What's new?

Version 2019

News Overview

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TOC

Basics

Discontinuation of Windows 7 ®
Microsoft will cease the support for Windows by January 2020. Due to compability matters HiCAD 2020 SP2 and HELiOS 2020 SP2 will thus be the final versions of our CAD and PDM systems which support Windows ® 7. The corresponding server operating systems (Windows Server 2008 R2 or older) will then no longer be supported either.

Discontinuation of "old" HiCAD itemisation
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 2020 onwards, only the "new" itemization will be supported. The itemisation by assemblies (the **Within assembly, ...** options in the old Itemisation dialogue) will only be supported by the new Standard Itemisation **as of HiCAD 2019 SP2**.

Service Pack 2 2019 (V 2402)

Improved performance

- Switch Drawing/AutoSave
  Switching between large drawings with HCM models via **Switch Drawing** docking window has been significantly improved with HiCAD 2019 SP2. This also applies to the AutoSave function.

  In test cases, the following performance optimisations (duration in seconds) could be achieved for switching the drawing to an empty one:

	HiCAD 2302	HiCAD 2402
Example drawing with 4,800 parts	10	5
Parametric drawing with 3,400 parts	16	1

  Which performance increase is ultimately achieved depends on the respective drawing.
- Waiting time when rotate view
  The waiting time after rotating a shaded view could be reduced with SP2. In various tests, an increase in performance of approx. 25% was achieved.
- Activating parts
  The activation of parts in very large constructions has been accelerated.
- Drawings with many texts
  Constructions usually also include many texts. These can be " regular" texts, dimensions, annotations and BOMs. With SP2, zooming and moving in constructions with many texts has been significantly accelerated, by a factor of 4-20 depending on the construction, so that editing here is much more fluid.
- Sectional views
  In drawings with more than 25,000 parts, sectional views are updated much faster as of SP2, up to a factor of 3.
- Referencing assemblies
  The performance when referencing assemblies has been increased by internal optimizations.

- Breaking up Referencing of parts/assemblies
  The break up referencing of parts and assemblies function has been improved. In the test, performance increases between factor 2 and 4 were measured, depending on the assembly.
- ICN Update
  The update of the ICN representations in case of drawings modifications has been accelerated by internal optimizations.
- Inserting referenced assemblies via Clipboard
  The waiting time when inserting referenced assemblies via the clipboard has been reduced with SP2 - especially in large drawings. In test examples, an increase of acceleration by a factor of 60 was achieved.

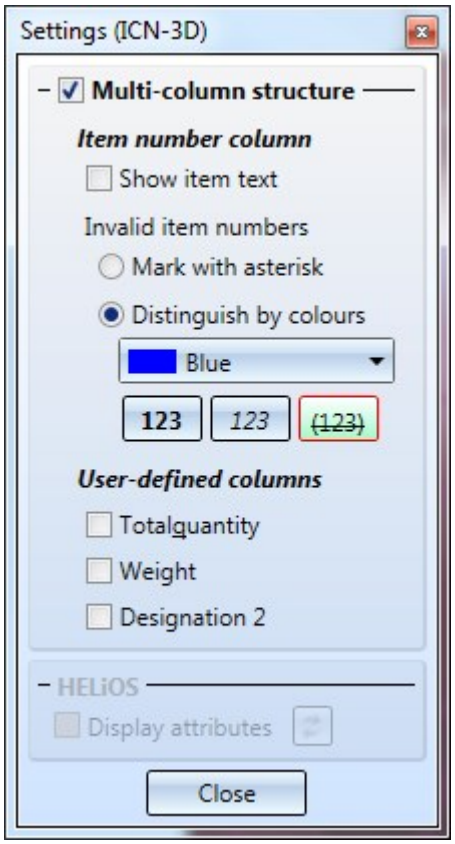
Docking window panoramas

The Visual Effects docking window has been renamed to Panoramas.

ICN enhancements

ICN - Representation of user columns

From HiCAD 2019 SP2 the ICN representation of up to three user columns is supported for the 3-D part structure. The representation can be switched on/off by activating/deactivating the checkbox.



The attributes displayed in the three user columns can be configured individually. The user columns are defined in the HDB files ICN3D_USER*n*.HDB (for use without HELiOS) or ICN3D_USER*n*_DB.HDB (for use with HELiOS). *n* is the number of the user column. The files are located in the HiCAD sys directory.

The following user columns are predefined by ISD:

User column	Attribute		File without HELiOS	File with HELiOS
1.	%06	Total number	ICN3D_ USER1.HDB	ICN3D_USER1_DB.HDB
2.	§01	Weight	ICN3D_ USER2.HDB	ICN3D_USER2_DB.HDB
3.	$02	Designation 2	ICN3D_ USER3.HDB	ICN3D_USER3_DB.HDB

For instance, the number of the shipping item, that is, the item number of the uncut installation profiles (attribute %PI), plays an important role in installation planning. However, the item number (attribute %02) is important for substructures. By defining corresponding user columns, you can display both item numbers in different columns.

To change the default setting of a user column, click on the corresponding HDB file, e.g. the file ICN3D_USER1_DB.HDB for the first user column:

```
HDB::HEADER="dwf_totalNumber"::SORTTYPE="INTEGER"
# HDB::HEADER="Versand-Pos..."::SORTTYPE="INTEGER"
#
# ::SORTTYPE="INTEGER" or "DOUBLE" or "STRING"
#
# Configuration file for the output of any number of
# HiCAD part attributes instead of the part name in the 3-D browser.
# TEXT - Commentary ( will not be displayed )
# ATTR - Attribute to be displayed
# TYP  - STRING(default), DOUBLE
# NKS  - Decimal places
# <H> - HiCAD part attribute
# <E> - Features

<H>::TEXT=""::ATTR="%06"::TYP="INTEGER"
# <H>::TEXT=""::ATTR="%PI"::TYP="INTEGER"
```

The row highlighted in red defines the contents of the first user column. The ISD default setting is the attribute %06, i.e. the total number. You can also enter the line more than once and mark the other lines as comments by placing the character # at the beginning of the line. In the file shown above, this is already applied for the attribute %PI, i.e. for the number of the shipping item. For example, you could delete the comment character in the %PI line and place it before the %06 line instead. The shipping item number would then be displayed in the ICN in the first user column.

See also HDX and HDB Files and Overview of HiCAD Attributes

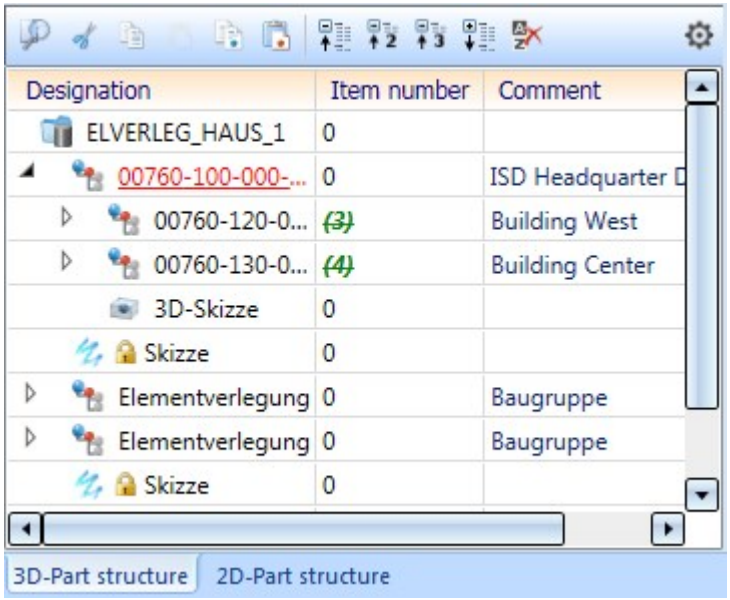
When saving HDB files, please make sure that ANSI is selected as character encoding.

ICN - Marking of invalid item numbers

In the Settings of the 3-D ICN you can define how invalid, i.e. not current, item numbers are to be marked:

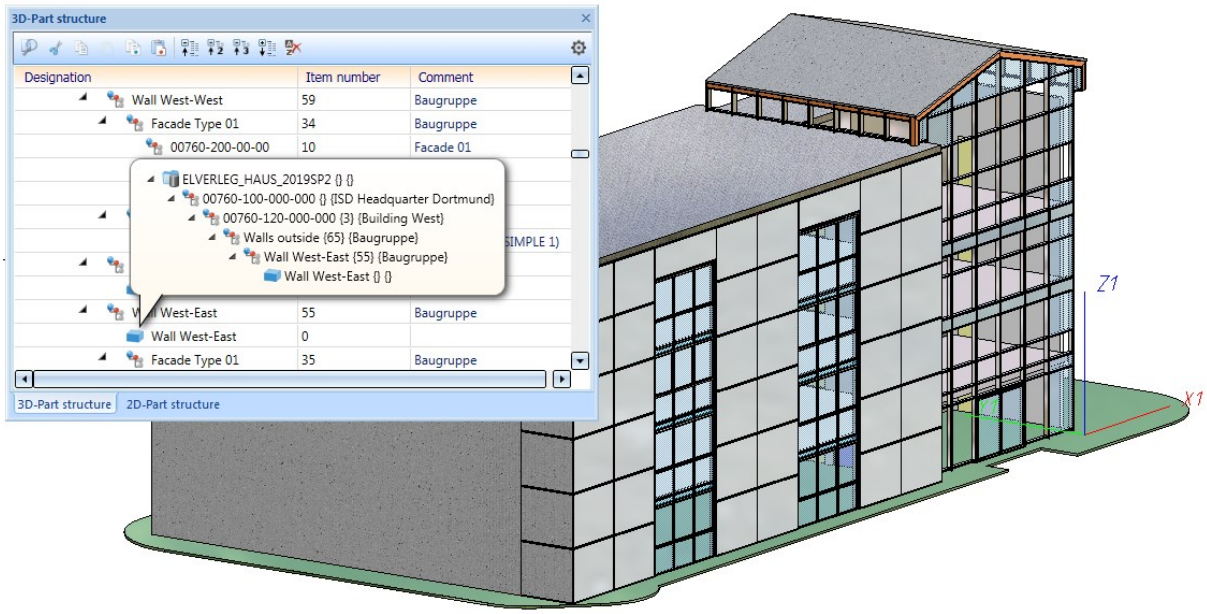
- with an asterisk * or
- by a colour representation.
  Select the desired colour in the list box. You can also change the font styles and variants by activating the buttons below. The fonts and variants **boldface**, *italics* und ~~strikethrough~~ are available as well as combinations of these.

Please note that item numbers with no current status are only marked accordingly the parameter **Set change mark when changing parts** is active in the Configuration Management under **Compatibility > Itemisation up to HiCAD 2017 > Updating.**



ICN - Tooltip with 3-D part structure

If you point to a part symbol in the 3-D part structure of the ICN with the cursor, now a tooltip with the corresponding superordinate structure is displayed. This can be especially helpful in large drawings with many parts.



ICN - Double click

The **Part attributes** dialogue window can be opened now by double-clicking on a part name with the left mouse button.

Settings for surface approximation

With the **Surface Approximation**  you define the number and determination of polygon points for 3-D quadrants. This setting is used, for example, when approximating polyhedra and shading 3-D models.

As of SP2, the ISD default settings in Configuration Management under **Modeling > Part creation > Polyhedron approximation** have been changed as follows:

Description	Value	Comment
Polyhedron approximation mode	Via distance tolerance with minimum number	
Number of division points per quadrant	6	min = 2; max = 100
Polyhedron approximation accuracy	2	min = 10*tol

The settings shown above are also used in the Configuration Templates (CSV files) for mechanical engineering and plant engineering included in the scope of delivery of HiCAD, which are loaded via the Parameter Configuration. The steel/metal configuration template uses 4 division points per quadrant.

Settings for dimensioning

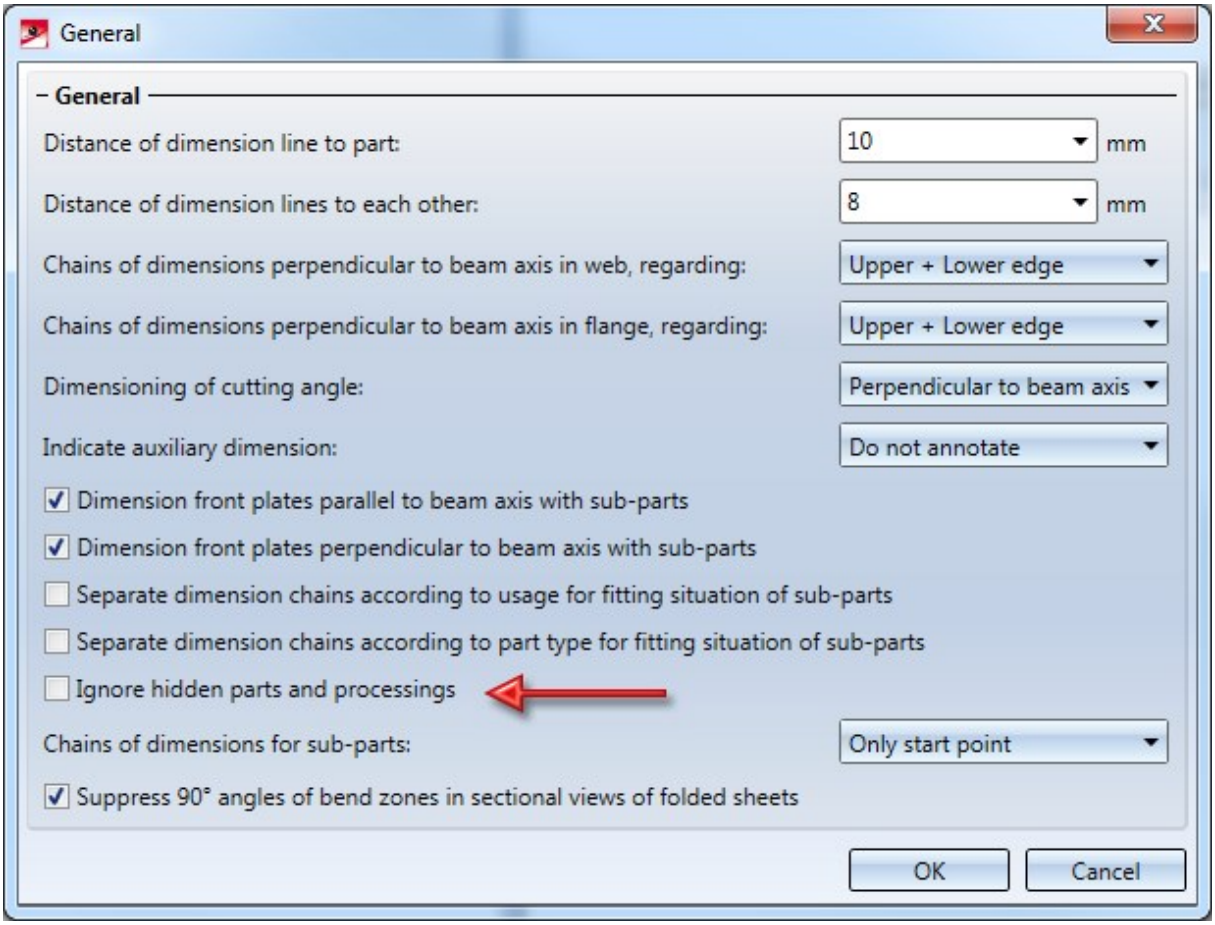
Bores/Boltings

In the **Settings for Conventional Dimensioning**  for drawing derivation with regard to **Bores/Boltings**, the ISD default setting for **Multiple display of successive, identical dimension chains if distance greater than** has been changed from 30 to 1500. This has led to better results in practice.

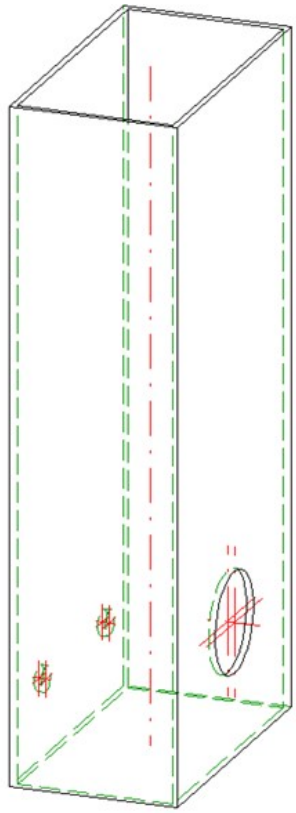
 The customer settings are preserved during update installations. If you want to use the new value as the default setting, you must change the file STAB3DPAR.DAT accordingly or - if available - the file STW_DIMSETTINGS.XML (parameter SUMUPEQUALDIST). Both files are located in the HiCAD sys directory.

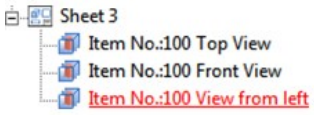
General - Hidden parts and processings

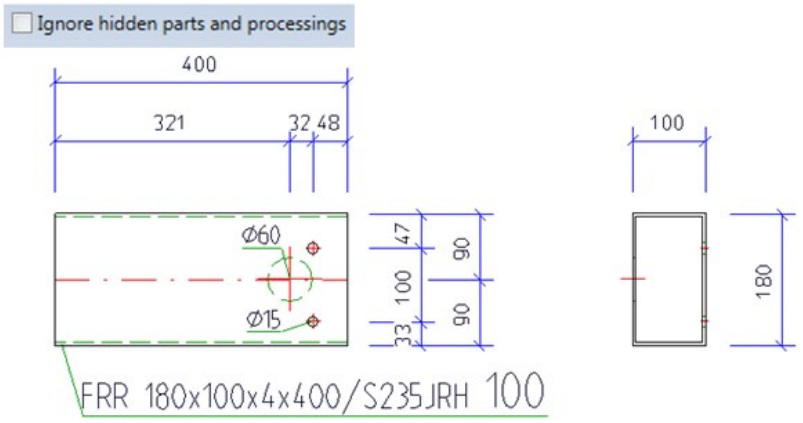
In the settings for "conventional" dimensioning, as of SP 2 it is possible to specify whether hidden parts and processings should be ignored. The setting predefined by ISD is **no**. This corresponds in the file STW_DIMSETTINGS.XML to the line </PARAM><PARAM Name="IGNOREHIDDENSUBPARTANDBORES" Typ="INT" Value="0"> or in the file STAB3DPAR.DAT (new customers) to the entry *ignore base points on invisible parts and bores? 1:yes, 0:no*

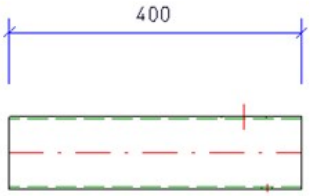


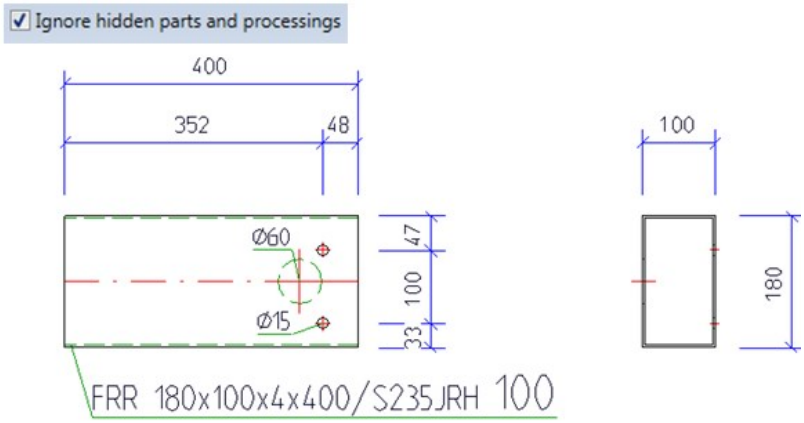
Please note that for drawing derivation this setting is only taken into account if the option **Load drawing parameters from configuration** is active.

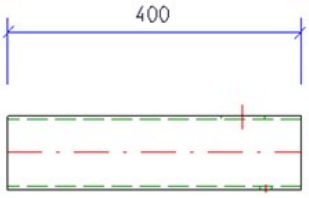



Sheet 3
Item No.:100 Top View
Item No.:100 Front View
Item No.:100 View from left


Ignore hidden parts and processings
400
321
32
48
100
Ø60
Ø15
47
100
33
90
90
180
FRR 180x100x4x400/S235JRH 100


400


Ignore hidden parts and processings
400
352
48
100
Ø60
Ø15
47
100
33
180
FRR 180x100x4x400/S235JRH 100

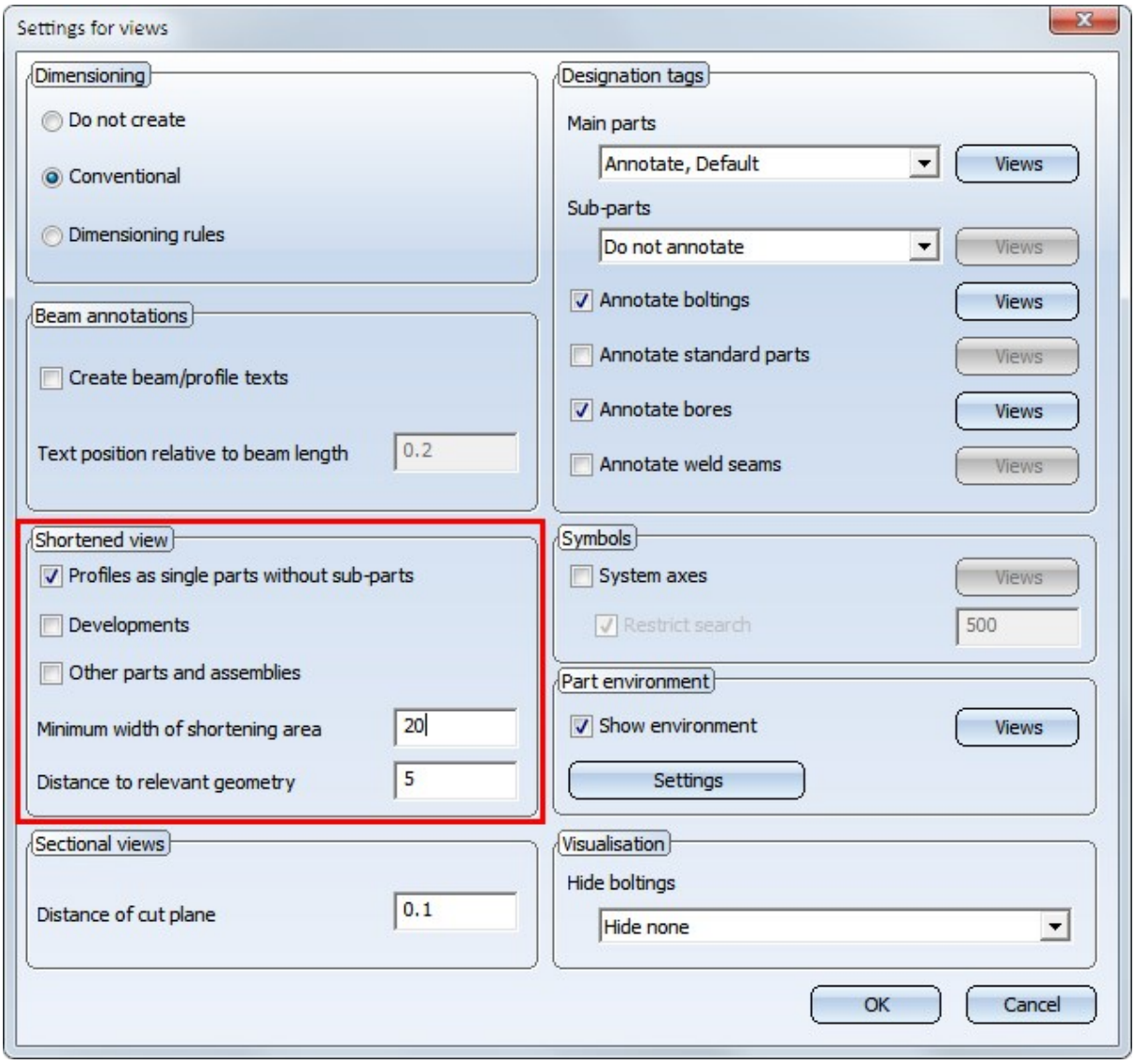

400

Drawing derivation

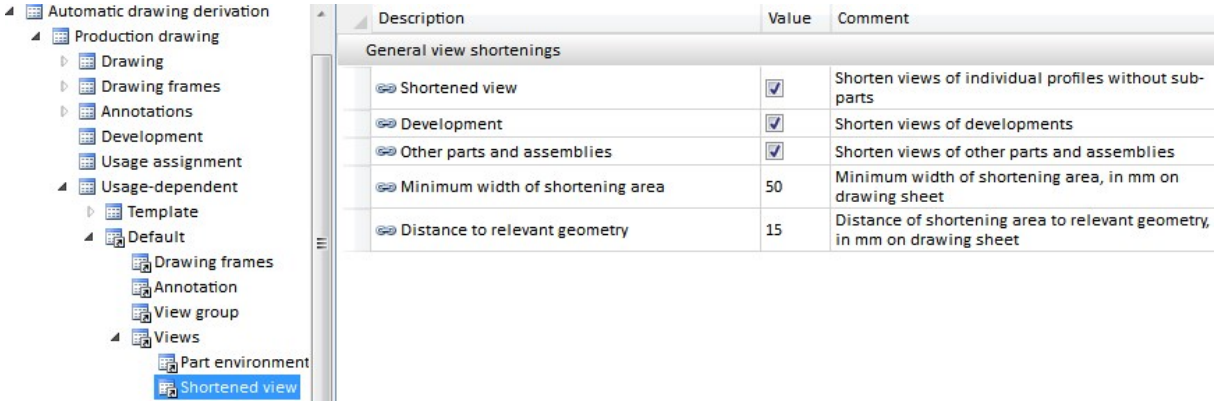
Drawing Derivation - Settings for shortened Views

In the Settings for Views window the texts for shortenings in drawing derivation have been improved.

Previously	From HiCAD 2019 SP2 onwards
Show views of beams as shortened views	Beams as single part without sub-parts
Show views of development as shortened views	Developments
Show other views as shortened views	Other parts and assemblies



The corresponding texts in the configuration management (**Automatic drawing derivation > Production drawing > Usage-dependent > ... > Views > Shortened view**) have been modified analogously.

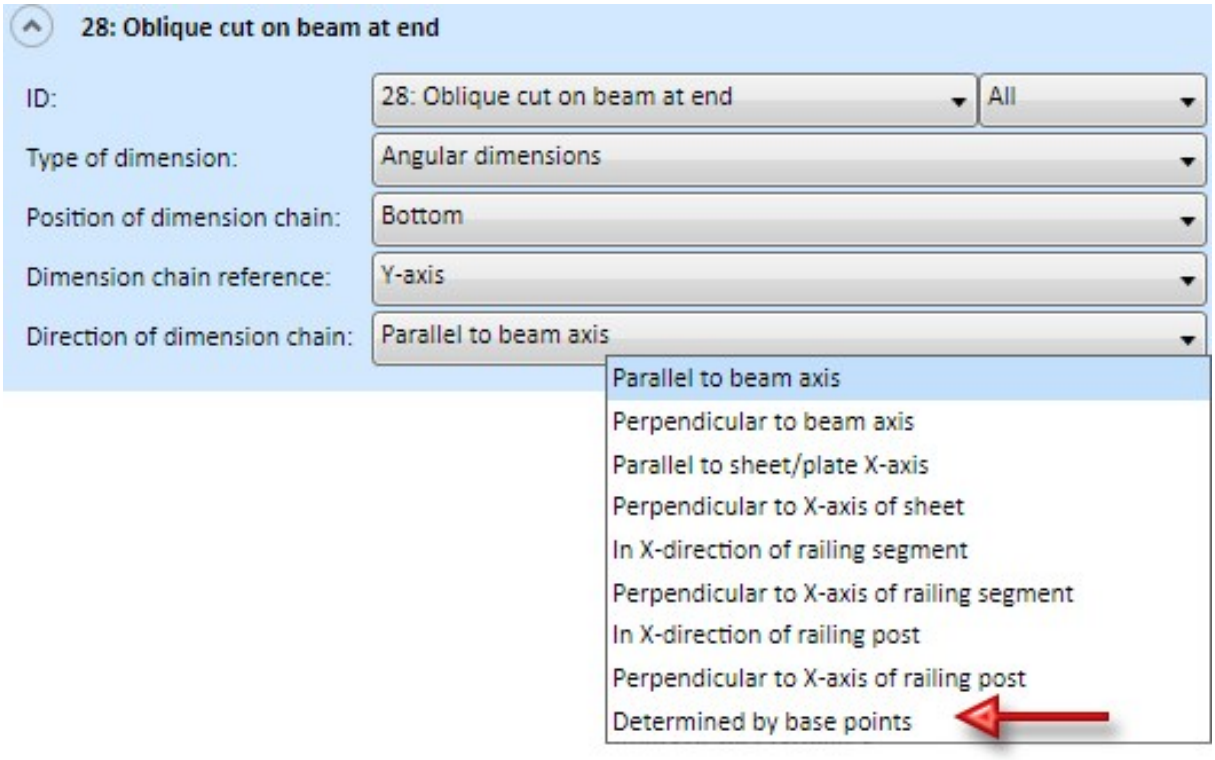


Drawing derivation - Changed configuration for Steel Engineering

The HiCAD parameter configuration for steel/metal engineering has been changed. The objective was to make workshop drawings in steel engineering easier to understand. For more information, see Steel Engineering - What's New.

Drawing derivation - Dimensioning rules

In the Dimensioning Settings, the **Direction determined by 2 points** option has been renamed to **Determined by base points** in the Direction of dimension chain selection box.

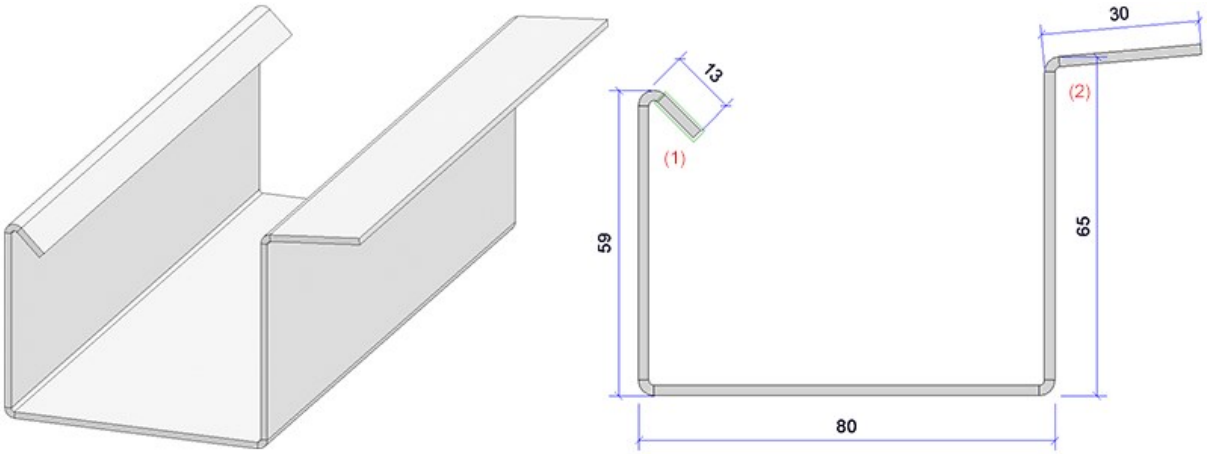


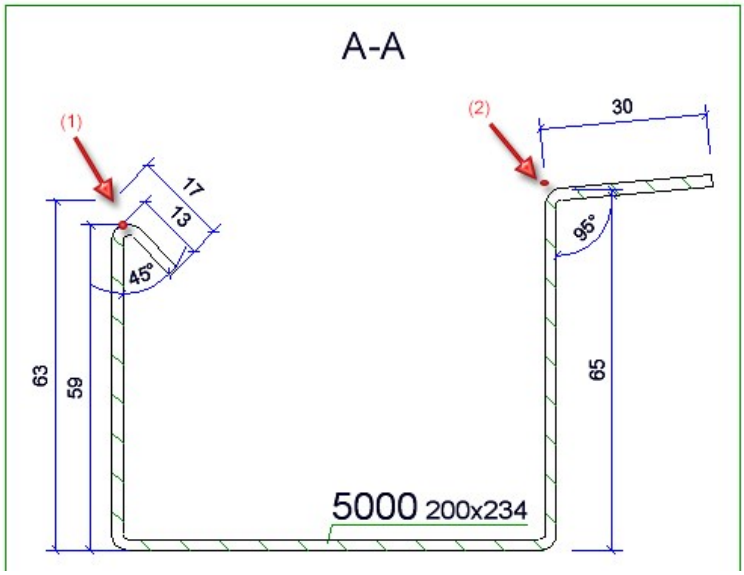
Drawing derivation - Dimensioning of sheets in sectional views

For dimensioning the flange length of sheets in sectional views, the following base point is being dimensioned:

- at an initial angle > or equal to 90°, the theoretical intersection of the flanges; and
- at an angle > 90° the tangential point at the edge of the bending zone.

The following graphic shows an example. The flange (1) was attached with an angle of 135°, the flange (2) with an angle of 85°. In the sectional view, the tangential point is dimensioned for flange (1) and the theoretical intersection point is dimensioned as the base point for tab (2).



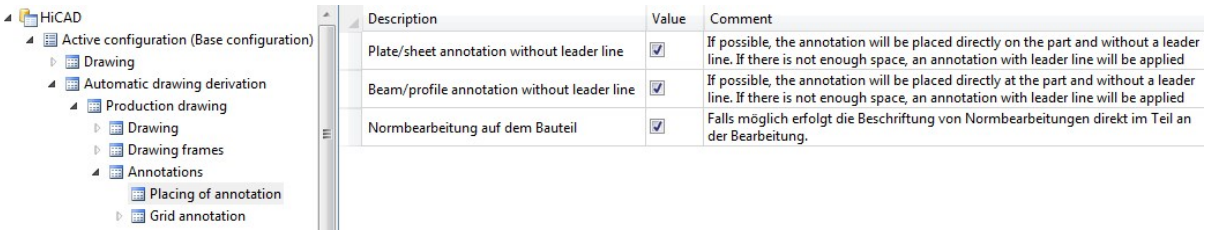


Drawing derivation - Improved placing of the annotation of standard processings

So far, the automatic placing of annotation for standard processings, e.g. threads, has often resulted in the leader line running unnecessarily across the hole bore pattern. This behaviour has been changed with HiCAD 2019 SP2. Now, the leader lines of standard processings always go to the nearest standard processing. In addition, the annotations can now also be placed on the part and not just outside as before. This applies to both production and mounting drawings. The settings are made via Configuration Management. There you can find via

- Automatic drawing derivation > Production drawing > Annotations > Placing of annotation or
- Automatic drawing derivation > Mounting drawing > Annotations > Placing of annotation

the parameter Standard processing on part.

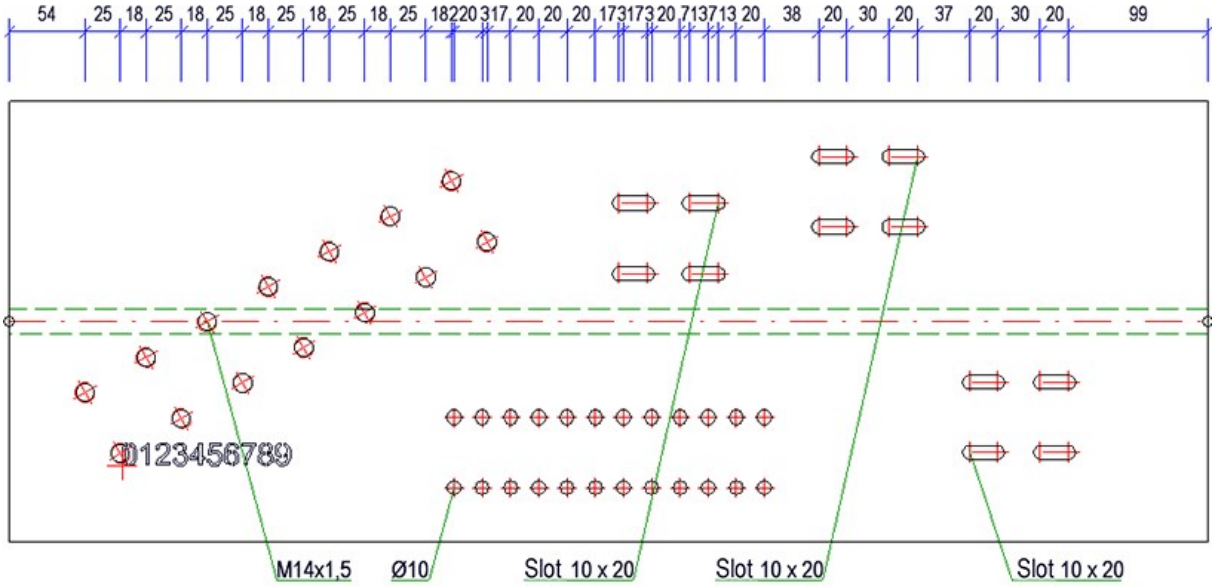


If the checkbox is active, the annotations of standard processings are located directly aside the processing - i.e. on the part - if there is enough space. If there is not enough space, the annotation is placed further away.

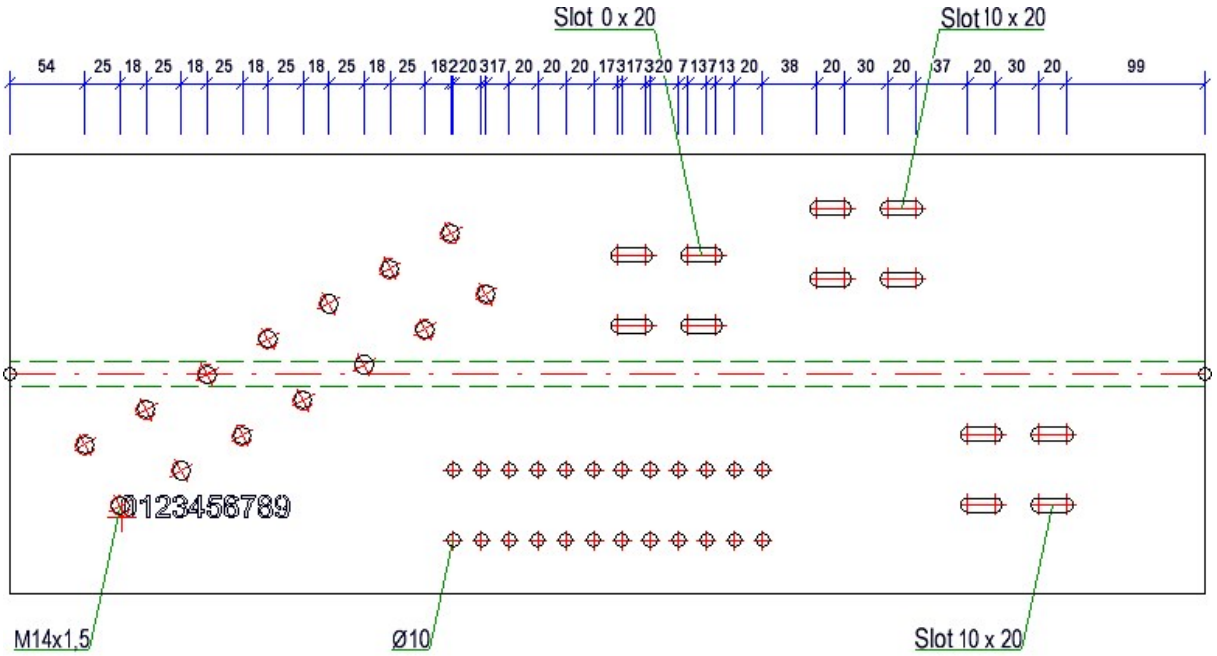
If the checkbox is inactive, the annotations are - as before - placed outside the part. The leader lines now always go to the nearest standard processing.

An example:

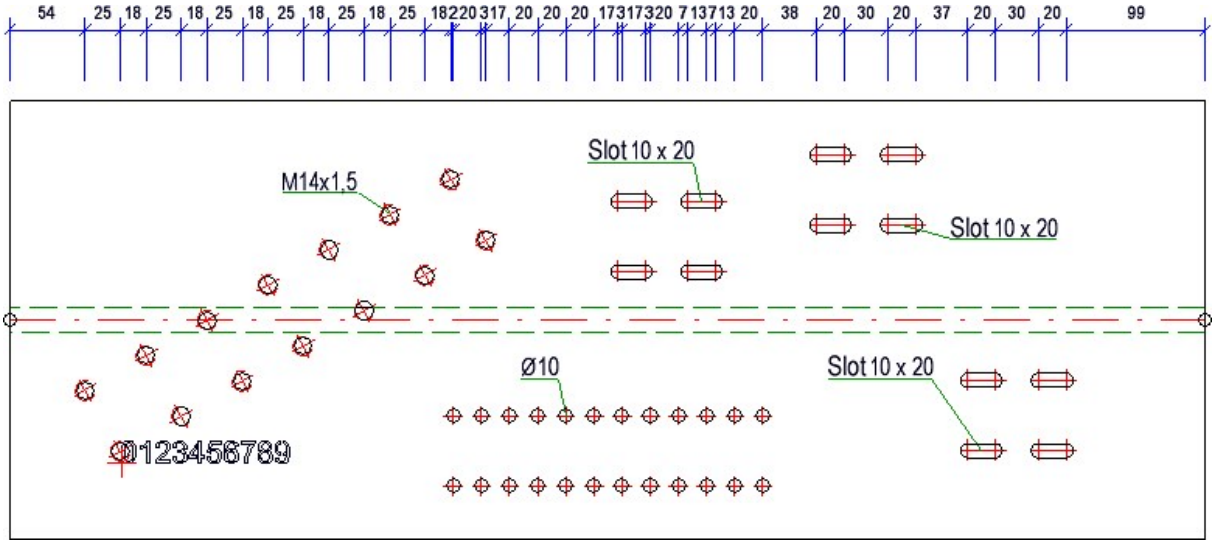
1. Placing of annotations in HiCAD 2019 SP1



2. Placing of annotations as of HiCAD 2019 SP2 - Standard Processing on part: no



3. Placing of annotations as of HiCAD 2019 SP2 - Standard Processing on part: yes



Mounting drawing

Overall drawing for sub-structures of element installation

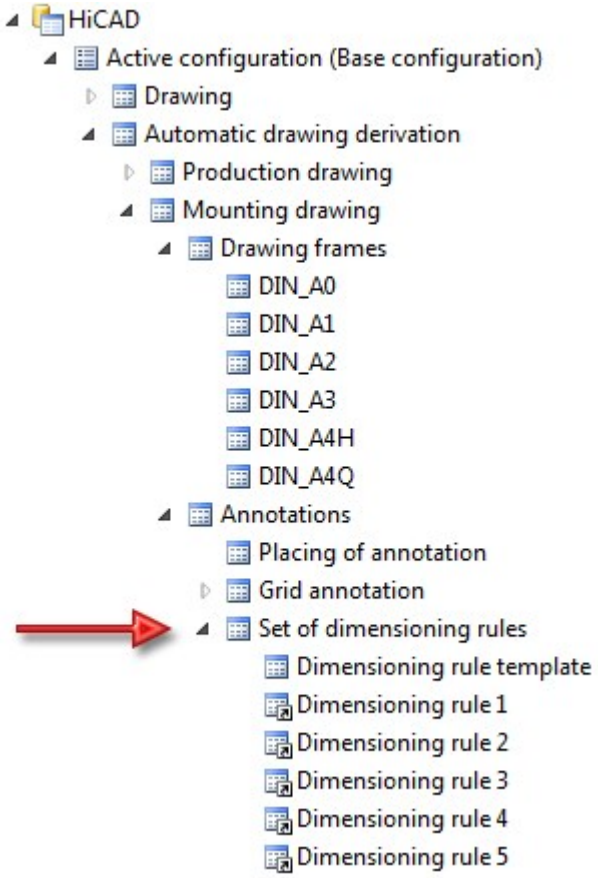
The Automatic drawing derivation function supports creating usage-dependent production drawings. Thus, individual dimensioning rules can be specified for different usages, such as girders, columns, certain beams, railings etc.By contrast, the creation of mounting drawings is being independent of the usage for parts and assemblies. Only the selected box is taken into account. In most cases this option is sufficient - the essential parts can be annotated automatically and grids can be dimensioned automatically as well.

However, in order to enable overall drawings with automatic creation of relevant dimensions for substructures of the element installation, dimensioning rule sets can also be defined in the Configuration management for mounting drawings - albeit to a very limited extent.

Hence, only three IDs are available:

- No dimension (in most cases this is the recommended default setting)
- 2003: Profile sub-parts of an assembly without main part (recommended for sub-structures)
- 2004: Sub-parts of an assembly via named isolated point (recommended for wall consoles of sub-structures)

These dimensioning rule sets can be used to define what is dimensioned and how (dimension type, position, reference, etc.) and in which view.



The following settings are recommended for sub-structures:

Dimensioning rule 1

Description	Value
Sequence	1
ID	2003: Profile sub-parts of an assembly without ma
Type of dimension	Chain dimensions
Position	Bottom
Reference	Start/End point of complete assembly
Direction	Parallel to beam axis
Show dimensions in views of type	From front
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None

Dimensioning rule 2

Description	Value
Sequence	1
ID	2003: Profile sub-parts of an assembly without mai
Type of dimension	Height above datum
Position	Left
Reference	Start/End point of complete assembly
Direction	Perpendicular to beam axis
Show dimensions in views of type	From front
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None

Dimensioning rule 3

Description	Value
Sequence	1
ID	2004: Sub-parts of an assembly via named isolated
Type of dimension	Height above datum
Position	Left
Reference	Start/End point of complete assembly
Direction	Perpendicular to beam axis
Show dimensions in views of type	From front
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None
Show dimensions in views of type	None

Tip for sub-structures/consoles:

When configuring the assemblies with wall consoles, you have the option to define a dimensioning-relevant point on the console assembly yourself. This point will be considered as decisive point (ID 2004).

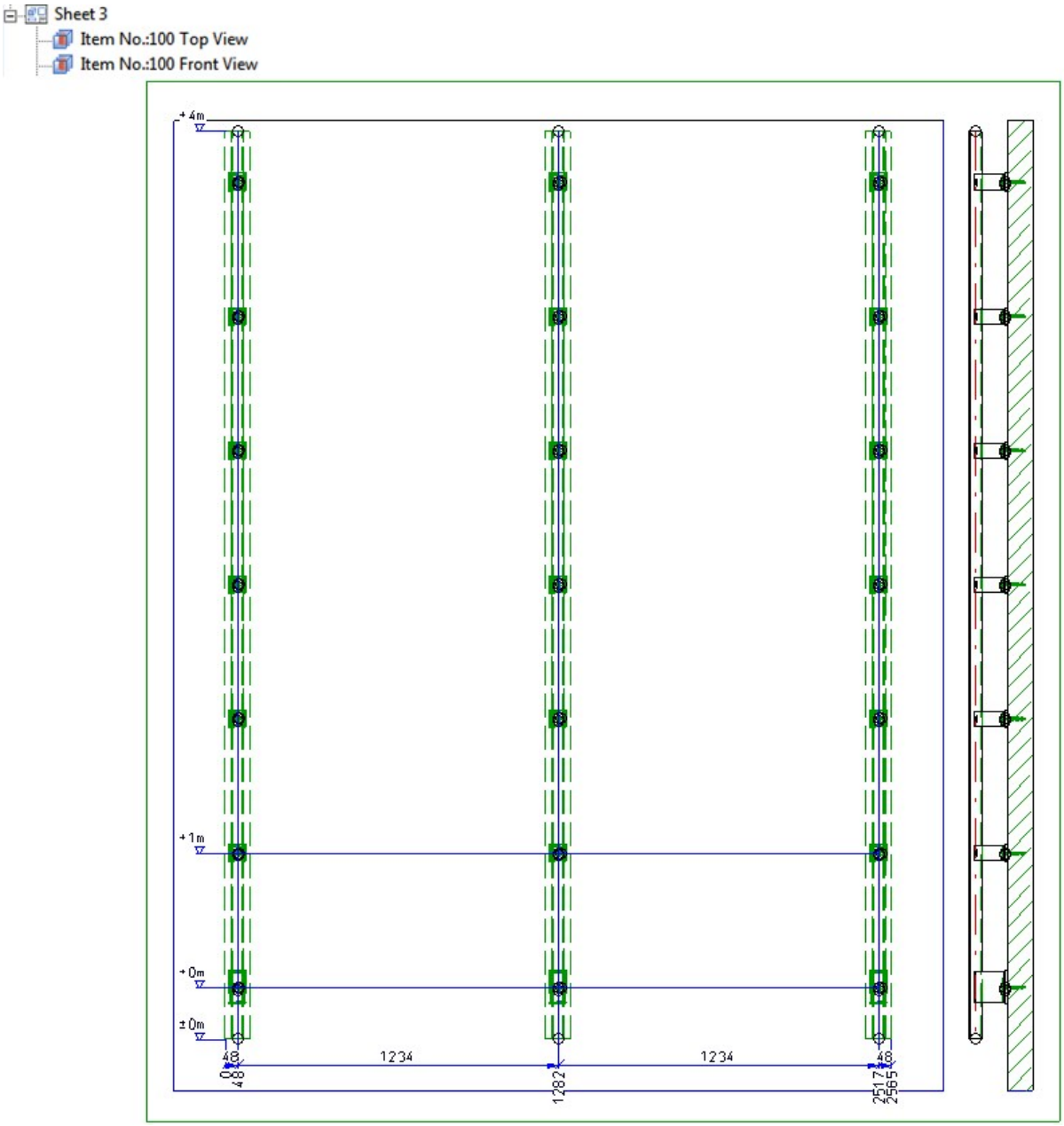
This decisive point **must** be a designated point, e.g. with the name **!**.

Please also take note of the hints in the area of Sub-structure.

An example:

The illustration shows a substructure with wall brackets (and dimensioning-relevant relevant point)

If the mounting drawing is created with the settings listed above, the following example may result:



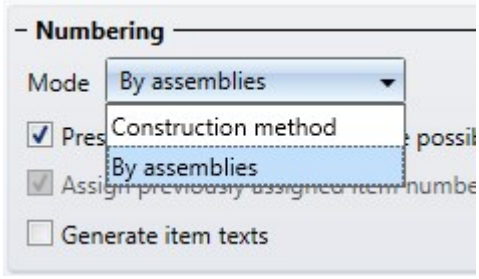
Settings for Sheet Metal Processing (ABWPAR.DAT / ABWCOL.DAT)

The basic settings for sheet metal processing, which were previously defined in the ABWPARDAT and ABWCOL.DAT file, can now be set in configuration management. For detailed information, see Configuration Management - What's New.

Itemisation

Itemisation by assemblies

As of SP2, HiCAD supports itemisation by assemblies with its new standard itemisation. For this, the tab **General** has been enhanced accordingly. Under **Mode**, you can now select whether the itemisation is to be continuous for the entire design or continuous within an assembly.



Example

In the following illustration, the factory settings have been used for the steel engineering. (1) Without item numbers, (2) Itemised in the mode **By assemblies**, (3) Itemised in the mode **Construction method**

❶

Designation	Comment	Item nu...
POSNEU_ANWENDUNGSBE...		
Assembly	Assembly	
Assembly LS 40x5	Assembly	
LS 40x5	L - Profile	
Bl 10	Bleche	
Assembly I-Beams	Assembly	
Assembly IPE 750x137	Assembly	
IPE 750x137	I - Profile	
Bl 10	Bleche	
Assembly IPE 500	Assembly	
IPE 500	I - Profile	
Bl 10	Bleche	

❷

Designation	Comment	Item nu...
POSNEU_ANWENDUNGSBE...		
Assembly	Assembly	1
Assembly LS 40x5	Assembly	1
LS 40x5	L - Profile	101
Bl 10	Bleche	100
Assembly I-Beams	Assembly	2
Assembly IPE 750x137	Assembly	2
IPE 750x137	I - Profile	101
Bl 10	Bleche	100
Assembly IPE 500	Assembly	1
IPE 500	I - Profile	101
Bl 10	Bleche	100

❸

Designation	Comment	Item nu...
POSNEU_ANWENDUNGSBE...		
Assembly	Assembly	1
Assembly LS 40x5	Assembly	5
LS 40x5	L - Profile	103
Bl 10	Bleche	100
Assembly I-Beams	Assembly	2
Assembly IPE 750x137	Assembly	4
IPE 750x137	I - Profile	102
Bl 10	Bleche	100
Assembly IPE 500	Assembly	3
IPE 500	I - Profile	101
Bl 10	Bleche	100

Consideration of processing direction of steel plates

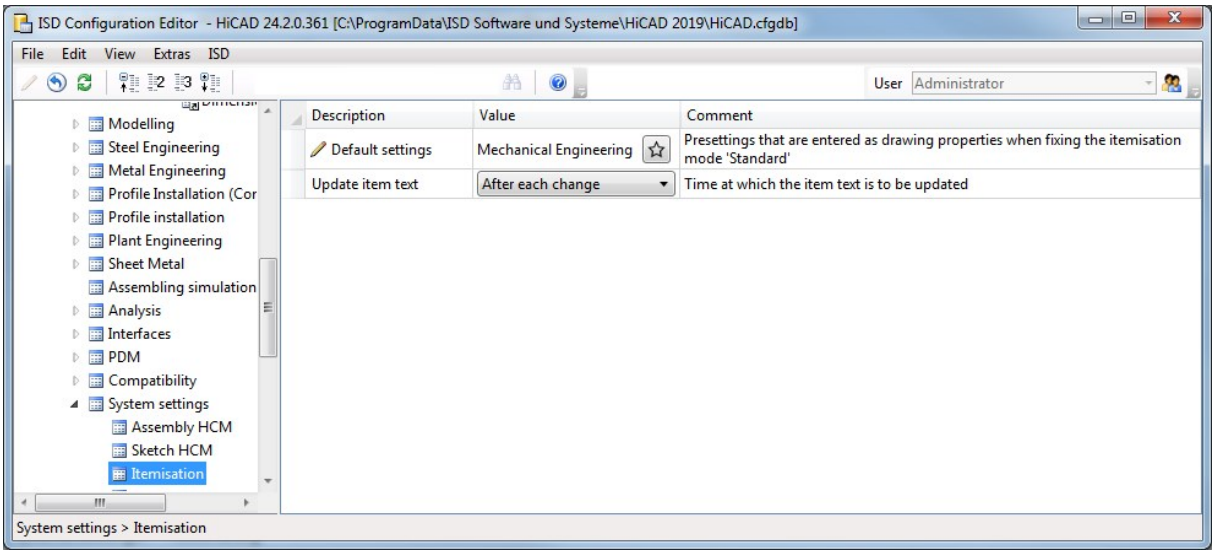
As of HiCAD 2019 SP1, the Processing direction can also be selected as a distinguishing criterion for the search of identical parts. Previously, this option was only available for sheet metal plates. From SP2 onwards, steel plates are now supported as well.

Setting time at which the item text is to be updated

Previously, item texts were updated after each modification to drawings. In large drawings, this can lead to performance loss, especially if attributes from superordinate assemblies are used in item texts. For this reason, the new parameter **Update item text** is from SP2 onwards available under **System settings > Itemisation** to set the time for updating item texts.

- **After each change** (as predefined by ISD) or
- **After itemisation**.

Please note, however, that the setting **After itemisation** has the disadvantage that it is not possible to recognize whether an item text is invalid.



Information - Distance between two 3-D points

The previous functions


Information, Direct distance 2 points (3-D)


Information, X-distance 2 points (3-D)


Information, Y-distance 2 points (3-D)


Information, Z-distance 2 points (3-D)

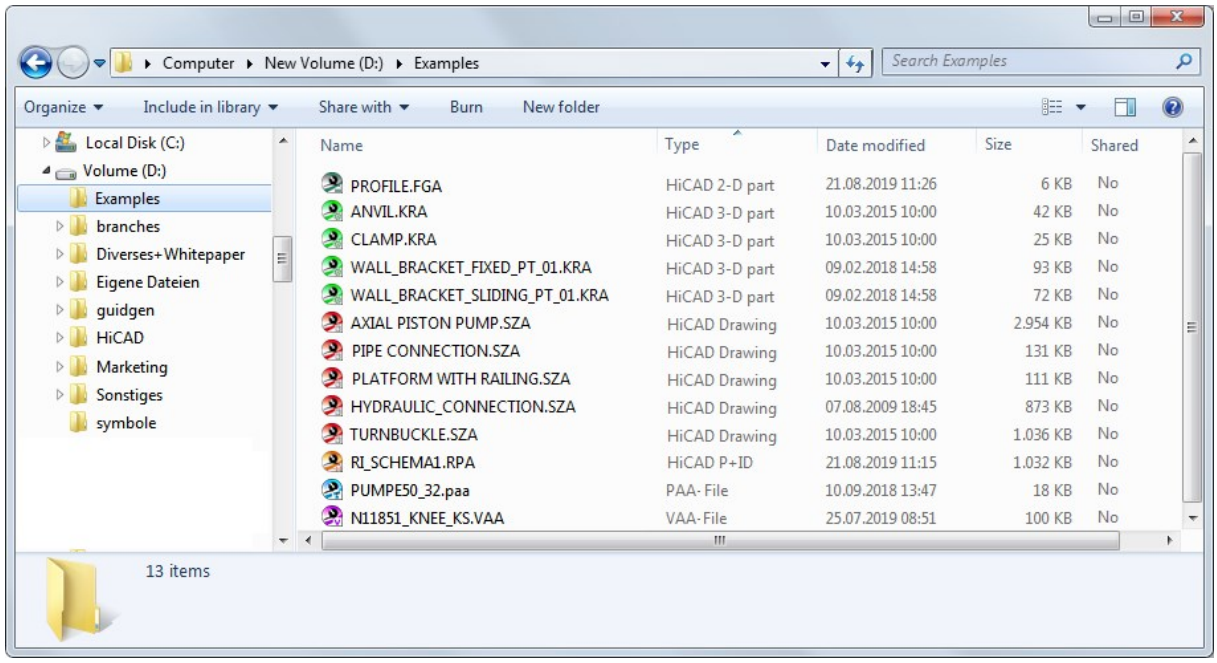

Information, XYZ-distance 2 points (3-D)

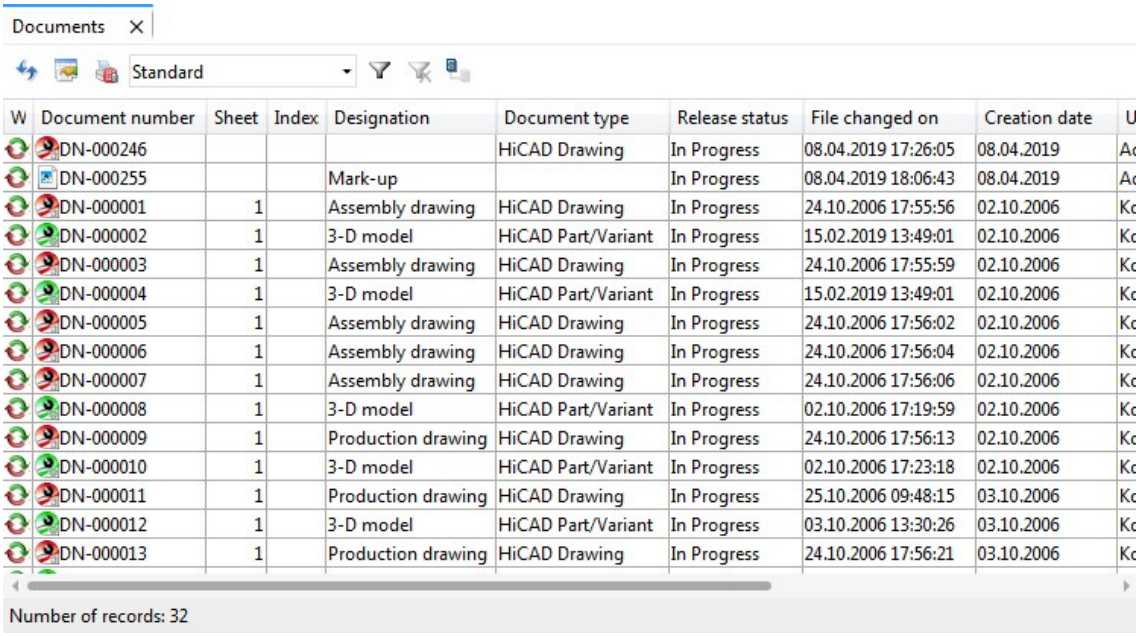
have been combined to one function **Information, distance 2 points (3D)**. The PullDown menu is therefore no longer used from SP2 onwards.

New icons for HiCAD file formats

From SP2 onwards, new icons are available for different HiCAD file formats, such as SZA, FGA, KRA, VAA, PAA. These icons are taken into account in Windows Explorer and thus also in HELiOS result lists.

	File format	File type
	.SZA	HiCAD drawing
	.KRA	HiCAD 3-D part
	.FGA	HiCAD 2-D part
	.VAD	HiCAD variant of previous versions
	.VAA	HiCAD variant
	.PAA	HiCAD part archive
	.RPA	HiCAD P+ID scheme
	.FIA	HICAD P+ID module

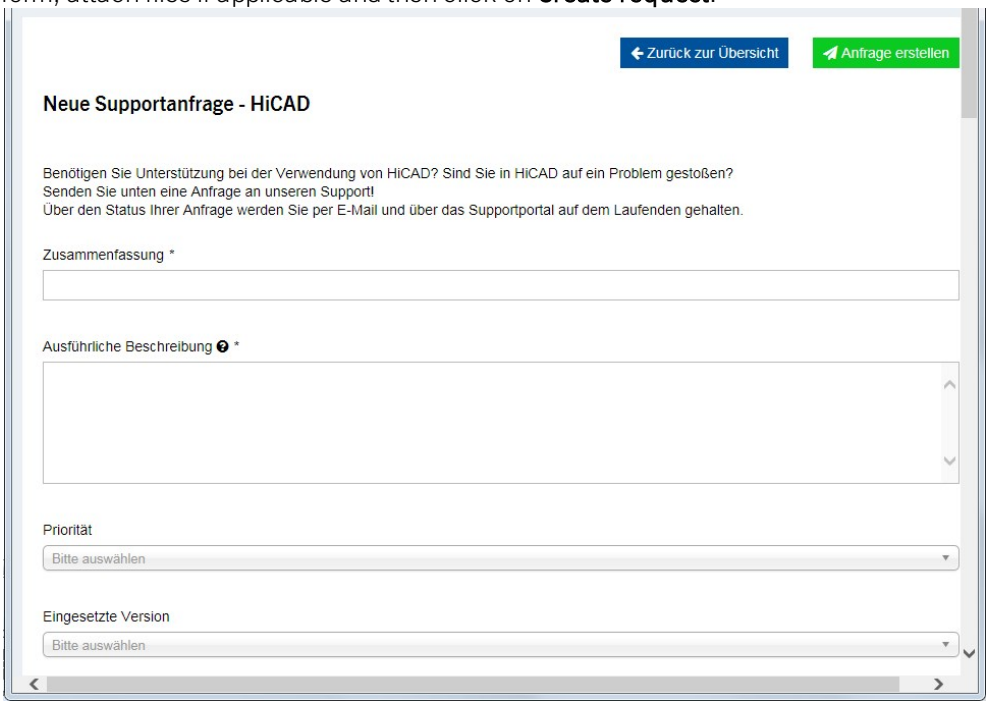




New support request

The function **New support request** is now available in **Help Topics and Information**

Use this function to send a support request directly from HiCAD to our Support. After calling the function, simply log in with your access data for my-ISD. Afterwards you will automatically enter the ISD support portal. Simply fill out the form, attach files if applicable and then click on **Create request**.



The new ISD support portal will be available for customers in Germany from 1 October 2019. For customers in other countries, the launch is scheduled for January 2020.

Service Pack 1 2019 (V 2401)

Improved performance

- Visualisation of Bitmaps
  As of SP1 Bitmaps in HiCAD drawings (for example logos in title blocks) are to be drawn in OpenGL for representation on screen. Thus, the performance is improved particularly for large bitmaps. An improvement by means of factor 60 could be targeted by testing with an example drawing.
- Switch drawing
  By optimising intern routines as pitch of threads for polyhedral models and catalogue queries for thread textures amongst others, the switch drawing could be accelerated.
- Rotating views
  With HiCAD 2018 SP2 rotating shaded views had been accelerated significantly, for shaded views with edges in particular. As of HiCAD 2019 SP1 the performance has again been considerably improved. (See also 3D - What's new?)
- Multiple selection
  These calculations have also been significantly accelerated by optimizing internal routines.
- Update referenced parts
  From SP2 onwards, sketches with many thousands of lines can be edited more quickly, even if HCM is not completely switched off. (see also Parametrics - What's new?)
- Accelerated graphic preview
  From HiCAD 2019 SP1, the graphic preview in the ICN is always drawn via OpenGL. The OpenGL mode is used, which most closely corresponds to the current view mode. This leads to a significant acceleration of the preview.

Licensing

- Both modules Profile Installation and Element Installation are now also part of the Trial Edition as well as HiCAD Trainee Suite.
- **HiCAD ALUCOBOND Suite exclusive Standard** and **HiCAD ALUCOBOND Suite exclusive Premium** are no longer available. As of now, there are only two ALUCOBOND Suites, namely **HiCAD ALUCOBOND Suite Standard** and **HiCAD ALUCOBOND Suite Premium**.

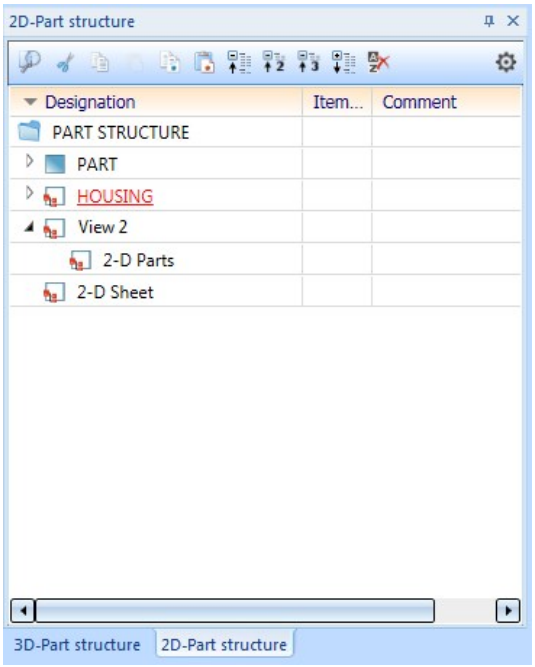
Maximum number of layers

So far, the amount of possible layers could be set via configuration management **System settings > 2-D lines** with the parameter **Maximum number of layers**. From Service Pack 1 onwards, this setting is not available. The amount of possible layers has been set to 1000 (layer 0 to 999).

New display of 2-D Part structure in ICN

Like the 3D-Part structure tab in the ICN, the 2D-Part structure tab has had a redesign:

- Improved performance, notably with complex drawings
- Tabular, multicolumn representation
- Sorting by click on the column header
- Individual configuration of the representation
- Fast switching of the representation of HiCAD and HELiOS attributes

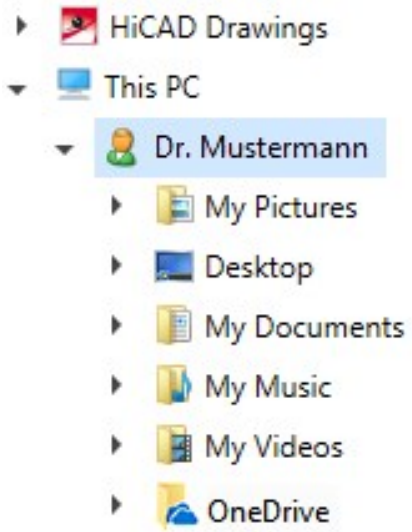


In connection with the new representation of the 2-D Part structure, system files have been renamed as follows:

Up to HiCAD 2019	As of HiCAD 2019 SP1
BRW_2DTEILATT.HDB	ICN2D_MULTIATTRVIEW.HDB
BRW_2DDBATTR.hdb	ICN2D_MULTIATTRVIEW_DB.HDB

Saving/Loading to or from cloud directories

If available, cloud directories such as Microsoft OneDrive or Dropbox are displayed when loading or saving drawings via HiCAD Explorer. These can be found in your user folder.

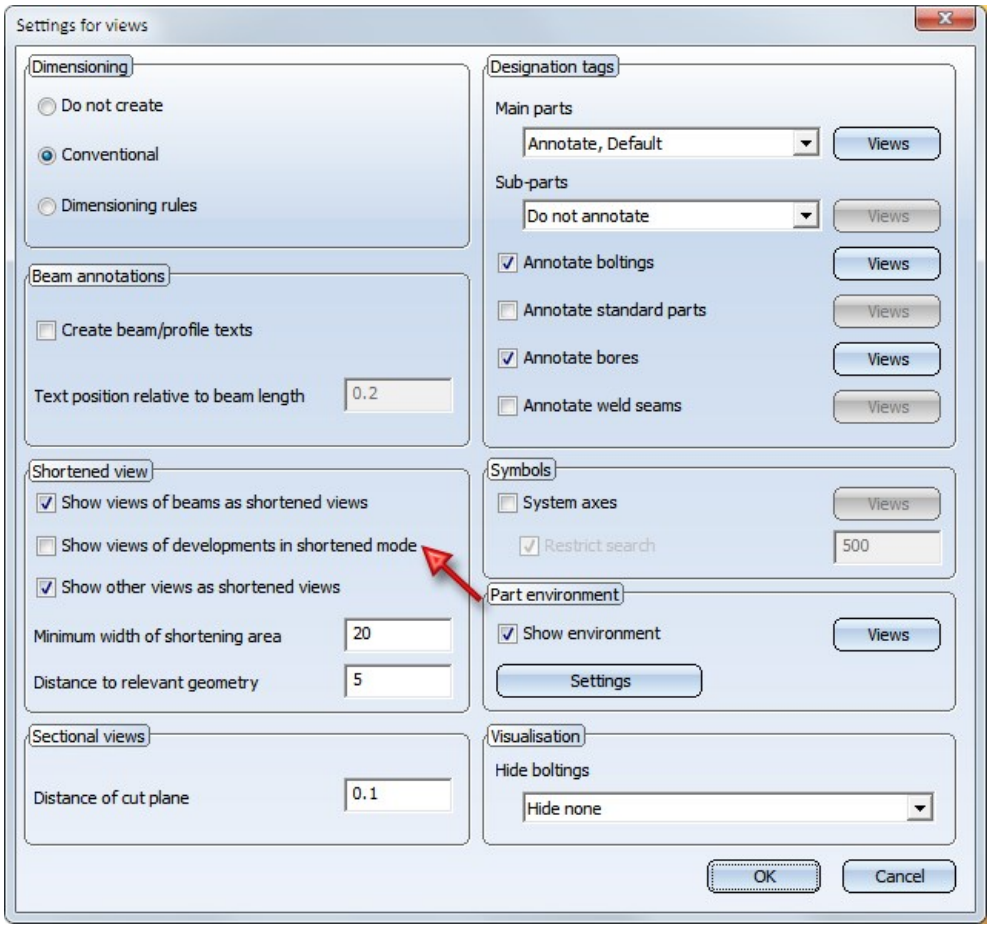


Display of installed Hotfixes in HiCAD Info window

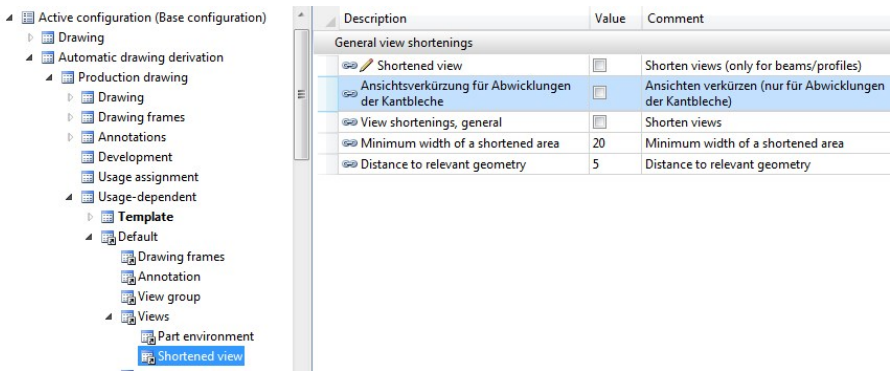
In HiCAD Info window, numbers of installed Hotfixes are being displayed in addition to the current HiCAD version number (including build number). To show the Info window, open **Help Topics and Information** and select **HiCAD**.

Drawing derivation - Shortened view of development

Views of developments of Sheet Metal can be created shortened as from now. The **Settings for views** window has been enhanced accordingly.



The pre-settings can be defined for each intended purpose under **..> Automatic drawing derivation > Production drawing > Usage-dependent >** ***NAME*** **> Views > Shortened view**, with the ***name*** being the name for the respective intended purpose.



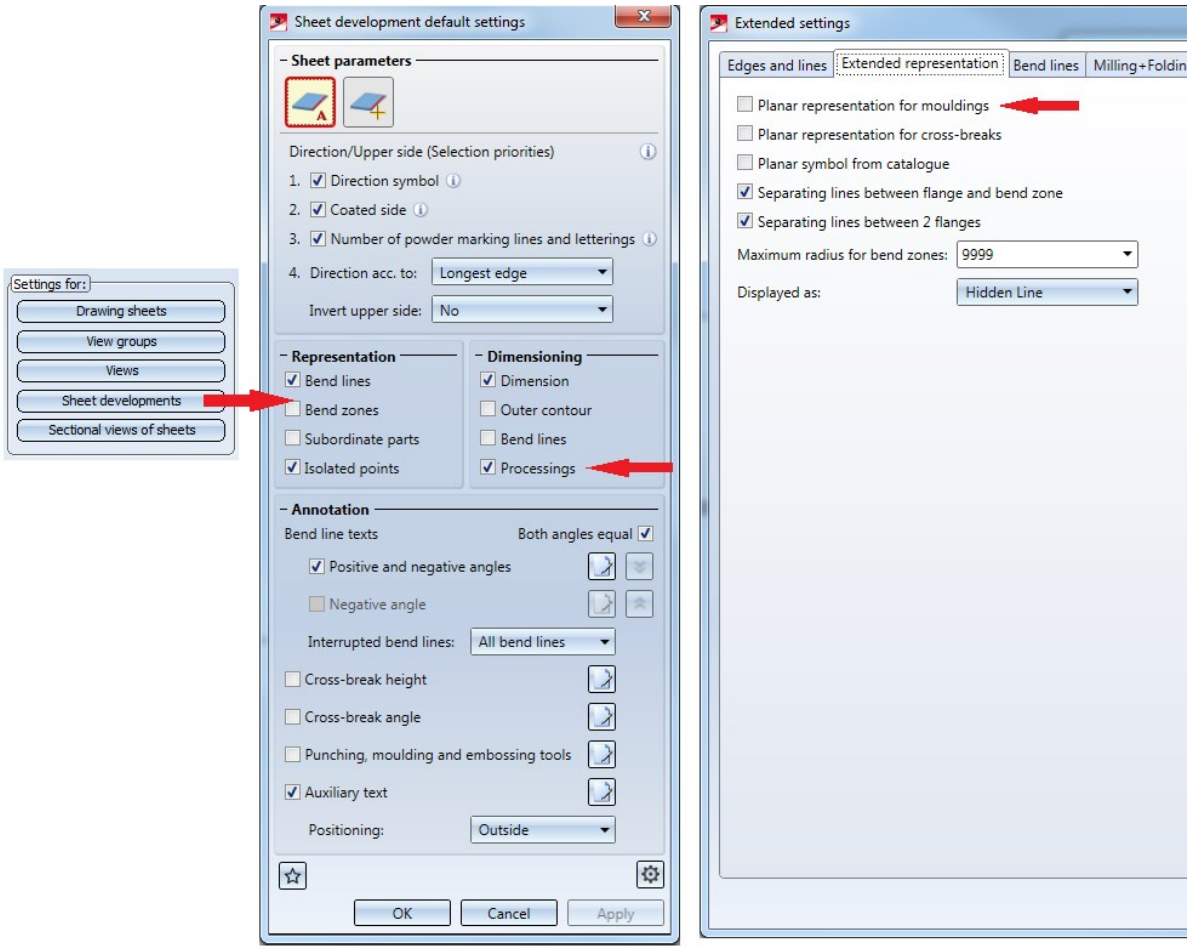
Drawing derivation - Dimensioning of the position of Sheet Metal

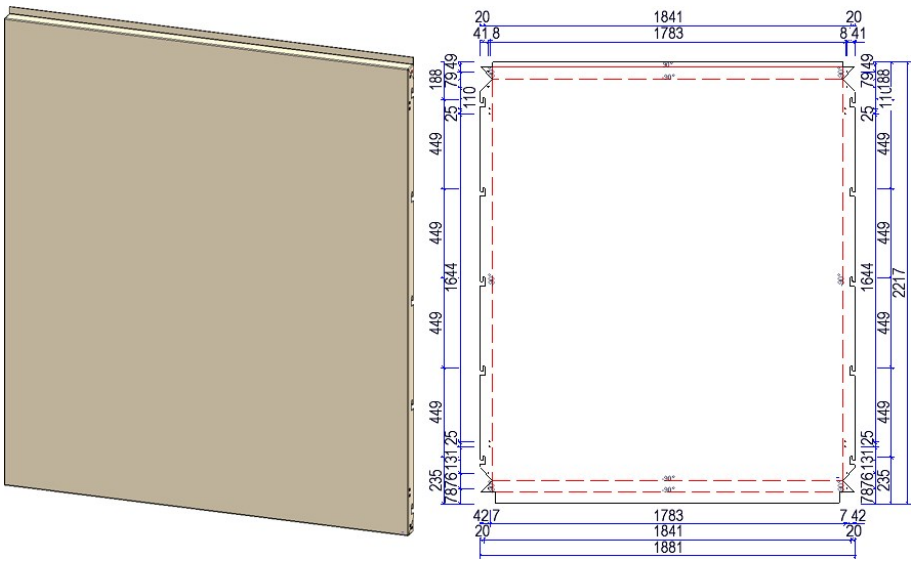
So far, only the position of Steel Engineering plates and beams has been dimensioned (as sub-parts). In practice, simple Sheet Metal plates consisting of one flange are in use instead of Steel Engineering plates. As of HiCAD 2019 SP1 also Sheet Metal plates are dimensioned automatically as attached parts of a welded assembly.

Drawing derivation - Dimensioning of Agrafes

If the construction contains sheets with agrafes, e.g. suspended ALUCOBOND® tray panels, the position of the agrafes in derived drawings is only dimensioned if the checkbox **Processings** is active in the settings for sheet development. However, agrafes are only dimensioned if the checkbox **Planar representation for mouldings** is NOT selected under **Extended settings > Extended representation**.

Start and end point of the agrafe are taken into consideration in dimensioning the outer contour.

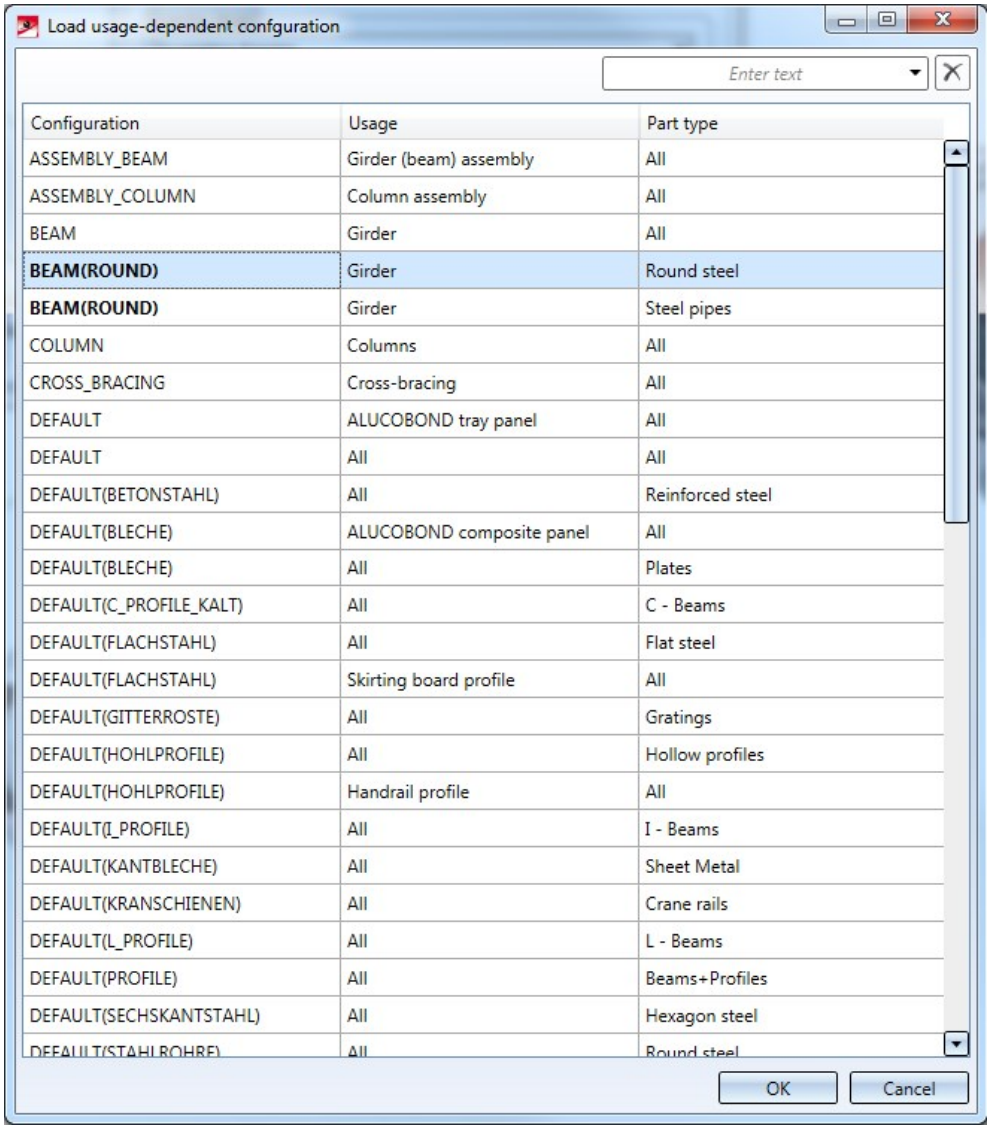




Drawing derivation - Usage-dependent configuration templates

The dialogue window Load usage-dependent configuration has been revised.

- On the left gap are the names of the available Configuration templates from the Configuration management (in Configuration management under **Automatic drawing derivation > Production drawing > Usage-dependent**).
- The middle column shows the usage type from which the configuration template is used.
- The right gap shows for which usage type the Configuration template is used.



The selection can be filtered using the search field at the top right of the dialogue window. Filtering can be cancelled by clicking on the delete icon.

The change affects the functions:

 **Drawing derivation** and

 **Rules for automatic dimensioning in derived drawings for dimension type "dimensioning rules"**

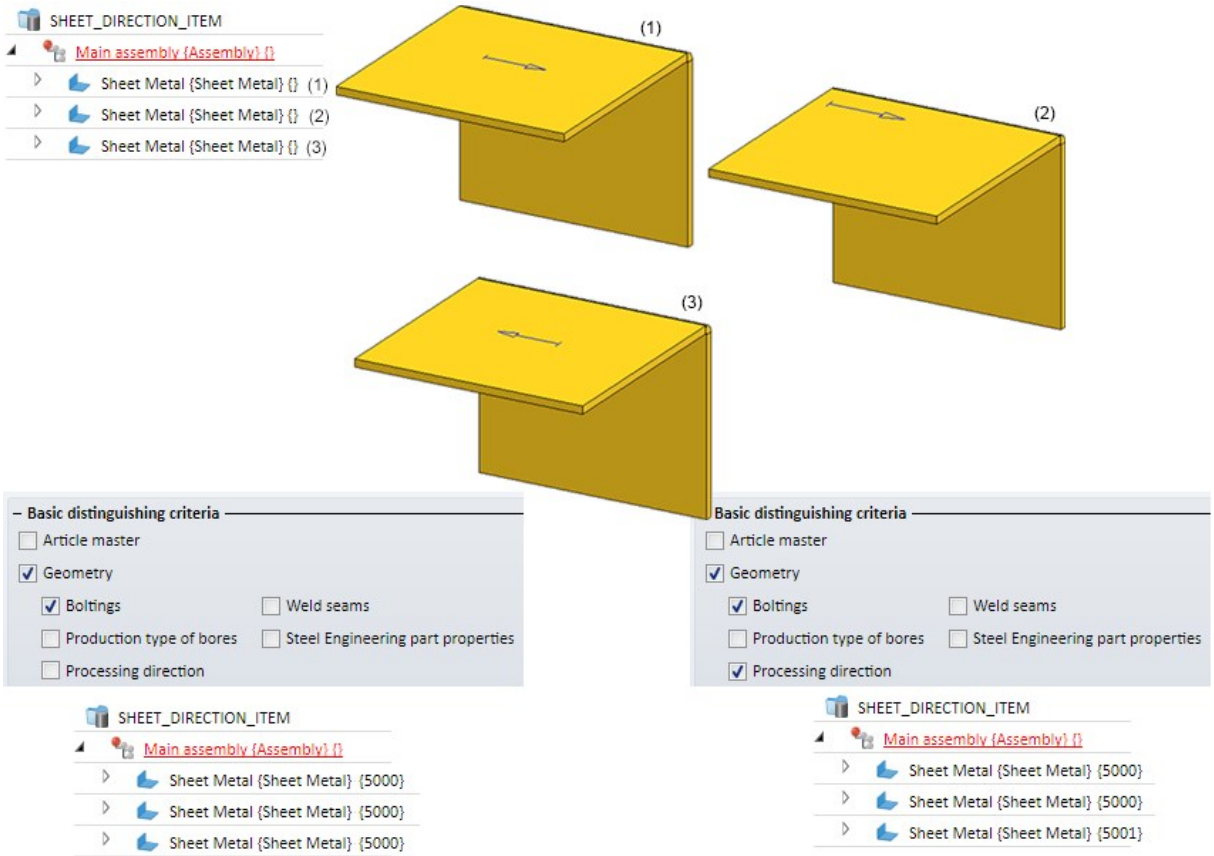
Itemisation - Consideration of Processing Direction

For the Itemisation a further distinctive criterion, the Processing Direction, can be chosen for Identical Part Search.

- Only direction symbols on top facet of flanges are being considered.
- Relevant to the comparison of Processing Directions are only your (processed) directions and **RTYP** entered in the catalogue (1: one-sided, 2: two-sided).
- Irrelevant to the comparison are thus the position of direction symbols (on which flange, where on the flange) and its geometry in particular.

An example:

The image depicts three geometrically alike sheets with Processing Direction. For Sheet 1 and Sheet 3 **Direction symbol one-sided 40 mm** was chosen, for Sheet 2 **Direction symbol two-sided 50 mm** was chosen. If the check-box Processing Direction is unticked during Itemisation, all Sheets are assigned to the same Item number. If the checkbox is ticked (despite different position of symbols on the flange), only Sheet 1 and 2 are assigned to the same Item number. Sheet 3 is assigned to a different Item number because the Processing Direction is opposing.



