     Drawing     Automatic drawing derivation	Update automatically calculated attributes when loading	No ~	Update automatically calculated attributes whose calculation is set to 'Always'. Attributes are calculated at the latest when itemising or manually updating.
Modelling     Surface/Edge functions     Grid	Update automatically calculated attributes before saving	No Y	Update automatically calculated attributes whose calculation is set to 'Manual/When itemising' before saving referenced parts and assemblies. The attributes Qty. (%01) and Qty. in assembly (%13) will not be updated.
III Miscellaneous	Calculate weight	Manual / When itemising ~	Assigns value to attribute \$01
Part creation	id not be updated. Excellaneous Calculate weight Calculate weight Approximation accuracy for weight calculation ft creation Calculate weight Calculate Cal		
Part properties     E	Calculate surface area	Manual / When itemising ~	Assigns value to attribute §10
Change of part structure      Weld seams	Calculate coated surface	Always ~	Assigns value to attribute \$SC
Steel Engineering	Transfer coated surface to	Coated surface ~	
Metal Engineering	Calculate volume	Manual / When itemising v	Assigns value to attribute §20
Profile Installation	Calculate quantity in assembly	Do not auto-calculate v	Assigns value to attribute %13
High Engineering     High Engineering     High Engineering	Calculate total quantity	V	Validly itemized parts are counted. Switching off the calculation deletes the total number.
Assembling simulation     Analysis	Material	<no catalogue="" entry=""> 🗙 🛐</no>	Material from catalogue

# Weld seam marking

With the new buttons in the **Weld seams** function under **Extended**, the weld seam can be displayed in accordance with the standard. In accordance with the DIN EN ISO 2553 standard, there are two display options: System A and System B. Previously, HiCAD only supported system A. As of SP2, weld seam marking according to system B is now also possible. In addition, a function for mirroring the weld seam using the annotation tag has been added.

Weld seams	×
<u></u>	a • • •
0	
1	
☆	
Parts	
	Select parts
Lines	
	SIDE 1
	(+/ A/
	<u></u>
	SIDE 2
Extended	
	Representation
	Representation     Standard      Reduced (complete)
	Representation     Standard  Reduced (complete)     System A  System B
Line spacing:	Representation  Standard Reduced (complete)  System A System B
Line spacing: Line length:	Representation  Standard Reduced (complete)  System A System B
Line spacing: Line length: <b>Toleran</b> e	Representation  Standard C Reduced (complete)  System A System B  To refor powder marking line search
Line spacing: Line length: <b>Toleranc</b> Distance tolerance:	Representation  Standard C Reduced (complete)  System A System B  To refor powder marking line search
Line spacing: Line length: <b>Toleranc</b> Distance tolerance: Min. length:	Representation  Standard Reduced (complete)  System A System B  To refor powder marking line search
Line spacing: Line length: <b>Toleranc</b> Distance tolerance: Min. length:	Representation  Standard Reduced (complete)  System A System B  To refor powder marking line search  To refor powder marking line search  To refere the search  Standard Standar

# Move and delete inflexion points

The **Move inflexion point** function can be used to move any inflexion point from form and positional tolerances, chamfer dimensions, weld seam symbols, edge markings and annotation tags. If there are multiple inflexion points, any inflexion point except the last one can be deleted.



### Magnetic snap-in of form and positional tolerances

New form and positional tolerances can be aligned horizontally or vertically to the existing form and positional tolerances in the same view.

If a form and positional tolerance moves at the same height as an existing one during dragging, this is indicated by showing a horizontal auxiliary line in the graphic. When the form and positional tolerance is dropped, it is aligned with this line. This also applies to vertical alignment. After the form and positional tolerance has been itemised, the displayed auxiliary line disappears. The reference point of the tag is used for alignment. If there are no other tags between two form and positional tolerances, parallelism is offered. Magnetic snap-in can be switched off by pressing the ALT key.



### Magnetic snap-in of annotation tags

If there are several annotations, the annotation tag to be moved can now be aligned horizontally or vertically with other annotations.

If an annotation tag moves at the same height as an existing tag during dragging, this is indicated in the graphic by showing a horizontal line. If you place the annotation down, it is aligned with this line. The same applies to vertical alignment.



#### Please note:

This functionality is only used to simplify the alignment of the annotation. The alignment is <u>not</u> associative. This means that the annotation tag is not updated if the tag to which it is aligned changes.

# View caption - information on the original view

As of SP2, the name, sheet name and sheet number of the original view can also be used in view captions for derived views, e.g. sectional or detail views.

			Fina	-	
	🔺 🙀 Attribute	s			
	🕁 🗛 De	viating view scale			
	🔂 🛪 Vie	w scale			
	🔂 🛪 Vie	w name			
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	A An	qle	_		
	W A Na	me of the original v	iew		
	TT A She	eet name of original	view		
	公A She	eet number of origin	nal view		
	Drawing	attribute			
	Document	t master of model	drawing		
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	🖻 📂 Folder				
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Number/Name	Scale	S 1 View Capacity	• 17 5.0 • 12 1.0 • 12 2.0	• ( <b>B</b> ) <u>7</u> <u>U</u> ) <del>9</del>	
View 1	1:1		Attribute		
Sheet 1		{Designation} / {N	ame of the original v	iew} / {Sheet num	per of original view}
Dop view	□ 1:1 □ □ 1:1	1			
Axonometry	1:1				
A	-A / Top view / 1				
		B			
		Н			
<b>V</b>					
	Ten view			Axonometry	
	Top view				
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Two functions have been added to the context menu for general 3-D parts (3-D) under **Standard Processings**. The functions are **Lettering** and **Cams**.

	Subtraction	
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С Р П	Standard Processings	
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ã Q	Part tools	
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enu	Lock against processing	•
xtm	Part/Assembly structure	•
onte	HELIOS	•
Ŭ	Properties	•

Exact representation of the view selection

If a view with quick representation is active or the current view selection contains a view with quick representation, the **Exact representation** function is available in the transparent toolbar.



Previously, this function could only be used to restore the exact representation of the active view. As of SP2, this applies to all views of the active view selection.

### Annotation - Base point symbol

Different base point symbols can be selected for 3-D annotations. From SP2, the **Circle** and **Circle (filled)** symbols are always displayed in the screen plane. The symbols therefore remain circular. Up to now, the symbols were spatially consumed, turning them into an ellipse.



# Referencing

Updating identical parts

With SP2, the parameters

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

are available again in the Configuration Editor under **System settings > Referencing**.

4	System settings	^	Description	Value	Comment	
	Assembly HCM		referenced parts	$\checkmark$		*
1	Itemisation		Show selection dialogue when deleting referenced parts			
1	Processing plane Scales Sketches		Automatic name selection for referenced parts without database entry			
	SpaceMouse		Compared part attributes for identical part search when referencing	Collection	1 entry per row, e.g. \$BB=Article designation \$BK= Article master ID	E
	Start configuration		Updating			
	Directories     Load/Save     Data save		Update sectional views, detail views and cut-outs when loading referenced parts			
]	🔳 Identification	=	Update colour	✓	Update colour for referenced parts	
	I Referencing		Update layer	$\checkmark$	Update layer for referenced parts	
1	Annotations		Update line type	$\checkmark$	Update line type in referenced parts	
1	Calculations		Synchronize item numbers/part attributes when updating file	For main parts and sub-parts v		
1	Visualisation		Automatically update referenced parts after each change	Yes ~	Updating of identical parts	
	<ul> <li>Feature</li> <li>2-D Lines</li> </ul>	-	Update identical parts of referenced parts before saving		Only important if referenced parts are not to be automatically updated	*

If there are several identical parts of a processed, referenced part in the drawing, the changes are also automatically made to the identical parts if the ISD is set by default. This automatism can be changed with the parameters mentioned above.

#### Referencing functions in the ribbon Drawing

The functions



Update all parts, from file and



can now be accessed directly from the **Drawing** ribbon via the menu of the  $\mathbb{P}$  **Update** function. Previously, these functions were only available in the context menu of referenced parts.



Another new feature is that a message is displayed when the functions



Update referenced identical parts and

#### Referenced part

are called if the active part is not referenced.



# Show/hide elements in view extended

The dialogue of the **Show/hide elements in view Solution** function has been improved and extended:

- support for multiple selection,
- saving the settings as favourites,
- transfer of settings from a reference view and
- simplified operation.

Show/hide elements in Views — Views — View	n view	,
View 1		•
- Representation		
Centre lines	¥90	
Centre crosses	¥90	
Steel Eng. Axes	¥90	
Steel Eng. Tracing lines	¥90	
Steel Eng. Bar elements	¥90	
Hatchings	¥90	
Free edges	¥90	
Tangential edges	<b>N</b>	
Polyhedron shell edges	<b>N</b>	
Weld seams	<b>N</b>	
Assembly points	<b>R</b>	
Isolated points	8	
Mesh diagonals	<b>P</b>	
Mesh in U-direction	<b>P</b>	
Mesh in V-direction	<b>F</b>	
Processing planes	<b>N</b>	
Lettering	<b>P</b>	
Cut edges	<b>I</b>	
∠ ☆		

The selected views are listed in the upper part of the dialogue window. After calling up the function, this is initially the active view or the current view list.

To select further views, simply click on the corresponding view frame in the drawing. If the frame of an already selected view is clicked again, the view is removed from the list. Alternatively, views can also be removed from the list by right-clicking on the corresponding entry in the list and then selecting the **Remove element(s) from list** function.

For the various elements, you can then select whether they are to be



hidden or

D

covered in HiddenLine mode.

The selected setting is always indicated by an orange frame.

☆	Favourites The current settings can be saved as favourites and used again at any time. They are saved in the ProgramData\ISD Software und Systeme\HiCAD 2024\Fa- vourites\VisGUI\ShowAndHideElements folder.
1	Select view from which properties are to be transferred
	This function can be used to adopt the settings of another view. To do this, select the corresponding view in the drawing.

The function is also available in the context menu of views and view lists.

### Lettering: Insertion point

In the **Lettering** function, you can select the insertion point from nine different positions. By activating the desired radio button, you can insert the lettering at this position.

Lettering	– 🗆 X
Westlich ▼         Arial         ▼         1         3.0         ¶         1.0         ▼         1Ξ         0.0         ▼	<u> </u>
(Item number (Part attribute)) - ISD	Attribute
Item number {Item number (Part attribute)}	🔽 Incl. rear side
	Reference/Insertion point
Favourites Select file>	
	OK Cancel

#### Insertion of standard parts from a copied table

KRA files created when copying tables are <u>not</u> saved in the HiCAD drawing. This means that if you pass the table copy on to a third party, you must also take the corresponding KRA file into account. If a drawing containing standard parts of this table is loaded in HiCAD and the KRA file does not exist, these standard parts cannot be edited with the functions for standard parts. For example, it is not possible to change the display. In this case, an additional message is displayed from SP2 onwards.

	×
The KRA file for the standard part could not be loaded. Note: KRA files created when copying tables are not saved in the drawing.	
ОК	

# Service Pack 1 2024 (V 2901)

Cams and cam processings

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

Cams		×
Continue tangentially	Select edge	•/
- Cam shape		

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

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

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



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



#### Lettering

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



### Replace standard parts

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



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

The following options are rejected with an error message:

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

# Please note:

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

### Referencing

#### Updating identical parts

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

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

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

#### Referenced assemblies with referenced parts

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

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

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

)orignation	Bart name			
SN-00001 (Slip-op.q)	91CAD576 *			
Join concor (sub origin	JICADDIO	nounco		
		atabace attributes	Product structure to HELIOS	

#### Only structurally changed assemblies

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

#### Also assemblies with new indices included

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

#### All assemblies with changed sub-parts

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

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

#### Example

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

J ISD Configuration Editor - HICAD 2	9.2.0.277 [0	C:\ProgramData\ISD Software und Systeme\HiCAD 2024\HiCAD.cfgdb]		- U >
e Edit View Extras ISD				
S 🕄 🕴 🔢 🖓	User 🦿 🦉			
System settings	*	Description	Value	Comment
Assembly HCM		Lock referenced parts for other users during processing?	Yes	
Sketch HCM		Lock non-updated, referenced parts against processing		
Itemisation Processing plane		Lock referenced parts if model drawing is read-only	No	Lock referenced parts against processing if model drawing is read-only?
III Scales		HELIOS		
Sketches		Update variants	Ask user 🗸	Update variants in case of new database index.
Units	=	Entered document master not in DB: -> Referenced part locked?		Lock ref. part against editing if entered document master is not in database?
Start configuration		DB document attribute to be written to part attribute "Designation"	NONE	
Directories		Lock parts against processing if HELiOS article master is locked	$\checkmark$	
Data save		Lock parts for other users via HELiOS article master	Yes	•
Identification		Update article master index	All parts with dialogue	
m Referencing		Allow part exchange in drawings	Not for released or read-only	
Annotations	+	Saving changed assemblies	All assemblies with changed sub-parts	Should assemblies whose referenced sub-parts have been updated also be saved?

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



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



Parts whose geometry has changed - in the example, the right-hand prism - are marked with the  $\checkmark$  symbol in the ICN. Due to the default setting in configuration management, all assemblies with changed sub-parts are

additionally marked with the **use** symbol. If the drawing is now saved, all marked assemblies and the prism with the changed geometry are listed in the **Save referenced parts** dialogue window.

me following referenced parts have been modified. Select the parts that are to be saved!						
		)(u d e o				
Designation	Part name	JUpdating	J[1_Sachnummer			
[ISN-033777 {Part document} {} *	65BF5DD7	Changed sub-parts	SN-033777			
ISN-033783 {Part document} {} *	50EF5C58	Changed sub-parts	SIN-033783			
SN-033778 /Part document\ 0.*	350CB6EE *	Modified	SN-033778			
Terr account of the randomical of A	0000000	Houmed	000770			
Also show unmodified parts	🔽 Display databas	e attributes 📃 Pr	roduct structure to HELiOS			

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

esignation	Part name	Updating	)(T_Sachnummer	
ZSN-033780 {Part document} {} * ZSN-033778 {Part document} {} *	A5756F88 350CB6EF *	Revision index, sub-parts Modified	SN-033780 SN-033778	

esignation	Part name	Updating	T Sachnummer	
SN-033778 {Part document} {} *	350CB6EF *	Modified	SN-033778	

## Dimensioning and annotations

Adopt tolerance



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

### Please note:

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

#### 3-D annotation with HELiOS data

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

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

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

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

#### Edge state - preview window

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

# Clean up intersections - Bore out inner corners

The Clean up intersections function has been extended.



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


#### Views

Create sectional view - Extensions

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

AA	On both ends Annotation on both ends of section path
A**B	<b>Consecutive letters</b> The ends and bends of the section path are annotated with consecutive letters (e.g. A-F).
A1 A2	Consecutive numbers The ends and bends of the section path are annotated with consecutive numbers (e.g. A1-A6).
A	On one end Annotation (and viewing direction arrow) only on one end of section path
↓↓	<b>No annotation</b> The section path is not labelled, but may have viewing direction arrows.

#### **Change section path**



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

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

Process sketch	×
Apply sketch	Cancel

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

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



**Change section limit** 



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

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

3-D V	ew							
New								
0 10		*		讔	30 30	00000		
Edit		_	_					
3	1 🙀		-	3	<b>*</b>	3	3	3
Other	5		Cha	ange	sect	ion I	imit	(3-D)
Transf	form			-				
1		<b>1</b>	2	-	1	<b>1</b>	٢	84 3
Other								

Fixed view point when changing scale

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

### Align views horizontally/vertically, via points

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



Align views horizontally, via points



Align views vertically, via points

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

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

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

#### Align and distribute views flush/centred

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



Align		Distribute	
	Align views left	← →	Distribute views horizontally
<b>7</b>	Centre views horizontally	<b>↑</b>	Distribute views vertically
<b>→</b>	Align views right		
<b>↑↑</b>	Align views at top		
<u>↑</u>	Centre views vertically		
	Align views at bottom		

#### Align

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



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

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



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

#### Distribute

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

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



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

### Time threshold for AutoQuickView (s)

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

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

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

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

陆 ISD Configuration Editor - HiCAD 29.2.0.277 [C:\ProgramData\ISD Software und Systeme\HiCAD 2024\HiCAD.cfgdb] - D X					
File Edit View Extras ISD					
/ 🕤 🖸 👫 🗠 🕄 👯 🖉		User 🖉 🦉			
A 📰 System settings 🔹 🖌 Description	Value	Comment			
Assembly HCM     Horizontal distance between views during view     arrangement	20 mm				
Itemisation     Processing plane     Vertical distance between views during view     arrangement	20 mm				
Cross out old cut-outs, sectional views and detail views in graphic					
Scale for new detail views	Increment scale Y				
Units Include new parts in list views	No ~	Hide created new parts in views with hidden parts? in the active view, new parts will always be shown.			
Include new parts in list views of mounting           Directories	Yes v	Hide created new parts in views with hidden parts? in the active view, new parts will always be shown.			
Load/Save Thread representation in shaded views	With thread texture ~				
Data save     Threshold value for simplified OpenGL     representation (Bounding box)	10	Parts appearing smaller than the specified number of pixels will be displayed in simplified mode. Concerns the representation types Shaded and Quick			
Annotations     Galculations     Calculations     E	1	Parts that would appear smaller than the specified number of pixels will not be displayed. Concerns the representation types Shaded and Quick			
Graphic Automatic locking of new, orthogonal views in sheet areas	Do not lock ~				
2-D     Automatic locking of new sectional/detail views     in sheet areas	Same as original view ~				
Workstation background     Automatic locking of created new development     views	Do not lock ~				
Wiews     Automatic locking of new views of derived     drawing	Do not lock v				
Frame of active view     AutoQuickView in new model drawings	Disable AutoQuickView ~	Applies to created new model drawings and to model drawings that were saved with HiCAD 2019 or older versions.			
☐ Hidden Line ► ☐ Insertion view ► ☐ Insertion view Time threshold for AutoQuickView (s)	0	Views whose calculation time is less than the threshold will not be set to QuickView.			
📰 View group 🔹 🔹 QuickView when saving	Save views only in exact r ~				
< ₩ →		#			

### Part and dimension orientation

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

Part orientation	×
Only allowed in itemized source mod	lels
ОК	

### Transform and clone - Move+rotate via planes

#### The previous function



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

by the function

# Move+Rotate part, via 2 planes.

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

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

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



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

Similarly, the previous function

Clone+Move part has been replaced by the function



# Major Release 2024 (V 2900)

### Dimensioning and annotations

### 3-D annotations - background cut-out

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



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



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

### New symbols in the context menus

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



#### Form and positional tolerances

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

#### New inflexion point

When setting form and positional tolerances, it was previously only possible to insert new inflexion points of the leader line by calling up the context menu (right mouse button) and then selecting the **New inflexion point** 

function. As of HiCAD 2024, this function can now also be called simply by pressing the CTRL key.

#### Other new features

The list of available symbols has been extended.



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



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



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



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



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



#### Simplified orientation of the dimension figure

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

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

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

Distance to previous element	0		
	General	Running	Height above
Parallel to dimension line	50% 💌	0	1.5
Perpendicular to dimension line	1.5	1.5	2.5
Position relative to dimension line	Above	-	
	Above		
	Below	*	

#### 3-D part annotation - Insert base point

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

## Coating of general parts

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

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

### Feature in part creation

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

This affects the following functions:



### Sketches

#### Simplified rotation of 3-D sketch elements

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



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

This affects the following functions:



Marking of sketches with deactivated HCM

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



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

Weld seam and weld symbols

Extended weld symbols

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

Weld pool fuse (not specified)



Fusible inlay



#### Reference line boundary arrow for 3-D weld seam symbols

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

BD Configuration Editor - HiCAD 29.0.0.122 [C:\Prog	gramData\ISD Software und Systeme\HiCAD 2024\HiCAD	).cfgdb]	– 🗆 X
File Edit View Extras ISD			
∕ 🕤 🛱 🕴 😫 🖽	AA   🞯 📙	User	- 🥂
⊿ 🚰 HiCAD 🔶	Description	Value	Comment
Active configuration (Base configuration)	Symbol colour	1: Dark Green 🔻	Default colour of symbol
Drawing	🖉 Symbol height	2	Default height of symbol
<ul> <li>Annotations</li> <li>Form/Positional tolerance</li> </ul>	Line colour	1: Dark Green 🔻	Default colour of line
▷ 📰 Text	Line type	1: •	Default line type
🛄 Dimensioning, 2-D 📃	Line type, dashed	3: +	Default line type for dashed lines
<ul> <li>Dimensioning, 3-D</li> <li>Fits table</li> </ul>	Length factor, dashed line	0.3	Length factor, dashed line (reference: symbol height)
<ul> <li>Profile Installation dimensions</li> <li>Edge state</li> </ul>	Height factor, site weld seam symbol	1.2	Height factor, site weld seam symbol (reference: symbol height)
<ul> <li>Bystem triangle</li> <li>Plot stamp</li> </ul>	Length factor, site weld seam symbol	1	Length factor, site weld seam symbol (reference: symbol height)
<ul> <li>▶ ₩ Weld symbols</li> <li>▶ ∰ Grid annotation</li> </ul>	Factor, radius of symbol for circumferential seam	0.25	Factor, radius of symbol for circumferential seam (reference: symbol height)
Automatic annotation	Text distance, horizontal	1 mm	Default distance of text, horizontal
Coating line in sectional view Coating line in sectional view Coating frames Coa	Text distance, vertical	1 mm	Default distance of text, vertical
	🖉 Arrow type	Arrow, open 🗸	
	/ Arrow height	0.66 mm	Height of dimension arrow
Automatic drawing derivation	🖉 Arrow length	2.3 mm	Length of dimension arrow
Drawing > Apportations > Weld symbols			

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

#### Show/hide weld seams by view

The functions



Hide part list in view selection

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



### Show part list in view selection

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



#### New symbols in the context menus

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



#### Views

#### Magnetic snap-in when moving

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

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



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

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

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

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



New dialogue for sectional views

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

Sectional view	×
- Original view	A -
View 1	(0) 
- Section path	
Select sketch	
Delete sketch after creation	
- Positioning	
Place view	() () () () () () () () () () () () () (
- Type	]
Depth: 1	*
Unfolded	1 to the second
- Limited	
Process limiting sketch	143
- 🗸 Ident (A) —	
- Caption (A-A)	
- View property	
Scale: 1:1	•
Show coating	
- 🗸 Hatching	
Preview OK Can	cel Apply

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



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

#### Multiple selection of views

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

#### **Rotating views**

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



3-D rotation point

#### Deleting dimensions in shortened views

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

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



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



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

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



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

#### Temporarily deactivate view shortening

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



#### Temporarily deactivate view shortening

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



#### Cancel temporarily activation of view shortening

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

### Consideration of intrusions in Hidden Line representations

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

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

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



#### Fast

Only collisions between straight edges and plane facets are considered.

#### Thorough

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

### Cams and cam processings

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

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

Cam processing	×
- Part to be processed	
Select part	
- Part with cams	
Select part	
- Properties	
Clearance in longitudinal direction:	0.1 -
Clearance in transverse direction:	0.1 •
Perpendicular to surface	
Corner processing:	Without processings 🔻
Width:	1 *
Depth:	1 *
ОК	Cancel Apply

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

### Update automatically calculated attributes when loading

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

### Extensions in lettering

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

#### Attributes

Attributes can be easily copied from the list box. Select the attribute and click **OK**. There are some restrictions. Attributes of superordinate parts may not be used. The lettering text must not depend on the position in the part structure. If the part is referenced, only attributes that are transferred via referencing may be used. This can be set in the Configuration Editor under **System settings** > **Referencing** > Area: **Updating**. Examples of non-permitted attributes: The attribute **Benennung 2** is configurable, i.e. non-permitted attributes can be used. The attribute **Positionsindex** is not permitted because parts with the same position number can have different item indices.

#### Multi-line signatures

You can now enter multi-line texts in the text editor of the lettering.

#### Save settings as favourites

Letterings can be saved as favourites. The file format is \*.FTD. In the HiCAD SYS directory you will find various FTD files with predefined lettering.

#### Showing and hiding letterings

The lettering can be shown or hidden in the view. To do this, right-click on the view frame and select the function Show/hide elements in view.

Z1 litem number 5

# **Catalogue Editor**

## Service Pack 2 2024 (V 2902)

### Units and unit categories

Since HiCAD 2023, the Catalogue Editor has supported unit categories (e.g. Length, Surface area, Volume, Weight) and units (e.g. mm, m, in, ft, kg) as column properties - initially only for fasteners. With SP2, the units have now been set in almost all catalogues, so that when installing in HiCAD it is clear in which unit the standard parts are available in the catalogue.

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

### Profile installation - new system attributes

New attributes for the sheet thicknesses of the sandwich inner and outer shells, the core thickness and the core material are available in the **System attributes** table in the **System settings** catalogue. When installing profiles from the catalogues at **Factory standards > Series > Roof Wall Facade > Room-closing profiles**, the content of the corresponding column is assigned to the respective system attribute.

Attribute name	Designation	Column
DWF_CORE_MAT	Core material	CORE_MATERIAL
DWF_T_CORE	Core thickness	CORE_THICKNESS
DWF_T_IN	Thickness of inner shell	THICKNESS_INSIDE
DWF_T_OUT	Thickness of outer shell	THICKNESS_OUTSIDE

### **ROMA** profiles

As of SP2, panels from the company Romakowski GmbH & Co. KG are now also available for profile installation at Factory standards > Series > Roof Wall Facade > Room-closing profiles > Romakowski.



### US Standard boltings

As of SP2, the HiCAD Bolting dialogue supports US standard bolting according to the **ASME B18.2.1** standard. For this purpose, the following tables in the **Fasteners** catalogue have been replaced by new tables with the same name.

Hexagon head bolt	<ul> <li>ASTM_A193_HEAVY_HEX</li> <li>ASTM_A307_HEAVY_HEX</li> <li>ASTM_A325_HEAVY_HEX</li> <li>ASTM_A354_HEAVY_HEX</li> <li>ASTM_A354_HEX</li> <li>ASTM_A440_HEAVY_HEX</li> </ul>
	= ASTM_A449_HEAV1_HEA
Hexagon head nut	<ul> <li>ASTM_A194_HEAVY_HEX_ FLAT</li> </ul>
	<ul> <li>ASTM_A194_HEAVY_HEX_ JAM</li> </ul>
	ASTM A194 HEAVY HEX
	ASTM_A563_HEAVY_HEX
Washer	ASME_B18_21_1_PW

#### Example of such a standard bolting:

Bolt

Hexagon head bolt according to ASME B18.2.1, size 1/2"-13 UNC. This refers to a screw with an outside diameter of 1/2 inch and 13 threads per inch, which represents a coarse thread (UNC - Unified Coarse Thread). The length of the screw depends on the specific application.

Nut

Hexagon nut according to ASME B18.2.2, matching the 1/2"-13 UNC size bolt. This nut is specifically designed to mate with the above bolt and provide a secure connection.

Washer

Flat washer according to ASME B18.22.1, suitable for 1/2" diameter bolts. The washer is used to distribute the load of the nut evenly on the material to be screwed, which reduces the risk of indentation or damage to the material.

### Flow drilling - NCX output and itemisation

Since HiCAD 2024 SP1, flow drillings with inch thread and metric thread from the suppliers

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

have been available in the catalogue **Factory standards > User-defined processings**. As of SP2, these are now taken into account for NCX output and itemisation.

### Semi-finished products/profiles with imperial dimensions

In the **Semi-finished products > Beams+Profiles** catalogue, the previous tables according to the AISC standard have been replaced by tables of the same name with imperial dimensions.

Catalogue	Table	File name					
I-Beams	AISC HP (imperial)	AISC_HP-SHAPES.IPT					
	AISC M (imperial)	AISC_M-SHAPES.IPT					
	AISC S (imperial)	AISC_S-SHAPES.IPT					
	AISC W (imperial)	AISC_W-SHAPES.IPT					
L-Beams	AISC L-isosceles (imperial)	AISC_LE-SHAPES.IPT					
	AISC L-non.isosceles(imperial)	AISC_LU-SHAPES.IPT					
T-Beam	AISC MT (imperial)	AISC_MT-SHAPES.IPT					
	AISC ST (imperial)	AISC_ST-SHAPES.IPT					
	AISC WT (imperial)	AISC_WT-SHAPES.IPT					
U-Beams	AISC C (imperial)	AISC_C-SHAPES.IPT					
	AISC MC (imperial)	AISC_MC-SHAPES.IPT					
Hollow profiles	AISC HSS RE (imperial)	AISC_HSS_RE_INCH.IPT					
	AISC HSS SQ (imperial)	AISC_HSS_SQ_INCH.IPT					
Steel pipes	AISC HSS RO (imperial)	AISC_HSS_RO_INCH.IPT					
	AISC PIPE (imperial)	AISC_PIPES_INCH.IPT					

This affects the following tables:

In addition, the **Factory standards/User-defined semi-finished products** catalogue has been expanded to include the following tables according to ASTM standard:

Catalogue	Table	File name
User-defined profiles > Flat steels	FL (imperial)	ASTM_FLAT_STEEL.IPT
User-defined plates	Plates (imperial)	ASTM_STEEL_PLATES.IPT

### Project and Folder selection

Projects and Folders can now be activated via the HELiOS menu in the menu bar. The menu has been extended accordingly.

HELIOS Settings	? ISD	
<ul> <li>Connect upo</li> <li>Login</li> <li>Remove assi</li> </ul>	on program launch gnment to HELiOS	Activate project
Project	>	Show active project
Folder	>	Activate folder
		Deactivate folder
		Show active folder

If a project or folder is active, the menu can be used to deactivate the project/folder, display the project/folder content and activate another project/folder.

Activate project	Activate folder
Deactivate project (PN-01-06-K)	Deactivate folder (AN-100/06)
Show active project (PN-01-06-K)	Show active folder (AN-100/06)

The title bar of the Catalogue Editor shows which projects and which folders are currently active:



### Threaded rods - Change of category

The tables with the category GEWINDEBOLZEN (Threaded rods) have been replaced in SP2 by tables of the same name with the category GEWINDEBOLZEN\_BEARB\_ (Threaded rods, processed). This affects the following tables:

- Fasteners > Bolts+Screws > Screw bolts > DIN 976-1 A
- Fasteners > Bolts+Screws > Screw bolts > DIN 976-1 B
- Factory standards > User-defined fasteners > User-defined bolts+screws > Fastenal > Rods and studs > Threaded rods

The change means that these threaded rods can be edited in HiCAD. This means that threaded rods with the same table entry can have different item numbers. This is the case, for example, if two identical threaded rods are inserted and then one of the parts is shortened.

### Further ALUCOBOND® colours

As of SP2, the **ALUCOBOND® Colour 539 C2 Light Gold** is also available in the HiCAD catalogues. This affects the tables:

#### ALUCOBOND Anodized in the catalogue Factory standards\Sheets\ALUCOBOND

### Look

3	36		ALUCOBOND 3mm IC2	3	ALUCOBOND 3mm	ALUCOBOND 3mm	3	0.01	1.5 R:ACP.ABW	R:ACP_outside.ABW	IC2
9	37	•	ALUCOBOND 4mm IC2	4	ALUCOBOND 4mm	ALUCOBOND 4mm	4	0.01	1.5 R:ACP.ABW	R:ACP_outside.ABW	IC2
15	38	•	ALUCOBOND 6mm IC2	6	ALUCOBOND 6mm	ALUCOBOND 6mm	6	0.01	1.5 R:ACP.ABW	R:ACP_outside.ABW	IC2
21	39		ALUCOBOND A2 3mm IC2	3	ALUCOBOND A2 3mm	ALUCOBOND A2 3mm	3	0.01	1.5 R:ACP.ABW	R:ACP_outside.ABW	IC2
27	40	•	ALUCOBOND A2 4mm IC2	4	ALUCOBOND A2 4mm	ALUCOBOND A2 4mm	4	0.01	1.5 R:ACP.ABW	R:ACP_outside.ABW	IC2
33	41	•	ALUCOBOND Plus 3mm IC2	3	ALUCOBOND Plus 3mm	ALUCOBOND Plus 3mm	3	0.01	1.5 R:ACP.ABW	R:ACP_outside.ABW	IC2
39	42		ALUCOBOND Plus 4mm IC2	4	ALUCOBOND Plus 4mm	ALUCOBOND Plus 4mm	4	0.01	1.5 R:ACP.ABW	R:ACP_outside.ABW	IC2

# Coating ALUCOBOND Anodized in the catalogue Factory standards\Surface treatment\Profile and Element Installation

Look

	ID MOD	STATUS	Designation	NUM	CAT_ITEM
1	1	•	ALUCOBOND I SATIN BROWN	0	876:165
2	5	•	ALUCOBOND IC0/EV1	1	876:185
3	6	•	ALUCOBOND IC2	2	876:17
4	3	•	ALUCOBOND IC31	31	876:178
5	2		ALUCOBOND IC32	32	876:159
6	4	•	ALUCOBOND IC34	34	876:96

New tables are also available:

- in the catalogue Factory standards/Sheets/ALUCOBOND:
  - ALUCOBOND legno PREMIUM WOOD
  - ALUCOBOND PREMIUM ANODISED
  - ALUCOBOND ROCCA
  - ALUCOBOND VINTAGE
- in the catalogue Factory standards/Surface treatment/Profile and Element Installation/ALUCOBOND:
  - Coating ALUCOBOND legno PREMIUM WOOD
  - Coating ALUCOBOND PREMIUM ANODISED
  - Coating ALUCOBOND Rocca
  - Coating ALUCOBOND Vintage

### CatalogueMaker- Factory standards

Tables from factory standards catalogues were not previously taken into account by the Catalogue Maker. As of SP2, this is now supported, i.e. your individual standard selection can now also contain standard parts from the **Factory standards** catalogue.