Itemisation - BOM-relevance of itemised parts

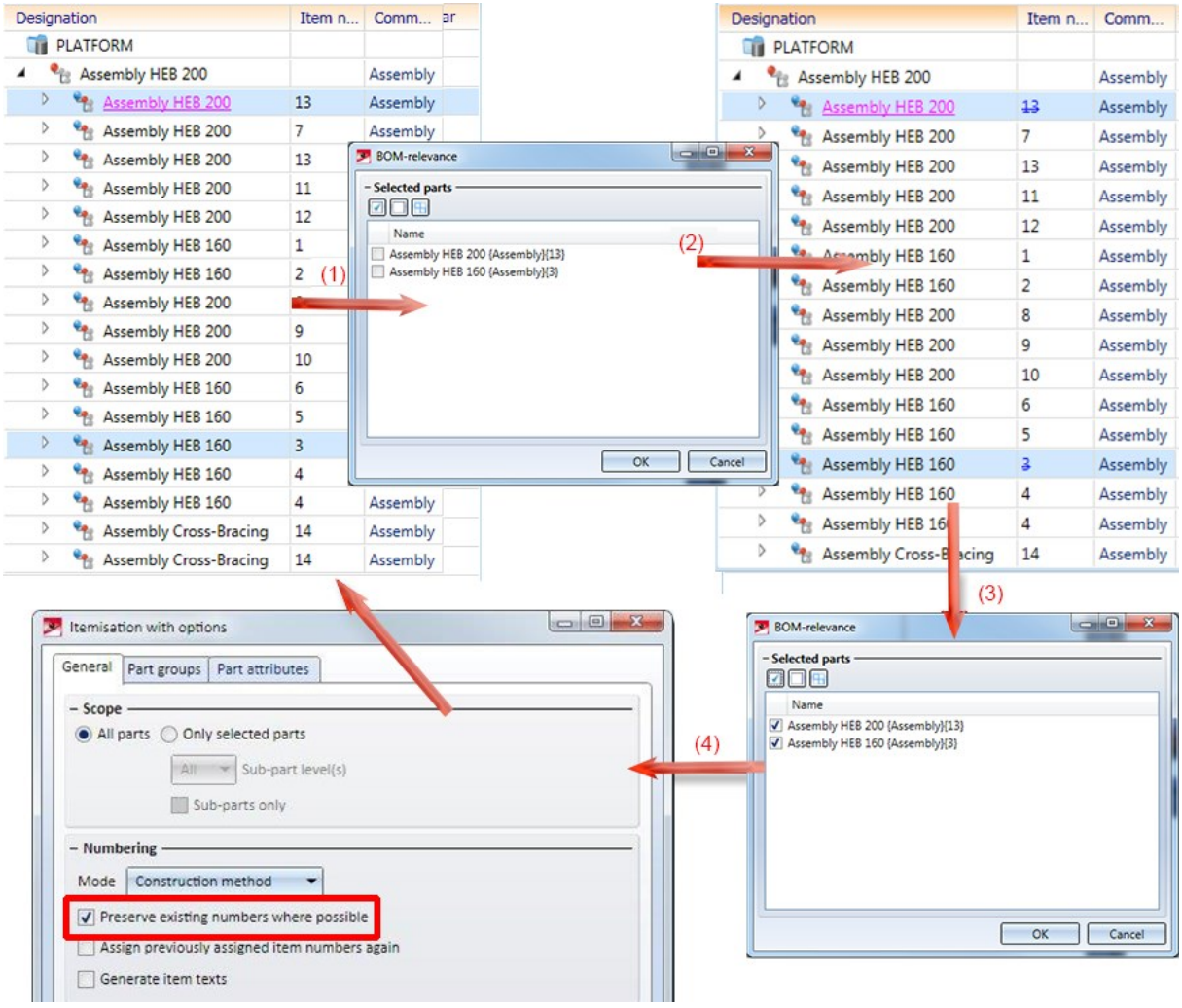
Previously, when the BOM relevance of a part was removed, the corresponding item data (item number, item text, and so on) was deleted. The item number of the part was then set to 0. As of HiCAD 2019 SP1, this behavior has been changed. If the BOM relevance of a part is removed, the item number is not removed, but only marked as invalid. However, the other item data, such as the item text, is not retained. Invalid item numbers are represented crossed out in the ICN.

If the BOM relevance is assigned to the part again later, the "old" item number is restored during automatic Itemisation, provided that the checkbox **Preserve existing numbers where possible** is active in the Itemisation settings. However, the current settings are used for the item text and not those that were active when the BOM relevance was canceled.

Use case:

In practice, it sometimes happens that parts lists are only created for a certain quantity of parts. To do this, you can remove the BOM relevance of the unwanted parts, create the BOM and then reassign the BOM relevance. Then start the automatic positioning again (with active checkbox **Preserve existing numbers where possible**).

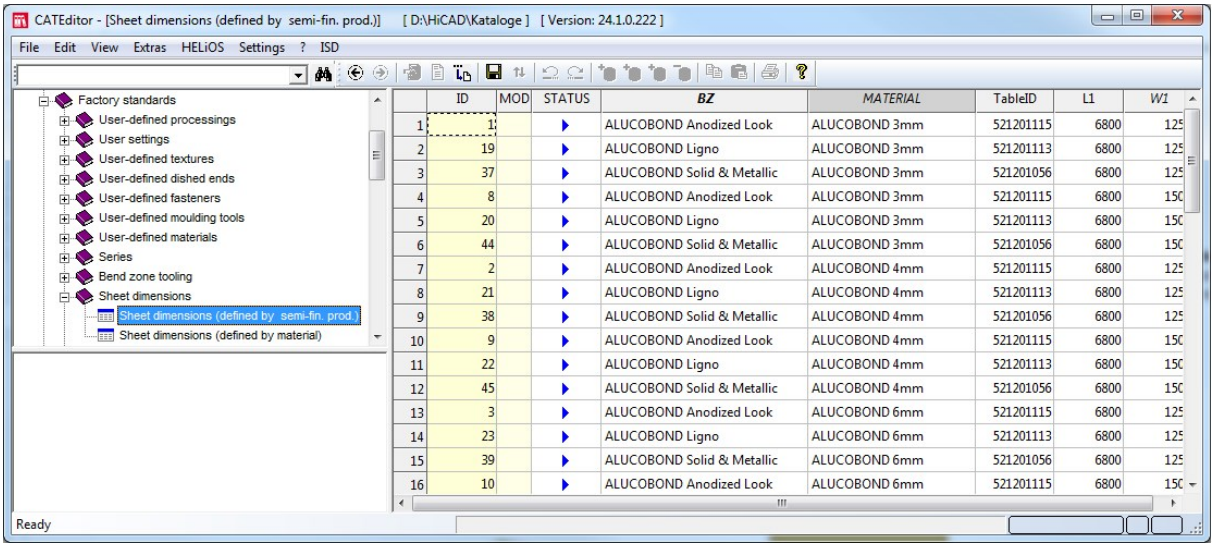
Example:



(1) Original model drawing, (2) Remove the BOM relevance for the parts with item numbers 13 and 3. The item numbers 13 and 3 are therefore invalid. (3) BOM relevance is set again. (4) The new positioning restores the invalid position numbers.

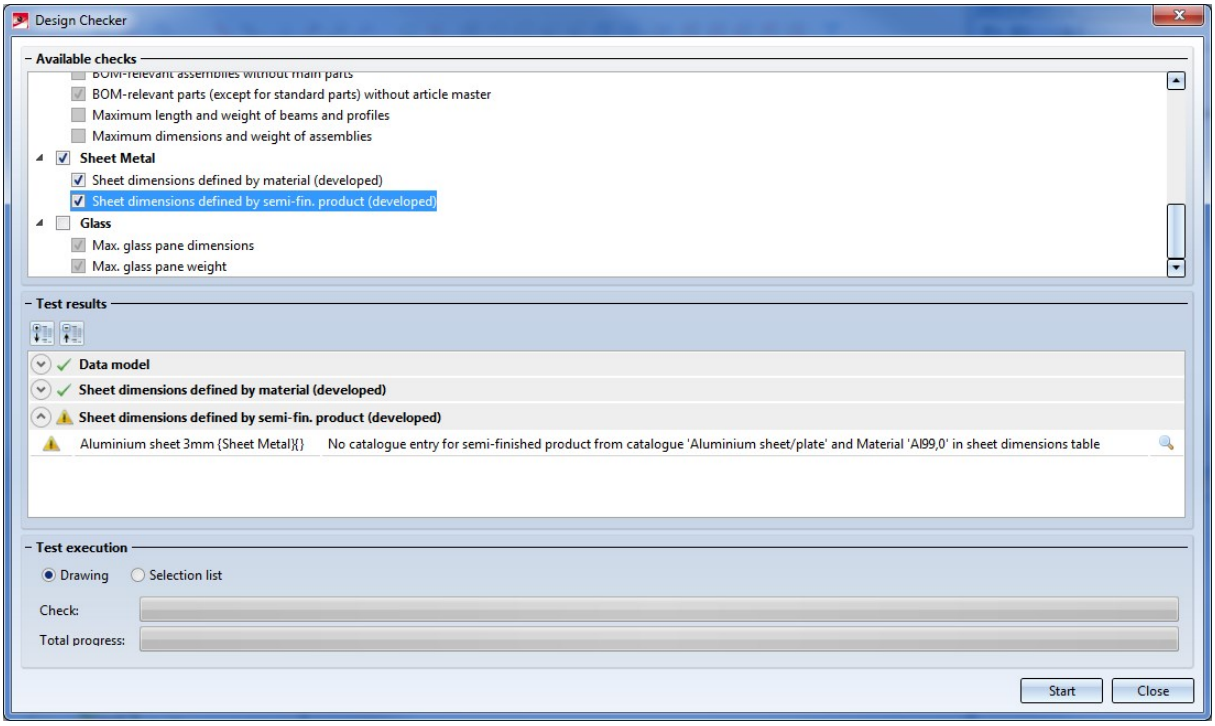
Design Checker - new check for ALUCOBOND Sheets

An additional check is available for Sheet Metal in the Design Checker. The new check **Sheet dimensions defined by semi-fin. prod. (developed)** checks the dimensioning for semi-finished products. For this, the table **Sheet dimensions (defined by semi-fin. prod.)** under **Factory Standards> Sheet dimensions** is evaluated in the CATeditor. The table shows the maximum length and width of the semi-finished product as a function of the TableID of the semi-finished product. The dimensions of the ALUCOBOND panels stored in this table are predefined by ISD. However, you can extend this table as required for other semi-finished products.



	ID	MOD	STATUS	BZ	MATERIAL	TableID	L1	W1
1	1			ALUCOBOND Anodized Look	ALUCOBOND 3mm	521201115	6800	125
2	19			ALUCOBOND Ligno	ALUCOBOND 3mm	521201113	6800	125
3	37			ALUCOBOND Solid & Metallic	ALUCOBOND 3mm	521201056	6800	125
4	8			ALUCOBOND Anodized Look	ALUCOBOND 3mm	521201115	6800	150
5	20			ALUCOBOND Ligno	ALUCOBOND 3mm	521201113	6800	150
6	44			ALUCOBOND Solid & Metallic	ALUCOBOND 3mm	521201056	6800	150
7	2			ALUCOBOND Anodized Look	ALUCOBOND 4mm	521201115	6800	125
8	21			ALUCOBOND Ligno	ALUCOBOND 4mm	521201113	6800	125
9	38			ALUCOBOND Solid & Metallic	ALUCOBOND 4mm	521201056	6800	125
10	9			ALUCOBOND Anodized Look	ALUCOBOND 4mm	521201115	6800	150
11	22			ALUCOBOND Ligno	ALUCOBOND 4mm	521201113	6800	150
12	45			ALUCOBOND Solid & Metallic	ALUCOBOND 4mm	521201056	6800	150
13	3			ALUCOBOND Anodized Look	ALUCOBOND 6mm	521201115	6800	125
14	23			ALUCOBOND Ligno	ALUCOBOND 6mm	521201113	6800	125
15	39			ALUCOBOND Solid & Metallic	ALUCOBOND 6mm	521201056	6800	125
16	10			ALUCOBOND Anodized Look	ALUCOBOND 6mm	521201115	6800	150

Example: Sheet dimensions defined by semi-fin. product



Multiple part selection

In previous versions, the selecting of parts in rectangle has behaved differently in shaded and non-shaded views as well as in referenced and "regular" parts. As of HiCAD 2019 SP1, the conduct is standardised as follows:

To select all parts/assemblies within a rectangle for processing, draw a rectangle by simultaneously pressing the left mouse button and the CTRL button. With regard to assemblies, the following should be observed:

- The behavior is independent of the representation mode.
- Referenced and non-referenced parts behave equally.
- If assembly points are assigned to one assembly and at least one of the assembly points lies within the rectangle, the complete assembly including all of its subordinate parts are being selected.
- If no assembly points are assigned to an assembly or none of the assigned assembly points lie within the rectangle, only those parts of the assembly are being selected which lie completely within the rectangle.
- A special treatment applies to sheets. If all flanges and bending zones of a sheet are in rectangle, the corresponding superordinate part is also selected.

- There is no special treatment for pipe parts.

Information window Collision check

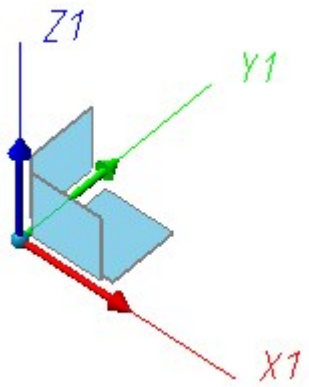
The collision check functions now also display a message if no collision was found:

Article master sync when saving

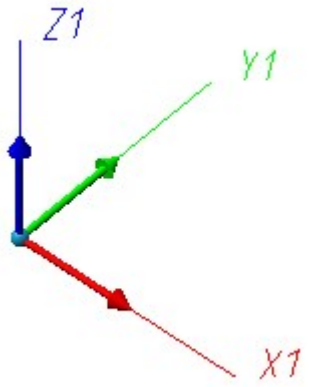
With the function **Article master sync when saving**  you can specify which HiCAD part attributes are to be assigned to HELiOS article master when saving. These attributes are then compared each time they are saved. As of SP1, this function is no longer available in the menu function under **Drawing > Itemisation/Detailing < Attr...**. Instead, the function is now available via the menu function **HELIOS PDM** under **Others > Link....**

Selection of CS axis, plane and zero point in drawing

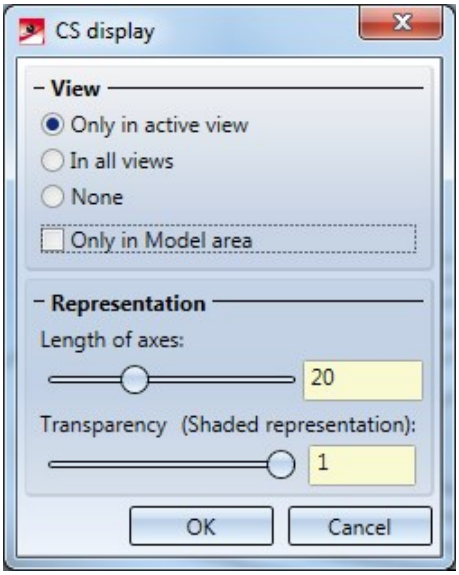
For functions that refer to axes, planes or the origin of the active coordinate system, it was previously always possible to select them via the context menu. Since HiCAD 2019 SP1, additional symbols are now displayed in the drawing, which can be clicked directly for selection.



Only the symbols that can be used are displayed; if, for instance, a direction is queried, only the coordinate axes and the origin (as start point of a direction) are available for selection:



You can specify the transparency with the option **Transparency (Shaded representation)** under **Drawing > Others > Extras > Settings, permanent > CS display** .

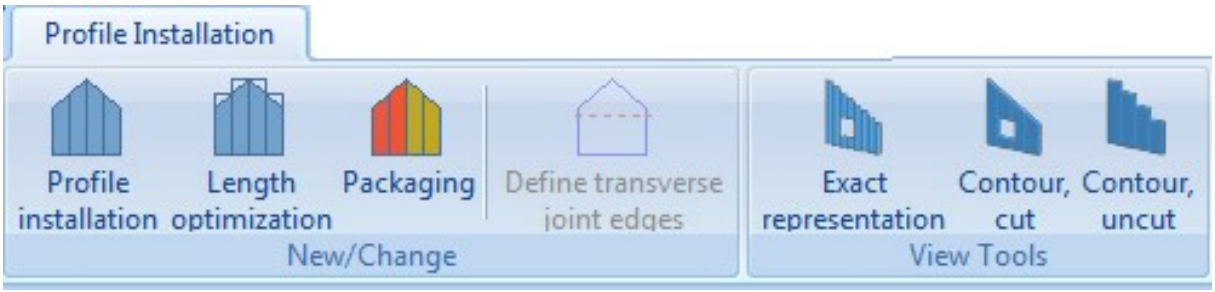


Major Release 2019 (V 2400)

New Module Profile Installation

The new module **Profile Installation** (optional) provides convenient options to place installation elements on walls and roof surfaces within a short time.

You can find the functions on the Ribbon **Profile Installation**.



Improved performance

- The HiCAD function dialogues work significantly faster because of internal adjustments (this concerns body-wise saving).
- The collision check works much faster by means of factor 3 as of HiCAD 2019.

Itemisation

As of HiCAD 2019 it is automatically determined which mode is being used for Itemisation, i.e. it is not possible to select the mode via Configuration Management.

It depends on the current drawing which mode is active:

- For drawings created with HiCAD 2019 or older drawings which have not been itemised yet, **Standard Itemisation** is being used (since HiCAD 2018).
- Conversely, the mode **Itemisation up to HiCAD 2017** is only used for drawings created with a version before HICAD 2019 and which have been -at least partially- itemised with the old mode.

Please note that the mode **Itemisation up to HiCAD 2017** will not be available anymore by HiCAD 2020. For converting Itemisations to the Standard Itemisation as of HiCAD 2018, the function **Convert to standard itemisation**  is available.

New features for Textures

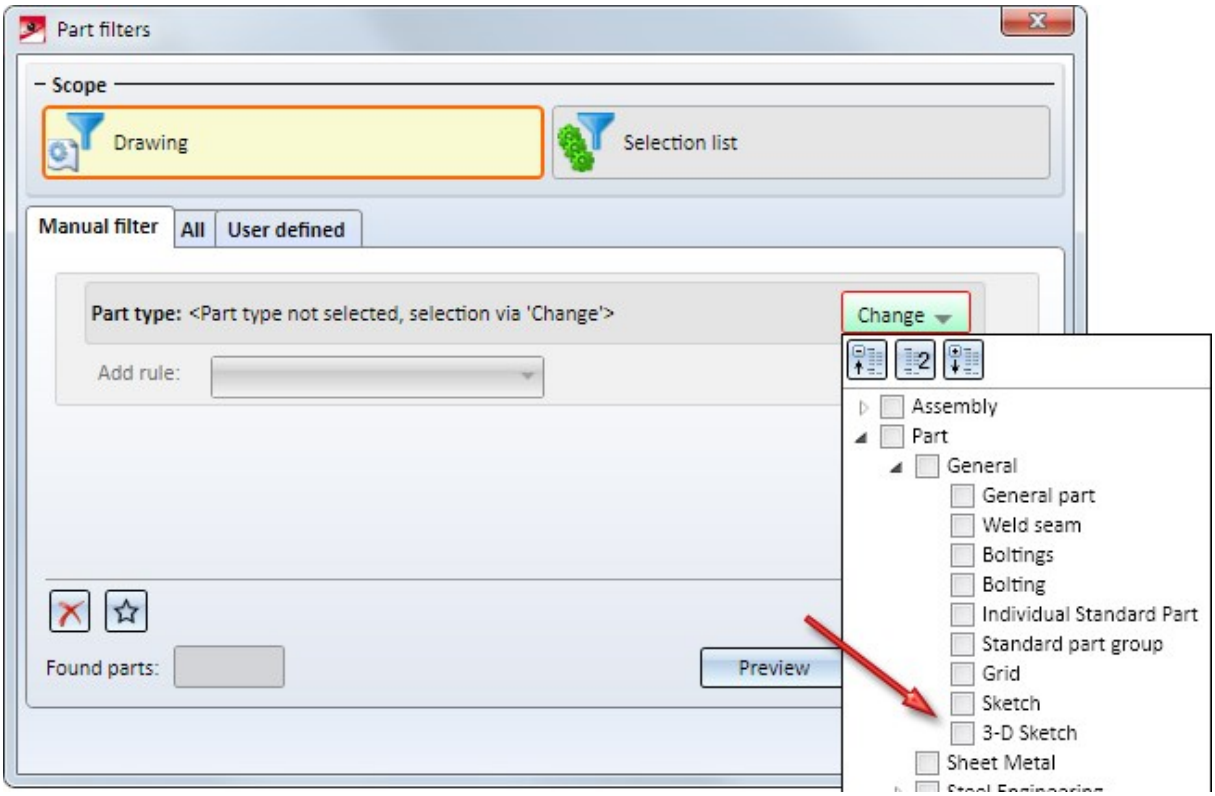
As of HiCAD 2019 Textures are
not any more assigned via the docking window **Visual Effects**. The following function is available instead (see also HiCAD 3D - What's New?)

	**Texture - New (3-D Part list)** applies texture to active part or active part list of a texture	via **3-D Standard > Tools > Attr...** or in the context menu of a part list via **Properties**
	**Texture - New (3-D Part)** applies texture to active part	via context menu for 3-D parts under **Properties**
	**Change texture, active part (3-D Part)** changes texture of active part	via context menu for 3-D parts
	**New texture** create new textures/changes existing textures	via **Drawing > Others > Extras**

In addition, HiCAD 2019 offers a multitude of textures. These are available in HiCAD Catalogue under **Textures and Colours > Textures**.

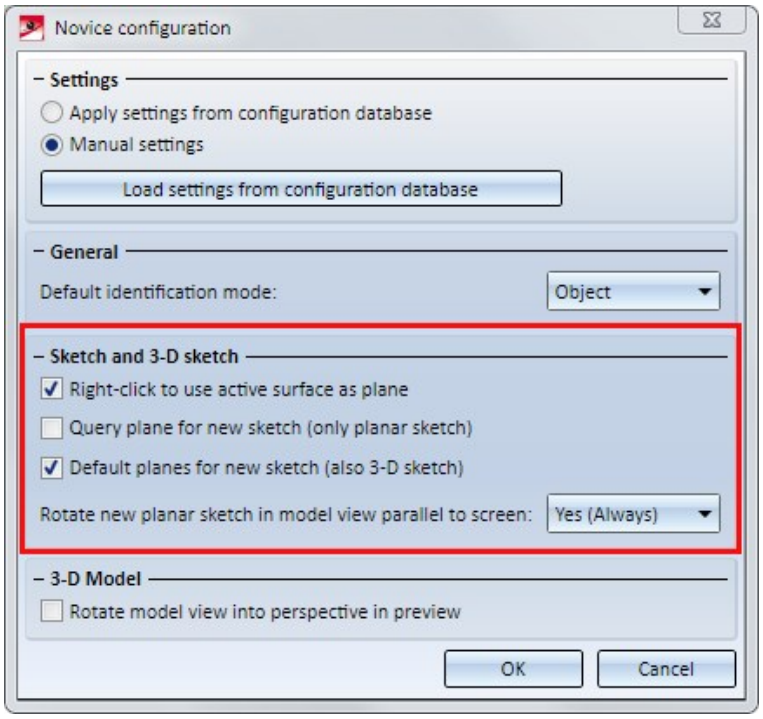
New Part Type 3-D Sketch

Planar sketches and 3-D sketches are different part types as of HiCAD 2019. Thus, Planar sketches  and 3-D sketches  can be marked differently and the Part filters for the Toolbar can be set detachedly for Planar sketches and 3-D sketches.

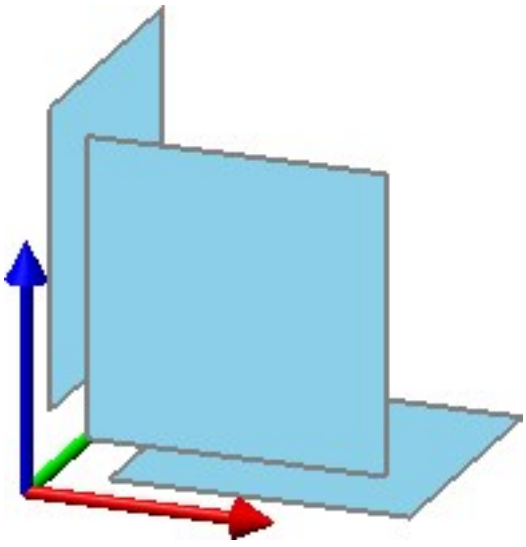


Novice - Configuration - Sketch

The Novice configuration settings have been revised:

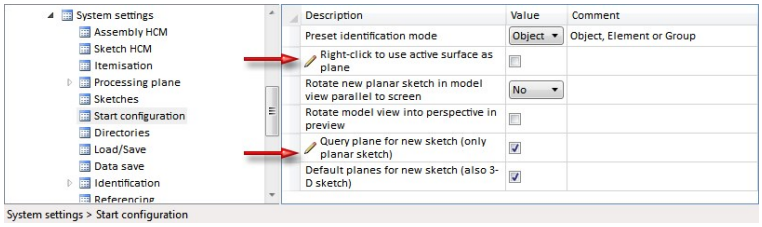


- The checkbox **Default planes for new sketch** is also applicable for 3D sketch. Here, a preview of the default planes is shown. The desired plane can be easily selected with the cursor.



The setting (Yes) affects planar sketches only if the checkbox **Query plane for new sketch** is ticked.

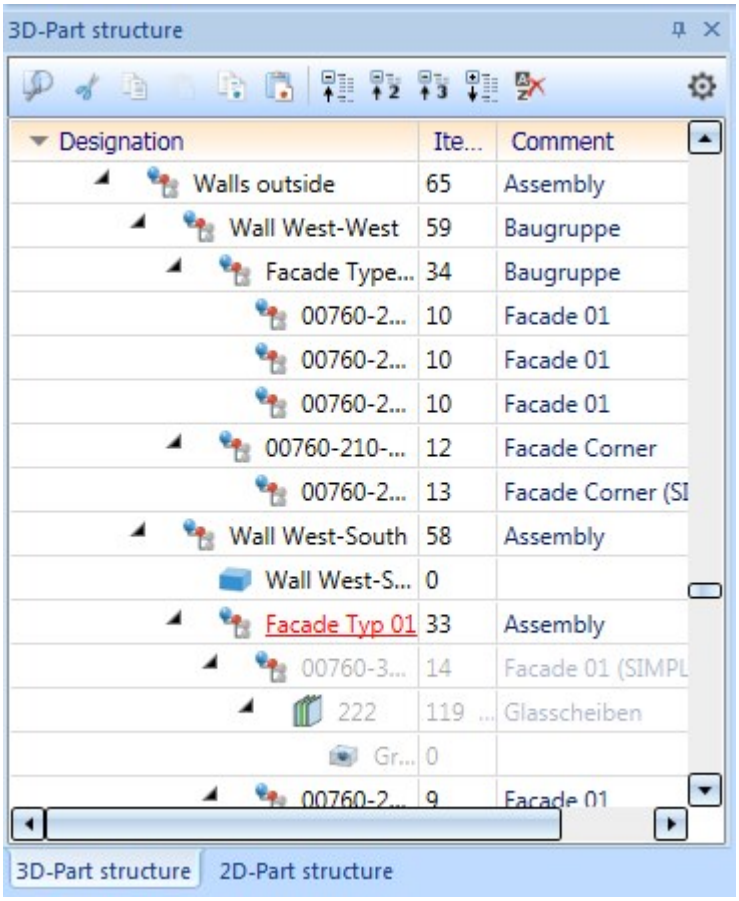
- If you have chosen the Novice Configuration in the **Extended settings** of the HiCAD installation, or via the **Parameter configuration** (ParKonfigComp.exe / ParKonfigUser.exe), this will also influence the settings in the Configuration Editor!



New Part Structure Representation in the ICN

The Part structure representation in the ICN has been redesigned:

- Improved performance, notably with complex drawings
- Tabular, multicolumn representation
- Sorting by click on the column header
- Individual configuration of the representation
- Fast switching of the representation of HiCAD and HELiOS attributes



In context with the new representation of the 3-D Part structure, the following system files have been renamed:

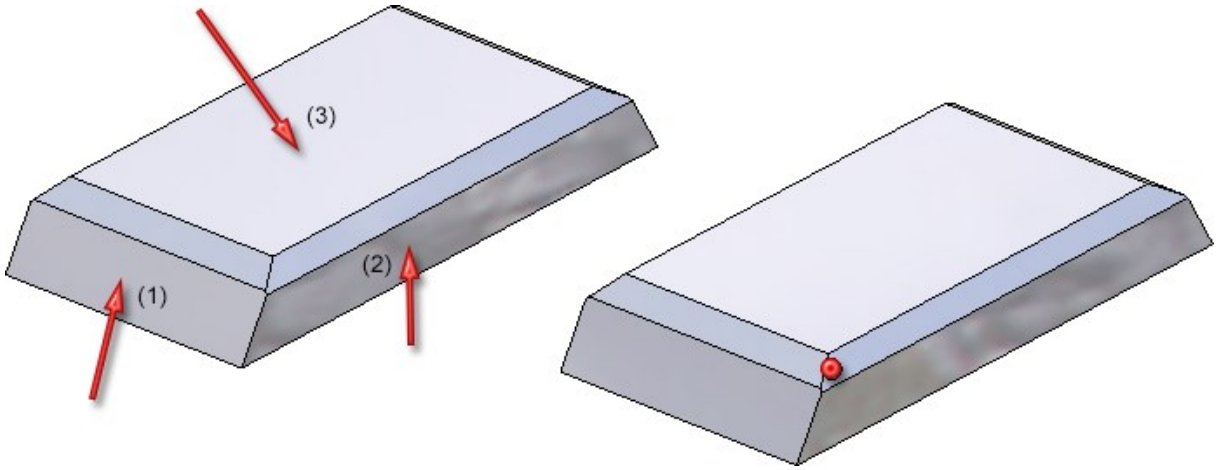
Up to HiCAD 2018	As of HiCAD 2019
BRW_3DTEILATT.HDB	ICN3D_MULTIATTRVIEW.HDB
BRW_3DDBATTR.hdb	ICN3D_MULTIATTRVIEW_DB.HDB

New 3-D Point option - Intersection of three planes

With the new function **Intersection of three planes**  it is possible to determine an intersection - even in theory - of three planes. This is often used for Element Installation to specify the origin of the Fitting CS in order to determine the exact fitting direction.

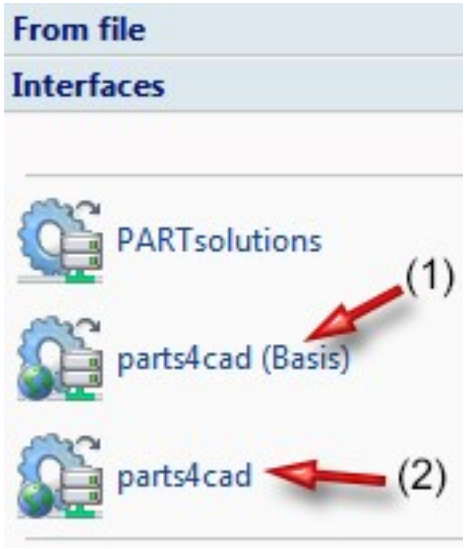
Example:

Here, the intersection of plane (1), (2), and (3) have been identified by selecting the respective area.



Free general access to PARTS4CAD

As of HiCAD 2019 you are granted with free access to more than 500 manufacturer's catalogues via the web catalogue parts4cad by Cadenas. The access to the extended catalogue range (DIN/ISO/EN… Norms + all 800 manufacturer's catalogues) is fee-based.



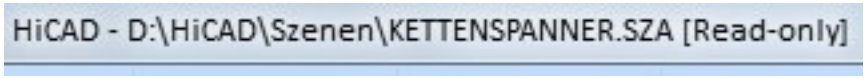
Standardization of terms

The following term has been standardized as of version 2019:

Previously	As of HiCAD 2019
Figur	2-D Part

Advanced display in the title block

Furthermore, it is displayed whether a read-only file has been opened in the title block.



A file is considered as read-only file, if

- it has been opened with the HELiOS function **Open, read only** ,
- the Workflow status is indicated with **released,**
- if it has been opened with **Open drawing, read-only** via Explorer in HiCAD or
- you do not have the required user rights for opening the file.

Select superordinate part

For selecting a superordinate part in a drawing, click on the part with the left mouse button and hold it. Click additionally on the right mouse button. HiCAD automatically selects the closest superordinate part and marks it. Clicking again on the right mouse button enables further search for parts.

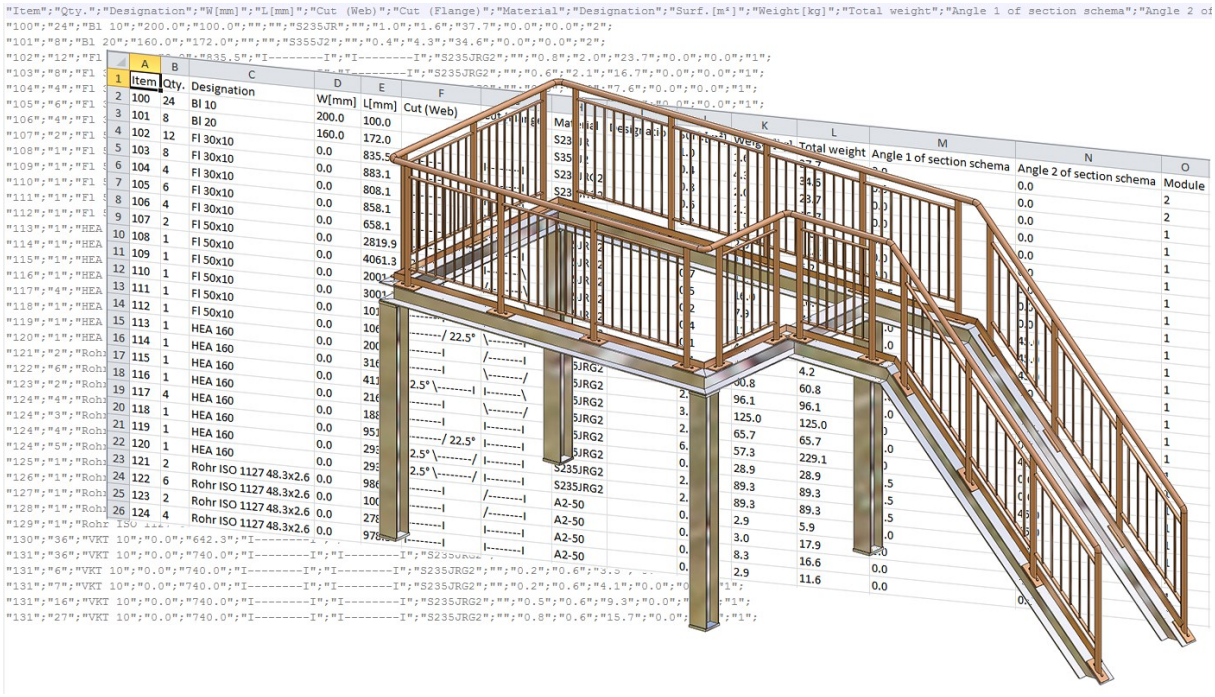
So far, this function has been enabled only for general selection of parts before calling a function. As of HiCAD 2019, this is also possible when a part selection is started via a new function dialogue, for instance the definition of exploded views.

QOMET-specific BOMs

HiCAD 2019 supports the export of QOMET-specific BOMs. QOMET is an ERP system specialised in steel and metal engineering.

The BOM can either be exported for the complete drawing or active part and contains particular columns of the Excel BOM in Steel Engineering (hicad_Stahlbau.xlsc) relevant for the QOMET system in particular. The file is exported as CSV.

The respective functions for exporting the BOM are available under **Drawing > Save/Print > Save as...**



Drawing derivation - Sectional views of Sheet Metal parts

As of HiCAD2019 continous sectional views are only created if the bending zone is parallel or perpendicular to the view CS.

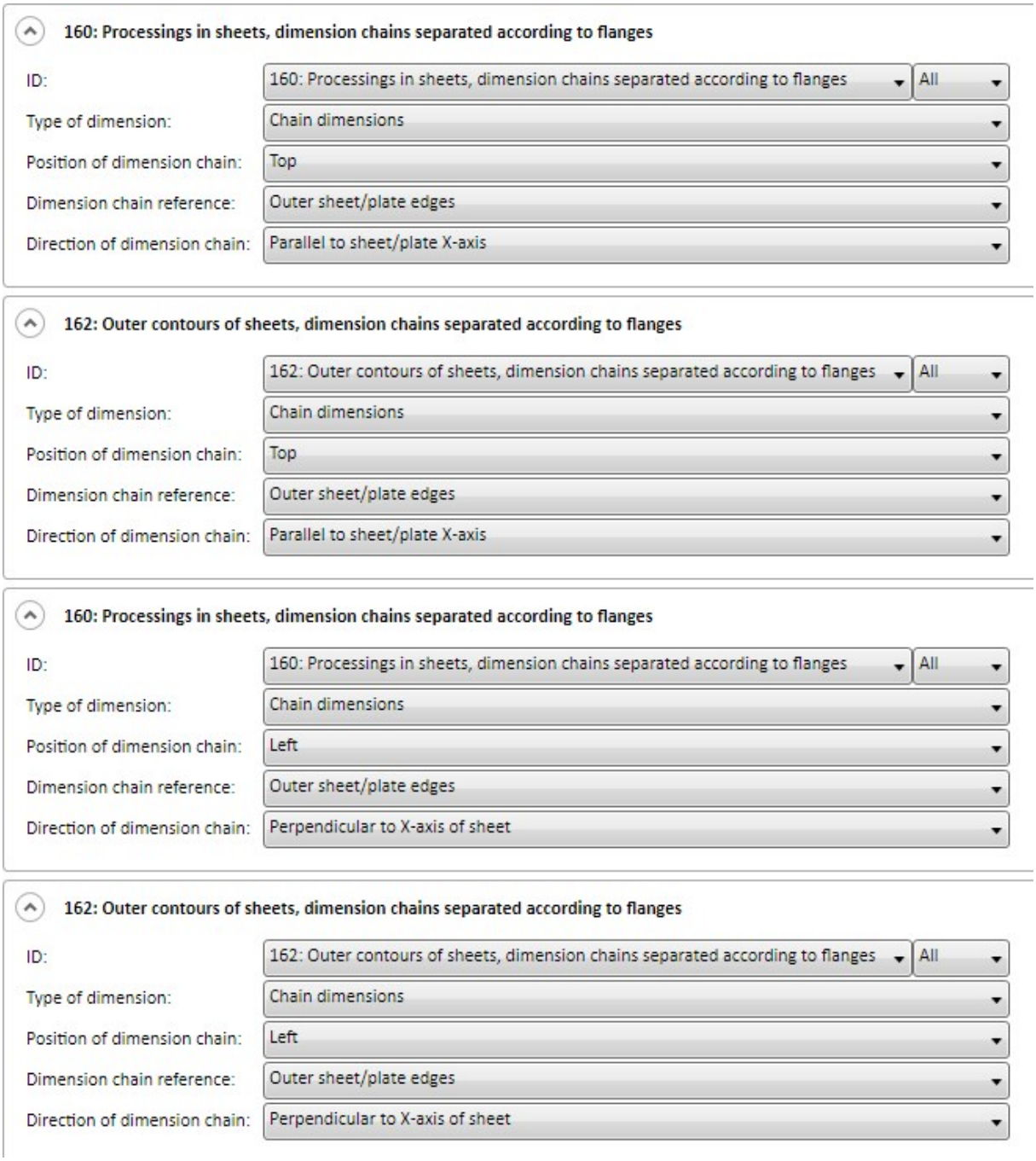
Drawing derivation - Enhanced dimensioning rules for Sheet Metal parts

In facade engineering, sheet metal is used as bent profile or tray panel. It is often requested that these sheet metals are also to be dimensioned automatically in standard views when creating workshop drawings (top view, side view, front view) and that the processings are taken into consideration next to the outer dimensions.

157	SHEETMETALBORES	Processings in sheets
158	SHEETMETALBORES_ VERT	Processings in sheets incl. processings perpendicular to view
159	SHEETMETALOUTLINE	Outer contours of sheets
160	SHEETMETALBORES_ SEPERATED	Processings in sheets, dimension chains separated according to flanges
161	SHEETMETALBORES_ VERT_SEPERATED	Processings in sheets incl. processings perpendicular to view, dimension chains separated according to flanges
162	SHEETMETALOUTLINE_ SEPERATED	Outer contours of sheets, dimension chains seperated according to flanges

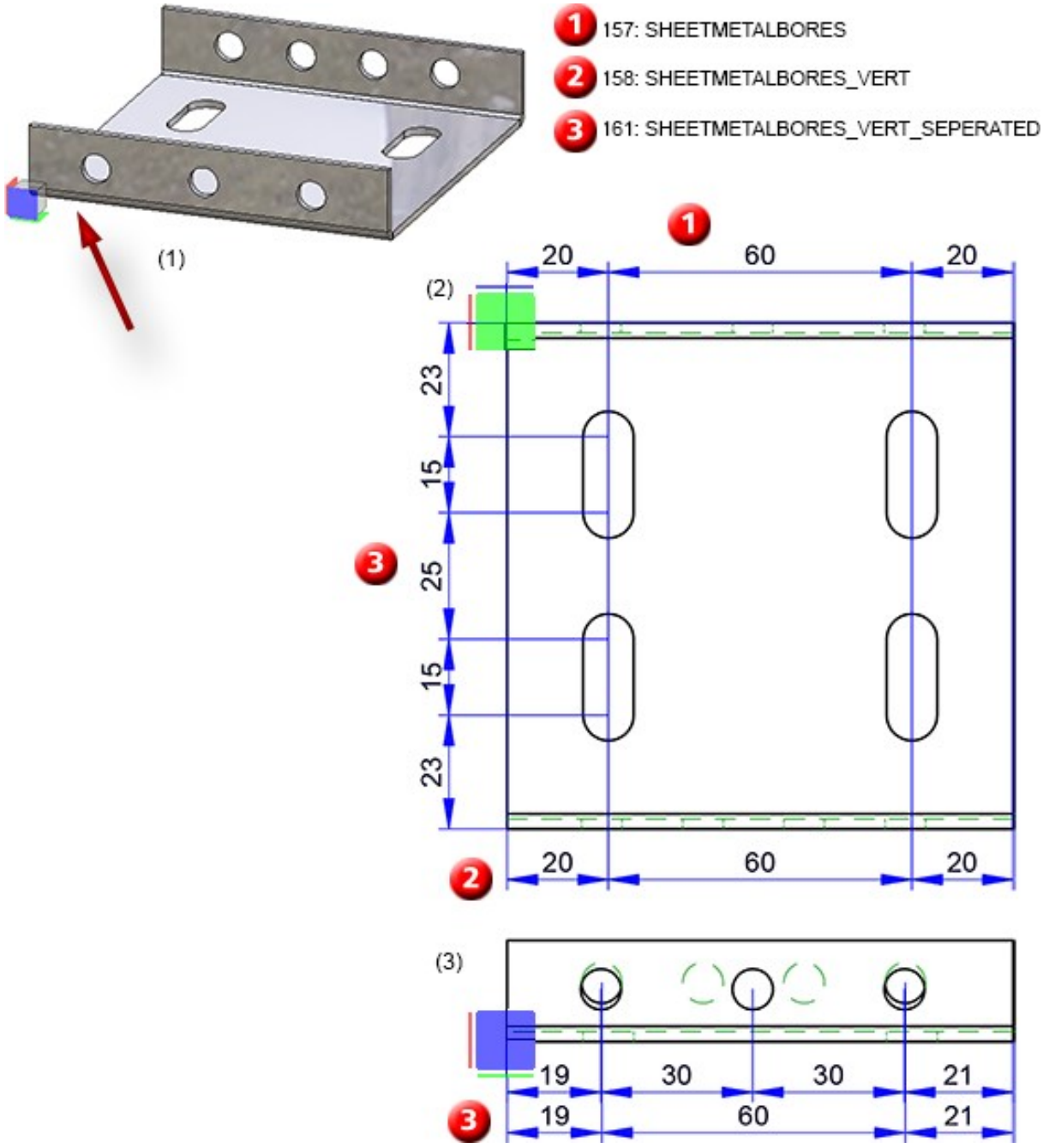
The sheet metal is treated as superordinate part including its flanges and bending zones. The prerequisite for these dimensioning rules is that an appropriate coordinate system has been defined with the functions via **Alignment of derived drawings**in the context menu for 3-D parts.

Recommended settings rule 157 - 162



Example:

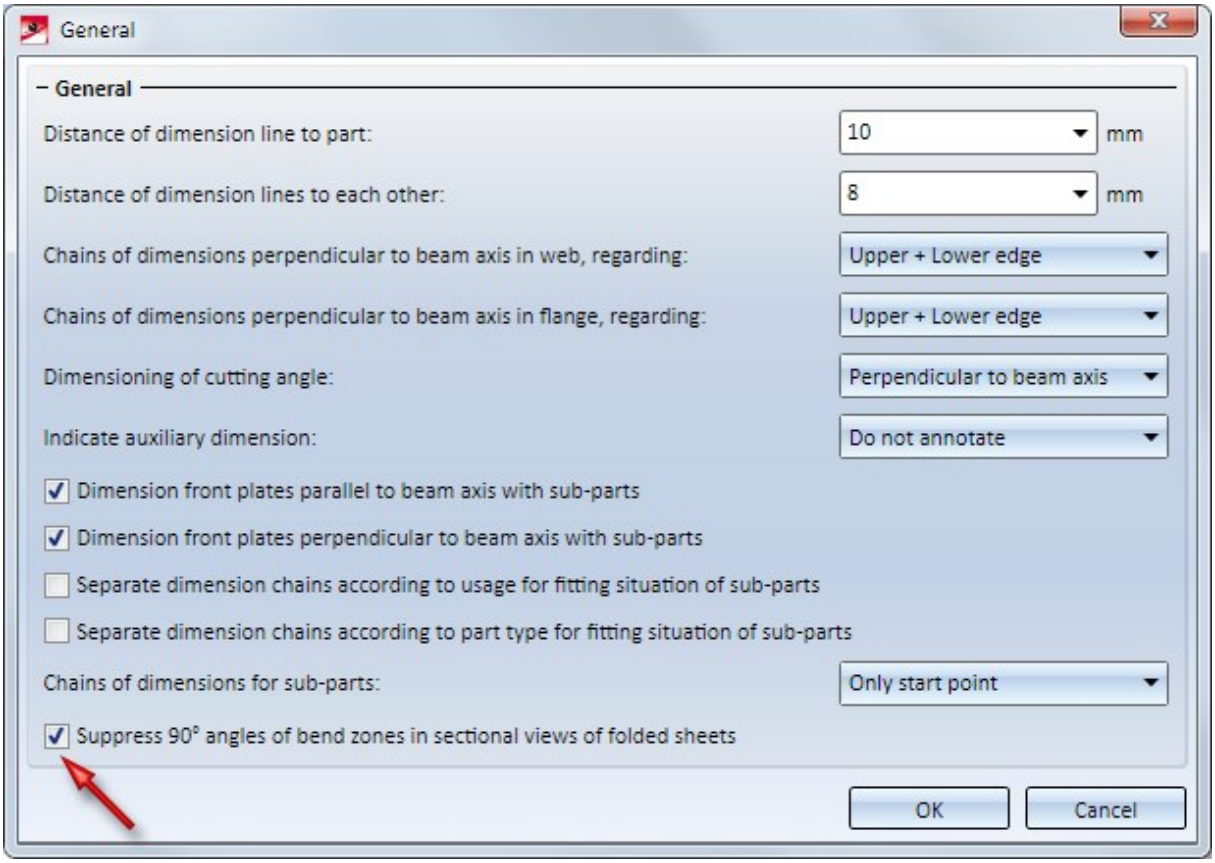
The following picture shows a sheet metal with different bores. For the alignment of derived drawings the top view has been placed in the flange surface (1). (2) shows the front view of the drawing derivation and (3) the top view. The configuration DEFAULT (SHEETMETAL) has been used as template - enhanced by rule 157, 158 und 161.



Drawing derivation - Settings for dimension type "conventional"

As of HiCD 2018 it was possible to determine whether the 90° bend angle should be dimensioned for drawing derivation in the sectional view of folded sheets. In order to disable the dimensioning of 90° bend angles the system data STAB3DPAR.DAT (for new customers) and STW_DIMSETTINGS.XML had to be adjusted manually so far.

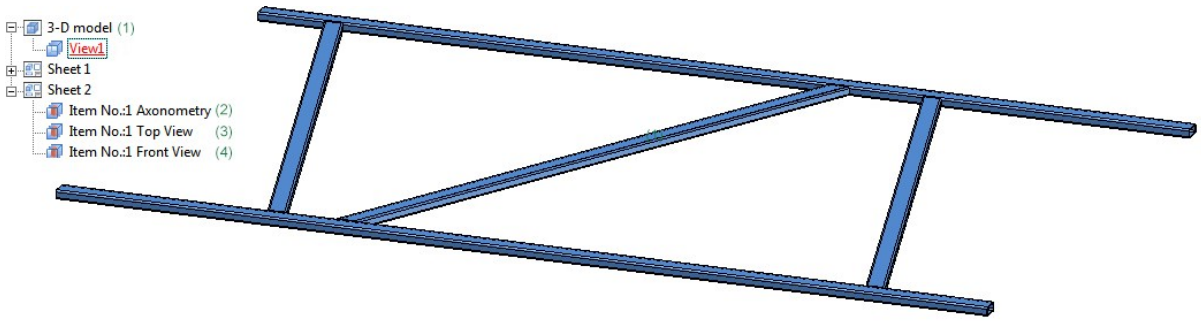
As of HiCAD 2019 this function can be set via **Dimensioning Settings**  . The dialogue window **General** has been enhanced with the checkbox **Supress 90° angles of bend zones in sectional views of folded sheets**.



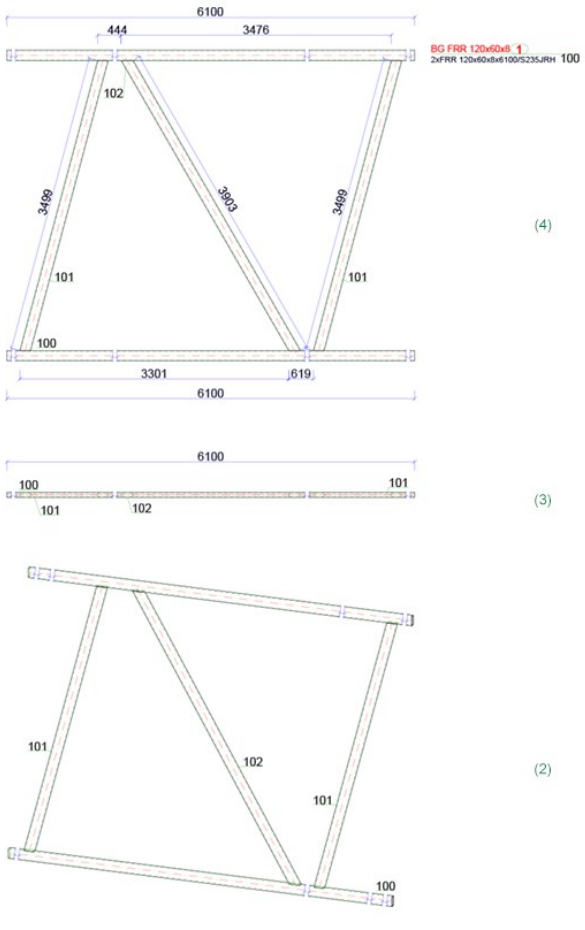
Drawing derivation - Shortened view

As of HiCAD 2019, shortenings are displayed simplified.

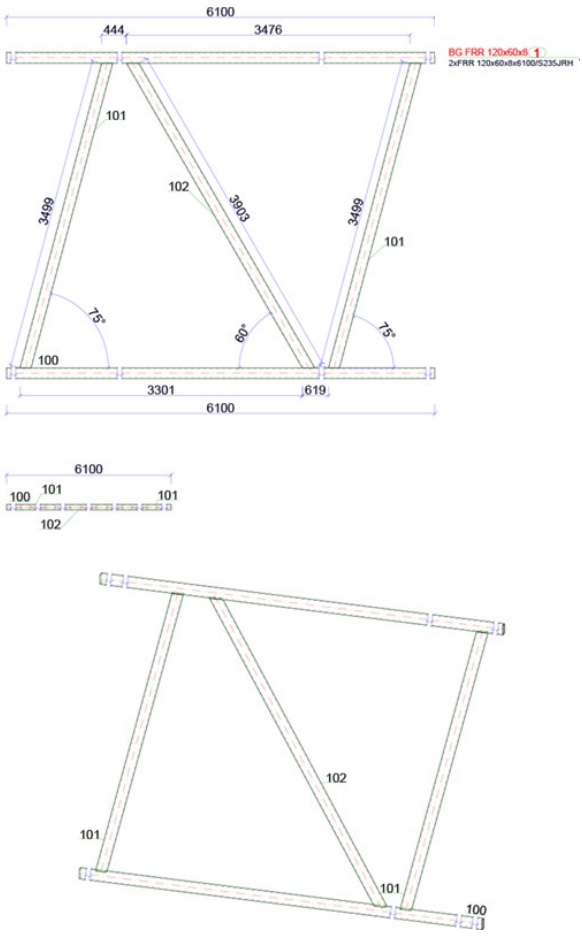
The following depiction shows the difference:



HiCAD 2019



< HiCAD 2019

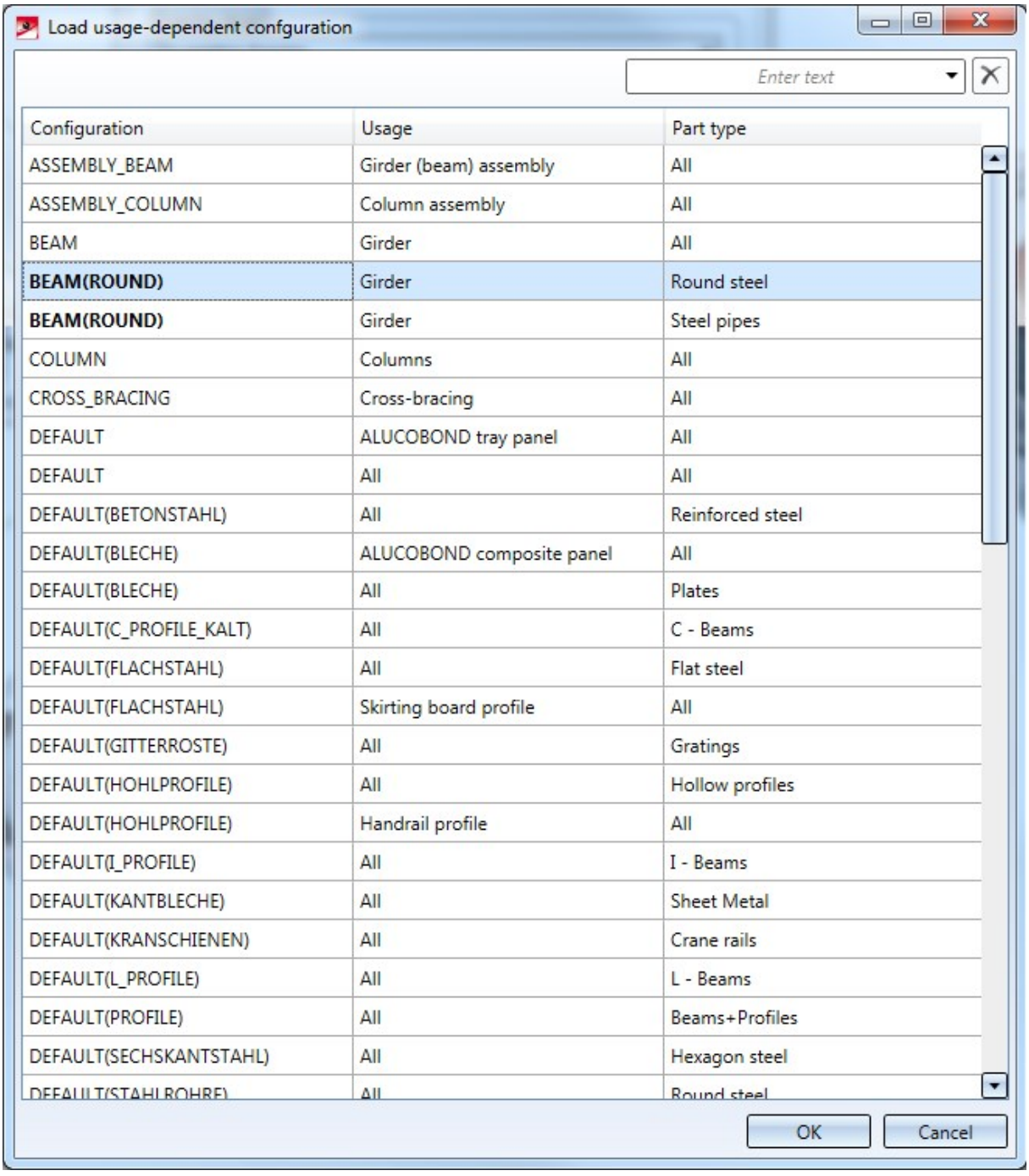


Loading usage-dependent configurations

The dialogue window for usage-dependent configurations has been rearranged more clearly. This concerns the following functions in particular:

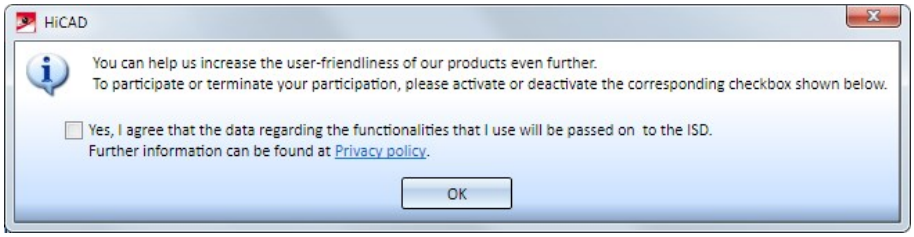
- Drawing Derivation and
- Dimensioning Rules Editor

The first column shows the configuration. The second column shows the usage for the configuration.



Customer feedback

Via the function Help > Customer feedback now also the Data protection notice: Collection of data for constant GUI optimisation can be opened.



The same applies to the HiCAD installation for the setup step **GUI optimisation** .

TABPAR.DAT -> Configuration management

So far, this system file has contained settings for working with a tablet as input device as well as regulation for time interval for double-click. This file will be cancelled as of HiCAD 2019 because tablets are not being supported as input device. The parameter **Time interval for double-click** can be defined via **System settings > Identification** in the configuration management.

AutoSync of Part Attributes and HELiOS Article Master Data

With the new function **Article master sync when saving** you can specify which part attributes are to be synchronized with which article master data. These attributes are synchronized automatically with each saving procedure.

Please note that the **Transfer part attributes to HELiOS** checkbox must be activated the ISD Configuration Management via **System settings > HELiOS**.

You find the function via Drawing > **Itemisation/Detailing > Attr...**

ISD License Manager - New Tab "Server Monitor"

WIth the new tab Server Monitor you can look at available and occupied modules. By means of the listed licenses you can easily find out which colleague blocks licenses and e.g. contact them in case if you need a license.

2-D

Service Pack 1 2019 (V 2401)

Maximum number of layers

In previous versions the possible maximum number of layers in HiCAD could be set in the Configuration Editor at **System settings > 2-D Lines** with the parameter **Maximum number of layers**. From Service Pack 1 onwards, this parameter will no longer be available. The number of possible layers has now been set to constantly 1000. (Layer 0 - 999).

Translation tool - Ignore upper/lower case letters

In the Configuration Editor you can now also specify whether upper and lower case letters are to be taken into account when localizing texts with the translation tool. The default setting is **Consider**.

You can find this option at **System settings > Miscellaneous > Translation -Upper and lower case letters**. For a change of this setting to take effect, HiCAD needs to be restarted.

The setting will not be saved in the drawing or in the text; it is a pure system setting.

Major Release 2019 (V 2400)

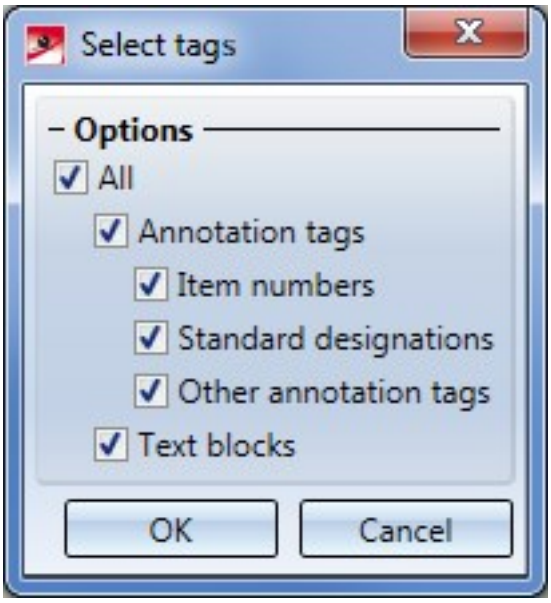
Delete annotation, in rectangle

The **Delete tags** menu of the **Delete text** function now features the new **In rectangle**  option.

This function deletes all annotations the insertion point of which lies within one or several rectangles. The actual deletion will only take place after a right-click.

One distinguishes between:

- Tags with item numbers (**Itemize part**)
- Tags with standard designations (**Standard designation**)
- Other designation tags (**Text with leader line**) and
- **Text blocks.**



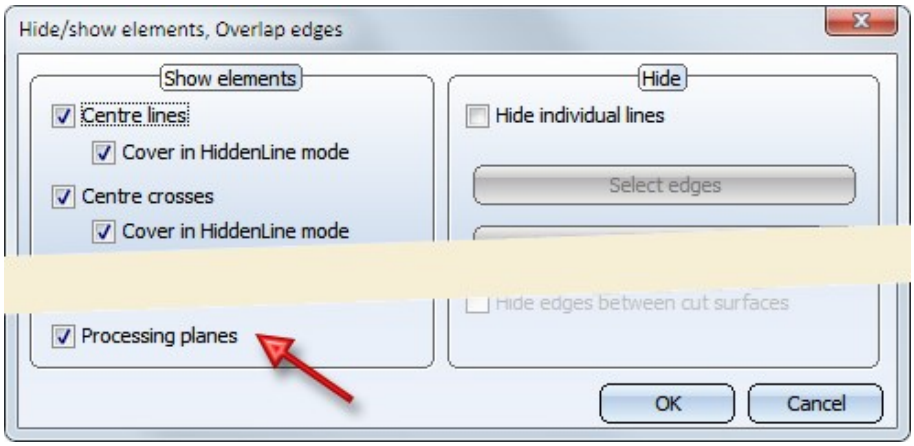
3-D

Service Pack 2 2019 (V 2402)

Views

Processing planes in shortened views

In previous versions, it was not possible to display shortened views in processing planes. From HiCAD 2019 SP2 onwards this will be possible, i.e. the switching of **Processing planes** in the dialogue window of the function Views > Properties > Elements  can now also be applied to shortened views. In particular, you can now hide and show processing planes in shortened views of workshop drawings.



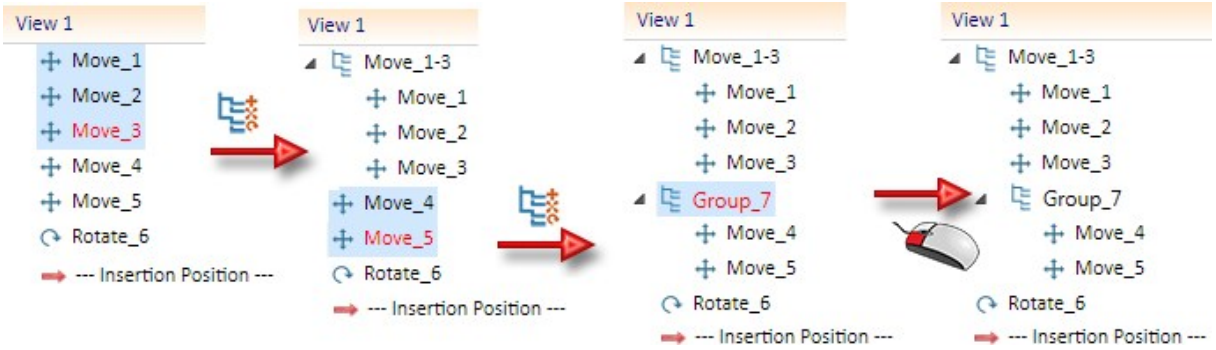
Exploded views - Combine transformations into set

In the Exploded views docking window you can now combine several transformations in succession into one group, e.g. in order to get clearer and better structured explosion logs for complex exploded layouts. For this purpose the following functions are available in the context menu of the explosion log:

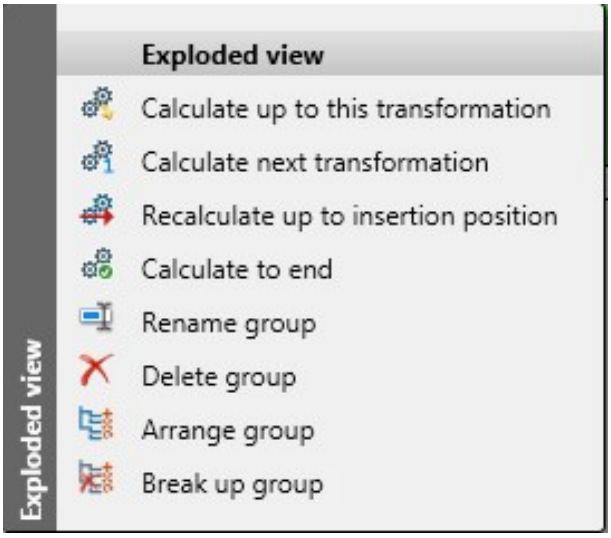
Combine transformations into group

Use this function to combine several transformations in succession into a group. You can also move other transformations into or out of this group by Drag & Drop.

A set can in turn contain other sub-groups.



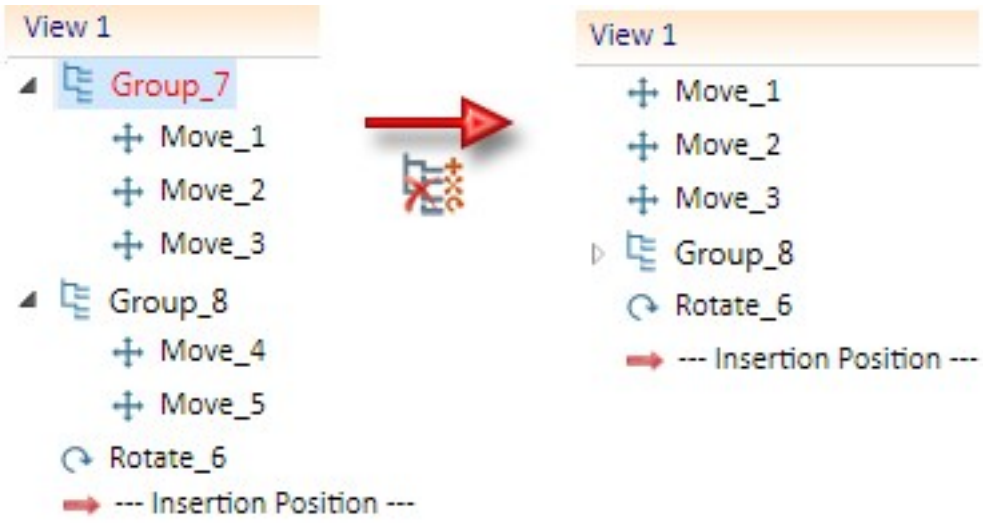
Right-clicking a group opens a context menu with further functions:



You can then change the name of a group, delete, arrange or break up a group again.

Break up group

Use this function to break up a marked group. Sub-groups contained in that group will be preserved.



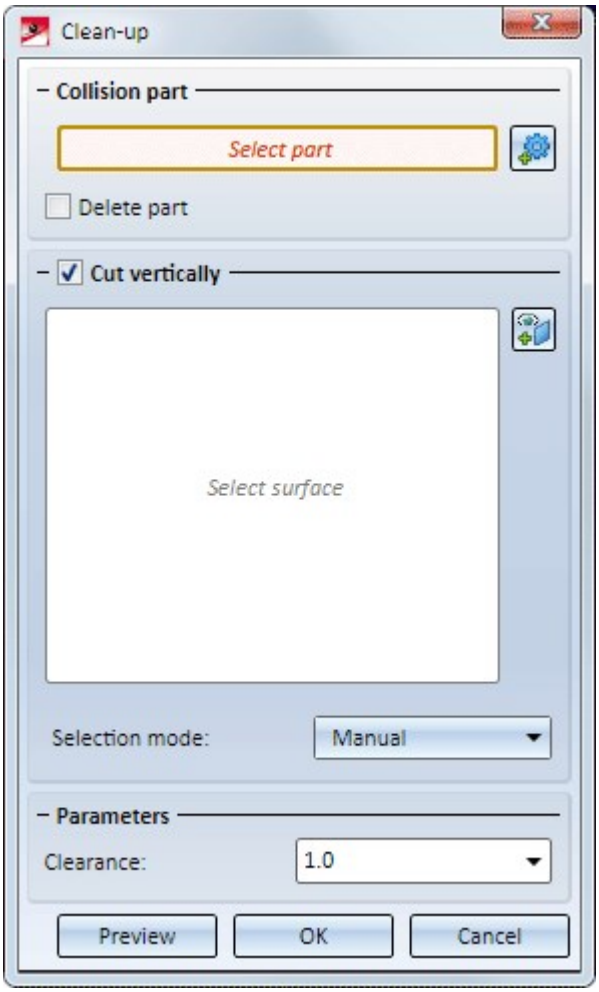
Important:

If you delete a complete set of transformations, all transformations contained therein will also be deleted.

Process / Model parts

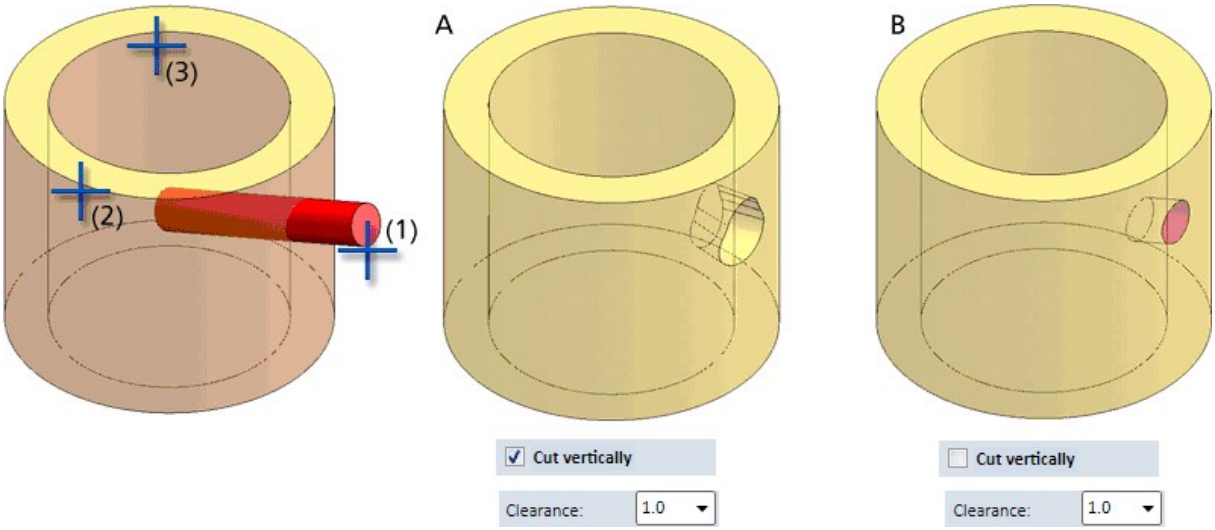
Clean up

The **Clean up**  function has been enhanced. It now offers the new **Cut vertically** checkbox.



- If the checkbox is activated, you need to determine the surfaces in relation to which the hole resulting from the intersection is to be placed in a vertical position. This means that the function behaves as before.
- If the checkbox is deactivated, the selection of surfaces is also deactivated. In this case the result of the function corresponds to a subtraction with clearance.

In the image below shows the collision part (1) is a cylinder intersecting with a pipe. The outer (2) and the inner surfaces (3) of the pipe have been selected here. A shows the result with clearance, B, in comparison, the result of a subtraction. A shows the result of a vertical cutting and clearance. The outer (2) and the inner surfaces (3) of the pipe have been selected here. B shows the result without vertical cutting.



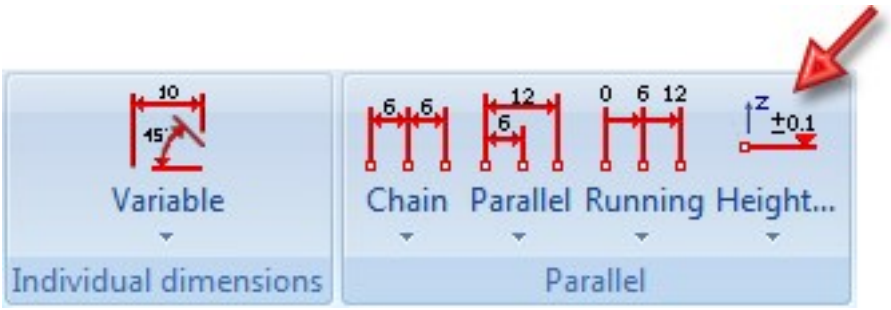
Part properties - Weight calculation

The weight calculation option in Configuration Management under **Modelling > Part properties**controls when the weight of parts (based on geometry) are to be recalculated. If the weight of a part changes, then these changes must be transmitted upwards in the parts structure, e.g. recalculation of the weight of superordinate assemblies, etc. Up to now, this transfer potentially occurred more often than the update of weights based on the geometry (manual entry of a weight, change of the BOM relevance, varying geometries etc). As of HiCAD 2019 SP2, this transmission of weights is also controlled by the **Weight calculation** option in configuration management, which can significantly improve performance in large drawings.

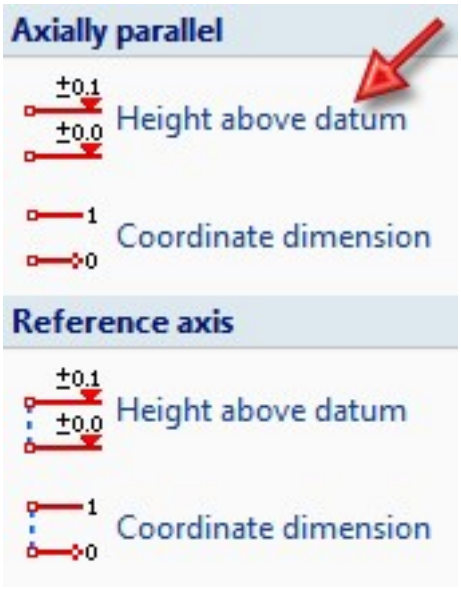
Dimensioning

Heights above datum

The **Create new height above datum, parallel to Z-axis** function can now be found at **3-D Dimensioning+Text > Parallel > Height...**.



The **Create new height above datum** function has been moved into the sub-menu.

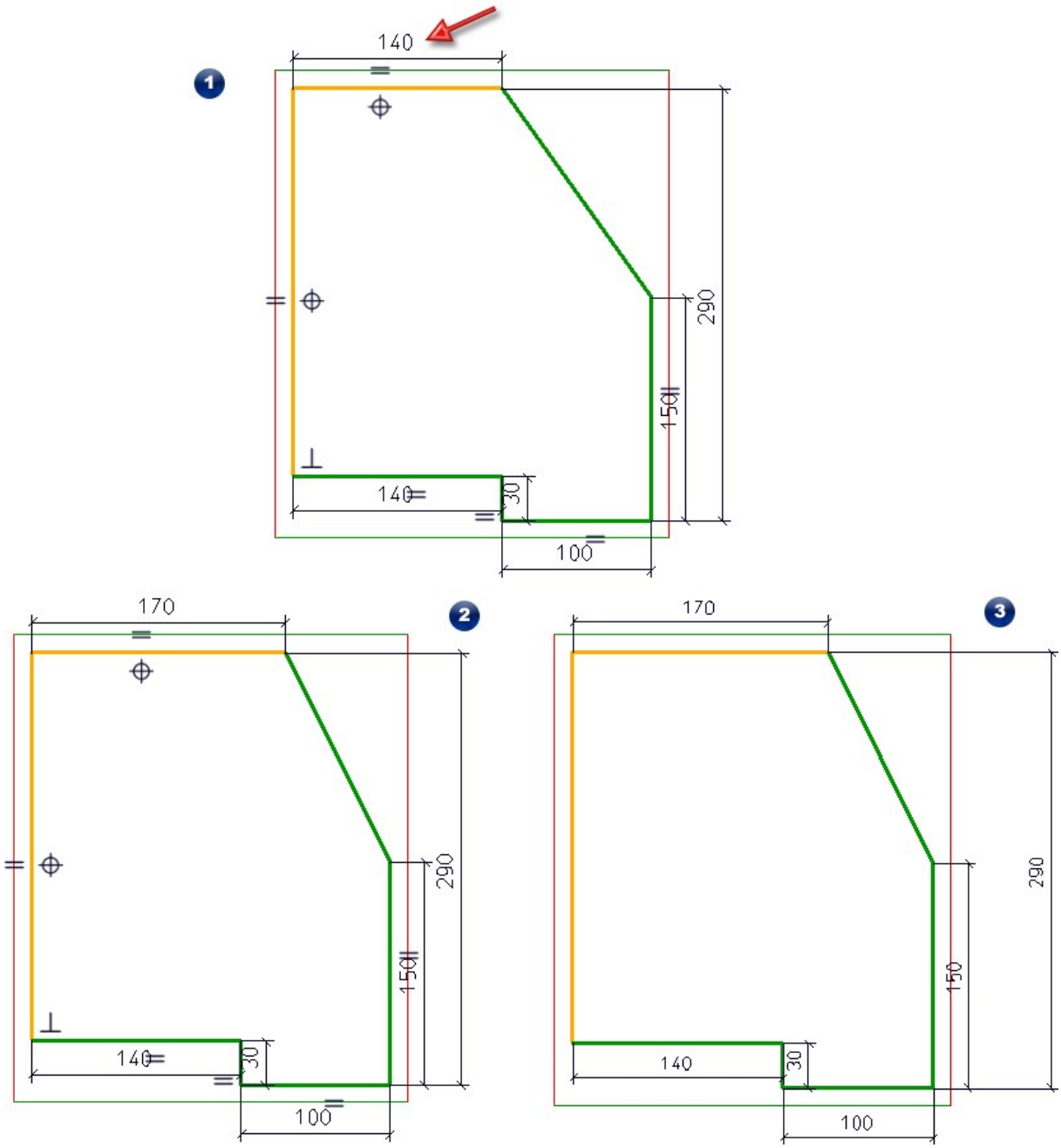


Placing of HCM dimensions in case of geometry changes

In older versions, HCM dimensions in sketches were placed with regard to their first base point. In case of geometry changes, the new dimension was placed in such a way that the length of the projection line of the first base point was identical to the length before the change, which frequently lead to unsatisfactory results. From SP2 onwards, the placement logic is different. The length of the shortest projection line after the change is now identical to the length of the shortest projection line before the change. In the process, cases may occur where the shortest projection line could be applied to different base points.

Example:

In the original sketch (1) the marked HCM dimension is changed from 140 to 170. (2) shows the result in HiCAD 2019 SP2, (3) shows the result in HiCAD 2018



Annotations

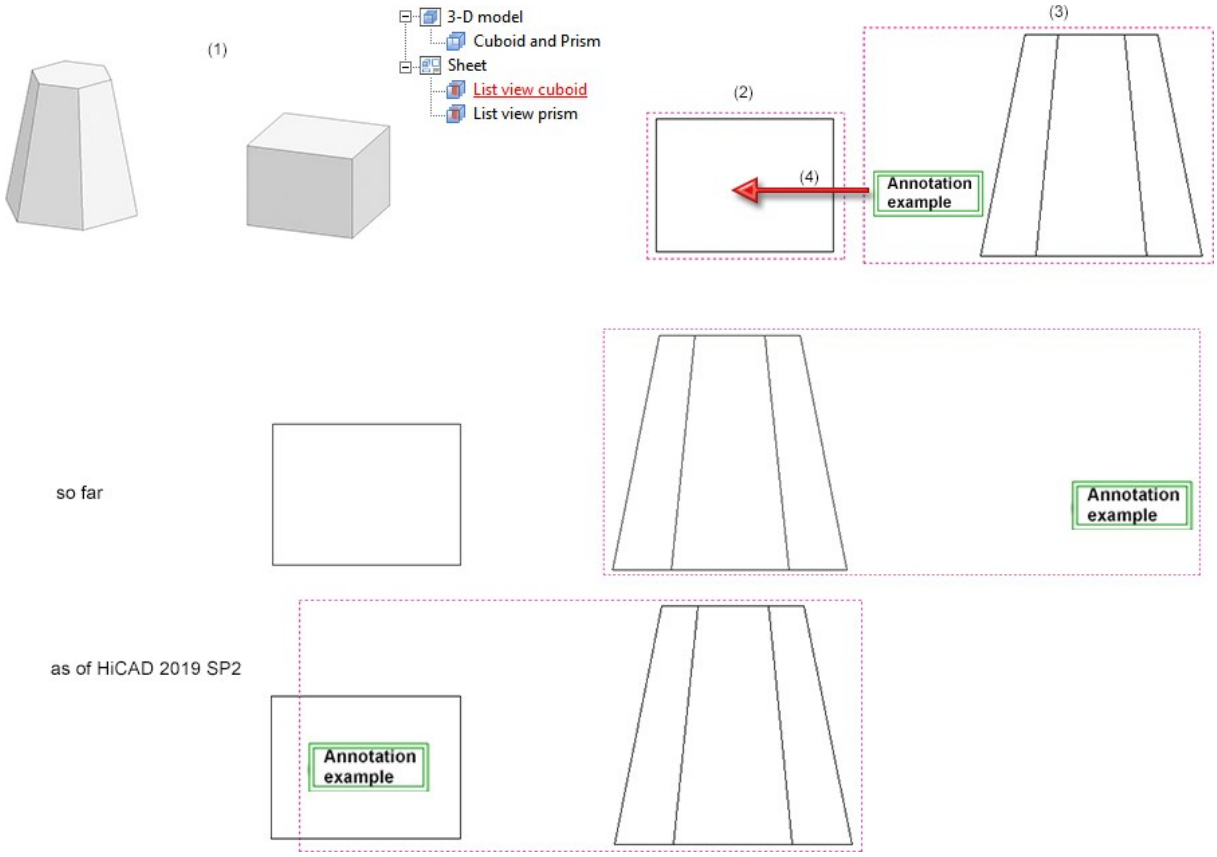
Move / Clone annotations

In previous versions, it could happen that annotations were displaced in a chaotic manner when you moved/cloned them in model drawings with several views, and accidentally selected a target point in a neighbouring view. This behaviour produced unwanted results.

This behaviour has now been changed: The target point now always refers to the original view.

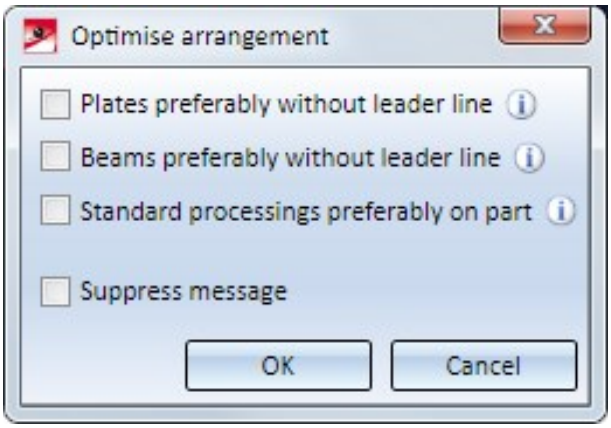
Example

The image below shows a 3-D model (1) containing a prism and a cuboid. Two list views have been created in the Sheet area- one list view with the cuboid (2) and one list view with the prism (3). The list view for the prism contains an annotation. If it is moved to the left (4), the behaviour is now different from that of previous HiCAD versions.



Improved placing of annotations for standard processings

In previous versions, leader lines frequently ran straight through processings (e.g. bore patterns) when using the automatic placing of annotations for standard processings. This effect has been corrected in HiCAD 2019 SP2; now, leader lines of processings will always end at the nearest standard processing. Furthermore, leader lines can now be placed either on or outside of parts. For this purpose the dialogue window of the **Optimise arrangement** function (**3-D Dimensioning+Text > Text > Part annotation with free text + leader line**) now contains the new **Standard processings preferably on part** checkbox.

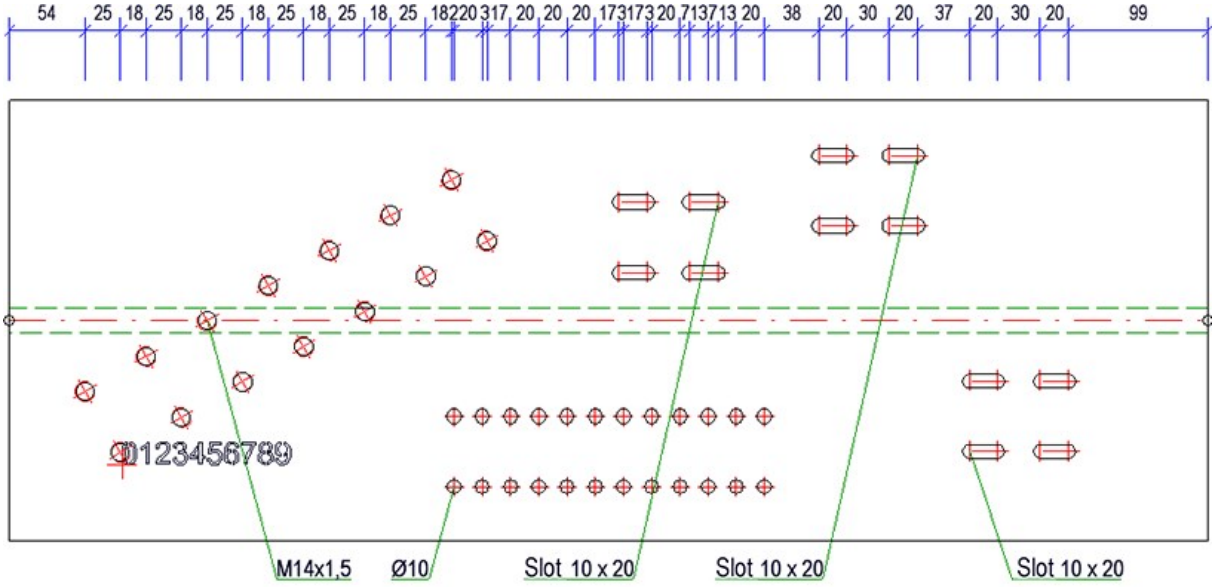


If the checkbox is active, the annotations of standard processings are placed directly next to the processing, i.e. on the part, if there is sufficient space. If there is not enough space, the annotation will be placed a little farther.

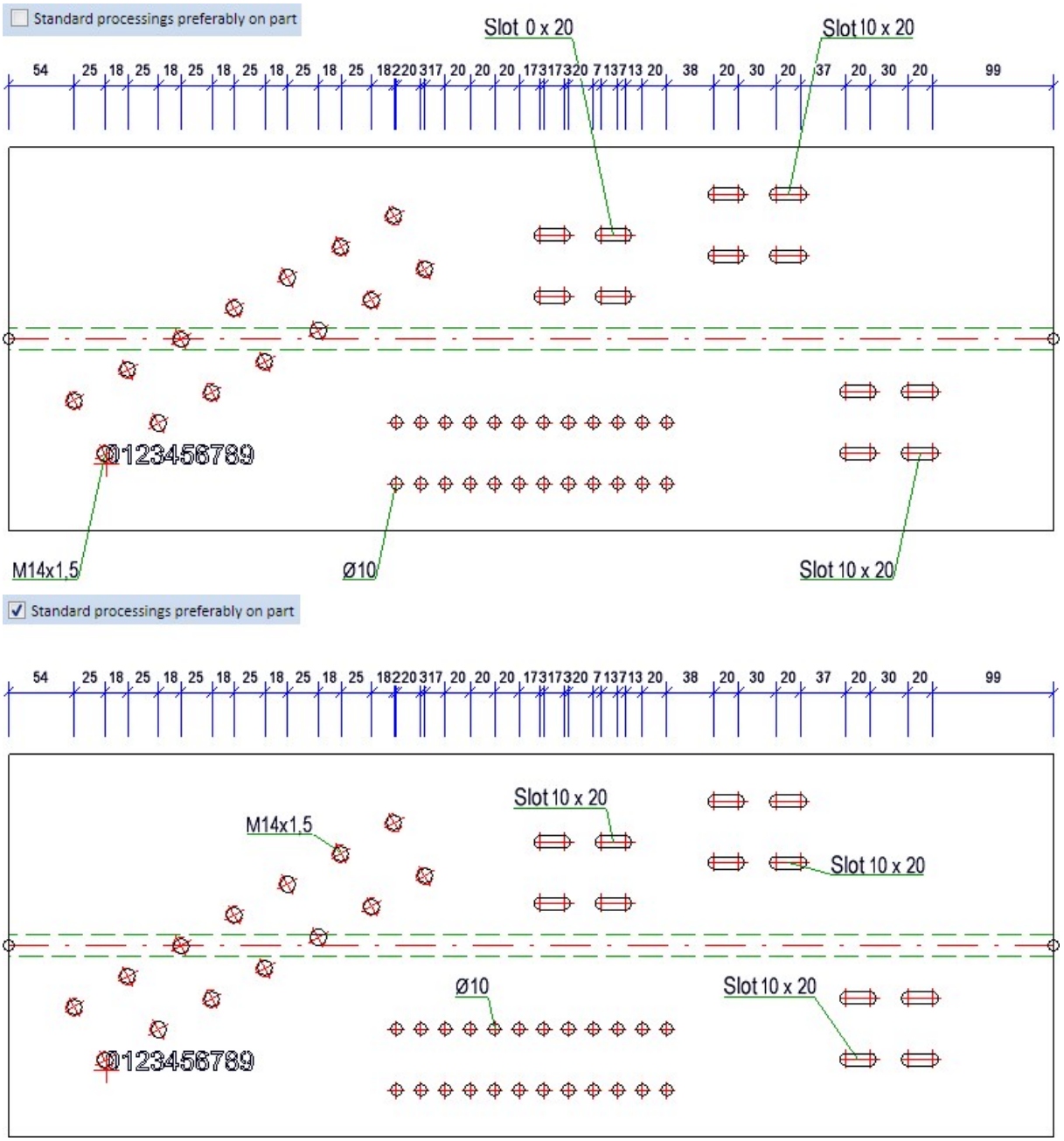
If the checkbox is deactivated, the annotations will be placed outside the part, as in previous versions. This is the default setting.

Example

1. Arrangement of annotations in HiCAD 2019 SP1



2. Arrangement of annotations in HiCAD 2019 SP2



Please note:

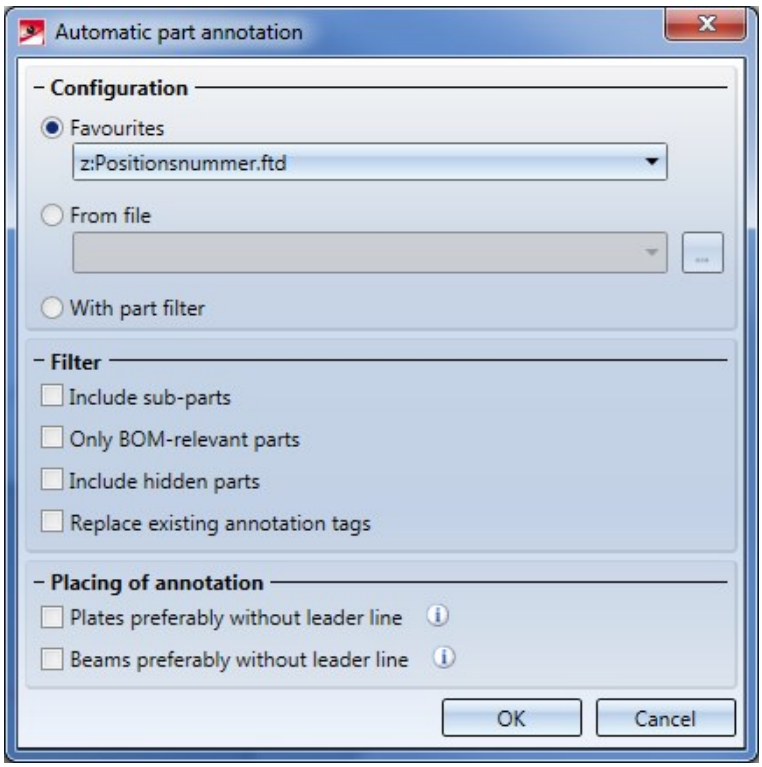
The behaviour of the arrangement of annotations can be set in the same way for automatically derived workshop and mounting drawings. This can be set in the Configuration Editor at

- **Automatic drawing derivation > Production drawing > Annotations > Placing of annotation** and
- **Automatic drawing derivation > Mounting drawing > Annotations > Placing of annotation**, respectively,

by setting the parameter **Standard processing on part** accordingly.

Automatic part annotation - Handling of hidden parts

The **Automatic part annotation** function has been modified to specify how hidden parts are to be treated in the active view. This is required, for example, in Steel Engineering, for halls or structures in particular. For this purpose, the **Include hidden parts** checkbox has been added in the dialogue window.

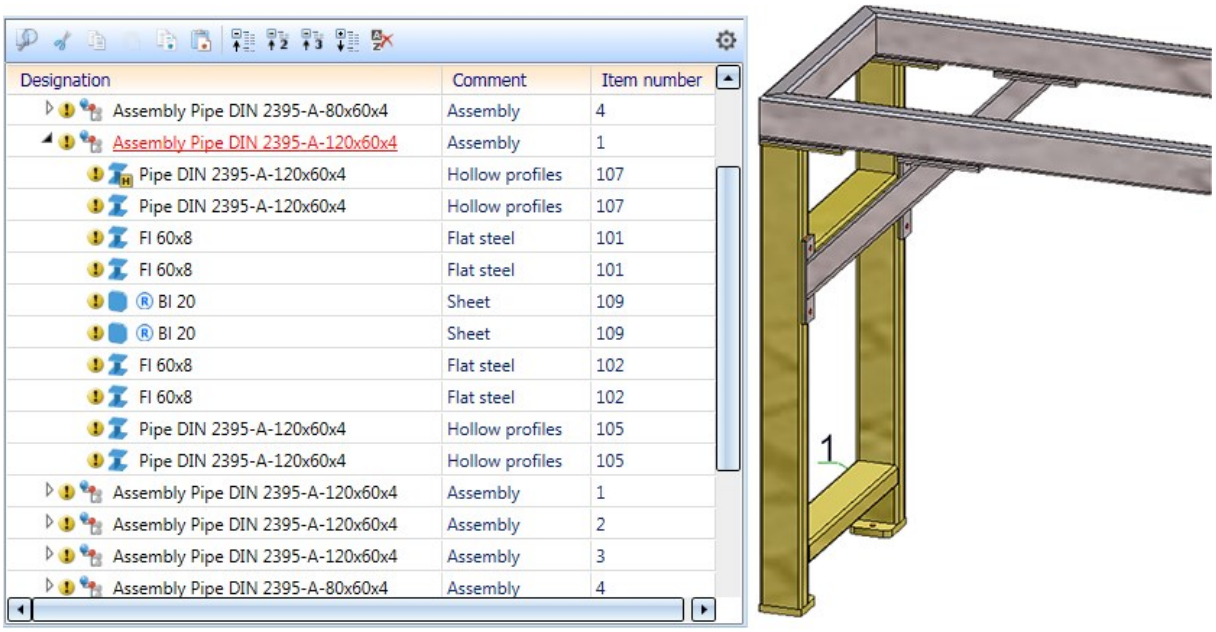


The **Include hidden parts** checkbox determines whether hidden parts are to be considered or not. Parts are considered as "hidden" if they are completely invisible in the active view.

- Representation types Glass model and HiddenLine dashed:
  No hidden parts occur in these views
- Representation types Shaded and HiddenLine:
  Hidden parts may occur here.

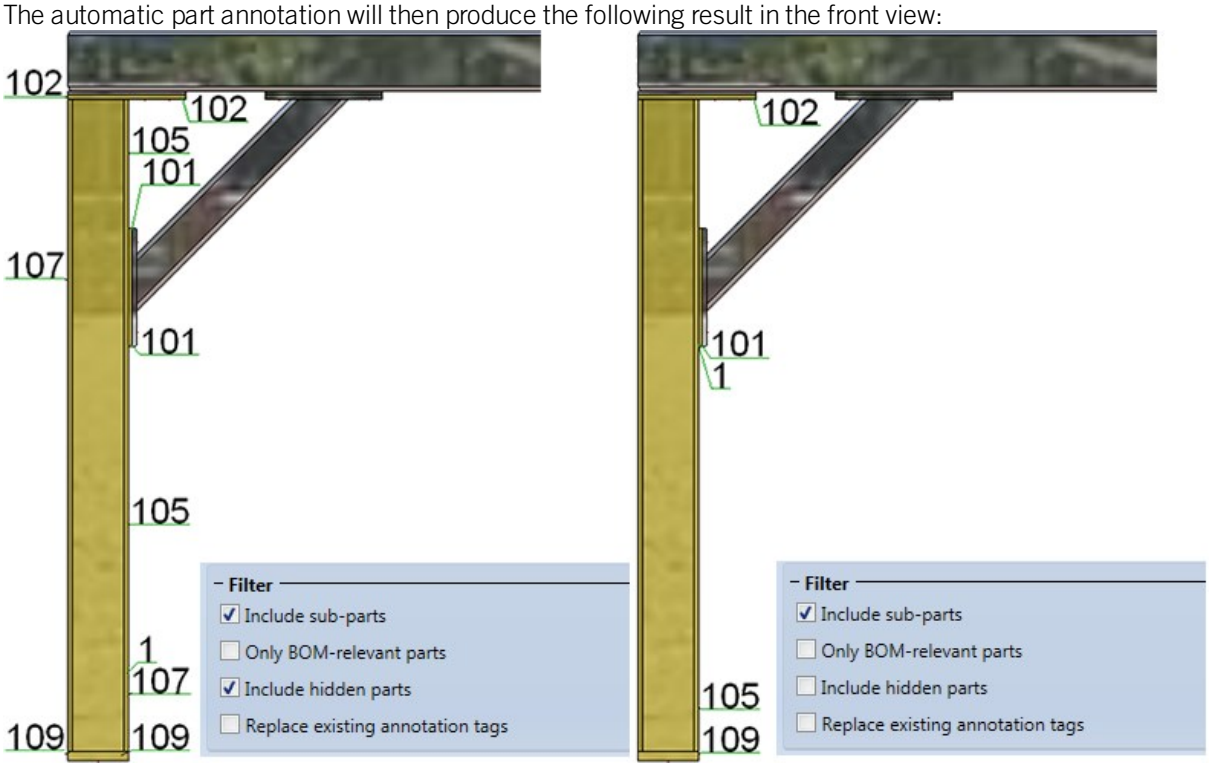
An example:

The image below show the detail of a support structure. Let's take a look at the assembly with the item number 1.



Designation	Comment	Item number
Assembly Pipe DIN 2395-A-80x60x4	Assembly	4
Assembly Pipe DIN 2395-A-120x60x4	Assembly	1
Pipe DIN 2395-A-120x60x4	Hollow profiles	107
Pipe DIN 2395-A-120x60x4	Hollow profiles	107
Fl 60x8	Flat steel	101
Fl 60x8	Flat steel	101
Bl 20	Sheet	109
Bl 20	Sheet	109
Fl 60x8	Flat steel	102
Fl 60x8	Flat steel	102
Pipe DIN 2395-A-120x60x4	Hollow profiles	105
Pipe DIN 2395-A-120x60x4	Hollow profiles	105
Assembly Pipe DIN 2395-A-120x60x4	Assembly	1
Assembly Pipe DIN 2395-A-120x60x4	Assembly	2
Assembly Pipe DIN 2395-A-120x60x4	Assembly	3
Assembly Pipe DIN 2395-A-80x60x4	Assembly	4

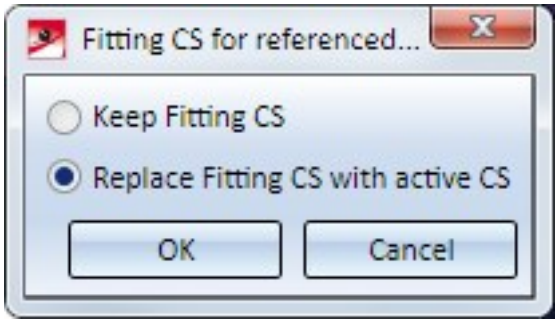
The automatic part annotation will then produce the following result in the front view:



Referencing

Fitting CS

When referencing 3-D parts, a Fitting CS, which will also be entered into the Feature log, will be automatically assigned to the parts. This applied to both the internal and the external referencing. In previous versions, fitting coordinate systems that existed during referencing were deleted. This behaviour has now been changed. If a Fitting CS already exists during referencing, the following message appears:



By activating of the desired option and confirming with OK you can either

- keep the existing Fitting CS, or
- replace the existing Fitting CS with the active CS.

Break up referencing of parts/assemblies

The breaking up of parts and assemblies has been speeded up. In tests, performance increases by factors between 2 and 4 could be achieved.

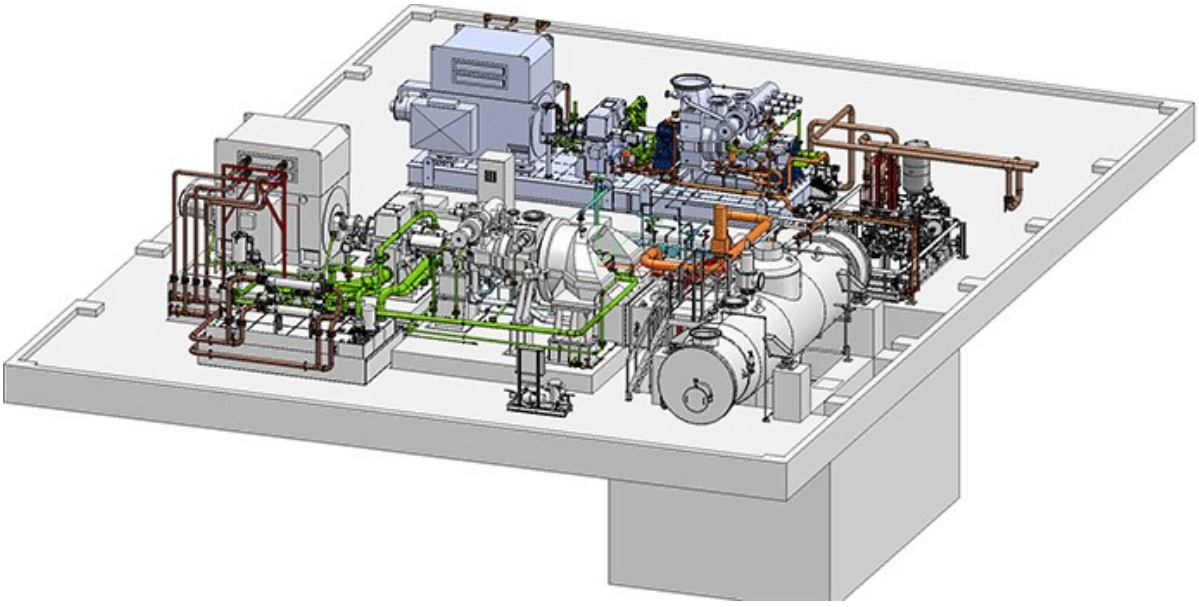
Service Pack 1 2019 (V 2401)

Discontinued functions
The functions of the **Assembling simulation** toolbar will be no longer available with the release of HiCAD 2020. Simulations created with these functions can still be played, but no new assembling simulations can be created any more.
From HiCAD 2020 onwards, the convenient simulation tool that was introduced with HiCAD 2014 can be used for motion simulations and physical simulations.

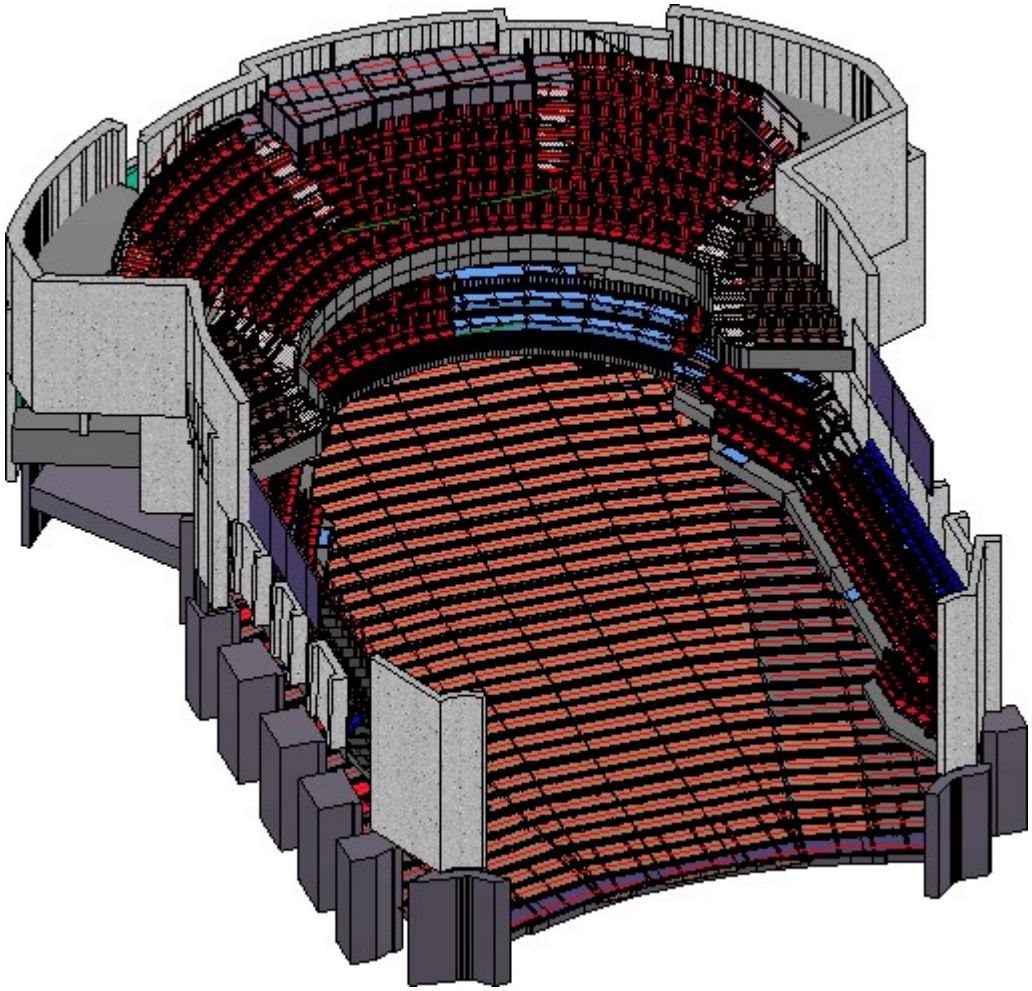
Views

Performance for rotations

In HiCAD 2018 SP2 the rotating of shaded views could be speeded up significantly - especially for shaded views with edges. In HiCAD 2019 SP1 the performance could be further increased. In a test we took a look at different model drawings with 20.000 parts (some of which were highly complex, imported parts), 60.000 parts and 90.000 parts. Examined were, for instance, the following model drawings:



20.000 with large imported parts (Image: TGM Kanis Turbinen GmbH, Nuremberg, Germany)



60.000 parts(Image: CAD Planung Arnold Matei, Mannheim, Germany)

Compared with HiCAD 2019 (Major Release) a significant increase of the frame rate (the number of images generated during rotating) could be achieved for the model drawings that we tested:

- Shaded with edges: Increased rotation speed by a factor of 4, and
- Shaded without edges: Increased rotation speed by a factor of 3.

Performance for sectional views, detail views and cut-outs

Thanks to the optimization of internal routines the generation of sectional views, detail views and cut-outs could be speeded up further.

Dynamic rotating with the mouse

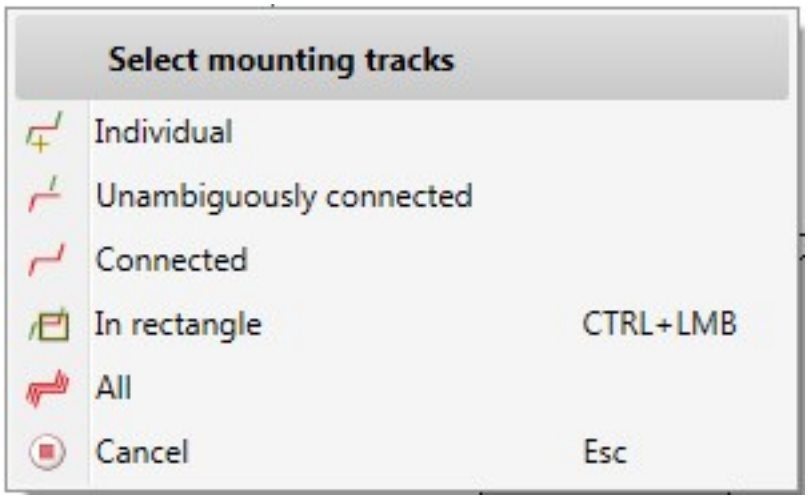
When performing a dynamic rotation with the mouse without a defined rotation point (MMB/mouse wheel), such a rotation point will be determined automatically. The position of this point depends on the part on which the cursor currently points, i.e. which part is currently highlighted in a different colour.

If the cursor points on a solid, the rotation will take place about a point on this solid. The rotation point will be determined automatically, by projecting the current cursor position perpendicular to the screen onto the next found surface. In previous versions this did not work for sketches.

From HiCAD 2019 SP1 onwards, a "reasonable" rotation point will be determined for sketches, too; this point can also be located on the frame of the sketch plane. The same principle applies to processing planes, i.e. you can also rotate about a point on the frame of the processing plane.

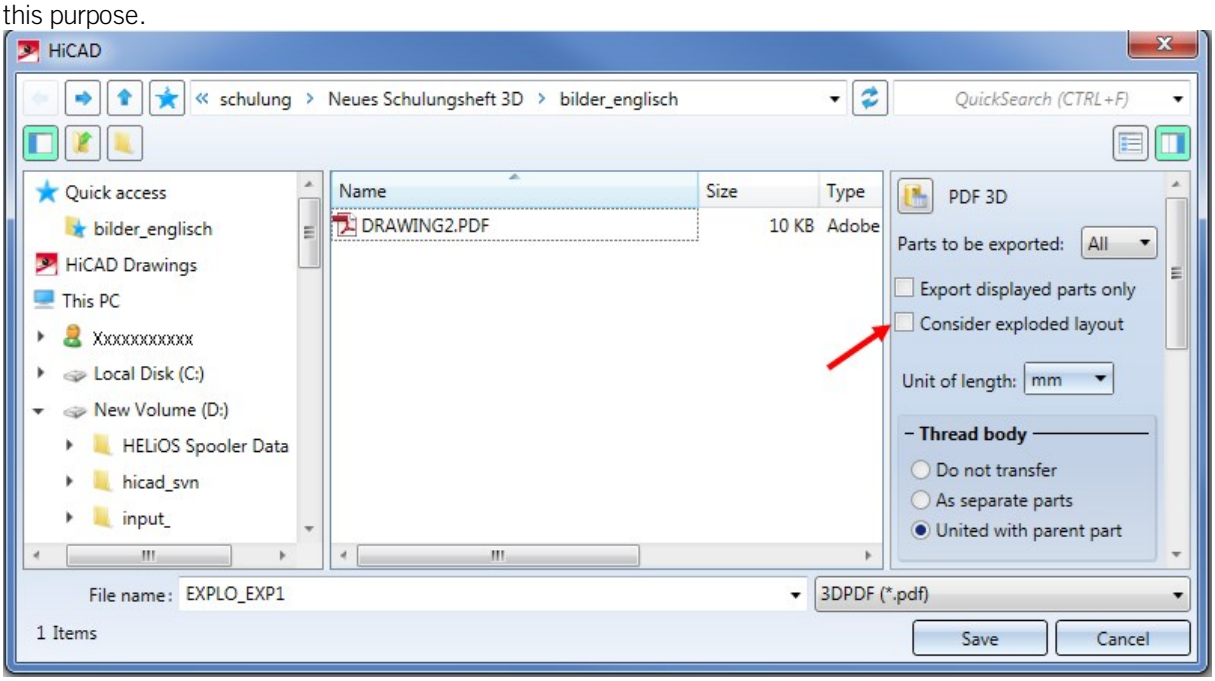
Exploded views

- When changing or deleting mounting tracks of exploded views you can now right-click to open a context menu with further selection functions:



- Please also note that exploded views can now be considered as
  - 3D-PDF (*.PDF)
  - Universal 3D (*.U3D) and
  - Product Representation Compact (*.PRC)

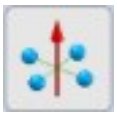
  files for exports. The Configuration Editor and the export dialogue windows have been expanded accordingly for this purpose.



  In the export file the parts will be displayed in the exploded layout. The mounting tracks, too, will be exported. While their colour will be exported from the model drawing, line type and font will not be considered.

- The name of the exploded view is now shown in the docking window. The name can be changed with the Views > Properties > Rename  function.

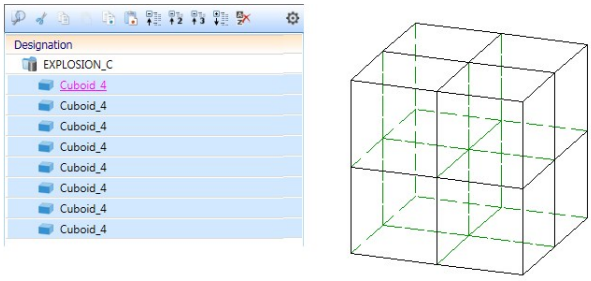
- Concentricity transformations can now be performed not just about an axis, but also about a point. For this purpose, two buttons have been added to the dialogue window:

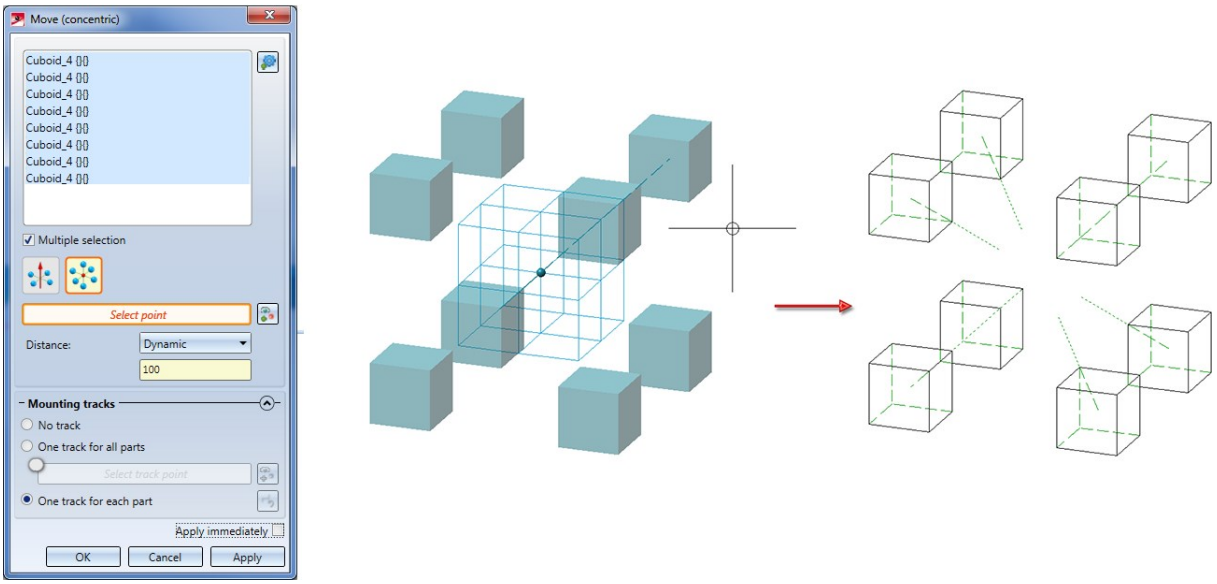
   **Concentric, about axis**

  Here the transformation takes place by selection of an axis. The moved parts will then be located on a circle the centre point of which lies on the selected axis and runs through the centroids of the selected parts. This means that the parts move within one circle plane, i.e. the transformation is two-dimensional.

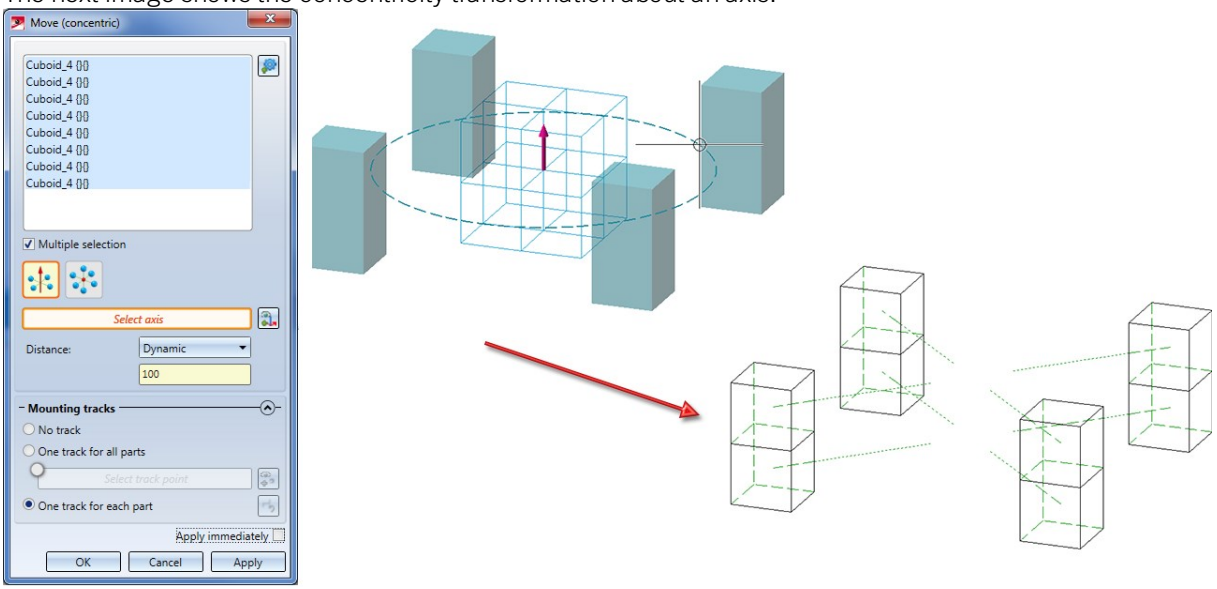
   **Concentric, about point**

  If you choose this option, the parts will move on a straight line running through the selected point and the centroid of the corresponding part, i.e. the transformation is three-dimensional.
  The following example illustrates the difference between the transformation types Concentric, about axis and Concentric, about point. The image below shows 8 neighbouring cuboids and one concentricity transformation about a point:





The next image shows the concentricity transformation about an axis:



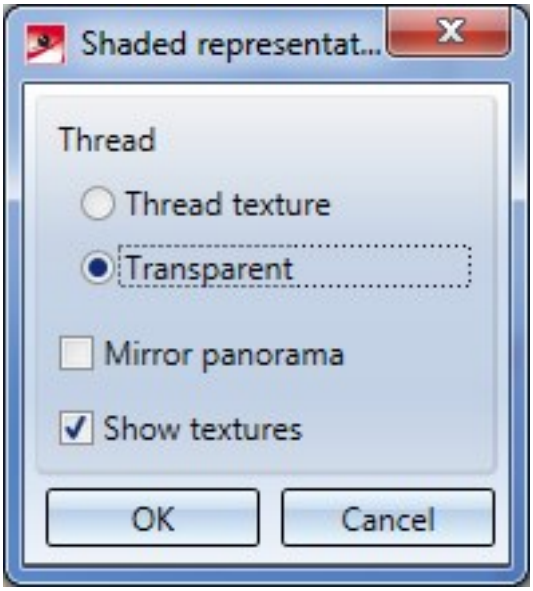
Switch visibility of textures on/off

With the functions

 **Shaded representation, active view** and

 **Shaded representation, all views**

the visibility of textures in the active views or in all views of a model drawing can be switched on or off. The menu of these functions has been expanded accordingly for this purpose.



Switching off the visibility of textures can be helpful during the processing of large model drawings with many textures.

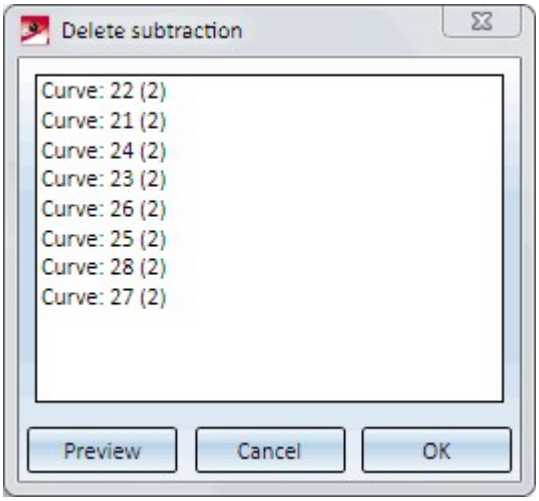
Model and process parts

Lettering

The **Lettering**  function dialogue window now always shows the last used settings.

Delete subtraction

The dialogue window of the **Delete subtraction**  function has been slightly modified.



Click on **Preview** to check what the result of the function will look like. The highlighted bores/subtractions will be removed when you click on **OK**.

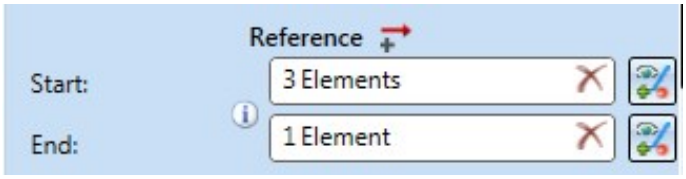
Parametric cloning

If you choose the **Clone parts, parametric**  function, the option **Linear pattern**, and, for distance determination,

- **Up to reference, with distance** or
- **Up to reference, with number**,

you can also choose an edge as end reference. In previous versions you were, after selecting an edge, always asked to identify a second edge (to determine a surface in this way).

Furthermore, you can now also determine several start and end references by clicking on the  symbol next to the corresponding input field and selecting the desired element. The number of the selected elements will be shown in the input field, e.g.:



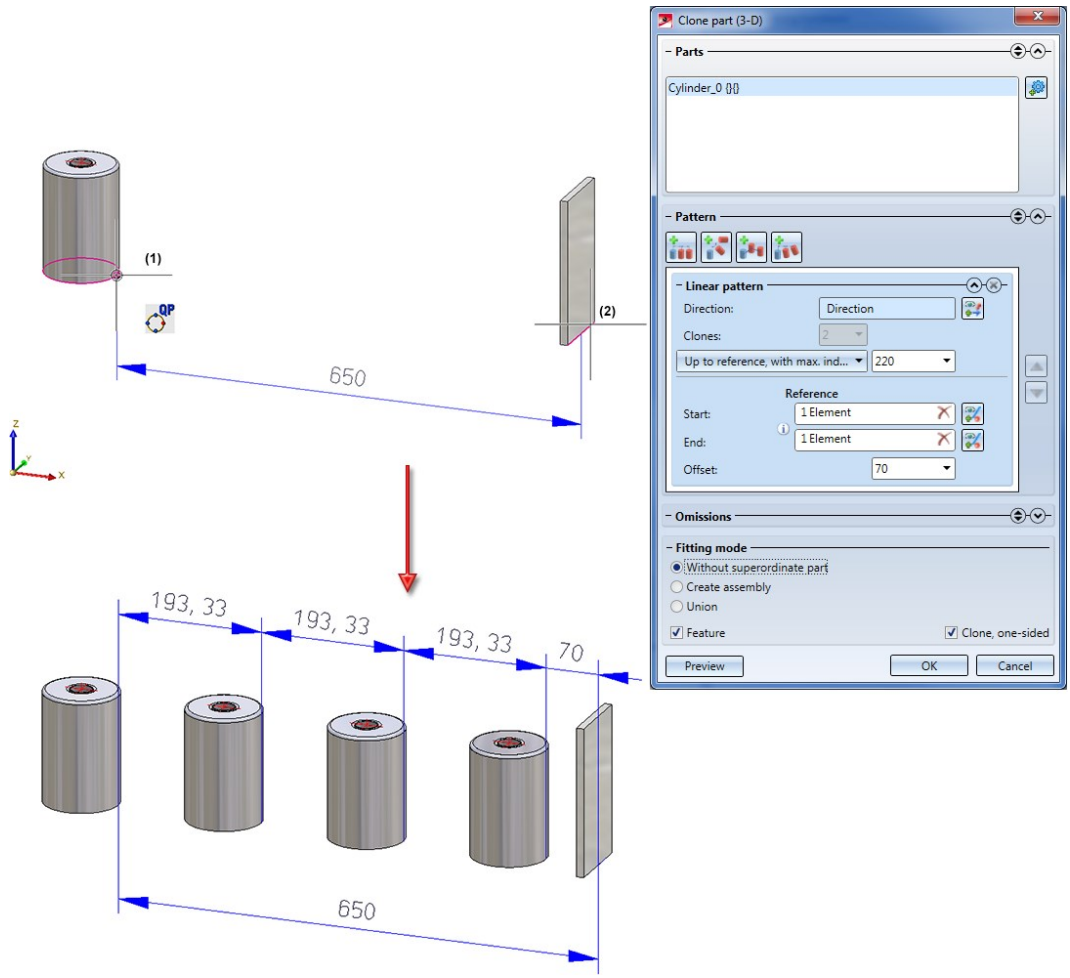
In this case, always the smallest distance resulting from the specified start and end references will be used. This distance can be shown by clicking on the symbol.

If you select an already selected element again, it will be removed from the list of references. To remove all start or end references, click on the **Delete** symbol in the respective input field.

New for linear patterns is the option **Up to reference, with max. indiv. distance**. The distance between the original and the last clone is determined here by the distance between the selected start reference and end reference. From this value and the specified maximum individual distance HiCAD will calculate the number of clones in such a way that their distance will not exceed the specified individual distance.

An example:

The cylinder shown below has been cloned with the settings shown in the dialogue window. The quad point (1) of the cylinder has been chosen as the start reference, the highlighted edge of the cuboid (2) has been chosen as the end reference, while the X-axis determines the direction:



External references on facets

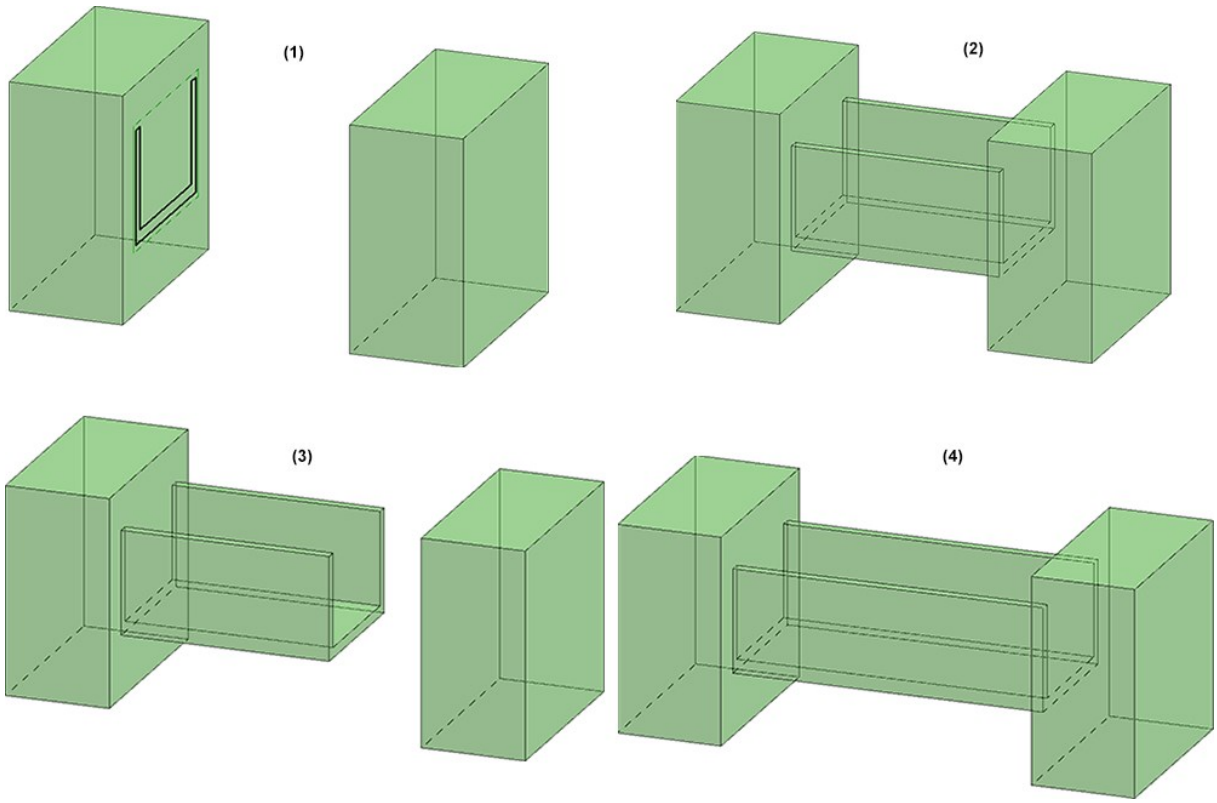
For the functions

- Add sweep, via translation,
- Subtract part, via translation or
- Trim to surface

the Feature setting **Do not use external references** was ignored in previous versions, i.e. external references were always considered. This behaviour has been changed now (see also Feature Technology - What's New?): If one of the above processings contains an external reference and is recalculated, while external references are deactivated, the processing will keep the previous result.

An example:

The image below shows two cuboids (1). On the left cuboid a sweep based on the shown sketch was created, up to the opposite surface of the right cuboid (2), i.e. there is an external reference here.

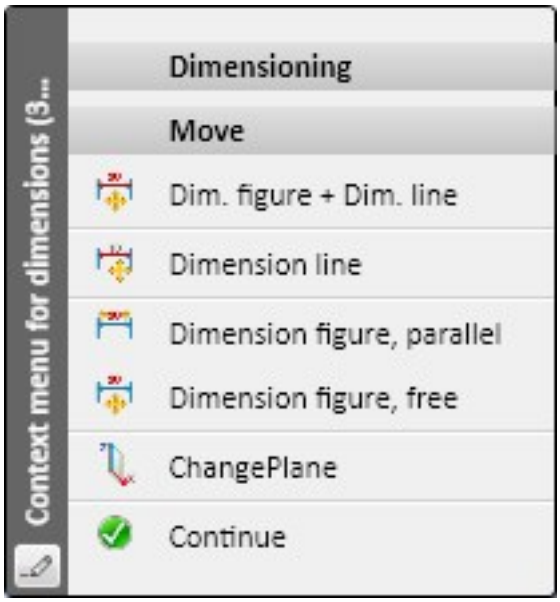


If you now move the right cuboid (3), and then perform a feature recalculation with the setting **Use external references**, the sweep will be adjusted accordingly (4). If the setting **Do not use external references** has been chosen, the original processing (3) well be preserved.

Dimensioning

Changed context menu

The **Move dimension** context menu has been slightly changed.



This context menu will be shown when you have selected a dimension you want to move and then right-click.

Changed dialogue texts for individual dimensions and structure dimensions

The user guidance texts in the info bar at the bottom left of the HiCAD screen have been improved and standardized in HiCAD 2019 SP1 and are now clearer and easier to understand.

Dimension / Action	Old text	New text *1, *2
General Selection of base points	Select 1st geometry element	Select base point (MMB = Cancel, RMB = Further options)
	Select 2nd geometry element	Select base point (MMB = Cancel, RMB = Further options)
	RMB = Change relocation mode	Drop dimension figure (MMB = Cancel, RMB = Further options)
General Expand structure dimension by selecting of further base points	Select next point	Select further base point (MMB = Cancel, RMB = Further options)
Individual dimension: Diameter / Radius Choice of object to be dimensioned	Edge or surface	Select circle, sphere, cylinder or torus (MMB = Cancel, RMB = Further options)
Structure dimension: Angular dimension Set base point	Select 1st geometry element	Select base point (MMB = Cancel, RMB = Further options)
Structure dimension: Arc dimension		
Select mid point	Mid point	Select mid point of arc (MMB = Cancel)
1st base point	Start point on arc	Select base point (MMB = Cancel, RMB = Further options)
2nd base point	End point on arc	Select base point (MMB = Cancel, RMB = Further options)

Variable dimensioning - Improvements

Since the Major Release 2019 the **Variable dimensioning** has been improved further with regard to the structure dimensions:

- Arbitrary base points can now be chosen,
- Besides points you can now choose edges for base point determination.

Sketches

Deriving of sketches

When deriving sketches from existing edges you can now, in certain situations, use reference edges that were created in sectional views as a result of the cut. This is possible with the following functions:

 Offset (via point and value)

 Double-sided offset with end caps

 Take over edges and

 Directed projection.

Process sketches via Feature log

In the **Process sketch** dialogue that can be opened via the Feature log of sketches used for the processing of parts (e.g. C-edge sweeps, extruded parts, revolved parts etc.) you can now also use the UNDO and REDO option, i.e. you can undo and restore executed actions in the **Process sketch** dialogue. For this to happen, dialogues in "deeper levels" must have been closed first.

An example: You fillet several corners of a sketch without leaving the **Fillet** function. After this, an UNDO cannot be performed for the individual fillets, but only after ending the **Fillet sketch** function.

After applying the processed sketch, an UNDO will completely take back the previously executed processing.

 For changes that are applied to the sketches in sectional views, detail views and cut-outs this is not possible.

Standard parts/processings

New standard parts

HiCAD 2019 SP1 offers many new fasteners, e.g. acc. to ASTM Standard (American Society for Testing and Materials). Please also see What's New in The Catalogue Editor?

Processing direction

New in the Standard Processings menu is the Processing direction  function. Use this function to indicate the processing direction and processing side of a part, e.g. a Sheet Metal flange, to determine the direction of its coating or texturing. This function was previously available on the Sheet Metal Ribbon tab.

Referencing

Updating of referenced parts

When you load a HiCAD model drawing, it can contain old indices of referenced parts. That is, the associated .KRA file of a referenced part has a higher version number than the part in the model drawing, or a higher index of the HELiOS document in the database may even exist, which requires the exchanging of this part against the part with the most recent index.

In this case, a dialogue listing the old parts and suggesting their updating will be displayed upon loading of the relevant model drawing. Upon confirming the updating as suggested, all parts in the model drawing will be updated to their most recent state. Depending on the size of the model drawing this process may take some time. In HiCAD 2019 SP1, however, the updating process could be speeded up significantly.

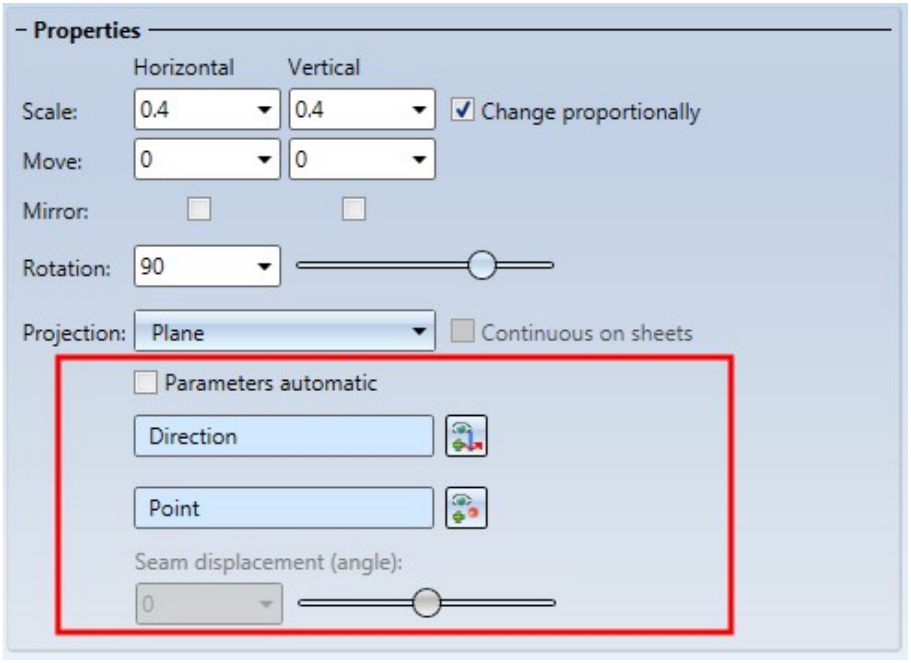
Textures

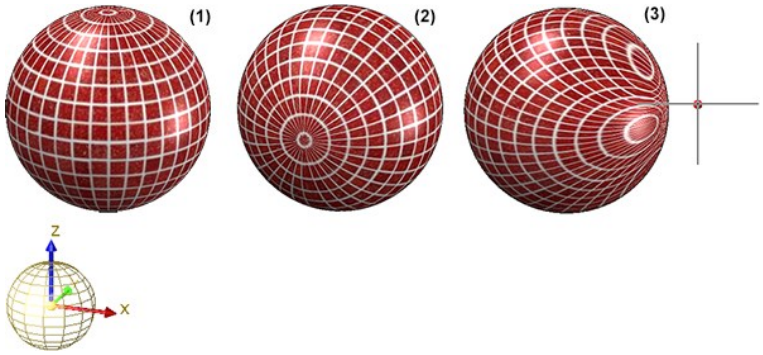
Setting the projection parameters

When assigning textures, you have now the option to set parameters for the selected projection method

- Cuboid,
- Plane,
- Cylinder or
- Sphere,

e.g. the cylinder axis or an angle for seam displacement.



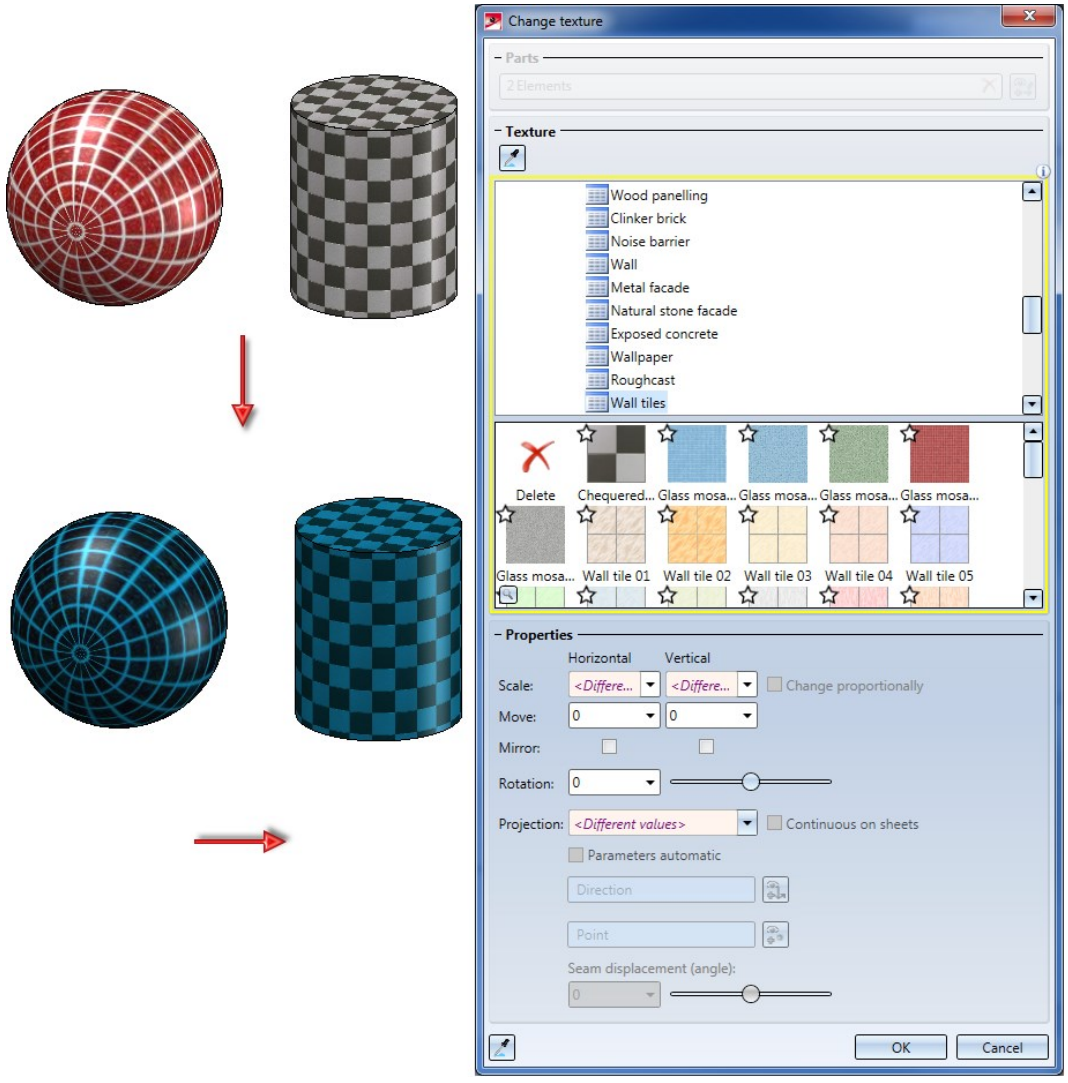


(1) Parameters,automatic, (2) Y-axis selected as axis, (3) Axis = Y-axis and additional point selected

Change texture - Multiple selection

The **Change texture** function also allows multiple part selection. The following rules apply:

- If the same settings apply for the selected and already textured elements, these will be shown in the dialogue window.
- If the settings differ, the fields containing different settings will either be displayed with a yellow border and marked with the **Different values** symbol, or the text **<Different values>** will appear in the field (depending on the field type).



New catalogue for user-defined textures

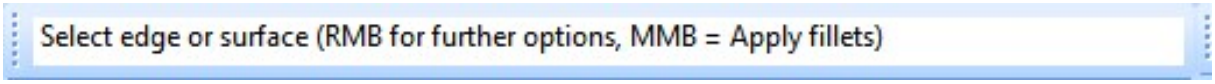
When defining your own textures with the **New texture** function, you can now save these to the catalogue **WERKSNORMEN (Factory standards) > User-defined textures**. There, a table called **Example textures** already exists. The texture image and the texture icon will be automatically saved to the HiCAD folder KATALOGE > TEXTUREN > IMAGE. Please note that new textures can only be added to the catalogue with this function. Their adding via the Catalogue Editor is not possible!

Major Release 2019 (V 2400)

Performance

Middle mouse button - Faster working process

For many functions, pressing the middle mouse button allows not only the exiting of the function (as long as data has been entered) but also enables a faster working process. For instance, chamfering or filleting 3-D solids or transforming sketch elements can be applied with the middle mouse button, after all necessary data has been entered. This means that switching to the dialogue window of the respective function and clicking on the **Apply** button will no longer be necessary. HiCAD points to the according functions in the dialogue text, for instance:



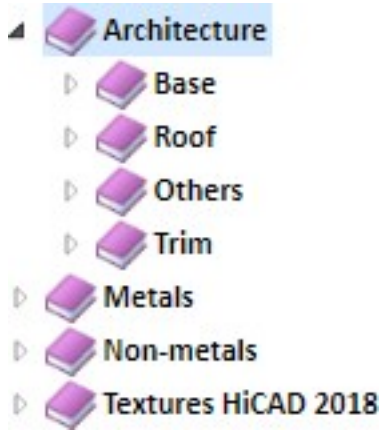
Currently, this function concerns the following 3-D functions:

- 3-D Standard > Process > Fillet
- 3-D Standard > Process > Chamfer
- 3-D-Standard > Process > Wall > Envelope
- 3-D-Standard > Process > Clone
- Sketch > Process > Fillet
- Sketch > Process > Chamfer
- Sketch > Transform > Move
- Sketch > Transform > Rotate
- Sketch > Transform > Mirror
- Sketch > Transform > Scal...
- Sketch > Transform > Move...
- Sketch > Clone > Move
- Sketch > Clone > Rotate
- Sketch > Clone > Mirror
- Sketch > Clone > Scal...
- Sketch > Process > Trim > By value
- Sketch > Process > Trim > To surface/line/point
- Sketch > Process > Trim > Number
- Sketch > Derive > Take...
- Sketch > Derive > Offset > Double-sided offset with end caps
- Sketch > Derive > Offset > Replace line

Part properties

New textures

HiCAD 2019 offers a multitude of new textures, e.g. for architecture, metals and non-metals. These are available in the HiCAD catalogue at **Textures and Colours > Textures**.



The previously available textures can be found at **Textures and Colours > Textures > Textures HiCAD 2018.**

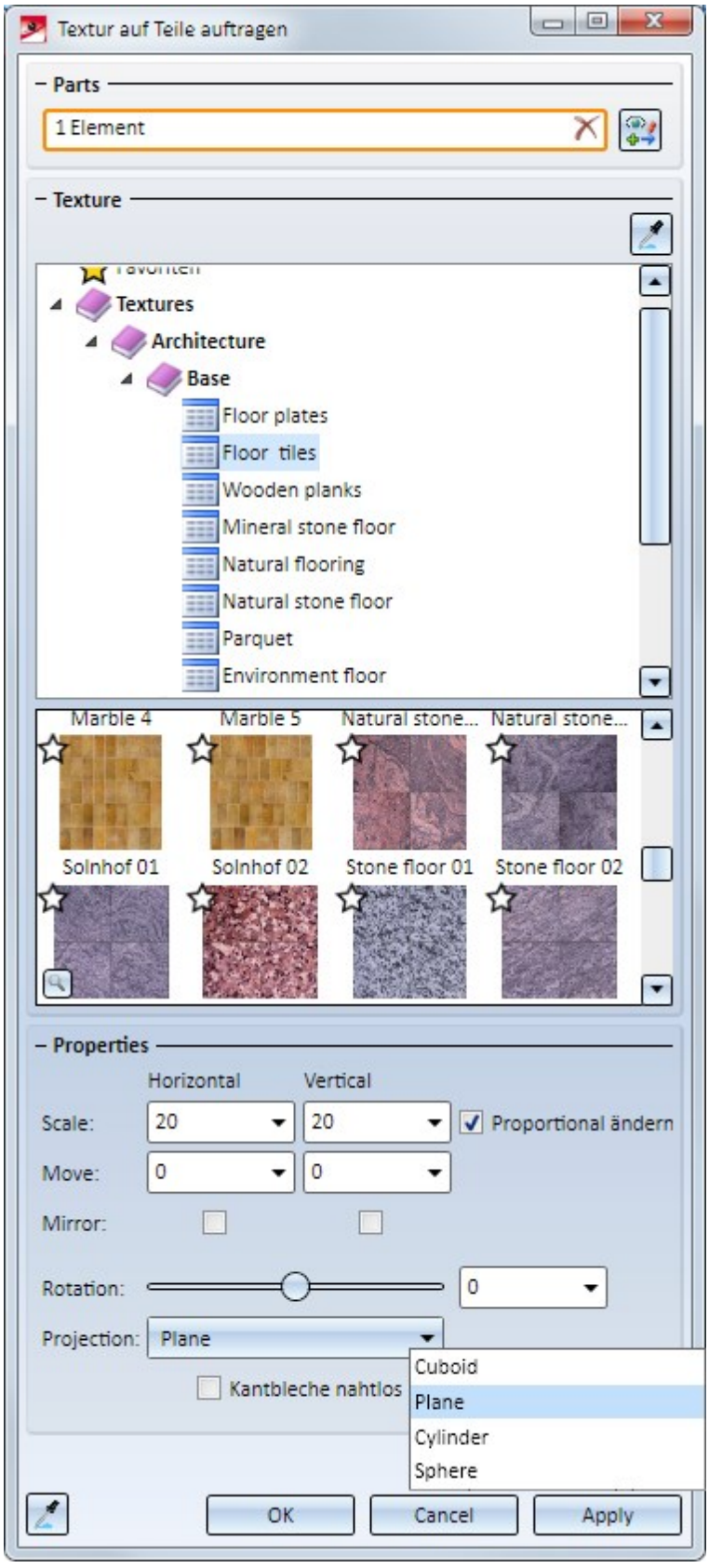
Also, new functions are available for the assigning, creating and changing of textures.

New dialogue for assigning of textures

As of HiCAD 2019, textures are no longer assigned via the **Visual effects** docking window. Instead, you can use the following functions:

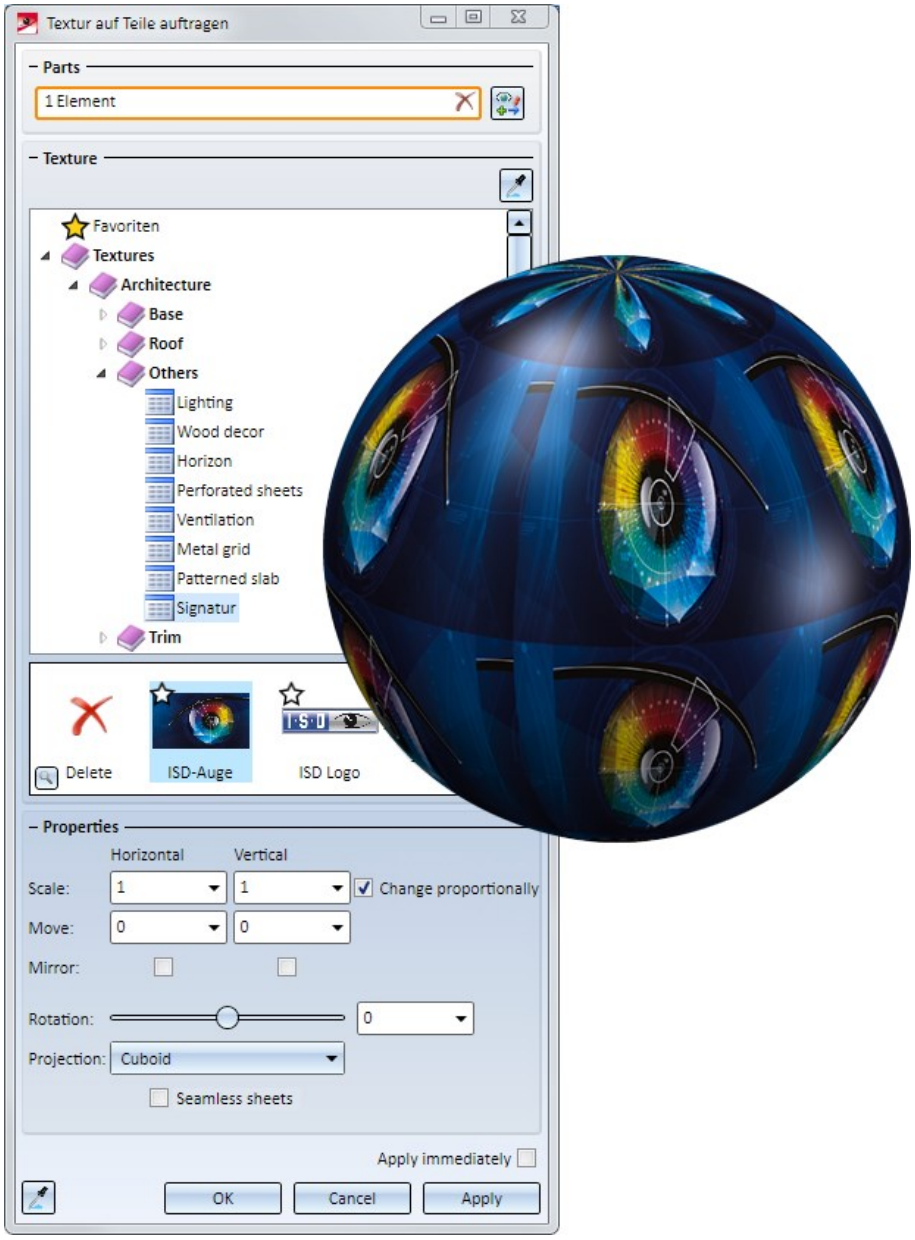
Texture - New (3-D part list) assigns a texture to the active part or the active part list	at **3-D Standard > Tools > Attr. > ...** or in the context menu of a part list at **Properties > ...**.
Texture - New (3-D part) assigns a texture to the active part	in the context menu for 3-D parts at **Properties > ...**
Change texture, active part (3-D part) changes the texture of the active part	in the context menu for 3-D parts

New dialogue windows for assigning and changing of textures are available, e.g.:



New Texture Editor

For creating of new textures and changing of textures that already exist in the catalogue the **New texture** function is available at **Drawing > Others > Extras > ...** :



Delete material

At **3-D Standard > Tools > Attr. > ...** you will now find the **Delete material** function that allows you to delete the material assignment of the active part or the active part list. You can also find the function in the context menu at **Properties > ...**.

Views

Pre-setting of the representation in model views and sheet views

The representation that is used when you create new drawings for model views and sheet views can be pre-set. One distinguishes between

- manually created drawings, i.e. drawings created with the **Drawing > New** function, and
- derived drawings, e.g. drawings created with the **Derive** or **Mounting drawing** function.

Manually created drawings

The representation of views that are directly generated upon creation of a new drawing with the **Drawing > New** function, is pre-set as follows:

- **Model view**
  Via the **HiCAD Start Centre**
- **Sheet views**
  Via the Configuration Editor at **Drawing > Views** with the parameter **Representation of Sheet view**. The default setting is Hidden Line.

The pre-set representation will also apply to subsequently created views of the current model drawing - as long as you choose a different representation. After this, the new representation will be the new pre-setting for the current model drawing.

Derived drawings

The representation of the model view in external drawings created by means of drawing derivation can be specified in the Configuration Editor at **Automatic drawing derivation > Representation of model view**. The default setting here is **Shaded with edges**.

The representation for sheet view can be specified in the dialogue windows of the following functions:

Drawing > Itemisation / Detailing > Derive

Drawing > Itemisation / Detailing > Derive > Create

Drawing > Itemisation / Detailing > MntDrw

Drawing > Save/Reference > Part (Option **Save as part**)
In this case the parameters **Representation of Sheet view**and**Representation of model view** will be considered.

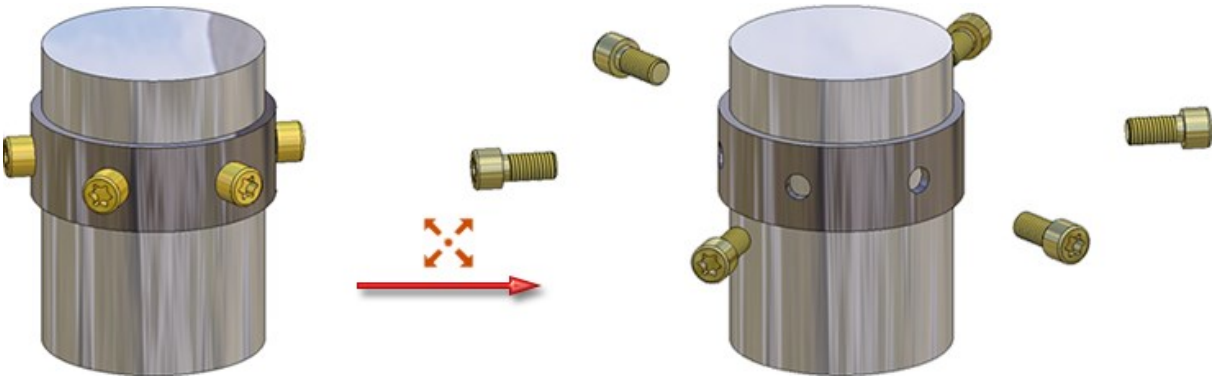
Collinear sheet edges

As of HiCAD 2019, collinear sheet edges are no longer displayed in HiddenLine mode. The corresponding checkbox has been removed from the **Show/Hide elements** dialogue window and the Configuration Editor. When opening model drawings created with HiCAD 2018 or older versions the representation that was set when the drawing was saved applies. However, you have the option to update the handling of collinear sheet edges to the new mechanism with the new **Update sheet edges** function (Views > Properties > Elements > ...). After this, you can hide collinear-tangential sheet edges with the HiddenLine representation. Collinear bent edges are always shown, and collinear mergeable edges are hidden.

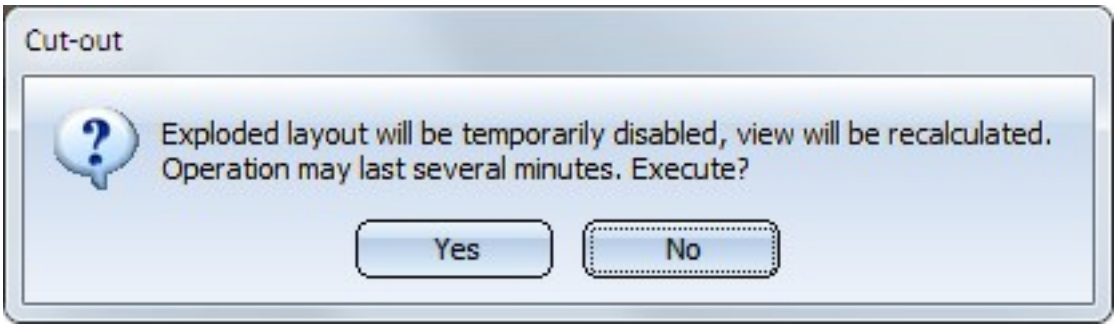
New options for exploded views

- Besides the Move and Rotate function, another transformation type for exploded views, Concentricity, is now available. Use this function to define concentric transformations in the Exploded view mode. This makes sense if you want to transform parts that are arranged concentrically around a common centre, e.g. bolts arranged on a polar or radial grid.

  In the case of parts that are arranged symmetrically around a common centre, the centroids of the parts are located on a circle. The Concentricity function moves the parts in such a way that their centroids will be located on a circle that is concentric to the aforementioned circle.



- From now on, sectional and detail views as well as cut-outs can be transformed into exploded views.
- Cut-outs can be created and edited in an exploded view. The explosion will be temporarily removed. In this case, an according notification will be displayed with the option to cancel the process.



- The functions of the user interface **Exploded view** and **Delete explosion** have been removed. As of now, these functions are available via **Views > Process > Expl...**

- The **End**  function from the tab **Exploded view** tab is not available any more. Use the according button from the drawing area while the Exploded view is active instead.
- The individual steps of the exploded view creation and the insertion position can be moved up and down in the**Exploded view** docking window via Drag & Drop.
- As soon as parts that belong to an exploded view are changed, the exploded view will be marked as "invalid". Like in sectional and detail views, such invalid views are marked with a red cross, like in sectional views and detail views. To update the exploded view, choose **Views > Edit > Update**.

  As long as the part selection is active, you can right-click in the drawing area to open the following context menu:

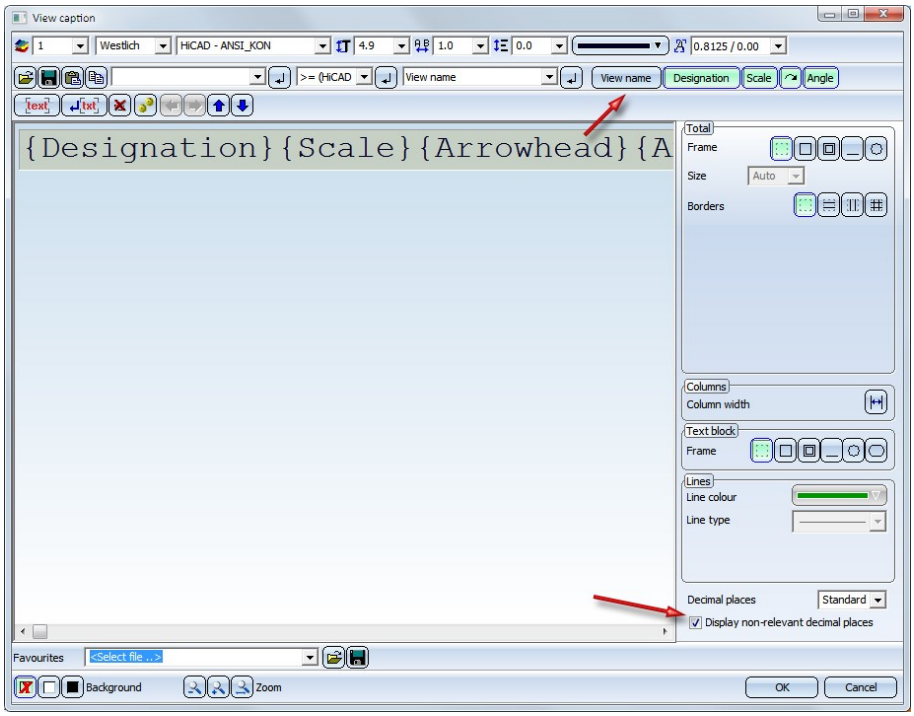
  

  With the function **Parts in rectangle** you can then select all parts within a drawn selection rectangle.

Captions in sectional views and detail views

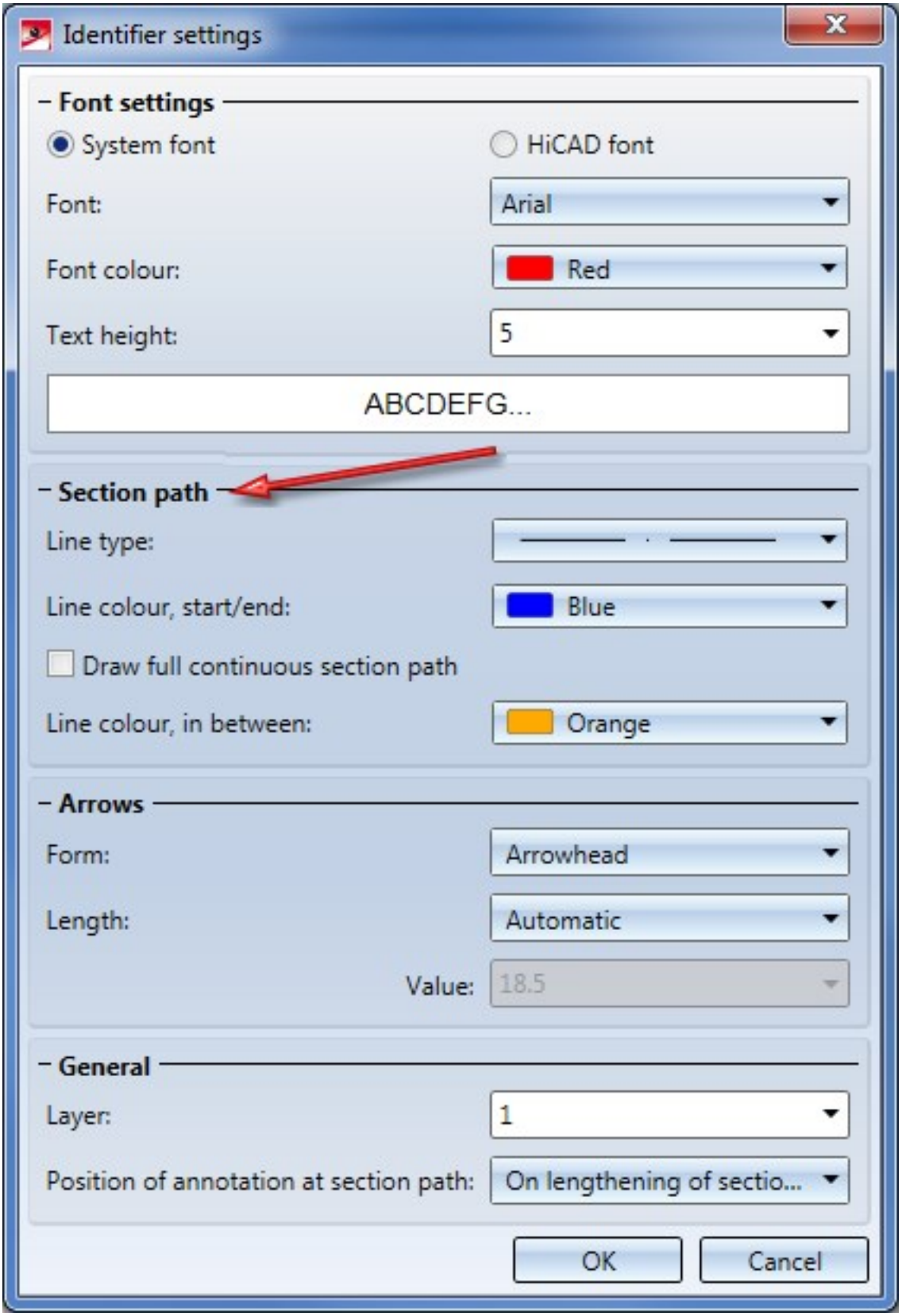
- The name of a view can now be used in captions for sectional views and detail views.
- You can now also specify the number of decimal places for the Angle in view captions, as with annotations. Furthermore, you can also show non-relevant decimal places, e.g. final zeroes, by activating the corresponding checkboxes. If you choose **Standard** for **Decimal places** (i.e. no number of decimal places has been specified in the FTD file), the setting from the Pos3DParNeu.dat file will be used for the number of decimal places (Default setting 0).
- The behaviour of the scale output in view captions has changed in HiCAD 2400. The scale will now also be shown if it is equal to the main scale.
  - For new view captions created with HiCAD 2400, or view captions to which the scale has been added subsequently, the scale will be output without a check for scale equality.
  - The behaviour of scale outputs in already existing view captions (created before V 2400) remains unchanged, i.e. the scale will only be output if it is not equal to the main scale.

  To apply the new behaviour to already existing captions, proceed as follows:
  - Edit the view caption and remove the scale.
  - Apply the changed caption with OK.
  - Edit the caption again and add the scale again.
  - Apply the changed caption with OK. The new scale will then show the new behaviour.

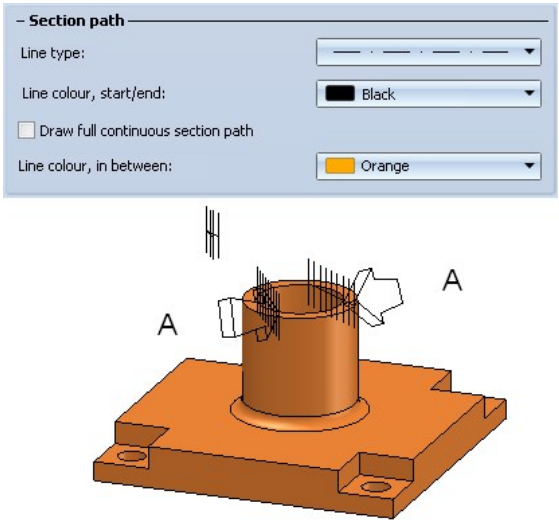


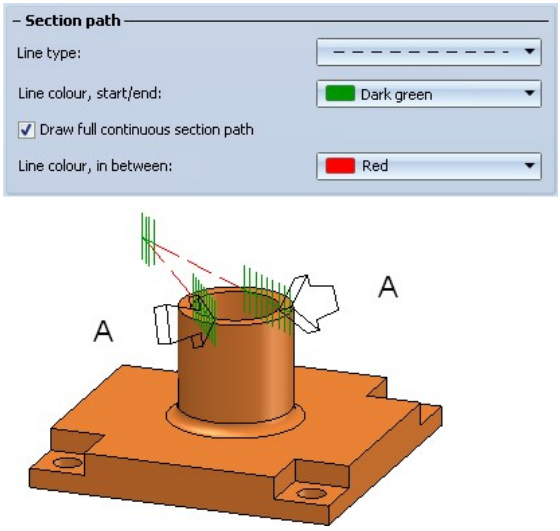
Representation of section paths

When creating sectional views, you can now use the  symbol to influence the representation of sectional paths. The dialogue window has been expanded for this purpose.


Identifier settings
Font settings
System font
HiCAD font
Font:
Arial
Font colour:
Red
Text height:
5
ABCDEFG...
Section path
Line type:
Line colour, start/end:
Blue
Draw full continuous section path
Line colour, in between:
Orange
Arrows
Form:
Arrowhead
Length:
Automatic
Value:
18.5
General
Layer:
1
Position of annotation at section path:
On lengthening of sectio...
OK
Cancel

Example:

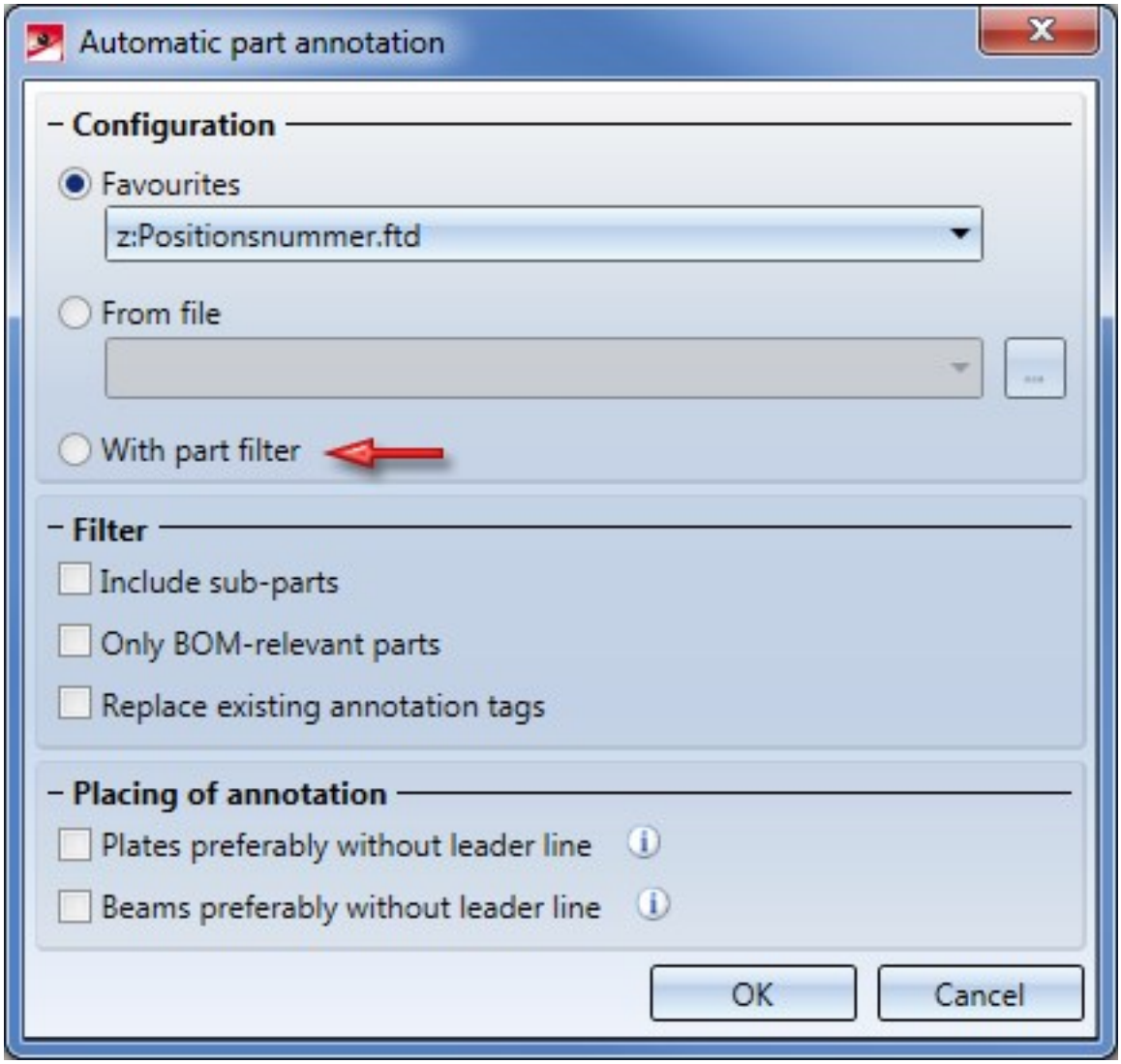




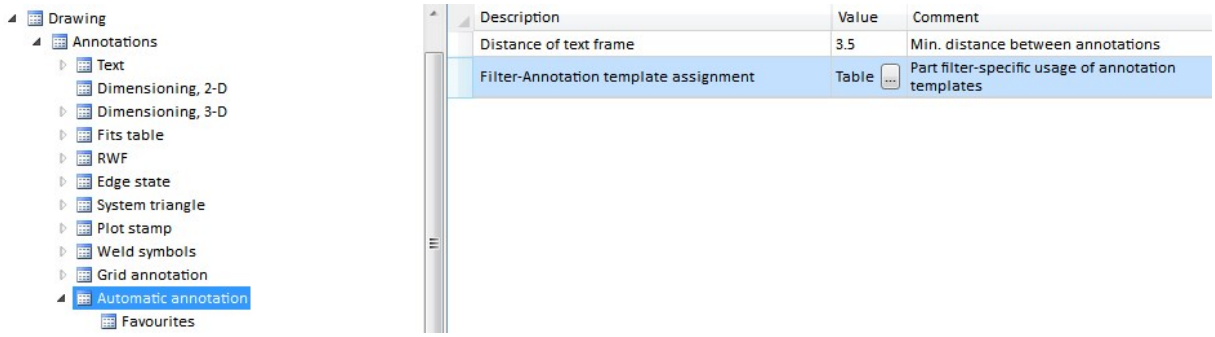
Dimensioning / Annotations

Using part filters for automatic part annotation

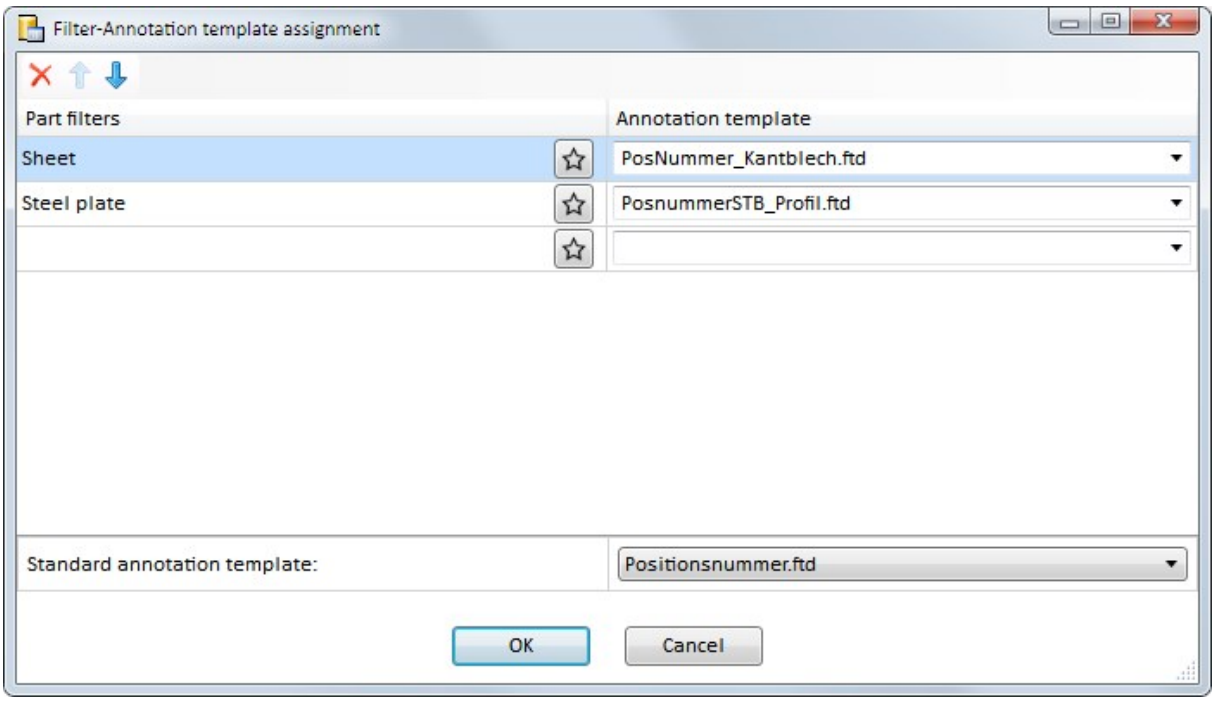
You have now the option to use part filters annotation templates for automatic part and part list annotation. Specific annotation templates can be assigned to these filters via the Configuration Editor, which allows you to use different annotation templates for beams, plates (Steel Engineering) and sheets (Sheet Metal). The filters that can be set in then Configuration Editor correspond to the filters for part search on the transparent toolbar. In the paragraph Using part filters for automatic part annotation explains how to use these filters.