# Service Pack 1 2024 (V 2901)

### Flow drillings

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

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

- M (	Ð 🕘 🕒	🔊 🗋 🐻		‡ ⊇ 9	2   to to to To   🖻 6	1 6 ?						
Factory standards		ID N	MOD	STATUS	Designation	Size	TYPE	Form	DN	P	DB	
User-defined building materials	1	1		•	Ø 9,2	G 1/8	R	short	9.728	0.907	9.2	_
User-defined processings	2	2		•	Ø 9,2	G 1/8	R	short-flat	9.728	0.907	9.2	
User-defined bores for blind rivet nuts	3	3		•	Ø 9,2	G 1/8	R	long	9.728	0.907	9.2	
User-defined bores for inim rivet ruts     User-defined bores for rivets     User-defined bores for rivets     User-defined bores for rivets     User-defined threads     User-defined flow drilling     O User-defined flow drilling     O User-defined flow drilling	4	4		•	Ø 9,2	G 1/8	R	long-flat	9.728	0.907	9.2	
	5	5		•	Ø 9,3 (for stainless steel)	G 1/8	R	short	9.728	0.907	9.3	
	6	6		•	Ø 9,3 (for stainless steel)	G 1/8	R	short-flat	9.728	0.907	9.3	
	7	7		•	Ø 9,3 (for stainless steel)	G 1/8	R	long	9.728	0.907	9.3	
Centerdrill inch thread	8	8		•	Ø 9,3 (for stainless steel)	G 1/8	R	long-flat	9.728	0.907	9.3	
Centerdrill metric thread	9	9		•	Ø 12,4	G 1/4	R	short	13.157	1.337	12.4	
	10	10		•	Ø 12,4	G 1/4	R	short-flat	13.157	1.337	12.4	
THERMORILL inch threads	11	11		•	Ø 12,4	G 1/4	R	long	13.157	1.337	12.4	
F-S GLOWDRILL	12	12		•	Ø 12,4	G 1/4	R	long-flat	13.157	1.337	12.4	
Glowdrill inch thread	13	13		•	Ø 15,9	G 3/8	R	short	16.662	1.337	15.9	
Glowdrill metric thread	14	14		•	Ø 15,9	G 3/8	R	short-flat	16.662	1.337	15.9	
User-defined countersinks	15	15		•	Ø 15,9	G 3/8	R	long	16.662	1.337	15.9	
	- 16	16		•	Ø 15,9	G 3/8	R	long-flat	16.662	1.337	15.9	
	17	17		•	Ø 19,9	G 1/2	R	short	20.955	1.814	19.9	
	18	18		•	Ø 19,9	G 1/2	R	short-flat	20.955	1.814	19.9	
	19	19		•	Ø 19,9	G 1/2	R	long	20.955	1.814	19.9	
	20	20		•	Ø 19,9	G 1/2	R	long-flat	20.955	1.814	19.9	
	21	21		•	Ø 25,4	G 3/4	R	short	26.441	1.814	25.4	
	22	22		•	Ø 25,4	G 3/4	R	short-flat	26.441	1.814	25.4	
	23	23		•	Ø 25,4	G 3/4	R	long	26.441	1.814	25.4	
	24	24		•	Ø 25,4	G 3/4	R	long-flat	26.441	1.814	25.4	

Flow drill holes can be inserted in HiCAD with the function 3-D Standard > Standard Processings > Bore/Thr

. The bores are displayed in HiCAD like normal drilled holes.
# Major Release 2024 (V 2900)

### Tool numbers in sheet metal processing

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

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

### User-defined columns in catalogue tables

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

		New column		×		
		Column propert	es			
Cut	Ctrl+X	Designation	EXAMPLE			
Сору	Ctrl+C	Displayed nam	Example		ERDI	Example
aste	Ctrl+V	Data type	Floating point number	-	4.773	
elete	Del	Category	Length	•	6.773	
New column		Unit	mm	-	8.773	
elete column		Comment		_	8.466	
					10.773	
olumn properties	2840	(Link attributes)	6		10.466	
liter in selection dia	logue	HiCAD		-	10.16	
Find"DIN 480"		HELIOS		-	12.16	
-lide column			Ignore when updating	_	14.16	
nac column				$\equiv$	16.16	
		(ок)	Car	ncel	18.16	

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

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

# Displayed names of table columns adapted

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

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

The following displayed names have been assigned:

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

### Fastenal - Threaded Rods

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

CATEditor - [ Catalogues\Factory standards\User-de	fined faste	ners\User-defined	bolts+scre	ws\Fastenal\Rods	and Studs\TI	hreaded Rod ]	[C:\HiCAD\k	ataloge ]	[Version: 2	9.0.0.97 ]			- 1	o x
e Edit View Extras HELiOS Settings ? ISD	•		110			149								
Eactory standards		ID MOD	STATUS	Designation	ARTICLE	Size	MATERIAL	OBERFL	TYPE	Thread	DN	Р	PITCH	LN
User-defined building materials	1	1		#4-40x3ft	47537	#4-40x3ft	AISI 304		AISI 304	#4-40	0.112	0.025	40	3
User-defined processings				#6-32x3ft	47538	#6-32v3ft	AISI 304		AISI 304	#6-32	0.138	0.0213	32	3
E Vser settings	-	2		#0.32x3ft	47520	#0.32x3ft	AIGI 204		AIGI 204	#0.32	0.150	0.0313	22	
User-defined moulding tools	-	3		#0-32X31L	47359	#0-32X3IL	AIGI 204		AISI 304	#0-32	0.104	0.0313	52	5
User_defined semi-finished products	4	4		#8-32X0TT	47490	#8-32X0Tt	AISI 304		AISI 304	#8-32	0.104	0.0313	32	0
Wer-defined textures	5	2		#10-24x3ft	4/540	#10-24x3ft	AISI 304		AISI 304	#10-24	0.19	0.0417	24	3
User-defined fasteners	6	6		#10-24x6ft	47401	#10-24x6ft	AISI 304		AISI 304	#10-24	0.19	0.0417	24	6
User-defined anchors	7	7	•	#10-24x12ft	47341	#10-24x12ft	AISI 304		AISI 304	#10-24	0.19	0.0417	24	12
User-defined studs	8	8	•	1/4"-20x3ft	47543	1/4"-20x3ft	AISI 304		AISI 304	1/4"-20	0.25	0.05	20	3
User-defined dowels	9	9		1/4"-20x6ft	47402	1/4"-20x6ft	AISI 304		AISI 304	1/4"-20	0.25	0.05	20	6
User-defined sockets	10	10	•	1/4"-20x12ft	47342	1/4"-20x12ft	AISI 304		AISI 304	1/4"-20	0.25	0.05	20	12
User-defined rivets	11	11	•	5/16"-18x3ft	47545	5/16"-18x3ft	AISI 304		AISI 304	5/16"-18	0.3125	0.0556	18	3
E Ser-defined elbows	12	12	•	5/16"-18x6ft	47403	5/16"-18x6ft	AISI 304		AISI 304	5/16"-18	0.3125	0.0556	18	6
User-defined bolts+screws	13	13	•	5/16"-18x12ft	47343	5/16"-18x12ft	AISI 304		AISI 304	5/16"-18	0.3125	0.0556	18	12
E 🚸 ALUCOBOND	14	14	•	3/8"-16x3ft	47563	3/8"-16x3ft	AISI 304		AISI 304	3/8"-16	0.375	0.0625	16	3
User-defined clinch studs	15	15	•	3/8"-16x6ft	47404	3/8"-16x6ft	AISI 304		AISI 304	3/8"-16	0.375	0.0625	16	6
Ejot	16	16		3/8"-16x12ft	47344	3/8"-16x12ft	AISI 304		AISI 304	3/8"-16	0.375	0.0625	16	12
Elerni	17	17	- <u>-</u>	7/16"-14x3ft	47565	7/16"-14x3ft	AISI 304		AISI 304	7/16"-14	0.4375	0.0714	14	3
H- Bolts	10	18		7/16"-14x6ft	47405	7/16"-14×6ft	AISI 304		AISI 304	7/16"-14	0.4375	0.0714	14	6
🗄 🚸 Lag Screws	10	10		7/16"-14x120	47245	7/16"-14×120	AISI 204		AISI 204	7/16"-14	0.4275	0.0714	14	12
H Achine Screws	19	19		1/0= 12.26	47543	1/2= 12.24	AIGI 204		AISI 304	1/2= 12	0.4575	0.0714	12	12
Rods and Studs	20	20		1/2 - 13x3ft	4/30/	1/2 - 13X3Tt	AISI 304		AISI 304	1/2 - 13	0.5	0.0769	13	3
Threaded Rod	21	21		1/2"-13x0ft	47406	1/2"-13x0ft	AISI 304		AISI 304	1/213	0.5	0.0769	13	0
E E E Ser-Drilling Screws	22	22		1/2"-13x12ft	47346	1/2"-13x12ft	AISI 304		AISI 304	1/2*-13	0.5	0.0769	13	12
	23	23	•	9/16"-12x3ft	47569	9/16"-12x3ft	AISI 304		AISI 304	9/16"-12	0.5625	0.0833	12	3
	24	24	•	9/16"-12x6ft	47407	9/16"-12x6ft	AISI 304		AISI 304	9/16"-12	0.5625	0.0833	12	6
	25	25		9/16"-12x12ft	47347	9/16"-12x12ft	AISI 304		AISI 304	9/16"-12	0.5625	0.0833	12	12
	26	26	•	5/8"-11x3ft	47585	5/8"-11x3ft	AISI 304		AISI 304	5/8"-11	0.625	0.0909	11	3
	27	27	•	5/8"-11x6ft	47408	5/8"-11x6ft	AISI 304		AISI 304	5/8"-11	0.625	0.0909	11	6
	28	28	•	5/8"-11x12ft	47348	5/8"-11x12ft	AISI 304		AISI 304	5/8"-11	0.625	0.0909	11	12
	29	29	•	3/4"-10x3ft	47587	3/4"-10x3ft	AISI 304		AISI 304	3/4"-10	0.75	0.1	10	3
	30	30	•	3/4"-10x6ft	47409	3/4"-10x6ft	AISI 304		AISI 304	3/4"-10	0.75	0.1	10	6
	31	31	•	3/4"-10x12ft	47349	3/4"-10x12ft	AISI 304		AISI 304	3/4"-10	0.75	0.1	10	12
	32	32		7/8"-9x3ft	47589	7/8"-9x3ft	AISI 304		AISI 304	7/8"-9	0.875	0.1111	9	3
Inner	22	33		7/8"-9x6ft	47410	7/8"-9x6ft	AISI 304		AISI 304	7/8"-9	0.875	0.1111	q	6
	33	34		7/8"-0-12#	47350	7/8"_0v12f+	AISI 304		AISI 304	7/8=_0	0.875	0.1111	0	12
	34	34		1" 0.04	47501	1" 0.04	AISI 204		AIGI 204	1" 0	1	0.125	9	12
	35	33		1-02.511	47411	1 -0X3IT	AIGI 204		AIGI 201	1 -0		0.125	8	3
	36	30		1 -8x6ft	4/411	I -8x6ft	AISI 304		AISI 304	1"-8	1	0.125	8	6
	37	37	•	1"-8x12ft	47351	1"-8x12ft	AISI 304		AISI 304	1"-8	1	0.125	8	12
ly											0 96	14:50:11	52120131	5

# Henkel Teroson foils

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

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

CATEditor - [ Catalogues\Factory standards\User-defined build	ding ma	terials\Foils	\Henkel\TEF	ROSON	I\TEROSON FO 2 SK1/SK2 ] [ C:\Hid	CAD\Kataloge	[ Version	: 29.0.0.97 ]		- 0
Edit View Extras HELiOS Settings ? ISD										
- M 🕑 🗩	10 0	ι.	11 2 9	2   *0	• *• *• *• *•   • • • • • • •					
E No Factory standards		ID	MOD ST	ATUS	Designation	ARTICLE	Size	MATERIAL	OBERFL	ΤΥΡΕ
E Ser-defined building materials	1		1	•	TEROSON FO 2 SK1 150x0.3mm	2919061	150x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
General building materials	2		2	•	TEROSON FO 2 SK1 200x0.3mm	2919062	200x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
Insulating material     Foils			4	•	TEROSON FO 2 SK1 300x0.3mm	2919064	300x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
Henkel	4	3	3	•	TEROSON FO 2 SK1 250x0.3mm	2919065	250x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
E- TEROSON	5		5		TEROSON FO 2 SK1 350x0.3mm	2919066	350x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
TEROSON FO 1 FOIL-TACK	6		6	•	TEROSON FO 2 SK1 400x0.3mm	2919067	400x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
TEROSON FO 1 FOIL-TACK DUO	7		7	•	TEROSON FO 2 SK1 500x0.3mm	2919068	500x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
TEROSON FO 150 FOIL-TACK M+S	8	1	8	•	TEROSON FO 2 SK2 150x0.3mm	2919069	150x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
TEROSON FO 2 SK1/SK2	9		9	•	TEROSON FO 2 SK2 200x0.3mm	2919070	200x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
	10	10	0		TEROSON FO 2 SK2 250x0.3mm	2919081	250x0.3	Fleece. Open to diffusion		Fleece. Open to diffusion
TEROSON FO 50 FOIL-TACK	11	1	1	•	TEROSON FO 2 SK2 300x0.3mm	2919082	300x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
TEROSON FO 50 FOIL-TACK DOO	12	10	2	•	TEROSON FO 2 SK2 350x0.3mm	2919083	350x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
TEROSON FO KSK M+S	13	13	3		TEROSON FO 2 SK2 400x0.3mm	2919084	400x0.3	Fleece, Open to diffusion		Fleece, Open to diffusion
TEROSON TA ALU	14	14	4	•	TEROSON FO 2 SK2 500x0.3mm	2919085	500x0.3	Fleece. Open to diffusion		Fleece. Open to diffusion
Y	-			_					[14 ][1	4:52:30 521201314

# User-defined tables with Steel Engineering plates

At Factory standards > User-defined semi-finished products > User-defined plates you now have the possibility to create tables with your own steel plates. These tables are then also offered for selection via the func-

#### tion Steel Engineering > Plate, new > Rectangular plate

CATEditor - [ Catalogues\Factory standards\User-defined semi-	-finished products	User-defin	ed plates\Raw plates ] [C:\	HiCAD\Katal	oge] [Versio	n: 29.0.0. <sup>-</sup>	102 ]			- 0	×
File Edit View Extras HELiOS Settings ? ISD											
- M   🗩	📲 🗈 ï <sub>b</sub> 🖶	14 🖸 9	2   to to to To   🖻 🖻								
E Sectory standards	ID MOD	STATUS	Designation	Size	MATERIAL	OBERFL	ТҮРЕ	DSTV	S	GEW	HGEW
User-defined building materials	1	•	DIN EN 10029-S235JR-3	3	S235JR		Plate	BL1	1	48	0
	2	•	DIN EN 10029-S355MC-3	3	\$355MC		Plate	BL2	1	48	0
User-defined moulding tools 3	3	•	DIN EN 10029-S235JR-4	4	S235JR		Plate	BL3	1.5	108	0
User-defined dished ends	4	•	DIN EN 10029-S355MC-4	4	\$355MC		Plate	BL4	1.5	108	0
User-defined plates	5	•	DIN EN 10029-S235JR-5	5	S235JR		Plate	BL5	1.55	88	0
Raw plates		_									
User-defined textures											
User-defined materials											
. User-defined cylinders											
E-Series											
H Send zone tooling											
Ready							(1.0	STV	12.15	14.22	45 10
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Rectangular plate	$\times$										
- Insertion point											
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Select insertion point											
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Select second point		Raw pla	ites								×
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- Plate parameters	<b>*</b> ₽	+	FING	1					All		
DIN EN 10029-S235JR-4 🔹		Sen	ni-finished product	s			MATERIA	L Desi	anation		
		5 500	in ministree produce	-			C22510	DINC	NI 10000	CODEID	2
Width (X): 105,890044		4 🥪	Plates				5235JK	DINE	IN 10029-	-5235JK-	3
			Plate				SSSSINC	DINE	IN 10029-	-3322100	3
Depth (Y): 56,50248							5235JR	DINE	IN 10029	-5235JR-	4
			Bulb plate (DIN	59220)			S355MC	DINE	N 10029	-S355M0	4
- General		1	X EN 10131				S235JR	DIN E	N 10029	-S235JR-	5
Referenced			JIS G 3193								
		Fact	ory standards								
Apply immediate	ely 🗌 🛛										
OK Cancel Apply	v		User-defined semi-	finished	products						
		4 6	User-defined pla	ates							
			D L								
			Kaw plates								
						Tur	o/Matorial				
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						Plat	te				-
		-				_					
		DINE	N 10029-S235JR-4 -	5235JR					JK	Can	cel

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

# **Point clouds**

# Service Pack 2 2024 (V 2902)

# **Clipping Box Manager**

With the revised Clipping Box Manager function, it is possible to transfer a multiple selection of views from the drawing to the dialogue window after starting.

The dialogue window then shows in the Views area how many views are selected. The selection of views can also be removed by clicking on the cross symbol. To activate a view, click on it with the left mouse button or,

after clicking on the icon, activate it in the drawing. If the view is already active, it is then deleted from the selection. The activated views are marked with an orange border in the drawing. As before, the functions and all clipping boxes in the drawing are listed under Clippings. The checkbox in front of the different representations of the point cloud indicates which clippings are used in the views. You can change this assignment here.



- (1) Area for selecting and removing views
- (2) (2) Clippings: Icons for creating, deleting and copying views of the point cloud
- (3) List of differently clipped point clouds
- (4) Active clipping Front view, Back wall in the 3-D model view

# Message in case of high memory utilisation

Process information has been added to the Point Cloud Converter. This allows you to monitor the conversion process and a warning is issued if, for example, the memory utilisation is too high.

🧠 P	oint cloud conve	ersion						- 0	×		
	Runtime	Target name	Target path	 File path	Size	- Settings					
1.15	00-00-29	ISD 001	CAHiCAD\Punk	C\\HiCAD\Punk	49.3 GB	Move point cloud into origin					
A	00100123		C:\HiCAD\Punk	 C:\HiCAD\Punk	17.8 GB				0		
	-	ISD Fortigung	CAHiCAD\Bunk	 CAHiCADABunk	0.5 CP	- Process information -			0		
		ISD Lesse	CALICAD Punk	 CALLICAD Punk	9.5 CD						
0		ISD_Lager	C:\HICAD\Punk	 C:\HICAD\Punk	8.3 GB	Name	Value				
						Total of physical memory	31.82 GE	<b>B</b>			
						PC memory in use	48%				
						Available PC memory	16.39 GE	3			
						Total of PC paging file	36.57 GB				
						Available in PC paging file	14.45 GB				
						Current directory	C:\HiCAD	/exe			
						Free memory space C:\	192.97 G	iB			
						Conversion program:					
						Name		Value			
						Program status		OK			
						File name		ISD_001			
						Free memory space (Target	path) C:\	192.97 GB			
						Paged memory size		1.21 GB			
						Paged system memory size	2	1.29 MB			
						Peak working set		1.23 GB			
						Working set		1.23 GB			
						Program processor time		00:00:13.875			
						No warnings					
								Close			

# Major Release 2024 (V 2900)

### **Clipping Box Manager**

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

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



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

Use the functions in the Clipping Box Manager to create, copy and delete views from the point cloud. The function **New** derives a new view of the complete point cloud without clipping box. If you select **Copy**, the active view of the point cloud with clipping box and recess is duplicated. The new view is automatically active.

If you then exit the manager with OK, all edits, e.g. Edit clipping box **Subtract**, refer to the active view. It is indicated by an activated checkbox **S**.

#### The Clipping Box Manager for Sheet views

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



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

# **Feature Technology**

# Service Pack 2 2024 (V 2902)

# **New Feature functions**

The feature formulas have been expanded in HiCAD 2024 SP2. You can find a list of the current functions here.

New functions	
Name	Description
part_of	(o: Object): Part - Returns the part to which the given geometry belongs (edge / surface / beam).
edge_start	(e: Edge): Point - Returns the starting point of an edge.
edge_end	e: Edge): Point - Returns the end point of an edge.
face_area	(f: Face): Real - Returns the surface of the facet.
part_volume	(p: Part): Real - Returns the volume of the part/assembly.
part_mass	(p: Part): Real - Returns the mass ("weight") of the part/assembly.
part_center_of_ volume	(p: Part): Point - Returns the centre of gravity of the part.
part_center_of_ mass	(p: Part): Point - Returns the centre of mass of the part.

New mathem	New mathematical functions							
Name	Description							
pi	(): Real - Returns the circle number PI.							
tau	(): Real - Returns the full revolution in radians (2x PI).							
е	(): Real - Returns the base of the natural logarithm.							
min	(x1: Real; x2: Real): Real - Returns the smaller of two numbers.							
max	(x1: Real; x2: Real): Real - Returns the larger of two numbers.							
clamp	(x: Real; min: Real; max: Real): Real - Returns a number bound in the range between min and max.							
atan2	(y: Real; x: Real): Real - Returns the directed angle based on the full circle (four quad- rants).							

# Service Pack 1 2024 (V 2901)

# Naming of feature functions

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

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

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

# Major Release 2024 (V 2900)

### Feature for part and sheet creation

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

### Create feature variant

When creating a Feature variant (as a VAA file), the assigned units are now taken into account and displayed

for information when saving the variant with the **Create feature variant function**. You can then use the Variant Editor to change the units in the VAA file.



# HCM

# Service Pack 2 2024 (V 2902)

Groups with isolated points

You can now use the **Group** A function to group together not only line elements but also isolated points. Line elements and isolated points that are contained in a group are only moved+rotated together by the HCM.



(1) The **D** and the isolated point form a group.

# Major Release 2024 (V 2900)

If you deactivate the HCM constraint for a sketch, this is indicated in the HCM window of the ICN. Click on the

Symbol to activate the HCM. If possible, HCM constraints are automatically assigned for the following sketches.

### HCM update when variables are changed

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

Designation	Value
Main assembly	
العام الع	100
🧕 length2	300
length3	300
🥶 width1	100
width2	200
🥶 width3	300

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

#### The variables of an assembly

💌 3-D Part HCM Settin 🗙
AutoRemove constraints (1)
✓ Preserve Drag constraints
AutoUpdate HCM model
OK Cancel

Settings for the 3-D Part HCM.

# **Configuration Management**

# Service Pack 2 2024 (V 2902)

# Referencing - Updating identical parts

In the Configuration Editor at **System settings > Referencing**, the parameters:

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

have been added again in SP2. This allows you to choose whether all identical parts in the current drawing should be updated automatically when changing externally or internally referenced parts.

Edit View Extras ISD				
0 😂   👫 12 13 👫		AA   🞯 🖕	User	Ŧ
System settings	*	Description	Value	Comment
Assembly HCM Sketch HCM		Show selection dialogue when deleting referenced parts		
Itemisation Processing plane		Automatic name selection for referenced parts without database entry		
Scales		Compared part attributes for identical part search when referencing	Collection	1 entry per row, e.g. \$BB=Article designation \$BK= Article master
Units		Updating		
Start configuration		Update sectional views, detail views and cut- outs when loading referenced parts		
I load/Save		Update colour	✓	Update colour for referenced par
Data save		Update layer	✓	Update layer for referenced parts
Identification		Update line type	✓	Update line type in referenced pa
Referencing	=	Synchronize item numbers/part attributes when updating file	For main parts and sub-parts	]
<ul> <li>Calculations</li> </ul>	-	Automatically update referenced parts after each change	Yes 🗸	Updating of identical parts
Graphic     G		Update identical parts of referenced parts before saving		Only important if referenced part are not to be automatically updated
<ul> <li>2-D Lines</li> <li>Miscellaneous</li> <li>Standard Parts</li> </ul>	•	Recover dimensions and weld symbols		Recovers dimensions and weld symbols the bodies of which can no longer be assigned due to the updating, by moving them to the

# Update automatically calculated attributes before saving

For the calculation of attributes that are set to **Manual / When itemising** in the Configuration Editor, you can specify whether or not these calculations should be performed automatically before saving referenced parts and assemblies. The parameter **Update automatically calculated attributes before saving** is available in the Configuration Editor at **Modelling > Part properties**. Weight (§01), Surface area (§10 and §SC) and Volume (§20) are updated. The attributes Total quantity. (%06) and Qty. in assembly (%13) are not updated.

# Service Pack 1 2024 (V 2901)

# Time threshold for AutoQuickView

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

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

### SpaceMouse®

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

### Referencing

#### Update identical parts

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

#### Referenced assemblies with referenced parts

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

### Additional column in the Packaging dialogue

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

# Development attributes for Steel Engineering plates

#### If the settings

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

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

# **Drawing Management**

#### Documents for general documents

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

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

#### Managing general 3-D parts via part filter

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

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

# Plant Engineering

#### Pipe length check no longer as macro

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

#### Insert plane flange

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

#### Settings for down-grade symbols

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

#### Interfaces

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

# Major Release 2024 (V 2900)

Part properties

#### Automatically calculated attributes

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

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

#### Parameter configuration

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



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

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

### CFGDBTool.exe

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

### Intersections in the Hidden Line representation

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

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

### Part master display by double click

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

#### Magnetic snap-in when moving views

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

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

#### Negative and positive position in identical part search

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

# Processing external drawings

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

# Processing sheet metal developments in production drawings

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

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

# Generate pipe spool drawing from sheet view

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

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

# Automatic BOMs for itemised source models

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

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

# **Bill of Materials / Report Manager**

# Service Pack 2 2024 (V 2902)

# Text file with separator

When exporting text files as a bill of materials, you can now also end each line with a separator. To do this,

select a separator in the Settings for the text file and then activate the option Separator at the end of each line

Export Text (Settings)	×
Create a text file with both lists (1)	
Output head data	
○ None	
All selected for output	
<ul> <li>All</li> </ul>	
Output data columns	
Visible	
Visible if selected for output	
<ul> <li>All selected for output</li> </ul>	
Round numerical values to the specified number of decimal places ()	
✓ Use alias names for column headers (if defined)	
Show unit abbreviations in column headers	
✓ Visualize levels (Structure list) ()	
Character for data record separation: #	
Separator at end of each line ()	
AutoAdjust column widths	
Character for field boundary: [No character]	
Decimal separator:	
File format: iso-8859-1 (Default) Y	
Query target file name (file dialogue)	
Target directory:	

The separator at the end of the line is active

```
Pos.#Menge#Teilename#Benennung#Länge [mm]#Breite [mm]#Höhe [mm]#
9#1#Blech#Blech#2491#862#3#
10#1#Blech#Blech#2491#862#}#
11#1#Blech#Blech#2491#862#3#
13#1#Blech#11-7-502-001#1441.05#986.05#4#
15#1#Tei1#B1 15#690#500#15#
21#1#Tei1#B1 12#800#450#12#
22#2#Tei1#B1 12#800#450#12#
23#1#Tei1#B1 12#800#450#12#
26#1#Tei1#B1 20#500#360#20#
29#1#Blech-Haupttei1#35-7-200-002#600#917.97#4#
34#1#538338010#Edelstahlblech 2mm#1240.4#1248.8#2#
35#1#538338010#Edelstahlblech 2mm#1240.4#1248.8#2#
36#1#Blech-Hauptteil#11-7-120-005#712#810.13#3#
37#1#Blech#45-2-0005-555#789.11#599.11#3#
38#2#Tei1#B1 20#384#150#20#
40#1#Teil#B1 12#300#300#12#
47#1#Tei1#B1 12#307.42#271.21#12#
48#1#Teil#B1 12#302.44#271.21#12#
```



### ERPlus interface - Excel export

When exporting the ERPlus BOM, the Report Manager is now executed in the background. The output is then based on the Excel template with the relevant files ERPlus.XLSX, ERPlus.CS and ERPlus.RM\_SETTINGS from the HiCAD installation directory under SYS. These are specifically only for ERPlus and cannot be selected via the normal BOM configuration.

# Set default configuration

For RM3 files that are transferred from HiCAD or HELiOS to the Report Manager without a configuration file,

you can choose **Set default configuration** (Menu bar > Settings > ...).

After calling up the function, activate the settings to be saved. All settings are active on the ISD side. Then specify HiCAD or HELiOS as the data source for the RM3 file. For HiCAD, the configuration is saved under Default-HiCAD.RM\_SETTINGS and for HELiOS under Default-HELiOS.RM\_SETTINGS. For example, you can deactivate column settings for HELiOS so that the order and visibility of the columns is adopted from the transfer file (RM3).



# Creating an RM3 file with the HELiOS API

The HELiOS API now supports the output of an RM3 file for the Report Manager from any node of the product structure. The function is IApiArticle::SaveProductStructureToRm3[Legacy].

# Major Release 2024 (V 2900)

# General adjustments

#### Empty rows in the structure list

With the new function **Extended settings (Structure list)...** 4 (under Settings) you insert empty rows in the structure list. You can determine the level up to which empty rows are to be inserted. In the output, the empty rows are taken into account in all selected formats.

e Export Settings	Tools								
HICAD	Extended settings (Structure list)	- 🖳 🛼 🦽	- 🕼 - 🕅 -	🌆 • ጆ • T • 🥯			↓1 ↓2 ↓	¢	0
1 11 12 13	Insert empty row after each assembly	Quantity list S	tructure List						
€ 6184B	up to level: 2 ×	Level	Item Qty.	Designation	L [mm]	W [mm]	Cut (Web)	Cut (Flange)	
🧊 6B4AE	up to level.	4 1	1	1 78045190	100.0	100.0			
■ 8309E.	Take into account during output:	2	1	1 6184B1FC	8.0	8.0			
E63BD3B5	Printer	2	2	1 6908A92E	40.0	50.0			
T DIN 47	Text	2	3	1 6B4AE391	5.0	5.0			
T DIN 4	Html	2	4	1 8309EA79	100.0	100.0			
DIN 4.	HICAD	2	6	2 DIN 2008-0 5v4v15	40.0	40.0			
150 47	Fuerd (devide)	2	7	2 DIN 913-M 10v12-45H	12.0				
150 47	Excel (xisx,xis)	2	8	1 DIN 6797-A 5.3-FSt	1210				
T ISO 47	<ul> <li>Excel (xml)</li> </ul>	2	9	1 ISO 4017-M 5x10-10.9	10.0				
T ISO 47		2	10	1 DIN 471-40x1.75					
1 ISO 47	OK Cancel								
1 ISO 4762-	M 6x20-10.9 (Hexagon socket cheese hea	A 1	2	1 E63BD3B5	174.0	116.0			
1 ISO 4762-	M 6x20-10.9 (Hexagon socket cheese hea	v 🕴 📜 2	1	1 06026465	116.0	174.0			
	>	. 2	2	1 301334E3	174.0	116.0			
N-00005 /Pinion	accembly (3)	2	3	2 DIN 472-32x1.2					
	usseniolyj (sj	2	4	1 DIN 472-47x1.75					
Designation	Value	2	5	8 ISO 4762-M 6x20-10.9	20.0				
:-D/3-D	3		2	1 Distance and be	47.0	47.0			
cceptance index			3	1 Pinion assembly	47.8	47.9			
Advance Order No.	0	2	2	2 DIN 625 - 6005	00.0	41.5			
Angle 1 of section s	chema		2	2 0111 025 0005					
Anale 2 of section s	chema		4	1 Gear wheel assembly	76.1	76.0			
Angle hottom/left -	. ¥7	2	1	1 F2A90D14	69.0	76.1			
Angle bottom/left	V7	2	2	2 DIN 625 - 6002					
angle bottom/left -	12 V7	2	3	1 ISO 2338-8x32-St	32.0				
ungle bottom/right	- XL								
Angle bottom/right	t - YZ	Σ							
Aperture angle		V <							

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

۵m

#### Auto-completion

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

Qua	ntity list Structure List							
u)	Designation	Filter (i)		۲	Attribute name	Category	Uni	t
e)	Level			~	%Stufe			
•	Material name			1	%Material			
N.	Item number			1	%Posnr			
2	Number			1	%Anzahl			
	Top view of cut			1	H_\$09			
•	Front views of cut			1	H_\$10			
	Article number	@.	8	✓	H_\$BB			
2	Weight 🚽	fx Contains	Boolean Contains(	Syste	em.String)		^	ram
3	Midah	fx EndsWith	Boolean EndsWith	(Syst	tem.String)			natra
~	Width	fx Equals	Boolean Equals(Sy	stem	n.String)			neure
5	Length	Jx StartsWith	Boolean StartsWit	h(Sys	stem.String)			netre
3	Total weight	Jx charAt	System.String chai	At(L	ouble)			1200
2	Iotal weight	Jx charCodeAt	System.Object cha	rCod	ieAt(Double)			
2	Total surface	Jx concat	System.String cond	at(S	ystem.Object[])			re metre
		Jx indexOf	Int32 indexOf(Syst	em.(	Object,Double)			
		Jx lastindexUf	Int32 lastindexOf(	syste	m.Object,Double)			
			Int32 localeCompo	are(S	ystem.Object)			
		Jx match	System.Object ma	tch(S	ystem.Object)	011 0		
		Jx replace	System.String repl	ace(S	system.Object,Syste	m.Object)	8	
_		Jx search	Int32 search(Syste	m.01	bject)			
	Expanded display	Jx slice	System.String slice	Dou	Die, system. Object)	Object Surteur Object		Consel
	expanded display V show only v	Jx split	Microsoft.JScript.A	ray	Joject split(System.	object, system. Object	/	Cancel
		Jx substr	System.String subs	tr(D	ouble,System.Objec	<i>U</i>		