The assigning takes place in the Configuration Editor at **Drawing > Annotations > Automatic annotation > Filter-Annotation template assignment**.



When you click on the ... symbol, the current assignment is displayed.



According to the ISD default setting, different annotation templates are used for sheets (Sheet Metal) and steel plates (Steel Engineering), namely, the templates that are offered in the table. For all other parts, the annotation template at the bottom of the window will be used.

Delete annotation, in rectangle

The **Delete, In rectangle** function has been added to the **Delete annotation** menu. Deletes all annotations within one or several selection rectangles. The annotations will only be deleted after a right-click.

Insert additional leader lines

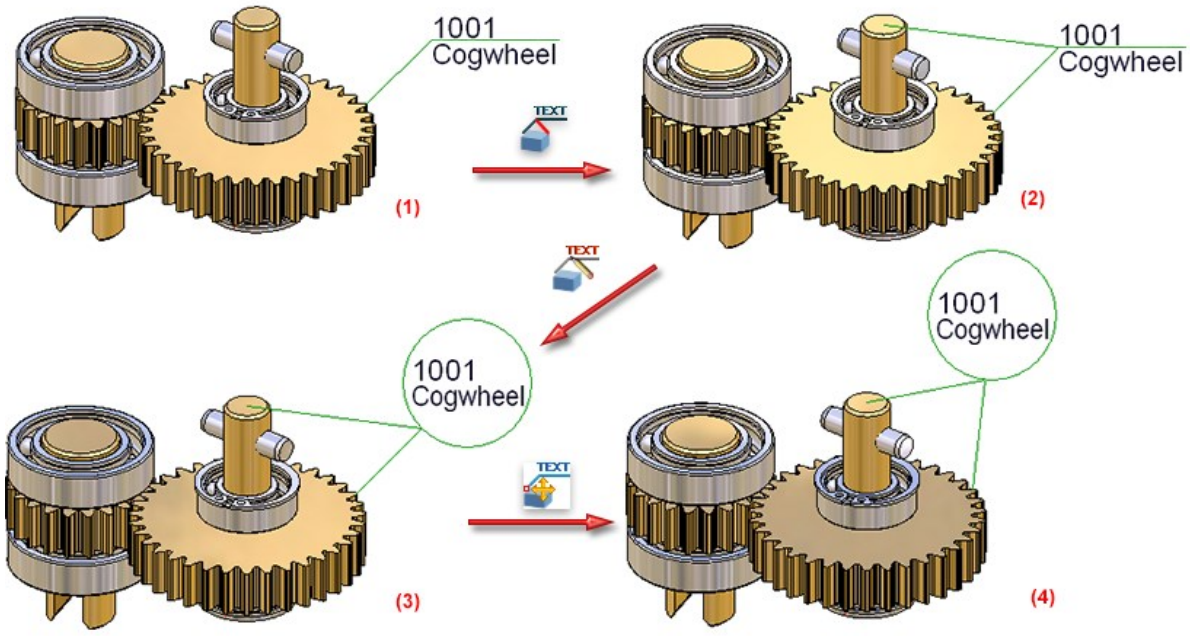
Use the **Insert base point** function to add additional leader lines to a part annotation. These leader lines have no own text block and always end at the text block of the selected part annotation. The function is only available in the context menu of part annotations.

Right-click on the part annotation for which you want to add further leader lines, and choose **Insert base point**.

Then, determine the reference point of the additional leader line. The line will be drawn from the text block of the selected annotation to the selected point. The start point of the additional leader line depends on the orientation of the selected part annotation (Reference/Insertion point), i.e. on the type of the selected complete frame.

Please note:

- When you move a part annotation, the additional leader lines assigned to it will also be moved.
- When you delete a part annotation, the additional leader lines assigned to it will also be deleted.
- When you edit an additional leader line, the edits will affect the complete annotation, i.e. it is not possible to change the line parameters of the additional leader line separately.



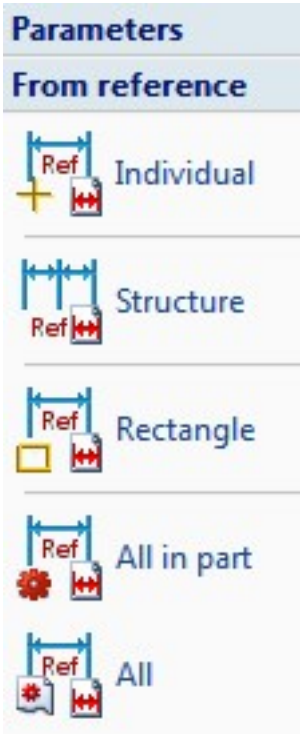
(1) Part with annotation; (2) Annotation with additional leader line, the base point is the centre of the frame; (3) Changed annotation settings; (4) Moving of annotation

Dimensioning of quad points

In older HiCAD versions, selected quad points were ignored during dimensioning in various cases. Instead, the quad point rotated 90° was used. From HiCAD 2019 onwards, quad points are handled like "real" point options, e.g. mid-points or centre points.

Apply parameters of reference dimensions

Use the functions of the **Parameters from reference dimension** menu to apply the parameters of existing dimensions to other dimensions of your drawing.



The operation of these functions has been facilitated in HiCAD 2019. The fist step after function call is the identification of the reference dimension, i.e. the dimension whose parameters you wish to apply. Then, choose (depending on the function) the dimensions to which you want to apply the parameters of the reference dimension.

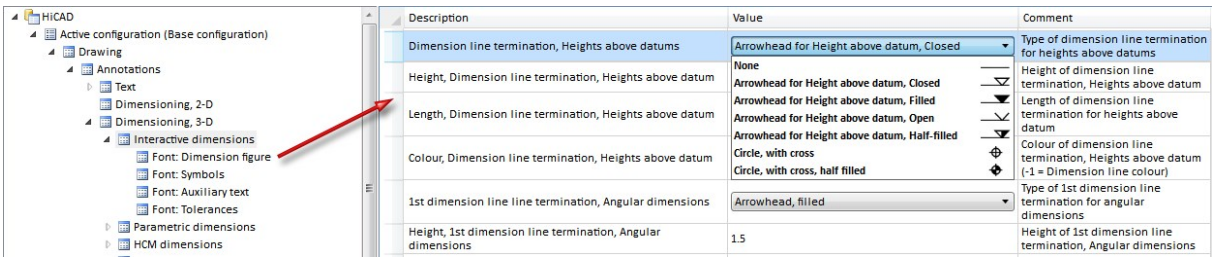
Height above datum - New symbols for dimension line terminations

For heights above datum new, additional symbols for dimension line terminations are available:

 Height above datum, Circle, with cross, unfilled, and

 Height above datum, Circle, with cross, half filled.

The dimension parameters and the setting options in the Configuration Editor at Drawing > Annotations > Dimensioning, 3-D > ... have been expanded accordingly.



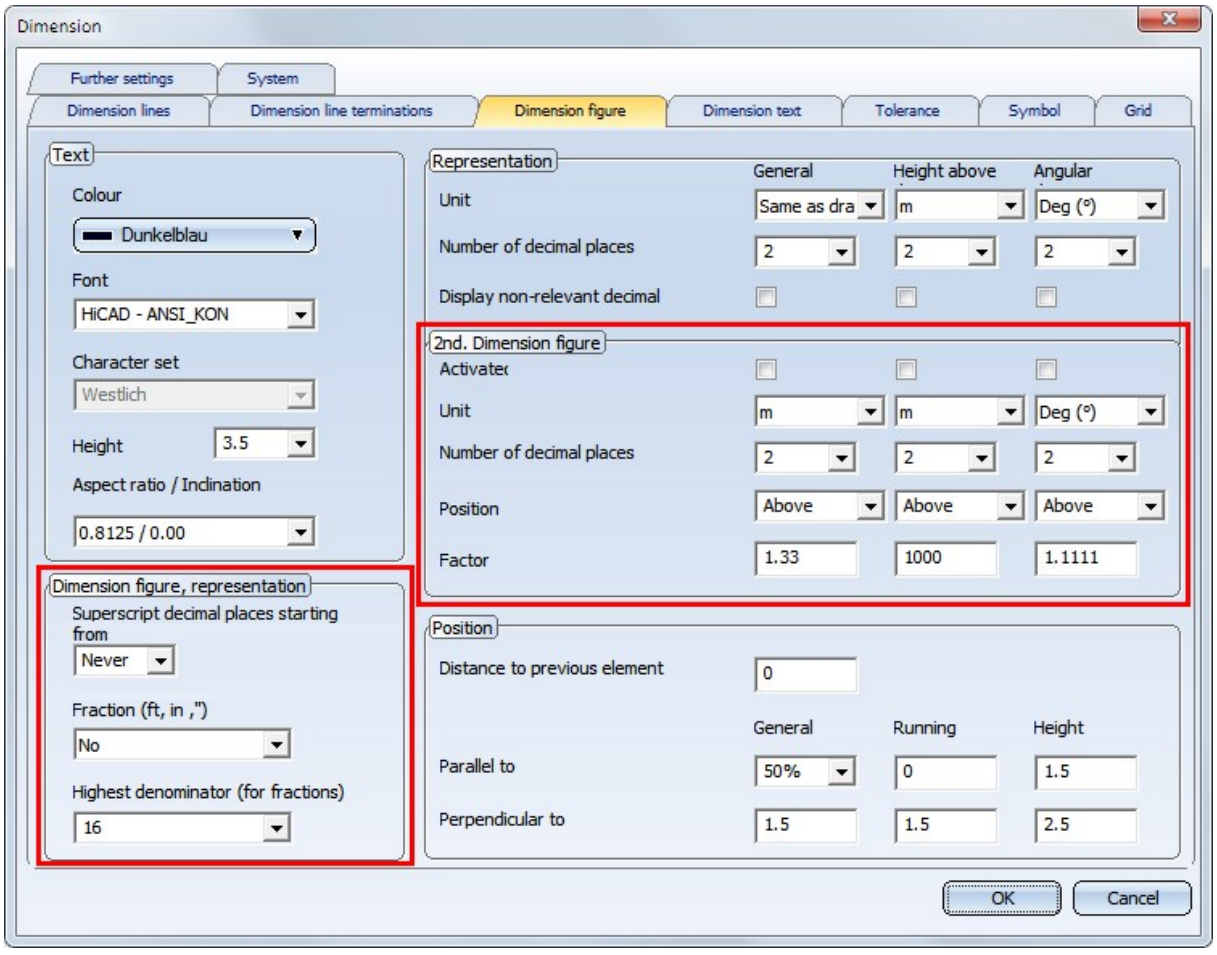
Dimension parameters - New options for 2nd dimension figure

From HiCAD 2019 onwards, new, additional dimension parameters are available. The **Dimension figure** tab has been adjusted accordingly.

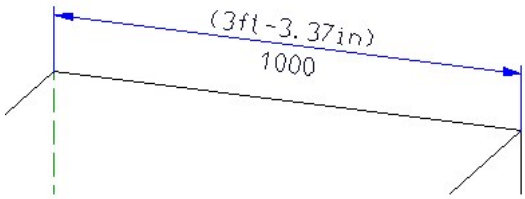
You can now also set the unit of measurement and the number of decimal places for the 2nd dimension figure. Additional settings such as superscripted decimal places or the conversion into fractures are taken over from the settings for the main dimension figure. If you choose a different setting than **None**, the entered conversion factor will no longer be considered and the 2nd dimension figure will be calculated via the selected unit.

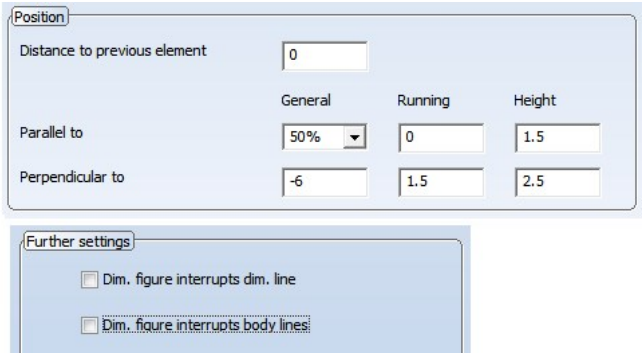
You use the **Position** parameter to determine whether the 2nd dimension figure will be shown above or below the main dimension figure.

The new parameters can be specified separately for general linear dimensions, heights above datum and angular dimensions.

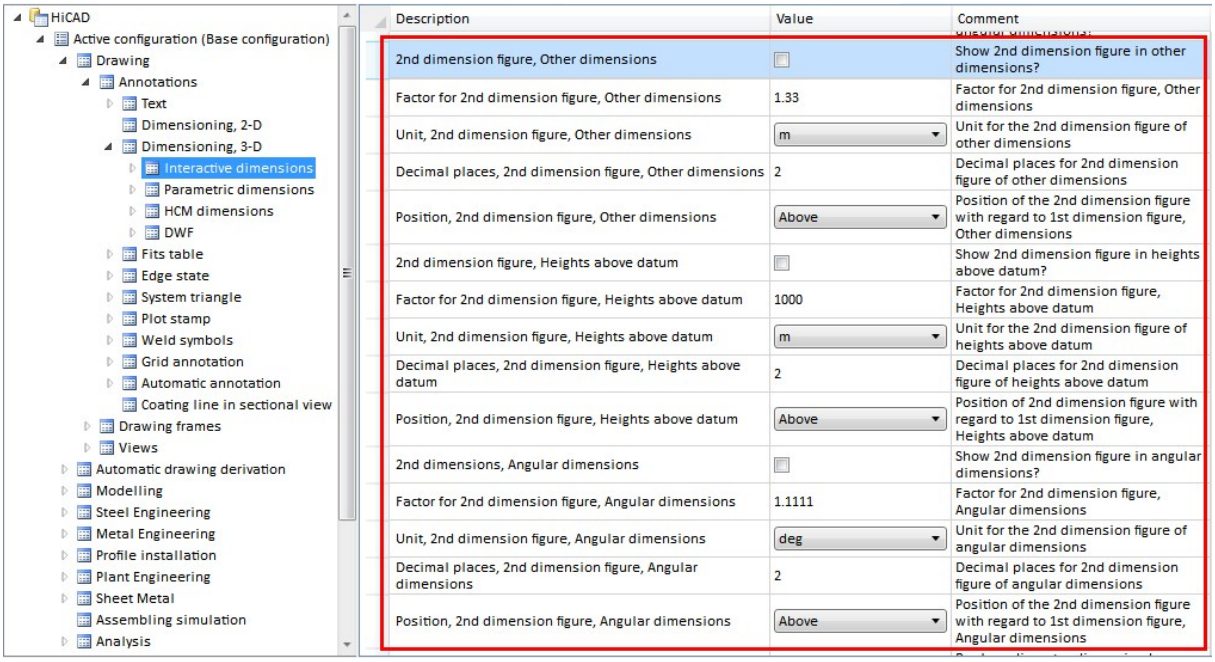


If you want to display two dimension figures with the first one being placed above, and the other one below the dimension line, you need to set the position of the dimension figure with regard to the dimension line accordingly in the **Position** area of the **Dimension figure** tab, and, on the **Further settings** tab, deactivate the interruption of the dimension line and body edges by the dimension figure.



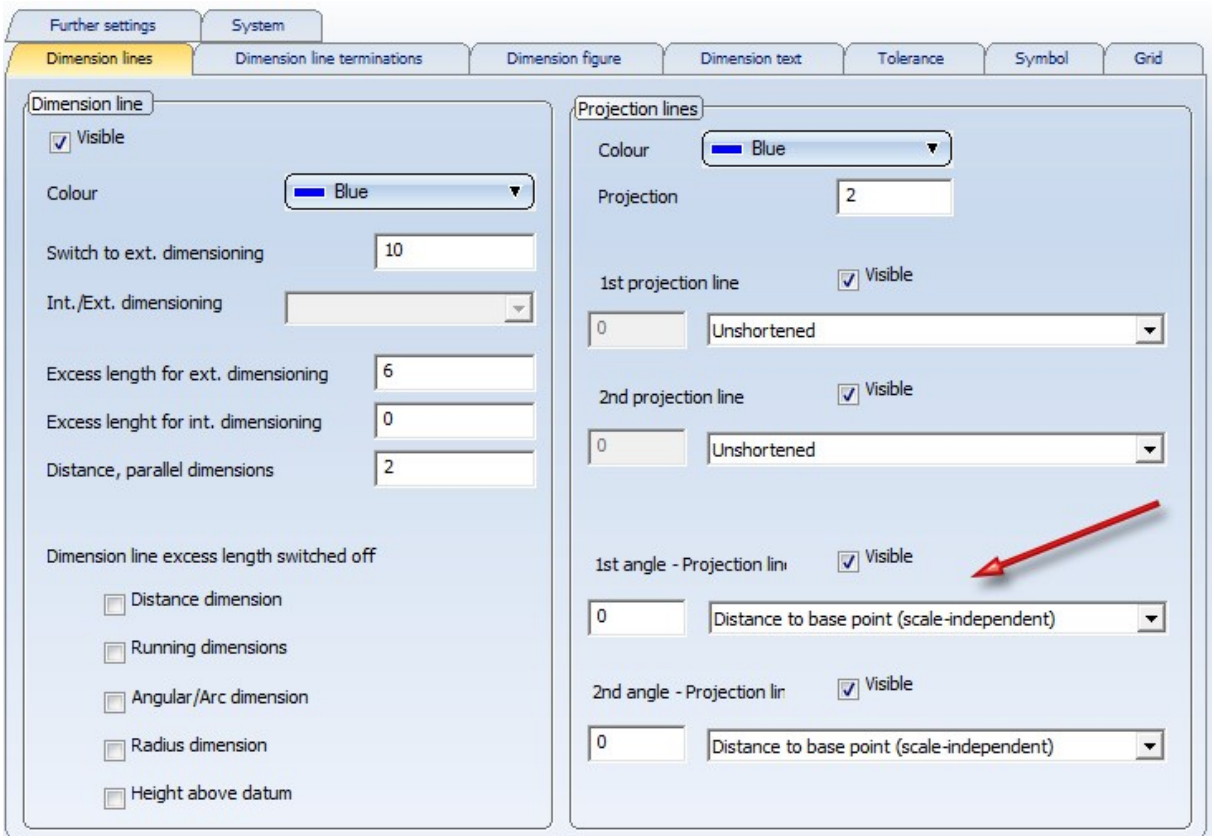


Furthermore, the setting options for 3-D dimensions in the Configuration Editor at **Drawing > Annotations > Dimensioning, 3-D > ...** have been expanded accordingly.



Separate settings for projection lines of angular dimensions

The pre-settings for projection lines of angular dimensions can now be specified separately. The **Dimension lines** tab of the Dimensioning settings dialogue window has been expanded for this purpose.



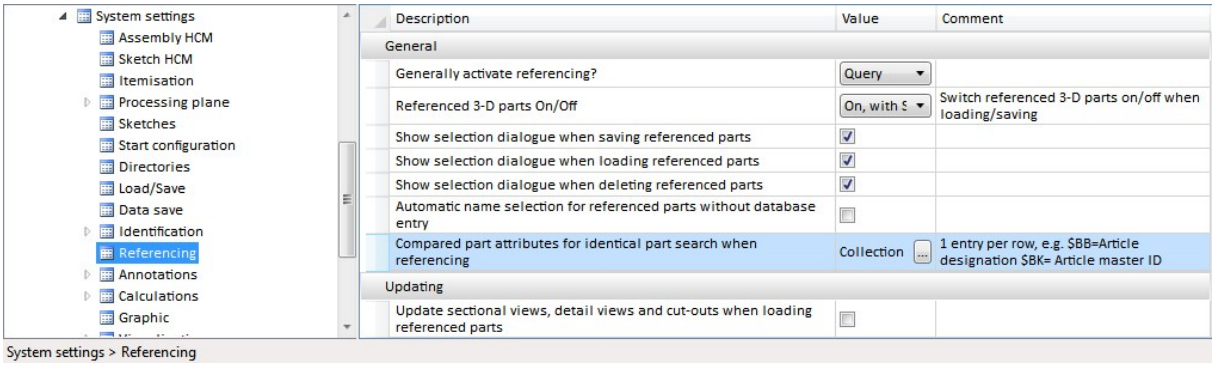
Variable dimensioning

The procedure for creating structure dimensions via variable dimensioning has changed. Now, you first need to specify all consecutive points before dropping the dimension.

Referencing

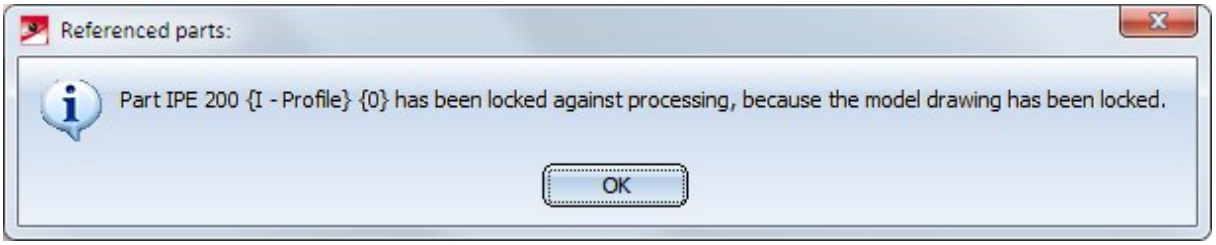
Locking referenced parts in read-only model drawings against editing

When working in write-protected, read-only model drawings you have the option to lock referenced parts of the drawing against accidental processing: In the Configuration Editor, the parameter **Lock referenced parts if model drawing is read-only** at **System settings > Referencing** is available for this purpose. The ISD default setting is **No**.

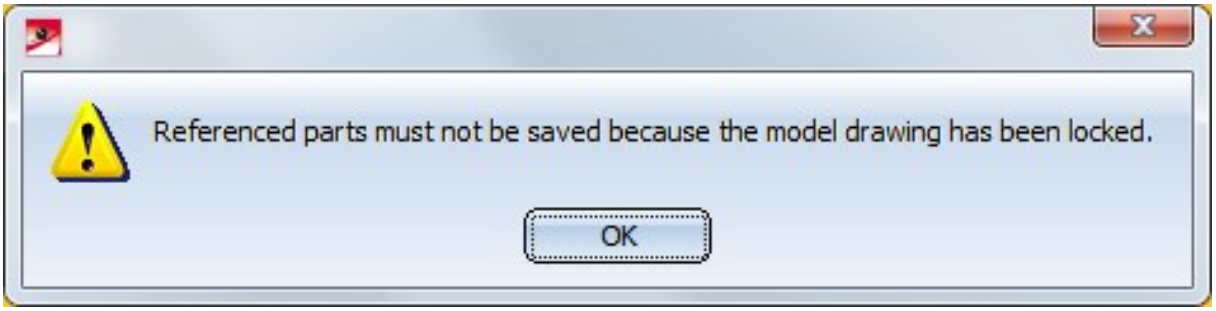


If the parameter is set to **Yes**, please note the following:

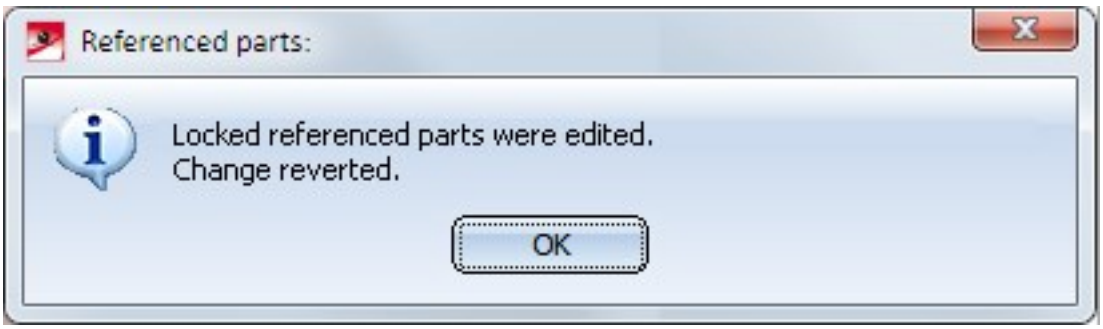
- A file is considered a "read-only" file, if
  - it has been opened with the HELiOS function "Open, read-only",
  - it has the Workflow status "Released",
  - it has been opened via the Explorer with the "Open drawing, read-only" option, or
  - you have not the permissions required for an opening of the file.
- If you attempt to modify a referenced part in a read-only file, the following message will be displayed:



- If you attempt to manually save a referenced part in a read-only file, the following message appears:



- During some operations you will only realize at the end that a referenced part was changed. In such cases the following message will be shown:



During simultaneous use of HELiOS please note the following:

When working with HELiOS, the parameter **Lock referenced parts if model drawing is read-only**, as well as the HELiOS setting **Allow part exchange in drawings** (HELiOS Options > Database > HiCAD) will be considered during updating and saving of referenced parts in read-only model drawings. This settings does not only affect the loading of a locked model drawing (referenced parts may not be updated from the computer), but also the saving of changed referenced parts. For example, if the setting option **Not for released or read-only** has been chosen, the message *Save rejected (Drawing is locked)!* has always been displayed upon saving of referenced parts. This applies even in cases where the parameter **Lock referenced parts if model drawing is read-only** in the Configuration Editor has been set to **No**. The dialogue for the saving of referenced parts will not appear either, but the changing of referenced parts will be allowed.

Dependent assemblies

At **3-D Standard > New > Assy.** > **...** you will find the new **Dependent assembly**  function.

This function allows you to replace an entire assembly by a dependent assembly, according to the same principle as with the Dependent part function.

This is only possible for original assemblies that are

- externally referenced and
- do not belong to a locked assembly.

Sketches

Point functions

As of HiCAD 2019 the **Point** functions on the **Sketch** tab are only available for planar sketches and 3-D sketches. For other parts such as assemblies, solids etc. please use the **Point** functions at **3-D Standard > Tools**.

Undo

UNDO actions now do no longer distinguish between sketches with and without HCM constraints, i.e. you can now use a step-wise UNDO on sketches with automatic HCM constraints. Also, the number of possible UNDO steps is now unlimited.

This is true both for planar sketches and 3-D sketches.

New part type 3-D sketch replacing c-edge part

In previous versions, only planar sketches had a special representation and a special behaviour, which was partly controlled by the purpose. It was geared towards creating and editing of parts and is very useful for this. For instance, the following applies to planar sketches with the purpose **Create/Edit**:

- They are displayed with broader lines when they are active.
- They are not hidden in shaded views when they are active.
- In sectional views, details and cut-outs they are not represented as cut sketches.

As of HiCAD 2019, 3-D sketches also benefit from these advantages. Here's an overview of what's new for 3-D sketches:

- 3-D sketches, too, can be assigned a purpose that influences their representation.
- 3-D sketches have, like other 3-D parts, a Part CS that cannot be modified.
- For 3-D sketches, too, the settings of the **Novice configuration** are considered.
- The context menus for planar sketches and 3-D sketches have been standardized.

Furthermore, one now distinguishes between the part types **Sketch** and **3-D sketch** in HiCAD 2019. Planar sketches  and 3-D sketches  are marked with different symbols in the ICN, and the part filters for the **Find** function can now be used separately for planar sketches and 3-D sketches.

The new part type **3-D sketch** replaces the C-edge parts (before HiCAD 2019). Where C-edge parts were allowed you can now use 3-D sketches.

When you load model drawings that were created and saved with versions before HiCAD 2019, HiCAD wil attempt to automatically convert the affected parts to 3-D sketches without changing the drawings, so that you can still benefit from the advantages of the part type **3-D sketch**. Notes on conversion can be found here.

Convert part into 3-D sketch

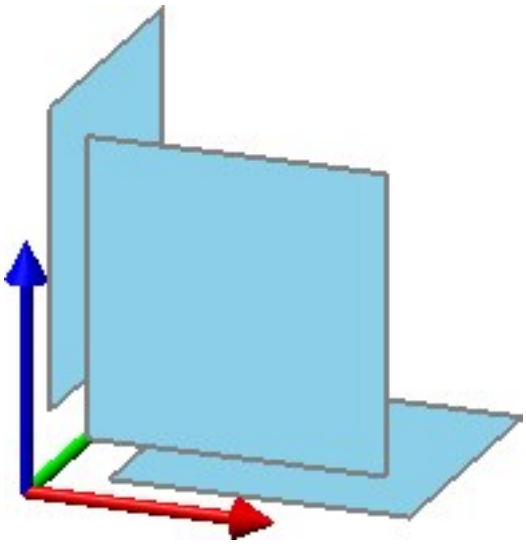
At **Sketch > New > 3-D Sketch**  **> ...** you can find the new Convert function. Use this function to convert parts with lines into 3-D sketches. This makes sense if, for instance, you want to import parts via the 3-D interface.

As is the case when you create a new 3-D sketch, you will be prompted to select a plane for Part CS definition.

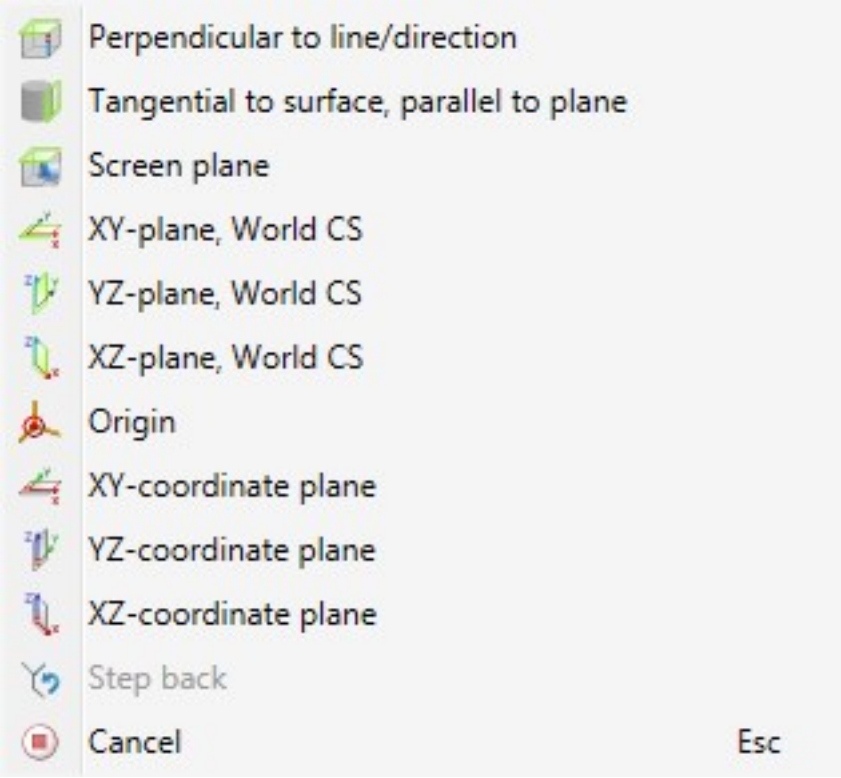
3-D sketches - Selection of plane

For 3-D sketches, too, you can now determine a plane in which the sketch is to be placed. Also, the settings of the Novice configuration are now considered for 3-D sketches.

If the **World CS planes fro new sketch (also 3-D sketch)** checkbox has been activated in the **Novice configuration** dialogue window, you can just select one of the displayed default planes with the cursor.

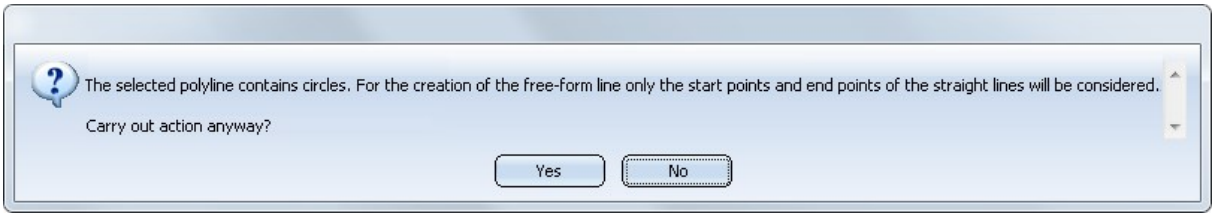


If you do not want this, or if the selectable default planes are not displayed, you can determine the plane by selecting points, edges, planes or processing planes. Right-clicking opens a context menu with further functions.

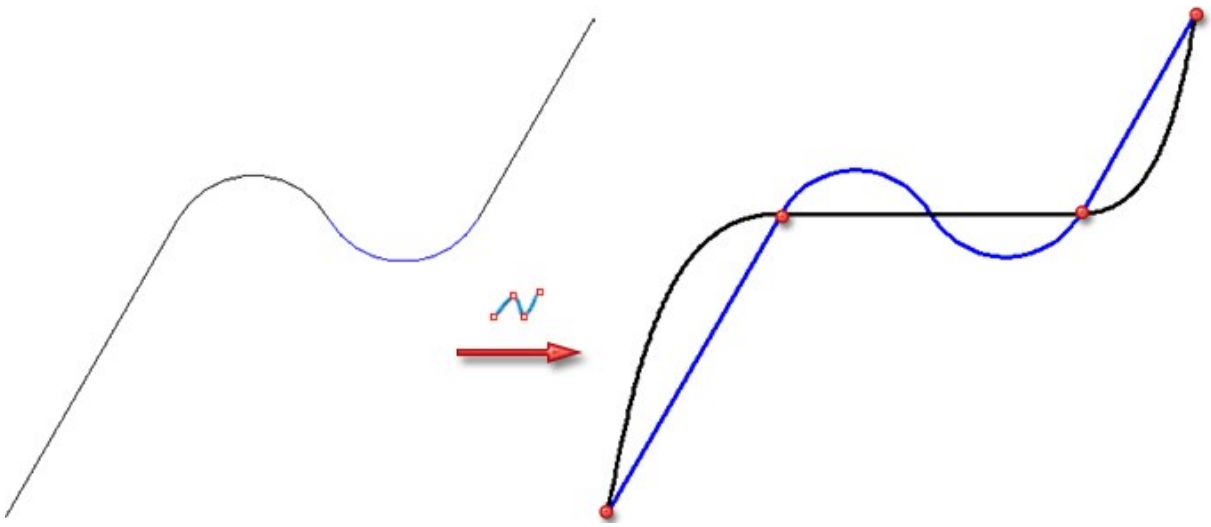


Deriving splines

Whan deriving splines from polylines it will now be checked whether the selected c-edge contains curves. If this is the case, an appropriate message will be issued, e.g.:



When you confirm this message with **Yes**, only the start points and end points of the straight line will be considered for the creation of the spline, as shown in the image below, where an Akima spline (including the arcs) has been derived from the sketch on the left hand side.



If you select **No**, the function will be cancelled, and no spline will be created.

To convert these polylines in such a way that splines can be directly derived from them, you can choose between the following options:

- Choose the Divide, Number  function. Enter a suitable number of segments and activate the **Convert curves into straight lines** option.
- Apply the Replace line  function to each curve of the polyline, activating the **Replace with straight lines** option.
- Directly convert the polyline with the C-edge -> NURBS  function.

Model and process parts

Point functions

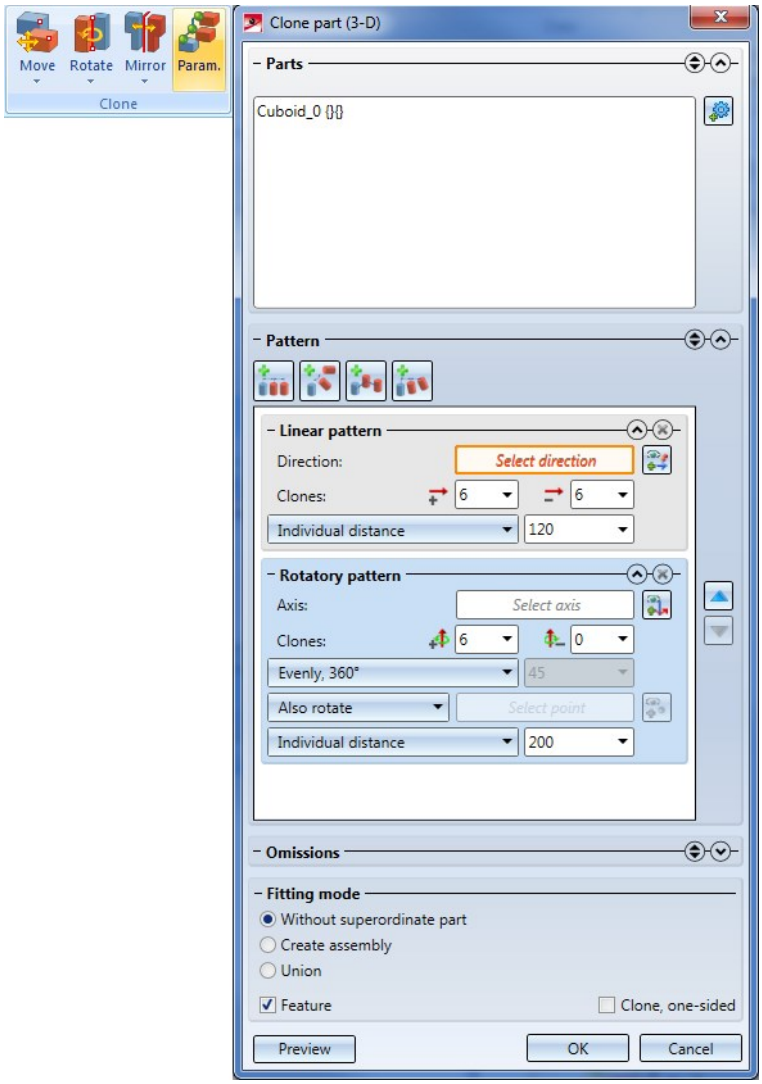
As of HiCAD 2019, the **Point** functions at **3-D Standard > Tools** are no longer available for planar sketches and 3-D sketches. Instead, use the **Point** functions on the **Sketch** tab for these part types.

Clone part, parametric

The previous functions

- Clone, Parametric along c-edge
- Clone, Parametric grid
- Clone, Rotate - parametric
- Clone, Parametric with displacement

have been combined into one new, central function called **Clone parts (parametric)**.



Others

Preview mode during part creation

When creating 3-D parts, a preview of the new part is normally displayed. Some functions show this preview automatically, others require a click on the **Preview** button.

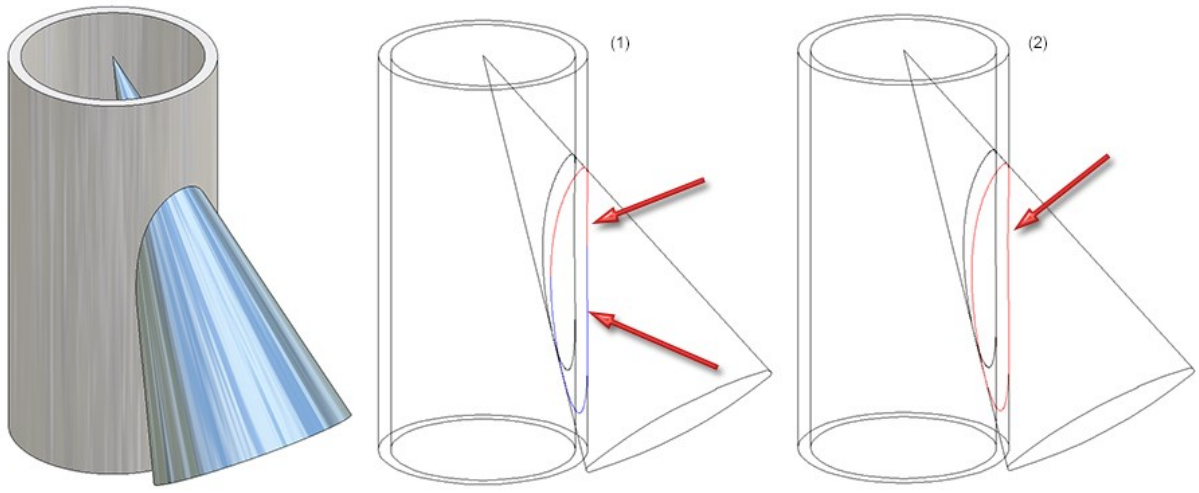
In the preview mode the model drawing can also be rotated dynamically by clicking with the MMB on the drawing area. Based on the current cursor position, the next surface beneath the cursor will be selected and the rotation will be performed about this surface. Some newer functions will also recognize when the cursor points to the preview of the new part to be created. In this case this preview can also be rotated. However, please note that the **Apply immediately** checkbox must not be active here. This currently applies only to the following functions:

- Insertion of solid primitives,
- Glass from sketch,
- Newer functions for Sheet Metal parts, such as Create base sheet or Attach flange, and
- Plane selection for sketch creation in the Novice configuration mode.

Improved intersection routines

The internal routines for intersections between cylinders and cones have been revised, resulting in improved results, e.g. during Boolean operations.

In the example below the cone has been subtracted from the cylinder. In previous versions, 2 section curves with 286 and 301 control points (1), respectively, were created in the process. In HiCAD 2019, only 1 section curve with 58 control points (2) is created.



Catalogue Editor

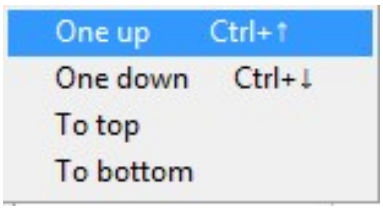
Service Pack 2 2019 (V 2402)

Change order of tables in catalogues

The order of tables in the catalogue structure tree can be now changed by means of the **Move table** function. This affects both the display in the Catalogue Editor and the selection via HiCAD. For instance, this function allows you to move frequently used tables to the top of the catalogue structure tree.

Proceed as follows to change the order:

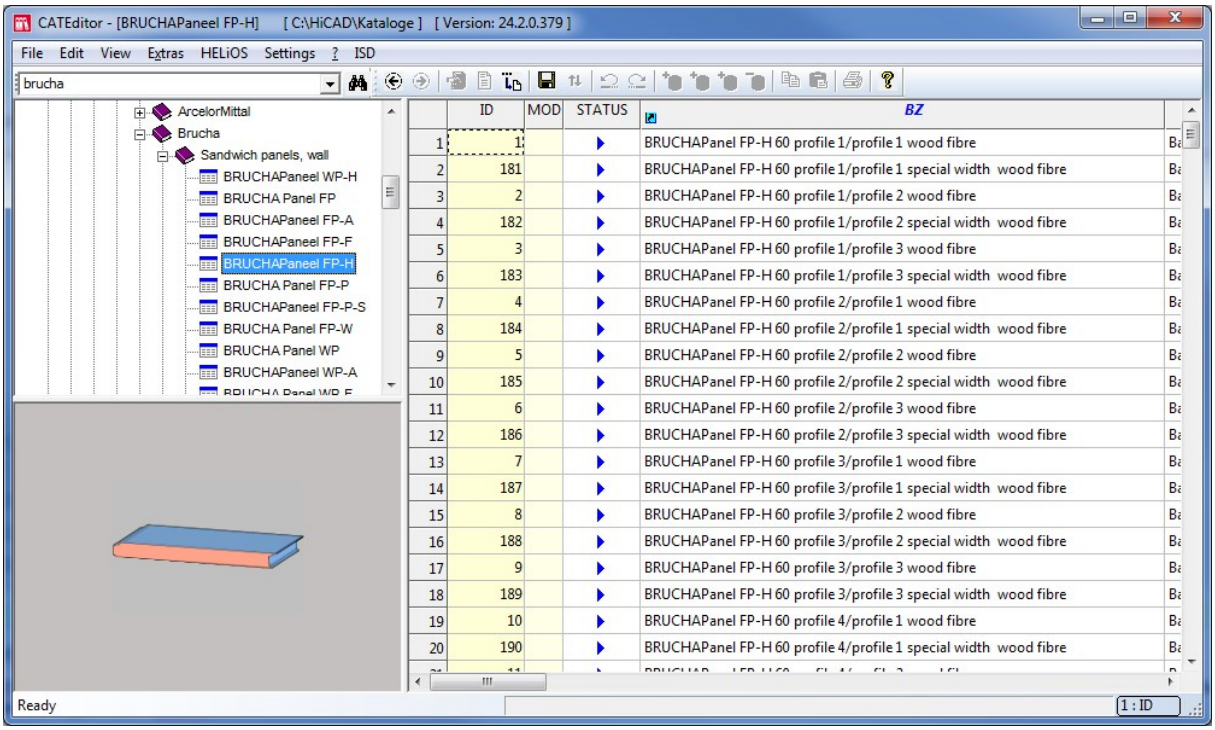
- Move the cursor on the name of the table to be moved, mark the table with a click and move the marked table up or down with the shortcuts CTRL+Arrow key.
- Right-click the table to be moved, choose **Move table** in the context menu, open the sub-menu of the function and choose the desired option:



Please note that tables, but not their sub-tables, can be moved.

Additional BRUCHA panels for element/profile installation

At **Factory standards > Series > Roof wall facade > Room-closing profiles > Brucha** you can now find new panels with different sizes by the company **Brucha** for element and profile installations.



Eternit® installation elements

HiCAD now also supports the installation of Eternit panels for element installations and sub-structures. These can be found in the following catalogues:

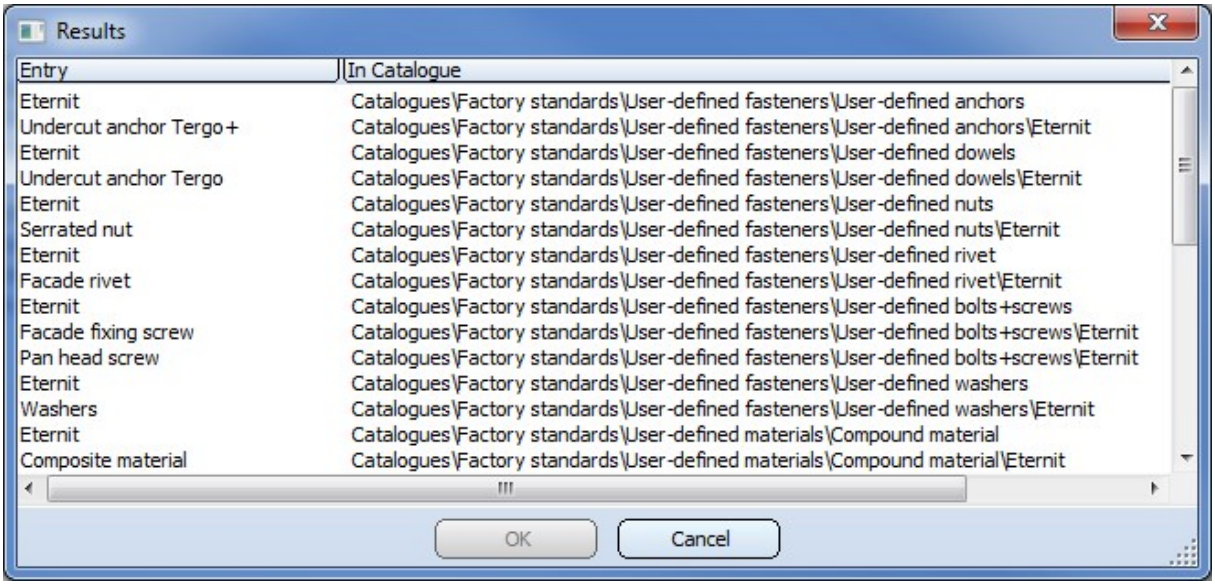
- **Factory standards > Installation Planning - Parts and Processings > Element installation > Installation elements > Eternit**.

  Available are:
  - EQUITONE Tergo and
  - EQUITONE riveted
- **Factory standards > Installation Planning - Parts and Processings > Sub-structure > Installation elements > Eternit**

  Available are:
  - Riveted to L-profile
  - Riveted to T-profile and
  - Systea UBE 25/2

Eternit® Standard Parts

New in the **Factory standards** catalogue are standard parts that are required for installing of Eternit panels / substructures. These are various Eternit fasteners, Eternit sleeves and Tergo / Tergo+ undercut anchors and dowels.



Eternit® semi-finished product data

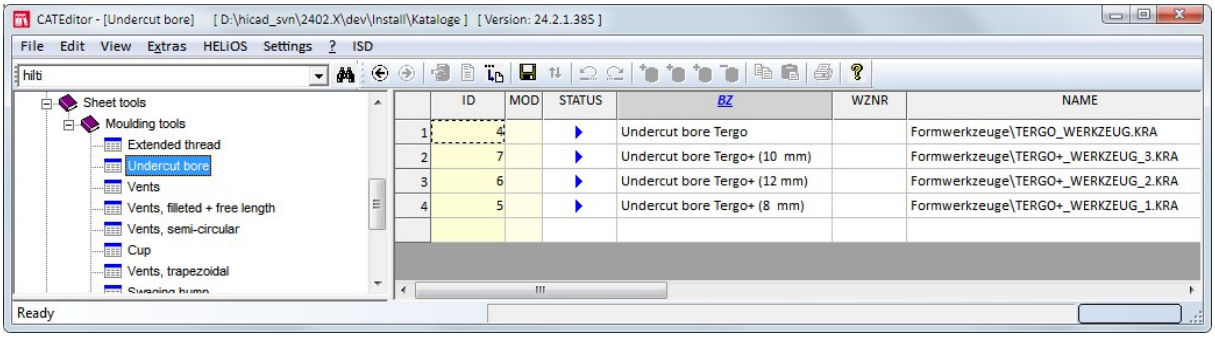
The semi-finished product data for Eternit panels can be found at **Factory standards > Sheets > Eternit** in the following tables:

- EQUITONE [linea]
- EQUITONE [materia]
- EQUITONE [natura]
- EQUITONE [natura] PRO
- EQUITONE [pictura]
- EQUITONE [tectiva]
- EQUITONE [textura]

The semi-finished product-dependent part colours are defined in the tables at **Factory standards > Surface treatment > Profile and Element Installation > Eternit**.

Eternit® undercut bores for Tergo / Tergo +

At **Sheet tools > Moulding tools** a new table called **Undercut bore** for Tergo and Tergo + is available.



The insertion in HiCAD takes place with the function **Sheet Metal > Further Tools > Mould**  .

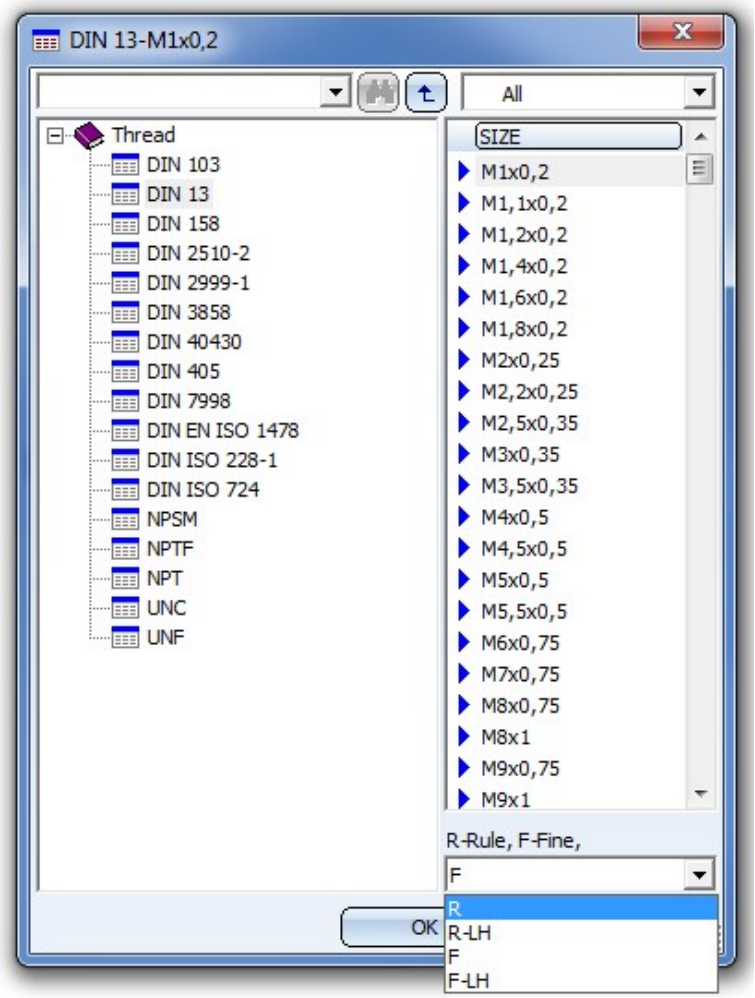
HILTI consoles for rear-ventilated facades

New in the catalogue **Factory standards > Purchased/Factory standard parts > Wall consoles** are (optional) HILTI consoles for rear-ventilated facades. Further information can be found in the **Wall Bracket** topic of the **Element Installation** Help chapter.

Service Pack 1 2019 (V 2401)

Thread types - Changed order

For many parts, different types of threads are available, e.g. F (Fine thread), R (regular thread etc.). The sorting of these tables also affects the order in the selection box that is displayed in HiCAD during insertion of objects. For some of these tables the sorting has changed in such a way that the different types will be available in HiCAD in the following order during part insertion:



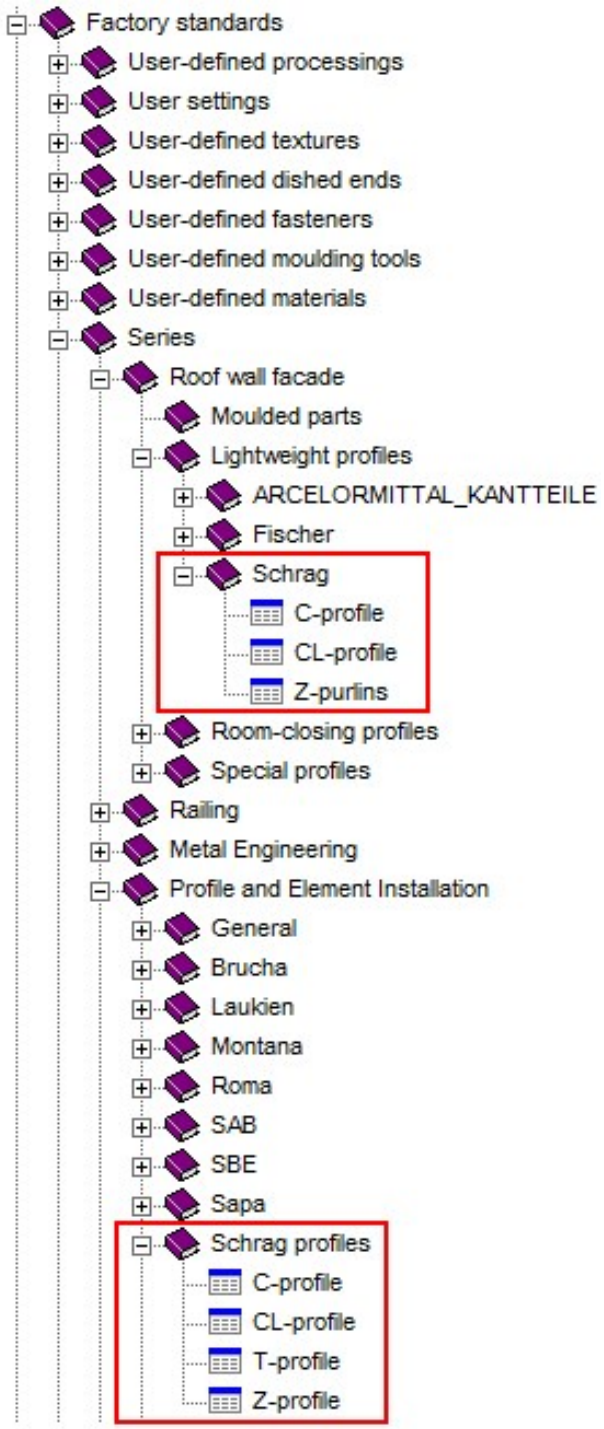
This concerns the tables of the catalogues

- Threads,
- Outer threads,
- Thread runout, and
- Thread undercut.

Threads with other type groups (e.g. various JIS or UNC threads) have remained unchanged.

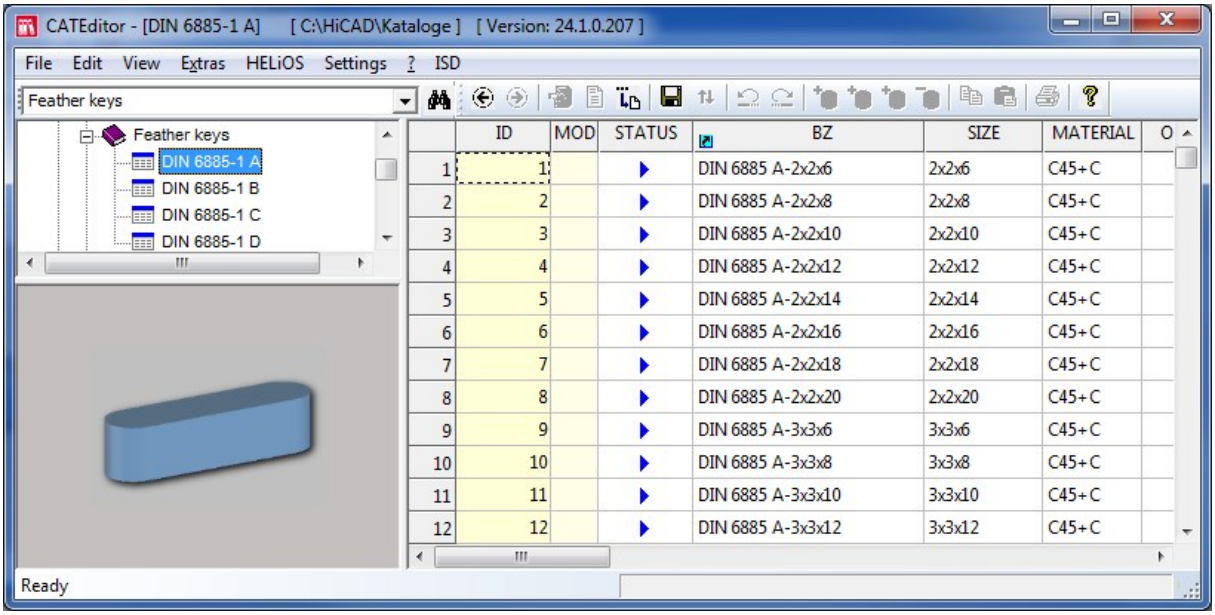
Schrag profiles

- The profiles of the company Schrag at **Factory standards > Series > Profile and Element Installation > Schrag profiles** have been updated according to the current description of the manufacturer.
- New are the Lightweight profiles at **Factory standards > Series > Lightweight profiles > Schrag**.



Feather keys

In previous versions, no material was assigned to feather keys. From HiCAD 2019 SP1 onwards, the material **C45+C** (unalloyed hardened and tempered steel) is assigned to the feather keys acc. to DIN 6885-1 A - D.



Other fasteners acc. to ASTM standard

As of HiCAD 2019 SP1 many new **Fasteners** acc. to ASTM Standard (**A**merican **S**ociety for **T**esting and **M**aterials) are available:

Table	Catalogue
ASTM A193 / A193M	Bolts+Screws\Hexagon head bolts
ASTM A307	Bolts+Screws\Hexagon head bolts
ASTM A325 / A325M	Bolts+Screws\Hexagon head bolts
ASTM A354	Bolts+Screws\Hexagon head bolts
ASTM A449	Bolts+Screws\Hexagon head bolts
ASTM A490	Bolts+Screws\Hexagon head bolts
ASTM F593	Bolts+Screws\Hexagon head bolts
ASTM A193 / A193M	Bolts+Screws\Cheese-head screws
ASTM F1554	Bolts+Screws\Anchor bolts
ASTM A194 / A194M	Nuts\Hexagon head bolts
ASTM A563	Nuts\Hexagon head bolts
ASTM A320 / A320M	Washers\Washers
ASTM F436 /F436M	Washers\Washers

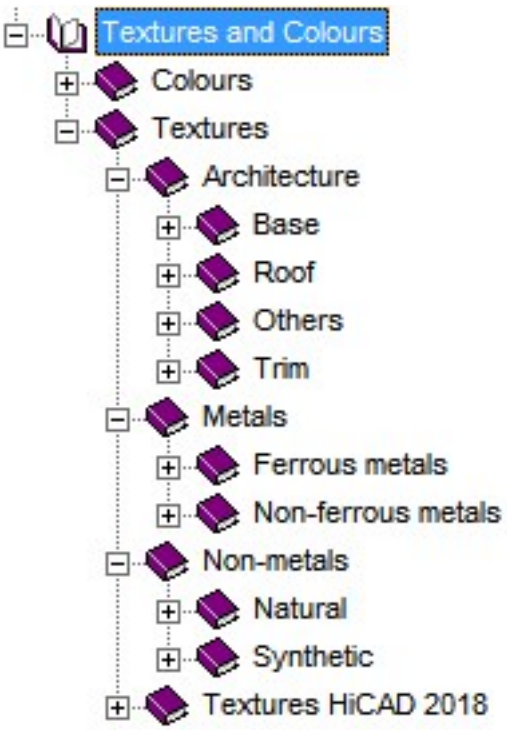
Major Release 2019 (V 2400)

New textures

In HiCAD 2019 the functionality for textures has been revised completely. As of HiCAD 2019, textures are no longer assigned via the **Visual effects** docking window. Instead, you can use the following functions (see also HiCAD 3-D - What's New?):

Texture - New (3-D part list) assigns a texture to the active part or the active part list	at **3-D Standard > Tools > Attr. > ...** or in the context menu of a part list at **Properties > ...**
Texture - New (3-D part) assigns a texture to the active part	in the context menu for 3-D parts at **Properties > ...**
Change texture, active part (3-D part) changes the texture of the active part	in the context menu for 3-D parts
New texture Definition of new textures / change existing textures	at **Drawing > Others > Extras > ...> Extras**

Furthermore, a multitude of textures is now available, which are stored in the folder **Textures and Colours > Textures**. The previous textures can be found at Textures HiCAD 2018.



DIN ISO 7379 Hexagon socket head shoulder screws

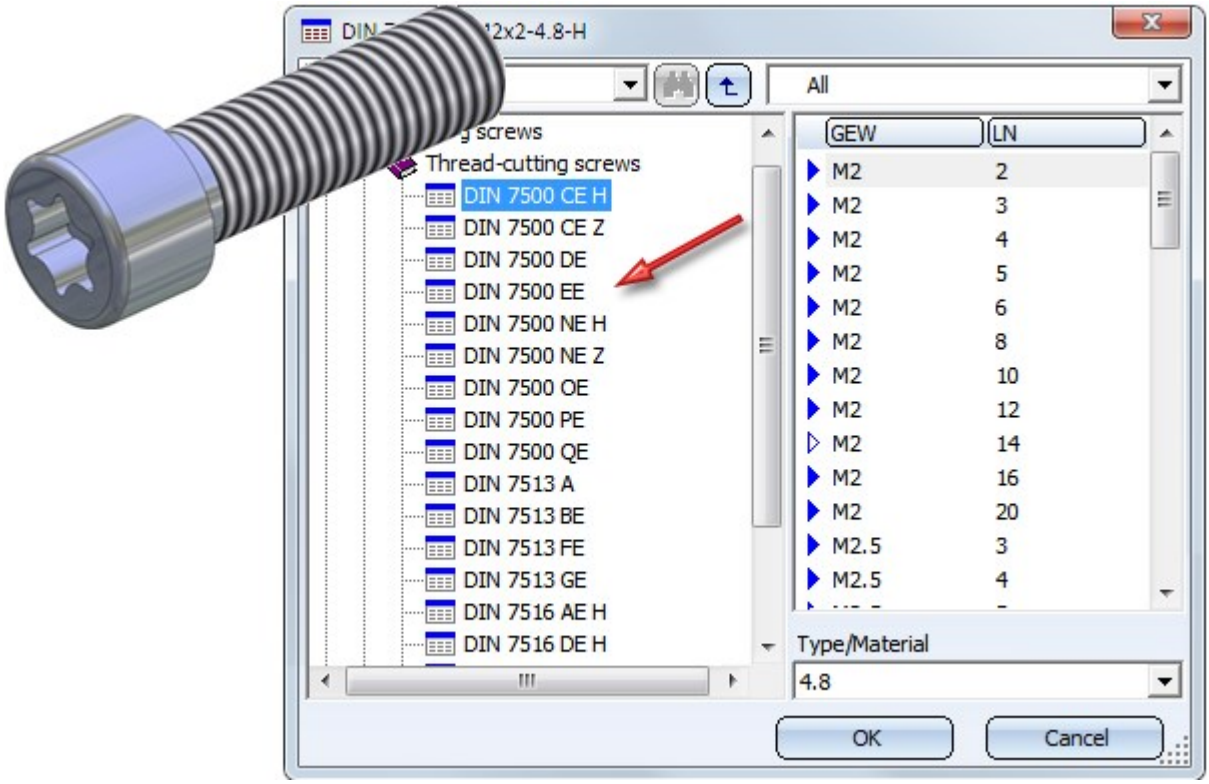
The **Fasteners > Bolts+Screws** catalogue now contains the new Hexagon socket head shoulder screws according to DIN ISO 7379 in the **Fitting screws** table.

DIN 444-B - Eye bolts

In the **Fasteners > Bolts+ Screws > Various bolts+screws** catalogue, the new Eye bolts according to DIN 944 have been added to the **Various bolts+screws** table.

DIN 7500 - Thread-cutting screws

At **Fasteners > Bolts+ Screws > Thread-cutting screws** you can now also find the new Thread-cutting screws according to DIN 7500.



New usages for BIM

New in the catalogue **Factory standards > Usage > Civil Engineering > Steel Engineering** is the table **Construction section**.

Construction sections and structure assemblies with this usage can now even be transferred to HELiOS if they are not BOM-relevant (see also What's New in Management + BIM).

Other new catalogue parts / processings

- **DIN 440 R Washers**
  As of HiCAD 2400, washers according to DIN 440, Form R will be available. You can find these washers at **Fasteners > Washers > Washers**, in the table **DIN 440 R**.
- **DIN 103 Threads**
  Furthermore, new left-handed threads according to DIN 103 (Type F-LH) are now available.

Dished ends - CUSTOM columns

In the tables that are used for part processing, e.g. the tables beneath **Processings, general**, **Fasteners**, **Semi-finished products** etc., you will find CUSTOM columns that allow you to assign customer-specific information to the corresponding parts.

These have now also been made available for **Dished ends**. This enhancements applies to all tables of the catalogues **Dished ends** and **Factory standards > User-defined dished ends**.

Catalogues/tables for profile installation

Depending on the catalogue to which they belong, tables can be assigned to a particular category. For profile installation, the category **Parts** is available.

When you create a new table for this category, the corresponding table columns will be generated automatically.

The tables that are supplied with HiCAD and can be found at **Factory standards > Installation planning - Parts and Processings > Profile installation** have already been assigned to this category.

ALUCOBOND® SZ-20 sub-structure

In previous versions the ALUCOBOND® SZ-20 sub-structure was available as a hat profile in the catalogue **Factory standards > Installation Planning -Parts and Processings > Sub-structure > Installation elements** and the table **ISD Example**.

From HiCAD 2019 onwards you will find it in the table **ALUCOBOND** of the same catalogue instead.

Feature Technology

Service Pack 2 2019 (V 2402)

Weight in dependent assemblies

When creating dependent assemblies it is now also possible to transfer the "Weight" attribute. In this case the weight will be automatically fixed in the dependent parts.

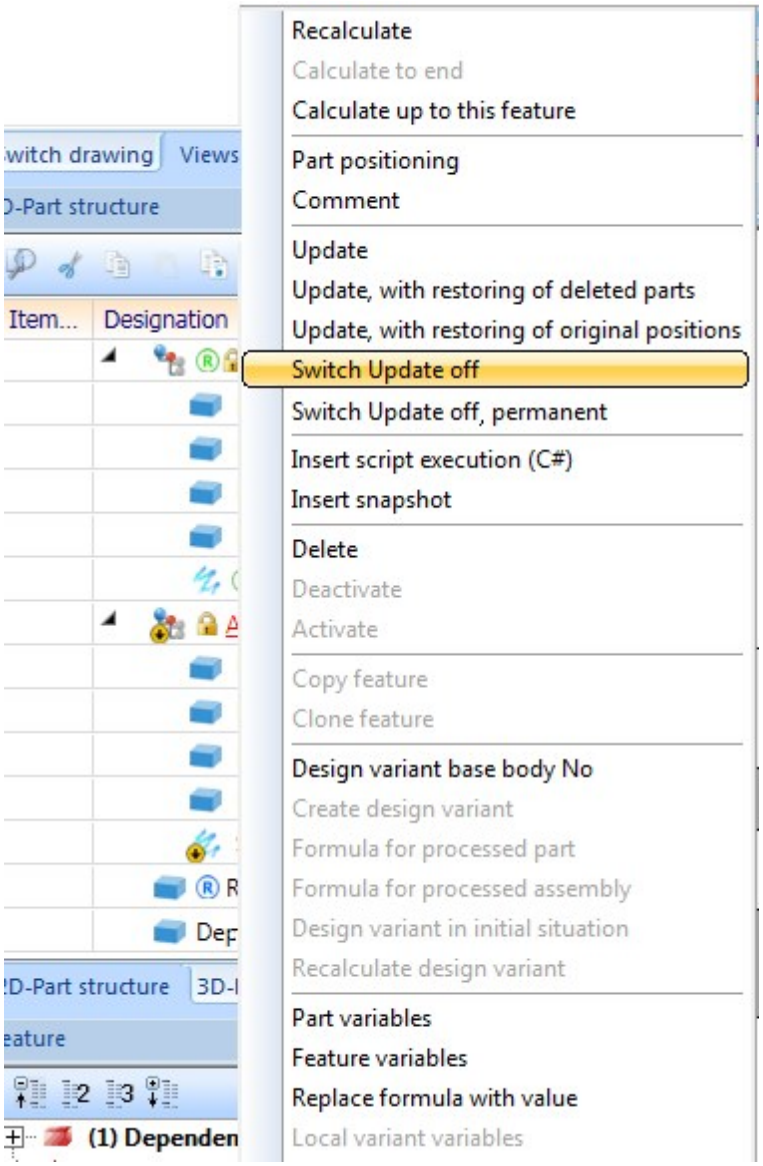
Dependent assemblies with internally referenced parts

You can now also create a dependent assembly if it contains internally referenced parts. These parts will be converted into dependent sub-parts when the dependent assembly is created.

However, the original assembly still needs to be externally referenced.

Switch off updating of dependent assemblies

The updating of dependent assemblies can now be switched off temporarily or permanently. For this purpose the context menu of the new options **Switch Update off** and **Switch update off, permanent** have been added to the context menu.

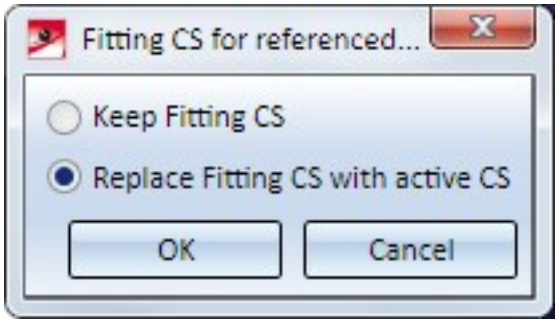


Switch Update off deactivates the updating only temporarily - after selecting this option an updating of the dependent assembly will be prevented, the corresponding Feature log entry will be renamed to **Dependent part (Deactivated)** and in the context menu the option **Switch Update on** will be offered.

Switch Update off, permanent also deactivates the updating, but permanently, i.e. this step cannot be undone (except with the UNDO) function. The corresponding Feature log entry will then be renamed to **Dependent part (Switched off)**, and the context menu entries for updating will no longer be displayed.

Fitting CS for referenced parts

3-D parts will automatically be assigned a Fitting CS when being referenced, which will be recorded correspondingly in the Feature log. This applies to both internal and external referencing. In previous versions, fitting coordinate systems that already existed during referencing were deleted. From HiCAD 2019 SP2 onwards, this behaviour has changed: If a Fitting CS exists when a part is being referenced, the following message appears:



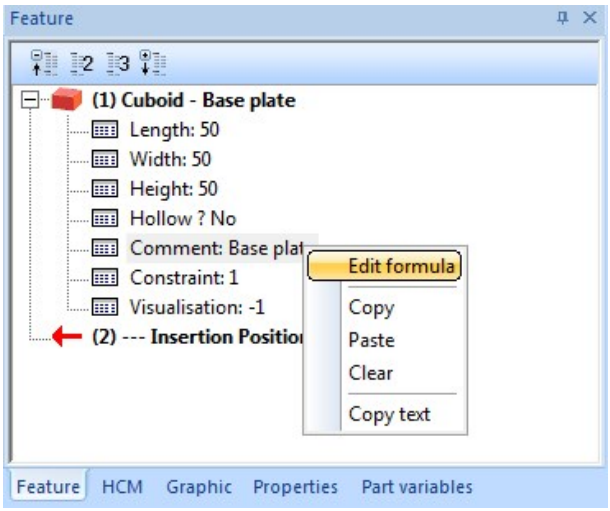
By activating the desired option and confirming with OK, you can either,

- keep the already existing Fitting CS, or
- replace the existing Fitting CS with the active CS.

Service Pack 1 2019 (V 2401)

Context menu for comments

Each Feature log entry in the ICN contains a sub-entry called **Comment**. Here you can enter a comment on this feature which will be shown next to the name of the feature. If you right-click this entry, a context menu will be displayed:



Edit formula opens a Formula Editor, allowing you to edit the comment you entered. The Formular Editor can also be opened with a double-click on the Comment entry. **Copy** copies the comment to the Clipboard, enabling you to **Paste** in into another feature by chosing the same-named function. **Clear** deletes the comment. **Copy text** copies the text of the comment to the Clipboard, allowing you to paste it in a different program (i.e. not HiCAD).

Clone processing: Edge selection after distance determination via "Up to reference..."

If you call the **Clone processings** function and, in the dialogue window, choose **Linear pattern** and determine the distance via **Up to reference, with distance** or **Up to reference, with number**, you can now also choose an edge as end reference.

In previous versions, the identification of a second edge (for the definition of a surface) was always required after choosing an edge.

Undo/Redo during processing of sketches

In the **Process sketch** dialogue, which can be called via the Feature log of a sketch used for the processing of parts (e.g. C-edge sweep, Extruded part, Revolved part, Add part etc.), the Undo and Redo option will now be available, allowing you to revoke or restore processings.

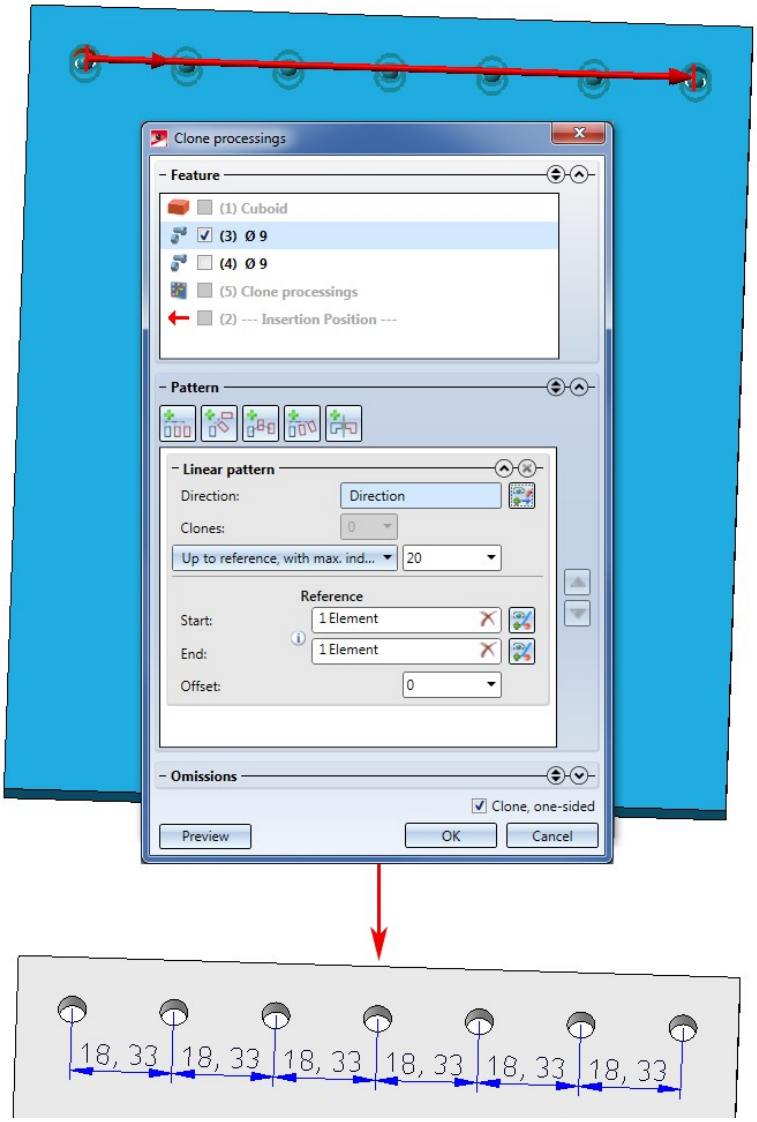
For changes of sketches in sectional views, detail views, cut-outs or for created c-edge sweeps this will not be possible.

Clone processing: Up to reference, with maximum individual distance

The **Clone processings** function now offers the new option **Up to reference, with max. indiv. distance**.

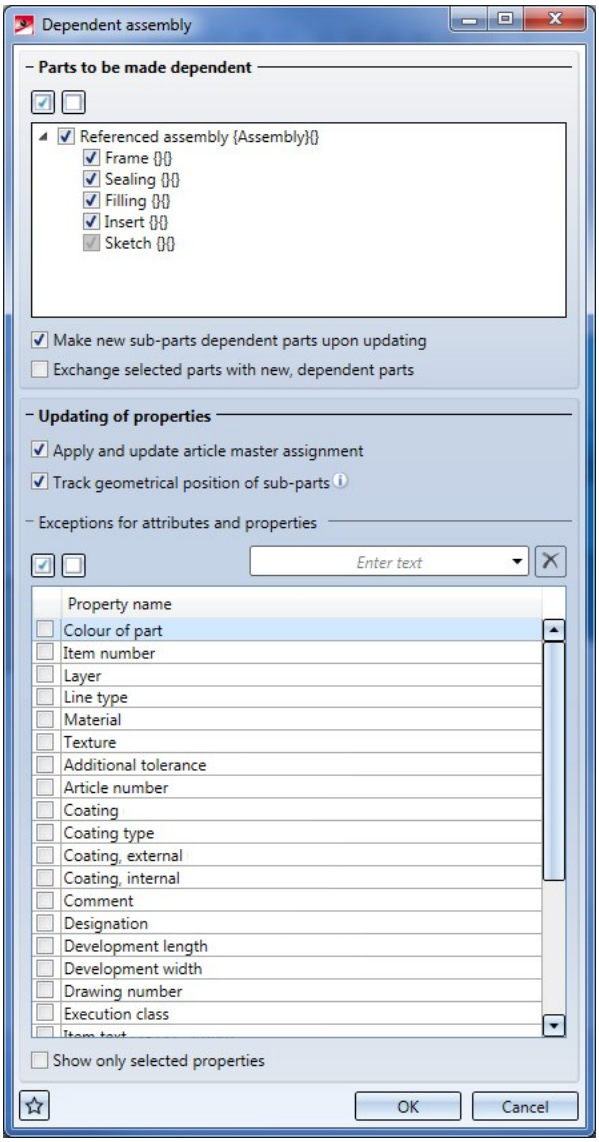
Here you need to specify two reference points and the desired maximum distance. If desired, you can specify an offset for the distance of the reference points. Specifying a number of clones is not necessary.

Based on the distance of the two reference points, as many clones as required will be created, with their distance to each other being smaller than the specified maximum distance.



Enhanced "Dependent assembly" function

The dialogue window of the **Dependent assembly** function has been revised and enhanced:



- You can now specify whether article master assignments are to be taken over into the derived assembly and updated.
- Beneath **Exceptions for attributes and properties** you can now search for specific properties via full text search, select or deselect all properties at once, save the currently active selection as a favourite and load it again later. Also, only the attribute names, but no longer the internal designations and data types are now displayed here.
- When changing nested dependent assemblies you can now choose whether the changed settings are also to be taken over into the contained dependent assemblies.
- Dependent assemblies are marked with the symbol in the ICN.
- A special handling of the transfer of item numbers has now been introduced.
- If desired, the dependent assembly can now be created in addition to the base assembly instead of replacing it.

- Dependent assemblies now offer the option to load the base part or (under certain conditions) exchange it with a different base part.
- Plant Engineering attributes will only be shown in the attributes list if the Plant Engineering module has been activated.
- The sheet thickness is dealt with separately: If the sheet thickness is calculated, the attribute will not be transferred; otherwise, it will always be transferred.

External references to facets

In previous versions, when using the processings **Add sweep, with translation**, **Subtract part, via translation** or **Trim to surface** (all on the 3-D Standard Ribbon tab) and specifying a surface (facet) as external reference the setting **Do not use external references** was ignored, i.e. external references were always considered.

This behaviour has been changed now: If one of the above processings contains an external reference and is recalculated, while external references are deactivated, the processing will keep the previous result.

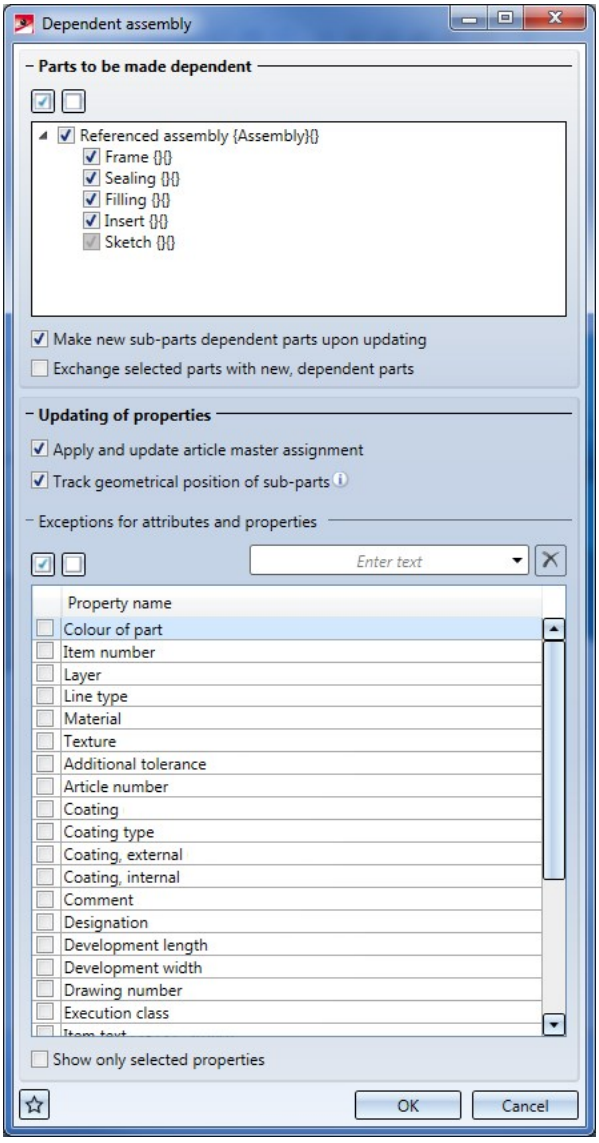
When opening old drawings, the facet parameter **External references in processings** will always be set to **Use** for affected processings. This happens to ensure that the behaviour known from older HiCAD versions will be preserved in these cases.

If you want to use the new behaviour in model drawings that were created with an older hiCAD version, you need to perform a feature recalculation.

Major Release 2019 (V 2400)

Dependent assemblies

In addition to the already existing **Dependent part** function, HiCAD now also offers the **Dependent assembly** function. You can use this function to replace a referenced assembly in the model drawing with a dependent assembly.



The difference between a dependent assembly and a referenced assembly is that the dependent assembly is only synchronized in one direction. That is, only changes to the original assembly will be taken over into the dependent assembly, and not the other way round. This can be useful if, for instance, you need to modify a part to enable its mounting, e.g. by applying bores to it. Referenced parts would also apply the bores in other model drawings, which is not the case when you use them in a dependent assembly. However, if the original drawing is changed, (e.g. by applying bores, but also ba adding or deleting parts), these changes will be taken over into the dependent assembly.

Parametrics (HCM)

Service Pack 2 2019 (V 2402)

Effect of HCM on assemblies

In previous versions, isolated points in assemblies were interpreted as movable points, so the HCM attempted to move the HCM assembly containing the HCM model as soon as such points occurred in constraints. This behaviour has now been changed.

The geometry of parts, in particular isolated points of assemblies, will be interpreted as "fixed" by a HCM model existing in a contained part. Constraints between these geometries and the part coordinate system of the contained part can no longer be set. Already existing constraints will, of course, be preserved.

However, constraints between the geometry and its sub-parts can still be set, which allows you to determine the position of the sub-parts.

Sketch HCM: Configuarble use of external references

In the Sketch HCM the constraints will now take into account whether the use of external references has been activated or deactivated. An external reference exists if a HCM constraint refers to a part that does not belong to this sketch.

In existing sketches the use of external references is active by default, in order to keep the existing behaviour of these sketches. Created new sketches behave according top the settings specified in the **Configuration Editor**: The value chosen at **System settings > Sketch HCM > Use external references** determines your default value. After new installations, external references are not used by default.

If you right-click on the empty area of the HCM window of the ICN, a context menu with the options **External reference: Use** or **... do not use**, respectively, will be displayed, which shows you the current setting for external references. After choosing this option, a dialogue window will be displayed, allowing you to choose between the settings **Use** and **Do not use**.

Point option M in new positional and dimensional constraints in sketches

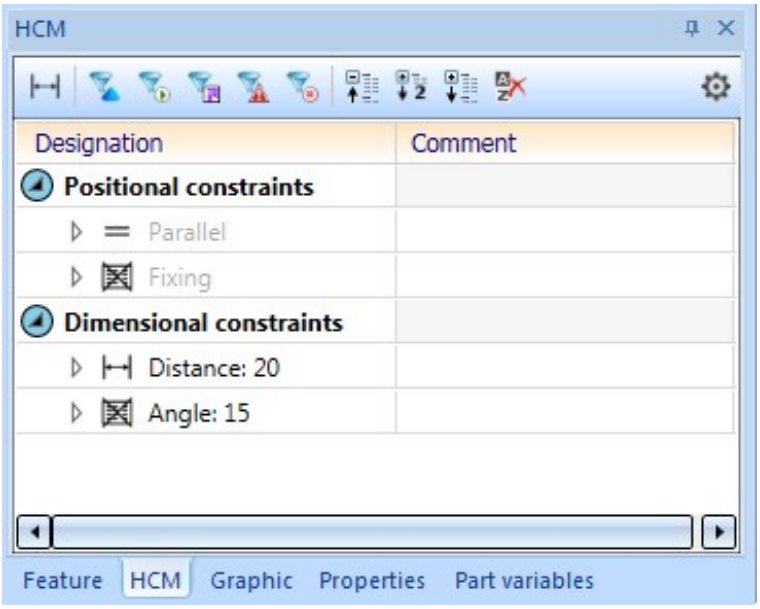
When defining new positional or dimensional constraints in sketches or 3-D sketches, the point option **(M) Midpoint of edge** will now no longer be suggested by the Autopilot, as this point option is only rarely used in practice.

However, it is still possible to select this point option manually.

Redesigned HCM window in the ICN

The HCM window for the display of constraints in sketches and 3-D assemblies in the ICN has had a redesign, now featuring a new, modern Look & Feel and a multi-column display.

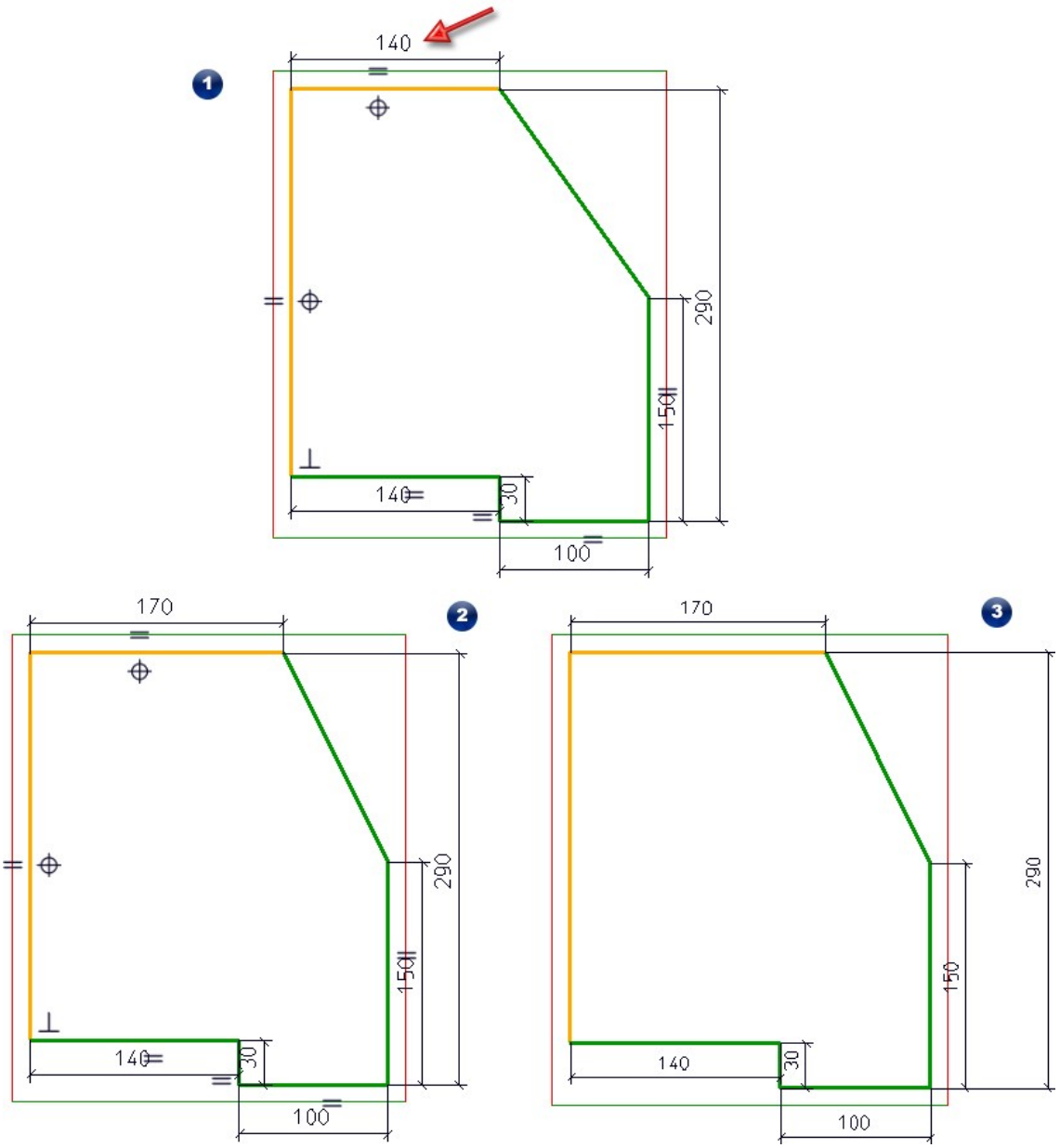
Also, you can now use filters such as **Deactivated constraints** or **Constraints with external references**, or the new function **Zoom to constraint**.



Placing of HCM dimensions in case of geometry changes

In older versions, HCM dimensions in sketches were placed with regard to their first base point. In case of geometry changes, the new dimension was placed in such a way that the length of the projection line of the first base point was identical to the length before the change, which frequently lead to unsatisfactory results. From SP2 onwards, the placement logic is different. The length of the shortest projection line after the change is now identical to the length of the shortest projection line before the change. In the process, cases may occur where the shortest projection line could be applied to different base points.

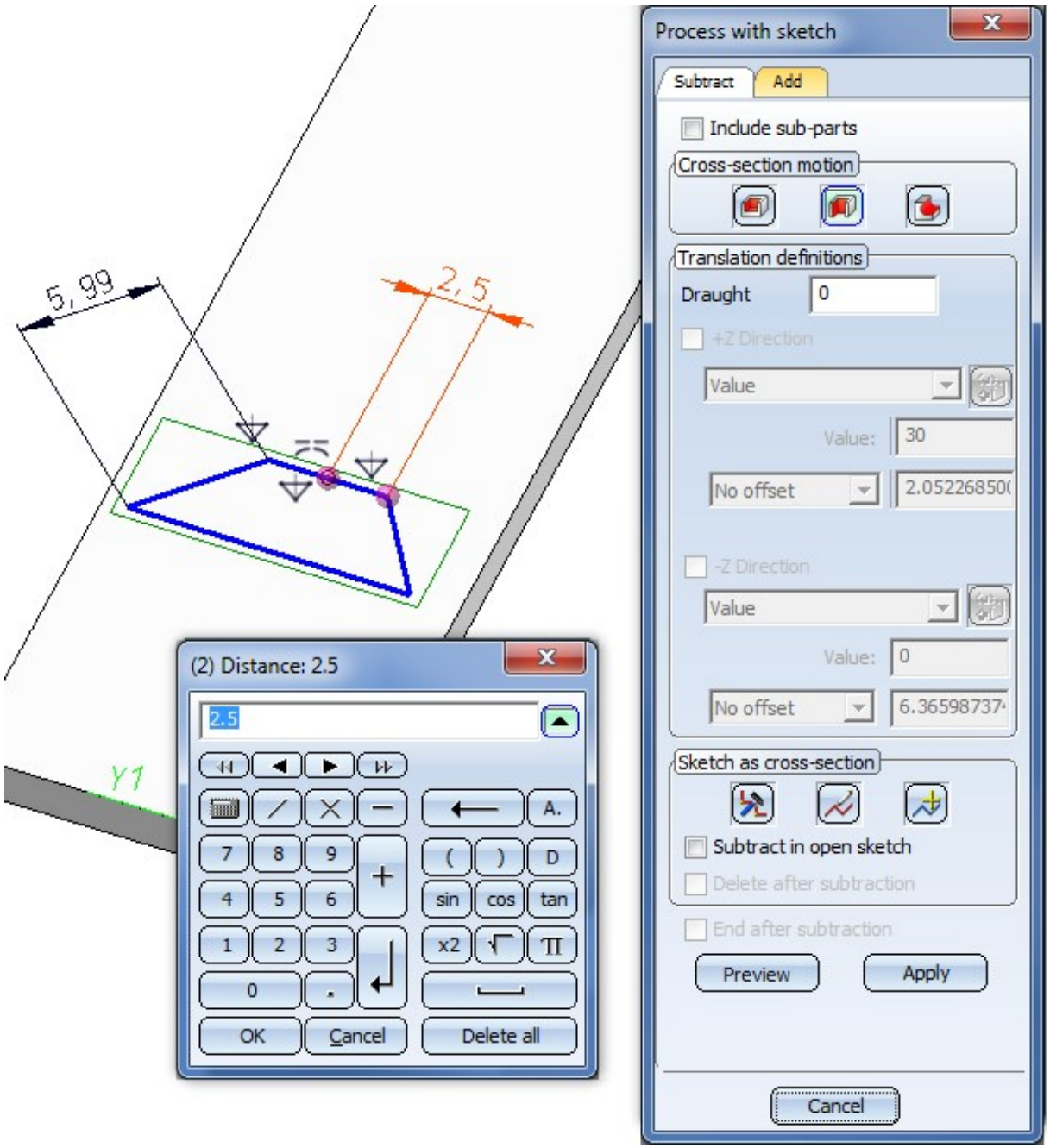
Example:



In the original sketch (1) the marked HCM dimension is changed from 140 to 170. (2) shows the result in HiCAD 2019 SP2, (3) shows the result in HiCAD 2018.

Changing HCM constraints while dialogue window is open

Youcan now change HCM constraints directly in the model drawing even while a dialogue window is open. This concerns, for example, the dialogues of the functions **Extruded solid**, **Revolved solid**, **C-edge sweep**, **Subtract** or **Add**.



Automatic assigning of HCM constraints for isolated points in sketches

If the HCM setting **Enter constraints** is active, positional constraints will be automatically set when creating isolated points in planar sketches or 3-D sketches. These depend on the chosen point option:

- **Snap point (I)** and **Centre (Z)**: **Coincidence**
- **Online to edge through point (O)** and **Online to edge through point (D)**: **Coincidence**with the relevant edge
- **Mid-point of an edge (M)**: **Mid-point** and **Coincidence**
- **Quad point (Q)**: **Equal distance** to X-and Y-axis and **Coincidence** with circle

Performance

- **Recalculations** in model drawings containing parts created with older HiCAD versions are now performed much faster: In tests we could measure a speed increase of more than 30%.
- The performance has also been increased for working with sketches with thousands of graphical elements while the HCM is activated.

Selection order for constraints

When creating a constraint that has not yet been fulfilled at the time of its creation, graphical elements or parts need to be transformed.

A good example for this would be a **Parallel** constraint between 2 non-parallel edges. In previous versions, one could not tell which of the two affected parts needs to be rotated in order to fulfil the constraint (except for cases where one of the parts has been determined by further constraints).

This behaviour has now been changed in such a way that the first selected element will not be transformed if possible.

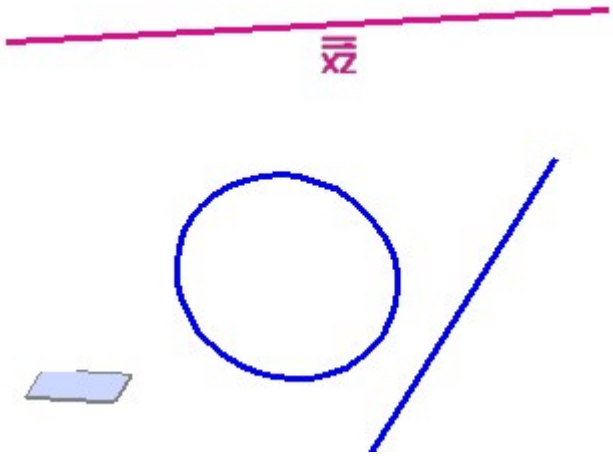
For **Symmetry** constraints, HiCAD will attempt to preserve first the symmetry axis, and then the first selected element.

For **Mid-point** constraints, the order of the queried elements has been changed: First, two elements, and then the mid-point will be requested. Accordingly, the mid-point instead of the selected elements.

Service Pack 1 2019 (V 2401)

Highlighting of HCM constraints

When you select a HCM constraint in the ICN, the sketch elements affected by it will be highlighted in your drawing. If the display of constraints has been deactivated, the concerning constraint will be displayed. If the constraint refers to an axis, plane or the origin of a coordinate system, these will also be displayed.

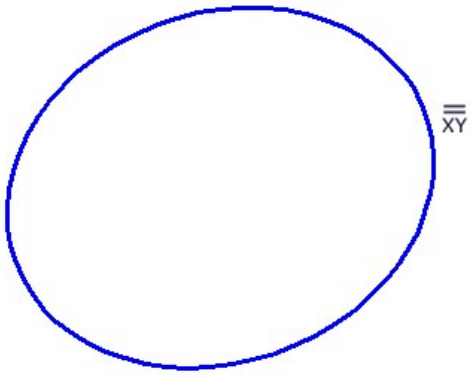


Example: The constraint "Parallel" between the line and the XZ-plane has been selected.

The highlighting will remain until you choose a different constraint or a different part.

Position of symbol for parallelism constraints of circles and ellipses

When the display of positional constraints for sketches has been activated, the symbol for parallelism of circles and ellipses placed parallel to a plane is no longer shown in the centre, but next to the circle/ellipse. This concerns both full circles/ellipses and arcs.



Excluded here are ellipses in planar sketches, if the parallelism refers to the main axis. Here, the symbol for the positional constraint will still be shown at the centre of the ellipse.

Editing of HCM constraints via "Process sketch" Feature in ICN

The editing of HCM constraints in sketches of a Feature step is now also possible if the editing of the sketch has been started by choosing **Edit sketch** for a Feature in the Feature log.

In previous versions this was only possible if the editing of the sketch was started by the opening of the dialogue window of the feature, and choosing **Sketch > Process sketch**.

HCM calculations on mirrored parts

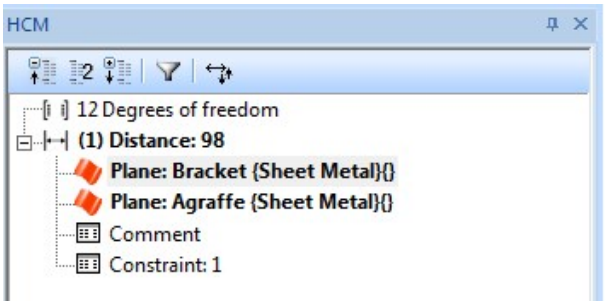
In previous versions it could happen in certain constellations that the HCM model reacted in an unexpected manner when perfoming mirrorings on one of its parts.

This behaviour has now be corrected, so that this problem no longer occurs.

Displayed references to Sheet Metal parts in the ICN

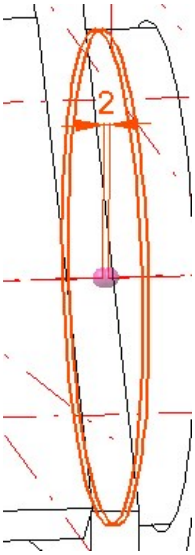
If you use HCM constraints in conjunction with Sheet Metal parts, references will often refer to flanges or bend zones. In older versions, the reference was often displayed as "Sheet flange" or "Bend zone" in the HCM window of the ICN.

Now, such reference are always displayed as "Sheet Metal", i.e. the entire sheet:



Marking colour for HCM dimensional constraints configurable

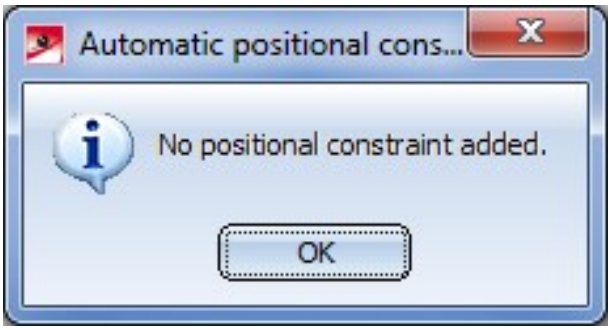
Selected HCM dimensional constraints are now highlighted in the drawing by the special colour **Marking 5** (default: Orange).



You can change this colour via the **Colour Editor** if desired.

Message displayed if functions were not applied

If the functions **Automatic positional constraint** or **Automatic dimensional constraint** have not created any constraints, an appropriate message will be displayed:



Rotatory degrees of freedom moved to mid point of graphical elements

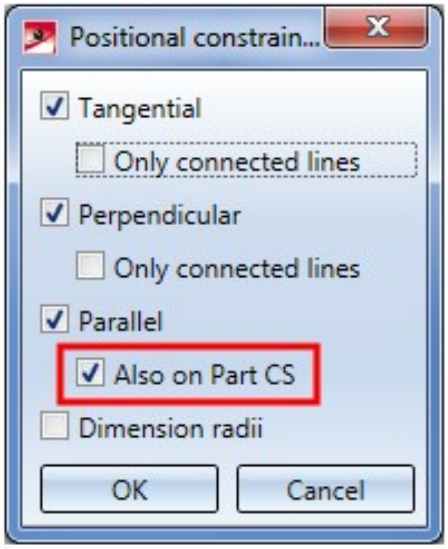
While the display of degrees of freedom in sketches is active, rotatory degrees of freedom will now always be shown on the mid point straight lines, circular arcs or elliptic arcs.

Performance

Through various optimizations the performance of HiCAD with regard to the loading of model drawings containing very many HCM constraints could be increased significantly.

Automatic positional constraints: Also on Part CS

The **Automatic positional constraints** function for planar sketches and 3-D sketches now contains, beneath the **Parallel** option, the additional **Also on Parts CS** option.



If this option has been activated, line elements running parallel to an axis (or, in 3-D sketches, also parallel to a plane) of the Part CS, will be assigned a **Parallel** constraint with regard to it. If you deactivate this option, HiCAD will show the same behaviour as previous versions, i.e. no parallelisms with regard to axes or planes of the Part CS will be considered, just parallelisms of lines.

Sketch HCM: Align to CS axis

The positional constraints for planar sketches and 3-D sketches were expanded by the new constraints **Align to CS axis**  . This constraint replaces the previous constraints **Align to X-/Y-/Z-axis**.

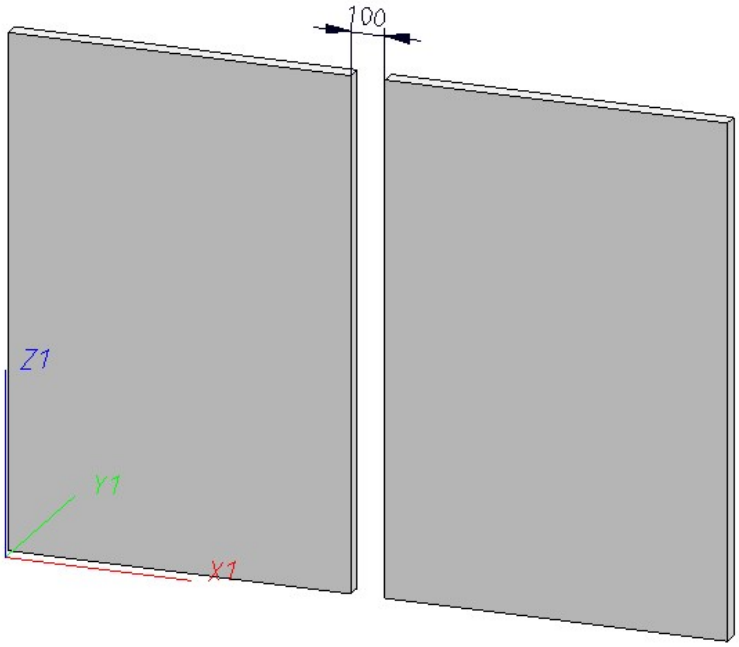
If you apply this positional constraint to a straight line of a sketch, HiCAD determines to which CS-axis the straight line has the smallest angle. A **Parallel** constraint will then be applied to this axis.

For planar sketches you can also apply this constraints to 2 points. In this case the angle between a virtual straight line running through these 2 points and the CS axes will be considered. An **Equal distance** constraint with regard to this CS axis will then be applied for the 2 points.

Preserve chirality

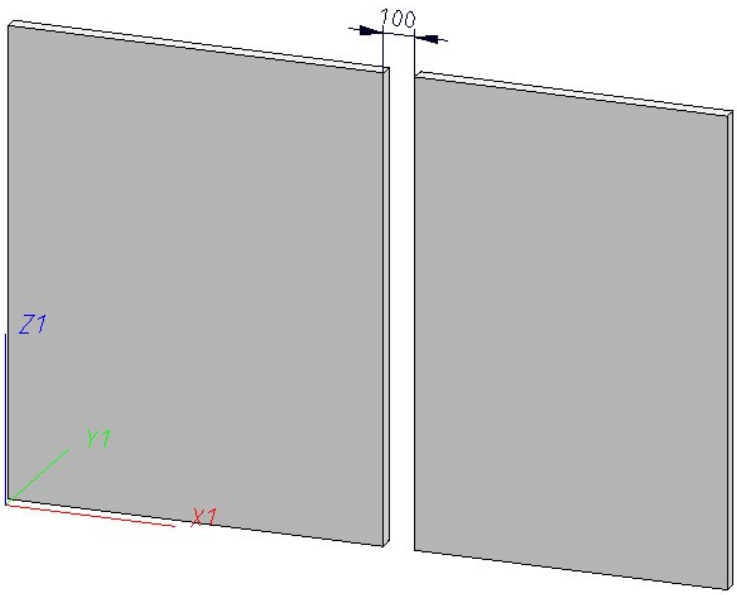
When changing dimensions, this could result in an unwanted switching of the chirality in previous HiCAD versions. In other words: While the positional and dimensional constraints were preserved, the direction of measurement could change unexpectedly.

This effect can be demonstrated by way of a very simple example of the two panels shown below:



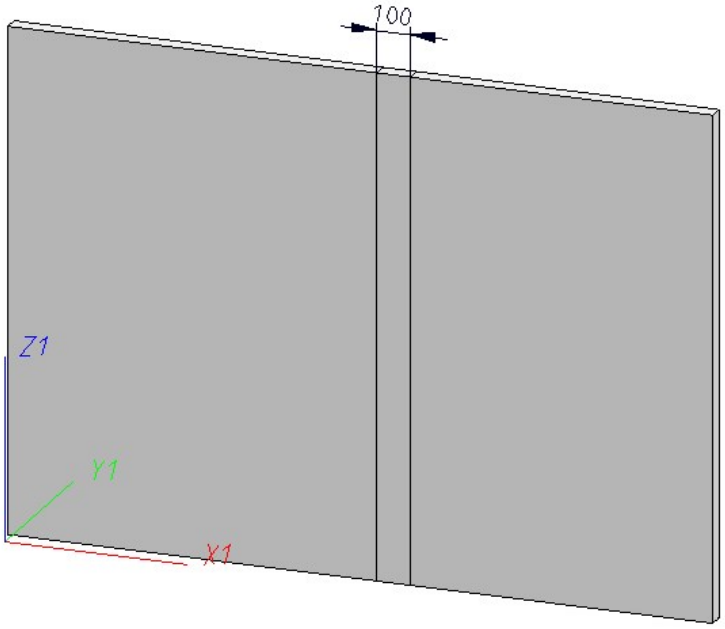
The left panel has been fixed. Both panels are kept on one plane by means of Coincidence constraints, and a Distance constraint defines a fixed distance between the edges on the borders of the 2 panels.

If you increased the width of the left panel by 200 mm in older HiCAD versions, the result looked as follows:



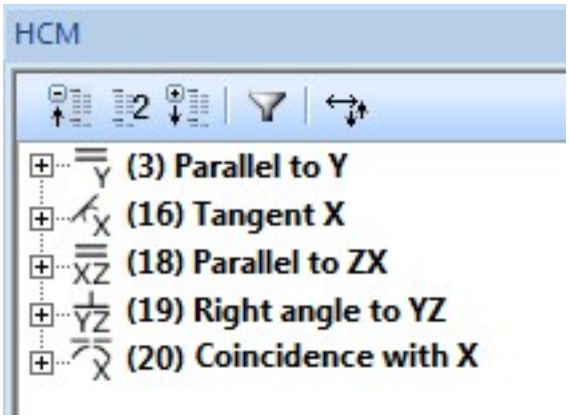
Formally speaking, this is still "correct" - the two edges still have a distance of 100 mm. However, this isn't usually the intended result.

From HiCAD 2401 onwards, it will be attempted to preserve the chirality after applying changes to the HCM system - here, the intended result looks as follows:

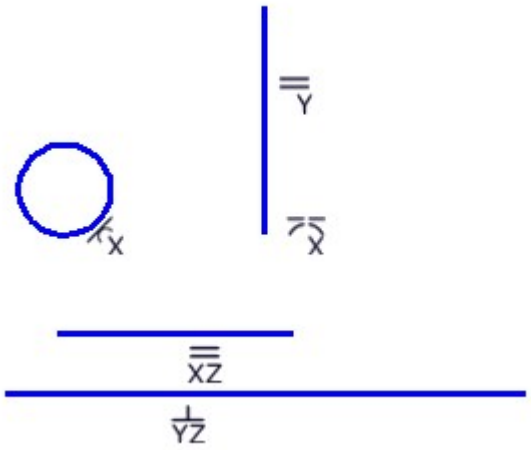


Display positional constraints with references to CS axes and planes in sketches

Positional constraints containing a reference to an axis or plane of the coordinate system will now be given particular attention: For instance, a parallelism constraint referring to the X-axis of the Sketch CS will not just be listed as **Parallel** in the ICN, but as **Parallel to X**.



Also, the symbols of the positional constraints will be displayed both in the ICN and (if the display of positional constraints in sketches has been activated) and in the sketch:



In this sketch you can see immediately that in this 3-D sketch the circle, for instance, has been fixed to the X-axis via a tangential constraint, and the lower line runs perpendicular to the YZ-plane.

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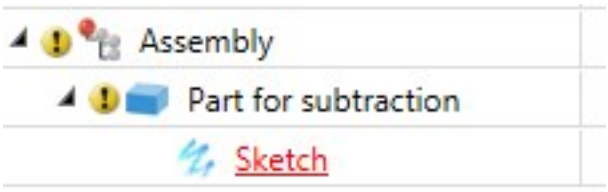
Selective automatic assigning of constraints in the sketch HCM

If the **Enter constraints**setting has been activated in the Sketch HCM, no distinguishing was made between "normal" constraints and constraints with external references in previous versions. To avoid errors in the design it may help in some cases not to work with constraints with external references. In this case the automatic assigning of constraints during drawing of sketches could not be used.

HiCAD 2019 offers the new option **Use external references**:

If this checkbox is deactivated, constraints that would contain external references will no longer be automatically created. This option is deactivated by default, which would be identical to the behaviour of HiCAD before Version 2019.



References to immediately superordinate parts or assemblies are not regarded as external references and will therefore be created even if the **Use external references** checkbox has been deactivated. An example of this is a sketch for a subtraction: If this sketch is located immediately beneath the part to which the subtraction is to be applied, it can still automatically obtain distance constraints to the edge of the part, e.g. when using the **Offset** function.

Switching of coordinate system during processing of 3-D sketches

When assigning constraints to a 3-D sketch, the Sketch HCM has always referred to the Part CS of the 3-D sketch. However, the active coordinate system was still displayed in the process, which could be rather confusing.

This has been changed now, so that the Part CS will automatically be displayed on a 3-D sketch when you choose a function that normally uses this type of coordinate system. After applying the changes, the display will switch back to the previously used coordinate system.

The behaviour of the Part HCM and the constraints has not changed; this change refers only to the display of the coordinate system during assigning of constraints.

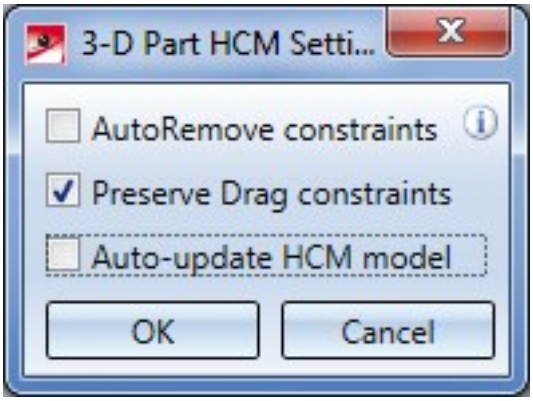
Consistent display of parametric dimensions

In previous versions the configurations available via the ParConfigComp.exe tool contained different configurations for the display HCM constraints and symbols. These inconsistencies have now been removed, so that the same settings are now used in every configuration:

- **Font size for HCM constraints**: 2.5
- **Size of symbols for positional constraints**: 20 pixel
- **Decimal places for other dimensions**: 2

Automatic updating

You have now the option to automatically update HCM models of superordinate assemblies when you apply changes to a part. This function can be activated or deactivated in the 3-D Part HCM Settings dialogue window.



After new installations this function is activated by default; after update installations, in contrast, it will be deactivated in order to keep the behaviour known from previous versions.

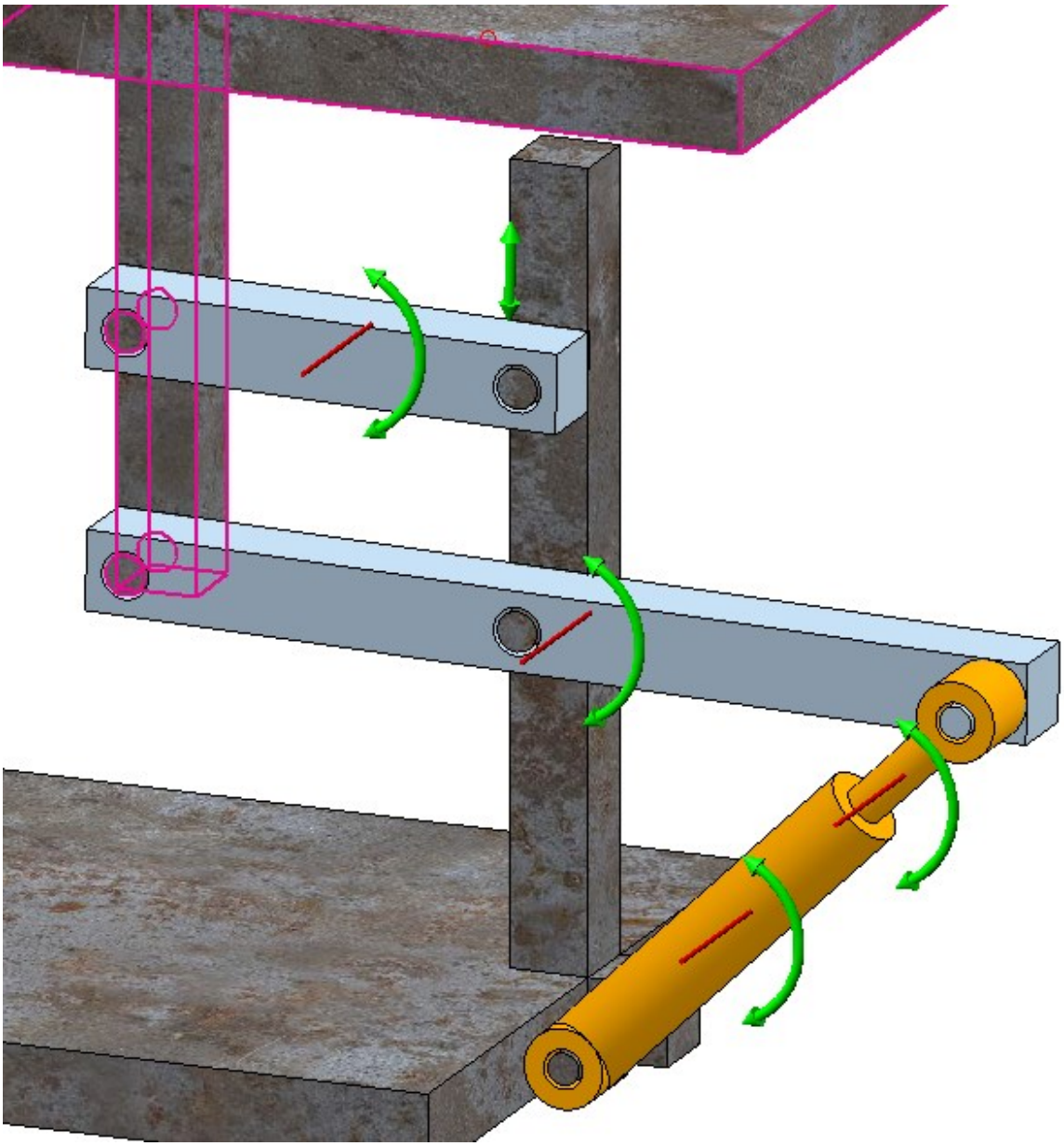
Improved UNDO in sketches with HCM model

When using the UNDO function on sketches, HiCAD no longer distinguishes between sketches with and without HCM constraints, i.e. a step-by-step UNDO is now also possible for sketches with automatic HCM constraints. The number of UNDO steps is unlimited.

This applies to both planar sketches and 3-D sketches.

Display of degrees of freedom in model drawings

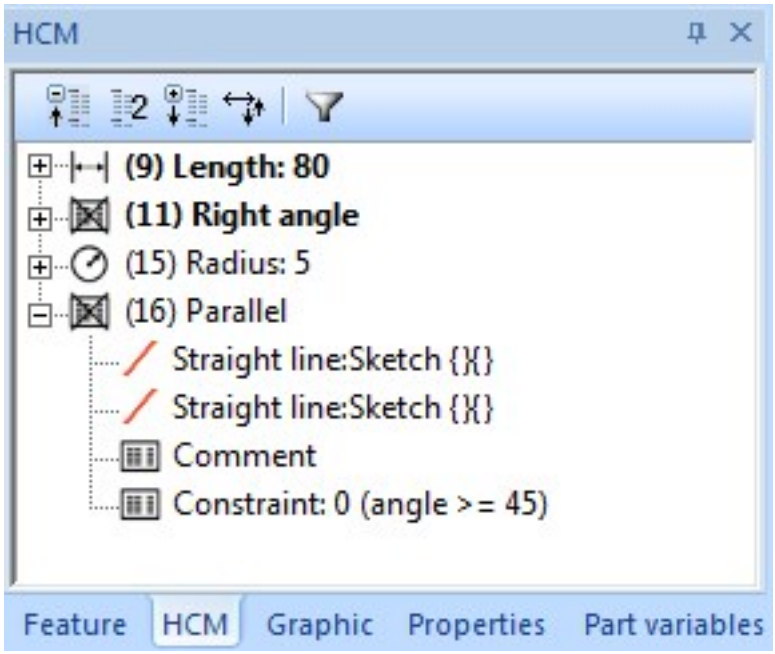
The 3-D Part HCM and the Sketch HCM now offer the new **Toggle visibility of degrees of freedom** function. This function allows you to switch the display of the degrees of freedom in the current HCM system in a drawing on or off.



If the display has been activated, arrows indicate how many and which axes are still available for possible movements and rotations.

Extensive descriptions of these functions and the meaning of the displayed arrows can be found in the relevant Help chapters for the Sketch HCM and the 3-D Part HCM.

Revised display of HCM constraints in the ICN



The HCM window in the ICN has been revised with particular focus on more user-friendliness. The changes include:

- New symbols for constraints.
- Geometries are shown in the same order in which they are shown in the part structure.
- Parts that are fixed or fully defined are marked with a corresponding symbol in the listing.
- Entries like "Condition fulfilled" or "Fully defined" are no longer explicitly listed.
- Constraints can be deleted via the DEL button. The shortcut CTRL+A selects all constraints.
- The toolbar now contains a button that allows you to switch between the "Sort by geometry" and "Sort by constraints" mode.
- Constraints with external references are indicated by a red "e" on their icon.
- The indication of errors in the HCM and in the Feature log have been standardized.
- You can now fix parts and assemblies via context menu function.

The changes for sketches are described here.

Positional constraint "Equal distance" also available for 3-D sketches and assemblies

The positional constraint  **Equal distance** can now be applied not only to planar sketches, but to 3-D sketches as well.

The **Equal distance** constraint is now also available in the 3-D Part HCM.

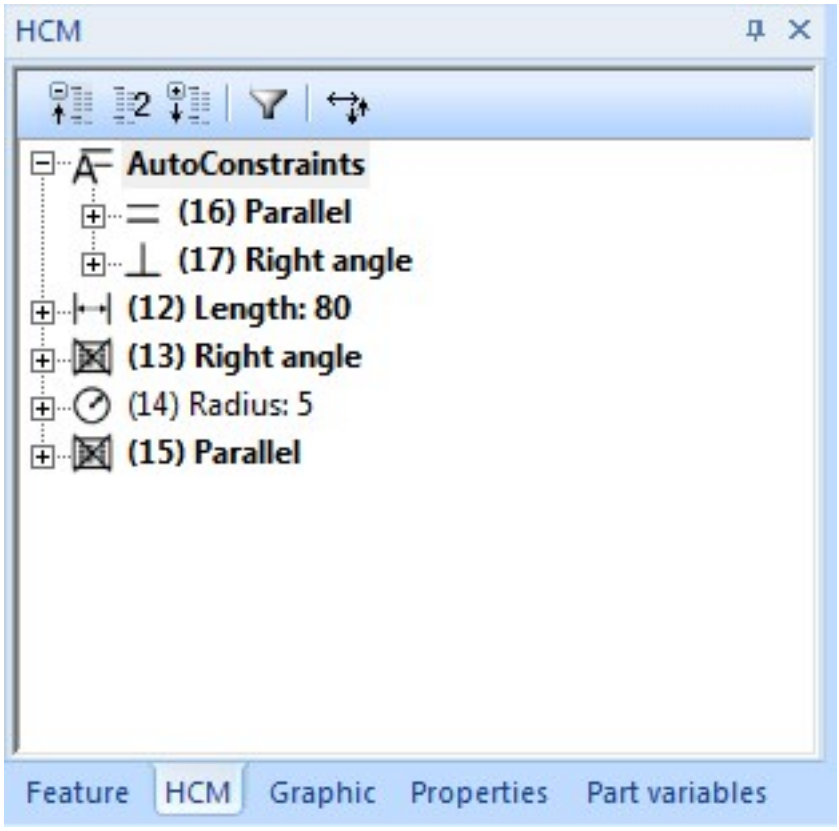
Redesigned HCM function groups

The structure of the Ribbon tabs has been optimized, so that the most frequently needed HCM functions are now directly available and need no longer be called from sub-menus.

You have now direct access to functions for dimensional constraints in X-, Y- and Z-direction (with the latter, of course, being not available for 2-D parts), as well as to Coincidence, Parallelism, Right angle and Fixing constraints.

Display of AutoConstraints in ICN

Positional constraints that have been created via the **Automatic positional constraints** function can now be found beneath an item called **AutoConstraints**. If desired, you can move these constraints out of this item via Drag & Drop , or choose the context menu function **Remove from AutoConstraints** to move the constraint out of the **AutoConstraints** item and list it among the "normal" constraints instead.

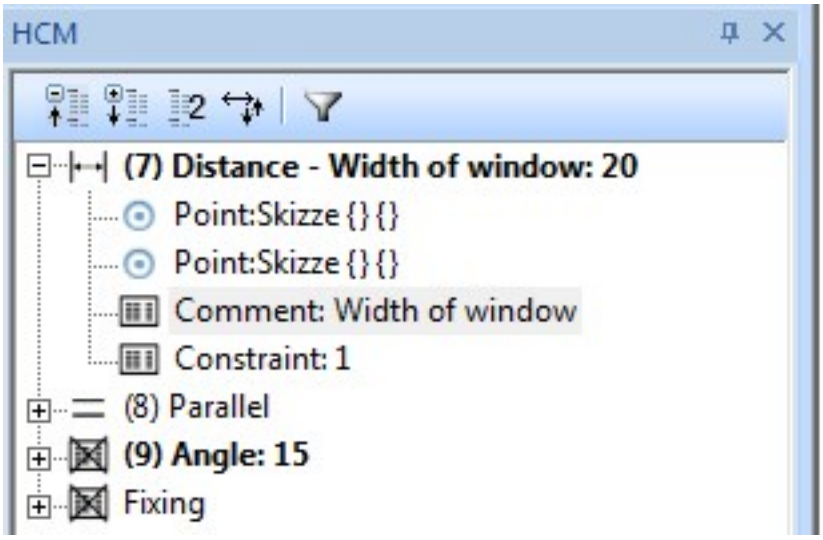


Positional constraints for free-form curves removed

The positional constraints for free-form curves, i.e. **Equal parameters**, **Equal direction**, **Equal 1st/2nd derivation** and **Equal bend** have been removed.

Comments for HCM constraints

You have now the option to specify Comments for HCM constraints in composite edges and in 3-D assemblies. These have no influence on the model; their purpose is to describe the meaning of individual or all constraints, especially in larger model drawings.



On the **HCM** tab of the **ICN**, expand the individual constraints and double-click the **Comment** entry. You can then specify a text for the Comment, which will then also be displayed after the designation of the constraint.

To delete a comment text again, double-click the **Comment** again and remove the text in the dialogue window and confirm with **OK**.

Configuration Management

Service Pack 2 2019 (V 2402)

Settings from ABWPAR.DAT and ABWCOL.DAT moved to Configuration Editor

The settings from the system files ABWPAR.DAT and ABWCOL.DAT have been moved to the Configuration Editor.

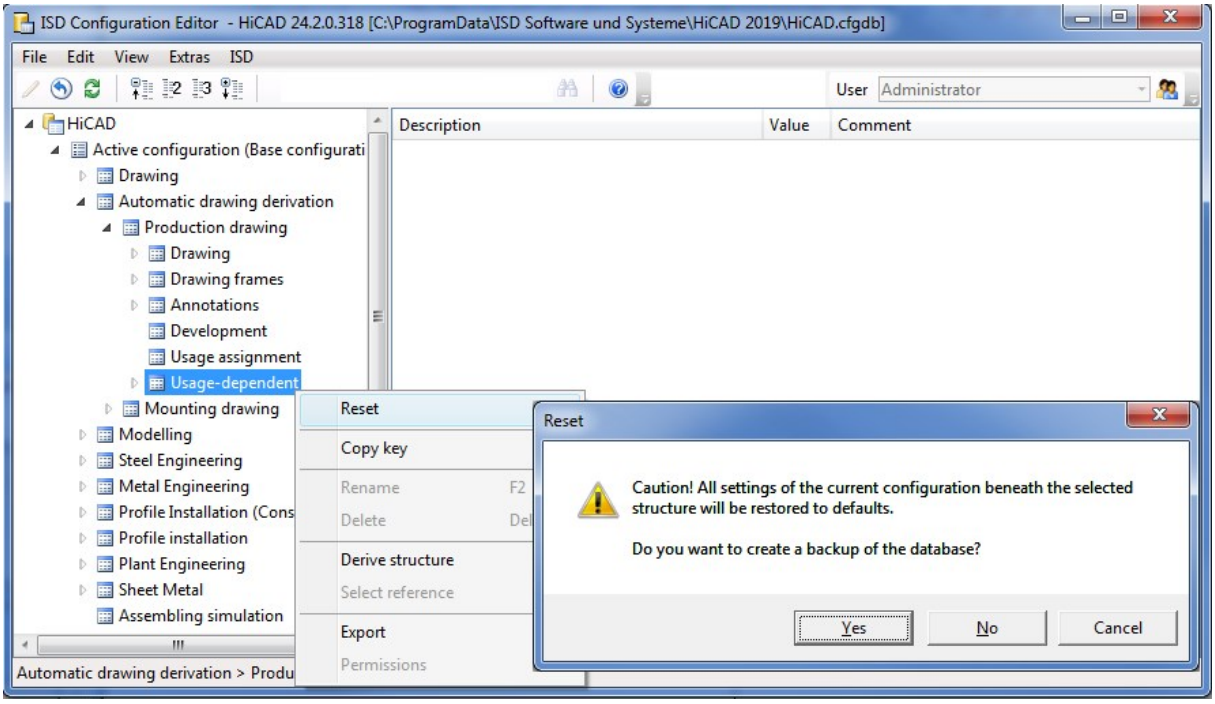
You can now find the settings from those DAT files in the following directories:

ABWPAR.DAT	Sheet Metal > Sheet development
	Sheet Metal > Default setting
	Compatibility > Sheet development up to HiCAD 2016
ABWCOL.DAT	Compatibility > Sheet development up to HiCAD 2016 > Extended settings

Reset all values in a node

In the Configuration Editor you have now the option to reset changed values for a defined node within the tree structure in one step.

For instance, if you have modified the default settings of the dimensioning rules in various directories, you can right-click the **Usage-dependent** entry, choose **Reset**, and, after confirming the security prompt with, restore the modified settings to defaults again. If you confirm with **Yes**, a backup ([Installation directory] > Configuration > HiCAD.cfgdb.[Date]) will be created, in case you may require the modified settings later again. If you choose **No**, the values will be reset without creating a backup.



Service Pack 1 2019 (V 2401)

User-friendlier catalogue selection option

The entries for **Material** selection at **Steel Engineering > Default material > Table ID for default material** and **Item ID for default material** have been replaced by a more convenient catalogue selection option.

Sorting of the ICN

The settings for the sorting of the ICN entries are no longer relevant for the new 2-D and 3-D ICN. The corresponding setting options have therefore be removed from the Configuration Editor.

All entries can now be directly sorted in the new ICN.

Major Release 2019 (V 2400)

Settings of TABPAR.DAT moved to Configuration Editor

The settings from the TABPAR.DAT file for a working with a tablet computer as input device and the specification of the time interval for double-clicks have been moved to the Configuration Editor (ISDConfigEditor.exe). The file has been withdrawn with the release of HiCAD 2019, since tablet computers as input devices are no longer supported. The Time interval for double click parameter can now be specified in the Configuration Editor, namely, at **System settings > Identification**.

Bill of Materials / Report Manager

Service Pack 2 2019 (V 2402)

New parameters for section schemas in BOMs

In the Configuration Editor at Steel Engineering, two new parameters for section schemas in BOMs are now available:

Cutting angle reference in section schema

Here you can choose to which leg a specified angle in the section schema is to refer:

- Cross-section or
- Beam axis

Do not show cutting angle in section schema if cut surface

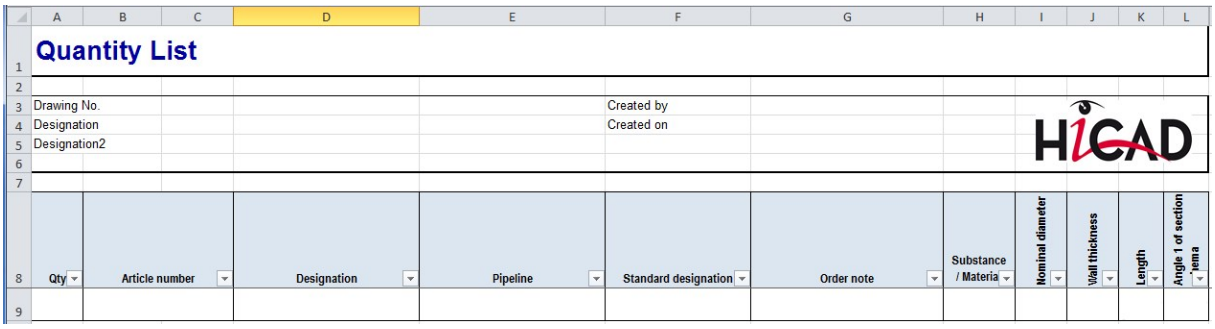
Use this parameter to specify the conditions under which no cutting angle is to be shown in the section schema, since the total length does not correspond to the trimmed length:

- The cut surface has been processed.
- The cut surface has been processed on one outer side or on both outer sides.

Service Pack 1 2019 (V 2401)

Plant Engineering BOMs

In Plant Engineering BOMs you can now find the new **Pipeline** (Attribute: **%PipeLineName**) column that indicates the pipeline to which the respective part belongs. The default template for plant Engineering BOMs, **Anlagenbau_ohne_DB_SZN.rms** (HiCAD sys directory) and the corresponding EXCEL template in the BOMTemplates folder have been adjusted accordingly.

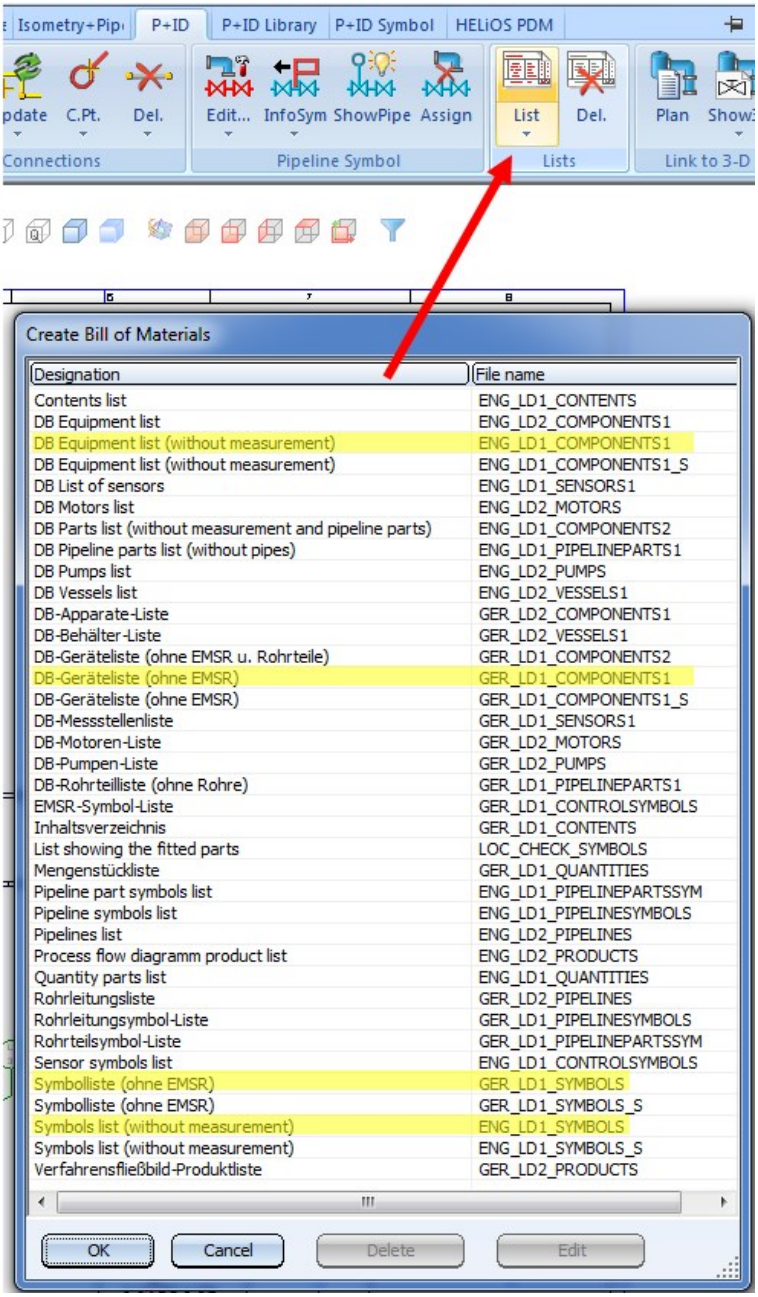


	A	B	C	D	E	F	G	H	I	J	K	L
1	**Quantity List**											
2												
3	Drawing No.					Created by						
4	Designation					Created on						
5	Designation2											
6												
7												
8	Qty	Article number		Designation	Pipeline	Standard designation	Order note	Substance / Materia	Nominal diameter	Wall thickness	Length	Angle 1 of section tema
9												

Major Release 2019 (V 2400)

Further Excel templates for P+ID BOMs

More templates for BOMs based on P+ID projects are now available.



Loose parts and bar lists in BOMs via product structure

You have now the option to configure the behaviour of Excel BOMs created on the basis of HELiOS product structures with regard to so-called "loose parts", i.e. parts that belong to no assembly: You can either show these parts as individual parts, or combine them into one set. You have also the option to change the name of such a set.

	A	B
1	**Script file**	PRODUCT_ISD.cs
2	**Export raw data**	false
3	**Language**	en
4	**Combine loose parts**	true
5		

Also, the new Report Manager template PRODUCT_ISD_HiCAD has been added to the existing template PRODUCT_ISD. The new template contains table sheets for bar lists and bar list summaries.

QOMET-specific BOMs

HiCAD 2019 supports the output of COMET-specific BOMs. QOMET is an ERP system developed specifically for use in the field of Steel and Metal Engineering.

The BOMs can either be output for the entire drawing or only for the active part, and have special columns in the Excel Steel Engineering BOM (hicad_Stahlbau.xlsc) that are relevant to the QOMET system. The BOMs will be output in the CSV format.

The function for the output of these BOMs can be found at **Drawing > Save/Print > Save as.. > Further...**:

also see HiCAD Basics - What's New?

Pre-mounted assemblies in BOMs

When working with pre-mounted assemblies in practice, you sometimes want only the assemblies, but not their sub-parts to be shown in BOMs. This can be achieved via the article attribute COMPONENT_PREINSTALLED, which needs to be assigned to the article master of the pre-mounted assembly.

also see HiCAD Management + BIM - What's New?

Variant Editor

Service Pack 1 2019 (V 2401)

Changing the part type ID during automatic deriving of variants

From HiCAD 2019 SP1 onwards, you can also change the part type ID, and thus also the classification of a VAA and a PAA file when automatically deriving variants.

The attribute assignment file (attribute_assignment.csv) supports the pseudo attribute **VAREDIT_KEEP_DB_IDS** for this purpose.

If this attribute has the value 1, the previous HELiOS IDs will be preserved during automatic deriving. i.e. no "actual" deriving takes place. From a HELiOS perspective, the resulting VAA or PAA file refers to the same parts. This is helpful for the adjustment of existing VAAs or PAAS, especially in conjunction with VAREDIT_MOVE_ON_SUCCESS.

Also, it is now possible to map the attribute ARTSCHLUESSEL (Type key) to a column in the file customer_list.csv. In this column the part type ID of the VAA or PAA file must then exist in the format AAAAA_BBBBBBB, with AAAAA being the coding for the industry according to **anbtlken_top.dat** and BBBBBBB the ID according to anbtlken.dat.

If the format should not be correct, the message

Classification not in the format AAAAA_BBBBBBB, with A = Industry and B = ID

will be issued in the output column.

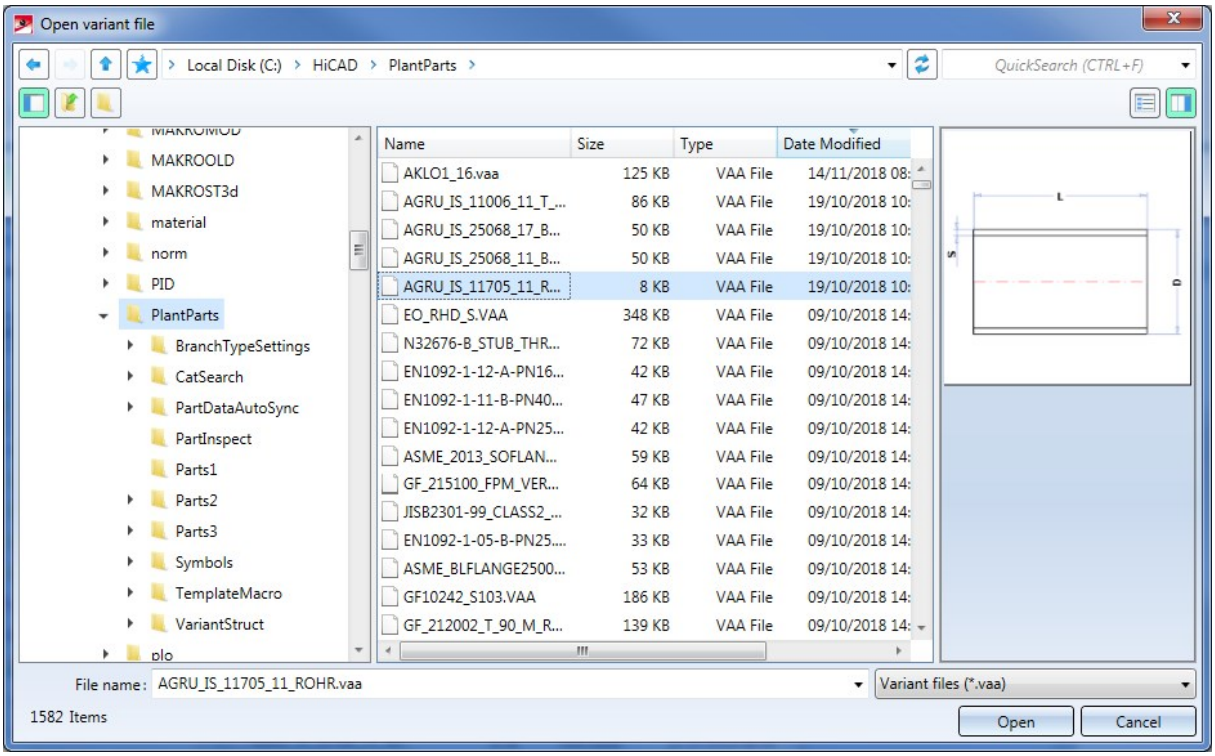
Major Release 2019 (V 2400)

VariantenEditor.exe - Revised file dialogue windows

The file dialogue windows of the Variant Editor for VAA files have been adjusted to Windows 10. This concerns the following Editor functions:

- Derive new,
- Open,
- Import,
- Export and
- Select graphic preview.

The relevant dialogue windows now corresponds to the file dialogue windows in HiCAD.

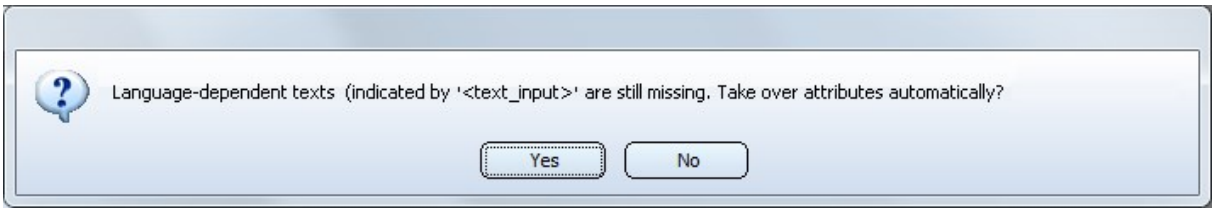


A detailed description of the **Open variant file** dialogue window can be found in the Dialogue for Opening and Saving of Files topic of the HiCAD **Basics** Help.

Attribute assignments - Take over attributes for all languages

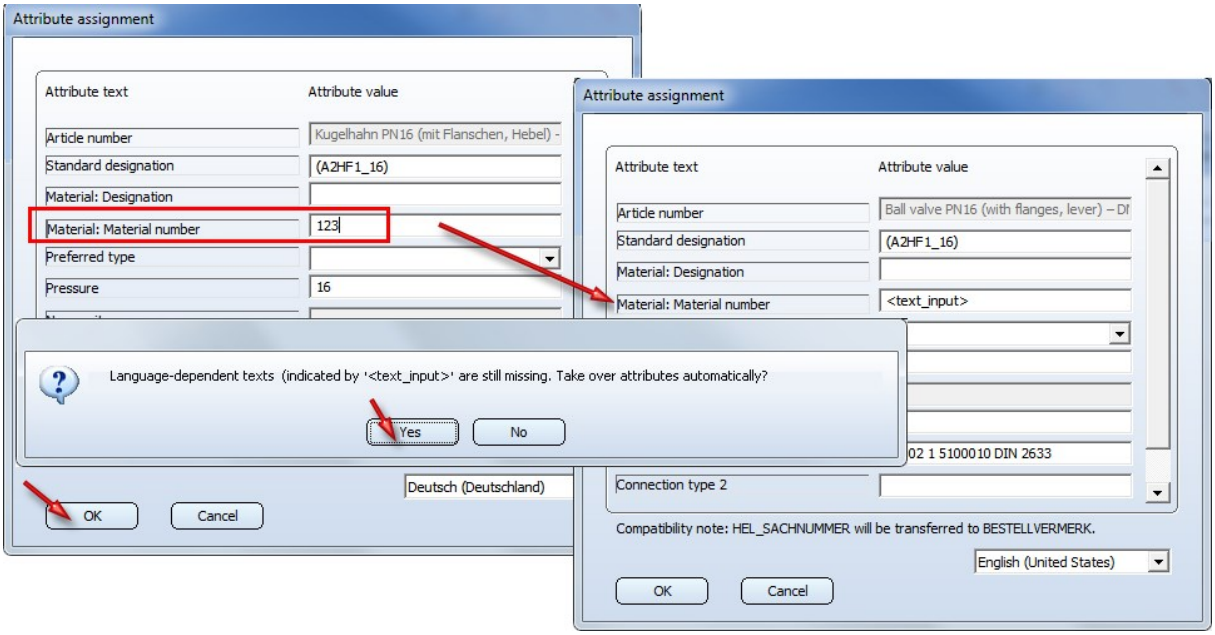
When you changed attribute assignments via the **A** button and confirmed with **OK** without adjusting the attribute in all languages in HiCAD 2018 (or older versions), the Variant Editor issues an appropriate message and entered the text **<text_input>** into the corresponding fields.

From HiCAD 2019 onwards, changed attribute assignments can be automatically be taken over to the other languages. If any attributes are missing, the following message will be displayed:



If you choose **Yes**, the attribute from the first language, in which has a value, will be transferred to the corresponding fields of the other language, if these are empty; otherwise, the current content of the field will remain unchanged.

If you choose **No**, the text **<text_input>** will be entered in the corresponding fields of the other languages.



Use of existing attribute masters of variants with varying sub-types

In cases where Plant Engineering variants with existing HELiOS article masters had to be used for the deriving of variants with varying sub-types, HiCAD 2018 only provided the possibility to enter, for each sub-type, a matching HELIOS URL directly during part data synchronization. This process is not only very tiresome, but also requires the necessary permission to create article masters in HELiOS at all.

The latter, in particular, may prove problematic in configurations in which HELiOS obtains the article masters "downstream" by other ERP systems. To provide help in such cases, the mechanism for automatic deriving has been enhanced in HiCAD 2019. The Head IDs can now also be directly provided in the **customer_list.csv** (=customerlist.csv) file.

For this purpose the new pseudo attributes

- VAREDIT_SUB_ID
- VAREDIT_GENERAL_ID and
- VAREDIT_MOVE_ON_SUCCESS

are available.

An example:

The following CSV file describes two VAAs, each with three sub-types. The sub-types are identified by their Type_ID here , i.e. through the HELiOS attribute **HEL_TYPNAME**. For each sub-type, one Head ID is provided for the general type, and one Head ID for the sub-type itself.

VAA name	Derived	Type_ID	Customer_GENERAL_ID	Customer_SUB_ID
Pipe	Pipe_derived	001	B0YHQHXIS0LVFL00003J7M	B0YHQHXIS0LVFL00003J7P
Pipe	Pipe_derived	002	B0YHQHXIS0LVFL00003J7M	B0YHQHXIS0LVFL00003J7Q
Pipe	Pipe_derived	003	B0YHQHXIS0LVFL00003J7M	B0YHQHXIS0LVFL00003J7R
Elbow	Elbow_derived	001	B0YHQHXIS0LVFL00003J7N	B0YHQHXIS0LVFL00003J7S
Elbow	Elbow_derived	002	B0YHQHXIS0LVFL00003J7N	B0YHQHXIS0LVFL00003J7T
Elbow	Elbow_derived	003	B0YHQHXIS0LVFL00003J7N	B0YHQHXIS0LVFL00003J7U

In the attribute mapping file **attributzuordnung.csv** (=attributemapping.csv) these data can be mapped to the VAA attributes as follows:

```
VAREDIT_VERBATIM_MARKER;#;
VAREDIT_BASE_VAA;VAAName;
VAREDIT_DERIVED_VAA;Derived;
VAREDIT_OVERWRITE_EXISTING_VAA;#1;
VAREDIT_LANGUAGE_CODE;#1031;
VAREDIT_BASE_DIR;#Path\to\PlantParts;
VAREDIT_DERIVED_DIR;#Path\for\derived\VAAs;
VAREDIT_STOP_ON_UTO_ERROR;#0;
VAREDIT_LST_FILE_NAME;#AutoDerived;
VAREDIT_OUTPUT_FORMAT;#SYLK;
VAREDIT_CLUSTERING_KEY;VAAName
VAREDIT_SUBTYPE_KEY;HEL_TYPNAME
HEL_TYPNAME;Typ_ID
VAREDIT_SUB_ID;Customer_SUB_ID
VAREDIT_GENERAL_ID;Customer_GENERAL_ID
```

The pseudo attribute attributes **VAREDIT_SUB_ID** and **VAREDIT_GENERAL_ID** ensure that the Head IDs from the **customer_list.csv** file are transferred into the VAAs.

The pseudo attribute **VAREDIT_MOVE_ON_SUCCESS** ensures that, after successful generation of all files, these files will be moved from DERIVED_DIR to BASE_DIR. For files that already exist in BASE_DIR, a backup with the file extension .bak will be created.

To transfer the VAA files prepared in this way to HELiOS, the tool **DBPlantDataImport.exe** must be called with the parameter **/X**. This allows a Plant Engineering compliant adjustment of existing article masters.

- It is assumed that document masters may be created. These are created for each VAA.
- The links between the article masters and to the document masters will be added.
- Article attributes will be overwritten. The values for attributes that are not relevant for Plant Engineering will remain unchanged.

The **DBPlantDataImport.exe** tool allows the removal of already transferred variants. When DBPlant-DataImport.exe is called with the parameter /X, only the document master will be removed. Article masters will remain unchanged.

Interfaces

Service Pack 2 (V 2402)

IFC Import/Export - ZIP files

HiCAD 2019 SP2 supports the import and export of zipped IFC files (file format .IFCZIP).

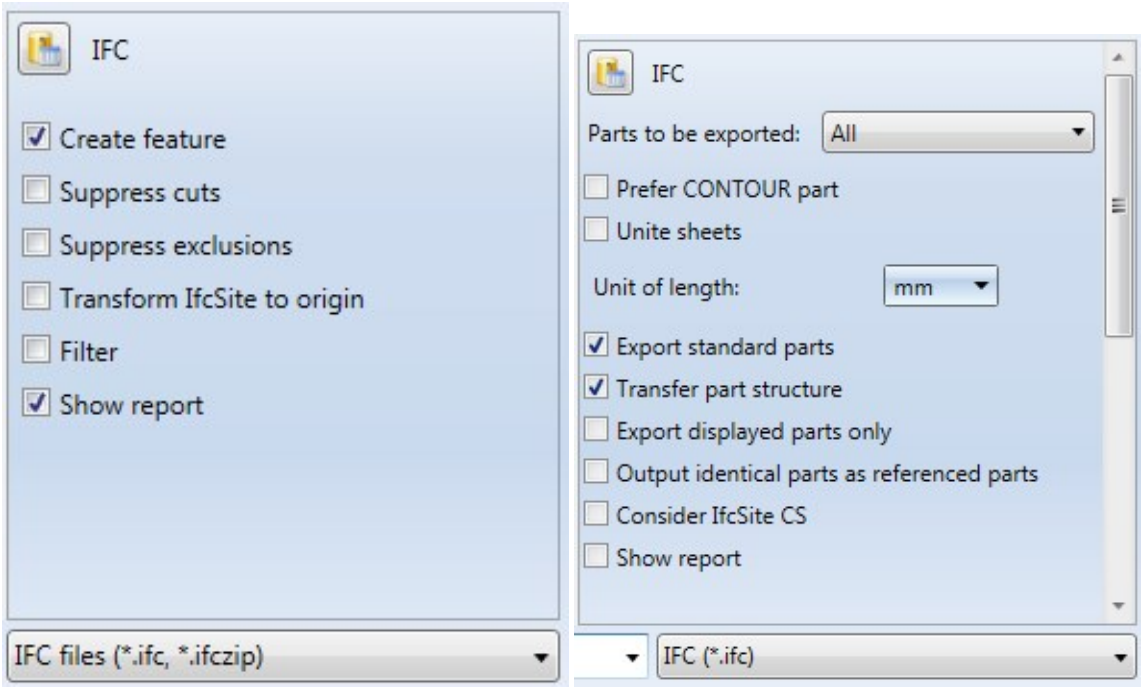
IFC Import/Export - Consider coordinate system

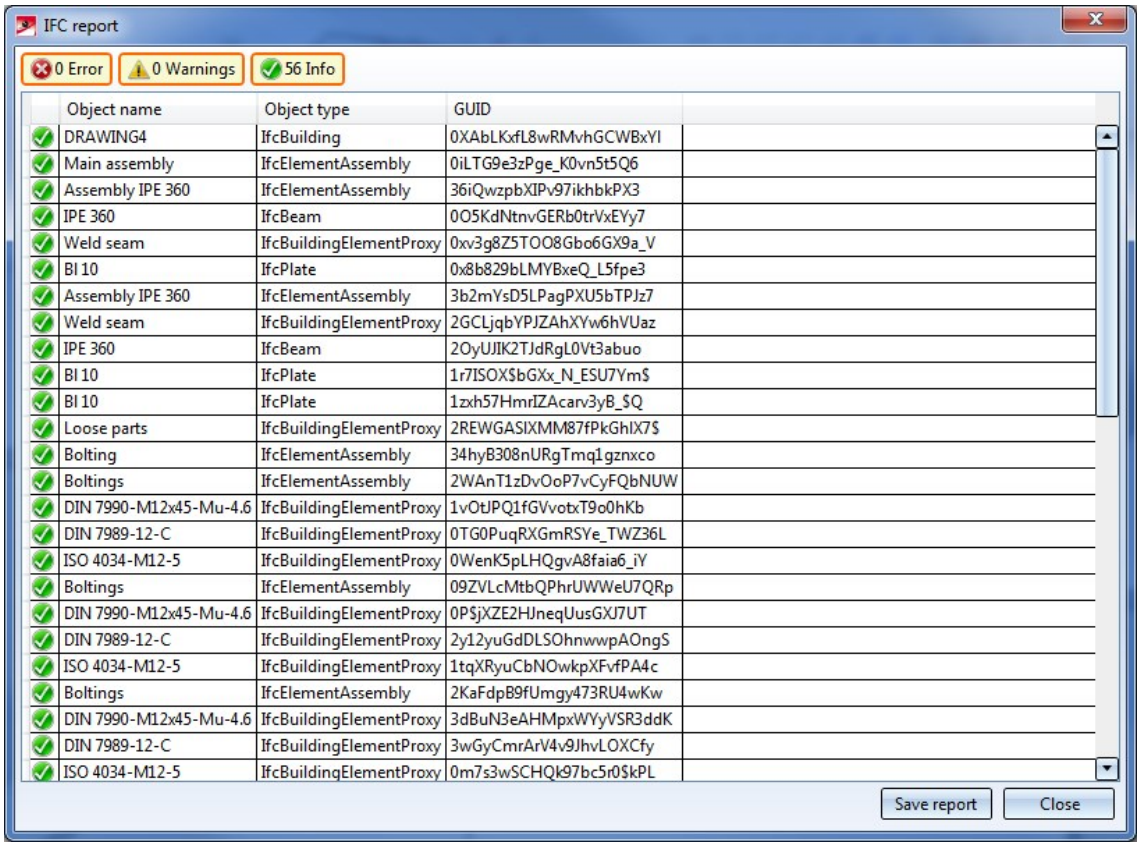
IFC data are often located far away from the origin of the coordinate system. However, HiCAD and other CAD systems require coordinates that are close to the origin. The necessary coordinate system transformation can now be realized for both import and export. For this purpose a corresponding checkbox has been added to the dialogue windows:

- **Import - Transform IfcSite to origin**
  If this checkbox has been activated, **IfcSite** (top element incl. sub-elements) will be transformed to the origin. The original coordinate system will be saves as Fitting CS with the name **IfcSite** in the Feature of the IfcSite assembly.
- **Export - Consider IfcSite CS**
  If this checkbox has been activated, the data model will be transformed to the Fitting CS with the name **IfcSite** saved in the Feature of the IfcSite assembly (top assembly), if any.

IFC Import/Export - Show report

For the import and export of IFC files a report listing possible errors, warnings and other information can now be displayed. For this purpose a corresponding checkbox has been added to the dialogue windows.





Click on an entry in the report to view further information on an object. Click on the **Save report** button to save the report as CSV file.

SketchUp export

The export of HiCAD drawing to the SketchUp format (.skp) could be speeded up significantly in HiCAD 2019 SP2. In test examples the time required for exports could be reduced from several hours to less than one minute.

DSTV-NC export to CAM format

For the export of the DSTV-NC data to the CAM format, an API-based solution in conjunction with the existing DSTV-NC interface is currently available.

This solution is currently only based on a script and has not been made generally available yet. After a successful test phase the CAM export is to be made a standard functionality. If you would like to take a look at the script in advance, please contact our Consulting team.

The configuration of the attribute output takes place directly in the API script. The DSTV-NC export parameters, too, can be set in the script. The script works as follows:

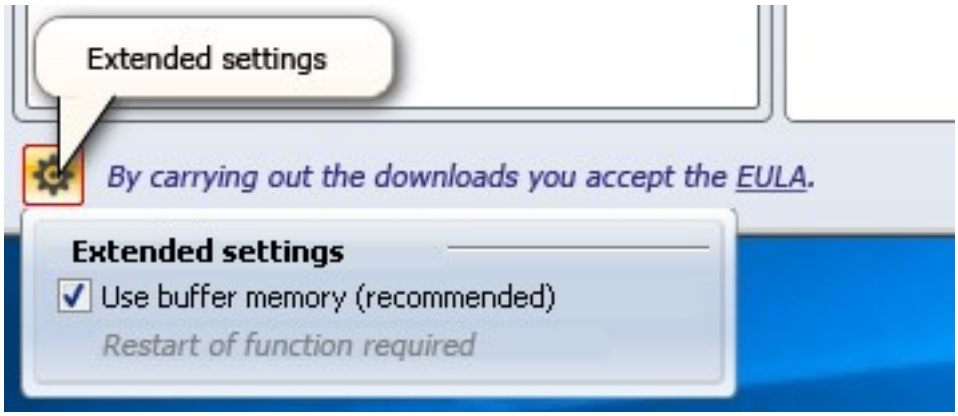
- Creation of the DSTV-NC files
- Creation of the header data (attributes and part structure)
- Merging of files into one CAM file

Service Pack 1 2019 (V 2401)

Cadenas PartSolutions and BIMCatalogs

The insertion dialogues that you can use directly on Cadenas Parts4cad or BIMcatalogs.net have been revised and improved.

The corresponding windows always show the current status of the catalogues on the external server.



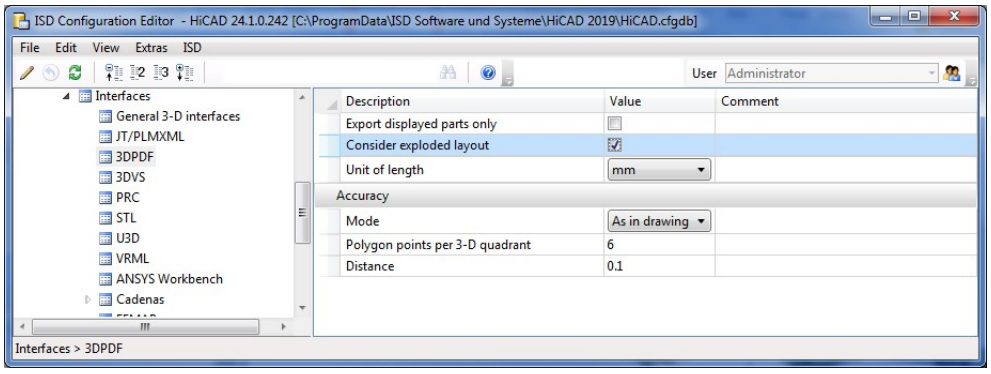
With an additional option under  **Extended settings** at the bottom of the dialogue window, you can also specify whether you want direct real-time access to the Web service or local intermediate storage in your HiCAD system (recommended).

3D PDF, Universal 3D, Product Representation Compact - Export of exploded layouts

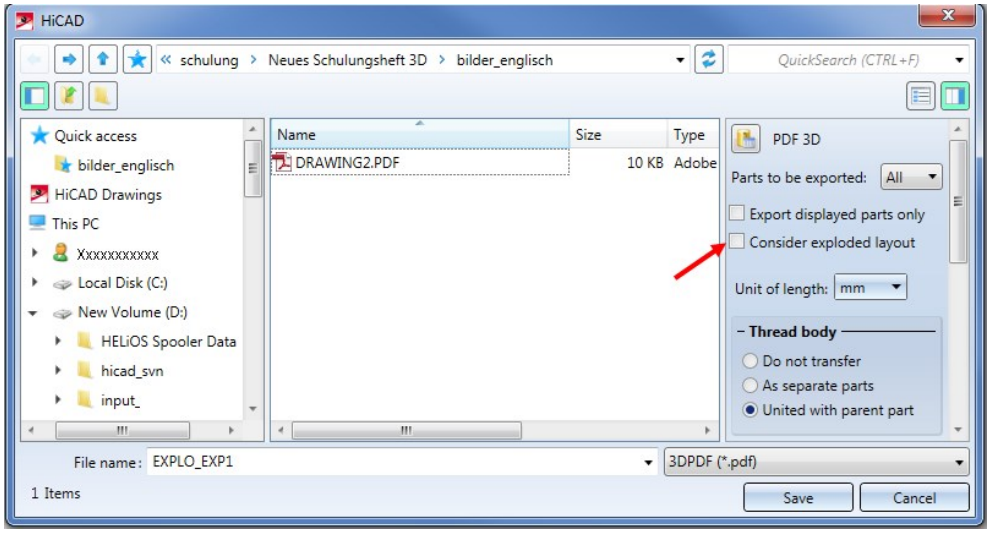
Exploded layouts, too, can now be considered for export as

- 3D-PDF (*.PDF)
- Universal 3D (*.U3D) and
- Product Representation Compact (*.PRC)

files. The Configuration Editor and the export dialogue have been expanded accordingly for this purpose. In the Configuration Editor at **Interfaces** the **Consider exploded layouts** checkbox is available for the above formats.



The same checkbox is available in the corresponding export dialogue window:



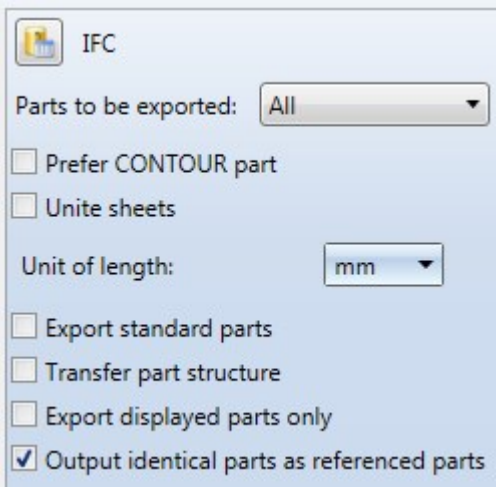
The parts will be shown in an exploded layout. Mounting tracks, too, will be exported, their colour will be taken from the model drawing. Line type and layer, however, will not be considered.

Performance for IFC export

A performance increase by a factor of up to 5 could be achieved for IFC import and export.

Furthermore, a downsizing of exported files by a factor of up to 8 compared to older HiCAD versions could be achieved.

For drawings with many identical parts you can use the option **Output identical parts as referenced parts** when exporting, which combines the same parts as referencing and thus reduces the amount of data.



STL

The insertion of STL files in an active HiCAD Drawing via **Drawing > Insert Part >**  **STL Format** has been adapted to the current algorithm and functionality of the import dialogue for STL files.

At this point, you can also use the **Separate parts** option to insert separate parts instead of a coherent solid primitive.

In addition to an increase in performance, it is also possible to insert binary STL files into the drawing.

Major Release 2019 (V 2400)

Update to CADfix 12

With the update to CADfix 12 the following format versions will now be available to HiCAD:

- CATIA V5 2018
- NX 12
- SolidWorks 2018
- Creo 5.0
- JT v9.0

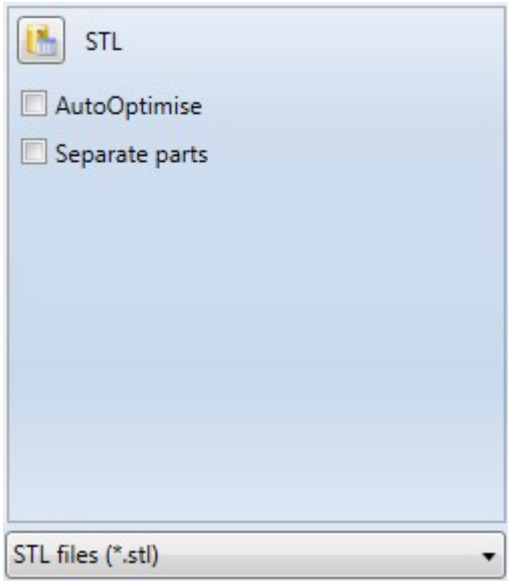
STL Import

Previously, ASCII based STL files have been applied via **Drawing > Insert Part >** **STL Format** in HiCAD.

As of HiCAD 2019 you are able to read ASCII based as well as binary STL files via the import dialogue window **Drawing > New/Open > Open >** **Step, Iges**

Select **STL files (*.stl)** as format.

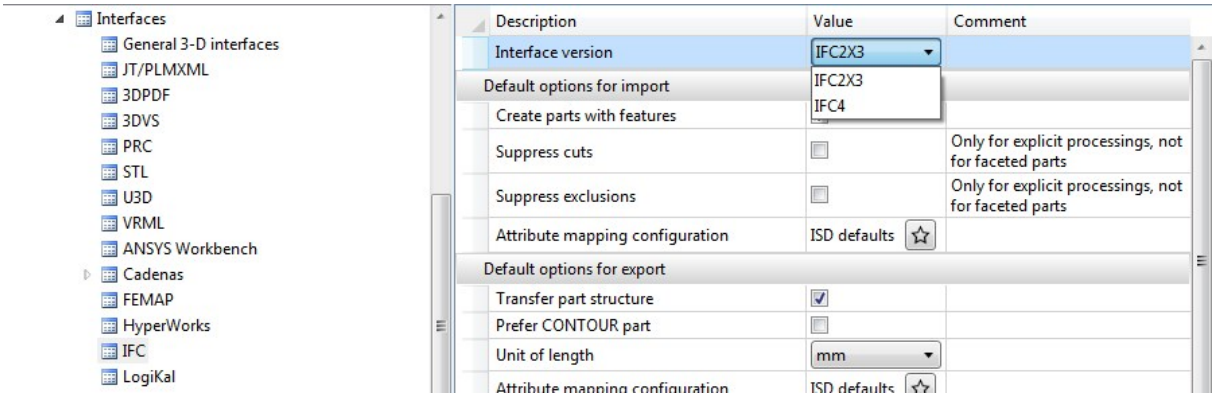
Further import options are available by clicking on the respective checkbox to either activate **AutoOptimise** or insert **Separate parts** instead of coherent solids.



IFC

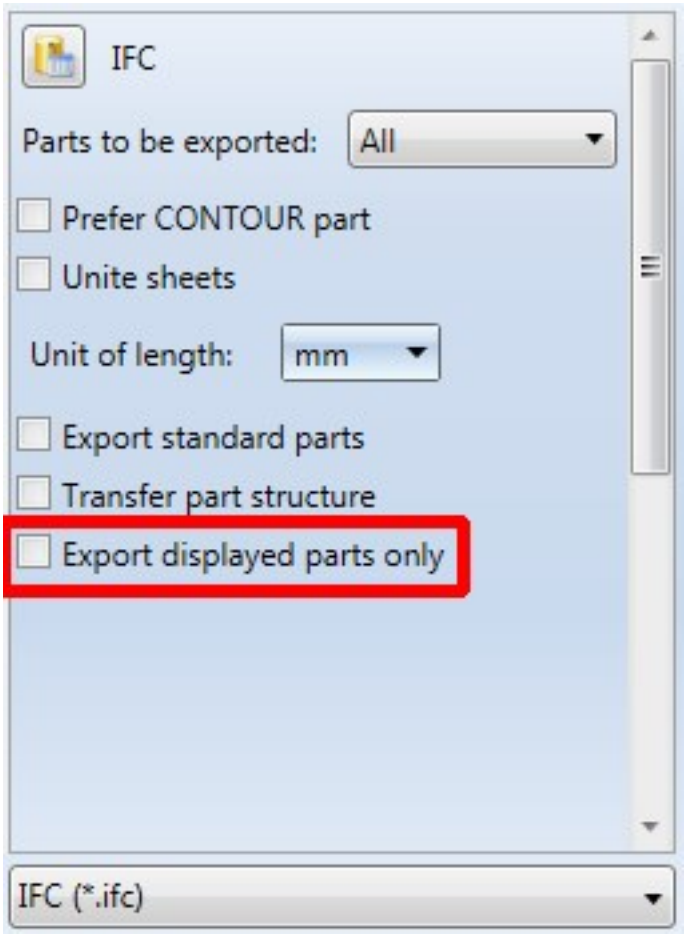
HiCAD 2019 supports the IFC4 format for import and export.

The IFC format version (IFC2x3 or IFC4) can be chosen in the Configuration Editor at **Interfaces > IFC > Interface version**



Furthermore, the performance of the IFC export has improved significantly.

The export in IFC format has been enhanced with the option to export displayed parts only instead of the whole drawing.