Placeholder and dot for displaying conditions

Quality list our	cture List								
Designation	Attribute name	Alias name	۲	Formula	Alignment	<b>73</b> 8	Category		
🔄 Total weight	_ATTR_@1	Total weight	~	[%Anzahl]*[H_§01]	Right Y	1	Weight	Ŷ	
Jotal surface	_ATTR_@2	Su	~	[H_§10]*[%Anzahl]	Right ~	1	Surface	v	Ī
এ New	_ATTR_@3		•	[H_\$BB].	Right ~	4	None	v	T
A.BC_       A.BC_         Dispatch item         Item number of         Main part num         Part type         Angle 1 of sec         Angle bottom         Curve radius a         Dispatch H [m         Installation with	Image: SK_       Image: SK_ <th>B-D () Advance ngth () Internal DSTV-part "H" () Module () N Qty. in assembly () Le 2 of section sch ure angle () Cor ius about z () () Height () I () Material leng</th> <th>Order I key of Leve lumber ema ( custom- custom- nstallat th (</th> <th>A statistviti     Bobiedin studiestri       A charAt     System.String chr       A charAt     System.String rep       A charAt     System.String rep       A search     Int32 search(System.String rep       A slice     System.String sub       A substring     System.String sub       A substring     System.String sub       A substring     System.String sub       A substring     System.String sub</th> <th>At(Double) rrCodeAt(Double) rrCodeAt(Double) cat(System.Object[Double) System.Object, Double) System.Object, Double) are(System.Object) tch(System.Object) tch(System.Object) stored by the system.Object) stored by the system.Object stored by the system.Object to the system.Object</th> <th>t) n.Object ect) Dbject)</th> <th>iect) t,System.Object)</th> <th>tem) t ID</th> <th></th>	B-D () Advance ngth () Internal DSTV-part "H" () Module () N Qty. in assembly () Le 2 of section sch ure angle () Cor ius about z () () Height () I () Material leng	Order I key of Leve lumber ema ( custom- custom- nstallat th (	A statistviti     Bobiedin studiestri       A charAt     System.String chr       A charAt     System.String rep       A charAt     System.String rep       A search     Int32 search(System.String rep       A slice     System.String sub       A substring     System.String sub       A substring     System.String sub       A substring     System.String sub       A substring     System.String sub	At(Double) rrCodeAt(Double) rrCodeAt(Double) cat(System.Object[Double) System.Object, Double) System.Object, Double) are(System.Object) tch(System.Object) tch(System.Object) stored by the system.Object) stored by the system.Object stored by the system.Object to the system.Object	t) n.Object ect) Dbject)	iect) t,System.Object)	tem) t ID	

Dot for selection for formula creation

#### **Round decimal places**

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

Export Text (Settings)	×
Create a text file with both lists (i)	
Output head data	
○ None	
<ul> <li>All selected for output</li> </ul>	
Output data columns	
<ul> <li>Visible</li> </ul>	
○ Visible if selected for output	
<ul> <li>All selected for output</li> </ul>	
Round numerical values to the specified number of decimal places	
Use alias names for column headers (if defined)	
Show unit abbreviations in column headers	
Visualize levels (Structure list) (i)	
~	

Settings for exporting the text BOM

#### Header and Footer Editor

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

Input:

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

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

Header and Footer Editor (printer output)			- D X
1 <b>1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 </b>	9 🦿 Grid: 🇮 🛱 DX: 🔤 12.00 DY:	12.00	
Page parameters:	^	₽ 7/27/2023	- 4:43 PM ()
Paper format: A4	7/27/2022 4.42 BM		
Orientation: Portrait Y	MMiller		
Page margins	DTM0815	TextBlock	e) fi
Left: 6.35 Top: 6.35	<u>م</u>	Text:	7/27/2023 - 4:43 PM MMiller DTM0815
Right: 6.35 Bottom: 6.35		Text format:	%Ss(Date) - %Ss(Time) %Ss(User)
Basic elements:		Ň	%Ss(Computer)
Text block	Formatted tout	Attribute nam	
Pixel graphic	%Ts(Text key) – Access to translatable texts	tauta a a 8/Ca(CDD)	
Document info data:	%Ss(Parameter name) - Access to system in	formation	Top
Auxiliary text 1	Date: %Ss(Date) Time: %Ss(Time)		Normal ~
Auxiliary text 2	Computer name: %Ss(Computer) User name: %Ss(User)		Normal ~
Auxiliary text 3		Text colour:	Black Y
Auxiliary text 4		Text height:	4.23
Auxiliary text 5		Text distance:	2,2,2,2
Back from authoriz.		Frame colour:	Black *
Created by		Border thickness:	0,0,0,0
Created on	,	Width:	48.42
File name:			Zoom: 1

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

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

### BOMs for HiCAD

#### Cancelling the insertion of the BOM

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

### Adjust text lengths

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

Export to HiCAD (Settings)	×			
Output data columns	Σ			
Visible				
Visible if selected for output				
Visualize levels (Structure list) (i)				
U Output rows in reverse order				
Number of rows per page: 200	Export to HiCAD (Column widths)			×
✓ Output column headers	Quantity list Structure List	🗹 A	dapt text to column w	vidth
On all pages		Alias name	Column width (i)	
✓ Use alias names for column headers (if definition)	n 🔿 Matarial name	Material	0.00	
Show unit abbreviations in column headers		Iviateriai	0.00	
	Item number	Item	7,00	~
AutoAdjust column widths	Number Number	Qty.	7,00	~
✓ Draw frame lines	Top view of cut	Cut (Flange)	35,00	~
Draw grid lines	Front views of cut	Cut (Web)	35,00	~
	Article number	Designation	25,00	~
Line colour:	🧾 🛃 Weight	Weight	0,00	~
Type of insertion: Replace an existing table or i	n 🛃 Width	w	15,00	~
Fitting point: Bottom left Y	Length	L	10,00	~
Ariel Decule	- 🛃 Total weight	Total weight	0,00	~
System font: Arial Regular	Total surface	Su	0,00	~
Bold Italic				
O HiCAD font: ANSI_KON	Expanded display 🖌 Show only visible columns	🚨 🚨 🗌	OK Cance	el
Width factor: 1	L			_

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

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

### Transfer visible structure list

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

Options	×
☑ Quantity display according to DIN (Stru	cture <mark>l</mark> ist)
Attribute name (user item number):	
Produkt.POSITIONSNUMMER	
Online Help	
Export visible structure (i)	
ОКС	ancel

# Use in the HDE reports

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

# Variant Editor

# Major Release 2024 (V 2900)

# Files managed by HELiOS

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

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

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

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



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

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

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

# Category and Unit

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

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

onnguration r Name	AGRU IS 11006 11 T KURZ		File selection	via file system	-	Derive new
Options	HiCAD variants		Language	English (United States)		
Part type	T-piece		Industry	English (United States)		U Open
Designation	1-piece		Malua far	Standard Plant Engineering		Edit repr.
Designation	agru Ind. T-piece, short, PPR grey, SDR111	SO S-5 (1 D	value for		<u> </u>	Structure
Number of rea	cords: 17			Variables + Attributes		Import
	L2	-		¬		Export
	2	designation	Generate	9		Save
4	A A A A A A A A A A A A A A A A A A A	Variable Value	Unit	Description		
1000	5.					Close
<b>L</b>		P 4.9	21 In	_	<b>-</b>	End
		P3 110	000 (mm)			Record
-	<sup>1</sup> <sup>1</sup>	L1 126.	500 (mm)			
Î		L2 122	000 (mm)			
-		L3 51.	000 (mm)			New
		S 10.	000 (mm)		_	
		CODE 11.0	06.0 (mm)		_	Find
_X		GW 1.1	80 (mm)	_		List
		N 110	000 (mm)	-		
	· · · · · · · · · · · · · · · · · · ·					Settings

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

### A Constant attributes

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

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

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

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

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

#### Variables + Attribute

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

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

Attribute text	Attribute text
Angle	cm
Area moment of inertia	dm
Density	ft
Elongation at break	in
Force	km
Integer	m
Length	mm
Limit torque	pt
Mass moment of inertia	yd
Mass per surface area	
Performance	
Pitch	
Pitch	
Pressure	
Relation	
Relation	
Section modulus	
Sound insulation	
Stressability	
Stretch limit	
Surface	
Surface area per length	
Temperature	
Tensile strength	
Test stress	
Thermal conductivity	
Thermal expansion coefficient	
Thermal transmittance	
Time	
Undefined	
Unitless number	
Mah	

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

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



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

#### Example:

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

Variable	Value	Unit	Variables + A	Attributes				
P P	110.000	(mm)	Variable	Data type	Category	Unit	Attri	
D3	110.000	(mm)	D	Double	Undefined	(mm)	D	[]
<b>[</b> ]	126.500	(mm)	D3 L1	Double Double	Undefined Undefined	(mm) (mm)	D3	[] []
L2	122.000	(mm)	L2	Double	Undefined	(mm)		[]
L3	51.000	(mm)	S	Double	Undefined	(mm)	WA	[]
s s	10.000	(mm)	- CODE GW	String Double	Undefined	(mm)	BE	[]
	11.006.0	(mm)	N N2	Double	Undefined	(mm)	NE	[]
GW	1.180	(mm)		Double	Ondenned	(mm)	INE	[]
N	110.000	(mm)						

Now change the assignment as shown below:

/ariable	Data tura	Catagory	Ulait	Attri
Variable		JCategory	JUnit	(Attri)
D	Double	Length	in	D []
D3	Double	Undefined	(mm)	D3 []
L1	Double	Undefined	(mm)	[]
L2	Double	Undefined	(mm)	[]
L3	Double	Undefined	(mm)	[]

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



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

Variable	Value	Unit	
P	110.000	in	
🔲 🛛 З	110.000	(mm)	Ì —

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

Attribute text	
cm	
dm	
ft	
in	
km	
m	
mm	
pt	
yd	

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

Variable	Value	Unit
🗖 🗖	4.331	in
D3	110.000	(mm)

# **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 API, you should ensure that the HiCAD version used is at least version 2502.5 or 2601.1 or newer. If you are unsure whether you are using corresponding adaptations, please talk to your administrator or contact the ISD in case of doubt.

# Service Pack 2 2024 (V 2902)

# API extension in 3-D

The HiCAD API now also supports the following 3-D functions:

- Dependent part ISD.CAD.Creators.DependentNodeCreator
- Deform ISD.CAD.Modifiers.Deform

# Develop beam

To develop curved steel engineering beams, the HiCAD API now supports the **Develop beam** function (Steel Engineering > Pull-down menu: Tools) with the following command:

ISD.CAD.Steel.DevelopBeam

# API for various View functions

HiCAD now offers various functions for creating views via the API:

- Detail view, Cuboid/Sphere
   : ViewCreator.CreateDetailView mit DetailViewSphereParams bzw.
   DetailViewCuboidParams
- Create new cut-out : View.CreateCutout
- Activate exploded view

# Order of drawing sheets

To change the order of drawing sheets, the HiCAD API provides the command

DrawingSheet.MoveTo

Free milling for sheet development

The HiCAD API now supports the processing of sheet developments with the **Free milling** Sheet Metal> PullDown-Menü: Extras). For this there is the class:



ISD.CAD.SheetMetal.FreeMilling
## Major Release 2024 (V 2900)

## Cam joints

The HiCAD API now also supports the Cam joint (3-D Standard > Standard Processings > Bore > Cams

- ). For this there is the new class:
- ISD.CAD.Modifiers.Cam/CamProcessing)

### Annotation tags

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

LabelCreation.OptimizeArrangement

## **Executing UI tasks**

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

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

## Event after drawing derivation

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

ISD.CAD.Steel.Drawings.WorkshopDrawings.SheetCreated

#### Form and positional tolerances

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

ISD.CAD.Dimensioning.FormPosTolerance.Create()

## Accessing fixed view points

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

View.FixPoint

# Interfaces

## Service Pack 2 2024 (V 2902)

## FlexiCAD XML

With the update to Version 2024 Service Pack 2, HiCAD supports the import of the XML interface by Flexi Jet. Files from Version 1.3 can be imported. All existing spatial element types are imported and assigned to the various levels in the part structure. The HiCAD part structure is based on the structure in the XML file. The

Hicad											
	1	> Lo	cal Dis	k (C:)	) > Hicae	) > sys >				~ 💈	QuickSearch (CT
	• • •	PlantPa	rts	^	Name	*	Size		Type	Chang	
		olo			l bomte	mplates			Folder	21, ^	
					Caden	as			Folder	21,	
		amu ha	4		CADfix				Folder	21,	
		omu_ne	-		CAM_I	nterface			Folder	21,	
	• • F	oneu			Ref DxfExp	ort			Folder	21,	
	• 🔍 F	rintdat	t		env_m	ap_panoramas			Folder	21,	
	۲ 🔍 F	unktwa	olken		lefilter				Folder	21,	
	۶ 👢 F	RefTeile	1		PDF3D				Folder	21,	
	• 👢 F	RefTeile	2		Sym				Folder	21,	
	• • F	Remote	Suppo		#PMO	D.DAT		1 KB	DAT File	11,	
		anhain			#SR.DA	AI .		1 KB	DAT File	21,	
		cpricip						1 KB	Adobe Acro.	11/	
		esourc	es					1 KD	Adobe Acro.	11/	
	• • •	Script			2d3dre	nortmanager D		1 KB	DAT File	11,	
	• • •	oplash			2d3dw	eldrenortmana	•	1 KB	DAT File	11,	
	) 👢 s	truntim	ne					1 KB	xmlfile	11.	
	) 👢 s	sys		~	2dstal	dichtung01 co		1 KB	xmlfile	11. ~	
<			>		<					>	
File	name :									All files	
1166 Item	IS									All files	
				-						STEP files	s (*.stp;*.step)
										STL files	(*.stl)
										IGES files	; (*.igs;*.iges)
										VDAFS fi	les (*.vda)
										CATIA file	es (*.CATPart;*.CATProdu
										ACIS files	; (*.SAT)
										Parasolid	files (*.x_t, *.xmt_txt, *.x
										Creo Para	ametric files (*.asm *.g *.
										NX files (	*.prt)
										SOLIDWO	ORKS files (*.sldasm, *.sld
										Inventor	files (*.ipt, *.iam)
										3-D DXF	DWG files (*.dxf, *.dwg)
										Solid Edd	ge files (*.par, *.asm, *.ps
										MicroSta	tion files (*.dgn)
										JT files (*	.jt)
										PLM XM	files (*.plmxml)
										IFC files (	*.ifc. *.ifczip)
											1

The FlexiCAD XML interface will be available with Patch 2902.1 of HiCAD 2024.

### IFC

#### IFC default settings for import

The default settings for importing an IFC file have been changed: When importing, features are no longer created automatically by default. This also increases performance.

If you want to create features, you must activate the Create feature checkbox.

#### Select IFC version when exporting

The dialogue window for the 3-D export of an IFC file has been adapted. Before saving, you can now choose between two IFC versions from the **Schema version** drop-down menu: IFC2X3 and IFC4.

The **Model view definition** has also been added. With IFC4, you can choose between **Reference View** and **Design Transfer View**, while **Coordination View** is set for IFC2X3.

#### NCX

#### **Compensation mode**

Under General in the NCX export dialogue, you can set a Compensation mode for milling tool paths:

- On contour
- With offset



#### Blind hole processing

Blind holes can be executed as a hole with one depth (From one side) or as a through hole (From both sides).



#### Thread description

Similar to the DSTV-NC export, you can also specify for the NCX interface whether threads should be written to the file as such or completely suppressed, or alternatively output as core drills.

Vrite	-	
Vrite		
Core drill		
ore ann Io not write		

#### Calculate cutting length of notched beams

In order to calculate the cutting length of notched beams and output it in the NCX file, the new option **The**oretically extend even with only one cutting plane + straight saw cut (output cutting length) has been added in the export dialogue under Cutting planes.

#### (Cutting plane)

- O Lengthen one of two cutting planes to theoretical intersection point + create saw cut
- (a) Theoretically extend even with only one cutting plane + straight saw cut (output cutting length)
- Two saw cuts for two cutting planes
- Directly indicate cut angles for flat steels

#### Flow drillings

Flow drillings can be inserted in HiCAD via **3-D Standard > Standard Processings > Bore/Thr.** These are also output accordingly by the NCX export.

## 2-D DXF/DWG: .dat file transfer confirmation

When importing and exporting2-D DXF/DWG files, you have the option of loading an older configuration file via the **Compatibility** tab of the Settings dialogue, for example to adopt settings from earlier HiCAD versions.

You will receive a confirmation message after successfully importing a corresponding file.



## KISSsoft 2024

HiCAD supports the new version of the KISSsoft plugin.

World Extras	Plugins		
Others	Plugins		
	۲ ۵ ۵ ۵ ۲	HELIOS Analyse mit ANSYS Analyse mit Patran Analyse mit FEMAP Analyse mit HyperMesh Analyse mit HyperMesh	
	K	ISSsoft	۲

#### Punch marks in ToPs-GEO export

With the update to Service Pack 2, you can optionally write punch marks to the file when exporting ToPs-GEO.

Punch marks are entered as a circle with a configurable diameter.

The setting is made in the file: \sys\CAMint\_HiCAD\_GEO.ini.

CAMint\_HiCAD\_GEO.ini: (excerpt)

CAMInt_HICAD_GEO.ini	× +
Datei Bearbeiten Ansicht	
<pre># Körnerpunkte # # # Körnerpunkte als Kreis ein 1 # Fester Körnerpunkt Kreisdum 0 # # #</pre>	tragen: nein (0), ja (1) rchmesser (0: unverändert aus Konstruktion übernehmen)

The file is not overwritten by update installations in the \sys\ directory, but is stored in the \templates\Default\sys\ directory.

To use the new setting, you must adjust the file in the \sys\ directory using the current one under \templates\Default\sys\. Of course, you can also overwrite the file in the \sys\ directory with the file from the \templates\Default\sys\ directory, but you should note that any adjustments in the file at \sys\CAMint\_HiCAD\_ GEO.ini that differ from the default settings will be overwritten and may have to be made again.

If there is an older version of the file in the \sys\ directory, no punch marks are output.

#### **ERPlus interface - Excel export**

When exporting the ERPlus BOMs, the Report Manager is now executed in the background. The output is then based on the Excel template with the relevant files sys\ERPlus.xlsx, sys\ERPlus.cs, sys\ERPlus.rm\_settings. These are specifically for ERPlus only and cannot be selected via the normal BOM configuration.

#### Version number for CATIA interface omitted

The name for the import and export of CATIA files has been adapted. The version number has been removed as the format versions for the interface have been changed and therefore not only V5 files can be imported and exported.

## Service Pack 1 2024 (V 2901)

## Update to CADfix 13

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

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

### Please note:

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

## Opening foreign formats without dialogue

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

Edit View Extras ISD				
) 🕄 🕴 🛿 🖓		aa   🞯 🖕		User
▲ Interfaces	*	Description	Value	Comment
Export		General settings for import		
General 3-D interfaces     Import		Import files directly during Drag&Drop		Dialog is suppressed during Drag&Drop, the default options (see below) are used.
		Default options for import		
3DVS		2-D DXF/DWG	ISD defaults 😭	]
Navisworks		ACIS	ISD defaults 😭	]
PRC STI		3-D DXF/DWG	ISD defaults 🟠	]
III U3D		CATIA V4	ISD defaults	]
		CATIA V5	ISD defaults 😭	]
ANSYS Workbench	=	MicroStation	ISD defaults 🟠	]
FEMAP		IFC	ISD defaults 🟠	]
GAMMA-RAY		/ IGES	ISD defaults	
HyperWorks		Inventor	ISD defaults	]
📰 LogiKal		л 🗸	ISD defaults	]
MCS Patran		Parasolid	ISD defaults	]
PDM	*	/ PLM XML	ISD defaults 🟠	]
	+	Creo Parametric	ISD defaults	

The default option will then be used.

## Combining the function for 3DFindit

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



## 3-D Import: Move to origin

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

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

P HiCAD					×
● ● ● ● ★ > Local Disk (C;) > HiCAD	> Szenen				✓ QuickSearch (CTRL+F)      ✓
<ul> <li>Quick access</li> <li>szenen</li> <li>HiCAD Drawings</li> <li>This PC (DEDTM066)</li> <li>Vetwork</li> </ul>	Name Cuboid_2.stp Cuboid_3.stp DRAWING1.STP	Size 15 KB 15 KB 7 KB	Type STP File STP File STP File	Changed on 04/12/2023 10:51 04/12/2023 10:51 04/12/2023 10:51	STEP Convert FFS into analytical surfaces Import layers AutoOptimise Repair Tool Move to origin igin if too far away from it
File name:					STEP files (*.stp;*.step)
5 items					Open Cancel

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

## DSTV-NC

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

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

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

DSTV-NC interface	X
- Output for	- File name
O 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
📝 Beam 🔰 Plate Extended	Parts: V Beams+Profiles V Plates V Contact surfaces
Calculate punch marks automatically (not for SM parts)	📝 Beam 🔰 Plate Extended
Display punch marks as points	Text: Item number
Edge distance: 10 mm	Font size: 10
- Powder marking line output	- Others
🝠 Beam 🐚 Plate Extended	Order number: HiCAD attribute
Destination: Write into block PU (powder mark ▼	Drawing number: HiCAD attribute
Contours, entire length of contact edge	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) ▼
O 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: Core drill
Side marking (not for SM parts) Length: 3	Countersink: Core drill
	Outer and inner contour:
	Maximum diameter: 200000 mm
	Max. length of approximation edges: 300 mm
	Write value for radius 0
	Expand outer contour values to 5 columns
	OK Cancel

## Major Release 2024 (V 2900)

### Access to 3Dfindit

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

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

#### 3Dfindit (Basis)

The free version offers you access to the available 3D catalogues that have a download contract with CADENAS (as of 06/2023, there are approx. 900 manufacturers).

#### 3Dfindit

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



No. 3 Sector 2015	dit							_		×
6	5567	Browse	3D CAD	& BIM ca	atalogs					Q
Filter	3D Shap	be 2D Sk	etch (	(?) Color	Grid	(Tatalog	→← gs Compare		EN	8
G	= > c	Pedrotti	>	> 🍪 e	BU-1BG -	Headed g	uide bush seli	flubrificating		
EBU-	1BG- 20 x	22								
Q 5	Search in tat	ole	$\langle \rangle$	К 🖸 Ма	ain variable	s 🗹 Seco	ndary variables			
	TYP	D1 (mm)	D2 [mm]	D3 [mm]	K [mm]	L [mm]	Execution ESECUZIONE	Material MATERIALE		
3 4	EBU-1BG EBU-1BG	16 18	22 26	26 30	5	22 22	Grinded Grinded	2.0598 / Grafite 200HB 2.0598 / Grafite 200HB		
5 🛃	EBU-1BG	20	28	32	6	22 🖌	Grinded	2.0598 / Grafite 200HB		
6	EBU-1BG	25	34	38	7	26	Grinded	2.0598 / Grafite 200HB		
0	EBU-IBG	32	40	45	-	30	Cripdod	2.0598 / Grafite 200HB		
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#### Registration is required for access.

The previous functions

- parts4cad (Basis),
- parts4cad und
- bimcatalogs

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

## New import formats: MicroStation and Solid Edge

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

MicroStation:

Parts and assemblies in the (3-D) geometry format .dgn. DGN Versions 7 to 8 are supported.

Solid Edge:

Parts and assemblies in the (3-D) geometry formats .par, asm and .psm Solid Edge versions V18 (2006) to 2022 are supported.

## IFC interface: Basic settings

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



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

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

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

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

eral DXF layer assignment Line types it of length Take from file As in drawing cort dimensions as Geometry nerate parts according to Block structure
it of length Take from file As in drawing Coort dimensions as Geometry nerate parts according to Block structure
Take from file   As in drawing   oort dimensions as   Geometry   nerate parts according to   Block structure
As in drawing  Cort dimensions as Geometry Therate parts according to Block structure
As in drawing
Geometry
Geometry   rerate parts according to Block structure
Block structure
Block structure

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

eneral DXF I	ayer assignm	ent Line types Font	s Colours	Others	Compatibility	
DXF colours						
DXF colour in	dex HICAD	colour				
2	4:	Red Red	8			
2	5:	Dark Blue				
3	1:	Dark Green	~			
4	3:	Blue	_			
2 C	2:	Blue				
6	6:	Orange				
7	0:	Black				
8	7:	Green				
9	8:	Red	_			
10	9:	Orange	•			
HiCAD colo	ur for geom	etric	•			
Hatching	s' <b>D</b>	ark Green	-			
natering	sD					
✓ Use the f	ollowing Hi	CAD colours for lines	with a fixe	d DXF lin	e thickness	
Line weight	HiCAD cold	bur				
Default	0:	Black				
0.01 - 0.14	7:	Green				
0.15 - 0.19	7:	Green				
0.20 - 0.29	4:	Red				
0.20 0.20	5:	Dark Blue				
0.50 - 0.59		Plack				
0.40 - 0.59	0:	DIdCK				

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

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

General		DXF layer assign	nment		DXF layer o	lefinition	
Line type assig	gnment	Fonts	Co	lour assignm	ent	Compatibility	у
$+$ $\times$							
HiCAD font	DXF style	name	Font fi	e	A	dditional width	correction
HiCAD - AI	MONOTX	т	monot	t.shx	1		
HiCAD - Al	STANDAR	D			1		
Arial	ARIAL		arial.ttf		1		
Arial Narrow	ARIAL Na	rrow	ArialN.t	tf	0	95	
Second and					<u> </u>		
ISOCPEUR	ISOCPEU	R	ISOCPE	UR.ttf		65	
ISOCPEUR	ISOCPEUI	2	ISOCPE	UR.ttf	0	65	

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

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

The configuration file keywords that are thus omitted include:

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

## 🕕 Please note:

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

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

# **Sheet Metal**

## Service Pack 2 2024 (V 2902)

## Cross-breaks in developments

In sheet developments, cross-break edges previously used the same setting as forming edges.

As of HiCAD SP2, you can set the line colour, line type and layer in the **Extended settings** on the **Edges** and lines tab for

- Cross-breaks with positive bend angle and
- Cross-breaks with negative bend angle.



## Manual changes to developments

Previously, automatically generated dimensions and annotations of a development that were changed manu-

ally were reset with the **Update development #** function. As of HiCAD SP2, all changes to the position and the properties (e.g. bracketing of the dimension figure) will beretained after synchronisation. This change

affects all texts generated by the **Develop sheet** function via the options **Dimensioning** and **Annotation** > **Bend line texts**.



(1) Dimensions and annotation of the bend lines are generated automatically,

(2) Chain dimension and bend angle are moved manually,

(3) After processing the sheet and updating the development, the changes are retained.

## Semi-finished product parameters

If you deactivated the semi-finished product in the sheet metal creation feature before HiCAD SP2, an error message was displayed for the parameters (bend radius) that were loaded from the semi-finished product. As of HiCAD SP2, the parameters are also deactivated if the semi-finished product is set to **No**. This behaviour affects the following functions::

•	New sheet along sketch	<b>نی</b>						
•	Create connecting sheet							
-	New sheet from surface							
	New sheet from solid	and						
	Pipes + Vessels							
	Feature	ах	F	eature				a ×
		ä	Γ	* * * *				13
		÷	-			Mahua		-
	Designation value	Ĥ	-	Designation	at along skotsh	Value		-
	(i) Sheet along sketch Sketch		-		et along sketch			
	Sami-finished product Ver			Ser	i-finished product	Var		
	Semi-finished product les	20.0	-	b Ser	i-finished product	RI 2 5 - AIQ	0	-
	Article number from catalogue VES	55,0	-	μ 🗖 Jen	le number from catalogue	VES	5,0	-
	Sheet thickness 25mm			She	et thickness	2.5mm		
	Mode Bend zon		-	More More	le	Bend zone		
	Milled side		-	Mill	ed side	Left		_
	Bend radius From sem	i-finished product (2.5 mm)	-	Ben	d radius	From semi-	finished product (2.5 mm)	
	Allowance method From sem	i-finished product (DIN6935)	-	Allo	wance method	From semi-	finished product (DIN6935	5
	Offset direction     Middle		-	Offs	et direction	Middle		-
	-		ſ	1			ן ר	
	Feature HCM Graphic Properties Variables			Feature HCM	Graphic Properties Varia	bles	, <u> </u>	
	Feature		4 X	Featur	e			4 X
	🍈 🖄 🏦 🍇 📠 🖪 👎 🚏 🖓		©.	. Li la	🛎 🛍 🏙 🔭 🏦	+2 ↓		¢
	Designation	Value		Desi	gnation		Value	
	▲ ● D 🗃 (1) Sheet along sketch			4	(1) Sheet along sketch			
	Sketch		_		Sketch			
	Semi-finished product	No		_	Semi-finished produced	uct	No	
	Sheet thickness	2 mm			Sheet thickness		2 mm	
	Mode	Bend zone		_	Mode Mode		Bend zone	
	Milled side	Left	_	2	Milled side		Left	
	Bend radius	From semi-finished product			Bend radius		1mm	
	Allowance method	From semi-finished product			Allowance method		DIN6935	
	Offset direction	Middle			Offset direction		Middle	
	Sheet width	100 mm			Sheet width		100 mm	
	Fitting direction	Double-sided			Fitting direction		Double-sided	
								الثاري

(1) Deactivating the semi-finished product before HiCAD SP2, (2) Deactivating the semi-finished product with HiCAD SP2

## Avoid double dimensions

If you activate **Outer contour**, the complete contour is dimensioned. If the measurement for the **Outer contour** matches the measurement for the **Dimension** during development, only the measurement for the **Dimension** will now be displayed.



(1) Evaluation of the dimension and outer contour parameters before SP2, (2) Evaluation from SP2 onwards

#### Deleting flanges and bend zones

Flanges and bend zones cannot be deleted from sheet metal parts with a feature log. After activating the flange or bend zone, the entire sheet metal part is deleted. From HiCAD SP2 without confirmation prompt. Deleting flanges and bend zones is only possible in the feature log.

#### Free milling for sheet developments

The HiCAD API now supports the processing of sheet developments with the **Free milling** For this purpose, there is the class



ISD.CAD.SheetMetal.FreeMilling

## Service Pack 1 2024 (V 2901)

**Revised Pipes + Vessels function** 

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

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

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

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

<ul> <li>Pipes + Vessels</li> <li>Insertion point</li> <li>Select insertion</li> <li>Type</li> <li>Type</li> <li>Sheet Metal O Solid</li> </ul>	point	
- Parameters		
Height (Z):	50 🔹	
<ul> <li>Sheet parameters</li> <li>✓ Use semi-finished product</li> <li>Aluminium sheet 2mm - Al99</li> </ul>	······································	
Thickness:	2	
Bend radius:	1 •	
Allowance method:	DIN6935 -	
Offset direction:	Second side 🔹	
Bend zone runout:	No special proc 🔻	
- Accuracy criterion		
Chord condition:	Length 👻	
Value:	15 🔹	
- General Referenced Article number: Aluminium sl	neet 2mm	

## Length change by selecting front edge

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



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



(1) Front edge, (2) Longitudinal edge

## Major Release 2024 (V 2900)

## Feature for Sheet Metal part creation

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

This affects the following functions in Sheet Metal:



## Free milling

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

After selecting the function, first identify the sheet metal part. For the sketch, you can either select an existing

sketch 🗐 and process it 🗹 if necessary or draw a new sketch 🖌 in the plane.

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

	Find	All
<ul> <li>✓ Factory stan</li> <li>✓ Composi</li> <li>✓ Stand</li> </ul>	dards te panels, groove form lard groove form	BZ           V 90°           V 135°           V 160°           V 90° acute           V 135° acute           V 135° acute           V 160° acute           Rectangle 10           Rectangle 15           Rectangle 14           V135° 3mm

Tools for the tool path



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

### Sheet development

Exclude annotations in sheet developments

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



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

#### Display of the catalogue symbols

The development of a sheet is always a 3-D part. To exclude catalogue symbols (bore patterns, moulding tools and punching tools) from the 3-D display and to display them as symbols, activate the respective checkbox 🗹

for the sheet development in the **Extended settings** > Tab: **Extended setting**.

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



(1) Parameters of the standard processing edges changed in the **Extended settings** of the development.

(2) Line properties taken from the catalogue.

Feature		φ×						
💼 💿 💩 📷 🖪 📲 📭 📲								
Designation	Value	•						
Bend line shortening								
Lengthen milling lines	No							
Punchings								
Planar representation for mouldings	No							
Planar symbol from catalogue	YES							
Keep line properties for catalogue symbols	No							
Planar representation for cross-breaks	No							

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

#### Tool numbers in sheet metal processing

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

Γ	10	M	DD STA	TUS	Designation	WZNR 1	NAME	SKETCH	TYP	PREVIEW	TOPSYMBOL	BOTTOMSYMBOL 2	WZNR_BOTTOM 2
	1	3		E	Beading 100/80/15/2	SI_100_80_15_2_0	Formwerkzeuge\BEADING.KRA	Formwerkzeuge\BEADING_Sketch.KRA	1	Formwerkzeuge\image\BEADING.bmp	Praegewerkzeuge\BEADING_TOP.F	A Praegewerkzeuge\BEADING_BOTTOM.FGA	SI_100_80_15_2_1
	2	1		E	Beading 20/20/5/2	SI_20_20_5_2_0	Formwerkzeuge\BEADING.KRA	Formwerkzeuge\BEADING_Sketch.KRA	1	Formwerkzeuge\image\BEADING.bmp	Praegewerkzeuge\BEADING_TOP.F	A Praegewerkzeuge\BEADING_BOTTOM.FGA	SI_20_20_5_2_1
	3	2		. 6	Beading 50/50/10/2	SI_50_50_10_2_0	Formwerkzeuge\BEADING.KRA	Formwerkzeuge\BEADING_Sketch.KRA	1	Formwerkzeuge\image\BEADING.bmp	Praegewerkzeuge\BEADING_TOP.Fr	A Praegewerkzeuge\BEADING_BOTTOM.FGA	SI_50_50_10_2_1

(1) Table in the catalogue with WZNR and TOPSYMBOL for the front side of the development(2) (2) WZNR\_BOTTOM and BOTTOMSYMBOL for the rear side of the development



(1) Sheet with beading

(2) Development of the front side with plane representation of the moulding tool (TOPSYMBOL) and tool number (WZNR)

(3) Development of the back side with plane representation of the moulding tool (BOTTOMSYMBOL) and tool number (WZNR\_BOTTOM)

Mitre with neighbouring sheets mode. To With the Corner/Mitre function you can now add a milling edge in the Mitre, with neighbours do this, you must activate the Sheet edge as milling edge option in the dialogue. Corner/Mitre - Mode - Edges / plane **%** Edge **%**/ Edge - Fitting parameters Joint: ΈF Groove form: V 90° 0.1 Clearance: • Projection (Factor): Bend zone adjustment: Linea - / Diameter: ✓ Lengthen Apply immediately OK Cancel Apply

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

## Apply length value

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



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



Figure 4.2. If this setting is active, the value or the total length is also available for the next edge after choosing Apply.