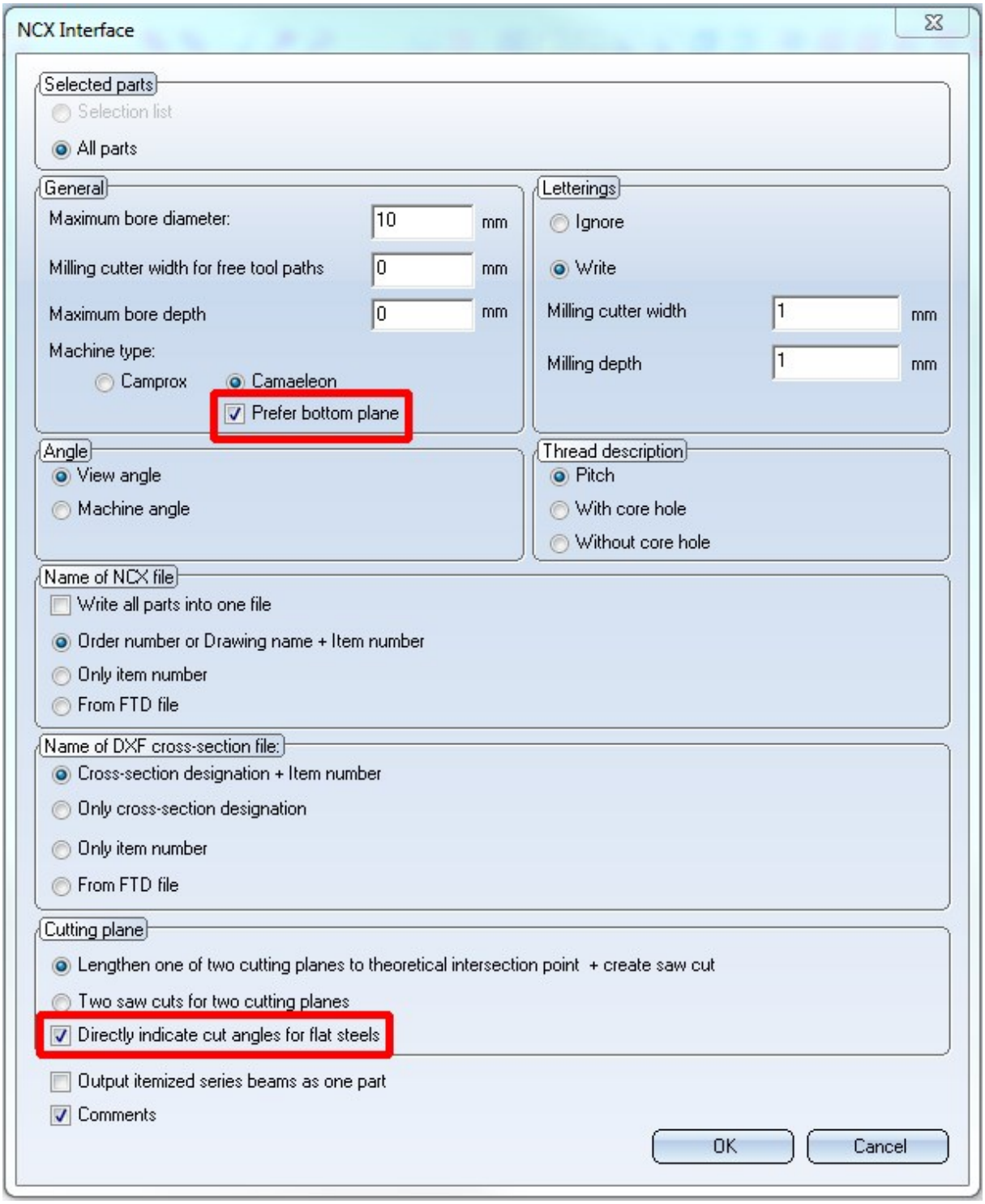


NCX Export of saw cuts

The NCX Export has been enhanced with two options for the optimum handling of saw cuts:

In order to avoid undesirable outcome, it is possible to prefer the bottom plane when exporting by activating the checkbox

It is also possible to export cut angles for flat sheets as such. (In previous versions, cut angles have been exported in 90° and the deviating real cut angle has been realized as saw cut afterwards)



Save 2-D part in DWG format

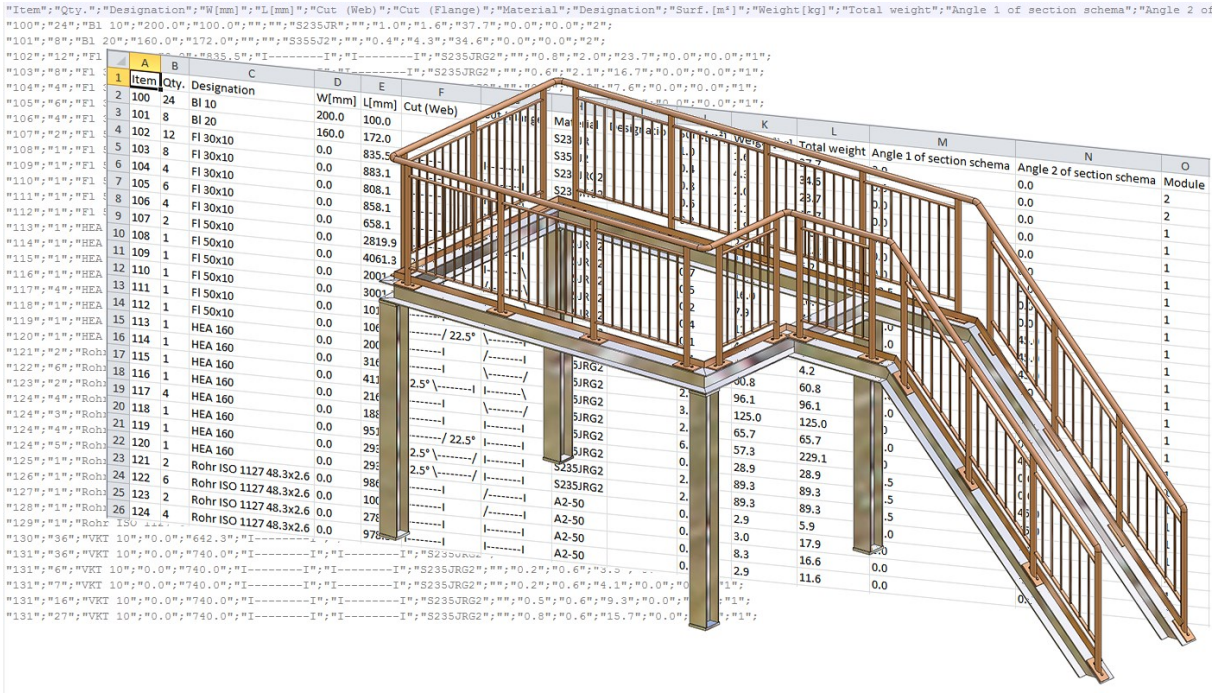
Apart from saving in DXF format, you can save 2-D parts in DWG format by clicking on **Drawing > Save/Print > Save as... > 2-D part DXF/DWG** and select **AutoCAD (*.dwg)** in the following dialogue window.

QOMET-specific BOM

HiCAD 2019 supports the export of QOMET-specific BOM. QOMET is an ERP system specialised in steel and metal engineering.

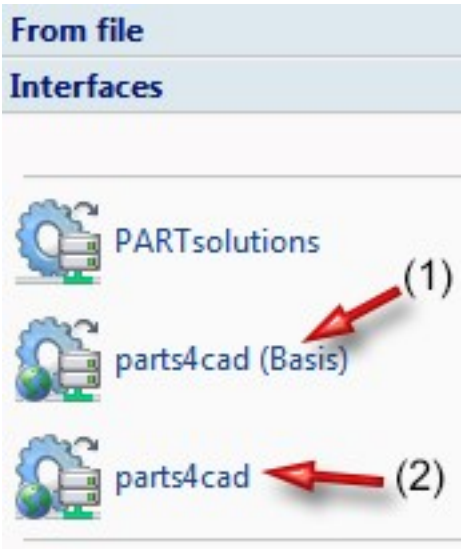
The BOM can either be exported for the complete drawing or active part and contains particular columns of the Steel Engineering Excel BOM (hicad_Stahlbau.xlsc) that are relevant for the QOMET system in particular. The file is exported in the CSV format.

The respective functions for exporting the BOM are available under **Drawing > Save/Print > Save as...** > **Further > ...** .



Free general access to PARTS4CAD

As of HiCAD 2019 you are granted with free access to more than 500 manufacturer's catalogues via the web catalogue parts4cad by Cadenas. The access to the extended catalogue range (DIN/ISO/EN… Norms + all 800 manufacturer's catalogues) is fee-based.



Export of Sheet Metal parts

The functions for data exchange available at **Sheet Metal > Sheet development** have been optimized in HiCAD 2019. The following functions are available here:

Export individual development

Develop and export sheets

Export sheet developments

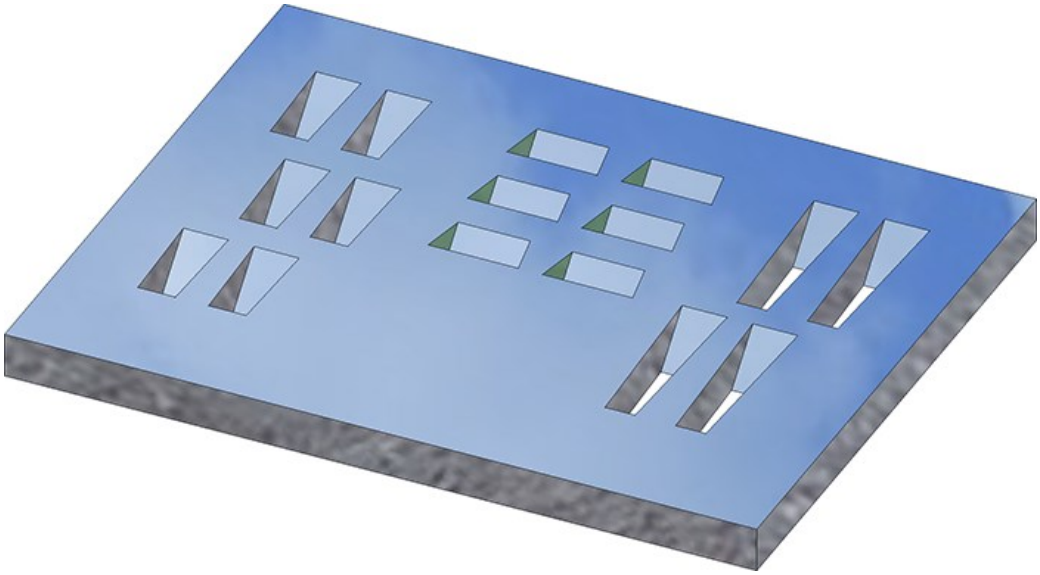
You can choose between the following formats:

- DXF/DWG,
- XML for CADMAN-B (from Version 7.5) by LVD and
- GEO for ToPs GEO

Extensive information can be found in the HiCAD Sheet Metal Help.

DSTV-NC

- From HiCAD 2019 onwards, DSTV Interfaces can be activated with any module that is based on HiCAD creator or HiCAD solution.
- When carrying out exports via DSTV-NC interface, rectangular subtractions running diagonally through the part will also be considered:



Sheet Metal

Service Pack 2 2019 (V 2402)

Settings from ABWPAR.DAT and ABWCOL.DAT moved to Configuration Editor

The settings from the system files ABWPAR.DAT and ABWCOL.DAT have been moved to the Configuration Editor.

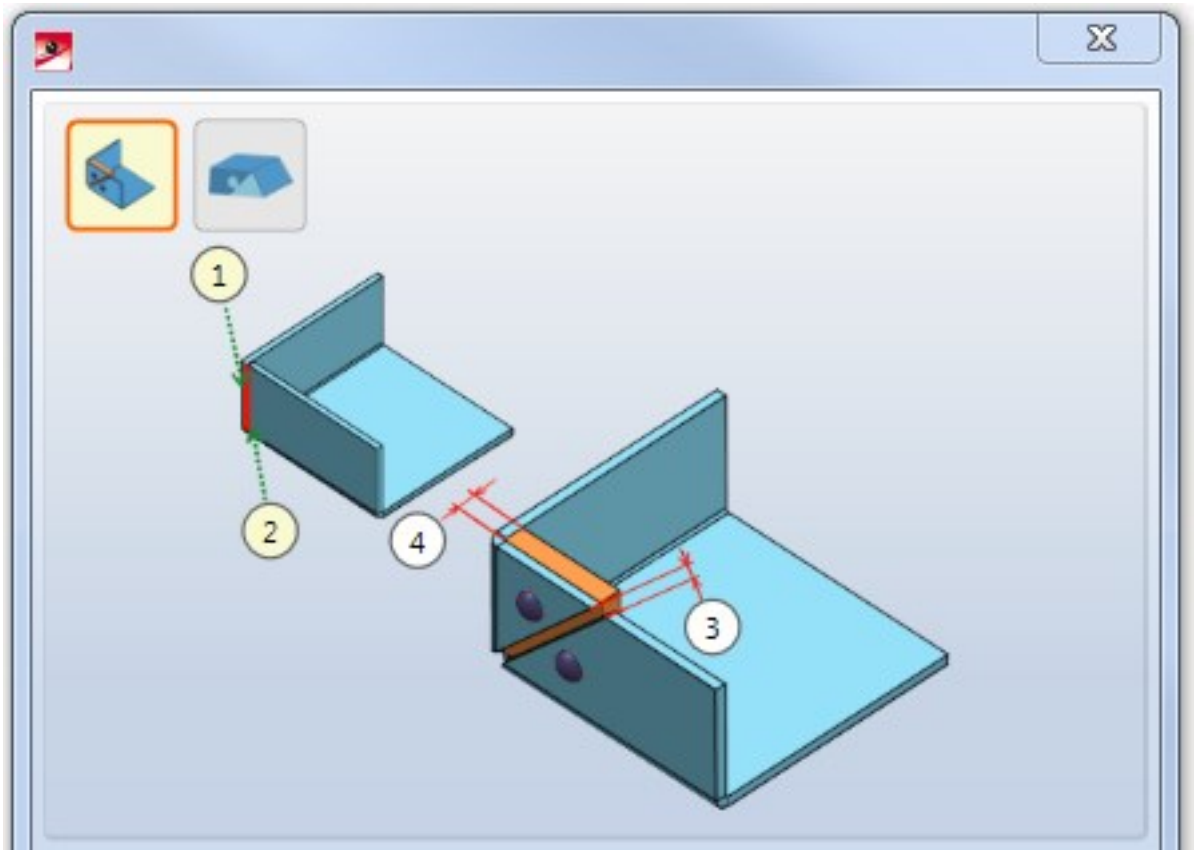
You can now find the settings from those DAT files in the following directories:

ABWPAR.DAT	Sheet Metal > Sheet development
	Sheet Metal > Default setting
	Compatibility > Sheet development up to HiCAD 2016
ABWCOL.DAT	Compatibility > Sheet development up to HiCAD 2016 > Extended settings

Sheet corner Design Variant

The two Design Variants **Sheet corner with stiffener** (riveted) and **Sheet corner with puzzle piece** have been revised and combined into one function. This also allows the insertion of sheet corners without stiffener.

The insertion of these Sheet Metal Design Variants takes place via the **Civil Engineering functions** docking window at **Sheet Metal > Sheet corner with stiffener or puzzle piece**  . In the displayed dialogue, you can then choose between clicking on the **Stiffener** or **Puzzle piece** icon.

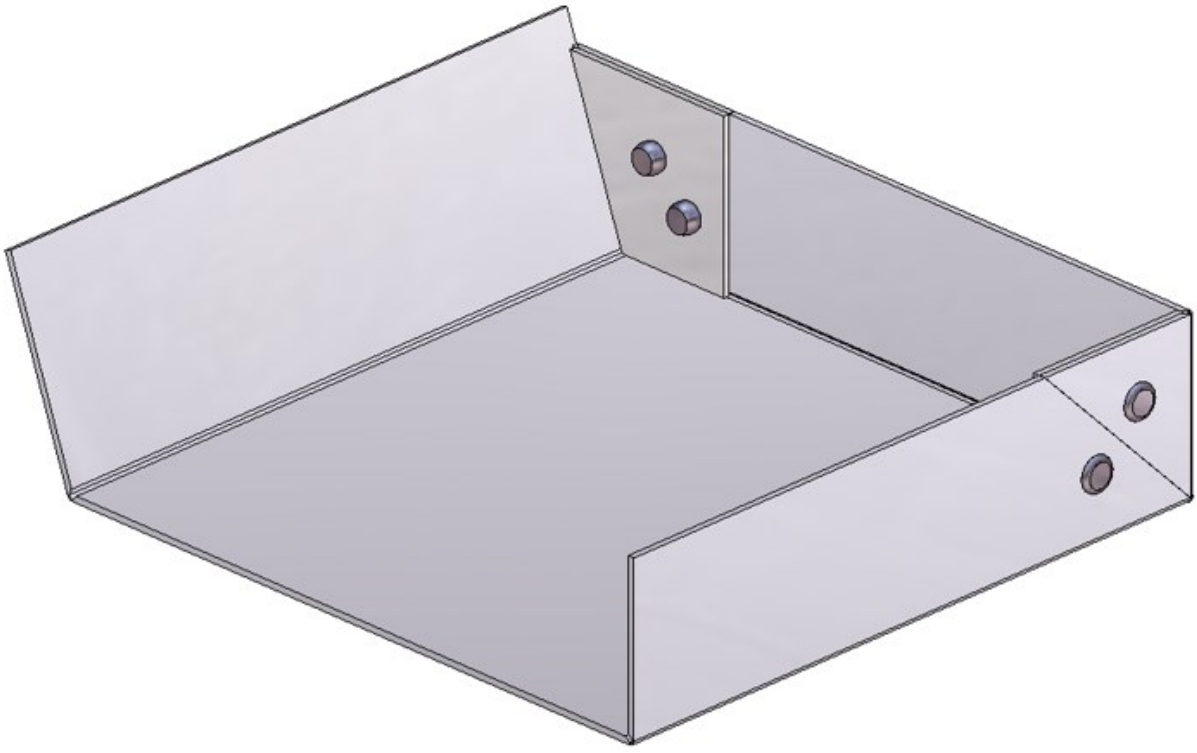


Sheet corner with stiffener	Use this Design Variant to create a sheet corner with or without stiffener between 2 sheet flanges. The flanges can have different lengths and different bend angles. For insertion, identify the two front sides of the flanges. Activate the **Use stiffener** checkbox if you want to create a riveted connection. From a multitude of available fasteners (e.g. SFS or Tit-gemeyer) select the one you need.
Sheet corner with puzzle piece	Use this Design Variant to create a sheet corner with a puzzle piece between 2 sheet flanges. The flanges can have different lengths and different bend angles. For insertion, identify the two front sides. The connection will be applied to the last identified flange. If you want a laser cut-out, activate the corresponding checkbox and enter the laser beam diameter for the cut out. If required, change the parameters of the puzzle piece.

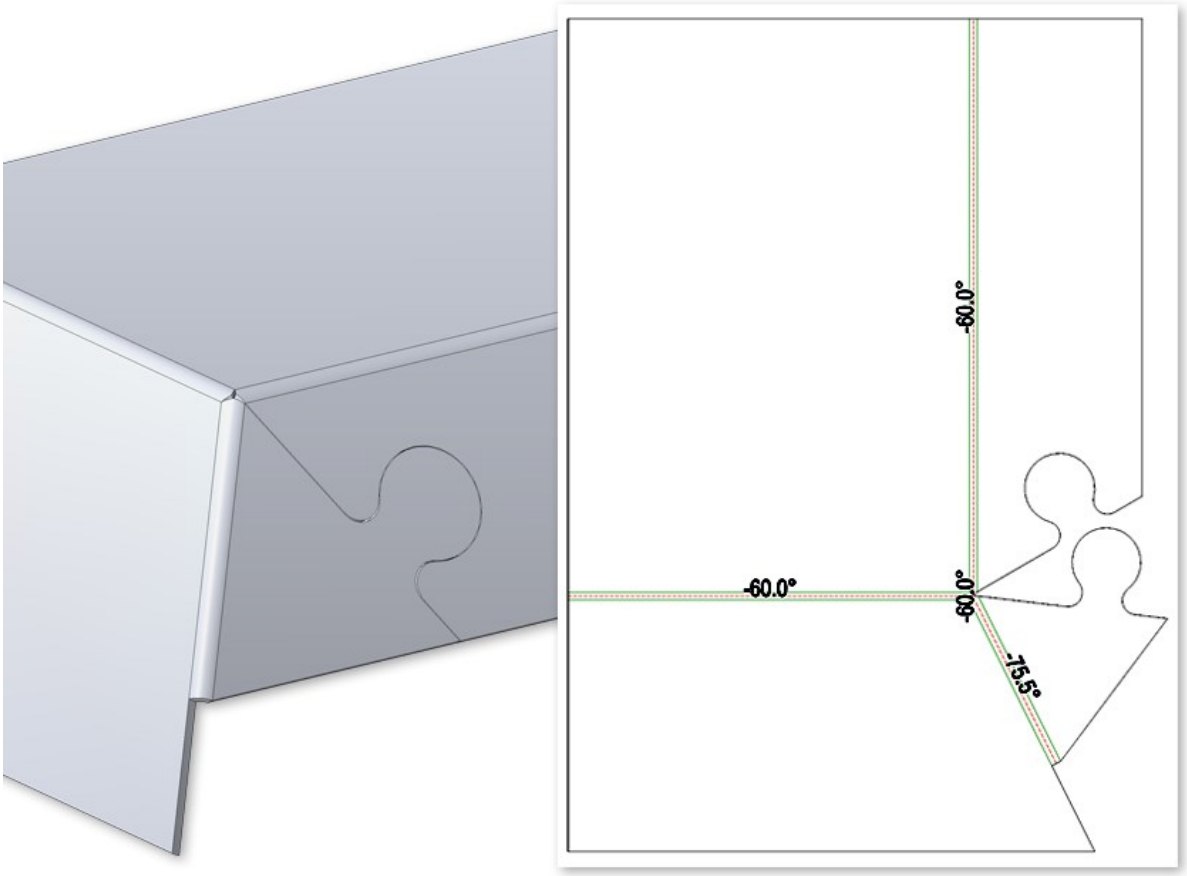
After entering all required data you can create the connection. Click **Preview** to check and, if required, modify the connection. If you click **Apply**, the sheet corner will be created, and the dialogue window remains open, allowing you to change the data or to switch to the other sub-variant, i.e. **Puzzle piece**  or **Stiffener**  . the data to other sheet corners. If you click **OK**,the connection will be created, and the window will be closed. Clicking **Cancel** closes the dialogue window and discards the data without creating the connection. .

Incorrect inputs are marked with this symbol . When you move the cursor over the symbol an error message will be displayed.

If the function cannot be executed with the entered data, this symbol will appear at the **OK** button. When you move the cursor over the symbol an error message will be displayed.



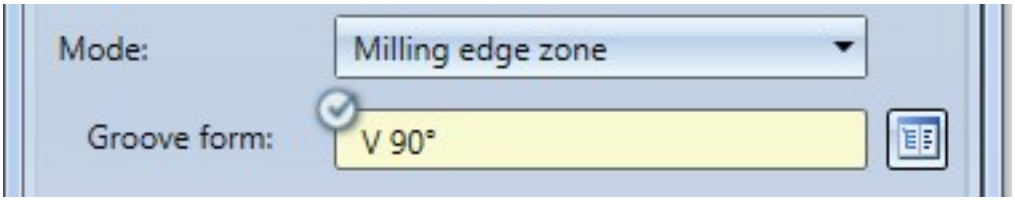
Sheet corner with stiffener



Sheet corner with puzzle piece

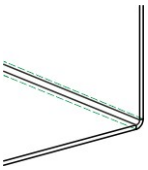
Folding with milling edge zone

Composite sheets, such as ALUCOBOND panels, are folded along milling edges. The resulting milling edge zones differ geometrically from the cylindrical bend zones. Therefore, the **Fold flange** function now offers the additional option to choose milling edge zones and milling tools.

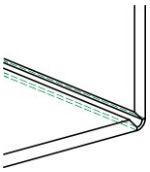


The following options are available:

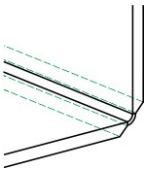
Bend zone
The bend zone has the specified bend radius.

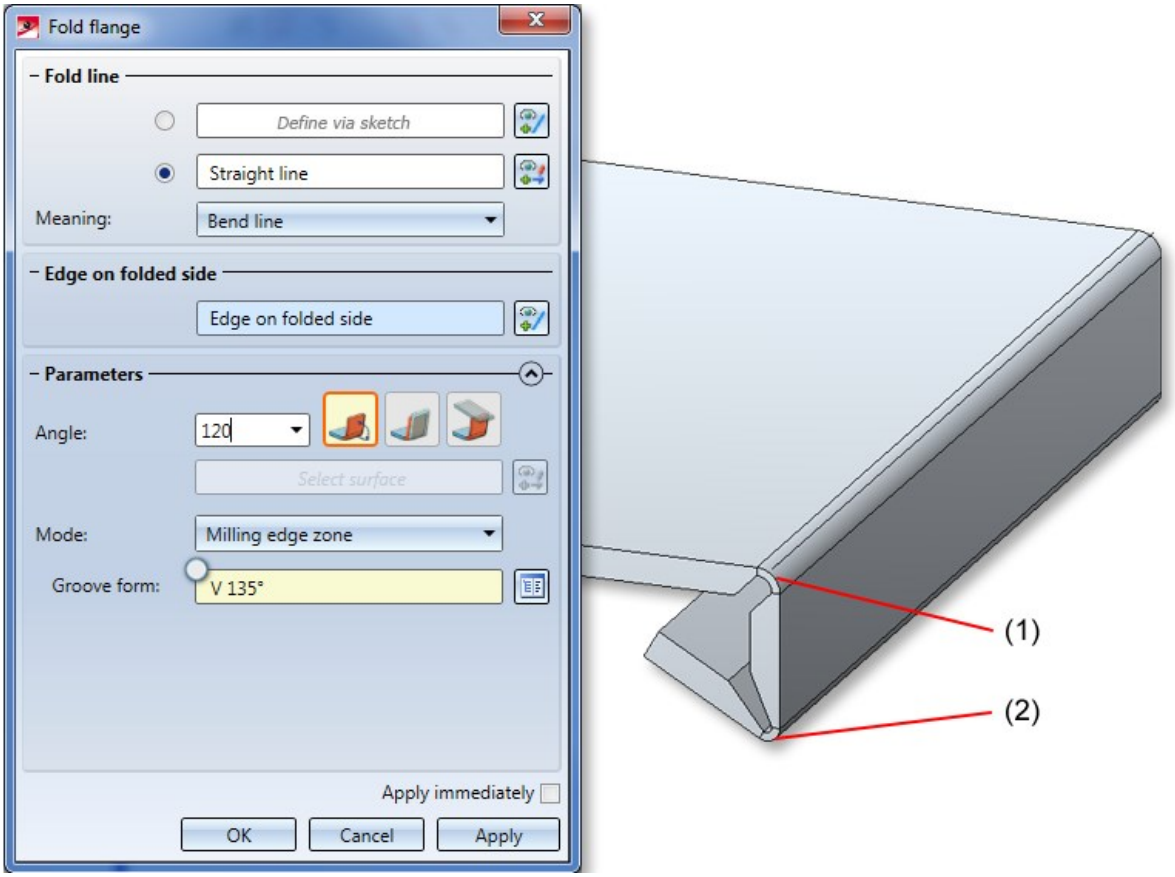


Milling edge zone
Bend zone with the selected tool (milled inside).



Milling edge zone inverted
Bend zone with the selected tool (milled outside).



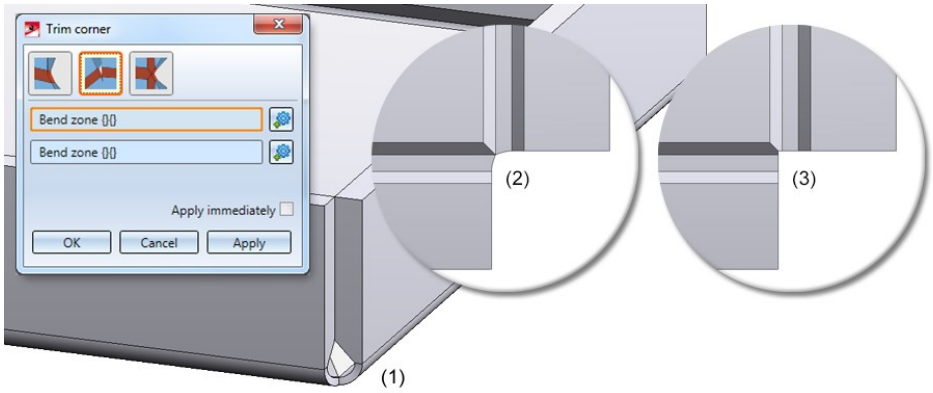


(1) Fold line defined by sketch, milling edge zone with groove form 90°
(2) Fold line on straight line (mid point of flange), milling edge zone with groove form 135°

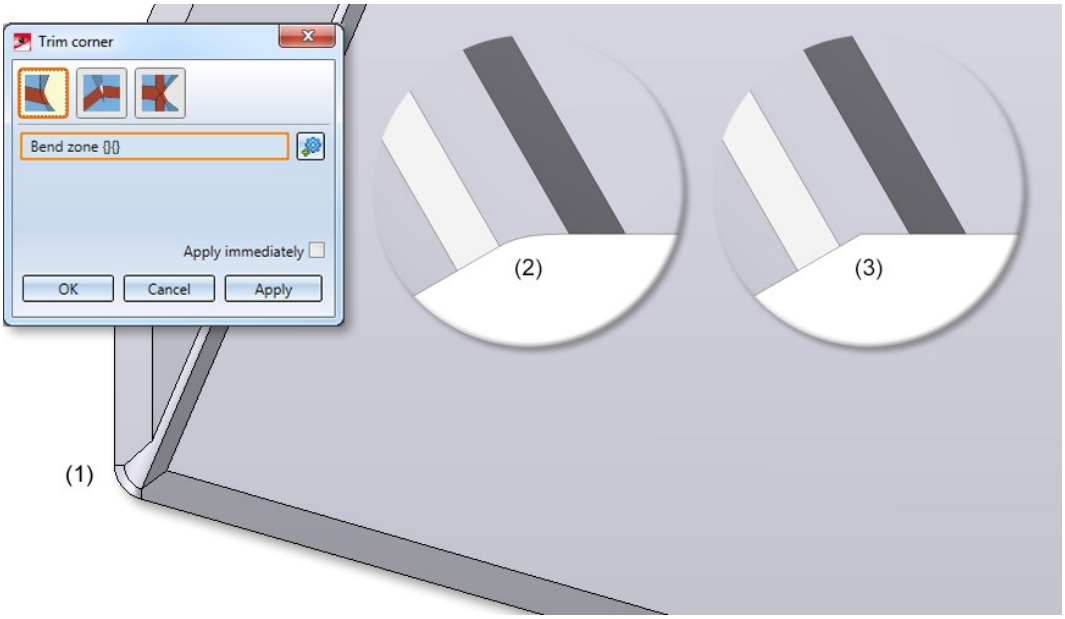
Bend zone ends suitable for milling

Some CAM systems (e.g. CobusNCAD) do not support arcs at bend zone ends. To create bend zones suitable for milling, you can now use the **Trim corner**  function to trim transitions with two or three milling edge zones in your sheet metal drawing. First activate the icon for the number of milling edge zones to be trimmed and then identify the milling edge zones. This creates 2 straight lines as an extension of the flange edges converging in the milling edge zone.

The ALUCOBOND SZ 20 cassettes were edited with the new **Trim corner** function having bend zone ends suitable for milling.



Trim corner, 2 milling edges
(1) Original situation
(2) View of original situation in the bending simulation (top view)
(3) Trimmed milling edge zone (top view)

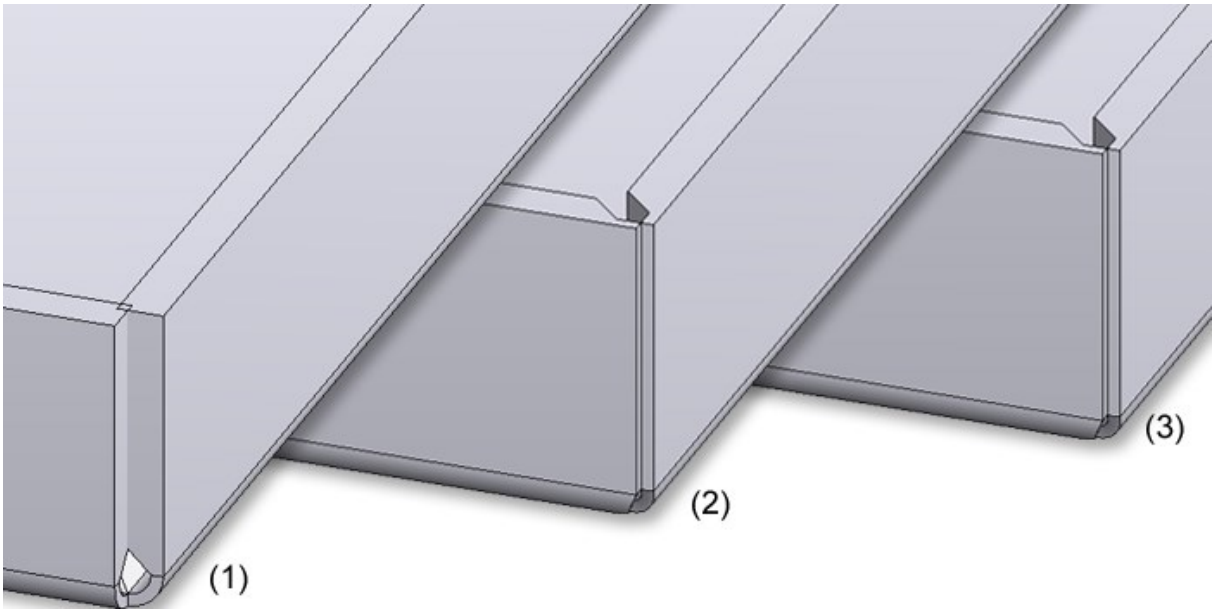


Trim corner, 1 milling edge zone
(1) Original situation
(2) View of original situation in the bending simulation (top view)
(3) Trimmed milling edge zone (top view)

Corner processing/Mitre

With composite panels (e.g. ALUCOBOND) it is often necessary to process the edges in such a way that a milling edge occurs. For this reason, it is now possible to finish these flanges in **Corner processing/Mitre within one sheet or between different sheets**  .

With the modes **Mitre cut** and **Mitre cut, with neighbours** you are allowed to select the option **Sheet edge as milling edge**  for fitting parameters. For the **Groove form** you can choose the matching milling tool from the **Composite panels, groove form** catalogue.

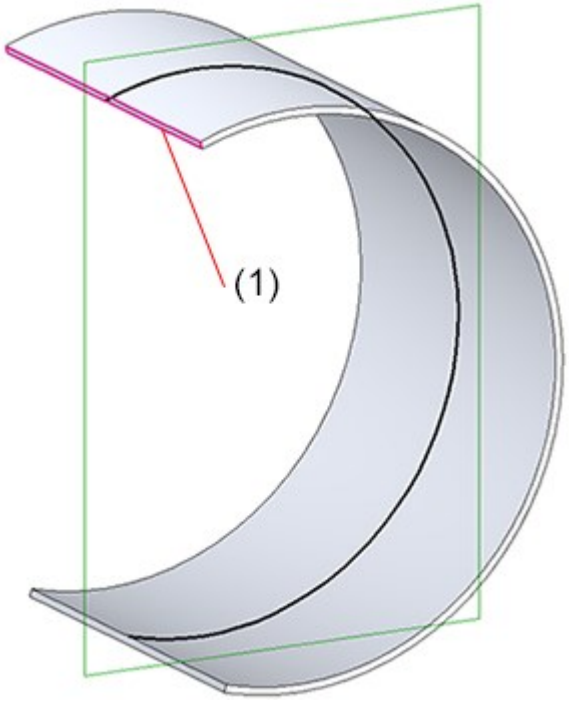


(1) Composite panels before processing
(2) After processing, mode Mitre cut, flanges with milling edges
(3) After processing, mode Mitre cut, with neighbours, flanges with milling edges, milling edge zone linear

Sheet from circular arc

When using the function **New sheet along sketch**  for the creation of a sheet from only one circular arc, one short, tangentially connected straight line element needed to be added at one end of that circular arc.

To simplify things, this element will now be added by HiCAD automatically as a flange (double tolerance, i.e. 0.002 mm) at the start of the circular arc.



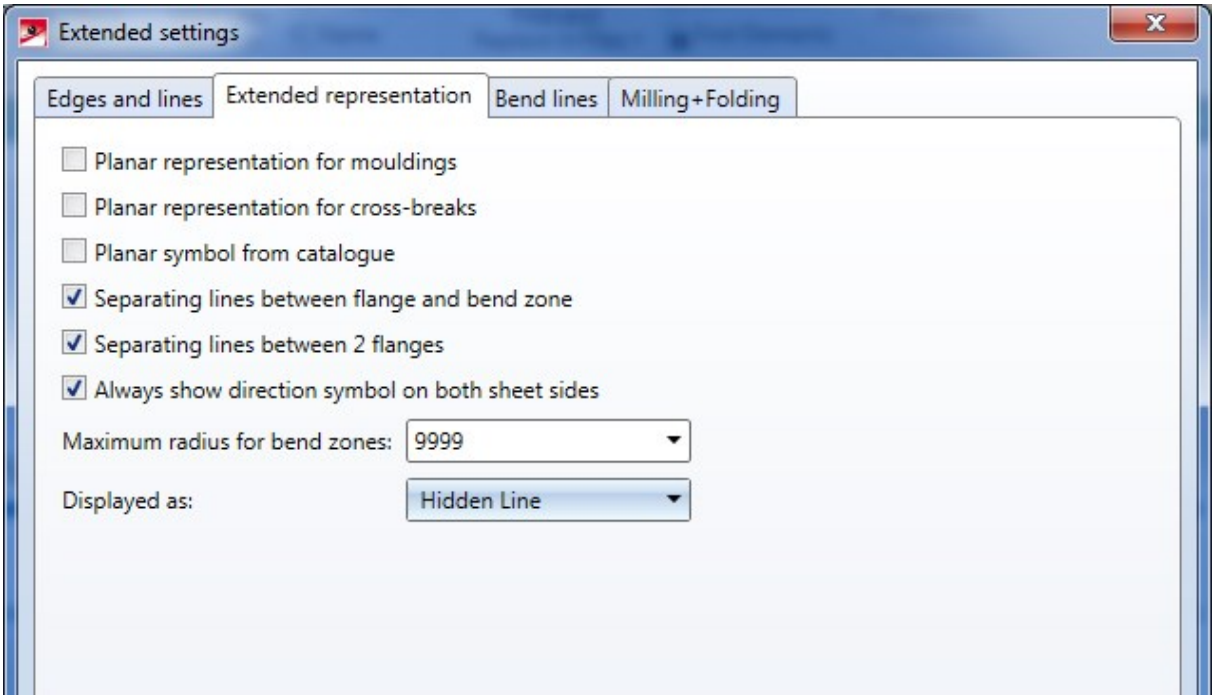
(1) Tangentially connected flange, 0,002 mm

Development

Visualisation of direction arrow when cutting ALUCOBOND trays to size

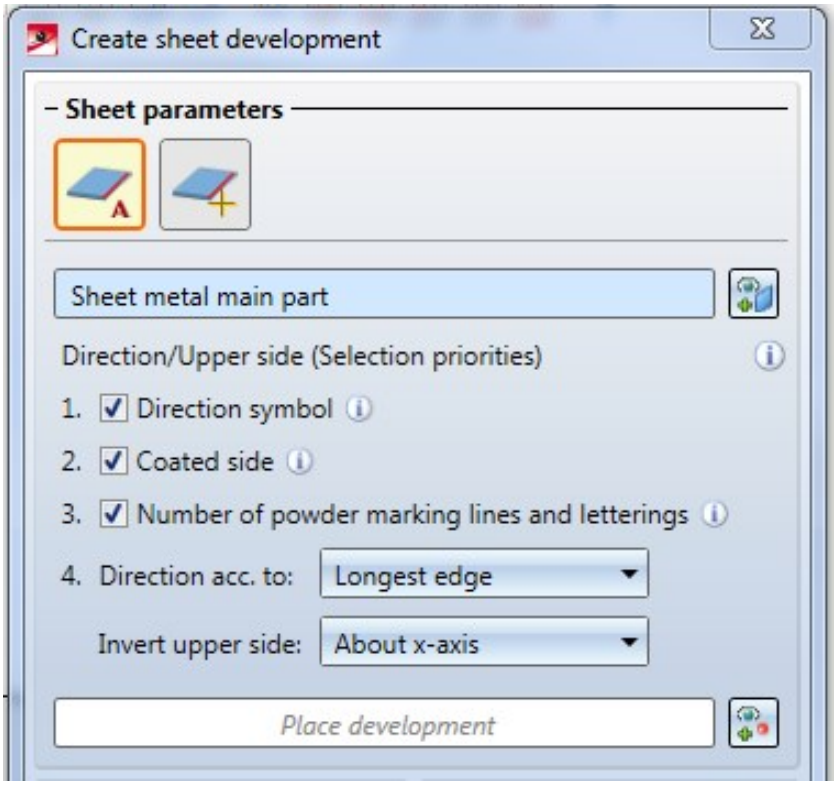
In previous versions the direction symbol for the processing direction was not visible in the DXF export if it was located on the rear side of the sheet development. On facade trays (e.g. ALUCOBOND trays) the direction symbol is often located on the front side of the trays. The development will then be carried out from the rear side in order to display the milling edges. Here, too, the direction symbol would not be visible in the export.

To visualize the direction symbol on the other sheet side, choose **Sheet development > Update> Default setting** and then, in the opened dialogue window, click on the **Extended settings** icon. In the next dialogue window, open the **Extended representation** tab and activate the **Always show direction symbol on both sheet sides** checkbox to copy the symbol to the other side of the sheet.



Favorites for developments

If you choose **Sheet alignment: Automatic**  in the **Create sheet development** dialogue window and select the Favourite COBUS NCAD, the option **Invert upper side: About x-axis** will now be set.



Changes to Favourites only affect initial installations; Favourites remain unchanged. Existing Favourites in an update installation remain unchanged.

Eternit

HiCAD now also supports the placing of Eternit panels .

Eternit semi-finished product data

The semi-finished product data for Eternit panels can be found in the Catalogue Editor, at **Factory standards > Sheets > Eternit**, where you will find the following tables:

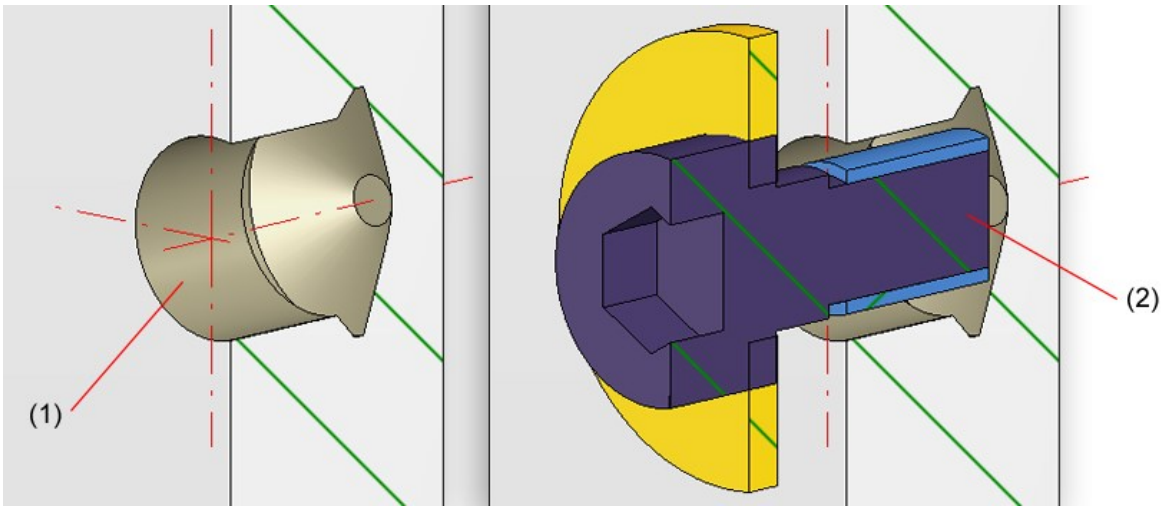
- EQUITONE [linea]
- EQUITONE [materia]
- EQUITONE [natura]
- EQUITONE [natura] PRO
- EQUITONE [pictura]
- EQUITONE [tectiva]
- EQUITONE [textura]

The semi-finished product dependent part colours are defined in the tables of **Factory standards >Surface treatment > Profile and Element Installation > Eternit**.

Eternit Tergo / Tergo+ undercut bores

In the Catalogue Editor at **Sheet tools > Moulding tools** you can find a new table called Undercut bore for Eternit Tergo und Tergo+ undercut bores.

Insertion in HiCAD takes place in the same way as the other moulding tools (e.g. beadings or vents), i.e. via the function **Sheet Metal > Further Tools > Moulding tools**  .



(1) Undercut bore
(2) Undercut bore with bolting

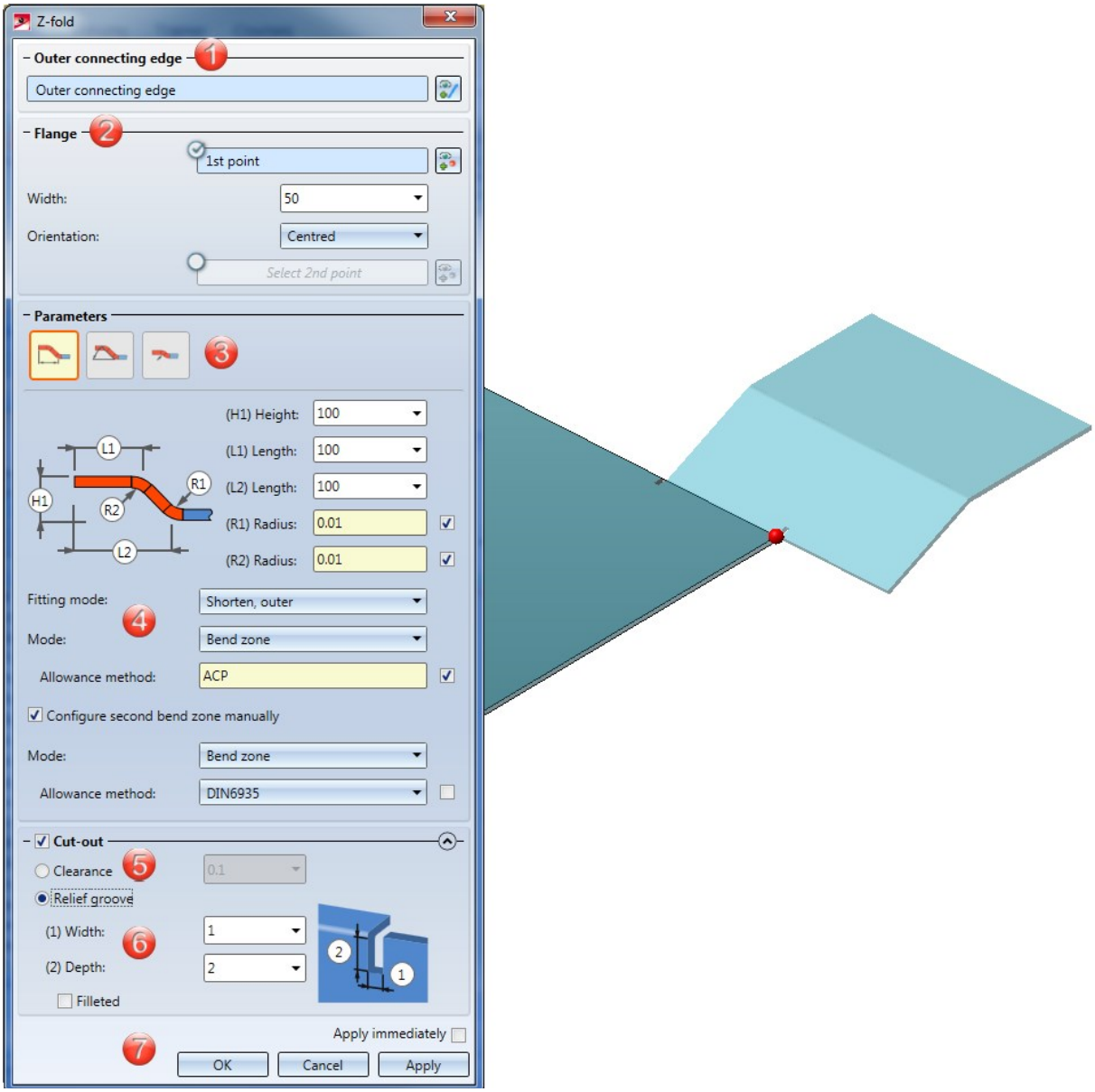
Sheet Metal parts with identical cross-sections

The settings for **Sheets with identical cross-sections** are now available as a sub-entry of **Sheet Metal**.

Service Pack 1 2019 (V 2401)

New Z-fold

The new **Z-fold** function allows you to attach 2 flanges to a sheet with one single working step. As soon as your inputs are sufficient, a preview of the new flanges will be displayed. That is, if you choose a connecting edge, the possible Z-fold with the standard values will be displayed. If you have activated **Width via point(s)**, the new flanges will only be shown after identifying a point and the width, or 2 points, provided that you have chosen the connecting edge. Furthermore, you will directly seethe change if, for instance, you switch from **Shorten, outer** to **Shorten, inner**. If the **Apply immediately** checkbox has been activated, the Z-fold will be immediately attached if the value inputs are sufficient and make sense, and can then be changed via the Feature log.



The selection of the connecting edge is decisive for the direction of the new Z-fold. After selecting the outer connecting edge, a preview of the new flanges will be displayed, provided that you have not activated any options that

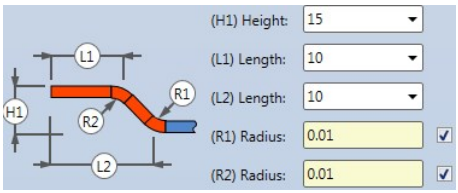
require further value inputs. If the **Apply immediately** option has been activated, the Z-fold will be inserted immediately. Changes can then be carried out the via Feature log.

If you want to identify a different edge, click on the  icon in the **Outer connecting edge** area and identify the edge.

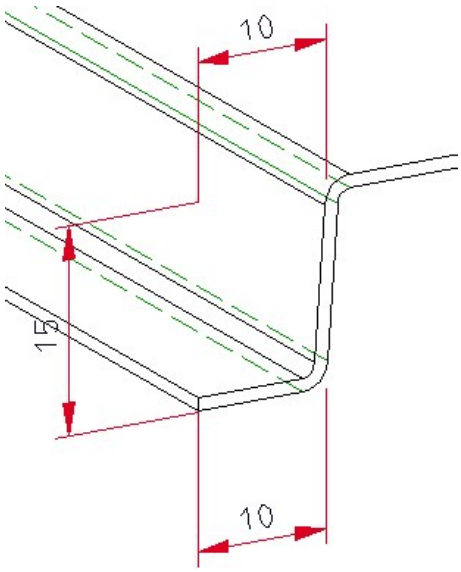
If you activate the option for the 1st point  , the width of the new folding can be freely selected - irrespective of the connecting edge. After choosing the 1st point in your drawing, you can set the **Width** and the **Orientation**; alternatively, you can identify a second point. If you want to choose a different point selection, click on the  icon and identify the point. If the points are located on an edge, this edge will become the connection edge, provided that you have not identified the connecting edge yet.

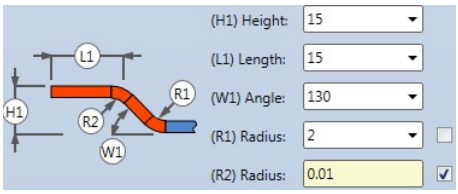
If you have activated the **Relief groove** option, the **Fitting mode** will only refer to the flange area. If the option has been deactivated, the Fitting mode (e.g. with shortening of the connecting flange) will refer to the complete connecting edge.

Parameters for Z-fold

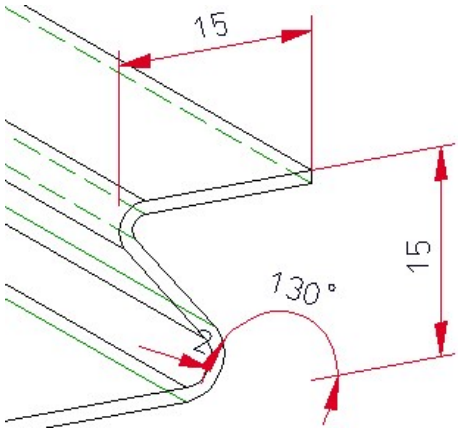


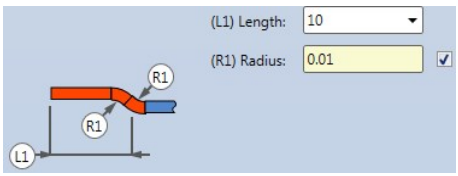
If this option has been chosen, you create the Z-fold by specifying Height and Length. The Radius is loaded from the semi-finished product in this example. Deactivate the checkbox ☐ to change the Radius.



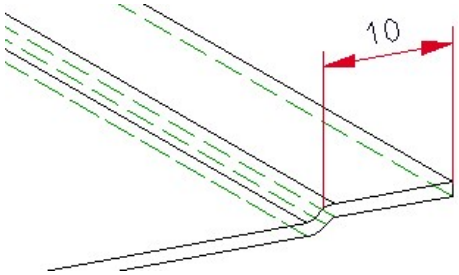


Here you define the Z-fold by entering an Angle. The first Radius here is entered manually, the second Radius will be loaded from the semi-finished product.



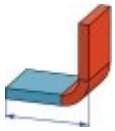


Here you create the Z-fold via an offset by the sheet thickness. The Radius will be loaded from the semi-finished product.



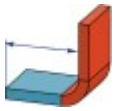
General parameters

The **Fitting mode** determines the processing of the existing flange.



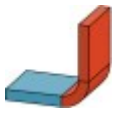
Shorten, outer

The connecting flange will be shortened if you use this mode. The outer flange surface intersects with the plane of the front surface before the shortening.



Shorten, inner

The connecting flange will be shortened. The inner flange surface intersects with the plane of the front surface before the shortening.

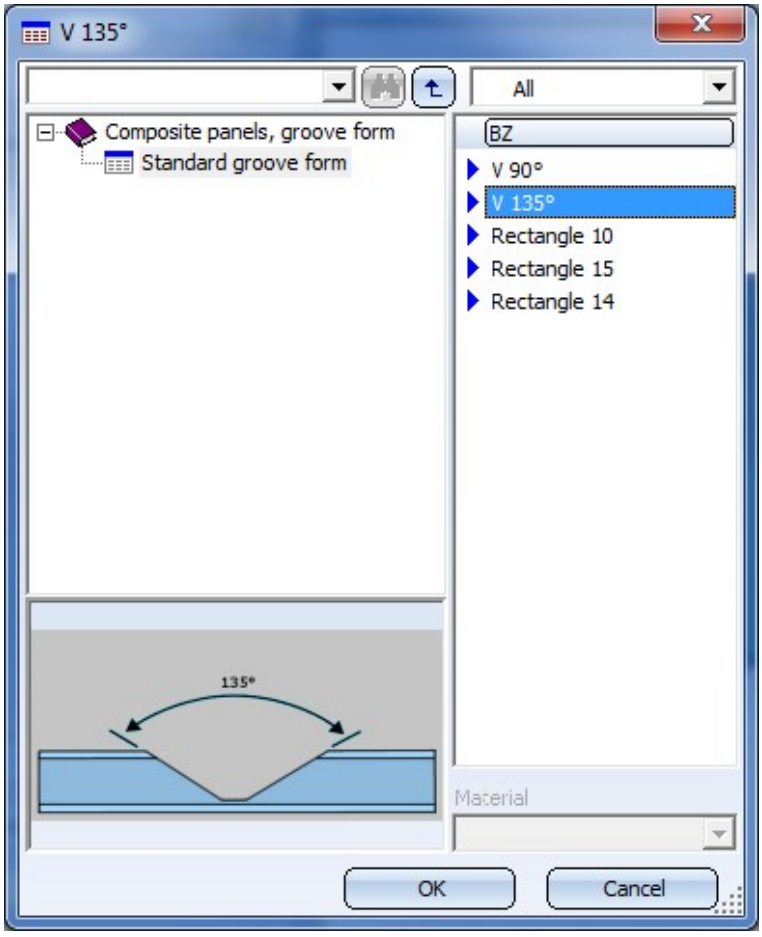


Without shortening

The connecting flange will remain unchanged.

For the **Mode** option you can choose between **Bend zone** and **Milling edge zone**. Milling edge zones are in practice used for composite panels. If you choose Bend zone, you can subsequently assign an **Allowance method**. If you choose Milling edge zone, you can subsequently choose a **Groove form**.

For the **Groove form** you can choose the matching milling tool from the **Composite panels, groove form** catalogue  .



If you have activated the checkmark  here, the tool will be assigned automatically.

Clearance/Relief groove

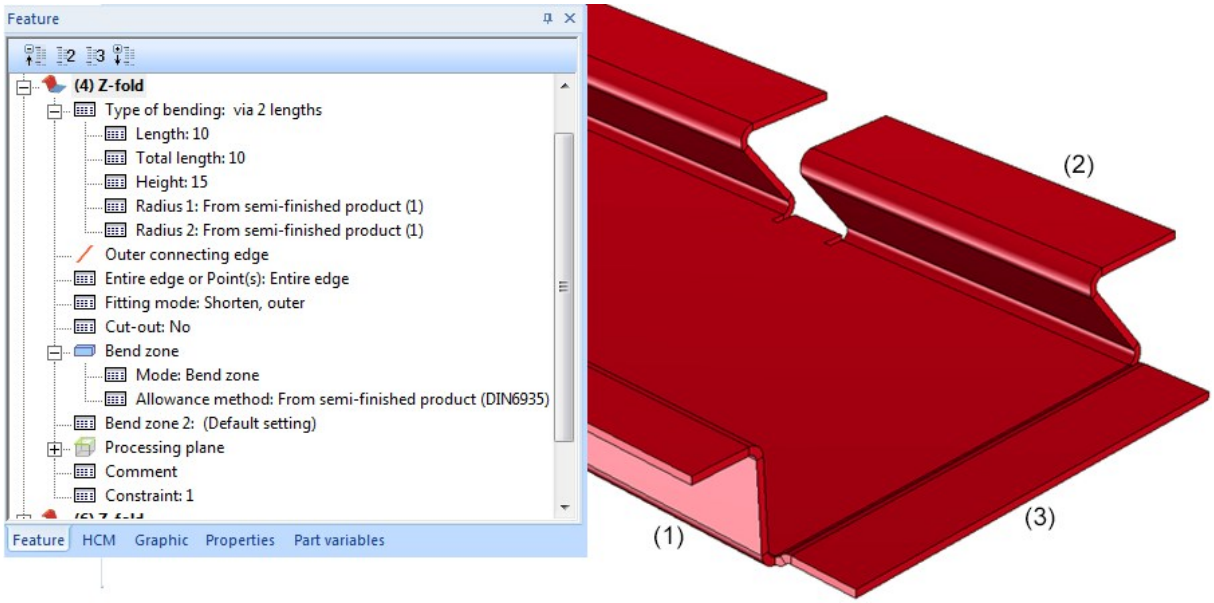
A **Clearance** will be created if you attach a flange between 2 points, requiring a shortening of the connecting surface due to the fitting mode. If you right-click on the input field you have the option to pick the clearance from the drawing.

If the flange is larger or smaller than the connecting edge, and the fitting mode shortens the connecting edge, it make sense to insert a **Relief groove**. Enter a **Width** and a **Depth** for the relief groove.

Apply/discard inputs

After entering all required data you have the following options: If you click **Apply** or press the MMB, the new Z-fold will be inserted into your drawing, but the dialogue window will remain open, allowing you to make adjustments if required. If you click **OK**, the Z-fold will be inserted and the window will be closed. If you choose **Cancel**, the window will be closed and the specified settings will be discarded. If you have activated the **Apply immediately** checkbox, the parameters will be applied on the spot.

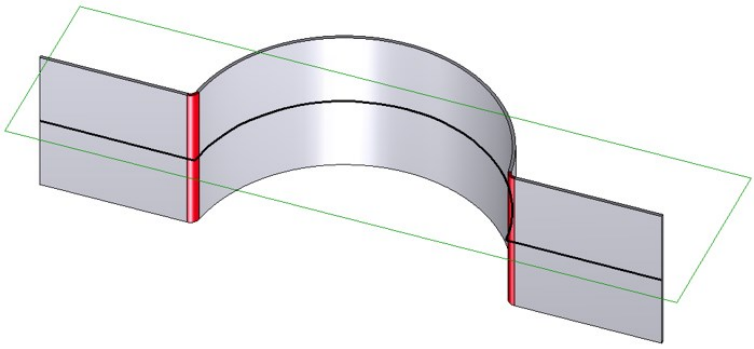
Example



(1) Z-fold with length, height and Feature log
(2) Z-fold with angle, width via points and relief groove
(3) Z-fold with offset by sheet thickness

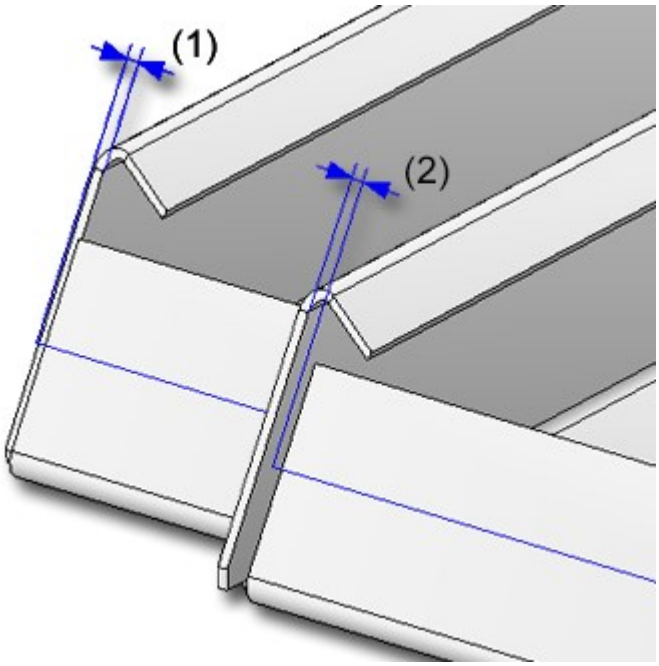
Sheet along sketch, with non-tangential transitions

The **Sheet along sketch** function now also considers non-tangential transitions between a straight line and a an arc. Where the line bends, a bend zone or a milling edge zone will be inserted:



Trim flanges, with negative clearance

When using the Trim functions on flanges or planes you have now the option to enter a negative value for the clearance.

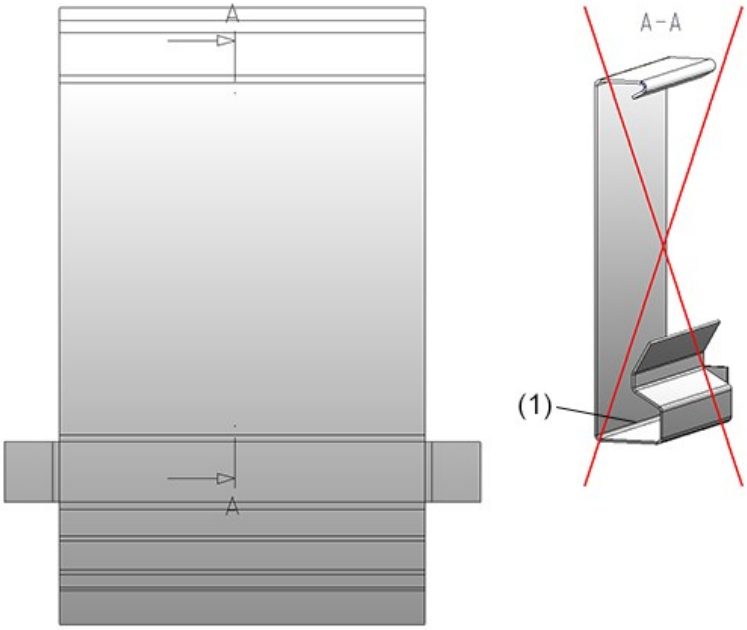


(1) Trimming the plane, Clearance 2 mm
(2) Trimming the plane, Clearance -2 mm

Bending simulation in sectional views and detail views

When performing bending simulations you can now also identify the edge for the base sheet in the sectional view or detail view. The bending simulation will then be performed in the original view. This will invalidate the sectional/detail view, which needs to be updated.

If you have selected an edge that was caused as a result of the cut, an error message will be displayed.



(1) Edge for base sheet

Please note:

The identification will not work if the sectional view has been created with the options **Unfolded** or **Only cut surface**.

Development

Line parameters for developments

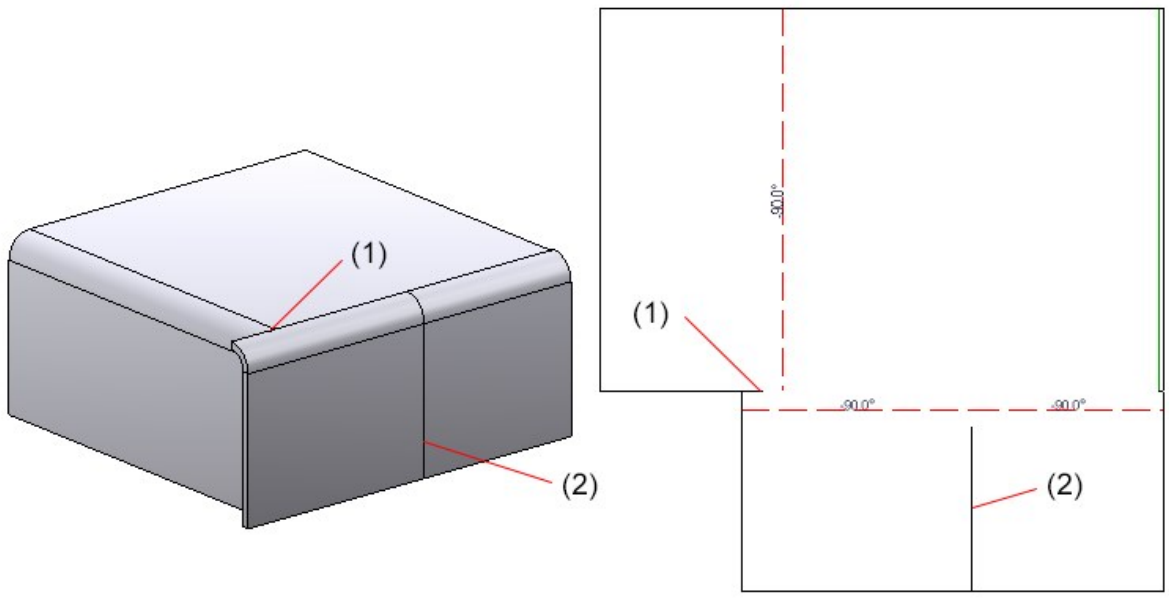
When you click on the **Extended settings** button of the **Sheet development parameters** dialogue window and open the **Edges and lines** tab, you will now find new checkboxes for the element types Crosshairs, Embossing lines, Direction symbols and Powder marking lines.

If the respective checkbox has been activated, the parameters set here will be considered for development. If the checkbox is deactivated, the pre-setting will be used.

Free edges

If two flanges abut when performing a development without clearance, a new line representing a separating cut will be inserted in the development. If the front surface of a bend zone directly abuts on a flange, here, too, a separating cut line will be inserted.

If you do not want to show separating cut lines between flanges, or between flanges and bend zones, deactivate the corresponding checkbox **Extended representation** tab.

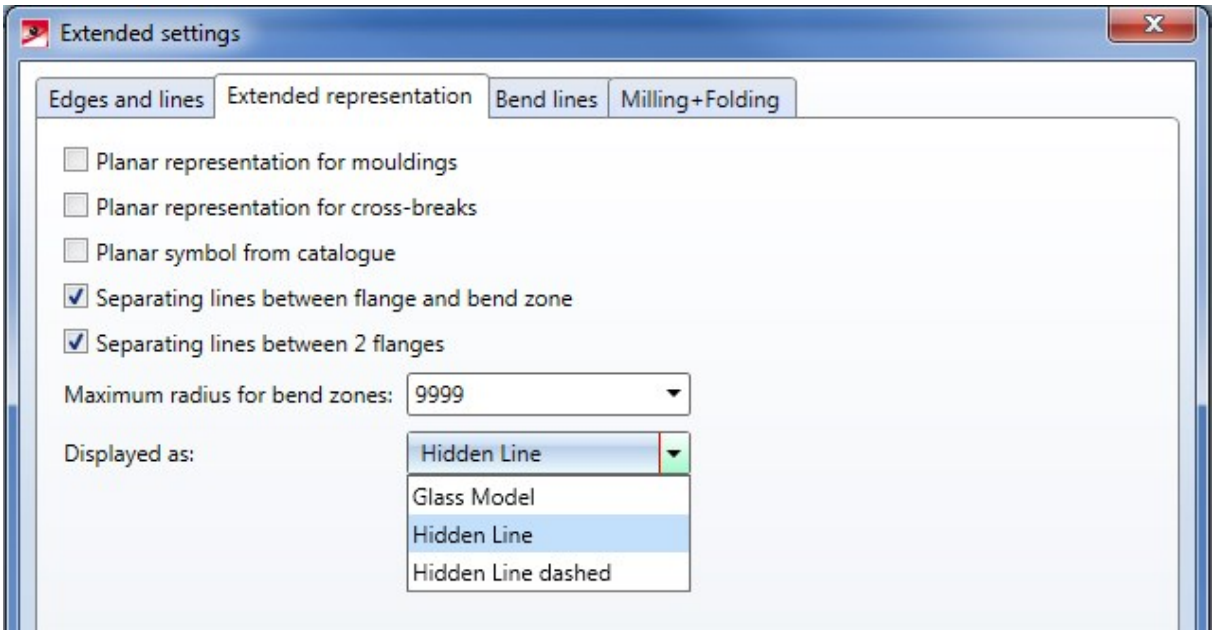


(1) Separation cut line between flange and bend zone
(2) Separation cut line between flanges

Representation of sheet developments

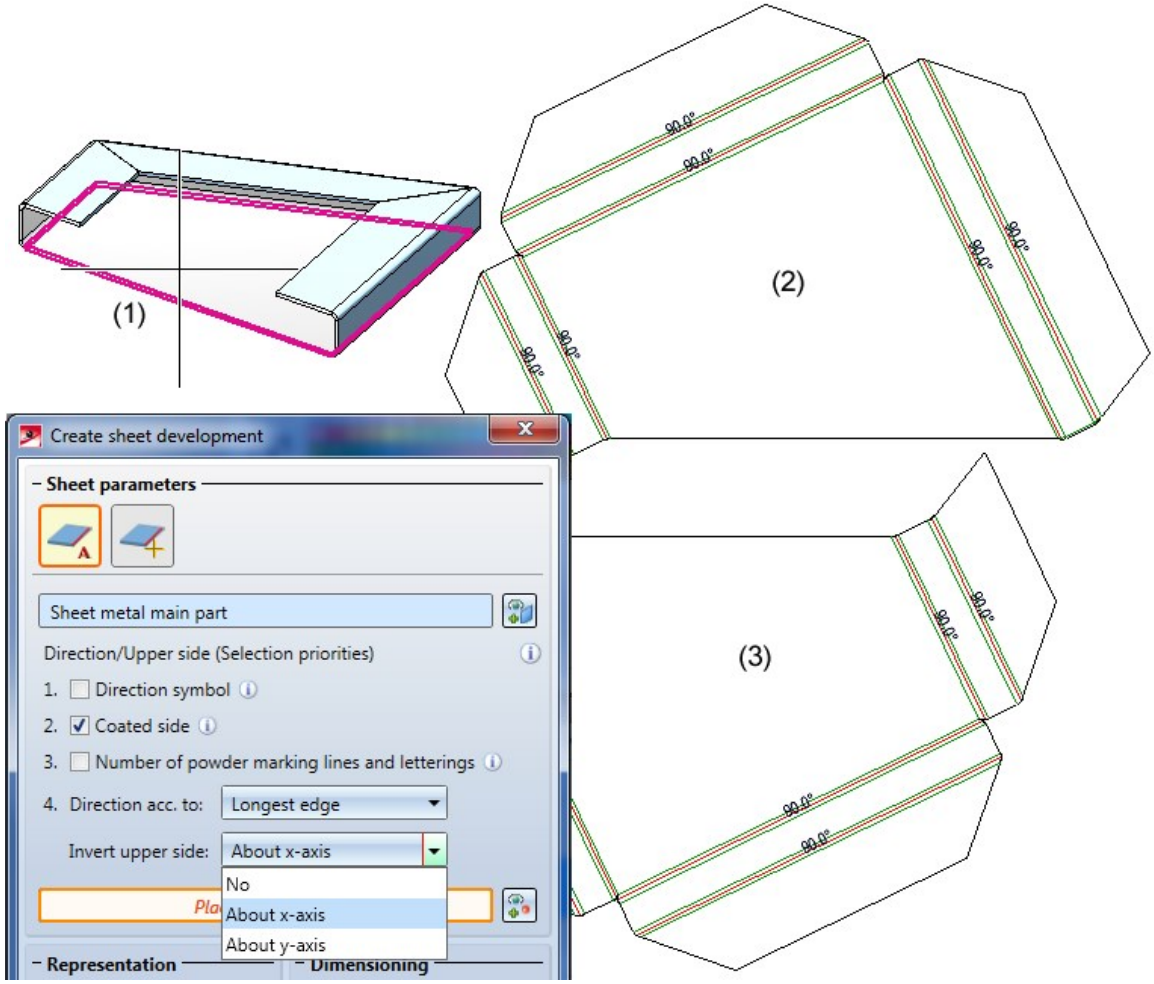
You can now select via the **Sheet development parameters** how a created development is to be displayed: Open the **Extended settings** dialogue window, go to the **Extended representation** tab and choose the desired representation from the **Displayed as:** selection list. The default value is **Hidden Line**.

The default setting for the sheet development and the changing of the Favourites also affects drawing derivations  for sheets, the export of developed sheets as DXF files  and the settings in the Configuration Editor (Sheet Metal> Sheet development > Default setting).



Invert axis direction for upper side

When milling composite panels the coated side is normally the bottom side. With the **Invert upper side** option you can rotate the upper side to the X- or Y-axis when creating the development.



(1) Sheet metal main part
(2) Upper side inverted, rotated about Y-axis
(3) Upper side inverted, rotated about X-axis

Export sheet developments as STEP files

The functions for data exchange of sheet developments have been enhanced. Besides DXF/DWG, XML and GEO you can now also export STEP files.

Export individual development	This function exports an individual development - either completely or as a section contour.
Develop and export sheets	Use this function to export the section contour or the complete development of Sheet Metal parts or Steel Engineering plates. In the process, a temporarily created development for the export will always be output.
Export sheet developments	Use this function to export multiple developments with different parameter settings, e.g. LVD and Bystronic, of one or different Sheet Metal parts.

For the export of STEP files you have the following options:

Transfer layers	The layer on which the edge is located (e.g. bend lines on Layer 2) will be transferred during export.
Transfer colours	The colour will be transferred during export. If you have deactivated this option, you cannot use the **Export fits information via surface colour** option (see below).
Export fits information via surface colour	By activating or deactivating this checkbox you can determine whether fits information are to be exported by means of a surface colour. For this to happen, the **Transfer colours** checkbox (see above) needs to be deactivated. The diameter of the corresponding bore will then be highlighted in a different colour in the imported STEP part. In the SYS directory of your HiCAD installation you find the configuration file FITINFO_COLOR.DAT, in which it is determined which RGB value are to be assigned to which fits information.
Export free points	By activating or deactivating this checkbox you can determine whether the exporting of free points in the model drawing should be allowed or suppressed (if they could cause problems in other places).
Export free edges	By activating or deactivating this checkbox you can determine whether the export of free edges in the model drawing should be allowed or suppressed (if they could cause problems in other places).
Thread body	Choose **As separate parts** or **United with parent part** if you want thread bodies to be exported as individual parts or united with their superordinate part, respectively, in the model drawing. Choose **Do not transfer** if you do not want the thread bodies to be exported.

Referenced parts during export

When carrying out exports using the functions

- **Develop and export sheets** and
- **Export sheet developments**

a warning message will be displayed if older versions of referenced sheets exists.

Processing direction

The **Processing direction**  function is not only relevant for Sheet Metal parts, but also for identifying the installation direction of tray panels in the Element Installation module. Therefore, this function has been moved, to the **Standard processings** function group of the 3-D Standard Ribbon tab.

Allowance method and bend radius from semi-finished products

The bend radius and the allowance method can be taken from semi-finished products in the dialogue windows of the following Sheet Metal functions:

- Attach flange
- Flange along sketch
- Z-fold
- Fold

The value will be shown in the dialogue window.

Standardized spelling of punching tools

The spelling of the punching tools (alias names) in the catalogue (at Factory standards) in HiCAD has been standardizes, which you will see when selecting the Punching tools function.

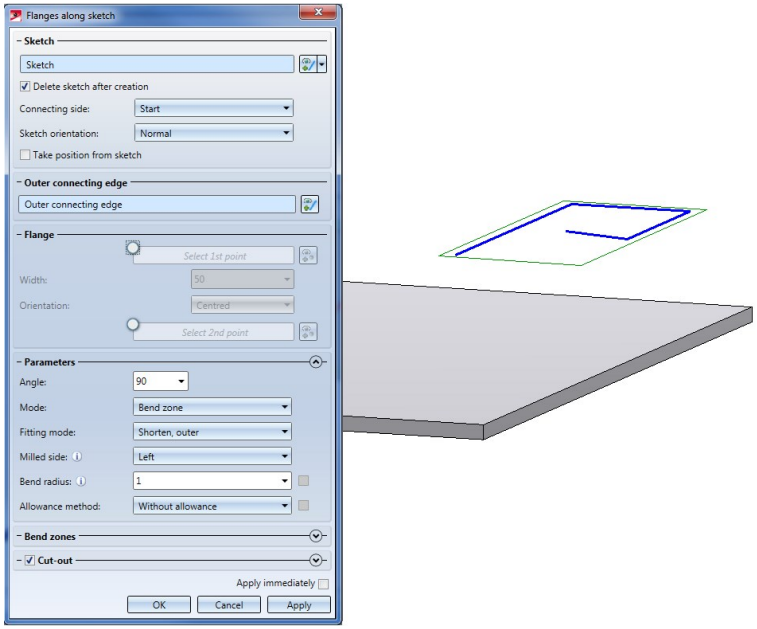
Cut off corner - 3 milling edge zones

Sheets for Element Installation panels will now be assigned notchings that are suitable for production when applying the Cut off corner function.

Major Release 2019 (V 2400)

Flange along sketch

You can use the completely revised **Flange along sketch** function to attach several flanges to a sheet in one step. As soon as your value inputs are sufficient, a preview of the new flanges will be displayed, i.e. when you choose a sketch and a connecting edge, the possible flanges are will be shown on the Sheet Metal part. If you have activated **Width via point(s)**, the new flanges will only be shown after identifying a point and the width, or two points, provided that you have selected the connecting edge. Furthermore, you can immediately see the change if, for instance, you switch from **Shorten, outer** to **Shorten, inner**. If the option **Apply immediately** is active, the flanges will be attached if the value input made sense, and can then only be changed via the Feature log.



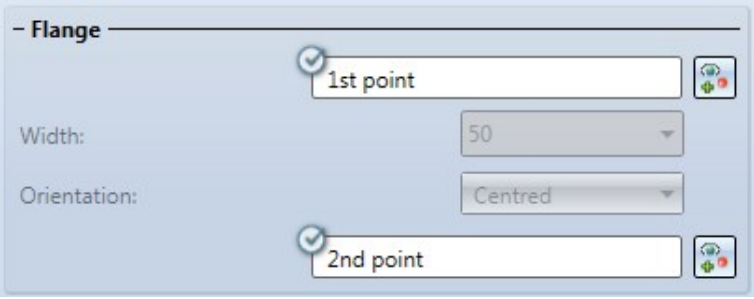
You can also click on the button at the top of the window and choose **New sketch in plane** to draw a new sketch using the Sketch functions. Then, click on the **Apply sketch** button. To change the sketch, click on the **Process sketch** button. This function can also be accessed via the button, The toolbar with the Sketch function and the dialogue window for Sketch processing will be displayed. When you end the processing of the sketch via the dialogue window you will get back to the **Flanges along sketch** dialogue window.

If you want to identify a different sketch, click on the icon in the **Sketch** area and chose the new sketch.

The selection of the connecting edge is decisive for the direction of the new flanges. After selecting the sketch and the connecting edge, a preview of the new flanges will be shown. If the **Apply immediately** option is active, the new flange will be inserted. Changes can then take place via the Feature log.

If you want to identify a different edge, click on the icon in the **Connecting edge** are and choose the different edge.

Flange



When you activate the option for the first point you can freely choose the width of the new flange, independently from the connecting edge.

After selecting the 1st point in the drawing, you can set the width and its orientation; alternatively, you can identify a 2nd point.

If you want to change the selected points, click on the icon for identification.

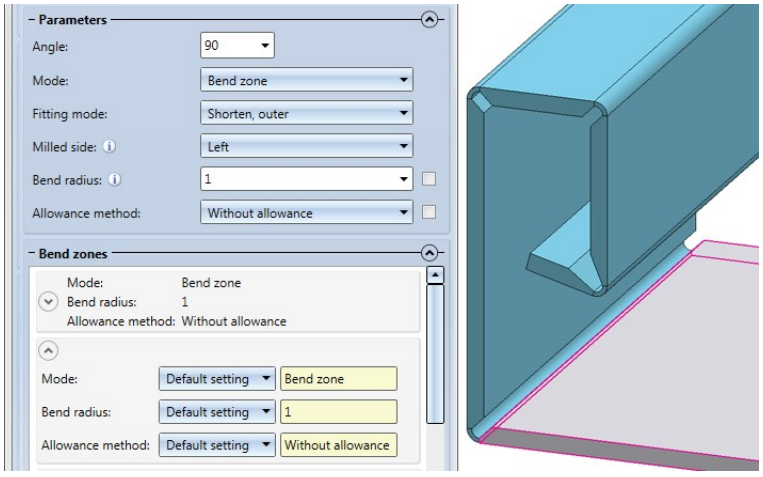
Parameters

Here you enter the bend angle of the first new flange.

For the **Mode** option you can choose between bend zone and milling edge zone. The milling edge zone is used in practice for compound sheets.

For the milled side, which will only be evaluated if the milling edge zone mode has been chosen, the direction of the sketch is decisive. The setting is used as the default setting for the complete Sheet Metal part and can, in the **Bend zones** area, or via the Feature, changed individually for each milling edge zone.

The bend radius will only be evaluated if the **Bend zone** mode has been chosen.

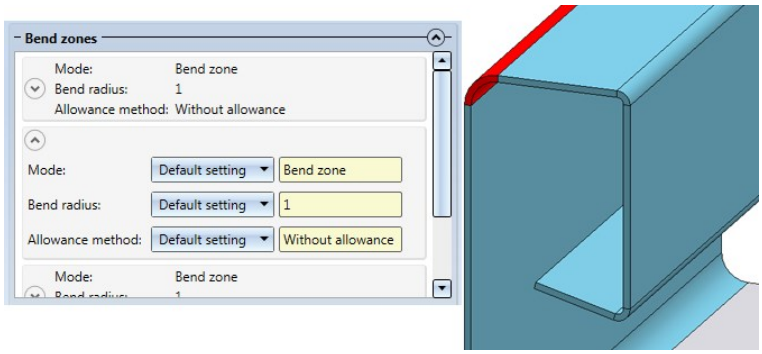


Bend zones

In the **Bend zone** area you can assign individual values to each bend zone.

In the expanded table you can choose between a bend zone or a milling edge zone for Mode. Depending on the chosen mode, you can then change the value for the bend radius or the milled side.

Depending on the processing type, the allowance method can be selected for each bend zone.



Relief groove

If the flange is smaller than the connecting edge and the fitting mode shortens the connecting sheet, it makes sense to insert a so-called relief groove. Enter the width, the depth and the type of the relief groove.

Apply or discard changes

After completion of your data input you can attach the new flanges. If you click **Apply** or press the MMB, the flanges will be attached, and the dialogue window will remain open, allowing you to change the data and apply them to a different connecting edge. Clicking **OK** attaches the flanges and closes the dialogue window. Clicking **Cancel** discards your value inputs and changes, without attaching the flanges. If the option **Apply immediately** is active ☑, the data will be applied directly.

Speed up design processes via MMB

In many functions, the middle mouse button (MMB) cannot only be used to cancel a function (as long as the entered data has not been confirmed yet), but also to speed up design processes. For instance, after creating a new base sheet, you can directly apply the data you entered in the dialogue window to another base sheet by pressing the MMB, without having to open a new dialogue window and pressing the Apply button again. This is possible with the following functions:

- Create base sheet
- New sheet from sketch
- New sheet along sketch
- Attach flange
- Corner/Mitre
- Trim
- Fold flange
- Develop sheet

Revised Sheet Metal Ribbon tab

To further improve the user-friendliness of the **Sheet Metal** tab, some functions were combined into new function groups and moved to the first, immediately visible level of the UI.