## Extension of the coating function

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

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

## Settings for 2-D DXF export

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

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

## Improved handling of crosshairs during bending simulations

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



Bores forming a circular pattern in a bend zone

## Transform + Clone in context menu

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

	<b>a</b>								
	Sheet flange								
	Options								
ď	Extended selection								
۲ř	Sheet metal main part								
	Visualisation								
	New								
	74 🛋 🕭 🚿 🖮 🛍 🖤 🌮								
	Transform								
	S - · · · · · · · · · · · · · · · · · ·								
	Others								
	Clone								
	S - T 12 17 18 1 17 17 J								
	Others								

## Attach flange function in context menu

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



Design variant SZ-20

## User-defined Steel Engineering plates in catalogue

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

# **Steel Engineering**

## Service Pack 2024 SP2 (V 2902)

### Civil Engineering - Part type catalogues

Due to many customer requests, the **Part type catalogue**, **3-D** function is once again available in the **Civil Engineering functions** docking window under **Civil Engineering**, **General > Civil Engineering - Part type catalogue**.

Insert new beam - modified and extended dialogue

The dialogue window of the Insert new beam 🥢 function has been slightly changed and extended.

- The Source are has been renamed to Cross-section and has been moved been moved to the top of the dialogue window, above the Fitting area.
- The option Adopt cross-section from reference profile has been moved to the Cross-section area and the symbol for Open/close configuration window for prototype beams has been changed for prototype beams.

💌 Beam 🛛 🗙	🛃 Beam 🛛 🗙
- Cross section	- Cross section
- Catalogue profile HEA 200	- Prototype beams
Previously, only the Centroid of the beam cross-section and the corresponding beam axis points could be selected as the reference point.



As of SP2, the **Middle** (=surface cross-section) of the bounding box, i.e. the smallest cuboid that completely encloses the beam, can also be used instead of the centroid. However, this only has an effect on asymmetrical beams and profiles, for example L-beams.



The functions Beam from sketch and Multi-part beam from sketch have been integrated into the dialogue.

- Cross section		() • I
- Sketch beam	d) -lt-h	
Select (multi-pai	rt) sketch	1
Select axis point	manually	70
Weight per length:	0	-
Commercial weight per length:	0	-
Surface area per length:	0	•
Material:		•
Designation:		•

The previous functions

Beam from sketch,

Beam from sketch, as sub-part, and

### Multi-part beam from sketch

have therefore been removed from the **Steel Engineering** Ribbon and moved to the **Steel Engineering** > **New** > **Beam** > ... pull-down menu, under Up to HiCAD 2022.

# Copying connections on API basis

Previously, only connections based on a design variant could be copied using the Copy connection/design

variant function. As of HiCAD 2024 SP2, this is also possible for connections that are realised as API variants. This affects the following connections:

- Cross-bracing (2601)
- Cross-bracing (2602)
- Cross-bracing (2603)
- Frame Corner (2203)
- Gusset Plate (2510)
- Stabilizing Pipe (2701)
- Stabilizing Pipe (2702)
- Purlin Joint (3204)
- Purlin Joint (3206)
- Bar Configurator (4001)

# **Railing Configurator**

### Post made of round steels

As of HiCAD 2024 SP2, profiles from the catalogue **Semi-finished products > Beams+Profiles > Round steel** can also be used as posts.

Post - Sub-structure	Post - Handrail	Handra	iil - Handrail	Skirting	oard - Skirting boar	d		
1) Walking line 2)	Post distribution	3) Post	4) Handrail	5) Infill	6) Skirting boa	rd		
All posts equal All posts equal	l: 0 •							
- Start post	Post	from standard	l beam	-	Round steel			;
Post:	Pipe	EN 10220 48.3	x2.6 - S235JRH		li fi	Find	•	
<ul> <li>Beam orientation</li> <li>Intermediate post(s)</li> </ul>					<ul> <li>Semi-finishe</li> <li>Beams+f</li> <li>Flat s</li> <li>Hollo</li> </ul>	ed products Profiles teel w profiles		RD (DIN 1013-1)
Variant: Post:	Post Pipe	from standard EN 10220 48.3	1 beam x2.6 - S235JRH		<ul> <li>Roun</li> <li>Steel</li> <li>Factory stan</li> </ul>	d steel pipes dards		DIN 175
✓ Beam orientation					,	$\sim$		DIN 50261
- Corner post/Transitio	on post		Previev				<del>O</del>	DIN 670
				ſ	R4 DIN 1013-1	8		OK A Cancel

### Straight cut on handrail

On the **Post - Handrail** tab, the **At lower edge of handrail** checkbox is also available for the **Trim pipes** variant. If the post is to be cut straight at the lower edge of the handrail, activate this checkbox.

- Post - Handrail			
Variant: Trim pipe	25	•	
At lower edge of handrai			
Width of obtuse end:	1 • i		
-	-		
(1)	(2)	(1)	(2)



#### Post - Sub-structure: Lateral connection with flat steels

On the **Post - Sub-structure** tab, there is another connection variant called **Lateral connection with flat steels** available. Here, the posts are attached to the side of the beam using connecting plates (sheet metal parts or flat steels)

### Post - Handrail - Console (round bar)

A further variant called **Console (round bar)** is available for connecting the handrail to the post. Here, the handrail is positioned in front of the wall console connection (own production).

Variant:	Vertical rods with cross	s-members 🔻
(1) Top:	<b>(i)</b>	) 150 🔻
(2) Bottom:		
✔ (1)+(2): Per	pendicular to handrail	
(3) Distance:		120 - 3
(4) Width:		
Rod rotation ang	jle:	
Trim to pos	t	
✓ Trim rods t	o cross-members	
Cross-members:		FI 30x6 - S235JR
Rods:	<b>v</b>	FI 20x 5 - S235JR

#### **Continuous infills**

As of SP2, infills can be configured for an entire segment independently of the posts. There is then one infill per segment. The components of the infill are summarised in an assembly called **Continuous infill**.

Railing Configurator							
Post - Sub-structure		Post - Handrail		Handra	il - Handrail	Skirting bo	ard - Skirting board
1) Walking line	2) Post	distribution	3) Pos	t	4) Handrail	5) Infill	6) Skirting board
O Infill, between posts	۲	nfill, continuous					

Two variants are available for continuous infills:

#### Segment infill through posts

Here, the knee rails run through the posts. The knee rails can be evenly distributed or have a certain distance to the handrail and finished floor.

Infill, contin	uous —					
/ariant:	Segment infill thro	ugh posts 🛛 🔻	]			
Knee rai	1	EN 10060- 16 - S	235IR		_	
Rotate 9	90°					3
Number	r of knee rails:		3	•		
Excess l	ength on first post (1):		100	•		
Excess l	ength at last post (2):		100	•	5	
Insertion	n depth in post:		10	•		
Distribu	ite even <mark>ly</mark>	<b>V</b>				
Clear di	stance to lower edge of	f handrail (3):	100	Ŧ		(4)
Clear di	stance to finished floor	(4):	100	~		
Clearan	ce for hole in nost (5)	<u> </u>	0	•		
	-					

#### Segment infill with spacers

In this variant, spacers are placed on the posts. The knee rails then pass through the spacers. They can be evenly distributed or have a certain distance to the handrail and finished floor.

L CII	p 0				
Infill, contir	nuous —				
Variant:	Segment infill wit	th spacers			
Distanc	ce piece	Holder 100x32x	18 - E155		
Switch	insertion direction				3
Knee ra	ail	EN 10060- 16 - 5	235JR		
Rotate	90°				
Numbe	er of knee rails:		3	i.	
Event	length on first post (1	).	100 •		
EXCESS			100 -	I	
Excess	length at last post (2)	:			4
Clear d	istance to lower edge	of handrail (3):	100 -		
Clear d	istance to finished flo	oor (4):	100 -		
Distribu	ute evenly				
	_				
			1		
				2	

# Staircase Configurator - Start/end platform as a step

From SP2, a new type of stair start and stair end is available on the **Calculation** tab of the Staircase Configurator, which can be used to install steps at a stair start/end instead of platforms.

Staircase C	Configurator					_	×
Calculation	Stringers	Platforms	Steps Assembling				
2					3 ⊻		
- Stair start				- Stair end	4		

If these options are selected, step variants can be selected for the stair start/end instead of a platform on the **Platforms** tab.

Matericase Confi	gurator	—	×
Calculation St	ringers Platforms Steps Assembling		 
Start platform:	Standardized grating step 👻		
Platforms:	Do not create		
End platform:	Grating platform		
	Standardized grating step		
- General para Dist. Start platf Distance end p	Free grating step Sheet metal step Sheet metal step w/o support profile		
- Start platforn	Wooden step		

Main assembly	Assembly	
4 1 % Staircase	Assembly	
👌 🌗 🍖 Accessories	Assembly	
A 1 % Segment	Assembly	
👂 🤚 🐏 Stringer	Assembly	
👂 🌗 🐏 Stringer	Assembly	
🜗 👭 P30-30-1000x305	Grating steps	
P30-30-1000x240	Grating steps	
P30-30-1000x240	Grating steps	
P30-30-1000x240	Grating steps	
🜗 👭 P30-30-1000x240	Grating steps	
P30-30-1000x240	Grating steps	
P30-30-1000x240	Grating steps	A A A A A A A A A A A A A A A A A A A
🔱 🖊 P30-30-1000x240	Grating steps	
🜗 🖊 P30-30-1000x240	Grating steps	
P30-30-1000x240	Grating steps	
P30-30-1000x240	Grating steps	
P30-30-1000x240	Grating steps	
🜗 👭 P30-30-1000x240	Grating steps	
P30-30-1000x240	Grating steps	
P30-30-1000x305	Grating steps	

## Redesigned Frame corner dialogue (2203)

The dialogue for the Frame corner (2203) connection has been redesigned in SP2.

The selection of the two I-beams to be connected now takes place directly in the dialogue window. The selection can also be corrected in the dialogue window. Previously, the beams had to be selected before the dialogue window was displayed.

eams							
st beam:	Select beam/profile		Usage: Girder	(beam) asse	embly		
nd beam:	Select beam/profile		Usage: Colun	n <mark>n assembl</mark> y	1		
tes Semi-fin	ished products Stiffeners	Bore grid	Galvanization	Filler plate	Weld seams	Reinforcement plate	

The semi-finished products of the individual components of the frame corner can be selected centrally on the new Semi-finished products tab. This also applies to the bolting on the front plate and the tension plate. The previous Bolting tab is therefore no longer available from SP2.

Plates	Semi-finished prod	ucts	Stiffeners	Bore grid	Galvaniz	ation	Filler plate	Weld seams	Reinforcement plate	
Plates - Plat Front Tensic Haun Haun Haun Web Ribs:	Semi-finished prod es plate: on plate / Top plate: ch: ched plate 2: ched plate 3: ched flange: reinforcement:	BI 10 BI 11 BI 11 BI 11 BI 11 BI 12 BI 12	Stiffeners 0 - S235JRG 4 - S235JRG 4 - S235JRG 4 - S235JRG 4 - S235JRG	Bore grid	Galvaniz	- Bol Frontin DIN ♥ F Assi ● ♥ Tensi DIN ♥ F Assi	Filler plate tings t plate: EN ISO 4014 it Invert gnment: Loose part Assembly (20 ion plate: EN ISO 4014 it Invert gnment: EN ISO 4014	Weld seams 4-M16-5.6 / M nd beam) 4-M16-5.6 / M	Reinforcement plate	
Haun Haun Web Ribs: Reinfi	ched plate 3: ched flange: reinforcement: prcement plate:	BI 14 BI 14 BI 14 BI 14 BI 8	0 - S235JRG 4 - S235JRG 4 - S235JRG 4 - S235JRG - S235JRG2	;2 ;2 ;2 ;2 ;2 ;2		⊂ Tensi DIN ▼ F Assi ⊙	Assembly (2) on plate: EN ISO 4014 iit Invert gnment: Loose part Assembly (1)	nd beam) <mark>4-M16-5.6 / M</mark> st beam)	<mark>16 (Ø</mark> ) 🔢	
- Usa Stiffe	ge ner: Stiffener verwrite usage			F						

As of SP2, a separate tab called **Reinforcement plates** is now available for this type of plates. The corner processing can be selected here if desired. The chamfers can either be chamfered (45°) or filleted. In addition, the reinforcement plates can be connected both horizontally or vertically. The vertical connection can also be made row by row.



By activating the corresponding checkboxes, you can individually define which rows/columns are to be linked, e.g.



# End plate (2102) - Base position

When connecting **End plates (2102)** to a beam, the position of the base can now be influenced. The **Shorten beam** checkbox has been added to the **Base** tab for this purpose. If the checkbox is active, the beam will be shortened by the height of the base when the plate is inserted.



(1) Shortening of the beam by the plate thickness including filler plates, (2) Beam additionally shortened by base height

# Strap joint (2310) - Chamfers of straps

With the **Strap joint (2310)**, it was previously only possible to fillet the corners of the strap. As of SP2, chamfering is also supported.

Strap joint (2310)	;
Strap Boltings Weld seams Stiffener Galvanization	
- Connection type O Without stiffener  With stiffener  Notch	- Semi-finished product: BI 10 (S235JR)
- Geometry Vertical:	- Bore grid (Positioning) (6) Last bore row - Strap end ▼ 50 ▼
Distances to fillet starts, 1st beam 🔹	
(1) Height:     180 ▼       (2) Top:     20 ▼       (3) Bottom:     20 ▼	Edit: Double-sided
Refer distances to notches	
Horizontal: (4) Distance: (5) Distance from flange	
(7) Radius: 10 ▼ - Bore grid Number X: 2 ▼ Number Y: 2 ▼ 50 ▼ 50 ▼	
	Preview OK Cancel

# Service Pack 2024 SP1 (V 2901)

# Development attributes for Steel Engineering plates

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

## **Railing Configurator**

#### Faster start

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

#### Distance between skirting board and post

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

Variant:	Skirting board, flat steel	•				
- Assemblin	ng				(2)	
Skirting	boards, All					
O Skirting	boards, Platforms					
Skirting board	d: FI 50x8 - S235JR		]			
(1)Gap at bot	tom:	20 🔻				
(2)Distance to	o post:	0 •				
- Fixing of	skirting board		1 1		(1)	
Internal	E.			2		
O Middle			1			
O Externa	I					
Trim ski	rting boards to post	-		_	2000000 41	
Width of obt	use end:	1 - 1				

#### Distance between handrail and cross-member

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

/ariant: Ver	ical rods with cro	ss-members	•		
1) Top:	(i	150 -			
2) Bottom:		100 🔻		1	3
🚺 (1)+(2): Perpe	ndicular to handra	il			
3) Distance:		120 👻		3	
4) Width:		1 -			
Rod rotation angle:		0 🗸		2	-
Trim to post					
✓ Trim rods to c	oss-members				
Cross-members:		FI 30x6 - S2	35JR		
Rods:	V	FI 20x5 - S2	35JR		

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

The following illustration shows the difference.



## Feature for mitre cuts

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

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



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

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

# Civil Engineering - Part type catalogue, 3-D

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

# Galvanization holes on beams

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

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

## **Clearance for stiffeners**

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

This applies to the following design variants:

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

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

Create	Usage (Stiffener):	Stiffener		III	
- Semi-finished	product		- Туре		
BI 10 ( S235JR	)	II	• Full stiffener	) Partial stiffener	Automatic
- Dimensions -					1
(1) Width: 🔽	Auto	100 -			
(2) Height:	Flush to 2nd plate	50 👻			1
- Internal corne	er			1	
Туре:	Int. fillet	•			
(3) Chamfer/Ra	dius: 🔽 Auto 🗍	10 -		6	5
- External share	-far			0	(2)
Without (4)	Fx: 30	*			-3
Fx, Fy (5)	Fy: 30	*			
full stiffener is round	ed to whole millimet	res. The clearanc	e is adjusted in the r	rocess	
					*-(7)
- Galvanization	(0) Distances	25	- Others		
	(a) Distance:	25 -	(7) Clearance:		1
Bottom	(10) Distance:	10	Chamfer for zi	nc plating	10
	(10) Diameter.	25			
Create	ener				
Somi-finished	araducti PL10 (53	251D \			
semi-imshed		.557()			

# Major Release 2024 (V 2900)

### Insert new beam

The function Insert new beam 🥒 has been extended:

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

#### Via catalogue...

With this option you insert beams from the allowed catalogues mentioned above at **Semi-finished** products or Factory standards.

Prototype beam...

With this option you can insert configurable prototype beams.

Beam and profiles from the catalogues Factory standards > Series and Factory standards > Factory beams can now also be inserted.

💌 Beam		×
- Fitting		
Guideline	<b>8</b> 4	Ν
Delete sketch af	ter creation	Ŧ±
- Source - Prototype beams I-beam / [S235JR0	 52]	\$
- Reference point -		
	<b>↓</b>	
Y	<b>⊕ ⊕</b>	
	¢ → →	
- General		]
Referenced	✓ BOM-relevant	
	Apply immedia	tely 🗌
	OK Cancel Ap	ply

### Connections

New connection - Gusset plate (2510)

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

Civil Engineering functions $\qquad = \times$						
, st , st <b>→</b>						
▲ Steel Engineering						
b General						
Connections						
Individual beam/profile						
Front side to front side						
Front side to web/flange side						
Web/Flange to Web/Flange						
Cross-bracing (2601)						
Cross-bracing (2602)						
Cross-bracing (2603)						
Gusset plate ( 2510 )						
K-bracing, welded						
K-bracing, bolted						
Stairs+Railings						
Size: (Large icons)						
Panoramas Exploded vi Function se Civil Engine						

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

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

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

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

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



#### Gusset plate connection with three steel pipes



Gusset plate connection with two steel pipes and one hollow profile

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



Connection of a flat steel profile

### Frame corners - placement points for stiffeners

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

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

The Stiffeners tab has been extended accordingly.

Zolumn connection, Frame corner (2203)	×
Plates Stiffeners Boltings Bore grid Galvanization Weld seams Filler plate	
Create     Image: finished and duct     Placement point     Placement point	
- Semi-finished product	
- Dimensions (1) Width:	
- Internal corner Type: Chamfered (3) Chamfer/Radius: Auto 10	,
- External chamfer Without (4) Fx: 10 - Others	
Fx, Fy     (5) Fy:     10     (7) Clearance:     (1) Clearance:       Image: Strain Stra	
Usage (Stiffener): Stiffener	
S ↔ Preview OK Cancel	



This extension also applies to Frame corner 2204.

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

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

Type: Two	-piece, insert horizon	tal 🔻		
(1) Distance:	0	•		
(2) Distance:	5	•		 
Hole clearance:	2	•		
Fillet corners	Radius: 2	*		
Manage			a a a a a a a a a a a a a a a a a a a	A
All	Quan	tity: 0 🔻		



Girder connection, filler plate not pre-planned

Cross-bracing (2601) with turnbuckle and blade screws

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



aams+Profiles Semi-finished products Fixing Connecting points Weld seams Weld		-	•	1 tot diagonal 2 2 of diagonal
Diagonals  Ist diagonal Pront flange Rear flange  Front flange Rear flange  Gusset plate: Pentagonal plate  Position of bracing member to gusset plate: Front  Turnbuckle  Turnbuckle  Ist diagonal PH  Create (3) Distance (Direct) 500  (4) 25 Boltings: Create Invert SP M20 DIN 1480 - St  Usage (Diagonals) Cross-bracing	eams+Profiles	Semi-finished products	xing Connecting points	Weld seams V Ist diagonal V 2nd diago
Ist diagonal   Ist diagonal   Image   Front flange   Image   Image <	– Diagonals –			
Front flange Rear flange   I   Distance from centre axis :   Qusset plate:   Pentagonal plate   Position of bracing member to gusset plate:   Pront     Ist diagonal   It diagonal   It diagonal     It diagonal </td <td>1st diagonal</td> <td>2nd diagonal</td> <td>[=]=]</td> <td>(2)</td>	1st diagonal	2nd diagonal	[=]=]	(2)
Front flange Rear flange   (1) Distance from centre axis :   Gusset plate:   Pentagonal plate •   Position of bracing member to gusset plate:   Front   •		0.0		
(1) Distance from centre axis : Gusset plate: Position of bracing member to gusset plate: Turnbuckle 1 Turnbuckle 1 Turnbuckle 1 Turnbuckle 1 Turnbuckle 1 1 1 1 1 1 1 1 1 1 1 1 1	Front flang     (1)	e 🔘 Rear flange		
Gusset plate: Position of bracing member to gusset plate: Turnbuckle Turnbuckle 1 t diagonal 2nd diagonal Creates (3) Distance (Direct) • 500 • (4) 25 • Boltings: Create Invert SP M20 DIN 1480 - St III	(I) Distance f	rom centre axis :	• 0 •	* *
Position of bracing member to gusset plate: Front  Turnbuckle  Ist diagonal 2nd diagonal  Creats  (3) Distance (Direct)  (4) 25  Boltings: Create Invert  SP M20 DIN 1480 - St  Usage (Diagonals)  Cross-bracing	Gusset plate:		Pentagonal plate 🔻	
- Turnbuckle Ist diagonal 2nd diagonal III VCreate (3) Distance (Direct)	Position of bra	cing member to gusset plat	e: Front 🔻	
Turnbuckle   1st diagonal   Ist diagonal   Ist diagonal     Is				
Turnbuckle 1st diagonal 2nd diagonal Create (3) Distance (Direct) • 500 • (4) 25 • Boltings: Create Invert SP M20 DIN 1480 - St Usage (Diagonals) Cross-bracing				
Turnbuckle   1st diagonal   Ist diagonal     Ist diagonal				
Ist diagonal   Ist diagonal   Ist diagonal     Ist diagonal				
1st diagonal     (3) Distance (Direct)     (4) 25   Boltings:   (2) Create   Invert   SP M20 DIN 1480 - St     (3) Distance (Direct)     (4) 25   (5) Create     (5) M20 DIN 1480 - St     (5) Cross-bracing	- Turnbuckle -			
Image: Create     (3) Distance (Direct)     (4) 25   Boltings: Image: Create     Invert     SP M20 DIN 1480 - St     Image: Usage (Diagonals)     Cross-bracing	1st diagonal	2nd diagonal	= =	
(3) Distance (Direct) • 500 • (4) 25 • Boltings: Create Invert SP M20 DIN 1480 - St III - Usage (Diagonals) Cross-bracing	✓ Create			
(3) Distance (Direct)				
(4) 25  Boltings: Create Invert SP M20 DIN 1480 - St Usage (Diagonals) Cross-bracing	(3) Distance	(Direct)	• 500 •	
Boltings: Create Invert SP M20 DIN 1480 - St Usage (Diagonals) Cross-bracing	(4) 25 •	·		
SP M20 DIN 1480 - St	Boltings: 🔽	Create 🗌 Invert		4
- Usage (Diagonals) Cross-bracing	SP M20 DIN	1480 - St		3
- Usage (Diagonals) Cross-bracing				
- Usage (Diagonals) Cross-bracing				
- Usage (Diagonals)				
Cross-bracing	- Usage (Diag	onals)		
	Cross-bracing	1		
	Cross-bracing	1		
	7			

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

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

Beams+Profiles Semi-finished products	Fixing	Conne					
- Bracing members							
FI 100x10 - S235JRG2	FI 100x10 - S235JRG2						
- Gusset plates		_					
BI 15 - S235JRG2							
- Boltings ( Gusset plates )							
DIN EN ISO 4014-M10-5.6 / M10 (Ø 11)	DIN EN ISO 4014-M10-5.6 / M10 (Ø 11)						
– Boltings ( Turnbuckle )							
DIN EN ISO 4017-M12-5.6 / M12 (Ø 13)		IF					
- Boltings ( Assignment )							
O Loose parts							

#### Cross-bracing without gusset plates and connecting plates

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

#### Point (as intersection point of axes)

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

Cross-bracing (2601)	×
Beams+Profiles Semi-finished products Fixing Connectin	ng points Weld seams 🗹 1st diagonal 🗹 2nd diagonal
A B C D HH	
- General Distance to reference: 1st point	
- Process corner None Chamfer/Radius: 10 -	- Chamfers + Fillets Gusset plates: - Value: 10 -
- Bore grid Number x: 2 ▼ Number y: 2 ▼ 50 ▼ 50 ▼ (8) Bore offset: 0 ▼	✓ Create     10:     100 ▼       Mount to flange     11:     50 ▼
	- Boltings ✓ Fit  Invert
	Connecting point without plates
	Preview OK 🚺 Cancel



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

#### Base plate + Anchor plate (2101)

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



#### Connections- Galvanization holes on beam

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

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

lates Fixing	Bore grid	Galvanization	Weld seams	Filler plate	Shear connector	Head studs	Ribs	
idees   mang	bore grid		There seems	riner place	onear connector	Theorem Status	1005	
On beam Ba	ase plate							
Processing	Holes			-				
🗸 Тор					1			
Bottom					<u>k</u>		-	
Radius:	20	*						
Diameter	20	-						
Diameter.	20				6			
(1) Distance X:	10	•		TU.	<u> </u>	4		
(2) Distance Y:	10	•						
					<u>p</u>			
							-	



# Civil Engineering - Part type catalogue 3-D

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

Metal Engineering / Facade Engineering	×			
₩II Find	- All -			
<ul> <li>Factory standards</li> <li>Part types</li> <li>Assemblies</li> <li>Sheet Metal construction</li> <li>Solid construction</li> <li>Mechanical construction</li> <li>Metal Engineering / Facade Engineering</li> <li>Steel Engineering</li> </ul>	Designation       TEXT         Insulation 3-D, soft       Creates from an existing sketch a 3-D			
Insulation 3-D, soft	OK Cancel			

In a first step, the entries under

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

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

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

#### An example:

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



### Parameter configuration

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

Edit View Extras ISD			
5 🖉   👫 12 13 👫	AA   💿 🖕		User
HICAD	Description	Value	Comment
Grant Configuration (Base configuration)     Graving     Automatic drawing derivation	Automatically update calculated attributes when loading	No	Updating of automatically calculated attributes whose calculation is set to 'Always'. The attributes are calculated at the latest during itemisation or manual updating
Modelling	Calculate weight	Manually / When itemising *	Assigns value to attribute \$01
Surface/Edge functions	Approximation accuracy for weight calculation	50	0 (imprecise but fast) - 100 (precise but slower)
Grid	Calculate surface area	Manually / When itemising ~	Assigns value to attribute §10
Miscellaneous	Calculate coated surface	Manually / When itemising ~	Assigns value to attribute §SC
Part creation     Part properties	Transfer coated surface to	Coated surface ~	
Change of part structure	Calculate volume	Manually / When itemising ×	Assigns value to attribute §20
Weld seams	Calculate quantity in assembly	Manually / When itemising ~	Assigns value to attribute %13
<ul> <li>Image: Steel Engineering</li> <li>Image: Image: Image:</li></ul>	Calculate total quantity		Validly itemized parts are counted. Switching off the calculation deletes total number.
Profile Installation     Image: Plant Engineering	Material	Material from catalogue	
Sheet Metal	Dimension calculation		
Assembling simulation     Analysis     Interfaces	Calculate dimensions	Manually / When itemising *	Determine the dimension for parts and assemblies from the geometry. Assigns the attribute Length (503), Width (502) and Height (504). For cert part types, dimension attributes are set even if the calculation from geometry is switched off here.
<ul> <li>E Compatibility</li> </ul>	Sort dimensions		When calculating dimensions, write the dimensions in descending orde size on the attributes Length (\$03), Width (\$02) and Height (\$04).
System settings Configurations	Calculate only length for beams		Restriction only applies if dimension calculation is activated. If this settir has been chosen, the height and width are not used for beams and any values transferred from the catalogue will be retained.
	Sheet Metal		
-	Calculate area from development contour	Manually / When itemising ~	Assigns value to attribute \$SOC. Calculated from the surface area of the development outline. Inner exclusions and processings are ignored
_	Calculate rectangular area of development	Manually / When itemising Y	Assigns value to attribute §S2D. Calculated from length * width of development
_	Calculate weight from rectangular area of development	Manually / When itemising Y	Assigns value to attribute \$CW
	Calculate dimensions of development	Manually / When itemising	Assigns value to attribute §L2D, §B2D, §T2D

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

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

### Feature when inserting Rectangular plates

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

## **Railing Configurator**

### Post connection top, with galvanising hole

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

Walking line	2) Post	t distribution	3) Post	4) Handrail	5) Infill	6) Skirting board
ost - Sub-structu	re	Post - Handrail	Handrail	- Handrail	Skirting board	d - Skirting board
Start post, indi	vidual vidual					
Start, interme	diate, end	posts				
Variant:	ost conne	ection, top		•		
• 0 •	<ul> <li>(1)</li> <li>(2)</li> <li>(3)</li> <li>(4)</li> <li>(8)</li> </ul>	20     ▼       40     ▼       75     ▼       40     ▼       80     ▼		☑ Wit Diamet	h galvanising ho er: 20	ole •
Fillet radius of	<ul> <li>(1)</li> <li>(2)</li> <li>(3)</li> <li>(4)</li> <li>(8)</li> <li>corners:</li> </ul>	20 ▼ 40 ▼ 75 ▼ 40 ▼ 80 ▼ 0 ▼ 90° rotated		<b>√</b> Wit Diamet	h galvanising ho	ole •



(1) without, (2) with galvanising hole

### Infill with bottom cross-member - Insert filling rods

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

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

ost - Sub-structure	Post -	Handrail	Handrai	l - Handrail	Skirting boa	rd - Skirting board	
Walking line	2) Post distrib	oution 3)	Post	4) Handrail	5) Infill	6) Skirting board	
Infill —							_
Varianti	utical cade with	hottom cross .					
vananc.	rtical rous with	bottom cross-r	ner •				
- Rods					_		٦
Rods:		FI 20x5 - S23	5JR	11			
(2) Distance:		120 -					
Rod rotation an	ngle:	0 •				2	
Connection Ha	andrail	Insert in han	drail	•			
connection, ne						1	
		(5) Depth:	10	<b>_</b>		• <u>•</u>	1
		(6) Clearances	1	-			
		(7) Fillet radiu	IS: 1	-			
		(8) Notch:	0	• (i)			
		(9) Notch rad	ius: 0	•			
Connection, cro	oss-member:	Insert in cross-member					
		(5) Denth	0	•			
		(6) Clearance		-			
		(7) Fillet and in					
		(/) Fillet radiu	IS; U	<u> </u>			
		(8) Notch:	0	<b>-</b> (i)			
		(9) Notch rad	ius: 0	•			
- Flanges		FL 30×6 - \$23	518		n		
w.n			2211		U		
(1) Bottom:		100 -					
Connection, Po	ost:	Do not trim		•			
Infill corner							
Corner infills can o	only be defined	if:					
1. the guidel	ine has corners						
2. no corner	posts are create	ed (see 'Post dis	tribution')				
Infill, transition –							
Transition infills ca	n only he defin	ed if					-[
<u>'I</u>							-



Filling rod, inserted in handrail

### Post-Handrail - Wall console (prefabricated part)

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



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

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

If you want to preset other drillings, you can do this by changing the table Factory standards > Purchased/Factory standard parts > Railing > Wall consoles > Prefabricated brackets (RAILING\_ BRACKET\_29\_ISD.IPT). There you have to change the columns

- CS\_CAT\_ITEM (Bore for handrail mounting) and
- W\_CAT\_ITEM (Bore for wall mounting)

accordingly.

<b></b> ₩ €	$  \odot  $	🗿 🗎 🐻	14	ລ ຕ∣*∎	10 10 T	) 🖻 🛍 🎒	8
Purchased/Factory standard parts		t	w1	nb	COLOR	CS_CAT_ITEM	W_CAT_ITEM
Agraffes	1	12	30	2	-1	2724:7	2701:4
E Spring clins	2	12	40	3	-1	2724:7	2701:4
E Railing	3	12	30	2	-1	2724:7	2701:4
Corner glass holder  Corner glass holder  Glass holder  Hand rail fixing  Hand rail ends	4	12	40	3	-1	2724:7	2701:4
Wall consoles  Wall consoles  Prefabricated brackets (old)  Prefabricated brackets  Sleeves							

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

### Usage for railing segments

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

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

Proceed as follows:

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

		ID	MOD	STATUS	Designation	CONFIGKEY
	1	4		•	Skirting board	SKIRTING
Prototype beams     Composite papels, groove form	2	12		•	Skirting board profile	SKIRTINGPROFILE
Sconposite paintis, groove form     Sconposite paintis, groove form     Sconposite paintis, groove form	3	6	;	•	Rod	WEBMEMBER
E. 🔖 Usage	4	7	,	•	Infill	FILLING
E Scivil Engineering	5	8		•	Railing	RAILING
General	6	9		•	Railing segment	RAILINGSEGMENT
Element Installation	7	14			Railing segment (oblique)	STAIR_RAILINGSEGMENT
Metal Engineering	8	16	j	•	Railing segment Glass	GLAS_RAILINGSEGMENT
Profile installation	9	17		-	Railing segment Knee	KNIE_RAILINGSEGMENT
E Steel Engineering	10	13		•	Glass	GLASSPANE
Assemblies	11	5			Cross-member	STRINGER
Plate types	12	1			Handrail	HANDRAIL
Railing	13	10		•	Handrail profile	RAILINGPROFILE
	14	3	6		Knee rail	KNEERAIL
E Glazing types	15	2			Post	POST
Factory beams	16	11			Post profile	POSTPROFILE
LogiKal materials	17	15		•	Wall hand rail	WALLHANDRAIL
Fasteners						

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

💾 Derive struct	ure	$\times$	💾 Derive struc	ture X
Code	)rawing.USAGE_DEPENDENTMASTER		Code	GE_DEPENDEN
Copy values	ASSEMBLY_BEAM	•	Copy values	
	OK Cancel			OK Cancel

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

- ▲ 📰 Usage-dependent
  - Template
  - Default
  - DEFAULT(BETONSTAHL)
  - DEFAULT(BLECHE)
  - DEFAULT(C\_PROFILE\_KALT)
  - DEFAULT(FLACHSTAHL)
  - DEFAULT(FLUTZ\_PROFILE\_281)
  - DEFAULT(Gratings)
  - DEFAULT(GLASSCHEIBEN)
  - DEFAULT(HOHLPROFILE)
  - DEFAULT(I\_PROFILE)
  - DEFAULT(KANTBLECHE)
  - DEFAULT(KRANSCHIENEN)
  - DEFAULT(L\_PROFILE)
  - DEFAULT(PROFILE)
  - DEFAULT(SECHSKANTSTAHL)
  - DEFAULT(STAHLROHRE)
  - DEFAULT(T\_PROFILE)
  - DEFAULT(U\_PROFILE)
  - DEFAULT(U\_PROFILE\_KALT)
  - DEFAULT(VIERKANTSTAHL)
  - DEFAULT(Z\_PROFILE)
  - GLAS\_RAILINGSEGMENT
  - ▷ 📷 ISD\_RW\_PANEL
  - ▶ ISD\_RW\_TRAPEZPROFILE(ASSEMBLY)
  - KNIE\_RAILINGSEGMENT
  - MC\_BAR
- 4. The new usages must now be assigned. To do this, open the entry Usage assignment, activate a row in the Railing area and click on New. A new rowis created. In the column, select the name of the usage in the selection list, e.g. Railing segment Glass, in the second column the part type and in the third column the name of the template, e.g. GLAS\_RAILINGSEGMENT.Now you can define the dimensioning rules for the new types of use in HiCAD with the dimensioning rule editor. To do this, open the corresponding templates, e.g. GLAS\_RAILINGSEGMENT, adjust the dimensioning rules accordingly and save the template..

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

	Q Part attributes				- 0	×
3D-Part structure	Part name	764434562		BOM-relevant	~	
P / B B B B B P P	Article number	Segment		Qty. per part	1	
<ul> <li>Designation</li> </ul>	Drawing number			Item number	1	
RAIL_CONFIG_GLASS	Execution class	EXC2 (EN 1090)	•			
1 🛣 HEA 400	Additional tolerance	Klasse 1 (EN 1090)	-			
💶 🛣 HEA 400	Width					
🌋 HEA 400	Length					
4 💵 🏤 Railing	Height					
🕨 🤳 🏤 Segment	Weight	41.0	Weight fixed			
🔺 🜗 🏤 Railing	Usage	Railing segment GI	355		X	1
👂 🜗 🐏 Segment	Designation 1					-
	Designation 2	1x Segment				]
	Comment					1
	System notes					1
	Part type	Assembly			-	]
		Apply change	es	C	ancel	

### A simple example:

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

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

P 🖌 🖻 🖱 🖻 🖪 🏗 🖡	2 + 3	P 🕅 🖓
<ul> <li>Designation</li> </ul>	I	Comment
RAIL_CONFIG_GLASS		
Assembly		Assembly
🔳 🗾 HEA 400	4	I-beam with parallel
🐠 🛣 HEA 400	3	I-beam with parallel
4 🜗 🍖 <u>Railing</u>	2	Assembly
🕨 🜒 🍖 Segment	1	Assembly
🔺 🜗 🍖 Railing	1	Assembly
👂 🦺 🐂 Segment	1	Assembly



### Example of an automatic drawing derivation



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## Extension of the BOM template for Steel Engineering

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

A	В	С	D	E	F	G	Н	1	J	K
Pro	file s	structure list								
			-							
Drawing	No		Customer						•	
Order No	140.	221474	Created by	Miller						
Order ter	v. vt	Example	Created by	Willer						
Mamina	~	New hall	Created on							TU
anning		NOW HUM								
Item	Qty.	Designation	Length (mm)	Material	Туре	Naming	Coating	urface area (m	Weight (kg)	Total weig
ssy Fa	acade - A	Axis 1 - C16020	•							
136	1	8 C16020	2390	S450GD 20	C-Profil			14,19	13,82	11
137	10	6 C16020	2495	S450GD 20	C-Profil			29,62	14,43	23
j.	24	Assy Facade - Axis	59040					43,81		341
ssy Fa	acade - A	Axis 1 - C20025		j						
146	1	1 C20025	7550	S450GD 25	C-Profil			67,93	60,05	6
147	1:	2 C20025	7500	S450GD 25	C-Profi			73,62	59,66	7
270		1 C20025	7550	S450GD 25	C-Profil			6,18	60,05	
	24	4	180600					147,73		143
ssy Fa	acade - /	Axis 1 - C20030								
160		4 C20030	7500	S450GD 30	C-Profil			24,27	70,73	2
161	1	8 C20030	7450	S450GD 30	C-Profi			48,22	70,25	5
	12	2	89600					72,49		84
ssy Fa	acade - /	Axis 5								
168		4 L EN 10056-1-100x100x8	220	S235JR	L - Profile	K1L		0,34	2,68	
169		4 L EN 10056-1-100x100x8	220	S235JR	L - Profile	K1R		0,34	2,68	
162		4 C20030	7350	S450GD 30	C-Profil			23,78	69,31	2
	12	2	31160					24,47		29
1										
ssy Fa	acade - /	Axis 7	8 14							
168	1:	2 L EN 10056-1-100x100x8	220	S235JR	L - Profile	K1L		1,03	2,68	
169	18	5 L EN 10056-1-100x100x8	220	S235JR	L - Profile	K1R		1,29	2,68	
171	10	6 L EN 10056-1-150x100x10	150	S235JR	L - Profile	K2		1,18	2,85	
173		1 L EN 10056-1-150x100x10	150	S235JR	L - Profile	K3L		0,07	2,85	
275	1	2 L EN 10056-1-150x100x10	110	S235JR	L - Profile	K5		0,11	2,09	
138		1 C16020	2240	S450GD 20	C-Profil			1,66	12,95	
139		1 C16030	3975	S450GD 30	C-Profil			2,90	33,75	

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

Steel plates ar	nd metal shee	ets				
rawing No.	Customer					_
rder No.	Created b	/			1 14	
rder text	Created or	1				
aming						Chu
		Item	100	Material	\$235JR	
		Number	2	Designation		
		Designation	BI 10	Coating		
	S. 1	Length (mm)	320	Surface area (m <sup>2</sup> )	0,22	
		Width (mm)	170	Weight (kg)	1,52	
				Total weight	1,52	
	í)	Item	101	Material	S235JR	
		Number	1	Designation		
		Designation	BI 10	Coating		
		Length (mm)	276	Surface area (m <sup>2</sup> )	0,04	
		Width (mm)	70	Weight (kg)	1,52	
	U/			Total weight	1,52	
				a statistica teoret H		
		Item	102	Material	S235JR	
	<b>[</b> ],	Number	1	Designation		
	1 J	Designation	BI 12	Coating		
	•	Length (mm)	229	Surface area (m <sup>2</sup> )	0,08	
		Width (mm)	170	Weight (kg)	3,66	
				Total weight	3,66	

### User-defined tables with Steel Engineering plates

At Factory standards > User-defined semi-finished products > User-defined plates you now have the possibility to create tables with your own steel plates. These tables are then also offered for selection via the func-

tion Steel Engineering > Plate, new > Rectangular plate



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

# **Drawing Management**

Service Pack 2 2024 (V 2902)

### Automatic itemisation by part groups with prefix

Since SP1, an item prefix can be defined for each part group in the parameter settings for **Itemisation 1...**, provided that the generation of item texts is activated on the **General** tab. A part is then only uniquely identified by item prefix + item number. In this case, the parts are not only differentiated according to the item number as a number, but the item prefix is also taken into account during itemisation. This means, for example, that an assembly and a beam can have the same item numbers but different prefixes.

Previously, automatic itemisation by part group with prefix was not supported by the drawing management. As of SP2, this is now possible.

### Attribute DOKUART (Document type) and HicadBIMConfig.exe

To ensure that the database with Drawing Management works error-free, all attributes and links required for drawing management must be available there. This is ensured by the HiCADBIMConfig.exe programme in the HiCAD exe directory.

Previously, the manual user definitions of the HELiOS attribute DOKUART were overwritten when HicadBIMConfig.exe was executed. As of SP2, this attribute is defined as an attribute of type Suggestion list when HicadBIMConfig.exe is executed, so that the manual definitions are retained when the programme is executed again.

## Service Pack 1 2024 (V 2901)

### Documents for general files

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

#### Creation of external documents

This parameter determines whether external documents are to be created for the active sheet only or for all sheets. The default setting is **Active sheet**.

#### HELIOS attribute for HiCAD sheet names Here you specify which HELIOS attribute the HiCAD sheet name should be assigned to. The HELIOS attribute BENENNUNG (Designation) is preset.

Two new document attributes are also available in this context:

#### HICADDOKUART

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

e.y..

Documents X						
🍫 🖳 🖼 Standard 🔹 🖌	- K					
O O Document number In In W Designatio	n Document type	Creation dat	Created by	File changed on	HICADDOKUART	Document type
😋 📂 🔒 DN-001106 🛛 🕒 📀 Sheet 1	Others	15.11.2023	Administrator	15.11.2023 10:20:18	Model drawing	PDF file
🍋 🍋 DN-001109 🛛 🔵 📀 Item No.: 1	Others	15.11.2023	Administrator	15.11.2023 10:20:53	Assembly drawing	PDF file
늘 📂 🔒 DN-001111 🛛 🕒 📀 Item No.: 10	0 Others	15.11.2023	Administrator	15.11.2023 10:20:57	Detail drawing Beam	PDF file
📴 🗁 🔊 DN-001113 🛛 🔵 🚱 Item No.: 10	01 Others	15.11.2023	Administrator	15.11.2023 10:21:00	Detail drawing Sheet	PDF file

#### HICADPROJNUMMER

This attribute is assigned the project of the HiCAD document..

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

## Managing general 3-D parts via part filter

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

🥖 Manage general 3-D parts	Via filter 🗸 🗸	Consider general 3-D parts when managing drawings (Filter: "Steel Engineering Drawing Management -> General parts")
Condition for 'Drawing is up to date'	No	Status 'Drawing is up to date' depends on derived drawing(s).
Only allow 2-level structures	Only dummy parts	Check before data transfer to HELiOS structure
Output associated assembly(-ies)	Via filter	Detect associated assembly(-ies) for individual parts? (via HELiOS article attribute "COMPONENT_REFASSEMBLY")

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

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

cope Drawin nual filter	All User-defined	Selection list	
Part to Pur BOM Add	t <b>ype:</b> General I- <b>relevant</b> is	Change - Yes - P	<ul> <li>↓</li> <li>↓</li> </ul>
× r sund r ∴	Save favourite Manage favourites General Mechanical Engineering Plant Engineering Sheet Metal	Ca Preview E	ase sensitive Execute Cancel
	Steel Engineering	<ul> <li>Assembly, Facade Engineering Assembly, Steel Engineering</li> <li>Drawing Management</li> <li>Indiv. part, Facade Engineering</li> <li>Indiv. part, General</li> <li>Indiv. part, Steel Engineering</li> </ul>	General parts

ISD default setting for general parts in the Drawing Management

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

Part filters - Scope Drawing Selection lis	.t
Manual filter     All     User-defined       Part type:     General part, 3-D Sketch       BOM-relevant     is     Yes       Add rule:	Change Ch
General parts	Preview  Preview Preview Preview  Preview Preview Preview Preview Preview Preview Preview Preview Preview Preview Preview Preview Preview Preview Preview Preview Pre

Then save the modified Favourites file under the same name.

### Simplified handling when referencing in project structures

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

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



## Major Release 2024 (V 2900)

### Remove parts from production drawings

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

Proceed as follows:

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

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

### Processing sheet metal developments in production drawings

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

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

ISD Configuration Editor - HiCAD 29.0.0.185 [C:\F	ProgramData\ISD Software und Systeme\HiCAD 2024\HiCAD.cfgdb]		– 🗆 X
File Edit View Extras ISD			
/ 🕥 🛱 🕴 🖳 🕄 🕄	A 🛛 💿 🖕		User 🖉 🧟
🔺 🛅 HiCAD	Description	Comment	
Active configuration (Base configuration)	Drawing number		
<ul> <li>Drawing</li> <li>Automatic drawing derivation</li> <li>Modelling</li> </ul>	Generate n-digit drawing number	No	<ul> <li>Generate n-digit drawing numbers, preceded by left-justified zeroes, when generating production drawings (HELiOS document attribute DRAWINGNUMBER)?</li> </ul>
Steel Engineering	Drawing number generation	By projects	Generate drawing numbers ?
Metal Engineering	Drawing number always dependent on drawing type	No	Generate identical drawing numbers for each drawing type?
Profile Installation     Plant Engineering	Production drawings		·
Sheet Metal      Accombling cimulation	Update production drawings	Yes	Load and update production drawings upon checkup and release ?
<ul> <li>Analysis</li> </ul>	Production drawings for mounting assemblies	Yes	Should production drawings be created for parts of the type "Assembly" ?
Interfaces	Production drawings for glass parts	<ul> <li>Create production drawings for glass parts?</li> </ul>	
A Drawing Management	Production drawing for unprocessed beams	Yes	Create production drawings for unprocessed beams?
Production drawings	Processing note for unprocessed beams	Collection	Unprocessed beams with processing note from the list (HiCAD attribute SBHW) obtain a production drawing anyway
Individual part type	Allow processing of sheet developments	No	Should the processing of sheet metal development be allowed in production drawings ?
<ul> <li>Revision clouds</li> <li>HiCAD-HELiOS interface</li> </ul>	Classify production drawing	Yes	<ul> <li>Classification (e.g. detail drawing, assembly drawing, etc) on HELIOS attribute "DOKUART"</li> </ul>
Compatibility	Show release status in title block	Yes	<ul> <li>Highlight non-released production drawings in title block</li> </ul>
System settings	Consider total number in production drawing	Yes	v Is the total number relevant for production drawings?
P 🚛 Configurations	Create HELiOS attributes from FTD file	Yes	Create HELiOS attributes BENENNUNG (Designation) and CACHNUMMER (ARTICLENUMBER) from FTD files for detail drawings (BIM_PDM_WSD_Designation.ftd, BIM_PDM_WSD_ArticleNumber.ftd)
	Create only one production drawing per part	No	<ul> <li>Create only one production drawing per part</li> </ul>
	Output associated assembly drawing Ves SILVA		<ul> <li>Write associated assembly drawing to HiCAD Drawing attribute _SZNATTRRAS, Assembly item number to _SZNATTRRIN (for detail drawing).</li> </ul>
	Article attributes for drawings with multiple parts	Collection	Article attributes to be transferred to document attributes for drawings containing multiple parts (Format: Article attribute;Document attribute)
	Article attributes for detail drawings	Collection	Which article attributes are to be transferred to document attributes in detail drawings? (Format : Article attribute;Document attribute)
PDM > Drawing Management > Production drawings			

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

	×
With the setting Allow processing of developments in production drawings, DXF data for sheet metal parts cannot be generated automatically.	
OK	

### Releasing assemblies with already released parts

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

### Automatic BOMs for itemised source models

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

Create BOMs	This setting determines whether or not Excel BOMs should be automatically generated for certain model drawings.
	The following settings are possible:
	None Excel BOMs are not generated automatically. This is the default setting.
	<ul> <li>Upon Checkup and Release</li> <li>The Excel BOMs are created automatically when checking and releasing drawings.</li> </ul>
	<ul> <li>Upon creation and update The Excel BOMs are created automatically when creating and updating drawings.</li> </ul>
	<ul> <li>Automatically upon Release The Excel BOMs are created automatically upon release of drawings.</li> </ul>
List of model drawings with external BOM	Here you define for which drawings external BOMs are to be created and managed. At present, this is only possible for itemised source models, i.e. for the model drawings in which the parts are itemised (see document attribute ITEMISATIONMODEL).

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

Project Explor	er × Product Explorer	Folder Explo	rer Article Cl	ass Explorer			
💫 Find				ta ta     ta			
↓ P2 → Bills of Materials				↔ ₽			
lame	Designation	i.		*			
aph3							
sheets							
bomco	rr						
<b>p</b> 1							
n2							
Bills	of Materials	,					
	Documents X						
		rd	• 7 %				
	🦘 🌯 🐼 🍓 Standar	(7. ) ·					
	**     **     **     **     **     **       O'     O'     Document number	In In W	Designation	Document type	Creation date	Created by	File changed on
	O O Document number	In In W	Designation	Document type Others	Creation date 12.05.2023	e Created by Administrator	File changed on 12.05.2023 09:33:2

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



- The templates HiCAD-DB\_Stahlbau\_BIM.DE.2900.0.xlsx and HiCAD-DB\_Stahlbau\_BIM.rm\_settings are used for BOM creation.
- For parts managed with the Drawing Management, the HiCAD attribute Item text \$INTXT is assigned the HELiOS attribute COMPONENT\_ITEMNR\_TEXT. For external parts / standard parts, the HiCAD attribute \$INXT is assigned the item number of the part..

## Updating manually created production drawings

Production drawings that are created manually with the function Drawing Management > Workshop > Draw-

ing > Manual settings

 $m{T}$  can now also be updated with this function when changes are made in the

ALC: N

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

**Metal Engineering** 

## Major Release 2024 (V 2900)

Multiple selection of items when importing from LogiKal to HiCAD

When importing several LogiKal items to HiCAD via Import > Several facades+inserts

6				
	Position	Brief description	Series	
	001	Window right	heroal IS Lighting Well Cover (IWC)	
	002	Window left	heroal IS Lighting Well Cover (LWC)	
	003	Window front	heroal IS Ligthting Well Cover (LWC)	



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

### Replacing individual profiles

Previously the function Process > Exchange via LogiKal, with facade profile 75 only applied to multi-part

facade engineering profiles that were installed via the function New > Insert facade profile 📂 L

With HiCAD 2024 this has been extended to single profiles whose installation was done via New > Individual



### Extended transfer of LogiKal attributes

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



### Note on Integer attributes:

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



### Note on "[Insertion] Escape door function":

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

### Facade/insert with sketch: Sketch on upper glass edge

A behavioural optimisation of the HiCAD/ LogiKal interaction concerns the function Metal Engineering > New

### > Sketch (Facade/insert with sketch)

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



### Civil Engineering - Part type catalogue, 3-D

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

Metal Engineering / Facade Engineering	×
Find         Factory standards         Factory standards	<ul> <li>All</li> <li>Designation TEXT</li> <li>Insulation 3-D, soft Creates from an existing sketch a 3-D</li> </ul>
Insulation 3-D. soft	OK Cancel

In the first step, therefore, the entries under

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

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

# Layout Planning

## Service Pack 2 2024 (V 2902)

## New connection for ALUCOBOND SZ 20

There is a new option in the **Connection, top** list box for the ALUCOBOND Tray panel SZ 20. You can now create a Z-profile as a folded sheet for the window connection.



Top connection: Window with profile

## Fastening for ALUCOBOND Tray panel SZ 20

The bores in the Z-profiles of ALUCOBOND Tray panel SZ 20 for fastening to the substructure are now optionally adjustable. This is useful, for example, if the bore is carried out on site. In the **Extended settings** for the Zprofiles of the ALUCOBOND Tray panel SZ 20 (with accessories), the option **Create bores for sub-structure** can now be deactivated.



(1) Without bores for the sub-structure, (2) With bores for the sub-structure

## Service Pack 1 2024 (V 2901)

### Wall bracket usage for parts without a steel engineering axis

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

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

🥦 Wall bracket usage	× 🗵	Wall bracket usage	×
- Fitting points	— - Fi	tting points	
Select left point		Select left point	
Select right point		Select right point	(@) •••
- Position	Po	osition	)
Standard	۲	Standard	
O Attic, front	C	Attic, front	
O Attic, top		Attic, top	
OK Cancel	- A:	xis ———	]
		Select start point	<b>*</b>
		Select end point	() ()
		ОК	Cancel

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

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



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

Feature	e		×
*		🗓 🍇 💼 💽 👯 💱	₽. Ø
Desi	gna	tion	Value
Þ	1	(1) Sheet along sketch	
4	<u>}</u>	(13) Fitting coordinate sys	
	Þ	Point	
1	Þ	<ul> <li>Point</li> </ul>	
	Þ	Point	
	1	Distance	0 mm
	(	Designation	
	1	show coor name	YES
	(	Error handling	0
	1	🔇 Processed part	-
	Þ 1	Processing plane	
	1	Comment	
	(	🗔 Constraint	1
4	ų,	(14) Wall bracket usage	
	(	Positioning	Standard
	4 (	📁 Geometry	
	¢	> 💿 Left	I (577,75; 1062,54; 1000)
	¢	> 💿 Right	I (581,75; 1062,54; 1000)
	¢	>      Start point	M (631,25; 864,54; 1000)
	٥	<ul> <li>End point</li> </ul>	M (631,25; 864,54; -1000)
	Þ 1	Processing plane	
	1	Comment	
	(	Constraint	1
1.0	-	(2) Insertion Position	
•			• •
Featu	re	HCM Graphic Properties	Variables



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

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

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



## Major Release 2024 (V 2900)

### Change of the structure in the docking window

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



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

### Annotating installation elements

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



#### Element numbering.

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

Element numbering	×
Numbering by levels	
Automatic Manual Canc	el

#### Numbering by levels

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

#### Automatic

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

#### Manual

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

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

- EINumbering\_PerLevel.ftd (for numbering by levels) {Level of element installation numbering (part attribute)}-{Running number of element installation numbering (part attribute)}
- EINumbering\_Consecutive.ftd (for consecutive numbering) {Consecutive number of element installation numbering (part attribute)}

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

### Example:



Left: Numbering by levels, Right: Consecutive numbering.

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

# **Profile Installation**

## Service Pack 2 2024 (V 2902)

### New attributes for sandwich panels

When installing sandwich panels from the catalogue, you can now access new system attributes. The system attributes are assigned to the contents of the corresponding columns in the catalogue. You can find the catalogues for sandwich panels at Factory standards > Series > Roof Wall Facade > Room-closing profiles,

Attribute	Description	Catalogue column
DWF_T_OUT	Thickness of outer shell	THICKNESS_OUTSIDE
DWF_T_IN	Thickness of inner shell	THICKNESS_INSIDE
DWF_T_CORE	Core thickness	CORE_THICKNESS
DWF_CORE_MAT	Core material	CORE_MATERIAL

## Profile images in the packaging list

In addition to the usual bills of materials, packaging lists can also be automatically generated in Excel from a profile relocation. As of HiCAD SP2, these packaging lists show the cross-section of the profiles of the respective package. The designation **A** and **B** on the cross-section stands for the positive and negative position

respectively. You must create the packages beforehand using the **Packaging** function. A packaging list is only generated if at least one package has been defined.



Displayed cross-sections of packaged profiles in Excel

## **ROMA** profiles

As of SP2, panels from the company Romakowski GmbH & Co. KG are now also available for profile installation at Factory standards > Series > Roof Wall Facade > Room-closing profiles > Romakowski.
## Service Pack 1 2024 (V 2901)

## Licensing

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

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

Civil Engineering functions	φ×
et 🚚 🔽 🚺 🗐	
Steel Engineering	
Civil Engineering, general	
Metal Engineering / Facade Engineering	
Sheet Metal	
▲ Layout planning	
⊿ New	
Element Installation	
Sub-structure	
Connection	
🔱 Wall bracket	
Sketch creation for sub-structure	
Preparation	
▶ Edit	
▷ SZ20 DV	
D Analysis	
✓ Variants	
🥵 Save variant for element installation	
Save variant for sub-structure	_
Annotation	
Template drawing	
Timber construction	

## Additional column in Packaging dialogue

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

Na	ame	Attribute	Number of profiles	Weight
Pa	ckage 1	Area 2.1	7	217.51
Pa	ckage 2	Area 2.5	6	164.16
7 Elen FALK 1 FALK 1 FALK 1 FALK 1 FALK 1 FALK 1 FALK 1	060 WB 060 WB 060 WB 060 WB 060 WB 060 WB 060 WB	(100 mm) box/b (100 mm) box/b (100 mm) box/b (100 mm) box/b (100 mm) box/b (100 mm) box/b	oox PIR {Sandwichpan oox PIR {Sandwichpan oox PIR {Sandwichpan oox PIR {Sandwichpan oox PIR {Sandwichpan oox PIR {Sandwichpan oox PIR {Sandwichpan	eel} { eel} { eel} { eel} { eel} { eel} { eel} {

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

## Rename openings

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

Global settings		Other properties			
Transverse joints	Openings	Specializations	5		
Door 1					
Window 1			-		
Opening 3					
Opening 4					
		Z1		Window 1	
			Door 1	Willdow 1	Opening 3
Distance to sketch line		. 11			oponinge
Top: 0 •	Right: 0	*	XI		
Bottom: 0 🗸	Left: 0	-			

## Major Release 2024 (V 2900)

### Preset default values for joint width towards sketch line

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

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

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

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

#### Example:



## **Enhancements for Coating**

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

Transverse joints	Openings	Specializations	1		
Global settings	Oti	ner properties			~
Installation width:	1066.8	•			1
Negative direction					
Maximum length:	12192	•			
Cut lengthening: 🕕	0	•			
Usage:					
Roof/Wall		II			
Minimum width: 🕕	100	•			
	No elements found				
Colours Use outside colour Paint RAL 2005					
✓ Use inside colour					
Paint RAL 6027		197			
Paint RAL 6027					

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

### Coating of one-piece profiles

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



(1) One-piece profile coated on both sides,

(2) One-piece profile coated on one side,

(3) Coating in the BOM

### Negative and positive position and identical part search

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

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



# **Plant Engineering**

## Service Pack 2 2024 (V 2902)

## SIKLA clamps

The range of available parts has been expanded in SP2 to include additional variants for pipe clamps from SIKLA.



The variants can be found in the list file sikla\_pipe\_supports.lst. It contains the following variants:

File	Designation	Туре
LA-HV-150.VAA	Loose bearing LA-HV-150	Pipe clamp
LA-HV-200.VAA	Loose bearing LA-HV-200	Pipe clamp
LA-HV-90.VAA	Loose bearing LA-HV-90	Pipe clamp
LC-HV-150.VAA	Loose bearing LC-HV-150	Pipe clamp
LC-HV-200.VAA	Loose bearing LC-HV-150	Pipe clamp
LC-HV-90.VAA	Loose bearing LC-HV-90	Pipe clamp
LD-HV-150.VAA	Loose bearing LD-HV-150	Pipe clamp
LD-HV-200.VAA	Loose bearing LD-HV-200	Pipe clamp
LD-HV-90.VAA	Loose bearing LD-HV-90	Pipe clamp
LK-HV-150.VAA	Loose bearing LK-HV-150	Pipe clamp

The fastenings of the clamps are modelled as sub-parts so that you have the option of subsequently adjusting their position.



## Support type and support type symbols for pipe clamps

The **Pipe clamp support type attribute** - attribute name PIPE\_SUPPORT\_TYPE - can be assigned to pipe clamps using the **Part attributes** function. By assigning this attribute, different pipe support types can be displayed in ROHR2.

Three additional support types are available in the Part attributes mask as of SP2:

- Rigid hanger,
- Spring hanger and
- Spring support.

In addition, the entries now also show the ROHR2 abbreviations so that you can assign them more easily.

- diting dites				
Part name:	LHS43_350_1		BOM-relevant: 🗹	
Article number:	TN-05560	Qty. per part:	1 •	
Drawing number:		Unit of quantity:		
Item text:		Item number:	0 -	
Coating, inside:		Coating, outside:		
Weight:	0.00 🗸	Surface area:		
Width:		Quantity 1:	0.00 -	
Length:		Quantity 2:	0.00 -	
Height:		Quantity 3:	0.00 -	
Decignation 1:	USEGA Horizontalschelle 43-350-1	Quantity of		
Designation 1.		Dia a alara da sia akiran		
Designation 2:	0x 110-05504	Pipe class designation:		
Pipeline:	Pipeline_0002			
Pipe clamp support type:	Rigid hanger (SH)		-	
Comment:	None			
DE attaileate 1.	Fixed point (FP)			
PE attribute 1:	Plain bearing (GL)			
PE attribute 3:	Guide bearing (FL)			
PE attribute 5:	Axial stop (AX)			
DE attaile to 7	Transverse stop (QS)			
PE attribute /:	Plain bearing+axial stop (GLAX)			
PE attribute 9:	Guide bearing+axial stop (FLAX)			a a
	Transverse stop+axial stop (QSAX)			
	Rigid hanger (SH)			/ 0)
	Spring hanger (FH)			

Furthermore, from SP2 it is possible to define support type-dependent symbols to be used for visualisation in an isometric drawing.

If a support type has been assigned to a pipe clamp and a symbol has been assigned to this support type, the original symbolic representation of the pipe clampin the isometric drawing is replaced by the symbol of the support type.



The symbols for the support types can be found in the PlantParts\Symbols\PipeSupport sub-folder.

In the same directory you will find the **file support\_type\_symbols.txt**, which makes the assignment between the support types and the symbols. The file uses the abbreviations known from ROHR2 for the support types. A symbol is assigned using a line of the form

Abbreviation="filename without extension"

#### Example:

FP="fixed\_point"

This means that the symbol fixed\_point.FGA is assigned to the support type FP.

## DBPlantDataImport - Attribute initialisation

The DbPlantDataImport.exe program is the central tool for updating the HELiOS database to the content required for the current version of HiCAD in the Plant Engineering module. **DBPlantDataImport** can be used to preset attribute values for articles and documents. These initializations are used when synchronising the part data with HELiOS in order to assign the attributes of newly created articles and documents.

As of SP2, the attribute values are initialized as shown below.

Resourcing	10	J[Selected Init Value
	COMPONENT_PROCUREMENT	Purchase
Unit of quantity	COMPONENT_QUANTITY_UNIT	pcs
Part type	COMPONENT_TYPE	Raw-part+Plant-design

Initialization, Article - defaults

ttribute designation	Attribute name	Selected init value
ocument type	DOCUMENT_TYPE	HiCAD Part/Variant

#### linitialization, Document - defaults

The initializations are saved in the **DbPlantDataImportConfig.xml** file in the HiCAD sys directory. The version of this file included in the scope of delivery from SP2 contains the attribute **AddDefaults** with the default value 1

#### <AddDefaults>1</AddDefault>

This setting ensures that the above-mentioned default initializations are inserted. Each time the configuration is saved, AddDefaults=0 is set so that once saved, the default settings are no longer changed automatically. By setting AddDefaults=1 again, the default initializations can be added again, e.g. after deleting the ISD initialization.

## Isometry and pipe spool drawing

### Angles in the rise triangles

As of SP2, the rise angles can be drawn into the rise triangles of an isometry. For this purpose, the **Automatic dimensioning** tab in the isometric settings has been expanded to include the **Dimension rise angles** checkbox for horizontal and vertical rise triangles.

Pipe shortening	Text o	bjects	2-D Drawin	ng elements	ltem	numbers
Automatic dimensioning	Texts/Lines	Optimise	positioning	Connections	Lists	Symbols
Dimension line distances			Dime	ension figure height		
1st dimension line Fu	uther dimension lines	Rise triangles:				
25 2	D	25		2,5 💌 mm		
Only write dimension f	igure next to rise trian	gles				
Determine reference	ne acc. to part diame	ter				
Vertical rise triangles		Hori	zontal rise triangles			
✓ Draw triangle	Line param	eters	Draw triangle	Line parar	meters	
Bange of angles	5-	85 *	Range of angles	5 -[	85 °	
			Dimension rise and			
Dimension lise angles			Dimension rise angle	es		
limonoion hunotonuos			Dimension hypotent	use		
Dimension hypotenuse						
Dimension hypotenuse     Hatch triangle	0		Hatch thangle	. 0	-	
<ul> <li>Dimension hypotenuse</li> <li>Hatch triangle</li> <li>Hatching line spacing</li> </ul>		V	Hatching line spacir		]	
<ul> <li>Dimension hypotenuse</li> <li>Hatch triangle</li> <li>Hatching line spacing</li> <li>Triangle distance</li> </ul>			Hatching line spacir Triangle distance		]	
<ul> <li>Dimension hypotenuse</li> <li>Hatch triangle</li> <li>Hatching line spacing</li> <li>Triangle distance</li> <li>Complete hatching</li> </ul>			Hatching line spaci Triangle distance Complete hatching		]	
<ul> <li>Dimension hypotenuse</li> <li>Hatch triangle</li> <li>Hatching line spacing</li> <li>Triangle distance</li> <li>Complete hatching</li> <li>Triangle length</li> </ul>			Hatching line spaci Triangle distance Complete hatching Triangle length		] ] ]	

The following image shows a vertical rise triangle with angle dimension.



### Recolour pipe part symbols

The colour of the symbolic representation in the isometry is defined when a pipe part is inserted. The corresponding **Plant Engineering Settings** on the **Part insertion** tab are taken into account in the process.

	n branching points	Y Fasteners	Flange o	connection, bolted
Part search	P+ID attr	ibute assignment Y	Weld gap	Straight pipe
Part insertion	Part selection	P+ID symbol assignment	Link to P+ID	Bills of Material
Vork with gu	i <mark>del</mark> ine			
Insert part as		Representa	ation type	
O Sub-part		🔘 Hollo	w body	
		🕥 Solid	body	
Part of active	e pipeline	<ul> <li>Hollo</li> </ul>	w bodies + Symbols	
		🚫 Solid	bodies + Symbols	
Use part cop	ies for Variants	V Hollo	w body, with all details	
Delete	all copies			
/ Take over pi	peline colour			
Set edge colo     Take over pin     Take over pin     Check attribu     Calculate mis	our to pipeline colour peline layer ite assignment sing weights from geo	metry		
Set edge colo     Take over pi     Take over pi     Check attribu     Calculate mis     Line colour for syr     Specify for pi     Specify for pi	our to pipeline colour peline layer te assignment sing weights from geo <u>mbolic representation</u> pes and elbows I other parts	metry	Default se	tting

Sometimes, however, it only becomes apparent later that the colour or line type of the symbolic representations should be changed. Particularly when printing, there is often a desire to differentiate the pipework from other drawing elements - such as dimensions - in the visualisation. Previously, the colour of the symbols had to be laboriously adjusted by hand. As of SP2, this is now possible in a single step with the **Recolour pipe** 

**part symbols** function. To do this, the **3-D edge parameters** window is opened after calling up the function. Immediately after selecting the desired parameters, they change colour in the isometry.