Sheet Metal functions	
New	
Attach	
Bend zone	
Process	
Change length	Corner/Mitre
	Flange
	Sheet
	Trim
Process	
Further Tools	
Sheet development	Development
	Update
	AllMet
	Export
Further functions	
Docking window	Design Variant
	Cost calculation

Export

The data exchange functions have been optimized further. With the following three functions you can output

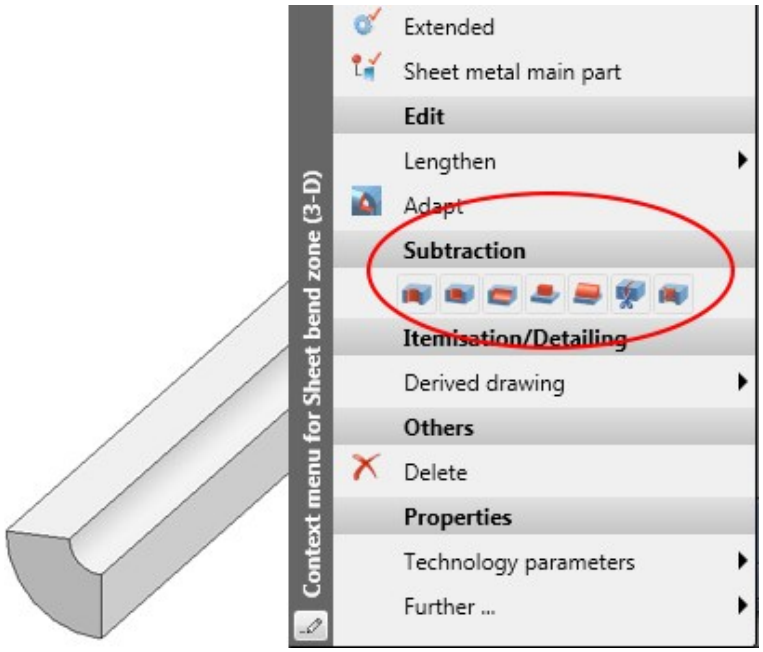
- DXF/DWG files
- XML files for CADMAN-B (from Version 7.5 onwards) byLVD or
- GEO files for ToPs GEO

Functions		Description
	**Export individual development**	This function exports a development - either completely or as section contour if you have chosen the DXF/DWG output. Also, you can export the sheet developments from HiCAD as XML file for CADMAN-B (from Version 7.5 onwards) by LVD. A prerequisite for the output of LVD data is the assigning of bending tools to the bend zone with the **Bend zone tooling**  . If you have chosen GEO as output format, the development will be exported from HiCAD to the ToPs GEO file format. The GEO data can be directly loaded into the individual modules (Laser etc.) of the ToPs family. The configuration of the file name takes place via the file SheetToDXF.FTD. You can customize this file in the Annotation Editor if desired. If this does not result in a proper name, the name will be "Sheet".
Export		
	**Develop and export sheets**	Use this function to export the section contour or the complete development of Sheet Metal parts or Steel Engineering plates if you have chosen the DXF/DWG output. In the process, a temporarily created development for the export will always be output. Also, you can export the sheet developments from HiCAD as XML file for CADMAN-B (from Version 7.5 onwards) by LVD. A prerequisite for the output of LVD data is the assigning of bending tools to the bend zone with the **Bend zone tooling**  . If you have chosen GEO as output format, the development will be exported from HiCAD to the ToPs GEO file format. The GEO data can be directly loaded into the individual modules (Laser etc.) of the ToPs family. The **Close & Report** button activates the Report Manager and creates a BOM. All attributes will be transferred to the Report Manager (as with drawing creation) and, additionally, the file name and the file path of the export. You can configure a RMS file with the Report Manager, determining which data are to be written.
	**Export sheet developments**	Use this function to export multiple developments with different parameter settings, e.g. LVD and Bystronic, of one or different Sheet Metal parts. You can choose between the following formats: ■ DXF/DWG, ■ XML for CADMAN-B (from Version 7.5) by LVD and ■ GEO for (ToPs GEO) A prerequisite for the output of LVD data is the assigning of bending tools to the bend zone with the **Bend zone tooling**  . If you have chosen GEO as output format, the development will be exported from HiCAD to the ToPs GEO file format. The GEO data can be directly loaded into the individual modules (Laser etc.) of the ToPs family. The **Close & Report** button activates the Report Manager and creates a BOM. All attributes will be transferred to the Report Manager (as with drawing creation) and, additionally, the file name and the file path of the export. You can configure a RMS file with the Report Manager, determining which data are to be written.

Enhanced context menu for bend zones

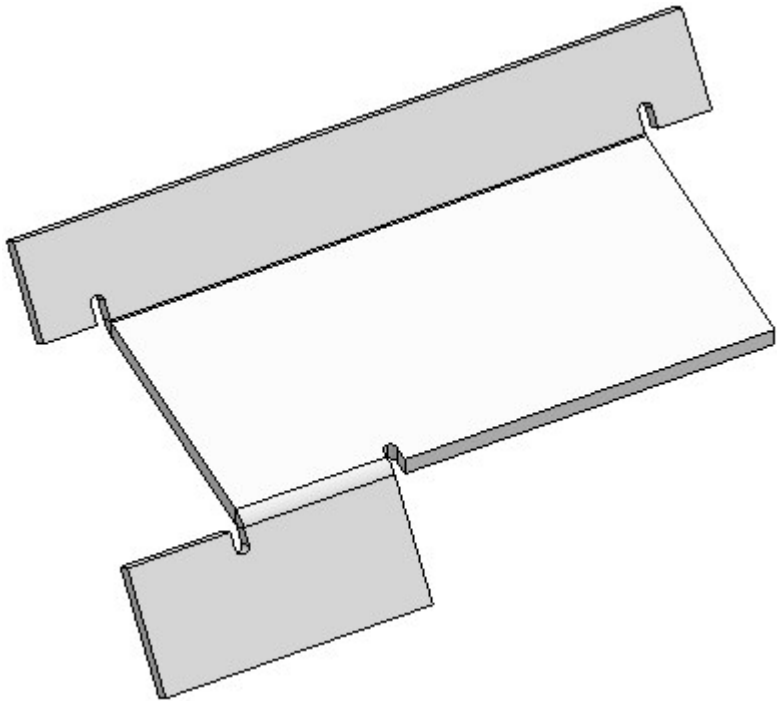
The context menu for bend zones has been expanded, now offering new Subtract and Divide functions. This allows you to apply straight cuts to or divide bend zones without having to switch to the functions on the 3-D Standard tab.

Right-click on a bend zone to open the context menu.



Attach long flange to short flange, with relief groove

When attaching flanges a relief groove is now applied to the new flange if it projects beyond the connecting edge.



Handling of collinear edges

Collinear sheet edges can no longer be shown in the HiddenLine representation. Therefore, the following setting options have been removed:

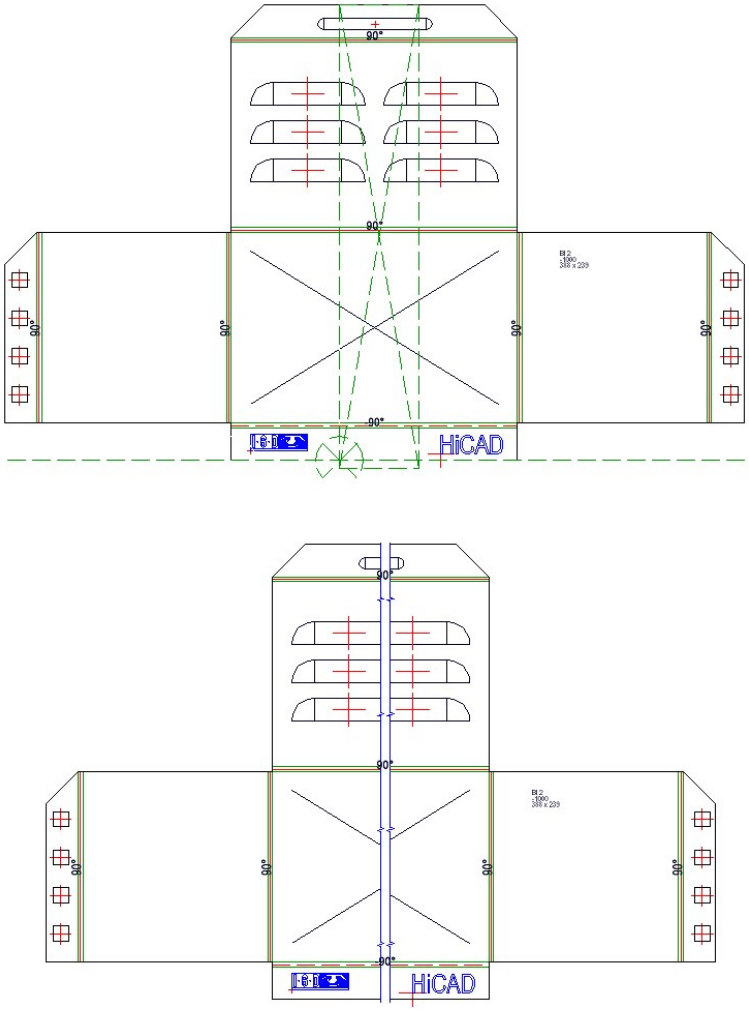
- in HiCAD at **Views > Properties > Show/Hide elements > Show collinear sheet edges** and
- in the Configuration Editor at **Drawing >Views > Hiding and overlapping of edges > Hide collinear sheet edges**.

For model drawings from 2018 and older you can no longer show collinear sheet edges in the HiddenLine representation. For these legacy drawings, the settings valid at the time of their saving apply. However, you have still the option to update the handling of collinear sheet edges in these legacy drawings to the new mechanism.

To do this, use the **Update sheet edges**  function. After applying the function you can hide collinear-tangential sheet edges in the HiddenLine representation if desired. Collinear bent edges will always be shown, while collinear mergeable edges will be hidden.

Preserve bend zone texts in shortened views

Bend zone texts will now be preserved in shortened views of sheet developments.



Lengthen flange or sheet

To improve user-friendliness, the user guidance texts for the lengthening of flanges and sheets in the info bar at the bottom left of the HiCAD screen have been revised.

Previously: 1st point; New: Vector of lengthening: Start point

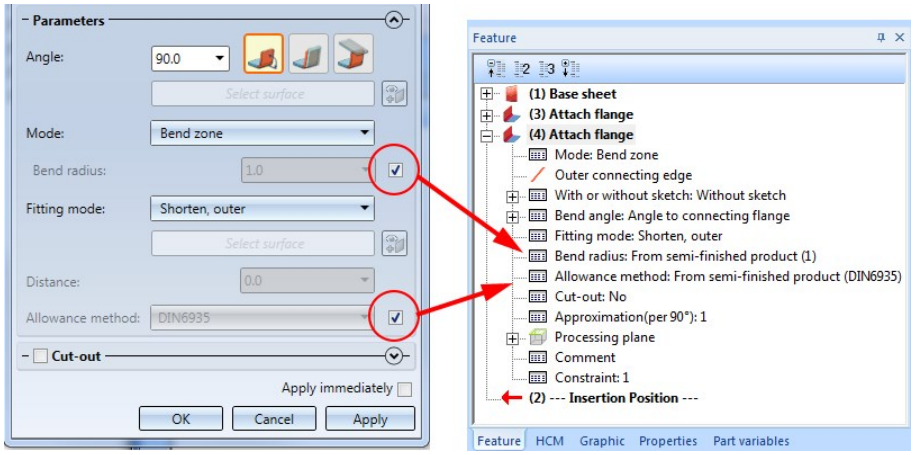
Previously: 2nd point; New: Vector of lengthening: End point

Allowance method and bend radius in Feature log - Indicate semi-finished product

In some Sheet Metal function dialogue windows you can activate a **From semi-finished product** option for bend radius and allowance method. If you have selected these options, you can see the relevant value in the Feature log.

This is possible for the following functions:

- Attach flange
- Sheet along sketch
- Flange along sketch



Outer contours of sheets, dimension chains separated according to flanges

Steel Engineering

Please note:

The functions of the **Itemisation/Detailing** function group on the **Drawing** tab are available in all HiCAD suites and modules.
From HiCAD 2018 onwards you will therefore find the descriptions of these functions in the Basics > Itemisation/Detailing topic.

News regarding these functions can be found in the Basics - What's New? topic, accordingly.

Service Pack 2 2019 Patch 1 (V 2402.1)

Railings on curved beams

Curved segments with equal radius and mid point are now handled like aligned, straight beams. That is, if there are several curved beam segments with equal radius and mid point in succession, the railing elements of these beams will be combined into one segment assembly. In this way, continuous hand rails and knee rails will be created on these beams, too. The updating and modification of existing railings (before Version 2019 SP2 Patch 1) with such areas continues to take place with individual segment assemblies, i.e. non-continuous, "interrupted" hand rails and knee rails.

In the image below a railing along two curved beams (1) and (2) has been created. **A** shows the result with a version before 2019 SP2, Patch1, **B** shows the result with Patch 1.


A
(1)
(2)
B

Service Pack 2 2019 (V 2402)

Configuration / Settings

Changed parameter configuration for Steel Engineering

The HiCAD Parameter configuration for Steel/Metal Engineering have been changed, with the aim to make Steel Engineering workshop drawings clearer and better intelligible.

The following modifications have been applied:

BOMs

Setting	previous	as of SP2	Parameters
Insertion of BOMs	Quantity list in view group	Quantity list in drawing frame	Changed in Configuration Editor for all Usages **Automatic drawing derivation > Production drawing > Usage-dependent > ... > View group > Insert BOM**
Position of BOMs in drawing frame	bottom right	bottom left	Changed in Configuration Editor for all Usages **Automatic drawing derivation > Production drawing > Usage-dependent > ... > View group > BOM: Position in view group**
Display of columns DSTV, Part type, Designation	yes	no	Changed in the **hicad_stahlbau.rms** in the HiCAD directories sys and sys/BOMTemplates. During installation the template sys/BOMTemplates/hicad_stahlbau.2400.0.rms will be copied to sys/hicad_stahlbau.rms.

Font size

Setting	previous	as of SP2	Parameters
Item numbers of sub-parts	5	3.5	Adjusted in the templates ofthe FTD file for annotations
Height of weld symbols	3.5	2	Changed in Configuration Editor **Drawing > Annotations > Weld symbols > Symbol height**
Identificatiopn of sections	5	4	Changed in Configuration Editor **Drawing > Views > Ident > Sections > Font height in original view**

View shortening

Setting	previous	as of SP2	Parameters
Minimum width of shortening area	20	50	Changed in Configuration Editor for all Usages except for Steel Engineering plates **Automatic drawing derivation > Production drawing > Usage-dependent > ... > Views > Shortened view > Minimum width of shortening area**
Distance to relevant geometry	5	15	Changed in Configuration Editor for all Usages except for Steel Engineering plates

Setting	previous	as of SP2	Parameters
			Automatic drawing derivation > Production drawing > Usage-dependent > ... > Views > Shortened view > Distance to relevant geometry
Shorten other parts and assemblies		ja	Changed in Configuration Editor for all Usages except for Steel Engineering plates Automatic drawing derivation > Production drawing > Usage-dependent > ... > Views > Shortened view > Other parts and assemblies

Dimensions

Setting	previous	as of SP2	Parameter
Distance between dimension line and geometry	10	9	The data are created from the file STW_DimSettimgs.xml in the HiCAD sys directory. This file will be created by HiCAD if it does not exist yet (i.e. during a new installation). In the created new file the values will be initialized as shown here. For update installations the parameters will remain unchanged, i.e. as defined by the user.
Distance between dimension lines	8	7	

New parameters for section schemas in BOMs

In the Configuration Editor at Steel Engineering, two new parameters for section schemas in BOMs are now available:

Cutting angle reference in section schema

Here you can choose to which leg a specified angle in the section schema is to refer:

- Cross-section or
- Beam axis

Do not show cutting angle in section schema if cut surface

Use this parameter to specify the conditions under which no cutting angle is to be shown in the section schema, since the total length does not correspond to the trimmed length:

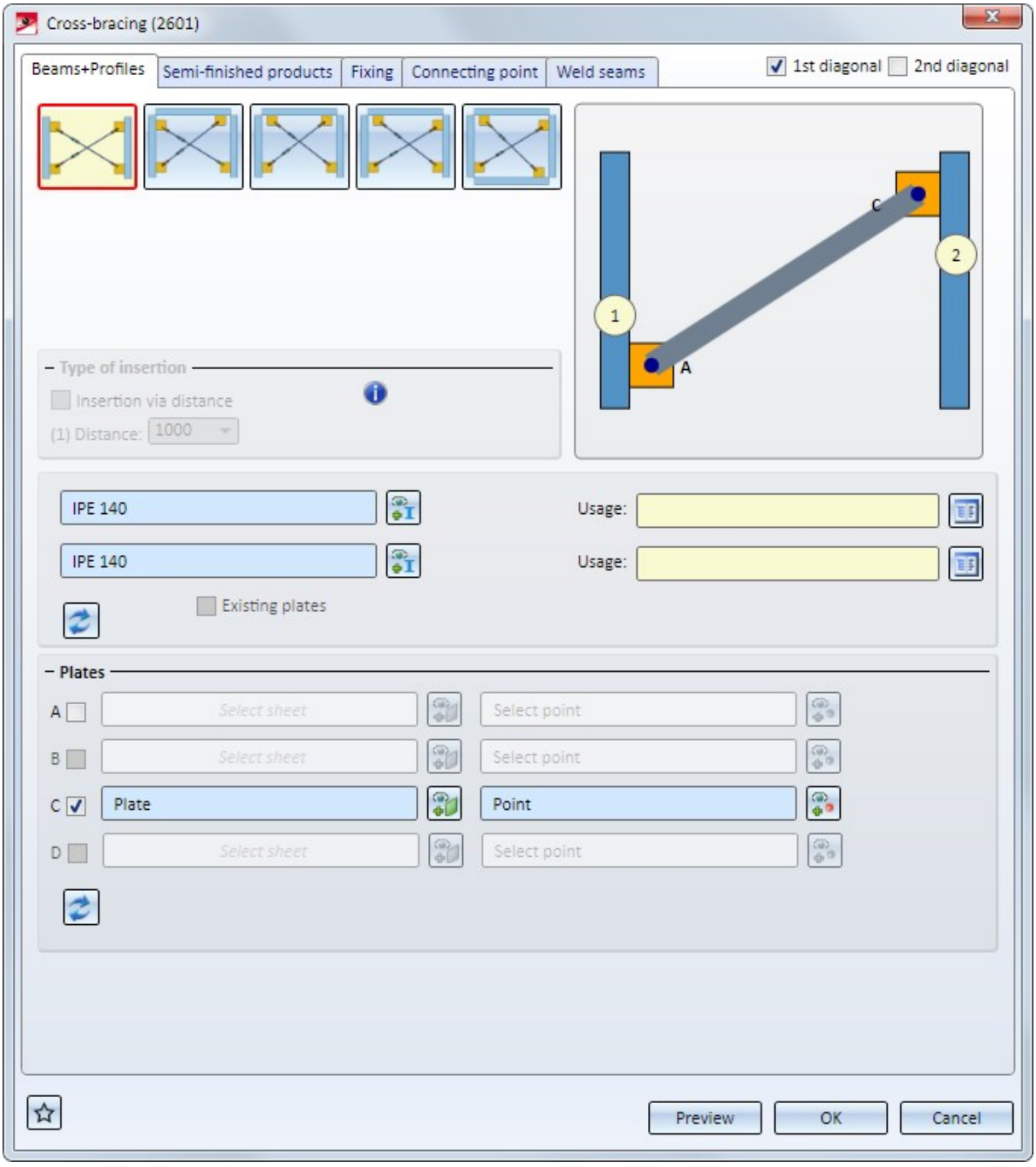
- The cut surface has been processed.
- The cut surface has been processed on one outer side or on both outer sides.

Connections

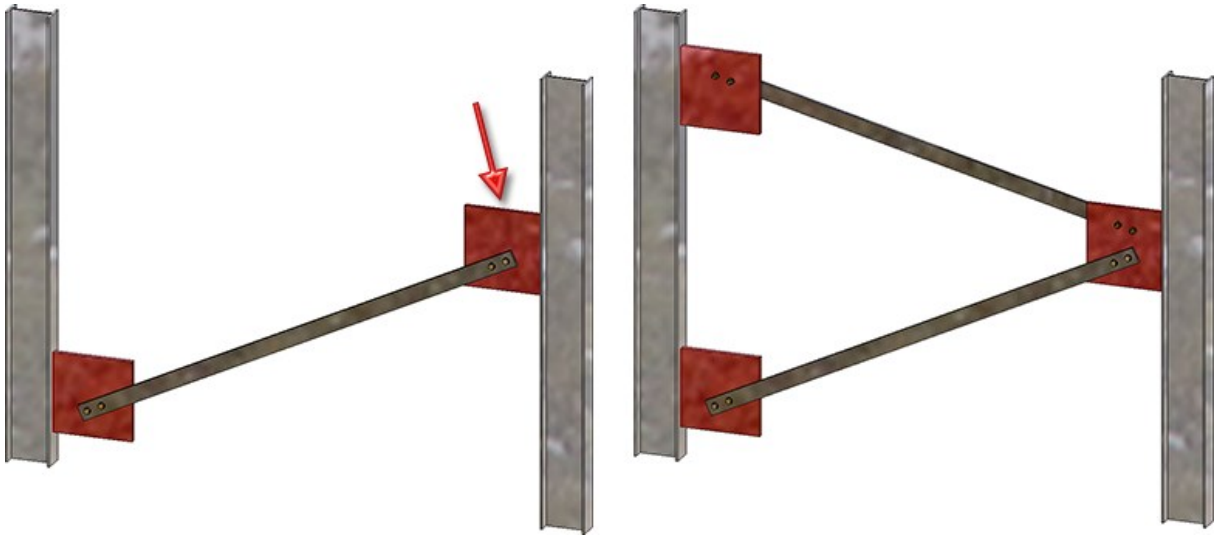
Cross-bracing (2601)

The previous Cross-bracing (1601) has had a redesign (analogous to Cross-bracing (2602)) and has been replaced with the new **Cross-bracing (2601)**. Use this Design Variant to insert cross-bracings as stabilizing elements between 2, 3 or 4 beams or plates.

When connecting profiles, you also have the option to mount the cross-bracing to already existing plates on the beams. This can make sense if, for instance, you want to insert two cross-bracings between 2 beams, and the second cross-bracing is to be mounted to the gusset plates of the first cross-bracing. Also new is the possibility to insert cross-bracings between plates (without assignment to any beams.



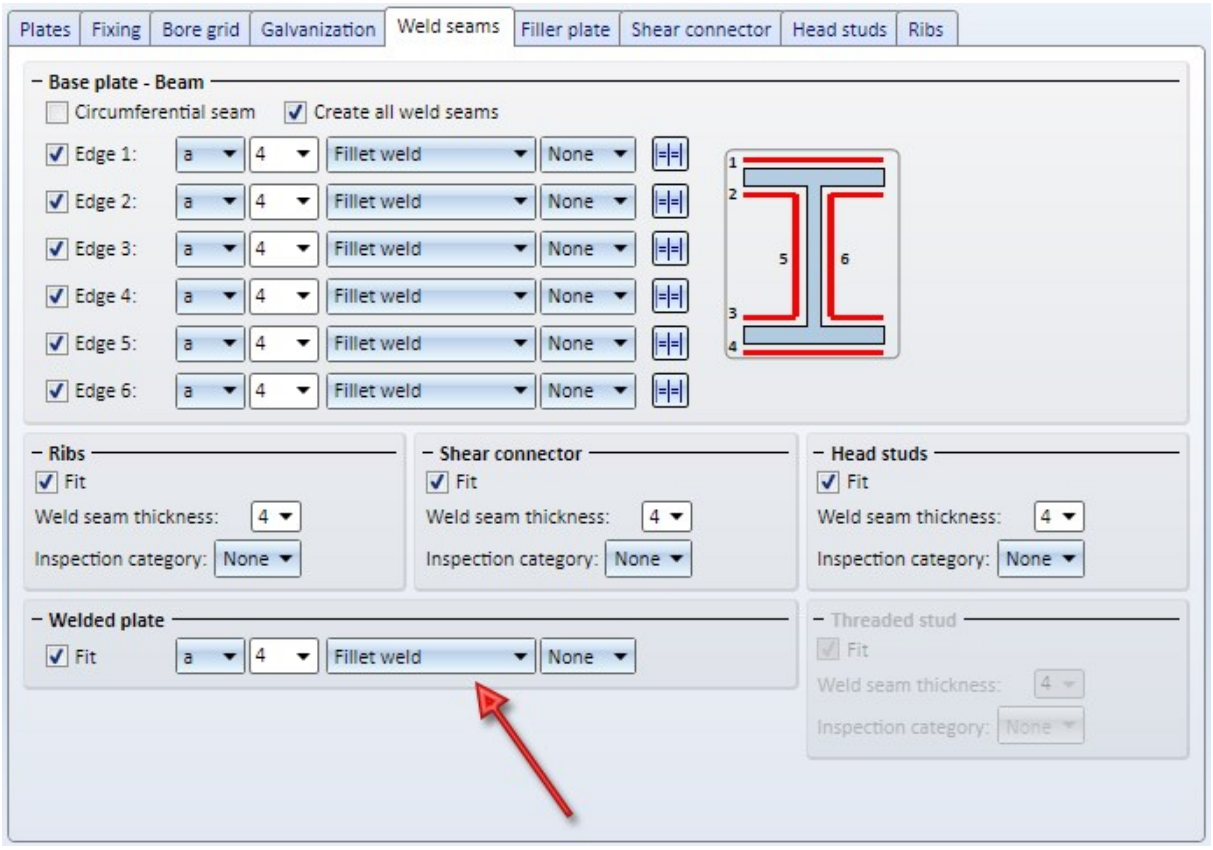
On the left hand side you can see a cross-bracing inserted between two beams. The gusset plates here were created together with the connection. Then, as you can see on the right hand side, another cross-bracing was inserted between the beams, mounted to the right gusset plate of the first cross-bracing (C).



The **Usage** attribute of the resulting assembly will be automatically initialized with **Cross-bracing**.

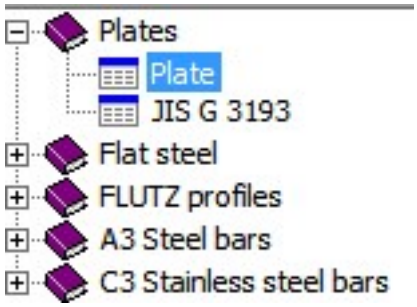
Base plate + Anchor plate (2101)

On the **Weld seams** tab of the **Base plate + Anchor plate (2101)** dialogue window you can now choose the weld seam type for welded on plates. The tab has been expanded accordingly for this purpose.



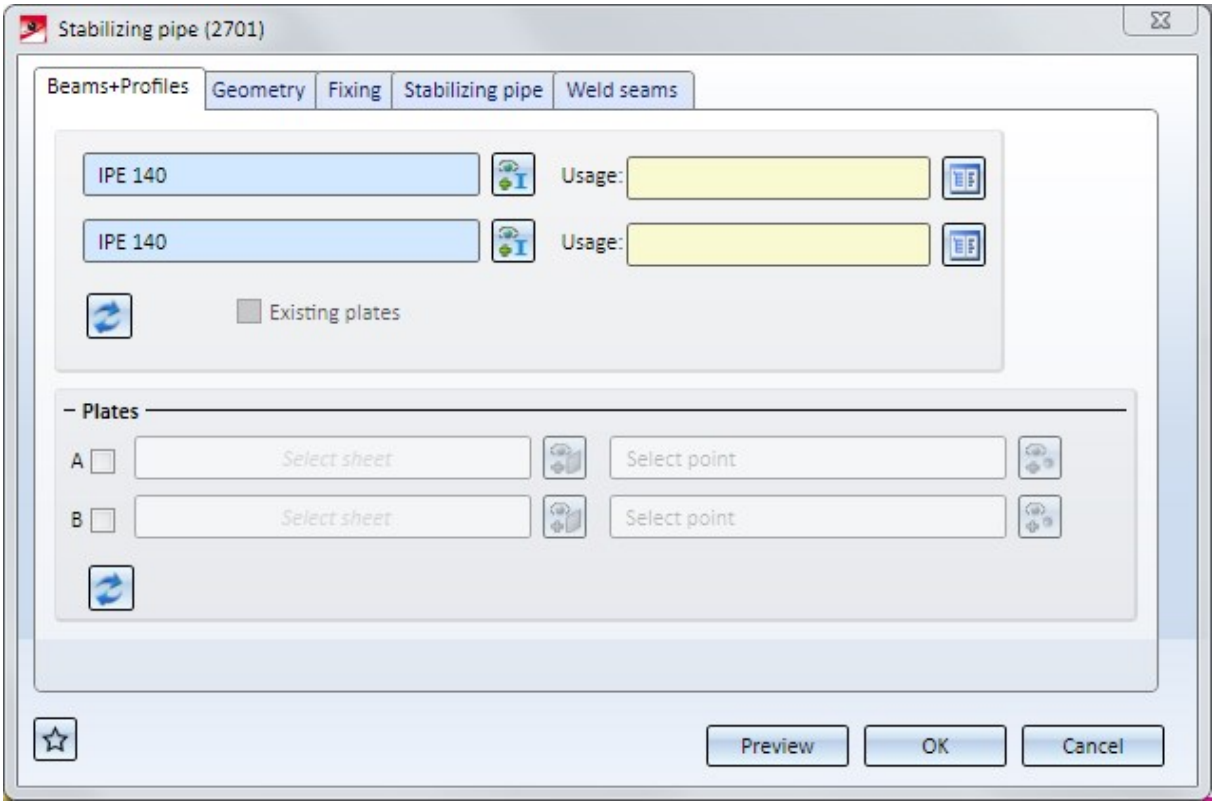
Cross-bracing (2602)

In previous versions,only plates could be selected as connecting plates. FromHiCAD 2019 SP2 onwards it is also possible to generate flat steels and wide steels.

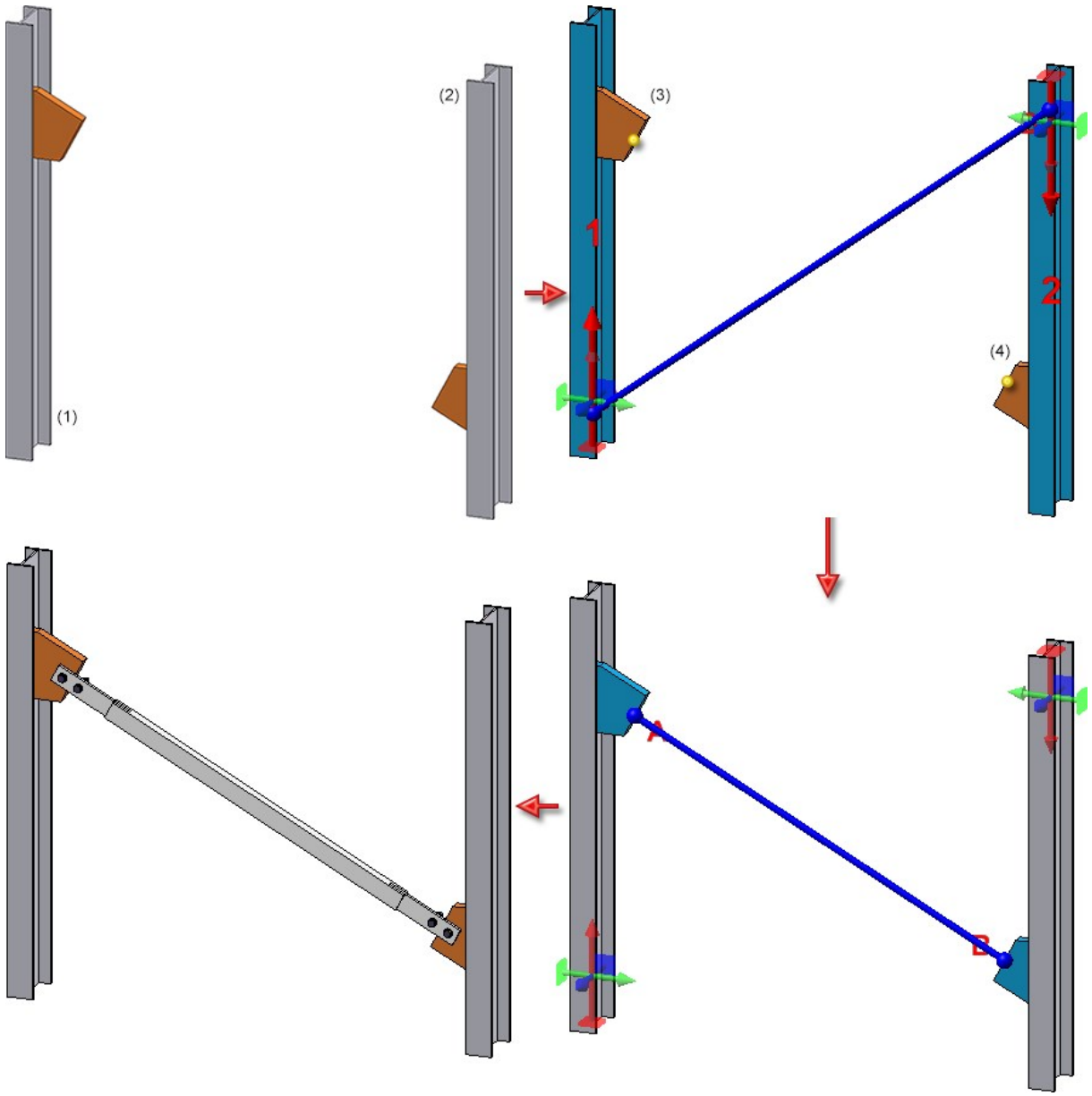


Stabilizing pipe

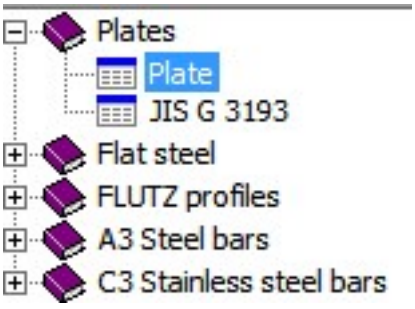
- As with the cross-bracings, you have now the option to use plates as gusset plates that already exist on a beam for stabilizing pipes, or to create a stabilizing pipe connection between two existing plates (that do not belong to any beam). The tab has been expanded accordingly for this purpose.



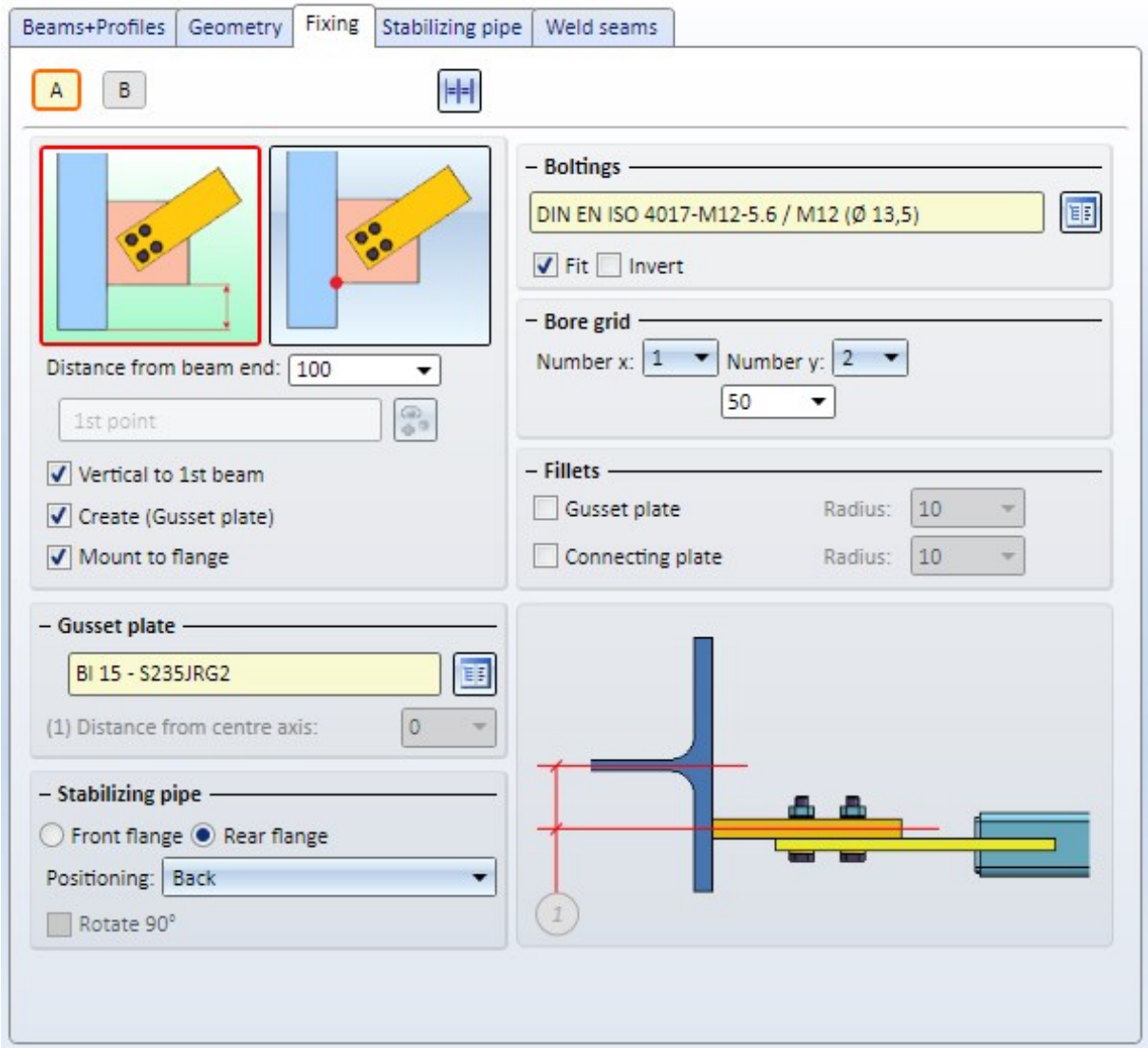
In the image below the two beams (1) und (2) were identified first. Then, in the dialogue window beneath **Plates**, checkbox **A** was activated. The gusset plate (3) on the first beam has been chosen as the plate and, as point, the mid point of the gusset plate edge. Next, it was proceeded likewise for Sheet B (4).



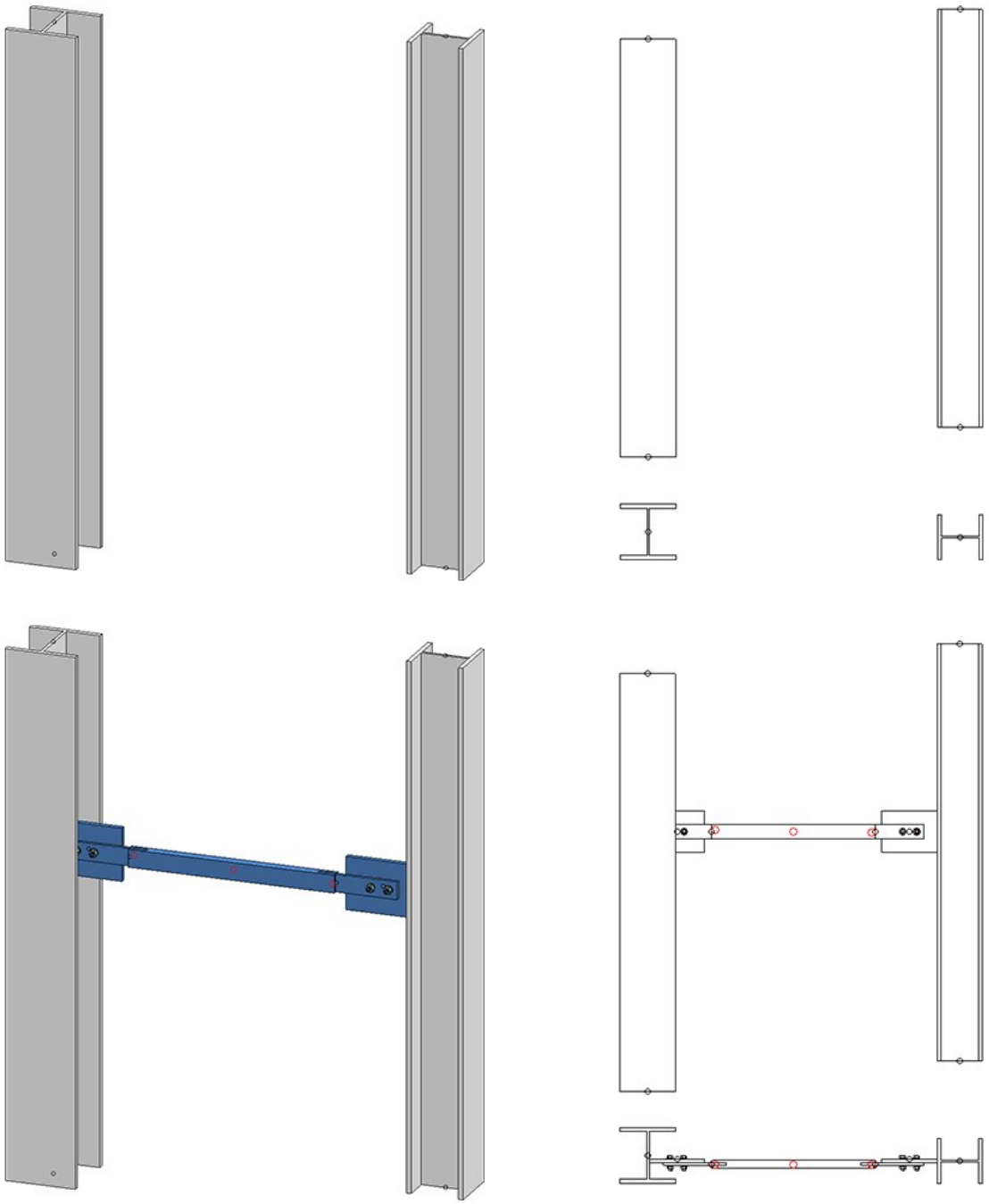
- As with the cross-bracing, the selection box for connecting plates has been expanded, i.e. here, too, connections on flat steels and wide steels can be generated.



- The insertion of gusset plates can now be deactivated if the stabilizing pipe is to be directly connected to the flange.


Beams+Profiles
Geometry
Fixing
Stabilizing pipe
Weld seams
A
B
Distance from beam end: 100
1st point
Vertical to 1st beam
Create (Gusset plate)
Mount to flange
Boltings
DIN EN ISO 4017-M12-5.6 / M12 (Ø 13,5)
Fit
Invert
Bore grid
Number x: 1
Number y: 2
50
Fillets
Gusset plate
Radius: 10
Connecting plate
Radius: 10
Gusset plate
BI 15 - S235JRG2
(1) Distance from centre axis: 0
Stabilizing pipe
Front flange
Rear flange
Positioning: Back
Rotate 90°
1

- For stabilizing pipe connections, a vertical alignment of gusset plates to the beam is now even possible if the beams have been offset to each other.

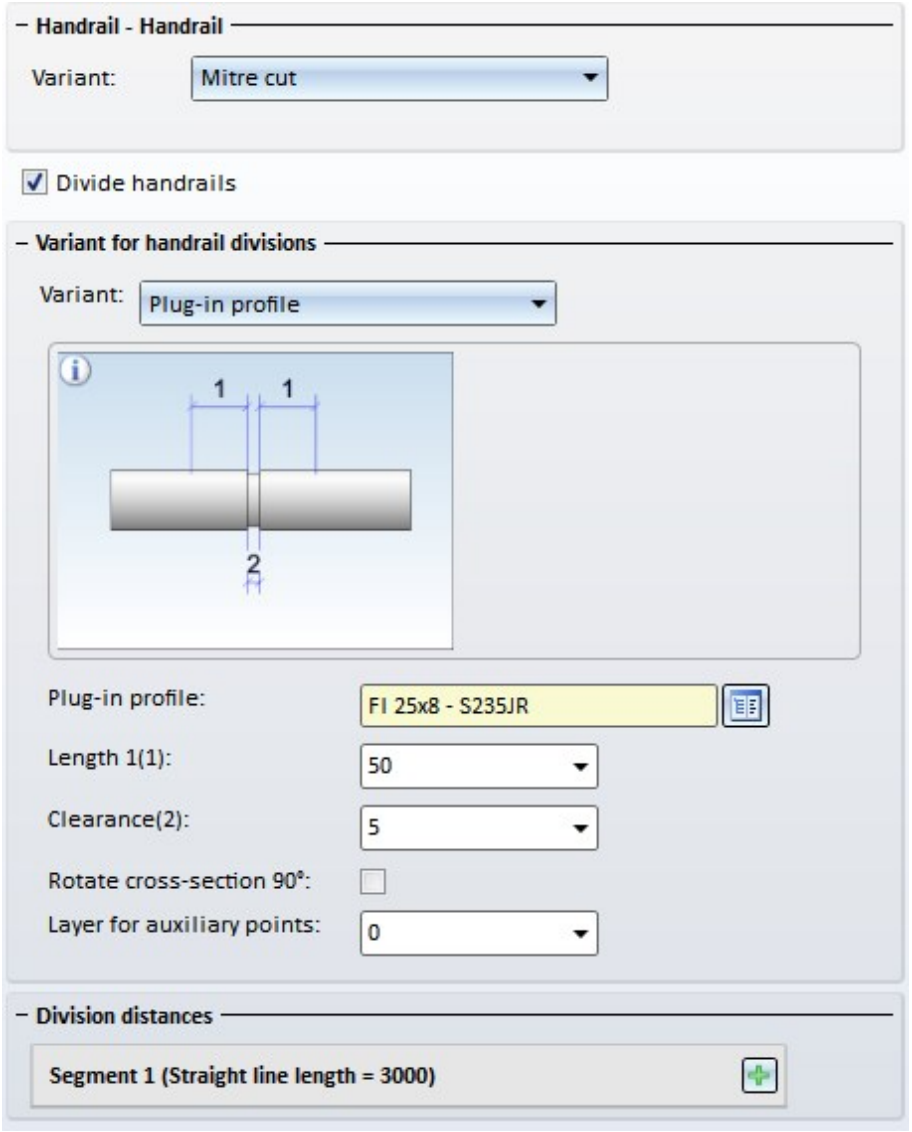


Stairs and railings

Dividing handrails

From SP2 onwards, handrails can be divided, e.g. in order to insert plug-in profiles or flat steel joints in long railings. For this purpose, an additional checkbox has been added on the **Handrail-Handrail** tab.

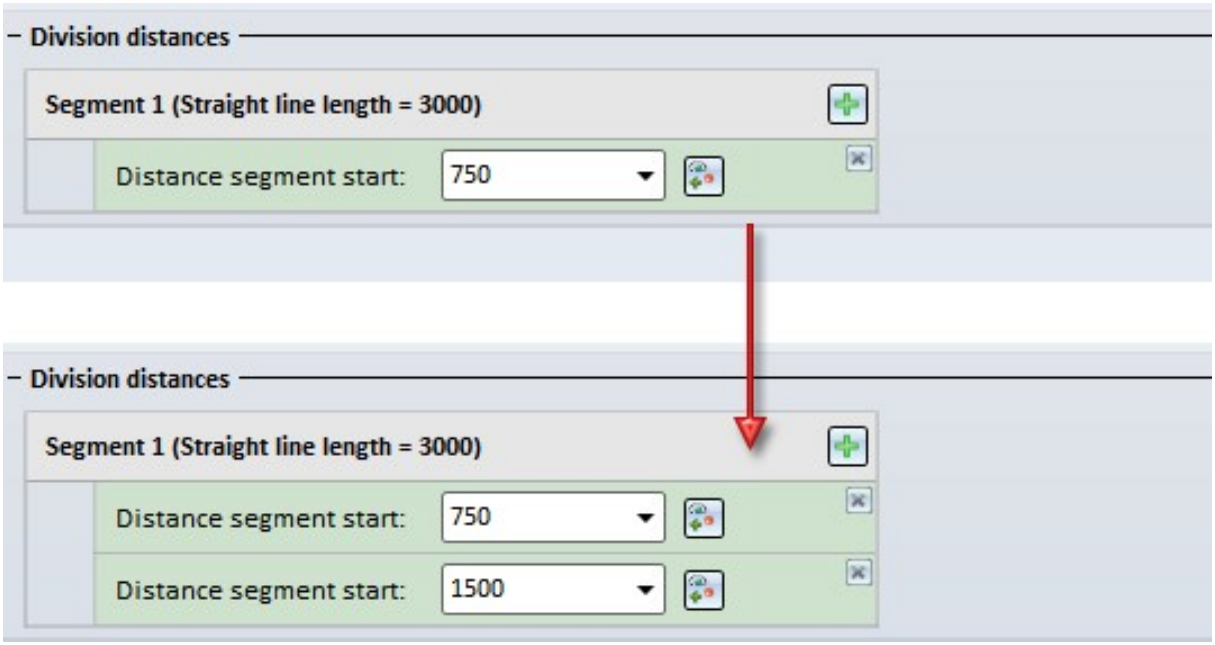
If the checkbox is active, the variant for handrail division and the division of the handrail can be specified by segments.



The divisions can be defined either by specifying the distance from the segment start, or by specifying a division point.

New division

Click on this symbol to add a new division, e.g.:



Enter the distance of the division from the segment start in the input field.

New division at selected point

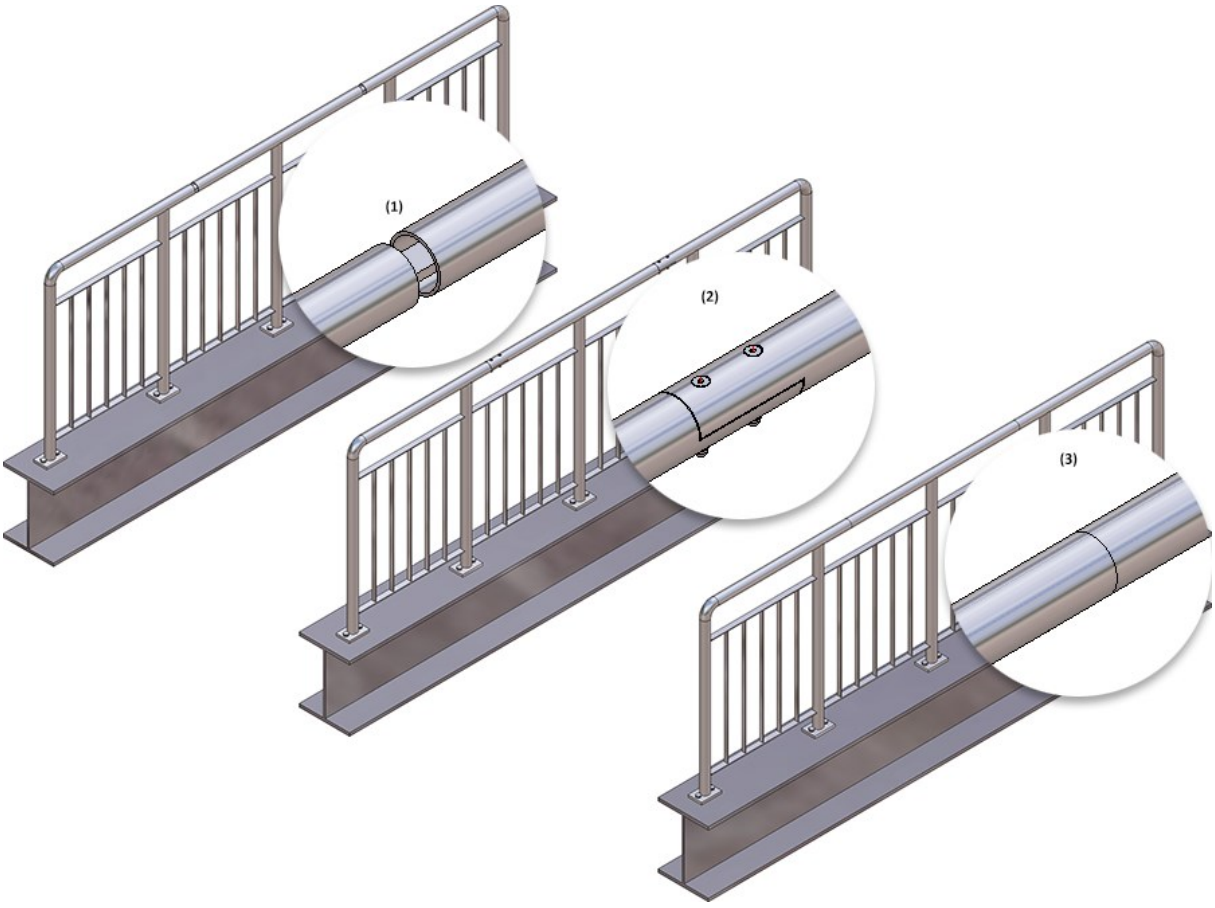
Click on this symbol do define the division by specifying a division point.

Divisions can be deleted at any time by clicking on the  symbol at the top right next to a division.

The following variants for handrail division have been predefined:

- Straight cut,
- Plug-in profile and
- Flat steel joint.

The image below shows a railing with divided handrails.

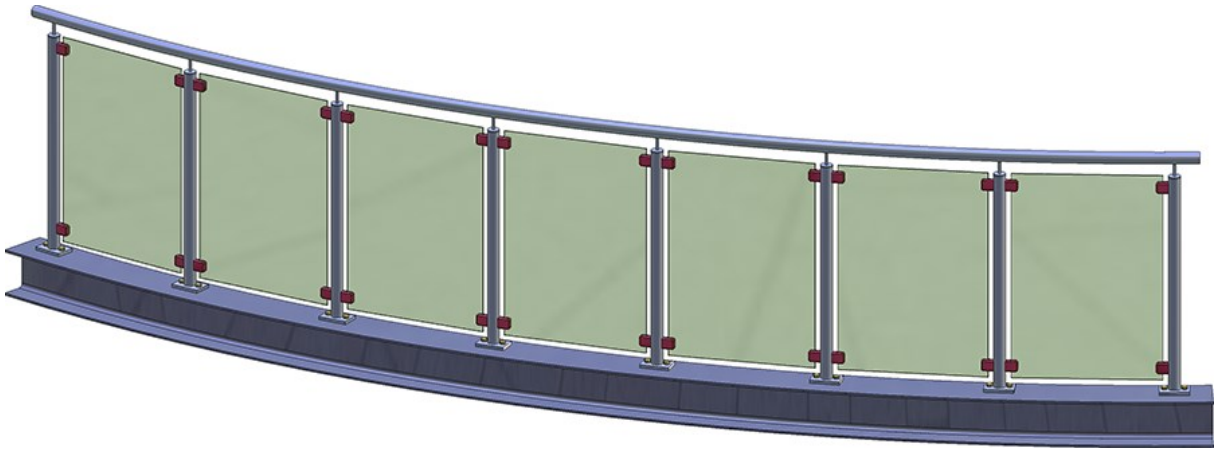


(1) Variant: Plug-in profile, (2) Variant: Flat steel joint, (3) Variant: Straight cut

Curved railings

From SP2 onwards, HiCAD allows the creation of curved railings in a plane. This applies to both railings along beams and railings along composite edges. However, this is only possible for curved beams or curved lines of a sketch. Curved beams placed along composite edges are currently not supported yet. The division can be defined either by specifying the distance from the segment start, or by specifying the division point.

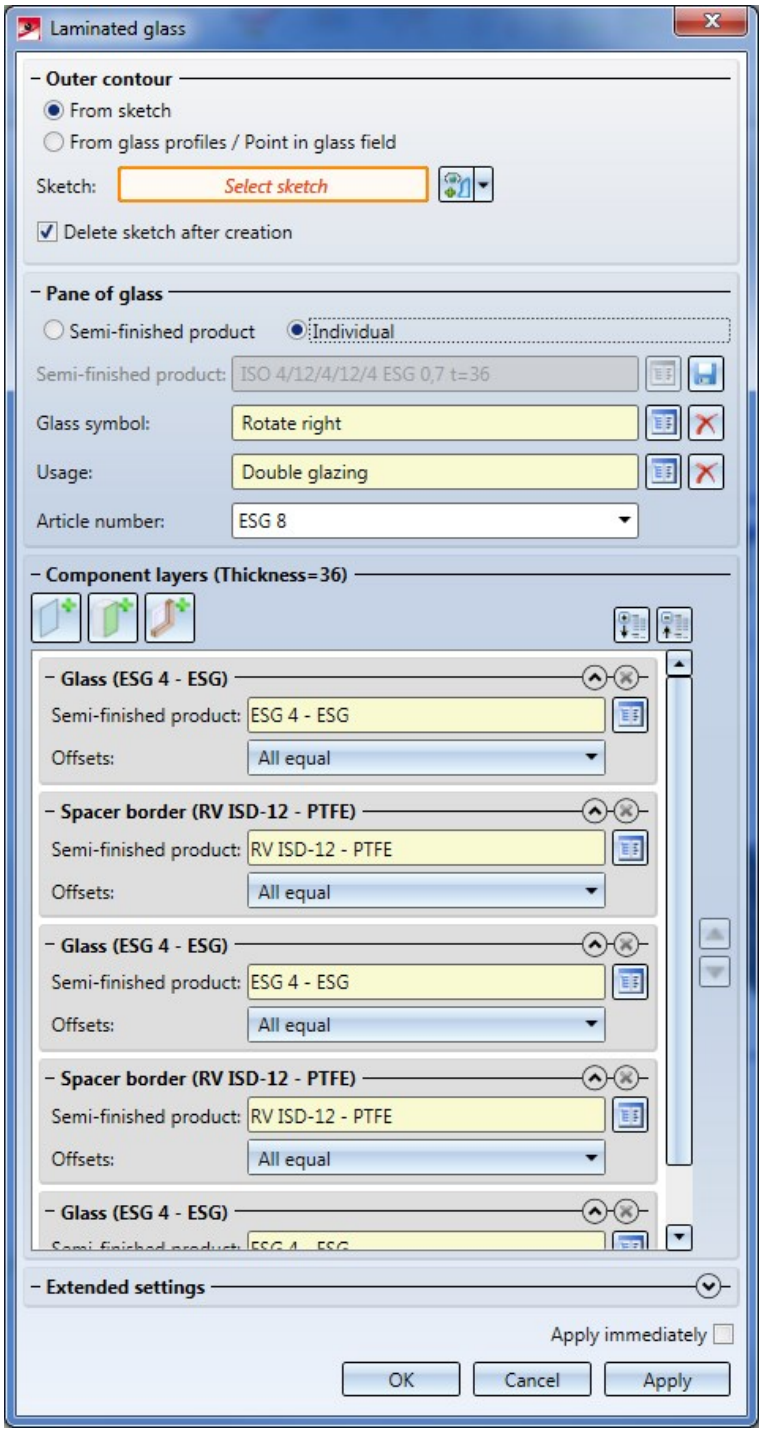
Curved railing, created on the basis of arcs



Curved railing, created along a curved beam

Glass insertion

The previous functions "Glass" and "Glass along sketch" have been combined into a new function called **Laminated glass**, which allows the creation of both glass panes in glass fields and the creation of glass panes from sketches.

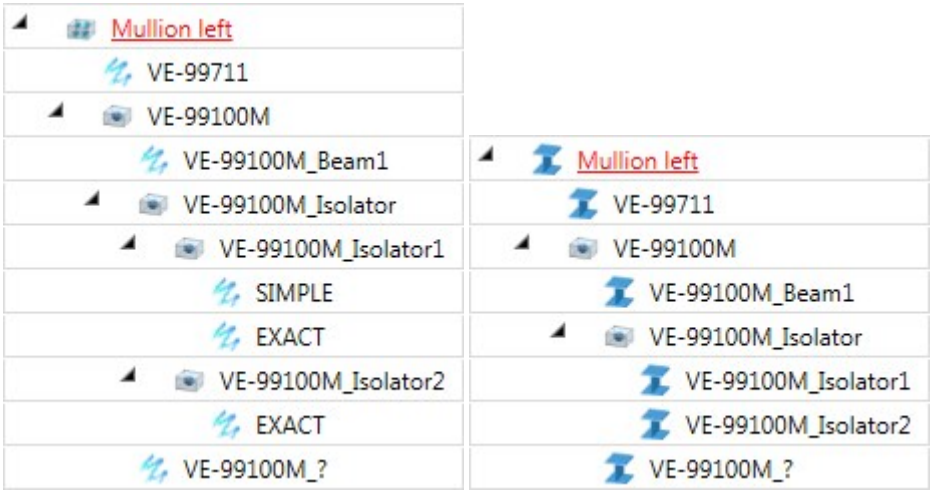


The Laminated glass dialogue window

Nested sketch-based series profiles

User-defined, sketch-based series beams and profiles consisting of several parts may now be nested arbitrarily in assemblies and dummy parts (exception: CUTOUT, CONTOUR and GLASS) - these must always be contained in

the uppermost element.



Nested beam structure and the resulting beam

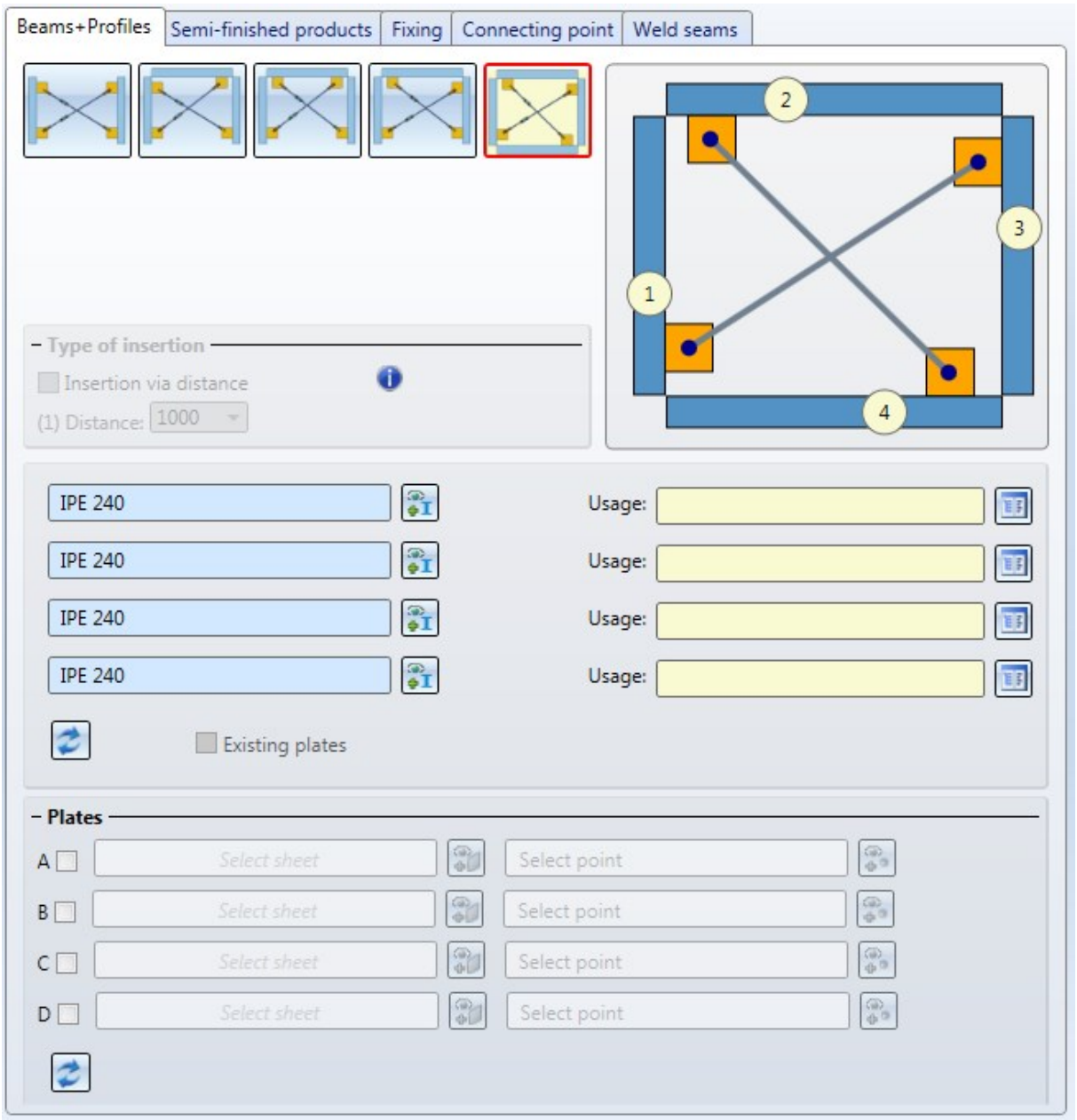
Service Pack 1 2019 (V 2401)

Connections

Cross-bracing (2602)

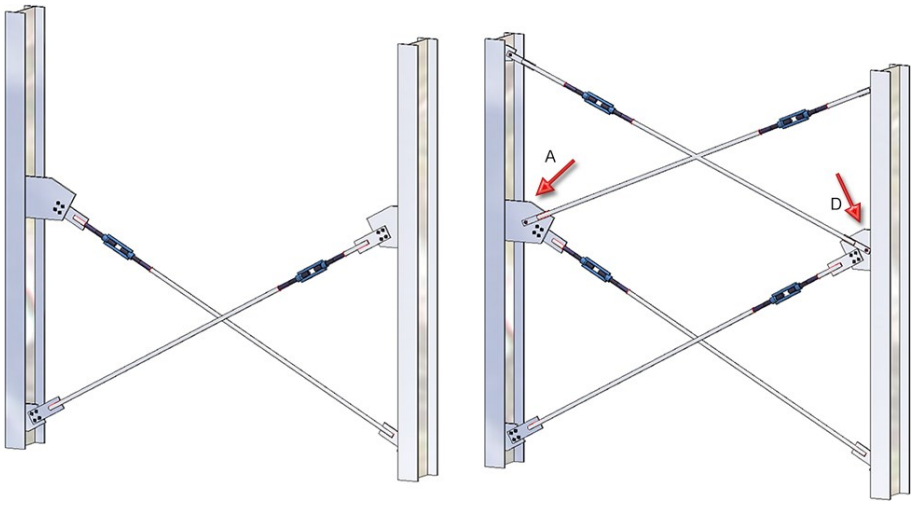
The **Cross-bracing (2602)** has been further enhanced. In previous versions, the connection automatically supplied the gusset plates. From HiCAD 2019 SP1 onwards you have the option to use already existing plates on the beams as gusset plates. This can be useful in situations where you want to insert two cross-bracings between two beams, and mount the second cross-bracing to the gusset plates of the first cross-bracing.

Also new is the option to insert a cross-bracing between plates (which are not assigned to any beam).

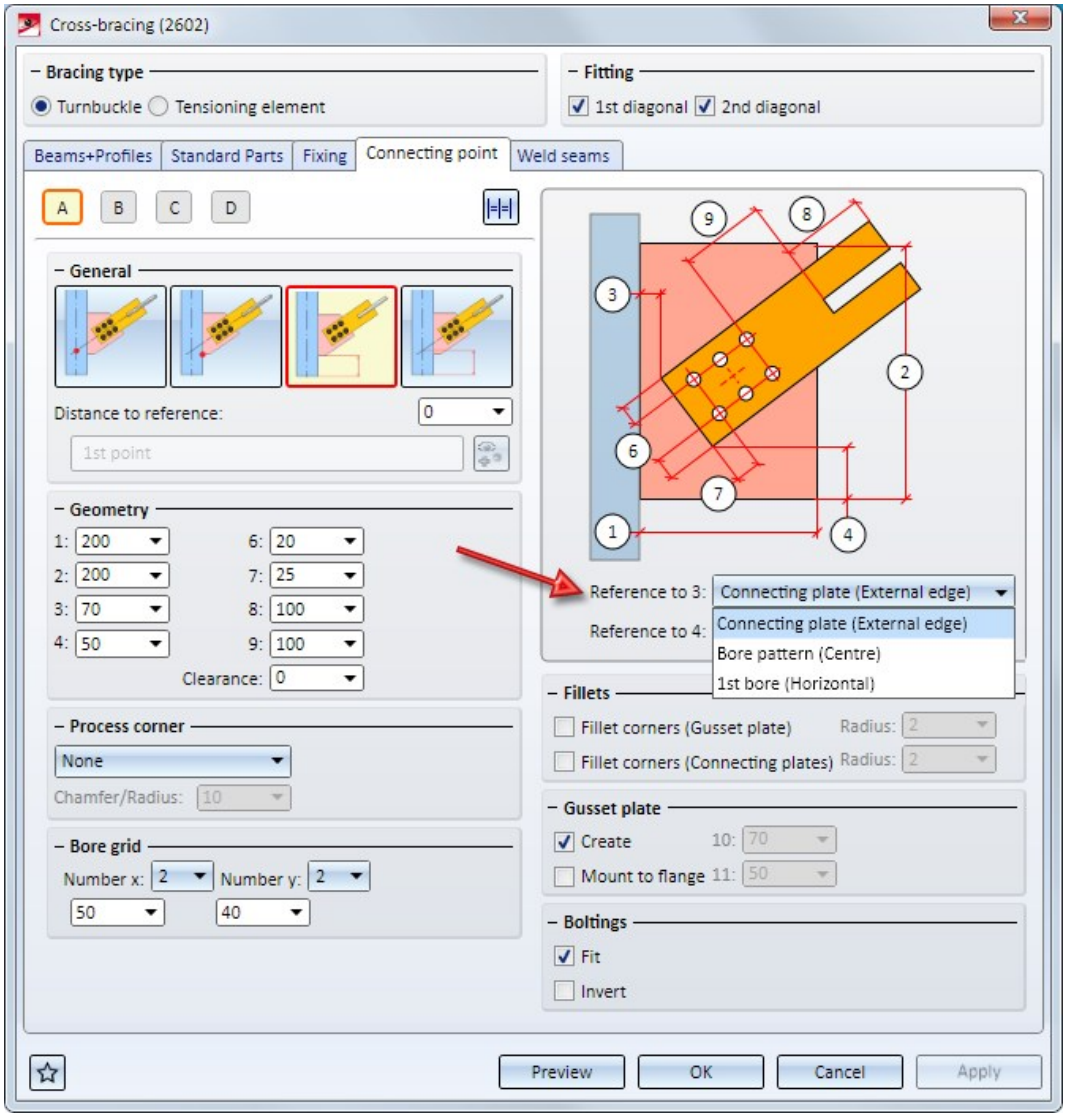


An example:

On the left hand side you see a cross-bracing inserted between two beams. The gusset plates here were created together with the connection. Then, as you can see on the right hand side, another cross-bracing was inserted between the beams, mounted to the gusset plates of the first cross-bracing (A and D).



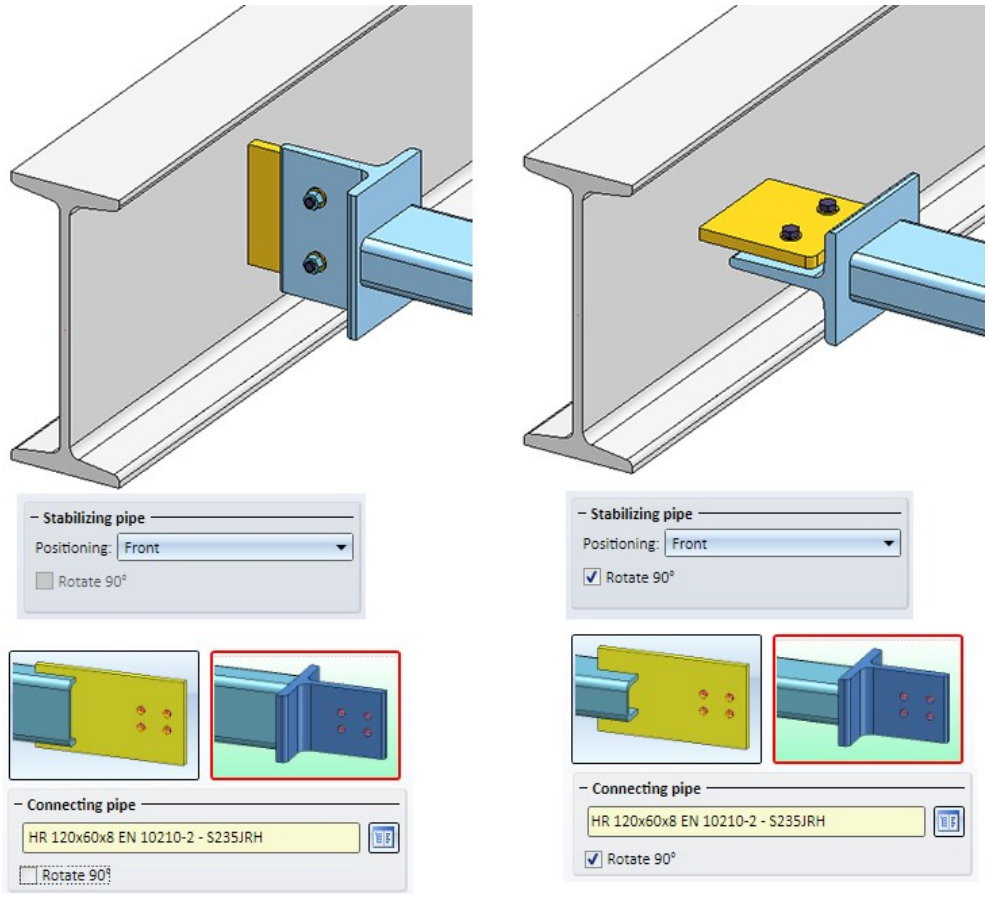
Furthermore, additional options are available for rectangular gusset plates, which influence the position of the connecting plate in relation to the gusset plate.


Cross-bracing (2602)
Bracing type
Turnbuckle
Tensioning element
Fitting
1st diagonal
2nd diagonal
Beams+Profiles
Standard Parts
Fixing
Connecting point
Weld seams
A
B
C
D
General
Distance to reference:
0
1st point
Geometry
1: 200
2: 200
3: 70
4: 50
6: 20
7: 25
8: 100
9: 100
Clearance: 0
Reference to 3:
Connecting plate (External edge)
Reference to 4:
Connecting plate (External edge)
Bore pattern (Centre)
1st bore (Horizontal)
Process corner
None
Chamfer/Radius: 10
Fillets
Fillet corners (Gusset plate)
Radius: 2
Fillet corners (Connecting plates)
Radius: 2
Gusset plate
Create
10: 70
Mount to flange
11: 50
Bore grid
Number x: 2
Number y: 2
50
40
Boltings
Fit
Invert
Preview
OK
Cancel
Apply

Stabilizing pipe

The **Fixing** tab of the **Stabilizing pipe** connection has been revised. Beneath **Stabilizing pipe** you can now determine whether the gusset plate is to be mounted to the front or back of the connecting sheet or T-profile, respectively. Also, you can rotate the connecting plate and T-profile 90° by activating the corresponding checkbox.

The image below shows a stabilizing pipe connection inserted perpendicular to the beam without (left) and with (right) 90° rotation of the gusset plate, the T-profile and the pipe.



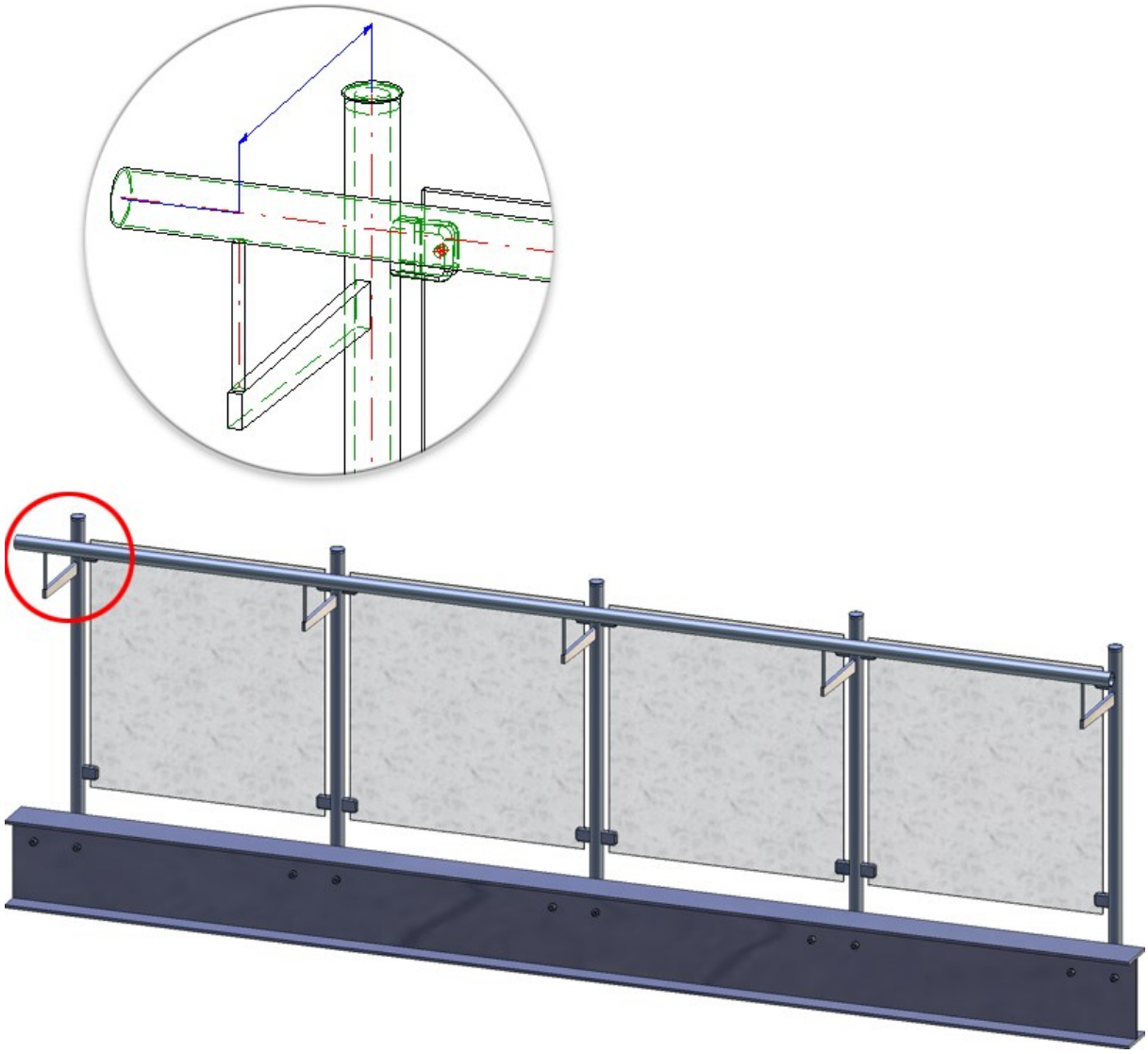
Stairs and Railings

Railings - Offset between post and handrail

For all posts you can now specify a lateral offset to the handrail. For this purpose a new checkbox has been added on the **Post** tab.

> Please note that specifying an offset currently only makes sense if **Variant: Console** has been selected on the **Post-Handrail** tab.

Example:

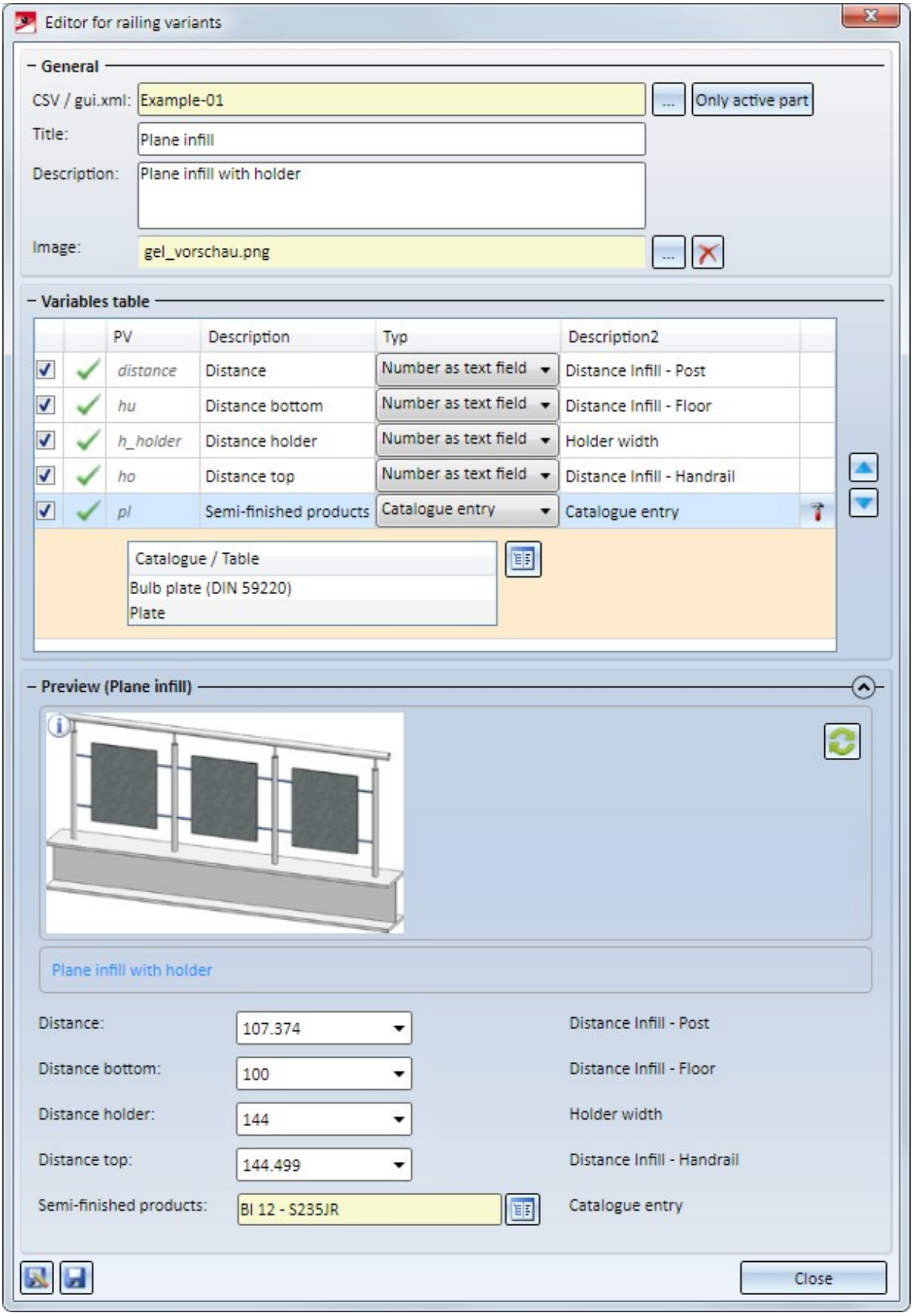


Editor for customer-specific railing variants

Railing components such as posts, handrails, transition infills, corner infills or the Post-Handrail connection can now also be saved as customer-specific variants. These variants will then be available in the Railing Configurator, allowing an individual composition of railings.

The creation of such variants requires the following steps:

- Construction of the desired railing component
- Parameterisation of the railing components by assigning ISD variables (mandatory) and free variables (optional) and - if desired - assigning of HCM constraints.
- Definition of the Fitting CS, and
- Saving of the variants with the **Variant Editor for Railings** that is available in the **Civil Engineering functions** at **Steel Engineering > Stairs+Railings > Railing**.


Editor for railing variants
General
CSV / gui.xml:
Example-01
Only active part
Title:
Plane infill
Description:
Plane infill with holder
Image:
gel_vorschau.png
Variables table
PV
Description
Typ
Description2
distance
Distance
Number as text field
Distance Infill - Post
hu
Distance bottom
Number as text field
Distance Infill - Floor
h_holder
Distance holder
Number as text field
Holder width
ho
Distance top
Number as text field
Distance Infill - Handrail
pl
Semi-finished products
Catalogue entry
Catalogue entry
Catalogue / Table
Bulb plate (DIN 59220)
Plate
Preview (Plane infill)
Plane infill with holder
Distance:
107.374
Distance Infill - Post
Distance bottom:
100
Distance Infill - Floor
Distance holder:
144
Holder width
Distance top:
144.499
Distance Infill - Handrail
Semi-finished products:
BI 12 - S235JR
Catalogue entry
Close

Other new features

Tracking of catalogue changes

If you have selected the default template **Steel/Metal Engineering** in the HiCAD Parameter Configuration (ParKonfigComp.exe) in previous versions, the tracking of catalogue changes was automatically switched on in the Configuration Editor at **System settings > catalogue**. From HiCAD 2019 SP1 onwards, the tracking will be switched off by default.

Specify default material

In the Configuration Editor at **Steel Engineering** you can specify a Default material for Steel Engineering parts.

In previous versions, the ID of the required table of the Material catalogue and the ID ofthe data record needed to be specified. From HiCAD 2019 SP1 onwards, the selection can be achieved much easier: Simply click on the  symbol and choose the desired material from the **Materials** or **User-defined materials** catalogue.

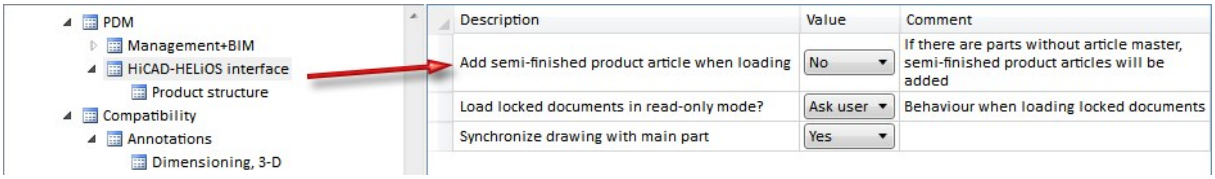
Major Release 2019 (V 2400)

Attached parts

Loading of the current article master of semi-finished products in attached parts

In previous versions, existing article masters of semi-finished products were not transferred to HiCAD when inserting attached parts such as Steel Engineering connections, stairs, railings, element installations, profile installations etc.

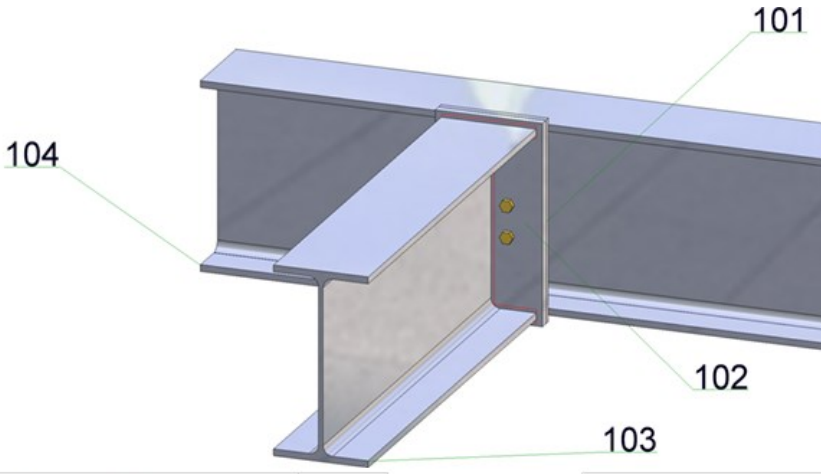
From HiCAD 2019 onwards this behaviour can be changed via a setting in the Configuration Editor: At **PDM > HiCAD-HELiOS Interface** you can find the parameter **Add semi-finished product article when loading**. The ISD default setting is **No**, i.e. the article master of semi-finished products will not be transferred.



For example, when transferring article masters of semi-finished products from user-defined catalogues, set the parameter to **Yes**.