	/		P	1	P
	3-D edge parameters		×		1D
⊕→¥	Block       Image: Constraint of the second sec	Coord (ine web)(.ayer)         I i APPLE           Image: Coord (ine web)(.ayer)         I i a APPLE           Image: Coord (ine web)(.ayer)         II i a APPLE           Image: Coord (ine APPLE)(.ayer		0- <i>1</i>	
		OK Cancel			



- In particular, symbols such as weld seams, flow symbols or even indicated parts are not recoloured. Dimensionings and rise triangles also retain their colour.
- The function is a post-processing step. This means that the colour is only changed for the current isometry. In particular, the colour is reset to the original colour when the isometry is regenerated.
- As only one crosshairs and one north arrow are generated, you may need to move both figures on the sheet using the 2-D part > Transform > Move > Move part, free (2-D) function.

### Create pipe spool drawing from pipe spool drawing

The **Pipe spool drawing** context menu, which you activate by right-clicking on a pipe in the pipe spool drawing, has been expanded to include the **From new part selection** function. Previously, this option was only available in the **Multiple selection** context menu.



It is now possible, for example, to create pipe spool drawings for individual pipes or pipe sections directly from an overall pipe spool drawing.

## Service Pack 1 2024 (V 2901)

## Length of inserted pipes

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

- Part attributes			
Part name:	N2448V2		BOM-relevant
Article number:	TN-02225	Qty. per part:	1
Drawing number:		Unit of quantity:	
Item text:		Item number:	0
Coating, inside:		Coating, outside:	
Weight:	•	Surface area:	
Width:	-	Quantity 1:	0
Length:	178.56 -	Quantity 2:	0
Height:	-	Quantity 3:	0
Designation 1:	Pipe		
Designation 2:	0x TN-02225	Pipe class designation:	RKL1_DIN

### Knee - Horizontal bending direction (VEERING\_RESTRICTION)

The new HELiOS attribute **VEERING\_RESTRICTION** complements the already existing attributes BENDING\_RESTRICTION (bending direction) and PLANE\_RESTRICTION (bending plane) in that also with VEERING\_RESTRICTION the route of a guideline can influence the part search.

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

The VEERING\_RESTRICTION attribute knows the following values:

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

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

#### Example - VEERING\_RESTRICTION

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



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

VEERING\_RESTRICTION = "\'3\*'\'2\*"

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

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

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

```
VEERING RESTRICTION = "\'3*'\'1*"
```

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

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

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

In the following example, the same composite edge has been fitted with a part whose VEERING\_ RESTRICTION has the value 3:



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

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



## Presettings for article attributes in the Configuration Editor

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



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

## Automatic checking of nominal diameters, pipe lengths and part structure

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



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

#### Nominal diameter matings

Here it is checked whether only parts with matching nominal diameters are connected with each other in the layout plan.

### Pipe lengths

It is checked whether inadmissible pipe length changes have been made in the layout plan.

#### Structure of the pipelines

The part structure of all pipelines is checked here.

a,		
Name	Description	
$\odot$		 
🔞 9 Erro	r	
TN-02233	Named point2 missing.(Below Error)	
TN-02233	Named point2 missing.(Below Error)	
TN-02217	Named point2 missing.(Below Error)	
TN-02233	Named point2 missing.(Below Error)	
TN-02233	Named point2 missing.(Below Error)	
TN-02217	Named point2 missing.(Below Error)	
TN-02233	Named point2 missing.(Below Error)	
TN-02233	Named point2 missing.(Below Error)	
TN-02217	Named point2 missing.(Below Error)	
🗼 12 Wa	rnings	
TN-02233	Named point1 missing.(Below Warning)	
TN-02217	Named point1 missing.(Below Warning)	
TN-01500	Connection 1 incorrect: Connection missing. (Below Warning)	
TN-01500	Connection 2 incorrect: Connection missing. (Below Warning)	
TN-02233	Named point1 missing.(Below Warning)	
TN-02217	Named point1 missing.(Below Warning)	
TN-01500	Connection 1 incorrect: Connection missing. (Below Warning)	
TN-01500	Connection 2 incorrect: Connection missing. (Below Warning)	
TN-02233	Named point1 missing.(Below Warning)	
TN-02217	Named point1 missing.(Below Warning)	
TN-01500	Connection 1 incorrect: Connection missing. (Below Warning)	
TN-01500	Connection 2 incorrect: Connection missing. (Below Warning)	
12 Info	ormation	
TN-05560	Named point10 missing.(Below RBL0201)	
TN-05626	Named point10 missing.(Below RBL0202)	

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

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

	P Che	ck unsuccessful.		_	×
		Za,			
		Name	Description		
A	40 1000 40 1000	$\odot$			<b>^</b>
<b>19-</b>		🔞 9 Error			
l 🔊		TN-02233	Named point2 missing.(Below Error)		
		TN-02233	Named point2 missing.(Below Error)		
		TN-02217	Named point2 missing.(Below Error)		
I I I I I I I I I I I I I I I I I I I		TN-02233	Named point2 missing.(Below Error)		
		TN-02233	Named point2 missing.(Below Error)		
31		TN-02217	Named point2 missing.(Below Error)		
		TN-02233	Named point2 missing.(Below Error)		
7		TN-02233	Named point2 missing.(Below Error)		
		TN-02217	Named point2 missing.(Below Error)		
🧧 📕 🎃 🦟	_	🔒 12 Warn	ings		
- andati	-	TN-02233	Named point1 missing.(Below Warning)		
		TN-02217	Named point1 missing.(Below Warning)		

## Changes/enhancements for pipe part insertion

Additional part information



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

Part insertion X
Part search Part insertion: Standard
- Additional parts
Weld seam gaps: Do not consider
Gaskets: Do not consider
Connecting parts
Type: Flange (20000) Connecting part:
Connection 2 Type: Butt-welded (10000) Connecting part
Connecting part
- Part information
Attribute Value
Nominal diameter (DN) 🔻 50
Standard designation 🔻 DIN 2633
Wall thickness 2.9mm
Outer diameter
Pressure 16
- Parult -
- result
- Selected part
Part type: Flange
Part: Welding neck flange DIN 2633
- Settings
General Straight pipes
☑ Immediately insert part after selection
AutoFlange
AutoReducer
Fix superordinate part: Select part
Image: Cancel



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

- Part information		- Part information	
Attribute	Value	Attribute	Value
Nominal diameter (DN) 🔻	50	Nominal diameter (DN) 🔻	50
Standard designation 🔻	DIN 2633	Standard designation 🔻	DIN 2633
-		-	
Wall thickness 🔹	2.9mm	Arbitrarily divisible	2.9mm
Outer diameter 🔹	60.3mm	Article code	60.3mm
Pressure 🔹	16	Article number	16
•		Connection type	
•		Designation	
-		Eavourite type	
-		HELIOS Revision ID	
		HELIOS Type name	
		Order note	
	2 7	Part type	
·		Thickness	



To change the attribute display of a row, simply select the desired attribute in the selection box. A click on this button deletes the current row of the attribute list.

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

### Undo/Redo

As of Service Pack 1, the Undo and Redo functions of the transparent toolbar or the quick access toolbar can

be used within the **Pipe parts I** be used within the **Pipe parts** 

	■ ( 母 <b>)</b> ( ) =
Drawing 3-D S	itandard Sheet Metal
Part insertion	×
Part search Part insertion: S	itraight pipe
- Additional parts	
Weld seam gaps: Do not co	onsider
Gaskets: Do not co	onsider
- Lengths Supplied length: 6000	Length allowance: 0
- Connecting parts	
Connection 1	
Type: Butt-welded (10000)	) Connecting part:
Connection 2	
Type: Butt-welded (10000)	) Connecting part:
B 41 6 - 41	
Attribute	Value
Nominal diameter (DN) 🔻	50
Standard designation 🔻	DIN 2448
Wall thickness 🔹 🔻	2mm
Outer diameter 🔹	60.3mm
- Result	
	Successful
	50003301

### Free point selection

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

-	AutoConnection	0
10-	Point 1 on cursor	1
20.	Point 2 on cursor	2
30.	Point 3 on cursor	3
-10-	Point 4 on cursor	4
	Reverse orientation	
F.	Switch axis	
•	Show part	

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



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

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



**Revised "Set all" option** 



In the **Part insertion** In the **Part insertion** in the **Selected part** area.



#### Insert everywhere

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



Then select the desired part after clicking on . If you have already selected and inserted a

part, clicking on will cause the part to be inserted automatically in the other corresponding places as well.

#### Example





### Highlight similar places

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



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

## Line blanks acc. to DIN2626

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

DIN2626 TYP A	Blind Disk Variant files: DIN2626-TYP_A-BD-PN10.VAA DIN2626-TYP_A-BD-PN100.VAA DIN2626-TYP_A-BD-PN160.VAA DIN2626-TYP_A-BD-PN160.VAA DIN2626-TYP_A-BD-PN25.VAA DIN2626-TYP_A-BD-PN40.VAA DIN2626-TYP_A-BD-PN6.VAA DIN2626-TYP_A-BD-PN63.VAA You can find these variants summarized in the list file DIN2626-TYP_A-BD.Ist.
DIN2626 TYP A	Perforated Disk Variant files: DIN2626-TYP_B-PD-PN10.VAA DIN2626-TYP_B-PD-PN100.VAA DIN2626-TYP_B-PD-PN160.VAA DIN2626-TYP_B-PD-PN160.VAA DIN2626-TYP_B-PD-PN25.VAA DIN2626-TYP_B-PD-PN40.VAA DIN2626-TYP_B-PD-PN6.VAA DIN2626-TYP_B-PD-PN63.VAA You can find these variants summarized in the list file DIN2626-TYP_B-PD.Ist.
DIN2626 TYP C	Orifice Plate DIN2626-TYP_C-OP-PN10.VAA DIN2626-TYP_C-OP-PN100.VAA DIN2626-TYP_C-OP-PN160.VAA DIN2626-TYP_C-OP-PN160.VAA DIN2626-TYP_C-OP-PN25.VAA DIN2626-TYP_C-OP-PN40.VAA DIN2626-TYP_C-OP-PN6.VAA DIN2626-TYP_C-OP-PN63.VAA You can find these variants summarized in the list file DIN2626-TYP_C-OP.Ist.

DIN2626 TYP D	Spectacle Blind As A Figure-8 Blank	
	Variant files: DIN2626-TYP_D-SB8B-PN10.VAA	
	DIN2626-TYP_D-SB8B-PN100.VAA	
	DIN2626-TYP_D-SB8B-PN16.VAA	
	DIN2626-TYP_D-SB8B-PN160.VAA	
	DIN2626-TYP_D-SB8B-PN25.VAA	
	DIN2626-TYP_D-SB8B-PN40.VAA	
	DIN2626-TYP_D-SB8B-PN6.VAA	
	DIN2626-TYP_D-SB8B-PN63.VAA	
	You can find these variants summarized in the list file <b>DIN2626-TYP_D-SB8B.lst</b> .	
DIN2626 TYP E1	Spectacle Blind As A Rotating Plate	
	The E1 subtype covers nominal sizes from DN 15 to DN 50.	
	Variant files:	
	DIN2626-TYP_E1-SBRP-PN10.VAA	
	<ul> <li>DIN2626-TYP_E1-SBRP-PN16.VAA</li> <li>DIN2626 TYP_E1 OPDP DN25 VAA</li> </ul>	
	<ul> <li>DIN2626-TYP_ET-SBRP-PN25.VAA</li> <li>DIN2626-TYP_E1-SBRP-PN40 VAA</li> </ul>	
	You can find these variants summarized in the	
	list file DIN2626-TYP_E1-SBRP.lst.	
DIN2626 TYP E2	Spectacle Blind As A Rotating Plate	
	The E2 subtype covers nominal sizes from DN 65 to DN 200.	
	Variant files:	
	<ul> <li>DIN2626-TYP_E2-SBRP-PN10.VAA</li> </ul>	
	<ul> <li>DIN2626-TYP_E2-SBRP-PN16.VAA</li> <li>DIN2626 TYP_E2 SBRP DN25 VAA</li> </ul>	
	<ul> <li>DIN2020-11P_E2-SBRP-PN25.VAA</li> <li>DIN2626-TYP F2-SBRP-PN40 VAA</li> </ul>	
	You can find these variants summarized in the	
	list file DIN2626-TYP_E2-SBRP.lst.	

## GF Piping Systems

### PROGEF

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



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

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

File	Designation	Туре
PROGEF_TEE_RED_MOLDED_SDR17_ 6.VAA	T 90° reduced, molded SDR 17.6	T-piece
PROGEF_TEE_RED_REDUCER_SDR11.VAA	T 90° butt welded reducer SDR11	T-piece

## ECOFIT

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


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

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

File	Designation	Туре
ECOFIT_SCREW_FITTING_SDR11.VAA	Bolting screw-in part SDR 11	Flange
ECOFIT_SCREW_FITTING_SDR17.VAA	Bolting screw-in part SDR 17	Flange
ECOFIT_STUB_FLANGE_A_SDR17.VAA	Butt welded collar Type A SDR 17	Flange
ECOFIT_STUB_FLANGE_B_SDR17.VAA	Butt welded collar Type B SDR 17	Flange
ECOFIT_TEE_45_SDR11.VAA	T 45° equal SDR 11	Branch
ECOFIT_TEE_45_SDR17.VAA	T 45° equal SDR 17	Branch
ECOFIT_TEE_RED_45_SDR11.VAA	T 45° reduced SDR 11	Branch
ECOFIT_TEE_RED_REDUCER_ SDR11.VAA	T 90° butt welded reducer SDR 11	T-piece
ECOFIT_TEE_RED_REDUCER_ SDR17.VAA	T 90° butt welded reducer SDR 17	T-piece
ECOFIT_TEE_RED_SHORT_SDR11.VAA	T 90° reduced SDR 11, short	T-piece
ECOFIT_TEE_RED_SHORT_SDR17.VAA	T 90° reduced SDR 17, short	T-piece
ECOFIT_TEE_SHORT_SDR11.VAA	T 90° SDR 11, short	T-piece
ECOFIT_TEE_SHORT_SDR17.VAA	T 90° SDR 17, short	T-piece
ECOFIT_UNION_NUT.VAA	Bolting - union nut	Fastener, unsym- metric

## ELGEF

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



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

File	Designation	Туре
ELGEF_BEND_45_SDR11.VAA	Segment bend 45° SDR 11	Knee
ELGEF_BEND_45_SDR17.VAA	Segment bend45° SDR 17	Knee
ELGEF_BEND_60_SDR11.VAA	Segment bend60° SDR 11	Knee
ELGEF_BEND_60_SDR17.VAA	Segment bend60° SDR 17	Knee
ELGEF_BEND_90_SDR11.VAA	Segment bend90° SDR 11	Knee
ELGEF_BEND_90_SDR17.VAA	Segment bend90° SDR 17	Knee
ELGEF_CAP_L_SDR11.VAA	End cap Type L SDR 11	Сар
ELGEF_CAP_L_SDR17.VAA	End cap Type L SDR 17	Сар
ELGEF_CAP_SDR11.VAA	End cap SDR 11	Сар
ELGEF_COUPLER_SDR11.VAA	Socket SDR 11	Other pipe part
ELGEF_COUPLER_SDR17.VAA	Socket SDR 17	Other pipe part
ELGEF_COUPLER_SDR26.VAA	Socket SDR 26	Other pipe part
ELGEF_ELBOW_45_SDR11.VAA	Elbow 45° SDR 11	Knee
ELGEF_ELBOW_90_SDR11.VAA	Elbow 90° SDR 11	Knee
ELGEF_ELBOW_L_15_SDR11.VAA	Elbow 15° Typ L SDR 11	Knee
ELGEF_ELBOW_L_15_SDR17.VAA	Elbow 15° Typ L SDR 17	Knee
ELGEF_ELBOW_L_30_SDR11.VAA	Elbow 30° Typ L SDR 11	Knee
ELGEF_ELBOW_L_30_SDR17.VAA	Elbow 30° Typ L SDR 17	Knee
ELGEF_ELBOW_L_45_SDR11.VAA	Elbow 45° Typ L SDR 11	Knee
ELGEF_ELBOW_L_45_SDR17.VAA	Elbow 45° Typ L SDR 17	Knee
ELGEF_ELBOW_L_90_SDR11.VAA	Elbow 90° Typ L SDR 11	Knee
ELGEF_ELBOW_L_90_SDR17.VAA	Elbow 90° Typ L SDR 17	Knee
ELGEF_REDUCER_BW_ SDR11.VAA	Butt welded reducer SDR 11	Reducer, con- centric
ELGEF_REDUCER_BW_ SDR17.VAA	Butt welded reducer SDR 17	Reducer, con- centric
ELGEF_REDUCER_SDR11.VAA	Reducer SDR 11	Reducer, con- centric
ELGEF_SPIGOT_CLAMP_ SDR11.VAA	Spigot clamp SDR 11	Saddle connection
ELGEF_TEE_DRILL_SDR11.VAA	T 90° tapping-T SDR 11	Corner valve
ELGEF_TEE_L_SDR11.VAA	T 90° Type L SDR 11	T-piece
ELGEF_TEE_RED_SADDLE.VAA	T 90° saddle connection reduced SDR 11	T-piece
ELGEF_TEE_RED_SDR11.VAA	T 90° reduced SDR 11	T-piece
ELGEF_TEE_SDR11.VAA	T 90° SDR 11	T-piece

## Polyethylene pipes acc. to DIN 8074

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

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

## Isometry and pipe spool drawing

#### Down-grade symbol

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

File Edit View Extras ISD						
		A 0		User		
<ul> <li>HiCAD</li> <li>Bactive configuration (Base configuration)</li> </ul>	*	Description Spreading of isometry over several drawings	Value	Comment		
Drawing		Create complete BOM when dividing isometry				
Automatic drawing derivation		Create only one BOM when dividing isometry				
Modelling		Down-grade symbol				
Steel Engineering		Distance between down-grade symbol and text	1 mm			
<ul> <li>Metal Engineering</li> <li>Profile Installation</li> </ul>	III.	Number of decimal places for angle representation	0			
4 📰 Plant Engineering		Display unit for down-grade	deg.	~		
Accessory parts Plant Engineering drawing check		Number of decimal places for percent representation	deg. Radiant			
Isometry and Pipe Spool Drawing		Number of decimal places for display	Percent			
III Layout plan ▷ III C-edge		Number of decimal places for per mille representation	gon Millimetre i	per metre		
P HD		Diagnosis options	Inches per t	foot		E
Bills of Materials		[Debug] Show trim frame	Per mil			
ROHR2		[Debug] Do not hide original pipeline				
📰 Symbol Editor	*	[Debug] No warnings				-

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

Pipe shortening	Text objects	2-D Drawing elements	Item numbers
utomatic dimensioning	Texts/Lines	Optimise positioning Connections	is Lists <mark>/ Symbols</mark>
Down-grade Symbol N Enlargemu Range of Offset from Text V Set or	symbol	Select  Select  1.00  0.1 - 10.0 deg.  1.0 mm SLOPE_CFG_UNI Text parameters on Attribute	
Defa	ault At	tribute selection Set %PIPE_SLOPE_CFG_UNIT	
		Attributes from 3-D part model Description Down-grade in degrees Down-grade in per mille Down-grade in percent Down-grade in unit from Config Ma Name of component connection	Wildcard %PIPE_SLOPE_ANGLE %PIPE_SLOPE_PERMILLE %PIPE_SLOPE_PERCENT anag %PIPE_SLOPE_CFG_UNIT CONNCON_NAME Apply
		Attributes from the database	Apply

#### Units in the isometry and pipe spool drawing settings

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

Pipe shortening	Text object	s 2-D D	rawing elements	Item	numbers
Automatic dimensioning	Texts/Lines	Optimise positioning	Connections	Lists	Symbols
Dimension line distances		(	Dimension figure height		
1st dimension line Further d	imension lines: Ris	e triangles:			
25 20	25	<u>.</u>	2.5 <b>v</b> mm		
Only write dimension figure n	ext to rise triangles				
Determine references lies and	ext to fise thangles				
	c, to part diameter	] [			
Vertical rise triangles		Horizontal rise triang	gles		
✓ Draw triangle	Line parameters	Draw triangle	Line param	eters	
Range of angles	5 - 8	Range of angle	es 5-	85 °	
Dimension rise angles		Dimension rise	angles		
Dimension hypotenuse		Dimension hyp	otenuse		
✓ Hatch triangle		✓ Hatch triangle			
Hatching line spacing	2 💌	Hatching line s	pacing		
Triangle distance		Triangle distan	ce 🚺 🚽		
			··		
Complete hatching		Complete hat	ching 🗾		
Triangle length	20 👻	Triangle length	20 👻		
Triangle surface area	0 1/2e-06	Triangle surfac	e area		
Position of dimension lines		Dimension types			
Always in vertical plane		▼ Part dimension	s		
Dimensions in all views		Dimensio	ns for all parts		
Repeat dimensions		Also dime			
Draw angular dimensions		Dimensio	ns for points of action		
Angular dimensions		Comer dimensi	ons		
Invert Angular dista	ince 0.2	Dimensions for	points of action		
	1				
Default					

The values can be entered in any desired unit and are then automatically converted into the unit of measurement preset in the Configuration Editor. If you point the cursor at the <sup>1</sup> symbol, the preset unit is displayed.



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

#### Generate pipe book

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



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

#### Microsoft® Excel is required for creating a pipe book.

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

After calling up the **Pipe book** function, the Report Manager is automatically started with a corresponding quantity list. This quantity list is the basis for the pipe book. You then generate the pipe book itself in the Report

Manager using the Create Excel document (with template or script)

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



#### Step 1: Create the isometry



Step 2: Generate a quantity list for the pipe book

Isometry+Pipe Spool Drawing > Edit lists > EditPL - > Pipe book

Report Manager										-	- 0
e <u>E</u> xport Settings <u>T</u> oc	ls			77							
🚺 🔣 HiCAD_PipeBo	ok	~ <b>Ľ</b>	6 🚯 🛷 -	<b>16</b> -	🕼 - 🤮 - ಶ - 🗍	[ + 🥥 +					¢ 0
		Quantity	ist Structure Lis								
PIPEBOOK_ISO	1	A ISO	Article number	Number	Designation	Standard	Length [mm]	Width [mm]	Height [mm]	Nominal diameter	Nominal diam
DIN 2633 (Welding	neck flange DIN 2633} {0}	1	TN-02225	1	Pipe	DIN 2448	100.00			32	
DIN 2448 {Pipe} {0}		2	TN-02225	1	Pipe	DIN 2448	100.00			32	
P DIN 2615 T1 (T-pie	ce} {0}	3	TN-02225	1	Pipe	DIN 2448	100.00			32	
DIN 2448 {Pipe} {0}		4	TN-02225	1	Pipe	DIN 2448	100.00			32	
T DIN 2615 T1 (T-pie	ce} {0}	5	TN-02225	1	Pipe	DIN 2448	100.00			32	
DIN 2448 {Pipe} {0}		6	TN-02225	1	Pipe	DIN 2448	100.00			32	
🖵 DIN 2615 T1 {T-pie	ce} {0}	7	TN-02225	1	Pipe	DIN 2448	100.00			32	
DIN 2448 (Pipe) (0)		8	TN-02225	1	Pipe	DIN 2448	100.00			32	
🖵 DIN 2615 T1 (T-pie	ce} {0}	9	TN-02225	1	Pipe	DIN 2448	100.00			32	
DIN 2448 (Pipe) (0)		10	TN-02225	1	Pipe	DIN 2448	100.00			32	
DIN 2633 (Welding	neck flange DIN 2633} {0}	11	TN-02225	1	Pipe	DIN 2448	100.00			32	
🖵 DIN 2615 T1 (T-pie	ce} {0}	12	TN-02225	1	Pipe	DIN 2448	100.00			32	
DIN 2448 (Pipe) (0)		13	TN-02225	1	Pipe	DIN 2448	100.00			32	
DIN 2605 (Flbow) (	0}	14	📜 TN-01619	1	Elbow	DIN 2605				32	
DIN 2448 (Pine) (0)		• 15	TN-01619	1	Elbow	DIN 2605				32	
		16	TN-01619	1	Elbow	DIN 2605				32	
		17	📜 TN-01619	1	Elbow	DIN 2605				32	
Designation	Value	18	TN-02283	1	T-piece	DIN 2615 T1				32	
		18	TN-02283	1	T-piece	DIN 2615 T1				32	
		19	📜 TN-02283	1	T-piece	DIN 2615 T1				32	
		19	TN-02283	1	T-piece	DIN 2615 T1				32	
		20	TN-02283	1	T-piece	DIN 2615 T1				32	
		20	TN-02283	1	T-piece	DIN 2615 T1				32	
		21	TN-02283	1	T-piece	DIN 2615 T1				32	
		21	TN-02283	1	T-piece	DIN 2615 T1				32	
		22	TN-02283	1	T-piece	DIN 2615 T1				32	
		23	TN-02283	1	T-piece	DIN 2615 T1				32	
		24	TN-02610	1	Welding neck flange DIN	DIN 2633				32	
		25	TN-02610	1	Welding neck flange DIN	DIN 2633				32	
		1									

#### Step 3: Create the Excel document

								Ţ													
I	3	5 - ಿ ಡಿ.⊮ =				PIPEBC	OK_ISO.S	ZA.G.xls	· - HELi	OS Plot Man	agement /	Add-In act	ive!				Martin	Kraus 🚺	ik) 🖪	- E	o x
	ile	Home Insert Pa	age Layout Formulas	Data F	Review	View He	lp A	crobat	٦	'ell me what	you want f	to do									$\Box$
Pa CI	ste pboa	→         Arial           →         B         I         U         ~           rd         53         Fr	• 10 • A* A* = □ • 20 • ▲ • ont 5	E = = Align	ŵr - 8 €≣ 3⊒ E ment	b Ger	neral ~ % Numl	9 500	.00 .00 .00 .00	📳 Conditio 👿 Format a 👿 Cell Style :	nal Format s Table ~ s ~ styles	ting ~	Ensert Cells Cells	e ~	∑ ~ A ↓ Z Sort Filte Edi	t & Find & er ~ Select ting	Ado	1-ins	Create P and Share A	DF Create PD link Share via O dobe Acrobat	) Fand utlook
Т4	6	• : × ✓	fx																		*
	A	В	c	D	E	F	G	Н	1	1	K	L	M	N	0	P	Q	R	S	T	f
1	Pip	e book																		-	
2 3 4 5 6	Draw Order Order Desig	ng No. No. Text nation			Customer Creator Created on															HÌG	٩D
7			Materia	documenta	ation			Weld s	eam doo	umentation		-			In	spection -	docume	ntation			_
9	Item	Part	Dimensions	Material	Batch	Certificate	Sear	m no.	WPS	Seam type	Welder	PWHT	VT	RT Test no	UT	MT/PT	HAT	PMI	WDP	Heat treatment	t protocol
11 12	24	Welding neck flange DIN 2633	DIN 2633 C 32x42.4					1													
13	1	Pipe	DIN 2448 - 42.4x2				$\leq$	Plant													
15	18	T-piece	DIN 2615 - 1-42.4x2-42.4x2				$\leq$	Plant													
17	2	Pipe	DIN 2448 - 42.4x2		-		$\leq$	Plant 4													
19	19	T-piece	DIN 2615 - 1-42.4x2-42.4x2				$\leq$	Plant													
21	3	Pipe	DIN 2448 - 42.4x2				$\leq$	Plant													
23	20	T-piece	DIN 2615 - 1-42.4x2-42.4x2				$\leq$	Plant													
25	4	Pipe	DIN 2448 - 42.4x2		-		>	Plant													
27	21	T-piece	DIN 2615 - 1-42.4x2-42.4x2				>	Plant													
29	5	Pipe	DIN 2448 - 42.4x2		-		$\leq$	Plant 10													
31	25	Welding neck flange DIN 2633	DIN 2633 C 32x42.4				$\leq$	Plant													
	4	Pipe book	÷		1	1	L		L	L	L	: •	.L	L	L	L			山		► + 100%

Create Excel document (with template or script)

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

				100		de et
Pine	DIN 2448 42 4×2			<u> </u>	Plant	
File	DIN 2440 - 42.4X2				2	
Tiniaca	DIN 2615 1 42 4v2 42 4v2				Plant	[ .
I-piece	DIN 2013 - 1-42.4A2-42.4A2				3	
Dine	DIN 2448 42 4v2		×		Plant	[ .
Fibe	0111 2440 - 42.482	· · · · · · · · · · · · · · · · · · ·			4	

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

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

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

# **Pipeline Tools**

## **Calculate transition**

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

Calculate transition			×
- (1) Arc parameters			
Designation:	Elbow		<b></b>
Radius:	270	Outer diameter:	0
Angle in degrees:	90	Wall thickness:	0
Nominal diameter (DN):	100	Nominal diameter (NPS):	0
- 🗹 (2) Arc parameters -			
Designation:	Elbow		II 🔊
Radius:	270	Outer diameter:	114.3
Angle in degrees:	90	Wall thickness:	6.3
Nominal diameter (DN):	100	Nominal diameter (NPS):	0

#### **Determine volume**

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

Volume calculation	×
Active pipeline Pipeline_0001	-
Volume	
<ul> <li>Cubic centimeter</li> <li>Litre</li> </ul>	
<ul> <li>Hectolitre</li> <li>Cubic metre</li> <li>Cubic inch</li> </ul>	
Cubic foot	
Parts evaluated for volume calculation are indicated	
ОК	

## **Guideline Editor**

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

# Major Release 2024 (V 2900)

## Component connections with flange parameters

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

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

S	Gelect part
S	elect point
Name:	•
- Connection	
Options:	Welded 🗸
Side:	None
ID:	10000
Q Specify co	onnecting direction
Thickness of flange plate	e: 0 🔻
- Dimensions	
Nominal diameter type:	Nominal diameter (DN) 🔻
Nominal diameter:	0 -
Wall thickness:	0 •
Outer diameter:	0 -

# Down-grade Editor - more units

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

- Mode	192
Select pipeline sections	
O Select entire pipelines	
O Select fixed points	
O Select fixed length sectio	ns
Down-grade	
Specify down-grade:	
0 -	• •
Show preview	o dag
	ueg.
- Options	Radiant
Cut elbows	%
Mitre cut straight pipes	Percent
Observe maximum lengt	n: gon
Dead alian at	gon
Dend pipe at:	mm/m
3 • x Outer diameter	. Millimetre per met
	in/ft
- Result	Inches per foot
	%o
	reima

## Isometry and pipe spool drawing

#### Generate pipe spool drawing from Sheet view

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

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

#### Changed default settings during isometry/pipe spool drawing creation

#### Orientation of 2-D drawing elements

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



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

Symbols Pipe shorte	ening Text objects	2-D Drawing elemen	ts Item numbers
Contents	Path	Orientation	(X-Pos (Y-Pos
North arrow		Left	
L Tripod		Bottom	
Bill of Materials (BOM)		Right	
🔊 Length list		Right	
Connection list		Right	

#### Data record assignment for the connection type

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

Automatic dimensioning	Texts/Lines Continise positioning Connections Lists
Connection type	Welded
	Assign item numbers: Within a pipeline
	🔽 Assign item numbers: At pipeline ends
	Use "Part.Connection" format
Symbol for connection typ	)e
for pipes and elbows	(within)  Symbol No. 131  Select
for pipes and elbows	(within) Symbol No. 131 Select
for pipes and elbows	(within) Symbol No. 131 Select
for pipes and elbows Data record and place of For joints within the pi	(within)  Symbol No. 131  Select
for pipes and elbows	(within)  Symbol No. 131  Select
for pipes and elbows Data record and place of For joints within the pi Record assignment ID number	(within)  Symbol No. 131  Select

This prevents the following message from appearing:



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



Insert connection pieces



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

Part insertion				×
Part search Part insertion	n:			
Active pipeline: RBL010	1			P 🔒
Straight pipe	Valve	Other pipe part	Сар	
Elbow	Corner valve	Knee	Double knee	
T-piece	3-way valve	Branch	Y-piece	
Cross-shaped branch	4-way valve			
Flange	Loose flange	Blank flange	Seal	
Reducer, symmetical	Reducer, asymmetrical			
Saddle connection	Elbolet	Fastener, symmetric	Fastener, unsymmetric	
Pipe clamp	Gauge part	Nozzle	Welding neck	
- Search conditions				
	R	KL1_DIN		
Pipe class	Default setting  R	N-00004		
Standard	Default setting 🔻			-
Connection type	Default setting 🔻			• 🚑 😔
Nominal diameter	Default setting 🔻 5	0		•
Show all				📖 👰 🌠 Select part
- Selected part				
Part type:				
Part:				
- Settings				
General Straight pipes				
Min. pipe length: 10	▼ Loose	flange Do not insert	-	
Max. pipe length: 60	00 🔹			
Observe supplied leng	gth			
<ul> <li>Observe length allows</li> <li>Insert connecting piece</li> </ul>	tes			
				]
				OK Cancel

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

If no connecting pieces are preselected in the Plant Engineering Settings, this is indicated by the ① symbol in the **Part insertion** dialogue window.

General Straight pipes				
Min. pipe length: 10	•	Loose flange	Do not insert	•
Max. pipe length: 6000	No connectir	a parts found i	n the settings.	
<ul> <li>Observe supplied length</li> <li>Observe length allowance</li> <li>Insert connecting pieces</li> </ul>	$\overline{\mathbf{V}}$	5,		

#### Please note:

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

#### Changed buttons

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



## Part data synchronisation

#### Indication of missing attributes

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

	Part data synchronisation					– 🗆 🗙			
			Edit document	master					×
	Create log files								
	Always open article data	mask. naster mask	Document						
	Message if Plant Eng. att	tributes missing							
	Message if Plant Eng. da	ssification missing	Basic informa	ation					
	Allow use of existing artic	cle masters	Project nu	mber: PN	I-01-06-K		Index:		
	Enocify attributes which mus	t not be avaruritti	Project nut	riber. Pro	1-01-00-K		muex.	- I-S-I O	
	Specify autobutes which mus	t not be overwrite	Constructi	on section:			Sheet:		
	Language for text attribute	values:	Drw. No.:				Itemisation:		
	[								
	WORKNOW	c	Designatio	n: Ba	III valve PN	16 👻	Release:	In Progress	
	Request article workflow	L	Document	category:			Document category	n 🔒 👻	
	Request document work	now						1	
	Workflow selection is only po	ossible if the part do	bes not yet exist in the	database.		Missing attribute valu	Jes	/	
	(cit		Gammant						
	EXAMPLE.VAA		(Comment			No values entered fi	or the following attributes	s:	
						DOCUMENT TYPE			
						boddinetti_inie			
						1			
						Ignore missing value	es (not recommended)?		
						Yes	No		
	Patro and a Patro indexe						~		
	Enter article; Folder-Indepe	ndent					~		
A	rticle Valve								
	Basic information								
	Particular International								
	Project number:	Project-indeper	ident 🛄 🟮	Project-related	: 🗸	lisol 🕥			
	Construction section:								
	Item No.:			Index:					
	Standard designation:	(A2HF1_16)		Release:	In Pro	ogress			
	Designation 1:	Ball valve PN 1	6 🗸	Part type:			-		
	Designation 2			Drawing/Manuf	e .				
	boolgilation Li			brattingmand					
									< l
	Extended information							Missing attribute values:	
	Material:			Unit of quantity			-	BESTELLVERMERK	
	Weight:		ka	Resourcina:			·		
	Dimensions:	4- 		Order note:				ОК	
1.1	entre le la								

#### Derived variants with different attributes

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

#### Synchronisation with catalogue - Units and categories

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

Attribute text	Attribute value	Category	Unit	-
Material: Material number				
Preferred type		<b>•</b>		_
Maximum nominal pressure		Unitless number	-	-
Nominal diameter				
Nominal diameter 3				
Outer diameter				
Outer diameter 3				
Wall thickness				
Wall thickness 3		Length	- cm	•
NPS (inch)				

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

## Create pipeline - Assigning of nominal diameters

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

🔽 Assign nominal diameter	DN 👻
🔘 from pipe class	DN NPS
from reference part	Select part
🔘 via direct setting	

- Search conditions			
Pipe class	Default setting 💌	RKL1_DIN RN-00004	
Standard	Default setting 💌		•
Connection type	Default setting 🔻		• 🗿 📀
Nominal diameter (DN)	Default setting 🔻		• 🔐 📀
Nominal diameter (NPS)	Default setting 🔻		•

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

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

Plant Engineering Settings	×
Bills of Materials Part search P+ID attribute Part placing on branching points Faste Part insertion Part selection P+ID symbol assignment	assignment Weld gap Straight pipe mers Flange connection, bolted
Work with guideline	
Insert part as Sub-part Main Part Part of active pipeline	<ul> <li>Representation type</li> <li>○ Hollow body</li> <li>○ Solid body</li> <li>○ Hollow bodies + Symbols</li> <li>○ Solid bodies - Symbols</li> </ul>
Use part copies for Variants	Hollow body, with all details
<ul> <li>Take over pipeline colour</li> <li>Set edge colour to pipeline colour</li> <li>Take over pipeline layer</li> <li>Check attribute assignment</li> </ul>	<ul> <li>only in mp</li> <li>only in inches</li> </ul>
Calculate missing weights from geometry Line colour for symbolic representation	
Specify for all other parts	
(	OK Cancel

Plant Engineering Settings	×
Bills of Materials         Part search         P+ID attribute assignment         Weld ga           Part insertion         Part selection         P+ID symbol assignment         Link to P+ID         Action           Part placing on branching points         Fasteners         Flange of	p Straight pipe ons during Load/Save onnection, bolted
Use presettings for placing of parts on branching points	
File selected as global default: C:\HiCAD\PlantParts\BranchTypeSettings\Default.BranchTypeDN.xml	
T       T         600x600       T         T       T         600x500       500x500         T       T         G00x450       500x450         450x450         T       T         G00x400       450x450         P       T       T       T         G00x400       450x400       400x400         P       T       T       T         G00x400       500x400       450x400       400x400	DN V DN V DN NPS 6 V To 600 V
	K Cancel

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

#### Important:

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



## Files managed by HELiOS

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

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

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

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



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

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

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

## Create feature variant - Units

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

💌 Ui	nits and categories for variables	×
(j)	H1(length,mm) L(length,in) S(length,in) T(length,cm) The listed units and categories were assumed for the variables. If this is not correct, you can change the units and the categories in the Variant Editor.	
	ОК	

## PAA Editor - Units and Categories

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

Attribute text	Attribute value	Category	Unit	
MATERIAL: Designation				
MATERIAL: Material number				
/ORZUGSTYP	0 = no	<b>•</b>		- 1
DRUCK	0 = no 1 = yes	Unitless number		] ] [
IENNWEITE	20	Undefined	▼ (mm) _	
ENNWEITE2		Undefined	▼ (mm) _	
ENNWEITE3		Length	▼ ft ▼	
VANDDICKE		Density Elongation at break	▲ (mm) _	
VANDDICKE2		Force	(mm)	
VANDDICKE3		Length	(mm)	
Compatibility note: HEL_SACHNUMM	1ER will be transferred to BESTELLVER	MERK. Mass moment of iner Performance	rtia ea	

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

## Jacob Push-in pipes - Length and Weight

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

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

Article number: Project number: Folder number: Designation 1: Designation 2: Standard: Extended information Material: Weight: Dimensions: Comment:	SN-026218 Jacob Push-in pipe 1000 mm	Index: Release: Part type: Drawing/Manuf.: Unit of quantity: Resourcing:	In Progress
Designation 1: Designation 2: Standard: Extended information Material: Weight: Dimensions: Comment:	Jacob Push-in pipe 1000 mm	Release: Part type: Drawing/Manuf.: Unit of quantity: Resourcing:	In Progress
Extended information Material: Weight: Dimensions: Comment:	 4,30 kg	Unit of quantity: Resourcing:	
Material: Weight: Dimensions: Comment:	4,30 kg	Unit of quantity: Resourcing:	• •
		Order note:	Jacob Push-in pipe 1000 mm
Index			
Index creator:	Administrator 06.09.2023	Created: Origin: Based on:	06.09.2023 Administrator
	Pipe part properties         Wall thickness (2):       1,4         Schedule:       1         Pressure:       1         Supplied length:       0.5	5 mm 99 m	Fitting Preferred type: ▼ Arbitrarily divisible: 0 = n0 ▼ Accessory set: ▼

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

## Check for invalid nominal diameter matings

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

This applies to the functions:

Check for invalid nominal diameter matings (active pipeline) and

Check for invalid nominal diameter matings (entire drawing)

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

Properties Feature HCM Graphic Variables Check nominal diameter mating (PE)

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

💌 Nomi	nal diameter check	-		×
	a,			
	Name	Description		
4 1000 4 1000	$\odot$			
	🔺 25 Warnin	gs		
	~CompJoint_0005	The nominal diameter at connection 1 does not match the connected part.		
	SN-056438	The nominal diameter at connection 1 does not match the connected part.		
	SN-050716	The nominal diameter at connection 8 does not match the connected part.		
	~CompJoint_0007	The nominal diameter at connection 1 does not match the connected part.		
	SN-050716	The nominal diameter at connection 7 does not match the connected part.		
	SN-056438	The nominal diameter at connection 1 does not match the connected part.		
	SN-050716	The nominal diameter at connection 9 does not match the connected part.		
	~CompJoint_0004	The nominal diameter at connection 4 does not match the connected part.		
	SN-050716	The nominal diameter at connection 1 does not match the connected part.		
	~CompJoint_0003	The nominal diameter at connection 1 does not match the connected part.		
l				
			(	)k

# New versions of the EN 1092-1 flanges

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

	Plane flanges						
	previously		from HiCAD 2024				
	EN1092-1-01-PN10.vaa	E	EN1092-1-01-PN100_SF.vaa				
	EN1092-1-01-PN100.vaa	E	EN1092-1-01-PN10_SF.vaa				
	EN1092-1-01-PN16.vaa	E	EN1092-1-01-PN16_SF.vaa				
	EN1092-1-01-PN25.vaa	E	EN1092-1-01-PN25_SF.vaa				
	EN1092-1-01-PN2_5.vaa	E	EN1092-1-01-PN2_5_SF.vaa				
EN1092-1-01-PN40.vaa E		E	EN1092-1-01-PN40_SF.vaa				
	EN1092-1-01-PN6.vaa	E	EN1092-1-01-PN63_SF.vaa				
	EN1092-1-01-PN63.vaa	E	EN1092-1-01-PN6_SF.vaa				
	Loo	os	e flanges				
	previously		from HiCAD 2024				
	EN1092-1-02-32-PN10.vaa		EN1092-1-02-32-PN10_LF.v	'aa			
	EN1092-1-02-32-PN16.vaa		EN1092-1-02-32-PN16_LF.v	'aa			
	EN1092-1-02-32-PN25.vaa		EN1092-1-02-32-PN25_LF.v	'aa			
	EN1092-1-02-32-PN2_5.vaa	3	EN1092-1-02-32-PN2_5_LF.	.vaa			
	EN1092-1-02-32-PN40.vaa		EN1092-1-02-32-PN40_LF.v	'aa			
	EN1092-1-02-32-PN6.vaa		EN1092-1-02-32-PN6_LF.va	a			
	EN1092-1-02-35-PN10.vaa		EN1092-1-02-35-PN10_LF.v	aa			
	EN1092-1-02-35-PN16.vaa		EN1092-1-02-35-PN16_LF.v	'aa			
	EN1092-1-02-35-PN25.vaa		EN1092-1-02-35-PN25_LF.v	'aa			
	EN1092-1-02-35-PN2_5.vaa	9	EN1092-1-02-35-PN2_5_LF.	.vaa			
	EN1092-1-02-35-PN40.vaa		EN1092-1-02-35-PN40_LF.v	'aa			
	EN1092-1-02-35-PN6.vaa		EN1092-1-02-35-PN6_LF.va	a			
	EN1092-1-02-36-PN10.vaa		EN1092-1-02-36-PN10_LF.v	aa			
	EN1092-1-02-36-PN16.vaa		EN1092-1-02-36-PN16_LF.v	aa			
	EN1092-1-02-36-PN2_5.vaa	3	EN1092-1-02-36-PN2_5_LF.	.vaa			
	EN1092-1-02-36-PN6.vaa		EN1092-1-02-36-PN6_LF.va	a			
	EN1092-1-02-37-PN10.vaa		EN1092-1-02-37-PN10_LF.v	aa			
	EN1092-1-02-37-PN16.vaa		EN1092-1-02-37-PN16_LF.v	aa			
	EN1092-1-02-37-PN2_5.vaa	3	EN1092-1-02-37-PN2_5_LF.	.vaa			
	EN1092-1-02-37-PN6.vaa		EN1092-1-02-37-PN6_LF.va	a			
	EN1092-1-04-34-PN10.vaa		EN1092-1-04-34-PN10_LF.v	aa			
	EN1092-1-04-34-PN16.vaa		EN1092-1-04-34-PN16_LF.v	aa			
	EN1092-1-04-34-PN25.vaa		EN1092-1-04-34-PN25_LF.v	aa			
	EN1092-1-04-34-PN40.vaa		EN1092-1-04-34-PN40 LF.v	aa			

## New masks for part search in Plant Engineering

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

Search co	nditions ——							
Article St	traight pipe, ro	und						
	Connection 1 Nominal diar Outer diame Connection 1 Connection 2	neter: ter (1): type:			nm			
	Pipe part prope Wall thicknes Schedule:	erties ss (2): mr	Fitting Prefe Arbitr	rred type:	•			
	Pressure: Supplied len	igth: r	n Acces	ssory set:	-			
Search res	sult	•						
Article nur	mber In In	W Designation		Part type	Designation	Standard designation	Creation dat	Created by
N-026189		Jacob Push-in pipe	200 mm	Raw-part+Plant-desig		(JACOB_EINSCHIEBF	06.09.2023	Administrator
Te OLOTOS								
SN-026190		Jacob Push-in pipe	200 mm	Raw-part+Plant-desig		(JACOB_EINSCHIEBF	06.09.2023	Administrator

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

# Placeholders in imperial units

When inserting placeholders, units can now be used.

DN20		-				
DN32		- C				٦
DN50			(		100	
DN65			10		Tim	
DN80			( )			
DN100			1 0-	- farmer	1	
DN150			1	-		
DN200						
DN300			~	78		
DN500				*	2.2	
DN750						
DN1000						_
arameter	's ———					
' <mark>aramete</mark> r Name	Value				_	
<b>arameter</b> Name Length	Value 150mm					
<b>arameter</b> Name Length Height	Value 150mm 165mm					

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

- DUMMY\_VALVE.VAA (metric units, stored in mm) and
- DUMMY\_VALVE\_USA.VAA (imperial units, stored in inches).

Name	DUMMY_VALVE_USA		File selection	via file system	•	Derive new
Options	, HiCAD variants		Language	English (United States)	▼ (+/-)	Open
Part type	Valve	(	Industry	Standard Plant Engineering	-	
Designation	Platzhalter-Armatur		Value for		-	Edit repr.
Number of red	cords: 12			Variables + Attributes	_	Structure
						Import
	12					Export
-	D1	Type designation	Generate	DN20		Save
	K1 P	Variable Va	alue Unit	Description		Class
90			1 435 in	Cube: Height of edges	-	Close
4		A01	1.435 in	Cube: Width of edges	- 1	End
		A1	4.000 -	Flange: Number of bores	_	Record
	A0 × A01	■ B1 ■	0.709 in	Flange: Width	-	
		D1	4.134 in	Flange: Diameter	-	
		D11	1.063 in	Pipe handwheel: Diameter		
	<u></u>	D12	0.551 in	Flange: Bore diameter	-	Find
7		<b>D</b> 2	1.181 in	Pipe valve: External diameter	-	List
	V	D6 0	3 937 in	Handwheel: Diameter	_	

# **Notes on HELiOS Updates**

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

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

#### Data backup before updating

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

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

#### • Log file for update

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

#### New mask format

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

#### MultiCAD interfaces

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

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

# **HELiOS Desktop**

# Service Pack 2 2024 (V 2902)

## Performance

With the update to Service Pack 2 of HELiOS 2024, further performance improvements have been achieved. The structure of result lists in the **Utilized items** context, for example, is now built up to 10 times faster.



# Language selection in Login dialogue

As of HELiOS 2024 Service Pack 2, you can select the user interface language when logging into the HELiOS Desktop.

Administrator	
은 PDM Administrators	-
🗌 Skip login	<b>■</b> •
	End Lo

## HELiOS Options: Save settings centrally

At the bottom of the **HELiOS Options** window, you can use the **Manage...** button to easily distribute individual settings (not only) of the HELiOS user interface from one system to other users or workstations.

New is the **Save settings centrally...** menu item, with which you can save the export of the settings directly in the central directory for the distribution of settings, which can be set up during the installation of HELiOS, for example.

	×	HELiOS Options
🕖 General 🔍 Find	Settings for import a	and export
Input Display Result lists Project and Folder structures Print / Convert	- Import Allow import on document No Yes and keep model/o Yes and delete model - Export	masters whose file was originally saved within another application: document structure /document structure
Import / Export	Show configuration dialogue	e when exporting
생 Workflow (2월) Log 조카 E-mail	<ul> <li>Files:</li> <li>Files of project:</li> <li>Files of folder:</li> </ul>	Settings Settings Settings
Preview	Generated file name for the e	export of
Database	Files:	Standard 👻
Document type	Files of project:	Standard 👻
HELiOS Desktop	Files of folder:	Standard 🔹
	Files via drag & drop:	Standard 👻
Manage •		Close Apply
Import settings		
Export settings		

## Accept active project / folder

The dialogue windows for selecting a project or folder have been extended to include the option of directly transferring the project or folder currently active in HELiOS with a single click.



# Structure in the project detail window

The new Structure tab of the project detail window shows you the sub-projects of the active project.

	🔋 PN-01-06-K (Project)	– 🗆 ×
General Assignments	Input / Output	* 🖻 🔿
Activate Edit Edit Workfly Genu	dit w status Roles Attribute values Roles Attribute values Find Combined search Find Combined Search Edit Combined Search Edit Combined Search Edit Combined Search Edit Combined Search Edit Combined Search Edit Combined Search Edit	e
Mask $ imes  $ Document	assignments $ imes  $ Article assignments $ imes  $ Folder assignment	ts × Structure ×
🔍 Find		
PN-01-06-K PN-01-06	м	<b>↔</b>
Name	Designation	
✓ ]] PN-01-06-K	Construction Documents	
🔁 PN-01-06-M	Mounting drawings	
PN-01-06-P	Production drawings	
# Convert file

For document masters, you will now find the new function **Convert file** under **Output**:

Docur	nents X	4	Create variant index	
4. 8			lmport file	
	🔄 🚰 🍓 Stan		Output	
0 0	Document numb		Show Version Management	nt type
	PDN-000001		Export file	awing
	20N-000002		a export me	rt/Variant
8 07	PDN-000003		Send file by e-mail	awing
	2DN-000004	Ę	Print (Spooler)	rt/Variant
3 5	PDN-000005	E	Print (Windows default printer)	awing
2 B	PDN-000006	-	a mine (windows delaan printer)	awing
1 B	PDN-000007	a	Convert file	awing
3 3	2DN-00008		Call report	rt/Variant
	PDN-000009	#	Conversion	awing
30	2DN-000010		Copy number	rt/Variant
80	PDN-000011	4	Copy HELiOS URL	awing
8 3	PN-000012	i i i	Show ID	rt/Variant
8 5	PDN-000013	Ē	Edit	awing
8 3	2DN-000014	å ,	Delete	rt/Variant
80	PDN-000015	_0	Delete	awing
R	2DN-000016		3-D model HiCA	D Part/Variant

With the help of this function, files saved in HELiOS can be saved in other file formats, even with applicationspecific options. A special selection dialogue is offered for this purpose.

	official for conversion.	
Format	Description	Conversion with
DWG	Conversion to DWG	Autocad
DWF	Conversion to DWF	Autocad
DWFX	Conversion to DWFX	Autocad
IGES	Conversion to IGES	Autocad
JT	Conversion to JT	Autocad
STEP	Conversion to STEP	Autocad
STL	Conversion to STL	Autocad
PDF	Conversion to PDF	Autocad
SAT	Conversion to SAT	Inventor
PNG	Conversion to PNG	Inventor
JPG	Conversion to JPG	Inventor
BMP	Conversion to BMP	Inventor

The export dialogue for the documents to a specific target directory is similar to the one you know from the functions Print (Spooler) or Export.

Output directory:	C:\Temp					
	Output to sub-	directory:		41		
utput converted	files:					
	- HoliosConvertedA	rehine 2024-04	17 14 02			
	HellosConvertedA	Archive-2024-04	·1/_14-02			
_						
🔊 🧑 Standard	•					
onvert O	Document number	In File size	Generated file name	Article	Project	Folder
Image: Section 1.	DN-000341	334,00 KB	DN-000341	SN-026377	3 20230830-1-1 🛛 🛞	
Image: Section 1.	DN-000342	323,50 KB	DN-000342	SN-026378	📀 20230830-1-1 🛛 📀	

The old **Convert** function and its sub-menu that existed in previous HELiOS versions **Output** have been removed.

Updated settings files for the conversion are supplied with new installations of the Spooler. Existing settings files are automatically updated after an update the first time the Spooler is accessed.

# Configuration of file names when printing and converting

In the HELiOS Options on the Print / Convert tab you can find new setting options:

Attribute assignments for the file name generation can be set for

- the output of Neutral formats via the HELiOS Spooler and
- the **Conversion** of documents

	<b>K</b> HELIOS Options						
📰 General 🔍 Find	Settings for Print / Convert						
🚮 Input	- Print documents (Spooler)						
🚰 Display	Print directly without calling up a dialogue if the input is unique Pre-definition of sheet areas to be printed:						
Result lists							
Project and Folder structures	User-defined     All     As last saved     Include model area  Final check for linked articles:     Must be located in the same selected project as the document to be printed     Must be located in the same selected folder as the document to be printed     Must have been selected if available Notes document:     Document must be explicitly selected if a notes document for it has been created or accepted						
Print / Convert							
Import / Export							
Workflow							
🚰 Log							
E-mail							
Preview							
Database							
Document type	File name for neutral format: Standard						
HELiOS Desktop	- Print result lists						
	Restart the Report Manager every time you print						
	- Convert documents						
	Final check for linked articles:						
	Must be in the same project as the document to be converted						
	Must be in the same folder as the document to be converted						
	Generated file name. Standard						
Manage •	Close Apply						

# Generated files names and configuration dialogues for export

Please note further new features and rearrangements in the HELIOS Options:

Under **Settings for import and export**, you can set the attribute assignment for the generated file name assignment for each of the following export processes individually:

- Files: Export of files, for example via the context menu of a document results list or from a document detail mask, also when exporting files via the API and when downloading files via the HELiOS Internet Server.
- Files of project: Export of the files of a project via Output > Export files of project...
- Files of folder: Export of the files of a project via Output > Export files of folder...
- Files via drag & drop: The assignment that you can set here then takes effect when exporting via drag & drop of document masters from HELiOS to the file explorer.

	1 2	
🗹 Files:	Settings	
Files of project:	Settings	
Files of folder:	Settings	
Generated file name for the e	export of	
Generated file name for the e Files:	export of Standard	•
Generated file name for the e Files: Files of project:	Standard Standard	•
Generated file name for the e Files: Files of project: Files of folder:	Standard Standard Standard Standard	•

A click on **Manage...** in one of the selection lists opens the **Attribute mapping for generating file names** dialogue window, allowing you to create new mappings or edit existing ones.

💥 Attri	oute mappings for	generating file names.
Attribute assignments:	Standard	- 🛃 🗡 🗎 🖬
		Close

A display of the set central directory for the distribution of settings can also be found at **HELiOS Options > General**.

# Export project files: Indices

In the Settings for export dialogue of the Export files of project.... function, a new option, Export all indices of a document, has been added:

Document	
Reserve exported documents also dir	rectly
Export also referenced documents fro structure	om document or model
File name	
<ul> <li>Attribute assignments</li> </ul>	
🗹 Replace invalid characters with un	nderscore (\/:*?<> ").
Individual specification	
O From document or model structure	
- Structure	
Create project as superordinate direct	tory
Also export sub-project structure wit	h content
- 1 1 2	
Export all indices of a document	
Export all indices of a document     Suppress dialogue	

If this option is deactivated (default setting), only the newest indices of project files are exported. If the option is activated, all other indices of a corresponding document are offered for selection/deselection in the export dialogue:

*2 🖂	Stand	ard 🝷	•
Export	0	Document number	Index
		DN-000209	
		PN-000216	
		DN-000210	
	1	PN-000001	
		2DN-000006	
		20000006 DN-000006	
<ul> <li>Image: A start of the start of</li></ul>	1	DN-000213	

## Export folder files: Indices

In the Settings for export dialogue of the Export files of project...

- If several indices are assigned, only export the highest assigned index: This option is active by default, meaning that only the highest index of a document assigned to the folder is exported. If the option is deactivated, the following export dialogue offers all indices assigned to the folders of a corresponding document for selection/deselection.
- Only export documents whose most recent index is also assigned to the respective folder: If an even higher index exists for a document assigned to the folder, which is not assigned to the folder, the assigned document with the lower index is still taken into account when calling Export files of folder... will still be taken into account. If this option is activated, documents in the folder are only included in the export dialogue if they are also assigned to the folder in their highest index. If a higher index exists "outside" the folder, documents with an older index are not taken into account.

Document	
Reserve exported docur	ments also directly
<ul> <li>Export also referenced of structure</li> </ul>	locuments from document or model
File name	
Attribute assignments	
🗹 Replace invalid chara	acters with underscore (\/:*?<> ").
Individual specification	
O From document or mod	del structure
Structure —	
Create folder as superor	dinate directory
🗌 Also export sub-folder s	tructure and content
If several indices are ass index	igned, only export the highest assigned
Only export documents to the respective folder	whose most recent index is also assigned
) Suppress dialogue	

#### Sheet structure, Model structure and Document reference up-to-dateness

Please note the following changes and enhancements when transferring the HiCAD model structure to HELiOS:

In previous HiCAD/ HELiOS versions, the sheet structure, i.e. the available sheet areas of an SZA document, was displayed in the Model structure tab of the document master.

As of HELiOS 2024 Service Pack 2, you will find the separate **Sheet structure** tab in the document detail mask for this purpose.

0	DN	-00	0013 (Document), Pr	oje	ct context: PN-01-06-K	-	□×
General Assignments Inp	ut	0	itput			1	r 🖪 🔿
		7					
Edit Preselect Open master data + Edit + W	E Vorkflo	dit ow s	Roles Attribu tatus value	l ute es	Notes Mark-up document + + Edit		
≪ Sheet structur × Model structur ★ Pii Pii → Standard	tı ×	So	ources × Targ	jets	imes   Utilized article $ imes  $ Graphic $ imes  $ Use		×»
Sheet structure	0	0	Document number	In	Designation		
✓ DN-000013	2	B	DN-000013		Production drawing		
🗊 3-D model							
🔛 Sheet 1 (Main views)							
Sheet 2 (Second view)							

The **Model structure** tab in turn displays the document structure of HiCAD files (.SZA, .KRA). In other words, it displays the document masters that were referenced by the corresponding SZA/KRA document.

😚 🏭 🏭 M Standard			Ţ			
Document	0 <sup>,</sup>	0 <sup>.</sup>	In	In	W	Designation
<ul> <li>DN-000001</li> </ul>		B		•	0	Draft
<ul> <li>DN-000002</li> </ul>		B		•	0	3-D model
DN-000016		B		•	0	3-D model
DN-000018		B		•	0	3-D model
DN-000020		B		•	0	3-D model
DN-000022				•	0	3-D model
DN-000024		B		•	Ð	3-D model
DN-000004		B		•	0	3-D model
DN-000008		B		•	0	3-D model
DN-000010		B		•	0	3-D model
DN-000012		B		•	0	3-D model
DN-000014		By		•	0	3-D model
DN-000034		B		•	0	3-D model
DN-000035		B		•	0	3-D model

The new database attribute **UI\_ReferencelsOutdated** is also available in the result list configuration of the model structure.

You can use this attribute to add the **Document reference up-to-dateness** column to the model structure result list.

Edit Aster d	lata el str	Assig Prese	elect Stan	nts Input Output	Attribute values Votes document · Mark-up · Mark-up · Mark-up · Mark-up		elete dit Jse	★ E
	- Av	ailabl :	e attr	ibutes	dit att	- -	e con	Configuration     Selected attributes     List     GR Context menu
, , , , , , , , , , , , , , , , , , ,				Designation Open, read-only Open + Edit Index Index up-to-dateness Workflow status Designation Document reference up-to-datent Article number Based on CC Changed on Changed on Changed on Changed on Check list Checkel by Checked on	Attribute name         UI_OpenItemReadOnly         UI_OpenEdititem         HEL_INDEX         UI_DocumentNewerIndex         VA_DocumentReleaseStatus         BENENNUNG         ERSATZFUER         CC         HEL_GEAENDERT_USER         HEL_GEAENDERT_USER         HEL_GEAENDERT_OATUM         HEL_LINKED_DATE         VA_ObjectWorkflowCheckL         GEPRUFT_NAM         GEPRUFT_DAT			Result list display:       Standard (Helios.RL.DocumentStructureView)          Type       Designation         Open, read-only       Open + Edit         Index       Index         Workflow status       Open jesignation
	Attr For the Sł	ibute a refe time now c	descr erence of refe	iption: , it is displayed whether there is a n rencing. n for attribute names	ewer version of the reference since			Width of selected column (px):

DN-000255 (Document), Project context: HiCAD-1123

– 🗆 ×

General Assignments I	nput Out	put				
Edit master data	Edit Workflow st	Role atus	Attribute values do	Notes cument •	Mark-up Edit	
Mask $\times$ Sheet structu	re × Mode	l structure	× Sources	× Targ	ets $ imes$ Utilized a	rticles $ imes$   Graphic $ imes$   Use
🐓 📲 🔭 😹 Standard		•				
Document	Open, r	Open In	Index up-to-da	Workflow	Designation	Document reference up-to-dateness
<ul> <li>DN-000255</li> </ul>		<b>D</b>	•	0		NOT ASSIGNED
DN-000256_ref	2	6	•	•	Part document	NOT ASSIGNED
DN-000258		6	a 🔴	0	Part document	NOT ASSIGNED

The following three states that a HiCAD reference can assume are then displayed in the result list:

- Not assigned Marks the display of the top part in the document structure as well as references for which no version information has been saved.
- Current: The referenced document has not been changed since the document structure was transferred. The reference is therefore up-to-date (please note that there may be outdated sub-parts for an assembly despite the reference being up-to-date).
- Old: The referenced document has been changed (and saved) since the document structure was transferred).



# Output product structure to Report Manager: Project context

Note the following behaviour adjustment when transferring the product structure to the Report Manager via the

Output functions

Dutput to Report Manager... and Cutput to Report Manager, new window... :

If there is an unambiguous project context for the top article, this is also transferred. If this is not the case, the currently active project is transferred.

Project Explorer Folder Explorer Proc	duct E	xplorer × Article Class Explorer Do	ocument Class E
🔍 Find			•
<ul> <li>SN-000001, , Slip-on gear mechanism, </li> </ul>			
> 🚫 1, 1, SN-000005, , Pinion assembly,		General	
> 🚫 2, 1, SN-000004, , Gear wheel assen		Add top article (to window)	•
<ul> <li>3, 1, SN-000003, , Housing assembl</li> </ul>	X	Remove top article (from window)	
👋 👔 1, 1, SN-000006, , Gearbox hou:	1	Reserve item	
🝈 2, 1, SN-000007, , Gearbox cove	Th.	Apply item edits	
🕥 3, 1, TN-64188, , Circlip for bore	Ťκ.	Cancel item editing	
Ѽ 4, 1, TN-64179, , Circlip for bore	°F	Show article	
Ѽ 4, 1, TN-64179, , Circlip for bore	٢	Edit article master	
🔯 5, 1, TN-33436, , Hexagon sock	*	Add to favourites	tandard part,
🔯 5, 1, TN-33436, , Hexagon sock		Input	tandard part,
🔯 5, 1, TN-33436, , Hexagon sock	10	Enter item (via Find article)	tandard part,
💭 5, 1, TN-33436, , Hexagon sock	N.	Enter item (via Enter article)	tandard part,
🕥 5, 1, TN-33436, , Hexagon sock	₽Ę.	Derive item	tandard part,
🕥 5, 1, TN-33436, , Hexagon sock	_	Output	tandard part,
🕥 5, 1, TN-33436, , Hexagon sock		Output to Report Manager	tandard part,
5, 1, TN-33436, , Hexagon sock		Output to Report Manager, new window	tandard part,
> 🕥 4, 1, SN-000002, , Clamping elemer		Output to new document	
💱 5, 1, SN-000015, , Sealing washer, ,	1	Output to existing document	
💱 6, 1, SN-000016, , Sealing washer, ,	19 B	Output structure versions	
💱 99, 1, SN-000017, , Lubricant RG120	1	Call report	

#### Product structure: Management of editing locks in the Locking Manager

In previous HELiOS versions, locks were set at user level when editing product structures. The corresponding sessions could be displayed in the Attribute Editor (Advanced functions > HELiOS Session).

With the update to Service Pack 2 (HELiOS Version 2902), the **1-level assemblies** tab has been added to the **Locking Manager** tool.

This tab shows the top parts of the product structure that are being processed so that they can be unlocked there if necessary.

Articles	X Documents X Folders	× Projects	× 1-level assem	blies $\times$
4 💌 💩	Standard 🔹	Unlock selected article(s	5)	
Article numbe	Designation	Host computer	Locked by	Locked on
	Clamping element	DEDTM066	Administrator	14.01.2022 09:57:39
N-000002				I HOTHEORE OFFICIES
1-000002	Clamping element			
N-000002	Camping element			
N-000002	Clamping element			
N-000002	Clamping element			
SN-000002				

The corresponding button in the attribute editor has been removed.

#### Permitted characters for user names

Please note that the characters permitted for user names have been adjusted and restricted. Punctuation marks such as comma, apostrophe, square brackets etc. and special characters such as @ cannot be used in user names.

A database version update from an older version to HELiOS 2024 Service Pack 2 or higher replaces invalid characters with spaces (except at the start and end of the character string, where they are simply removed).



The pre-Windows 2000 login name "a.\" contains at least one of the following characters: / []:; | = , + \* ? < > @ " If you continue the process, the invalid characters will be replaced with underscores (\_). Would you like to continue?

## Complete decimal representation of FLOAT attributes in the product structure

By default, HELiOS displays the decimal places configured in the Attribute Editor for FLOAT attributes. This may result in many zeros being displayed as decimal places.

In result lists, in addition to the display with a fixed number of decimal places, a complete display can also be selected for FLOAT attributes, which lists all decimal places but has no zeros at the end.

🔎 Dec	imal places	×
Con	nplete	
03		
	ОК	Cancel

If, in this case, the FLOAT value does not have any decimal places other than 0, the decimal separator is also omitted from the result list display.

For the display of FLOAT attributes in the product structure, it is also possible to use the complete display instead of the setting for the decimal places from the Attribute Editor.

To do this, you must edit the system file **pv\_konfig.dat**.

In future HELiOS versions, the technology of the HELiOS product structure will be revised and this manual step will no longer be necessary.

# Service Pack 1 2024 (V 2901)

#### Search in sub-projects/-folders

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

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

A 0.

	K HELiOS Options
General	Settings for the search of records
🔍 Find	
🛃 Input	- General
🚰 Display	Offer search in sub-projects/-folders for:
Result lists	Find document
Project and Folder structures	Find article
Articles X	
🖘 🛄 🐱 🍓 Sta	ndard 🔹 🖌 🦹
Article r Show content	related to selected project
Articles X	
😽 🐱 🌆 Standard	- 7 K
Article number In In	W Designation

## Please note:

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

If the options for searching in sub-projects/folders have been deactivated, this also affects the search templates. If the user selects a search template in which the search was originally used in sub-projects / subfolders, the search will no longer take place in the sub-projects / sub-folders despite the selection of the search template. In the case of results list templates, the search in sub-projects / sub-folders is only controlled via the **With substructure** checkbox. The corresponding setting from the search template is irrelevant here.

Name	Result list display	Search template	W	/ith sub-struc	ture
Standard	Standard (Helios.RL.Article )				
Articles (recursive)	Standard (Helios.RL.Article )	Articles (recursive)	•		
Articles (only current level)	Standard (Helios.RL.Article )	Articles (only current level)	•		
Articles (recursive/template)	Standard (Helios.RL.Article )	Articles (only current level)		<b>v</b>	

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

🐈 📉 👚 뒞 Double click a row with a result list template to change it.		
Name Result list display Search templ	late	
Standard Standard (Helios.RL.Article )		
Articles (recursive) Standard (Helios.RL.Article ) Articles (recur	rsive)	0
Articles (only current level) Standard (Helios.RL.Article ) Articles (only	current level	) 🌍
Articles (recursive/template) Standard (Helios.RL.Article ) Articles (recur	rsive/templa	te) 📀

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

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

When using the functions:

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

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

cle			
Basic information			
Article number:	SN-026247	Index:	
Project number:	PN-01-06-K, Construction 🛄 🏮		1.2.N
Folder number:	Folder-independent 🗔 📁		
Designation 1:	Pinion -	Release:	In Progress
Designation 2:		Part type:	Individual part 🔹
Standard:		Drawing/Manuf.:	
Extended information			
Material:		Unit of quantity:	pcs 🔹
Weight:	0.317941 • kg	Resourcing:	Self-manufacturing 🔹
Dimensions:		Order note:	
Comment:			×
-			

The workflow of the original object is preset.

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

If it is desired that the workflow of the original object is always adopted without giving the user a choice, administrators can set the Database option **Apply workflow of last state for revision** to **No**. The workflow selection will then restricted for the above functions in such a way that only the workflow of the original object will be applied.

Display		
	General T LICAD	
Posult lists		
Result lists	General settings	CAD System Local Session
Project and Folder		
	Global generation rule for file names and numbers	No
Print	Revise the new document.	No
Import/Export	Print only released documents	No
mpontexport	Optional Drag _Drop for follow-on sheet	No
Workflow	Vault Server	Yes
	Allow forced releases	Yes
Log	Test statuses in role workflows can be ignored	No
	Link E-mail	E-Mail Attachment
Database	Apply workflow of last state for revision	No
Decument true	Issue error message for undetected master data objects	No
Document type	Article attribute for defined weight	Yes
HELiOS Desktop	Article attribute for surface area	FLAECHE
	Article attribute for material (weight-relevant)	MATERIAL
	Article attribute for body volume (weight-relevant)	
	Article or material attribute for specific weight	RHO
	Master link for product structure	
	Default language	en
	Default configuration for product structure output	PRODUCT_Stahlbau_01.m_settings
	Attribute links in search masks	No
	Multiple classification for articles	No
	Consider only active group when determining user rights	No
	Update PDM items during product structure sync	No
	Allow updating of released product structures	No
	Changeability of old indices by Administrator	No
	Article master, Part type for semi-finished products	Raw-part+Plant-design
	Default settings for date format	DD.MM.YYYY hh:mm:ss
	Allow assignment of objects to read-only projects	Yes
	Changeability of document attribute HEL_ORIGINALPATHNAME by administrator	Yes

#### Deleting and renaming attribute assignments

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

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

e:	Standard	- 🛃 🏹 📄
Con	🕂 Rename	
_	Standard	nverter
	Cancel	ОК

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

## Please note:

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

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

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

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

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

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

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

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

	HELIOS import tool for settings	? – 🗆 🗙
	Import setting	
Settings file: D:\install_temp\configfiles\export	12-12-2023.zip	
Central directory: D:\install_temp\configfiles		
Import to central directory		
		Import Import all
	15	

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

Extended settings		- 0	×
HELIOS HELIOS Inter	faces General		
System data source	HeliosBauteil		
User	hicad		_
Password	••••		
✓ Use Vault Server	Address localhost	Port 9000	
✓ Use Spooler	Server localhost	Port 35725	
Use central direct	ory for configuration data		

# Major Release 2024 (V 2900)

#### Improved user interface

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

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

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

<b>⊡</b> Export	
Select the components that you want to export:	
Tab layout and positions The tab layouts and window positions (for e.g. Project Explorer, Favourites list, etc.) in the HELiOS Desktop main window.	
Tab layout The tab divisions within the windows (e.g. tabs in the detail dialogue, document tabs in the Project Explorer, etc.).	
All adapted masks	
Result list templates Result list templates and contained result list representations for all result lists	
Attribute assignments 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	
Context menu edits Customizations of the context menus differing from defaults	
HELIOS Options Special settings from the HELIOS options or various areas	
General Settings	
Settings for the search of data records	
Input of data records	
Display 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	
Settings for workflows	
Settings of directories for logs	
Elios Desktop Settings for the HELiOS Desktop	
	ОК

# HELiOS Options: Search

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

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

	HELIOS Options
General	Settings for the search of records
Input Display Project and Folder structures Print	- General         Offer search in sub-projects/-folders for:
MANAGE •	CLOSE

#### HELiOS Options: Selectable tabs

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

Wir	ndow type:	Show document	•	
	Tab			
•	Mask			
<	Graphic			
$\checkmark$	Targets			
$\checkmark$	Sources			
$\checkmark$	Model stru	icture		
$\checkmark$	Sheet strue	cture		1
	Utilized art	ticles		

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

🚯 SN-000003 (Article), Product context: SN-000003 — 🗆 🗙	
General Assignments Input / Output 🖈 🖾 🔿	
Edit     Preselect     Edit     Roles     Attribute values       General     Edit     Edit	
« Graphic × Targets × Use × Sources × Product structur × Utilized articles × 🕂 💌	
Q. → ↓a 1	Mask
47	Graphic
<ul> <li>SN-000003, Housing assembly, In Progress, Assembly,</li> <li>1, 1, SN-000006, Gearbox housing, In Progress, Individual part, ., 2.58</li> <li>2, 1, SN-000007, Gearbox cover, In Progress, Individual part, ., 2.15</li> <li>3, 1, TN-64188, Circlip for bore, DIN 472-47x1.75, In Progress, Standard part, .,</li> <li>4, 1, TN-64179, Circlip for bore, DIN 472-32x1.2, In Progress, Standard part, .,</li> <li>4, 1, TN-64179, Circlip for bore, DIN 472-32x1.2, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> <li>5, 1, TN-33436, Hexagon socket cheese head screw, ISO 4762-M 6x20-10.9, In Progress, Standard part, .,</li> </ul>	Targets Use Sources Product structure Utilized articles

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

# **HELiOS Options: Classification**

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

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

- Classification -
- Allow classification of documents
  - Show classification tabs in Find document dialogue window
- Allow classification of articles
  - Show classification tabs in Find article dialogue window



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

#### Improvements and extension of the HELiOS URLs

The functionality of the HELIOS URLs has been revised and improved.

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

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

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

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

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

#### Export file

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

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

Export file								×
The file name can be set for the se cannot be influenced.	lected d	ocument mast	ers. The file name for further	files, which	are also considered e	.g. due to the do	cument or model stru	ucture,
- Type of export								
• Export	C:\Use	rs\sobaempetm	acit\Documents					
O Send by e-mail								
Options:								
As ZIP archive	ZIP arc	hive name						.zip
With export information								
4 T								
Standard	•							
Export O Document num	ber In	File size	Generated file name	Extensio	Article		Project	
Image: Second		501,75 KB	DN-000003	sza	SN-00002	8	PN-01-06-K	Q
Image: Second state of the second state of		400,92 KB	DN-000004	kra	SN-000003	😣	PN-01-06-K	0
Image: Second		304,38 KB	DN-000005	sza	SN-000003	8	PN-01-06-K	0
								Þ
Number of records: 3								
Generated file name: Standard		•	Z Replace invalid character	rs with unde	rscore.		CANCEL	ОК



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

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

Functions	Assignment	Edit
Create DXF	Group	
Export	Group	
Move projects	Group	
Change classification structure	Group	
Change document class assignment	Group	

There is no separate user function for import.

# Export files of projects / Export files of folders

In the context menus of projects and folders you will find the new functions **Export files of project**  $\bigcirc$  or

#### Export files of folder and under Output.

r

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

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

Project Exp	plorer × Fol	der Explorer	Product Explorer	Article Class Explorer	Document Class Explore	er ToE	Do	
Final PN-01-	-06-K (Project)		<b>-</b> 14	+			×	
General	Assignments	Input / Or	utput			*		
Enter De	erive, with Deriv content co	re, w/o ntent	Print docum of project	t Copy Copy number HELIOS URL	JD ID			ty
Mask	× Docum	ent assignmen	ts × Article assignmen	ts × Folder assignments ×				
				-				-
								-
								-
	Basic informati	on						
	Destant	DNI 04 00				_		-
	Project:	PN-01-00	o-K		1.5.1			-
	Assignment	:						
	Designation	Construc	tion Documents					E
	Project type	Order		Project Manager	: Buchmann			F
	Comment:							
								l r
	Customer							н
	Name:	ISD Softv	vare und Systeme Gmbł	H Customer No.:	1000			н
	Street:	Hauert 4		E-mail:	hotline@isdcad.de			н
	Code/Tourn		44227 Dortmund					н
	Couerrown.		44227 Dominand					H
								H
								11
								Н
								HHH
								HHHH



# Send file by e-mail: As Zip archive

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

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



#### HELiOS Options: Attribute assignment and Export settings

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

These can then be selected in the export dialogue.

Current.	Cill	Heare\ of Laroy atros					
Send by e-mail	Citt	oscialized certification	and bocuments				
Detioner							
D As 7ID archive	ZIP	archive name					
As ZIP archive		an entre manne					
With export in	rormation						
🔊 👿 Standard	-	•					
Standard port O Doc	• ument number	In File size	Generated file name	Extension	Article		Project
Standard	ument number N-00002	In File size a 1172,04 KB	Generated file name DN-000002-a-PN-01-06-K-SN-000002-	Extension kra	Article SN-000002	8	Project PN-01-06-K
Standard port O Doc	ument number N-00002	In File size a 1172,04 KB	Generated file name DN-000002-a-PN-01-06-K-SN-000002-	Extension kra	Article SN-00002	3	Project PN-01-06-K
port O Doc	ument number N-000002	In File size a 1172,04 KB	Generated file name DN-000002-a-PN-01-06-K-SN-000002-	Extension kra	Article SN-000002	8	Project PN-01-06-K
Standard     Standard     O Doc     O	ument number N-000002	In File size a 1172,04 KB	Generated file name DN-000002-a-PN-01-06-K-SN-000002-	Extension kra	Article SN-000002	8	Project PN-01-06-K
Standard port O Doc C C C C C C C C C C C C C C C C C C C	ument number N-000002	In File size a 1172,04 KB	Generated file name DN-000002-a-PN-01-06-K-SN-000002-	Extension kra	Article SN-000002	S	Project PN-01-06-K

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

– Attribute assignments ———			
Generation of file names for output:	Configure		
Attribute assignment for file name:	Standard	•	
	Standard		
	ISD_Mapping_02		

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

At HELIOS Options > Import/Export you can also set for Export file, as well as for the new functions Export

files of project  $\textcircled{B} \xrightarrow{}$  and Export files of folder  $\textcircled{D} \xrightarrow{}$ . In each case, or the export settings dialogue should be executed or skipped for each export process.

#### Please note:

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

# Local changes during export, printing and conversion

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

	expoi	rt ———							
Export			C:\L	lser	s\.organawetal	\Desktop\Test			
O Send b	y e-m	nail							
Options:									
🗹 As Z	ZIP ar	chive	Heli	osE	xportedArchiv	ve-2023-09-14_14-38			,zi
🗹 Wit	h exp	ort information							
5 💌	Stand	ard	•						
xport	0	Document nur	nber	In	File size	Generated file name	Extension	Article	Project
<ul> <li>Image: A start of the start of</li></ul>		DN-000320			0,00 KB	DN-000320	txt		PN-01-06-
	f reco	rds: 1	_					_	
umber of									
lumber o	nent		Error						

You can then decide whether to **Cancel** or to export the document with **OK** regardless of the local changes. The same applies to **Print (Spooler)**, **Send file by e-mail** and **Convert**.

# Result list - Target date display for projects and articles

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

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

חו		ment	18	Article	Folder
			Type	Designation	Attribute name
1		(Stab)	ijpe I	Project type	PROJECT TYPE
it				PROJECTI OCK	PROJECTLOCK
i				Responsible person	SACHBEARBEITER
			1	Roles	VA_ObjectWorkflowRoleStatus
			۲	Roles	VA_ObjectWorkflowRoleStatusInfo
				ROOTPROJECT	ROOTPROJECT
				SCHUTZGAS	SCHUTZGAS
1				SCHWEISSNAHTVORBEREITUNG	SCHWEISSNAHTVORBEREITUNG
1				SCHWEISSVERFAHREN1	SCHWEISSVERFAHREN1
1				SCHWEISSVERFAHREN2	SCHWEISSVERFAHREN2
1				SCHWEISSZUSATZ1	SCHWEISSZUSATZ1
1				SCHWEISSZUSATZ2	SCHWEISSZUSATZ2
1				Street (Kunde)	STRASSE (ARCHITEKT)
1				Street (Kunde)	STRASSE (BAUHERR)
				Street (Kunde)	STRASSE (KUNDE)
				Street (Kunde)	STRASSE (STATIKER)
	<b>v</b>		۲	Target date	VA_ObjectWorkflowTargetDate

n	- 41/21	lable :	ttribute	×	
04	- Avai			3	
04	Filter:				
05	O Ar	ticle			
05			Type	Designation	Attribute name
05			1	Seal	DICHTUNG
05				Section modulus WY	MOMENT WY
05				Section modulus WZ	MOMENT_WZ
05				Session ID	HEL_SESSIONID
05				Sheet thickness	PLATETHICKNESS
05			1	Source or target link	VA_ObjectLinks
05				HiCAD hatching (Werkstoff)	SRAF (MATERIAL)
06				Standard	NORM
06				Standard part designation	HEL_AUSPRAEGUNGID
06				Standard part designation	HEL_NORMID
06				Status (Werkstoff)	STATUS (MATERIAL)
00				Supplied length [m]	LIEFERLAENGE
06				Surface [mm <sup>2</sup> ]	FLAECHE
06				Surface	OBERFLAECHE
06				Surface area per length [mm <sup>2</sup> ]	FLAECHE_M
06				TABLE3 (Werkstoff)	TABLE3 (MATERIAL)
07			1	Target date	VA_ObjectWorkflowTargetDate
07 07 07 07	Attrib Show	oute de rs an io	escription on if a t	n: arget date has been defined for the	workflow of the object.
07 04	Sho	w col	umn for	attribute names	

47				Star	ndard	•	7 7	Deres I	
Artic	le nu	ımbei	In	dex	Index up-to-date	Wor	kflow status	Part type	Target date

# User rights: Move folders

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

Functions	Assignment		Edit
Customize interface configuration	Group		
Work project-independent	Group		
Work folder independent	Group	1	
Move folders	Group	1	



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

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

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

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

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

## Initialised attributes

Note the instructions for initialising attributes:

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

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

## Workflow selection for article index

As with indexing of individual articles or documents, it is also possible to make a workflow selection for articles



			Index informatio	n						
			Index creator:		Administrator					
			Index date:		14.09.2023					
			index dator				Land	-		
			Index text:							
		1								
2										
	documents	to be revised an	d specify for whi	ch d	locuments a dialogue	isto	he dis	played. Index texts entered	there will be applied to do	ruments
elect the o	documents Standard	to be revised an	d specify for whi	ch d	locuments a dialogue	is to	be dis	played. Index texts entered	d here will be applied to do	cuments.
elect the o	documents Standard	to be revised an	d specify for whi	ch d	locuments a dialogue	: is to	be dis	played. Index texts entered	I here will be applied to do	cuments.
elect the of the	documents Standard Display	to be revised an	d specify for whi • Ō	ch d Öʻ	locuments a dialogue Document number	is to	be dis	played. Index texts entered	I here will be applied to do Name of link class	cuments. Creat
elect the offer	documents Standard Display	to be revised an	d specify for whi Ö	ch d Ö	Documents a dialogue	is to	be dis In W	olayed. Index texts entered c Designation Exploded view	I here will be applied to do Name of link class Pauteil-Konstruktion	Creat 03.10.
elect the of the	documents Standard Display	to be revised an	d specify for whi Ö	ch d Ö	Documents a dialogue	is to	be dis	olayed. Index texts entered c Designation Exploded view Assembly simulation	d here will be applied to do Name of link class ● Bauteil-Konstruktion ● Bauteil-Konstruktion	Creat 03.10 02.10
elect the of for the of the	documents Standard Display C C	to be revised an	d specify for whi Ö	ch d Ö	Documents a dialogue	is to	be dis	olayed. Index texts entered c Designation Exploded view Assembly simulation Gear wheel geometry 2-	I here will be applied to do Name of link class	Creat 03.10. 03.10. 03.10. 03.10.
elect the of the construction Revise	documents Standard Display C C C C C	to be revised an	d specify for whi Ö Ö	ch d Ö	locuments a dialogue Document number DN-000028 DN-000027 DN-000026 DN-000001	is to	be dis In W 0 ( 0 ( 0 (	olayed. Index texts entered c Designation Exploded view Assembly simulation Gear wheel geometry 2- Draught	d here will be applied to do Name of link class	Creat 03.10. 03.10. 03.10. 03.10. 02.10.
elect the of the construction Revise Construction Cons	documents Standard Display C C C C C	to be revised an	d specify for whi Ö Ö Ö Ö Ö Ö Ö	ch d	locuments a dialogue Document number DN-000028 DN-000027 DN-000026 DN-000001	is to	be dis In W C C C C C	olayed. Index texts entered c Designation Exploded view Assembly simulation Gear wheel geometry 2- Draught	d here will be applied to do Name of link class	Crea 03.10. 03.10. 03.10. 02.10.
elect the solution Revise C C C Number of	documents Standard Display C C C D S D S D S D S S S S S S S S S S	to be revised an	d specify for white	ch d Ö	Documents a dialogue Document number DN-000028 DN-000027 DN-000026 DN-000001	is to	be dis In W • € • €	olayed. Index texts entered c Designation Exploded view Assembly simulation Gear wheel geometry 2- Draught	d here will be applied to do Name of link class	Crea 03.10 03.10 03.10 02.10
elect the elect the constant cons	documents Standard Display 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2	to be revised an	d specify for white Ö O O O O O O O O O O O O O	ch d	locuments a dialogue Document number DN-000028 DN-000027 DN-000026 DN-000001	is to	be dis	olayed. Index texts entered c Designation Exploded view Assembly simulation Gear wheel geometry 2- Draught	I here will be applied to do Name of link class Bauteil-Konstruktion Bauteil-Konstruktion Bauteil-Konstruktion	Crea 03.10 03.10 03.10 02.10
elect the elect the for Revise C C Number of C C C C C C C C C C C C C	documents Standard Display C C C Display C C C C C C C C C C C C C C C C C C C	to be revised an	d specify for white Ö C C C C C C C C C C C C C	ch d	locuments a dialogue Document number DN-000028 DN-000027 DN-000026 DN-000001	is to	be dis	olayed. Index texts entered c Designation Exploded view Assembly simulation Gear wheel geometry 2- Draught	I here will be applied to do Name of link class Bauteil-Konstruktion Bauteil-Konstruktion Bauteil-Konstruktion	Crea 03.10 03.10 03.10 02.10

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

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

## New HiCAD Viewer format

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



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

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

 The above information also applies if only HELiOS is updated when using an older HiCAD version with HELiOS.

#### Italian and Polish remote maintenance

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

# Using the new Report Manager with HDE reports

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

# **HELIOS in HICAD**

# Service Pack 2 2024 (V 2902)

#### Model/Document structure and Sheet structure

Please note the following changes and enhancements in the transfer:

In previous HiCAD/ HELiOS versions, the **Sheet structure**, i.e. the available sheet areas of an SZA document, was displayed in the **Model structure** tab of the document master. As of HELiOS 2024 SP2, you will find the separate **Sheet structure** tab in the document detail mask for this purpose.

[	DN	-000	0013 (Document), Pr	oje	ct context: PN-01-06-K		-	n x
General Assignments Inp	ut	Ou	tput				*	•
Edit master data Preselect Open + Edit • V	E Vorkflo	dit ow s	tatus ieneral nurces × Targ	ute es	Notes Mark-up document • • • Delete Edit ×   Utilized article ×   Graphic ×   Us	e	)	×   »   •
🔸 📲 👫 🐱 Standard			-					
Sheet structure	0	0	Document number	In	Designation			
✓ DN-000013		B	DN-000013		Production drawing			
🗊 3-D model								
🖾 Sheet 1 (Main views)								
Sheet 2 (Second view)								

The **Model structure** tab in turn displays the document structure of HiCAD files (.SZA, .KRA). In other words, it displays the document masters that were referenced by the corresponding SZA/KRA document.

$\ll$ Sheet structure $\times$	Model structure	×	So	urce	es	× Targets		
🖘 📲 👫 🐼 Sta	andard		•					
Document	0 <sup>,</sup>	0	In	In	W	Designation		
<ul> <li>DN-000001</li> </ul>		B		•	0	Draft		
<ul> <li>DN-000002</li> </ul>			}	•	0	3-D model		
DN-000016		D		•	0	3-D model		
DN-000018		B		•	0	3-D model		
DN-000020				•	0	3-D model		
DN-000022				•	0	3-D model		
DN-000024		B		•	0	3-D model		
DN-000004	E	B		•	0	3-D model		
DN-000008		B		•	0	3-D model		
DN-000010				•	0	3-D model		
DN-000012		D		•	0	3-D model		
DN-000014		B		•	0	3-D model		
DN-000034		B		•	0	3-D model		
DN-000035	27			•	0	3-D model		

By saving the model/document structure of SZA/KRA files, the document where-used list (in the **Use** tab of the document master mask) can also be used.

					_	DN-000256_ret	f (Document)			
Genera	al As	signment	ts Input Outp	out						* 🖻 🔿
Edit	t Pro data	eselect +	Open Edit - Edit • Workflow sta	tus nera	Roles	Attribute values	otes Mark-up	Delete Edit		
« Sou	rces	$\times$	Targets $\times$	Jtiliz	zed artic	les $\times$ Graphic	× Use	×F	olders	× » 🛟 🔹
🍫 Lev	vels: All		] All indices 🛛 🦉	St	andard	-	<b>A</b> <i>K</i>			
Level C	Open, re	Open +	Document number	In	Index	Workflow status	Document type	Designatio	on Creation dat	Created by
1	2	B	PDN-000255			•	HiCAD Drawing		27.11.2023	Administrator

# Document reference up-to-dateness

The new database attribute **UI\_ReferencelsOutdated** is now available in the result list configuration of the **Model structure**.

				DN-000	225 (Document), Project co	onte	ct: HiC	CAD-1123 — 🗆 🗙
Genera	a l	Assi	gnmer	nts Input Output				* 🖻 📀
Edir master	data	Pres	elect	Open + Edit - Workflow status General	ibute Notes Mark-up		Delete Edit	
	∎ ∎ †		Stan	dard -		~	USE	
Docur V DN					😹 Edit a	ttrib	ute co	nfiguration
	- Av	ailab	le attr	ibutes		-		- Selected attributes
	Filter	r:				-		List De Context menu
		Docu	ment					↑         ↓           Result list display:         Standard (Helios.RL.DocumentStructureView) ✓
			Туре	Designation	Attribute name			
			1	Open, read-only	UI_OpenItemReadOnly			V Type Designation
			۲	Open + Edit	UI_OpenEditItem			Open, read-only
				Index	HEL_INDEX			Open + Edit
		🕑 🤍 Index up-to-dateness	Index up-to-dateness	UI_DocumentNewerIndex			🔲 🗐 Index	
			Workflow status	Workflow status	VA_DocumentReleaseStatus			Index up-to-dateness
				Designation	BENENNUNG			Workflow status
			۲	Document reference up-to-dateness	UI_ReferencelsOutdated			Besignation
				Article number	SACHNUMMER			
				Based on	ERSATZFUER		-	
				сс	CC		-	
				Changed by	HEL_GEAENDERT_USER			
Rows:				Changed on	HEL_GEAENDERT_DATUM			
				Changed on	HEL_LINKED_DATE			
			۲	Check list	VA_ObjectWorkflowCheckL			
			۲	Check list	VA_ObjectWorkflowCheckL			
				Checked by	GEPRUFT_NAM			
				Checked on	GEPRUFT_DAT	-		
	Attr For the	ribute a ref time	erence of ref	iption: , it is displayed whether there is a newe erencing.	r version of the reference since	e		Width of selected column (px):
	S 💟	how	colum	n for attribute names				Capital OX
								Carter
You can use this to add the **Document reference up-to-dateness** column to the **Model structure** result list.

	DN-0	000255	(Do	cument), Projec	t context:	HiCAD-1123	– 🗆 ×
General Assignments Inp	ut Out	put					
Edit master data	Edit Vorkflow st	R atus eneral	oles	Attribute values do	Notes cument +	Mark-up Delete	
$Mask \qquad \qquad \times   \ Sheet \ structure$	× Mode	l structu	ire :	× Sources	× Targ	ets $ imes  $ Utilized art	icles $ imes$   Graphic $ imes$   Use
🐓 📲 👬 🐷 Standard		•					
Document	Open, r	Open	In	Index up-to-da	Workflow	Designation	Document reference up-to-dateness
<ul> <li>DN-000255</li> </ul>	-	B		•	•		NOT ASSIGNED
DN-000256_ref	2	D		•	0	Part document	NOT ASSIGNED
DN-000258		6	a	•	0	Part document	NOT ASSIGNED

The following three states that a reference can assume are then displayed in the result list:

- NOT ASSIGNED: Marks the display of the header in the document structure as well as references for which no version information has been saved.
- Current: The referenced document has not been changed since the document structure was transferred. The reference is therefore up-to-date. (Please note that there may be outdated sub-parts for an assembly despite the reference being up-to-date).
- Old: The referenced document has been changed (and saved) in the meantime since the document structure was transferred).

« Model structu X Sources	×	Ta	rget	s		$\times \mid$ Utilized article $\times \mid$ Graphic
😽 📲 🛉 🐷 Standard			•			
Document	0	0	In	In	W	Document reference up-to-dateness
✓ DN-000230		B		•	0	NOT ASSIGNED
✓ DN-000224	2	B		•	0	OCurrent
DN-000229		B		•	0	Old

## Project and Folder selection in the Catalogue Editor

The menu of the HiCAD Catalogue Editor has been extended: Projects and Folders can now be activated via **HELiOS** in the menu bar.

Activate project
Deactivate project Show active project
Activate folder Deactivate folder

If a project or folder is active, the project/folder can be deactivated via the menu, the project/folder content can be displayed and another project/folder can be activated.

Activate project	Activate folder
Deactivate project (PN-01-06-K)	Deactivate folder (AN-100/06)
Show active project (PN-01-06-K)	Show active folder (AN-100/06)

The Catalogue Editor title bar shows which projects and folders are currently active.

CATEditor - [Catalogues] [C:\HiCAD\Kataloge] [Version: 29.2.0.374] [Active project: PN-01-06-K, Active folder: AN-100/06] File Edit View Extras HELiOS Settings ? ISD

## Service Pack 1 2024 (V 2901)

### Workflow selection

In earlier HiCAD versions, a dialogue for workflow selection appeared first when calling up new Drawing func-

#### tions, e.g. New drawing with database, with new article master

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

Basic information					
Article number:	SN-025767		Index:	I C D	
Project number:	PN-T01-01_a,	🟮		1.2.0	U
Folder number:	Folder-independent	🎾			
Designation 1:		•	Release:	In Progress	
Designation 2:			Part type:		•
Standard:			Drawing/Manuf.:		
Extended information					
Material:			Unit of quantity:		•
Weight:		▼ kg	Resourcing:		-
Dimensions:	-		Order note:		
Comment:					

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

### HiCAD 3-D annotation with HELiOS data

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

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

## Major Release 2024 (V 2900)

### Performance

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

## Article master display by double-click

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

Edit View Extras ISD						
5 2 11 12 13 11		AA   🞯 🖕	Use	er	-	
▲ System settings	*	Description	Value		Comment	
Assembly HCM		DB project	From document management	~	Database project	
<ul> <li>Sketch HCM</li> <li>Itemisation</li> </ul>		🥖 Check default DB links	Check, with query before correction	×	Check database connection (and correct if required)	
Processing plane		Use HELiOS/HiCAD Default Solution				
Scales Sketches Units Stat configuration		Handling of article master	Query	~	Handling of manually assigned article master during exchanging of ser finished products	
iii) Start configuration iii) Directories iii) Load/Save iii) Data save b iii) Identification		Taking over of semi-finished product attributes	Collection		Which semi-finished product attributes are to b taken over to manual artii masters ? (Format : Semi- finished product attribute;Article attribute)	
Referencing     Annotations     Galculations		Attributes for BOM via product structure	Only selected attributes	v	Selection of attributes to I transferred for BOM trans via product structure	
☐ Graphic ▷ ☐ Visualisation	ш	Transfer part attributes to HELiOS			Transfer part attributes of changed parts to article master when saving	
<ul> <li>Feature</li> <li>2-D Lines</li> <li>Miscellaneous</li> </ul>		Transfer product structure to HELiOS			Transfer product structure changed parts when savin 1 level	
Standard Parts HELIOS		When double-clicking on a part, display article master instead of part attribute mask				
Configurations	+					

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

# **HELiOS MS Office Interface**

## Major Release 2024 (V 2900)

## Send file by e-mail

Both via the context menu of document result lists in HELiOS and from document detail windows, you have the

option of sending corresponding files via the function **Send file by e-mail** ... to the standard e-mail programme in your local system.

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

	and the second s	Create follow-off sheetin		
Documents X	30	Create follow-on sheet, with link		
🍫 🖳 🐱 🚠	00	Derive variant	1	
Document number		Create variant index	ment type	Creation dat
PDN-000256		Output	Drawing	10.05.2023
PDN-000257		Show Version Management	Drawing	11.05.2023
PDN-000258	<b>N</b>		Drawing	12.05.2023
PDN-000261		Send file by e-mail	Drawing	16.05.2023
PDN-000262		Print (Spooler)	Drawing	16.05.2023
PDN-000263		Print (Windows default printer)	Drawing	16.05.2023
PDN-000264		C	Drawing	16.05.2023
PDN-000265	-	Convert	Drawing	16.05.2023
PDN-000266		Call report	Drawing	17.05.2023
PDN-000267	#	Copy number	Drawing	17.05.2023
PDN-000268			Drawing	23.05.2023
PDN-000269	1 100	COPY HELIOS URL	Drawing	24.05.2023
ADN-000270	2	Show ID		24.05.2023
ADN-000271		Edit		24.05.2023
PDN-000273	×	Delete	Drawing	25.05.2023
PDN-000274	2		Drawing	25.05.2023

-					
ant numbers In					
ent number in	Generated file name	Extensio	Article	Project	Folder
00001	DN-000001PN-01-06-K-SN	sza	SN-000001 📀	PN-01-06-K 💿	AN-100/06
00002	DN-000002PN-01-06-K-SN	kra	SN-000002 💿	PN-01-06-K 💿	
00003	DN-000003PN-01-06-K-SN	sza	SN-00002 📀	PN-01-06-K 💿	
00275	DN-000275PN-01-06-K-SN	sza	SN-026123 🛛 🗔	PN-01-06-K 💿	
00300	DN-000300PN-01-06-K	docx		PN-01-06-K 🚫	
00320	DN-000320PN-01-06-K	txt		PN-01-06-K 💿	
	0002 0003 0275 00300 00320	0002         DN-00002PN-01-06-K-SN           0003         DN-00003PN-01-06-K-SN           0275         DN-000275PN-01-06-K-SN           0300         DN-000300PN-01-06-K           0320         DN-000320PN-01-06-K	0002         DN-000002PN-01-06-K-SN         kra           0003         DN-000003PN-01-06-K-SN         sza           0275         DN-000275PN-01-06-K-SN         sza           0300         DN-000300PN-01-06-K         docx           0320         DN-000320PN-01-06-K         txt	N002         DN-000002PN-01-06-K-sh         kta         SN-000002         Etc.           N003         DN-000003PN-01-06-K-sh         sza         SN-000002         Image: SN-000002	N002         DN-000002PN-01-06-K-SN         kra         SN-000002         Immediate         PN-01-06-K         Immediate         Immediate

# **HELiOS Vault Server**

## Major Release 2024 (V 2900)

## Server Monitor: HELiOS Automation Service

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

This controls ERP interfaces and other tasks.

•	XIDU					
tatus	Server application	Eve	nt description			
			💠 Add server mon	toring		
		Choose a serv	rer application: HELiOS Automat	on Service 🔹		
		Server:				
		Port:	35729			
		- Configura	tion —			
		Name:		2		 -
		Version:		\$		
		Interval:	60	5		
		Log				

General	Settin	Setting:	<b>s</b> iOS Automatio	on Service
HELIOS Spooler HELIOS Vault Server HELIOS Article Synchronization HELIOS Automation Service	– General — Port: Interval:	35729 60	5	
	– Time-out – Response tim	30	s	
				Cancel

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#### Headquarter Dortmund

ISD Software und Systeme GmbH Hauert 4 D-44227 Dortmund Phone +49 231 9793-0 info@isdgroup.com

ISD locations worldwide at www.isdgroup.com

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