Example:

In this example a new model drawing was created and entered into the database. Then, two IPE-beams were inserted and mounted together using the **Beam to web, with 2 plates and stiffener (1211)** connection. The plates **Bl 15** and **Bl 10** were chosen from the semi-finished products catalogue, and also have an article master in the HELiOS database. The image below shows, by means of the ICN structure, the difference between the different settings in the Configuration Editor:



Add semi-finished product article when loading: No

Add semi-finished product article when loading: Yes

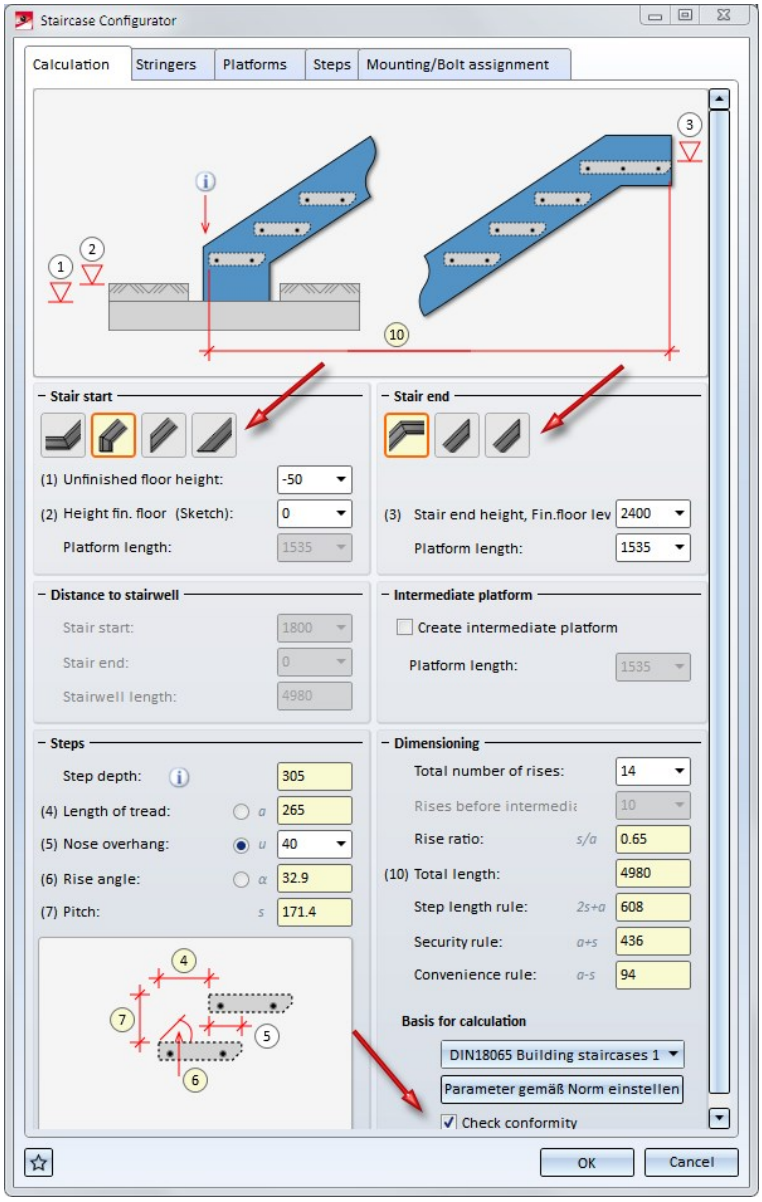
Designation	Item number	Comment
DN-004862		
SN-080572		Assembly
Assembly IPE 300	1	Assembly
SN-036552	100	IPE 300
Bl 15	103	Plates
Weld seam		
Assembly IPE 300	2	Assembly
SN-036552	101	IPE 300
Bl 10	102	Plates
Bl 10	104	Plates
Weld seam		
Loose parts		Structure assembly
Bolting		

Designation	Item number	Comment
DN-004862		
SN-080572		Assembly
Assembly IPE 300	1	Assembly
SN-036552	100	IPE 300
SN-079783	103	Plates
Weld seam		
Assembly IPE 300	2	Assembly
SN-036552	101	IPE 300
SN-079780	102	Plates
SN-079779	104	Plates
Weld seam		
Loose parts		Structure assembly
Bolting		

Stairs and railings

Enhanced Staircase Configurator dialogue window

The **Calculation** tab of the Staircase Configurator is now much more intuitive. The different types of stair starts and ends can now be chosen via symbols. The info graphic of the staircase in the dialogue window will be automatically adjusted in the process.



Also new is the **Check standard conformity** checkbox. If it is active, HiCAD will check on the basis of the specified parameters whether the staircase can be constructed, taking the calculation rules into consideration. If this is not the case, the symbol will be displayed at the **OK** button. If you move the cursor over the symbol, the tab and the input fields containing incorrect values will be marked with the symbol. Move your cursor onto this symbol to obtain further information and correct the staircase parameter. You can also click on the **Set parameters according to standard** button to correct the wrong values automatically.

Concrete stairs

Concrete stairs can no longer be inserted as sub-parts. The corresponding checkbox has been removed from the dialogue window.

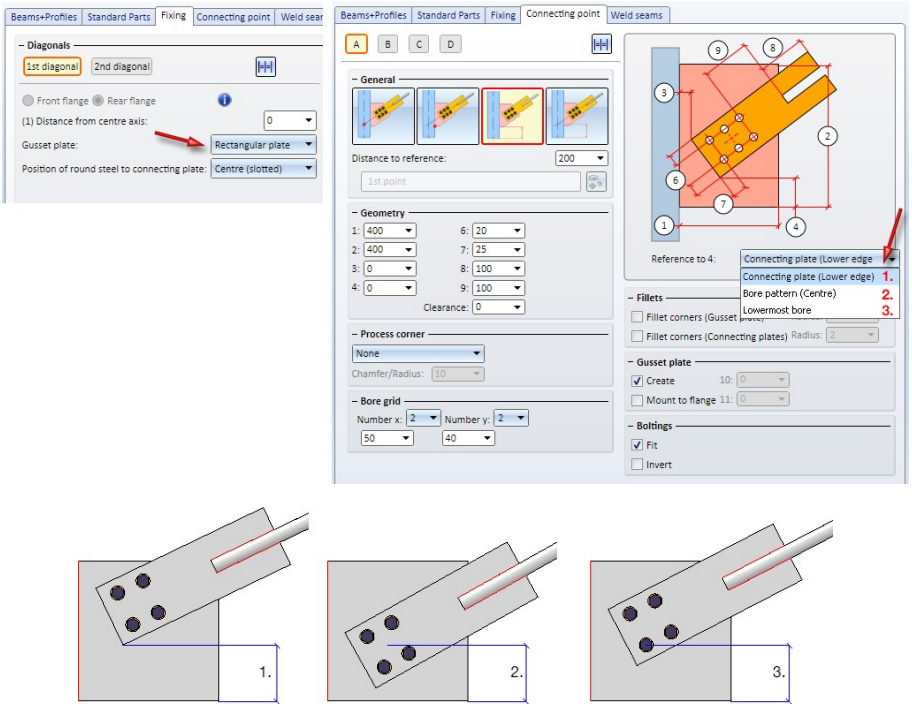
Connections

Free plate (1103) - Part structure

- If the part that is active when you call the Free plate (1103) function is already a sub-part to an assembly, the plate will be inserted as a sub-part to this assembly. Otherwise, the plate will be inserted as a main part in the drawing.
- The **Insertion as sub-part** checkbox has been renamed to **Insertion as attached part**. If this checkbox is active, HiCAD will proceed as follows with regard to the part structure:
  - If the active part already belongs to a BOM-relevant assembly, the plate will be assigned as a sub-part to this assembly.
  - If the active part does not belong to a BOM-relevant assembly, a new BOM-relevant assembly will be created, to which the active part and the and the plate will be subordinated.

Cross-bracing (2602) - Position of connection of gusset plate

If you choose a rectangular plate as gusset plate on the **Fixing** tab of the Cross-bracing (2602) dialogue window, various selection options on the **Connecting point** tab will be available to you, allowing you to influence the position of the connection on the gusset plate.



This enables you to determine the centre of the connecting sheet and place the bore in the middle of the gusset plate more easily.

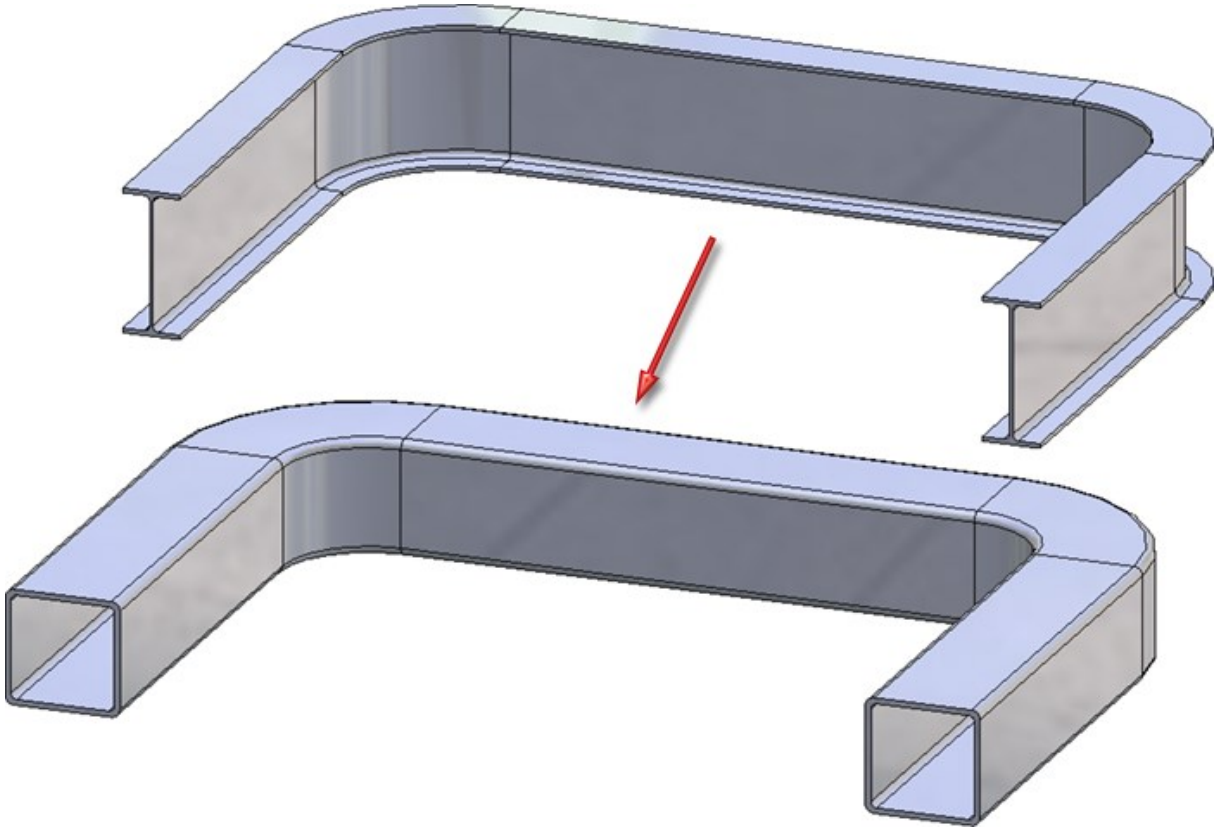
Standardized UNDO

The UNDO behaviour of all Steel Engineering connections has been stabilized and standardized. If a connection is inserted multiple times, i.e. without exiting the function, one UNDO step will refer to the last inserted connection, i.e. the insertion of one connection will be regarded as one step.

Process beams

Exchanging beams created along composite edges

From HiCAD 2019 onwards beams created along composite edges can be exchanged.



Glass insertion

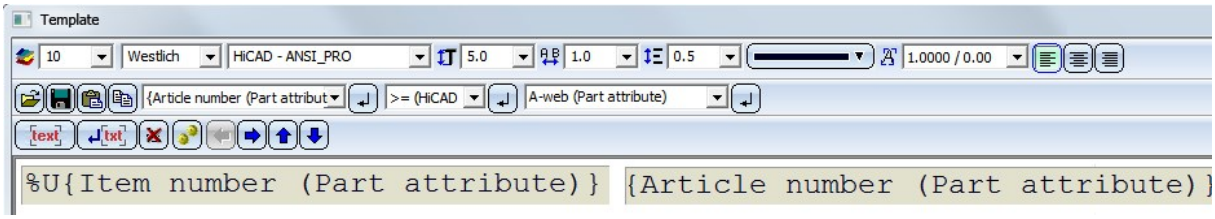
DXF export of multi-layered glass panes

When carrying out DXF exports via **Sheet Metal > Sheet development > Export > Sheets** 

glass panes will also be considered for export as of HiCAD 2019. If there are multi-layered glass panes, the individual glass layers will be exported.

For annotation purposes and file name composition the attributes of the total glass pane can be used. In this context, please observe the information given in the Attributes of superordinate parts paragraph of the **Attributes in annotation tags** topic.

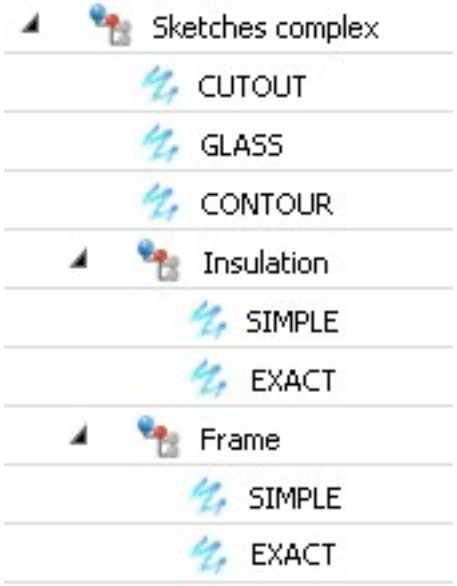
For instance, if you want the name of the DXF file to be composed of the item number of the complete pane and the article number of the glass pane, this can be achieved in the manual settings for the file name by using the following text blocks:



Series

Sketch-based series

User-defined series can now not only be created from 2-D parts, but also from 3-D parts that contain assemblies and sketches.



Steel Engineering - Management + BIM

Service Pack 2 2019 (V 2402)

Remove project-dependency

New in the menu **Management+BIM > Extras > Information** is the **Remove project-dependency**  function. .
Use this function to deactivate the project-dependency of the active assembly. This means that the article attribute STAHLBAUAUTO (**Project-related** checkbox) will be set to **No**, for both the assembly and all its sub-parts.

In this way, the assembly will be defined as a part for reuse. As a result, it will keep its the article master, even if it is inserted in other projects.

Checkup and Release of production drawings

Checkup and Release can now take place directly in the production drawing. In previous versions these options were greyed out if a production drawing had been loaded.

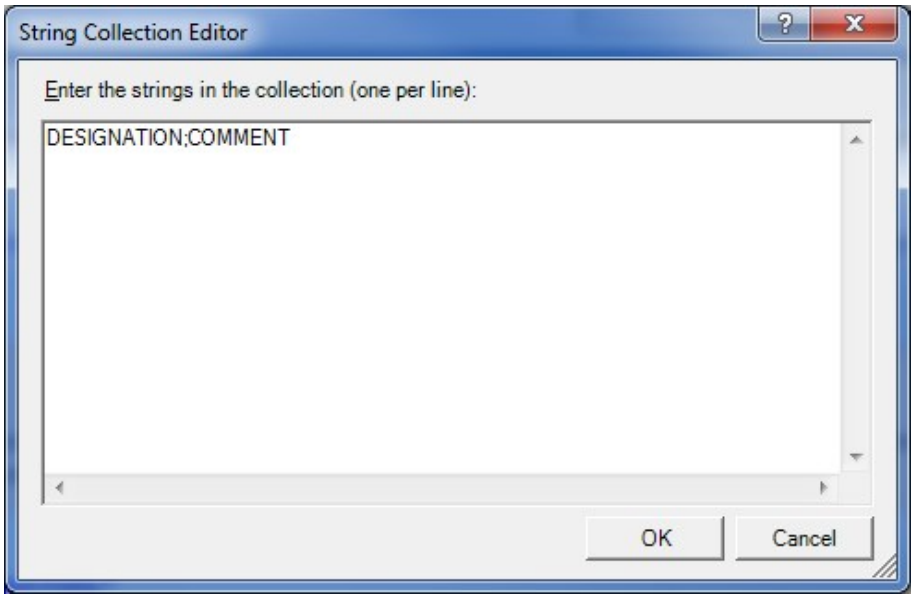
Article attributes for drawings with several parts

As with detail drawings, you can now also transfer article attributes of individual parts to document attributes of drawings with multiple parts. For this purpose the new parameter **Article attributes for drawings with multiple parts** has been added at **PDM > Management+BIM > Production drawings**.

There, click on the  symbol and define the desired assignments. Each assignment must be defined in one row, in the following format:

Article attribute;Document attribute

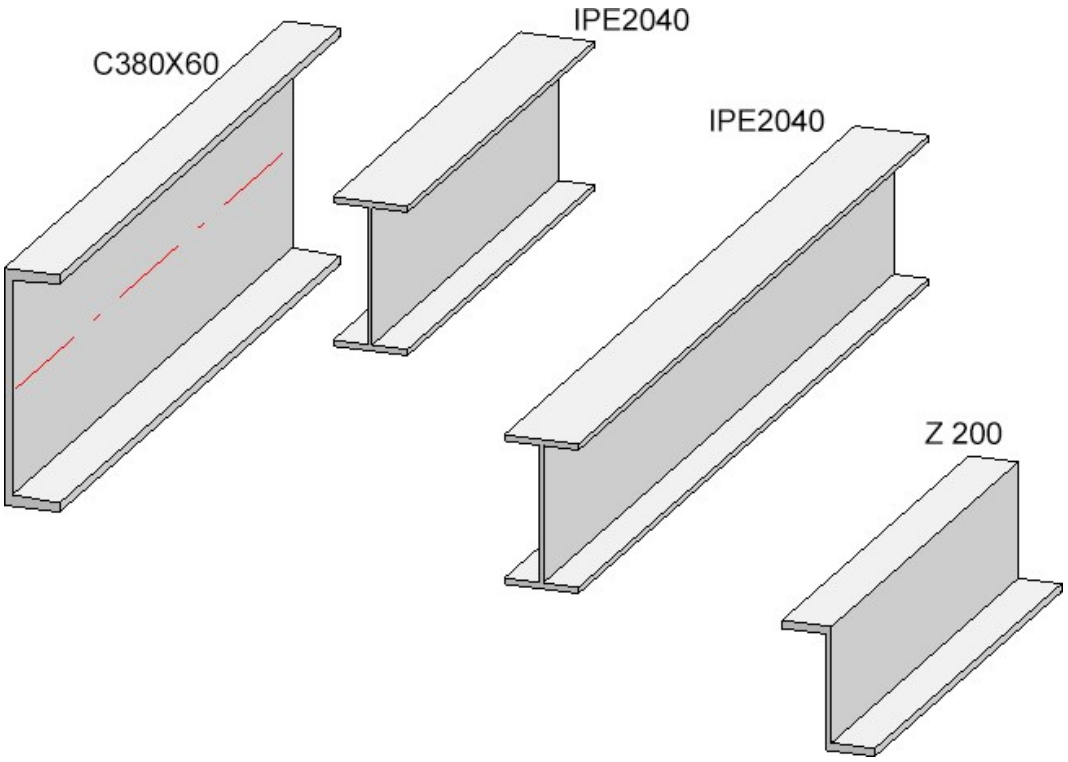
For instance, if you want to assign the article attribute NORMBEZEICHNUNG (Standard designation) of the parts to the document attribute BEMERKUNG (Comment) of the drawing, the assignment must look like this:



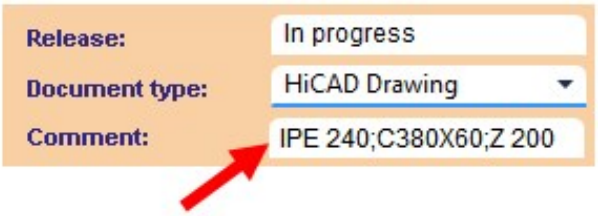
The article attribute NORMBEZEICHNUNG (Standard designation) of the parts will then be assigned to the document attribute BEMERKUNG (Comment) of the drawing, separated by a semicolon. Identical article attributes will be transferred only once in the process.

An example:

The model drawing consists of the 4 displayed beams with the indicated standard designations shown below.



If you now create a workshop drawing containing all parts, the following string will be assigned to the document attribute BEMERKUNG (Comment) of the drawing:

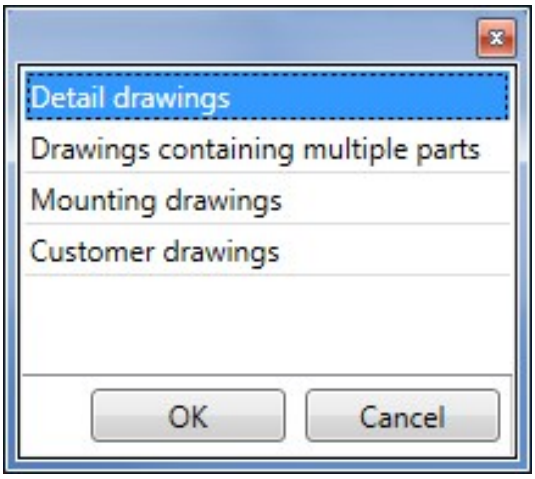


Please note that the attribute BEMERKUNG (Comment) that was used in the example is not contained in the dialogue masks by default, but needs to be added via the HELiOS Mask Editor if required!

Rework of drawing

When saving drawings that were created via **Management+BIM**, you have the option to execute an individual script for automatic reworking. For this purpose the new parameter **Rework of drawing** has been added at **PDM > Management+BIM > Production drawings**.

There you can choose between detail drawings, drawings containing multiple parts, mounting drawings and customer drawings. Mark the corresponding entry in the listbox to determine for which drawing types a rework via script is to be carried out, e.g.



The scripts that need to be executed must have certain names. The scripts that are supplied together with HiCAD have already been predefined as follows:

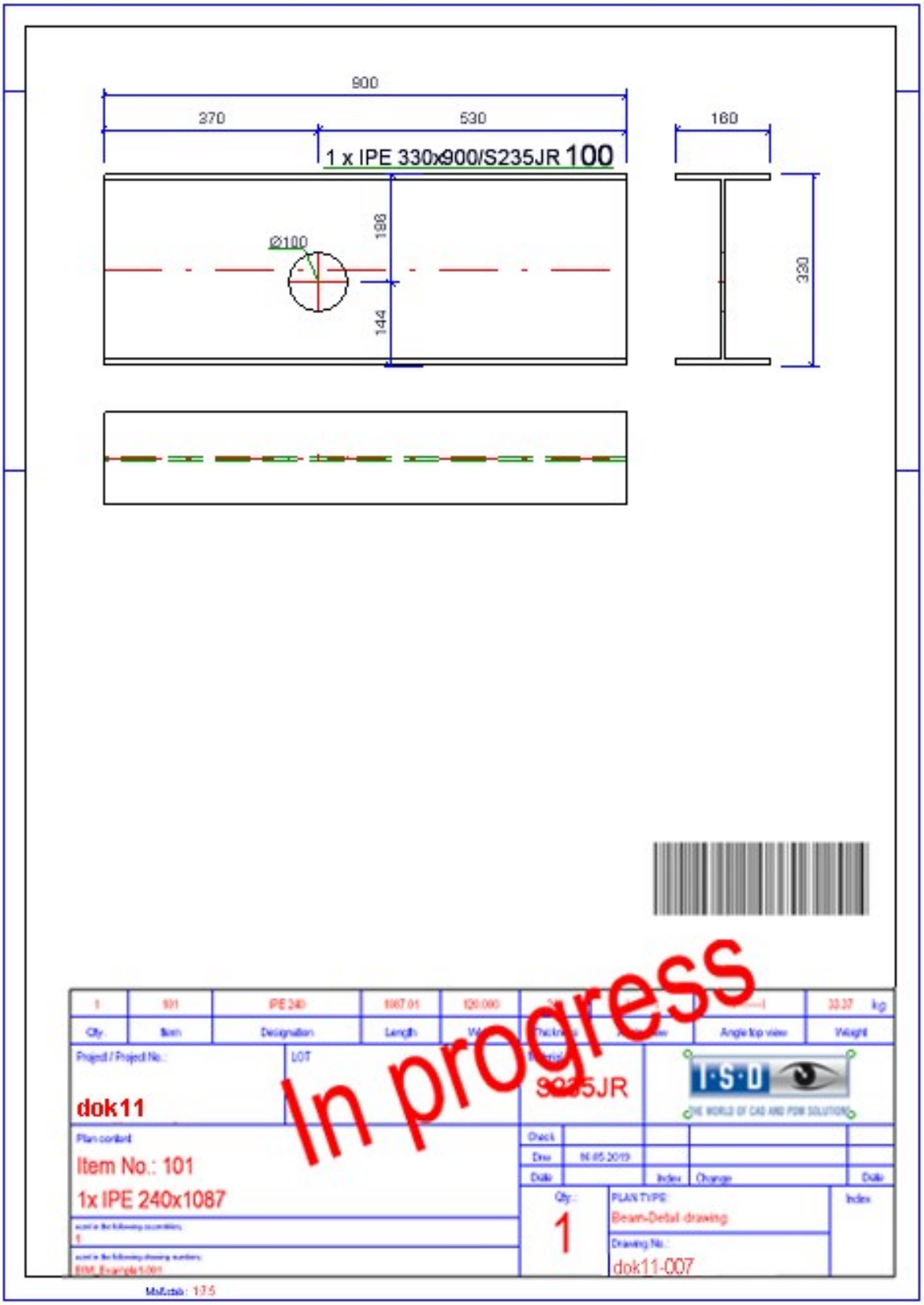
Drawing type	Name of script	Active part
Detail drawing	UpdateSingleWSD.cs	Part or Assembly
Drawings containing multiple parts	UpdateMultiWSD.cs	Main assembly
Customer drawings	UpdateCUD.cs	Main assembly
Mounting drawings	UpdateMTD.cs	Main assembly

The directory that is assigned to File group 2 in the FILEGRUP.DAT file has been preset as the path of the scripts. By default, this is the HiCAD directory MAKROST3D.

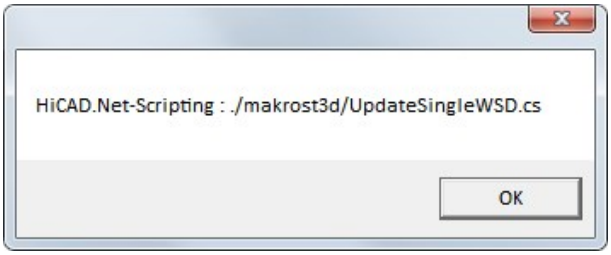
The file UpdateSingleWSD.CS:

```
// <debug/>

using System;
using System.Windows.Forms;
using ISD.CAD.Base;
using ISD.CAD.Data;
using ISD.CAD.Creators;
using ISD.CAD.Contexts;
using ISD.CAD.Modifiers;
using ISD.BaseTypes;
using ISD.Scripting;

class Script : ScriptBase
{
    [Context(typeof(UnconstrainedContext))]
    public static void Main()
    {
        // Das aktive Teil
        Part activepart = Context.ActiveNode as Part;

        MessageBox.Show("HiCAD.Net-Scripting : ./makrost3d/UpdateSingleWSD.cs");
        // Insert your code here
    }

    public static UnconstrainedContext Context
    {
        get { return BaseContext as UnconstrainedContext; }
    }
}
```

These script templates can be expanded by any API commands. For example, you could insert a bitmap with a corresponding bar code into all detail drawings in this way:



Please note however that a very good knowledge of the API is required for the creation of such individual scripts, and that the scripts (when they are activated) will be run upon every saving of the drawing, i.e. upon manual saving, creation and updating of the drawing, and its checkup and release. An appropriate message will be displayed in HiCAD after drawing creation, e.g.



Please note:

A very good knowledge of the API is required for the creation of the individual scripts.

As far as the example with the bar code is concerned: It will not be sufficient to merely add the insertion of a bar code in the script. Instead you need to ensure that the script also contains a check for an already existing bitmap and replace it. Otherwise the bitmap will be contained multiple times. HiCAD will not perform any such check.

Service Pack 1 2019 (V 2401)

Changed settings in Configuration Editor

- The parameter **Create part document** in the Configuration Editor at **PDM > Management + BIM** is no longer available.
- New at **PDM > Management + BIM** are the parameters **Transfer non-BOM-relevant construction sections** and **Updating of referenced parts**.

Non-BOM-relevant construction sections and product structure

HiCAD model drawings in Steel Engineering are often split into so-called construction sections, e.g. frameworks, platforms, columns, platform support structures etc. These construction sections are normally non-BOM-relevant structure assemblies, which were previously not transferred to the HELiOS product structure. From HiCAD 2019 SP1 onwards, the new usages

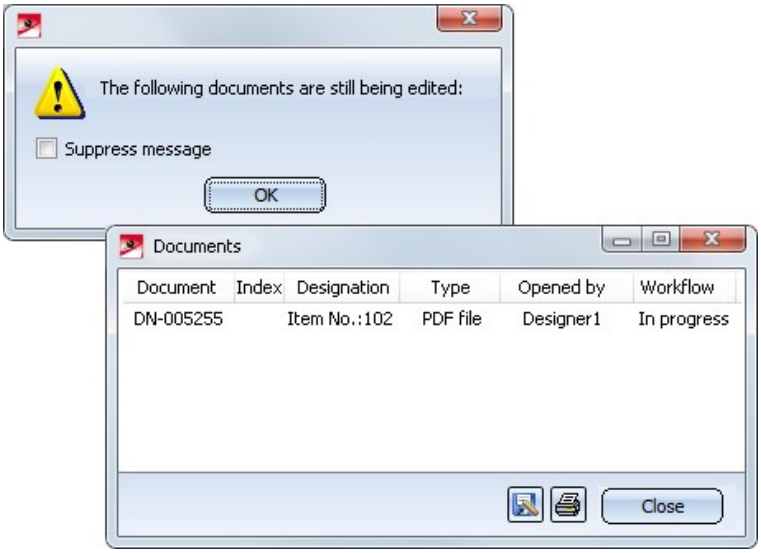
- Construction section or
- Production section

are available. Construction sections/structure assemblies can now also be transferred to the HELiOS product structure if they are not BOM-relevant. For this to happen, the parameter **Transfer non-BOM-relevant construction sections** in the Configuration Editor must have been set to **Yes**.

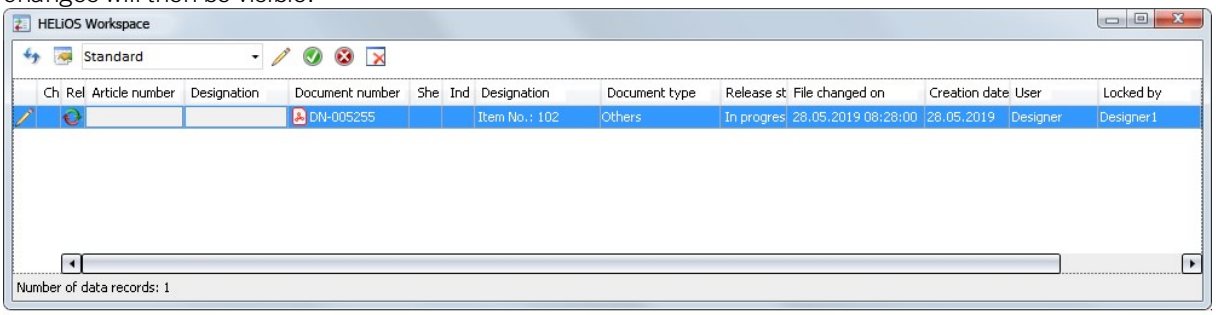
No updating of drawings while production documents are open

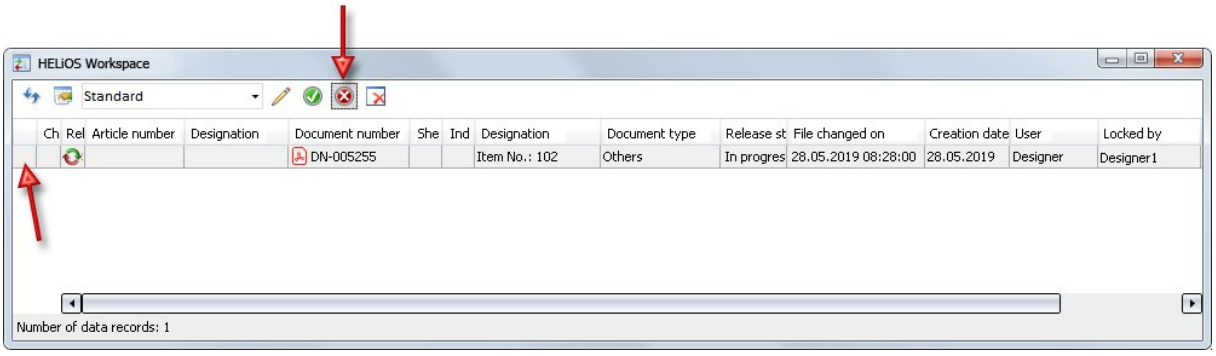
If an external production document, e.g. a PDF file, has been opened in HELiOS, a drawing that is linked to it cannot be updated in HiCAD.

- In such as case HiCAD will issue the message shown below and list the documents in another window:



In this situation you have the option to end the editing by choosing (**Extras > Show workspace** ) and clicking on the  symbol in the **HELiOS workspace** dialogue window. When you then perform an update, the opened external document, too, will be updated in the background. When you open this document again, all changes will then be visible.





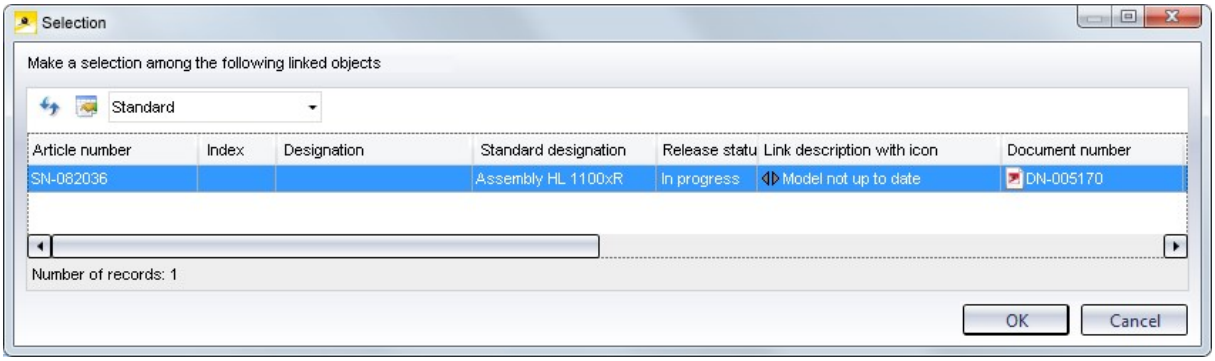
Updating of models with old referenced parts

Like production drawings, you can now also manage construction sections (i.e. referenced part models) in HiCAD. For this purpose, the new links **Model up to date** and **Model not up to date** have been introduced.

In the Configuration Editor at **PDM > Management+BIM** you will find the parameter **Updating of referenced parts** that enables you to specify whether the updating of BIM-managed, referenced parts is to take place in all models.

If you select **Yes**, HiCAD will offer you the additional function **Update models with old referenced parts**  . This function enables you to update all models containing old referenced parts in one step.

When you call the function, all models containing old parts will be listed.



Choose the required model and click on **OK**.

Major Release 2019 (V 2400)

New functions for linked documents

The **Linked documents** function now has a pull-down menu with further functions for linked documents:

	**Model drawing with part** Opens the model drawing containing the active part. For instance, if you work with referenced part drawings and call this function in the original model drawing, the corresponding part model drawing will be opened.
	**Production drawing** Opens the production drawing of the active part.
	**Production drawings** Displays a list with all production drawings in which the active part occurs, i.e. also the assembly drawings. Choose the desired documents and close the window with **OK**.
	**Mounting/customer drawings** Displays a list of all mounting drawings and customer drawings in which the active part occurs. Choose the desired documents and close the window with **OK**.
	**Itemised source model** Opens the itemised source model of the active part, i.e. the model in which the part has been itemised. see also: ITEMISATIONMODEL attribute

Derive, active assembly(-ies) without sub-parts

The **Derive, active assembly(-ies) with sub-parts** function now has a pull-down menu with the **Derive, active assembly(-ies) without sub-parts** function.

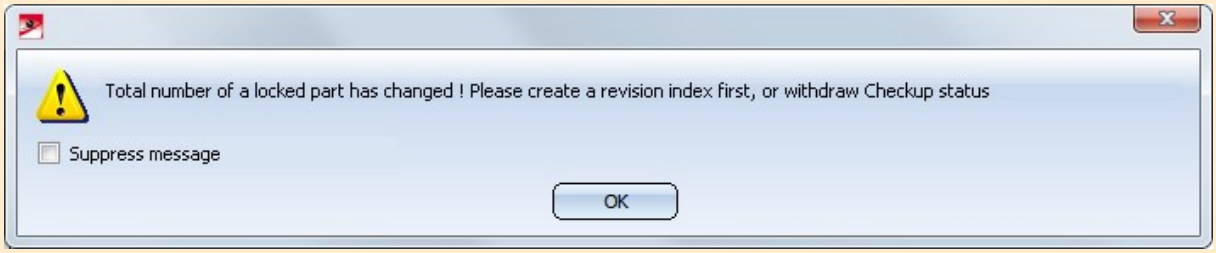
Here, too, the active assembly will be copied, but with the following difference:

- The article masters, indices and item numbers of the sub-parts are preserved.
- The Workflow status of the sub-parts remains unchanged.
- The links of the sub-parts to the drawings are preserved.

This function can make sense if you insert a released assembly a second time, and want to add just another beam there, without changing the sub-parts.

When itemising the derived assembly, only the item numbers of the assembly will be continued; the sub-parts will retain their original item numbers.

Please note that this function will only make sense if in the Configuration Editor the setting **PDM > Management + BIM > Production drawings > Consider total number in producton drawing** has been set to **No**. Otherwise the following message will be displayed when you save the model drawing:



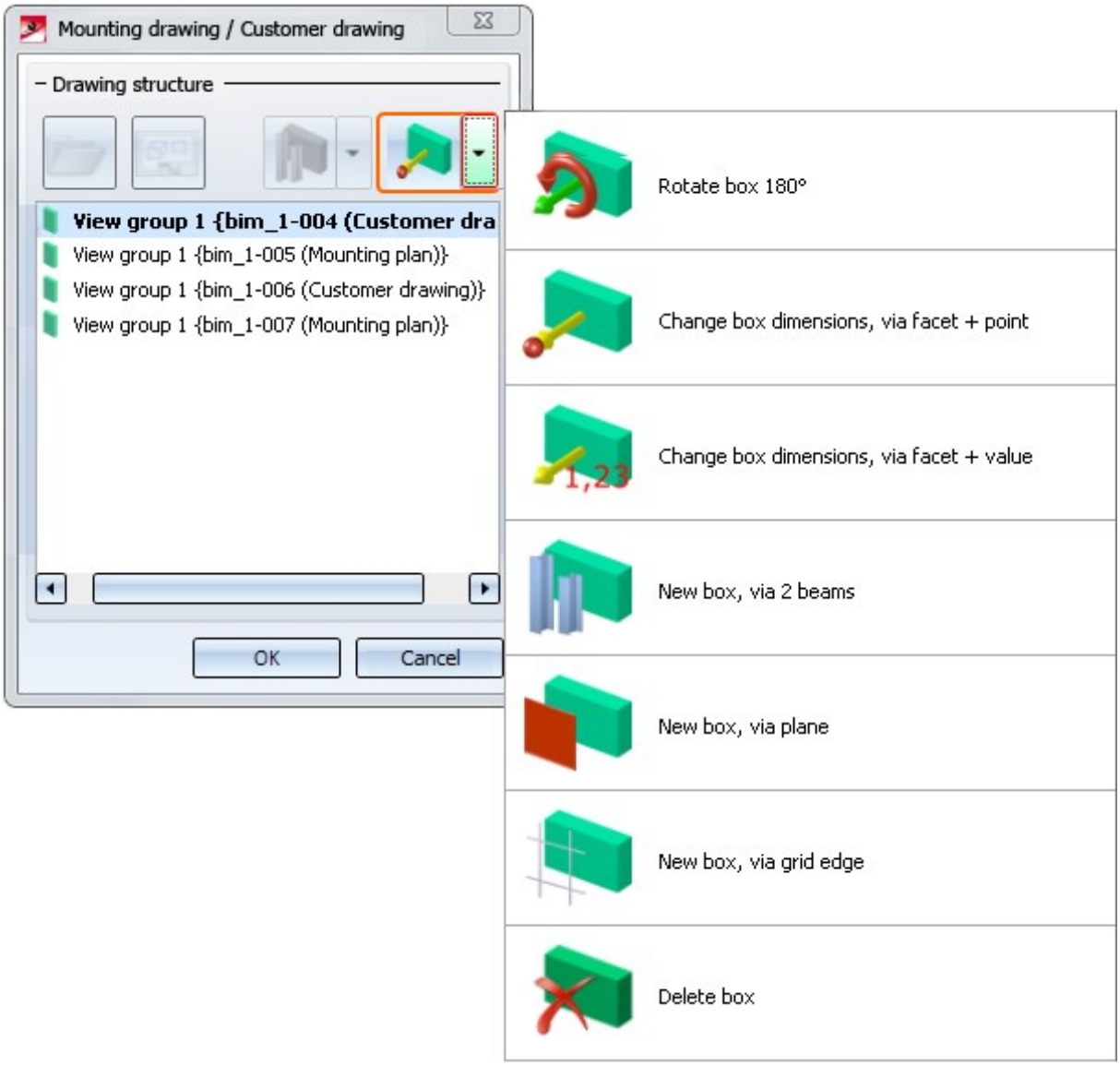
After clicking on **OK**, the corresponding sub-parts will be listed for your information. For these sub-parts of the copied assembly a revision index would have to be created, or the Workflow status would have to be reset, which would, in principle, be contrary to the function's purpose.

Set workflow status for active assemblies back, without sub-parts

Use the new **Active assembly(-ies) without sub-parts**  function to reset drawings and all their sub-parts from the Workflow status **Checkup** to the Workflow status **In progress** again. Drawings for sub-parts of the assembly will not be considered here.

Change details of mounting/customer drawings - New context menu functions

The context menu for the changing of box dimensions of the **Detail** function has been adjusted to that of the **Mounting drawing** function on the **Drawing** tab. This means that you can now select new boxes, or delete boxes in mounting drawings and customer drawings that have been created via the Management + BIM module.



If you delete boxes in a mounting/customer drawing, the associated mounting/customer plan will be removed from HELiOS, too. In this case the **Update drawing** function will not be required.

Automatically create/update drawing for active assembly

New in the pull-down menu of the **Management + BIM > Workshop > Drawing** function is the **Active assembly(-ies) with sub-parts**  function.

You use this function to

- automatically create workshop/detail drawings and
- update workshop/detail drawings after modifying the assembly models. Mounting drawings and customer drawings cannot be created with this function, but updated. However, such updating is only possible if the drawings are **In progress**. In this context, please also read the information given in the topics Customer Drawings - Revision index, Change Released Drawings and Mounting Drawing - Revision index.

either for only the active assembly or for a selection of several assemblies.

The settings for drawing derivation (i.e. for the drawing sheets, view groups, views etc.) will not be specified in the dialogue here, but loaded from appropriate pre-setting files instead. This means that the **Drawing derivation** dialogue window will not be displayed when you call this function, which speeds up the drawing creation process.

Otherwise, the function largely works in the same way as the **Management + BIM > Workshop > Drawing**  function - with the difference that here, non-existent drawings will be generated immediately without any further queries. Only then will HiCAD check

- which drawings - also mounting drawings and customer drawings - are no longer up to date, and
- whether parts for which a drawing existed had been deleted in the model drawing.

New setting options in the Configuration Editor

In the Configuration Editor at **PDM > Management + BIM** additional parameters are available.

PDM > Management + BIM

Update production drawings

Via this parameter you can determine whether production drawings are to be loaded and updated upon Checkup and Release. The default setting is **Yes**.

Choosing **No** can make sense if only the Workflow status of an assembly, but not the content of the drawing is changed. When you choose this setting, the drawings will, for instance, not be loaded/updated if the Workflow status changes from **Checkup** to **Released**, or if the index changes.

This also applies if, after a release of drawings, the index is changed with the functions available at **Management + BIM > Workshop > Change**, but then it is decided to make no changes.

In this way the waiting times in connection with the updating are reduced.

Product structure transfer

This parameter determines the conditions for a product structure transfer to HELiOS.

- **Only if model drawing is valid (green)**
  The product structure will only be transferred if the BIM model drawing is valid, that is, if a corresponding article exists in the HELiOS database for each part. A valid model drawing is marked with the  symbol in the ICN. This is the default setting.
- **Upon each saving with BIM**
  The product structure will be transferred each time you save with the Management + BIM module - even if the model drawing has not been completely passed on to HELiOS yet.

PDM > Management + BIM > Production drawing

Production drawings for glasses

When creating production drawings for glasses you distinguish between rectangular glass panes and modelled glass panes (i.e. non-rectangular and/or unprocessed glass panes).

If you set the parameter **Production drawing for glasses** to **Yes**, production drawings will be created both for rectangular panes and modelled panes. This is the default setting.

If you choose the setting **Only for modelled glass panes** instead, rectangular glasses will not be considered for production drawing creation.

> **Please note:**
>
> This parameter only applies to glass panes from the HiCAD catalogue, i.e. from the catalogues **Factory standards > Glass panes** and **Factory standards > Multi-layered glass panes**!
>
> All other glass types, e.g. from LogiKal, are automatically considered modelled glass panes with production drawing.

Rearrange views when updating production drawings and mounting

Use this parameter to specify for external, PDM-managed drawings whether you want the views to be rearranged automatically in production drawings or mounting drawings upon updates. Switching this option off makes sense if, for instance, you have deliberately moved some views and want to keep their new position.

The default setting is **Rearrange**.

drawings

PDM > Management + BIM > External production documents

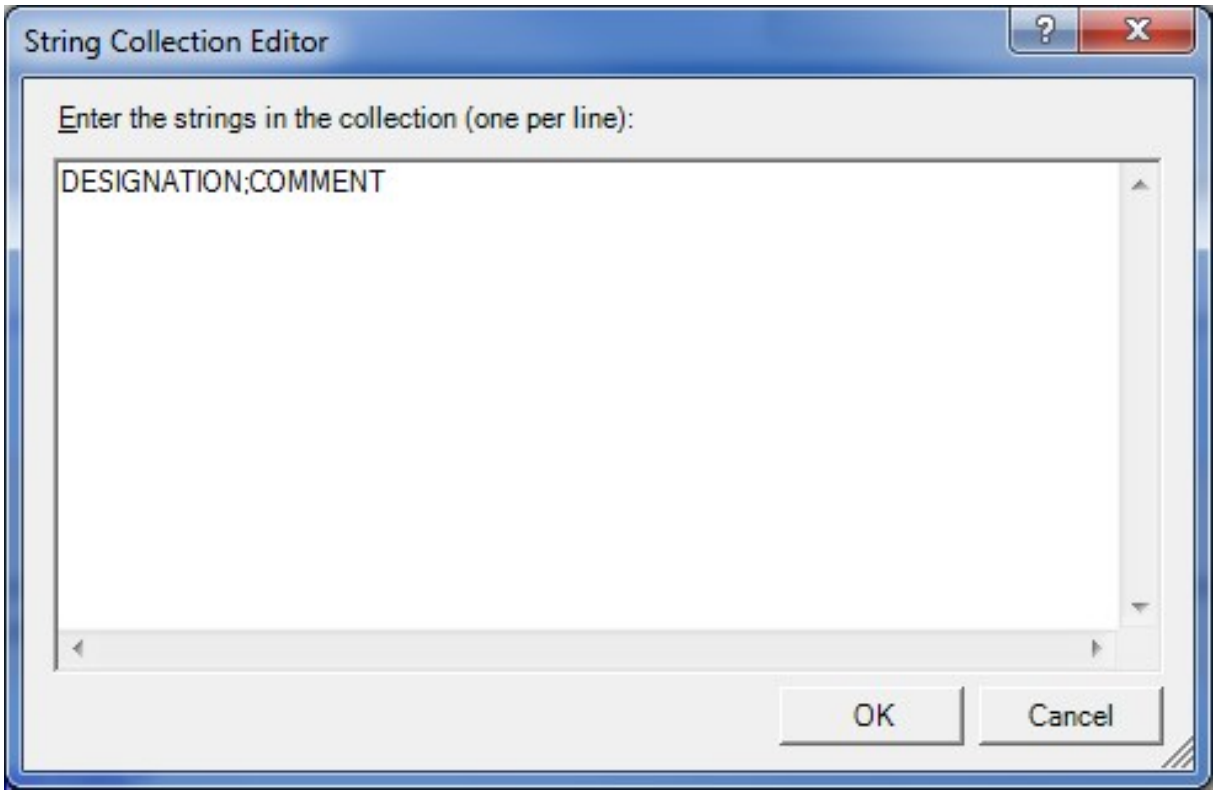
Article attributes for CAM documents

Here you determine which article attributes are to be assigned to the CAM document attributes when the CAM data are created.

To transfer article attributes of the semi-finished product to CAM document attributes, click on the ... button and define the desired assignments. Each assignment must be written in one line, as shown below.

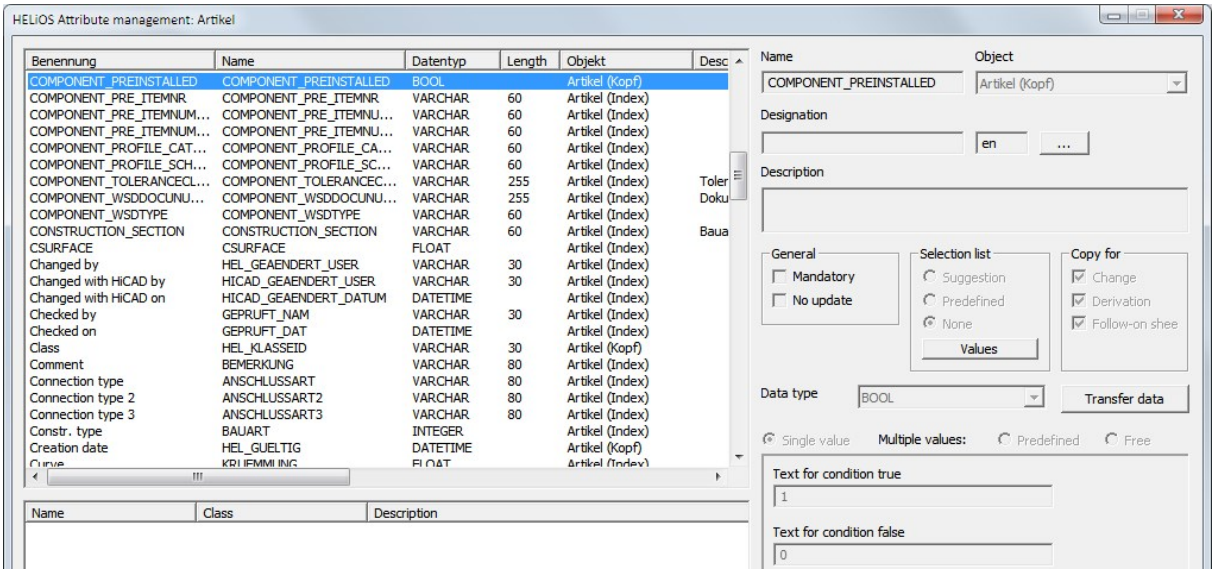
Article attribute;Document attribute

For instance, if you want to assign the article attribute DESIGNATION of the semi-finished product to the CAM document attribute COMMENT, you must do it like this:



Pre-mounted assemblies in BOMs

When working with pre-mounted assemblies in practice, it is sometimes required that only the assemblies, but not their sub-parts are displayed. This can be achieved with the article attribute **COMPONENT_PREINSTALLED**, which needs to be assigned to the article master of the pre-mounted assembly.



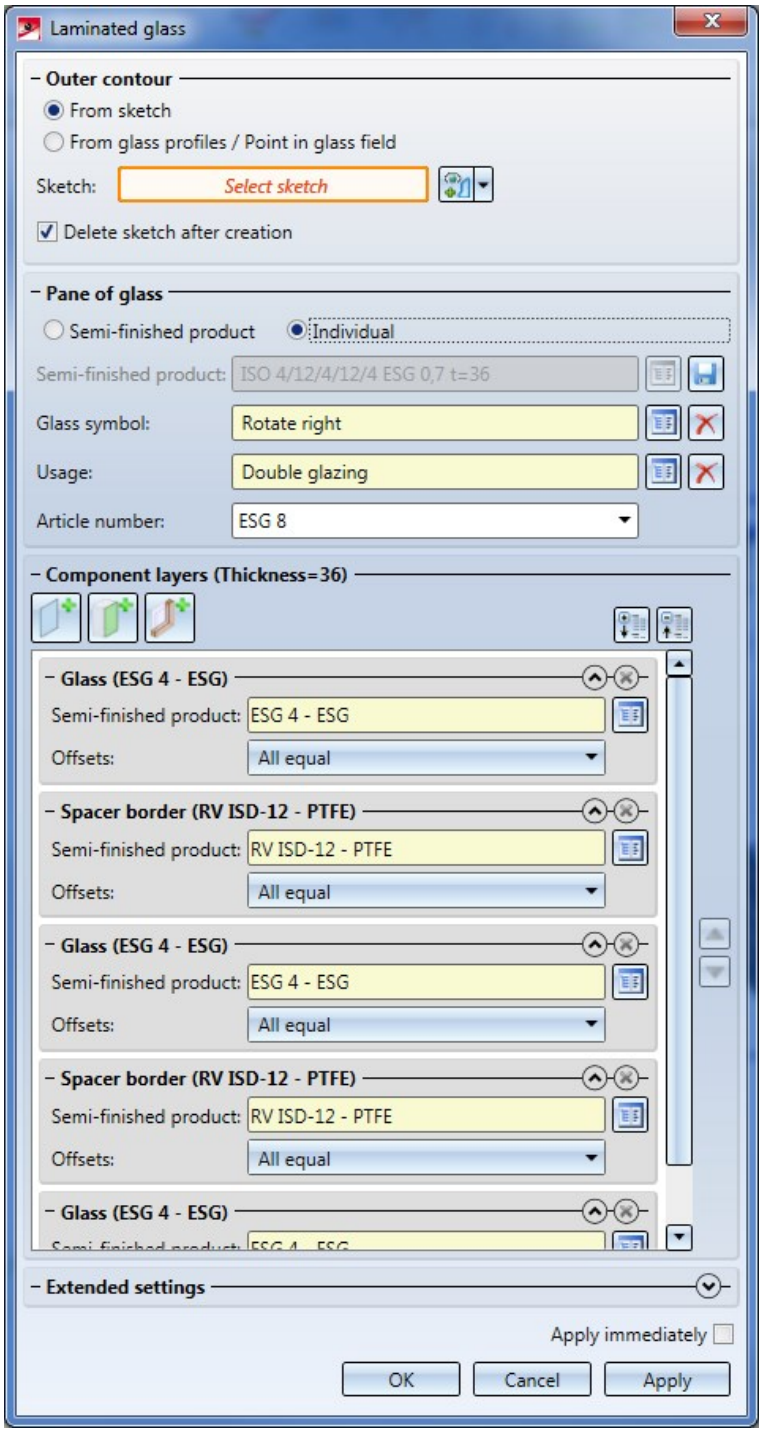
To ensure that the attribute is shown in the HELiOS article masks, you need to adjust the masks in HELiOS accordingly, i.e. the attribute **COMPONENT_PREINSTALLED** must be added as a new data field in the mask, creating a new checkbox in this way. If you activate this checkbox in the article master mask of an assembly, this assembly is regarded as "pre-mounted".

Metal Engineering

Service Pack 2 2019 (V 2402)

Glass insertion

The previous functions **Glass** and Glass along sketch have been combined into a new function called **Laminated glass**, which allows the creation of both glass panes in glass fields and the creation of glass panes from sketches.

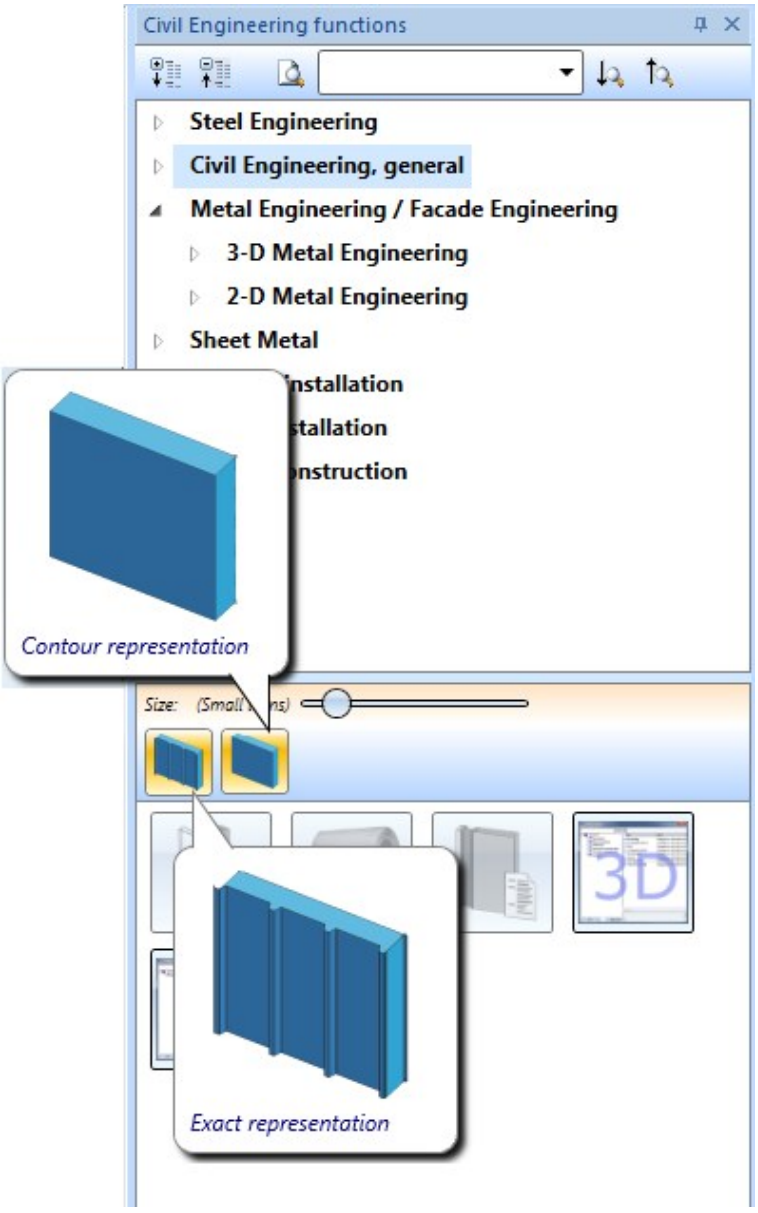


Major Release 2019 (V 2400)

Toggling between exact and contour representation

The buttons for toggling between exact and contour representation that could previously been found at **Metal Engineering > Others** have been moved.

You will now find them in the **Civil Engineering functions** docking window at **Civil Engineering, general**.



Taking over of U-values of LogiKal glasses

When importing glasses from LogiKal, the U-value (thermal transfer coefficient) of the glasses will be taken over into the System attribute §26.

Element Installation

Service Pack 2 2019 (V 2402)

The Dialogue Editor for customized installation elements

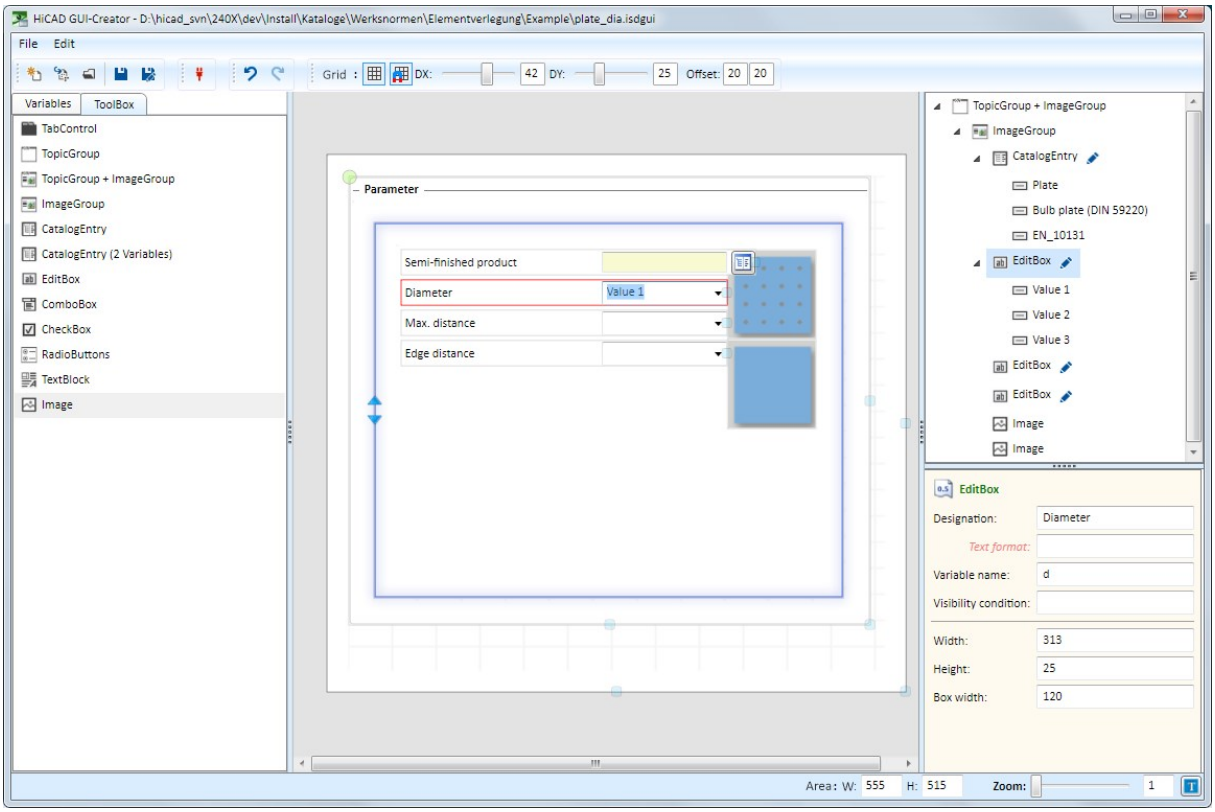
HiCAD offers the optionto create customized installation elements for facades (Element installation and Sub-structure). The dialogue which is shown when selecting these customized installation elements can in turn also be customized. Excluded from customization are the areas **Sketch**, **Type** and **Joint width towards sketch line**.

In previous versions, the XML file for the dialogue had to be created manually. From HiCAD 2019 SP2 onwards, a convenient tool,the **HiCAD GUI Creator**, will be available for such tasks. Just run the EXE file to start the tool.

HiCADGUICreatorApp.exe

in the HiCAD EXE directory.

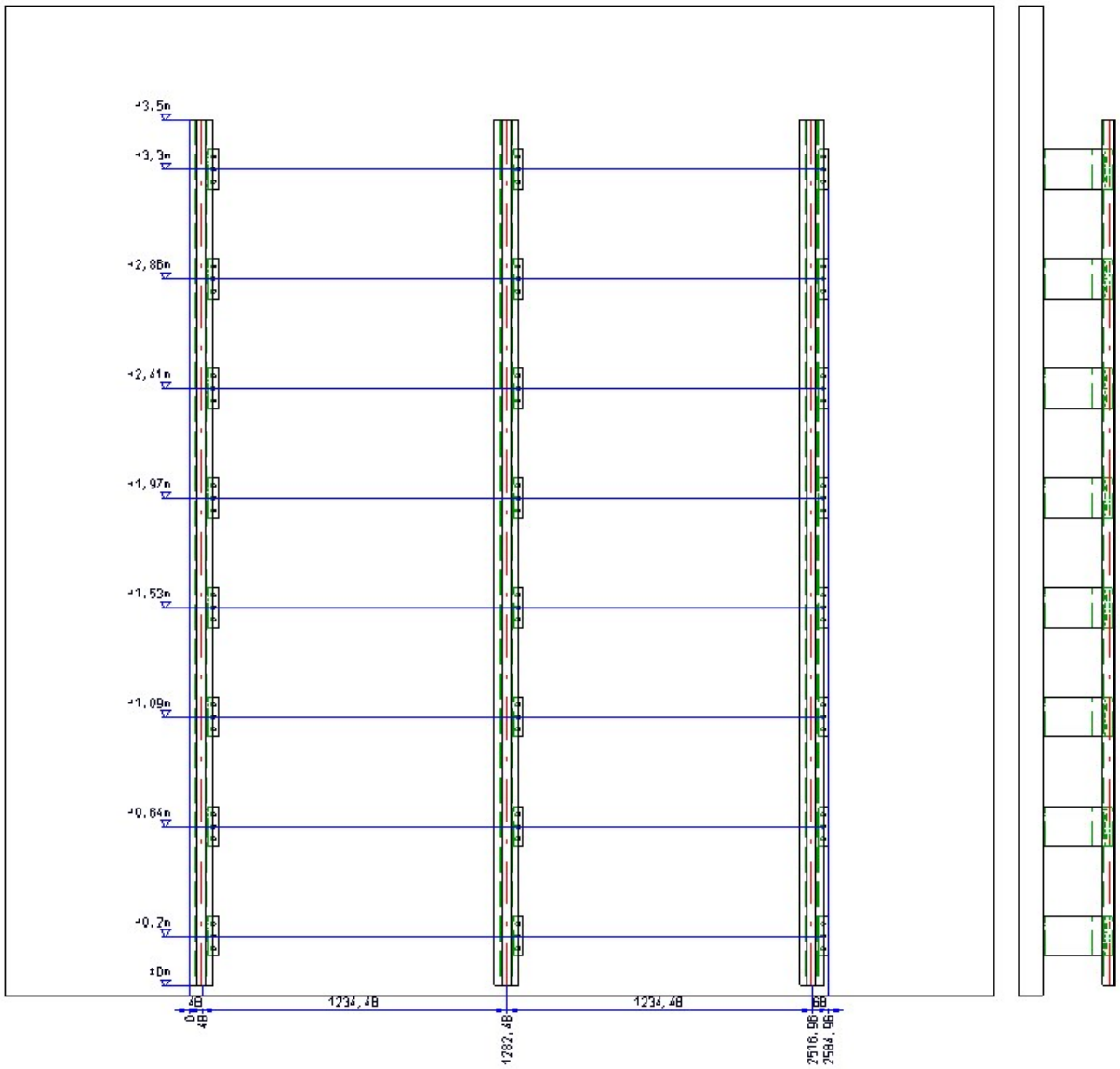
After calling the Editor, the **HiCAD GUI-Creator** dialogue window will be displayed.



The **HiCAD GUI Creator** in this version supports only dialogues for Element Installation. In future versions, the Editor will become a universal Dialogue Editor.

Mounting drawings for sub-structures

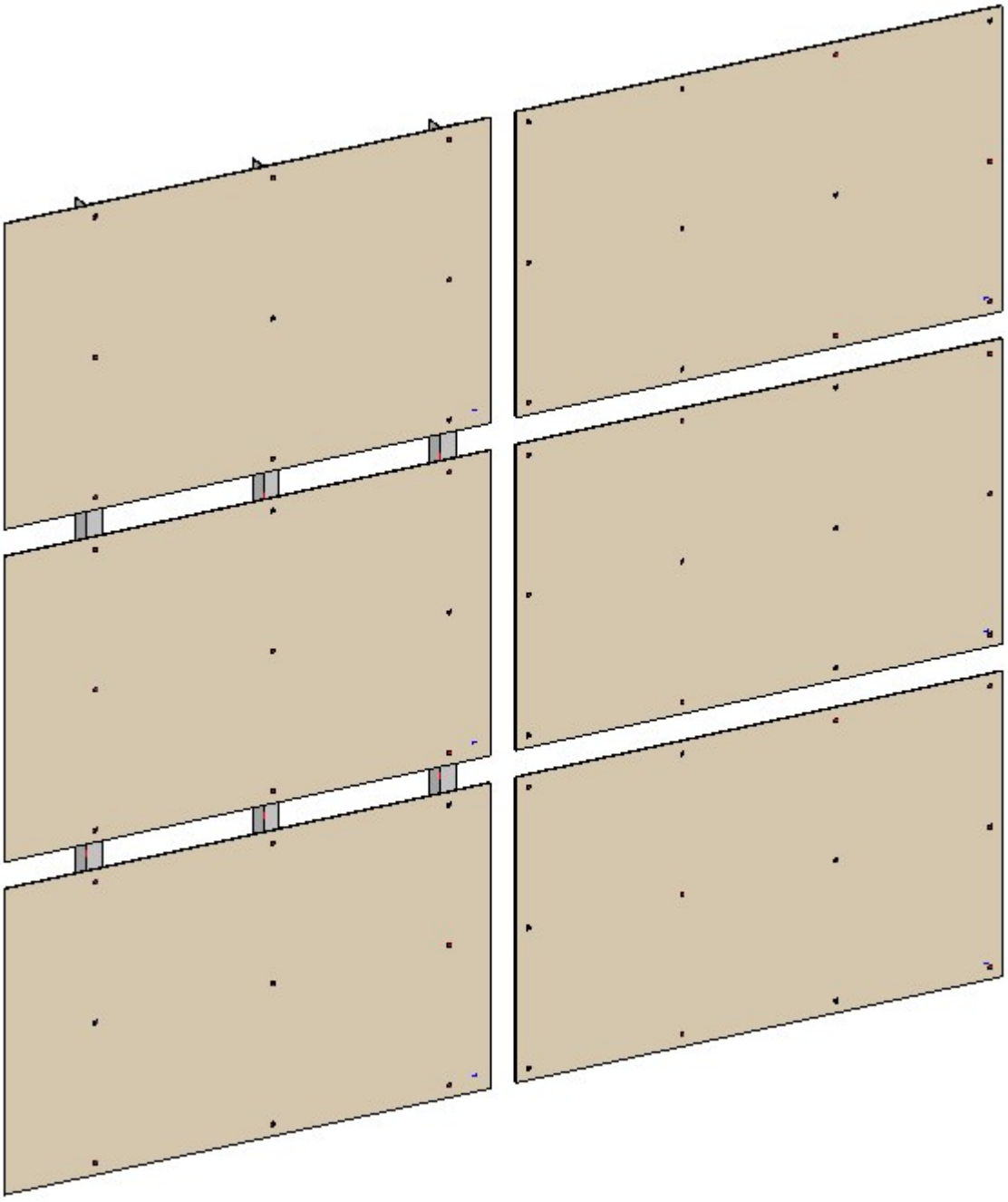
You have now the option to automatically derive mounting drawings for sub-structures.



Prior to this, a one-time configuration of the dimensioning rules will be required.

ALUCOBOND riveted with sub-structure

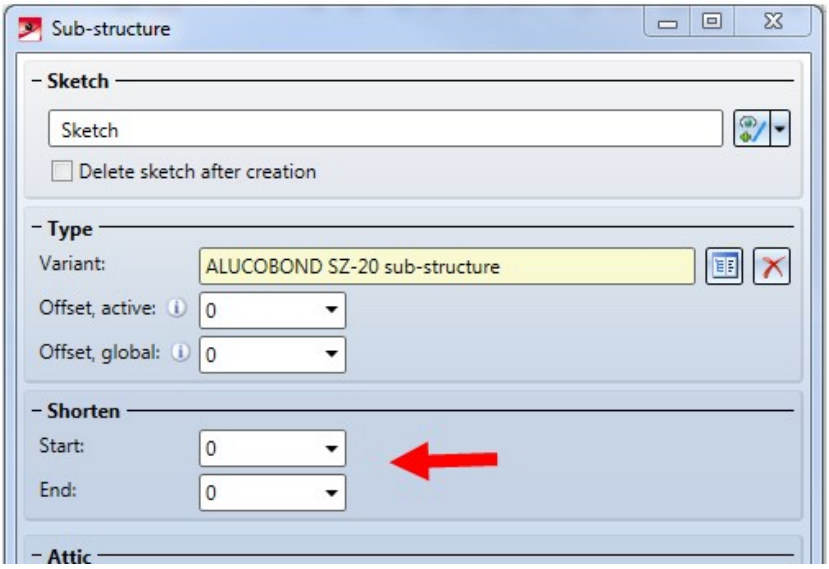
Bores and rivets connecting element installations to sub-structures will now be adapted to the sub-structure in visibly riveted ALUCOBOND panels.



Left: ALUCOBOND riveted, with connected sub-structure; Right: ALUCOBOND riveted, without sub-structure

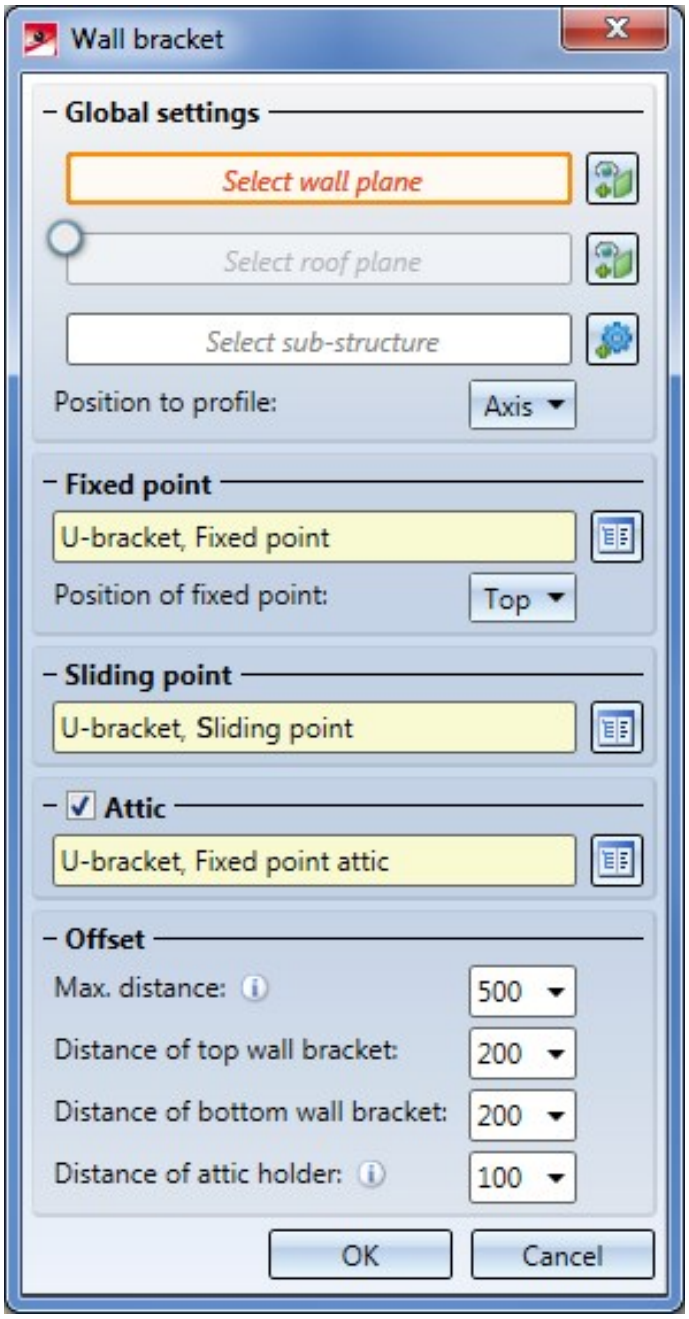
Shortening of sub-structures

When inserting sub-structures, you can now define, for each graphical element, a separate shortening for the start and the end of beams.



Creation of wall brackets

Use the **Wall bracket** function to automatically create wall brackets for sub-structures.

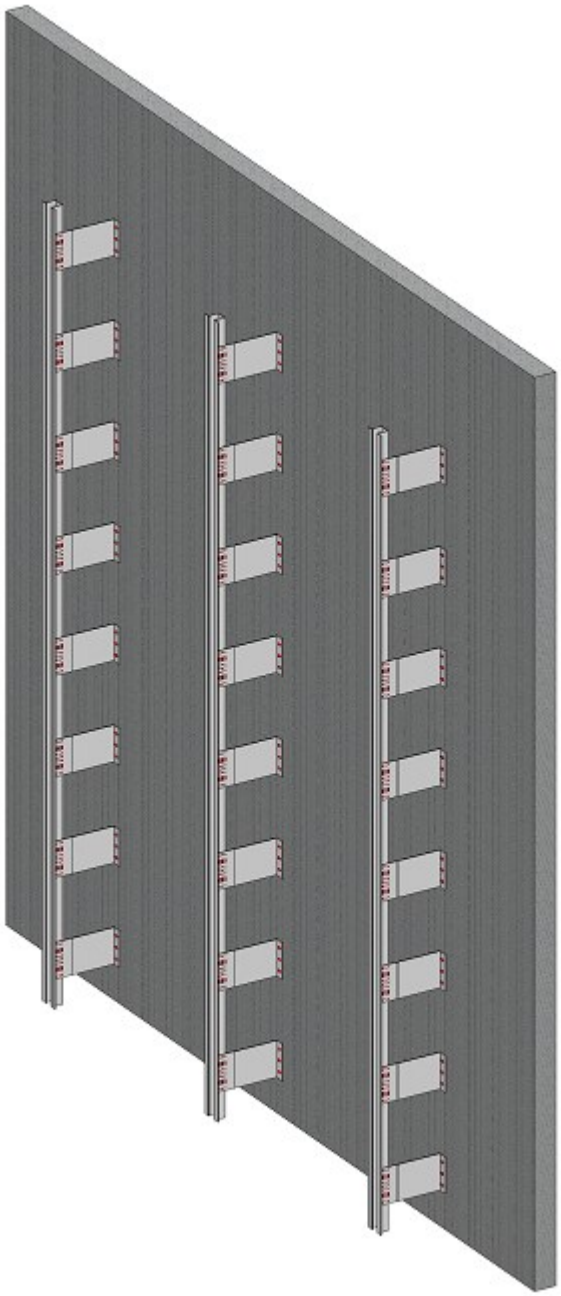


These can be created not only on walls, but also on roofs, e.g. when inserting attics.

Via the **Position to profile** setting you can mount the bracket in centred position under the sub-structure beams (e.g. for U-shaped brackets), or can be aligned to the beams on the left or right (e.g. for L-shaped brackets).

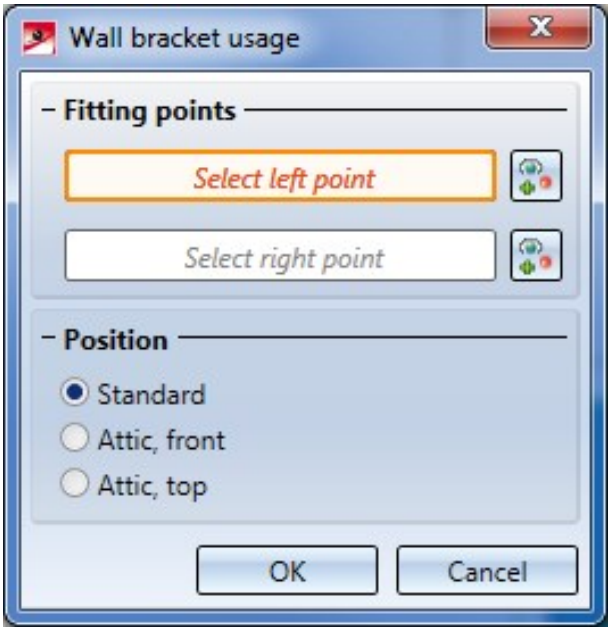
One of the brackets will be used as **Fixed point**, the other brackets as **Sliding points**. You can choose whether the top bracket or the bottom bracket is to be the fixed point, or whether the fixed point is to be located near the middle of the beam.

For the top and the bottom wall bracket a specified **Distance** to the end of the beam is used. The other brackets are evenly distributed in such a way that a **Maximum distance** is observed.



You can create your own wall brackets, save them to the catalogue system and insert them automatically afterwards. These brackets can be conveniently created from wall consoles in the catalogue, e.g. from the new Hilti consoles for rear-ventilated facades.

For a horizontal positioning of the wall brackets to the beams the new **Wall bracket usage** function usage has been added, which allows you to determine the distance of the wall brackets to the beam axes. In the Create Own Profiles for Sub-structures topic you can see a concrete example of how to use this function.


Wall bracket usage
Fitting points
Select left point
Select right point
Position
Standard
Attic, front
Attic, top
OK
Cancel

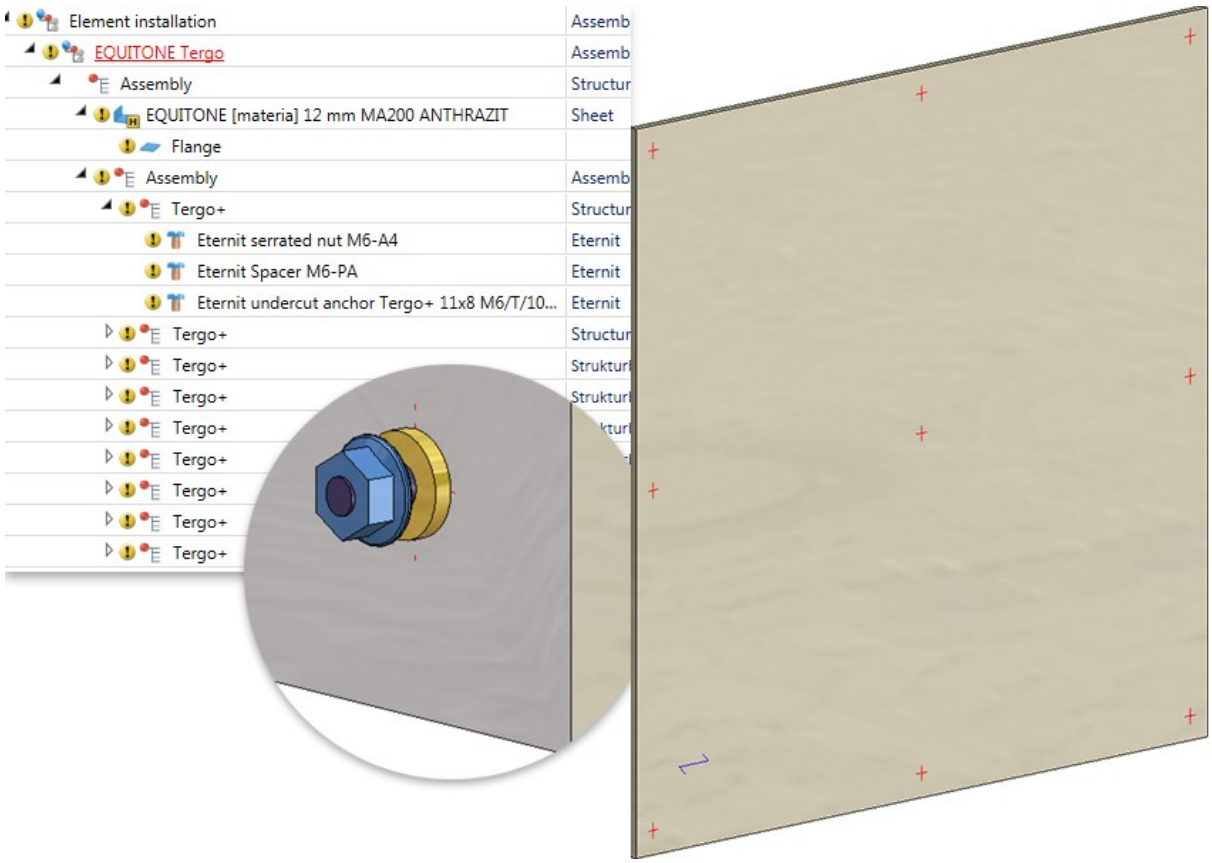
Eternit® installation elements

HiCAD 2019 SP2 supports the installation of Eternit panels. These can be found in the following catalogues:

- **Factory standards > Installation Planning - Parts and Processings > Element installation > Installation elements > Eternit**

  Available are:
  - EQUITONE Tergo and
  - EQUITONE riveted
- **Factory standards > Installation Planning - Parts and Processings > Sub-structure > Installation elements > Eternit**

  Available are:
  - Riveted to L-profile
  - Riveted to T-profle and
  - Systea UBE 25/2

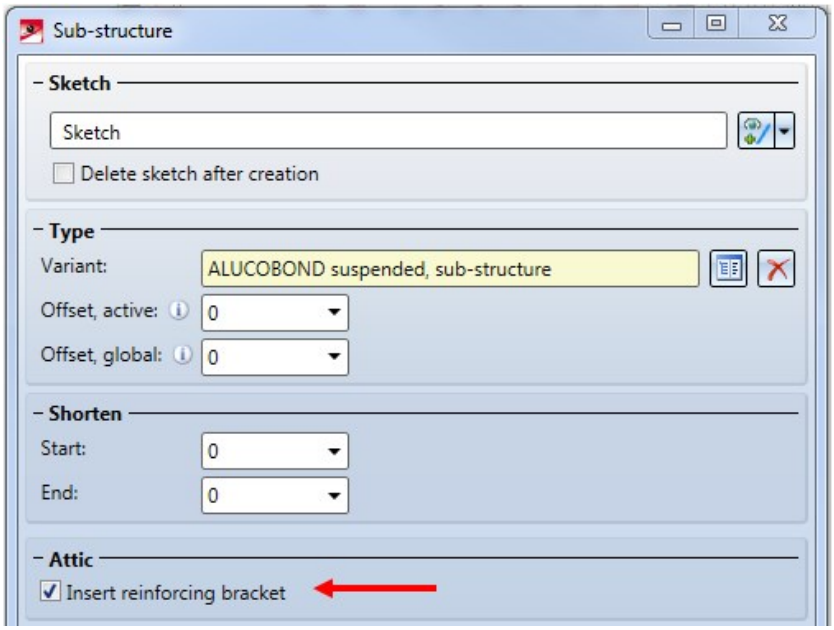


EQUITONE Tergo, Fixing type Tergo+

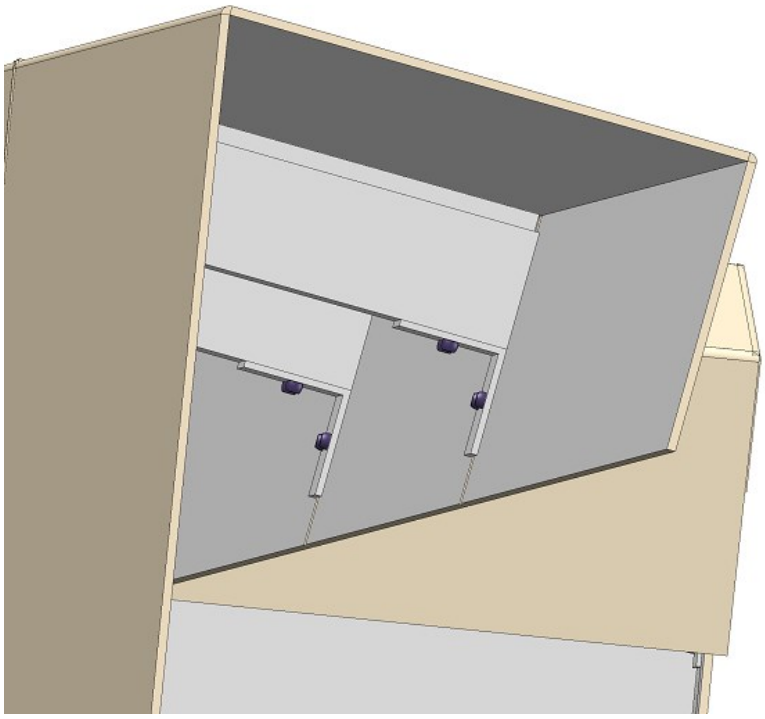
Since EQUITONE Tergo elements must be installed according to certain static requirements, the new **Create sketch for sub-structure** function has been introduces, which automatically creates a sketch serving as the basis for a Tergo element installation.

ALUCOBOND® sub-structure: Reinforcing bracket for rear attic flange

The new **Insert reinforcing bracket** option can be applied to **ALUCOBOND suspended** and **ALUCOBOND SZ 20** sub-structures.

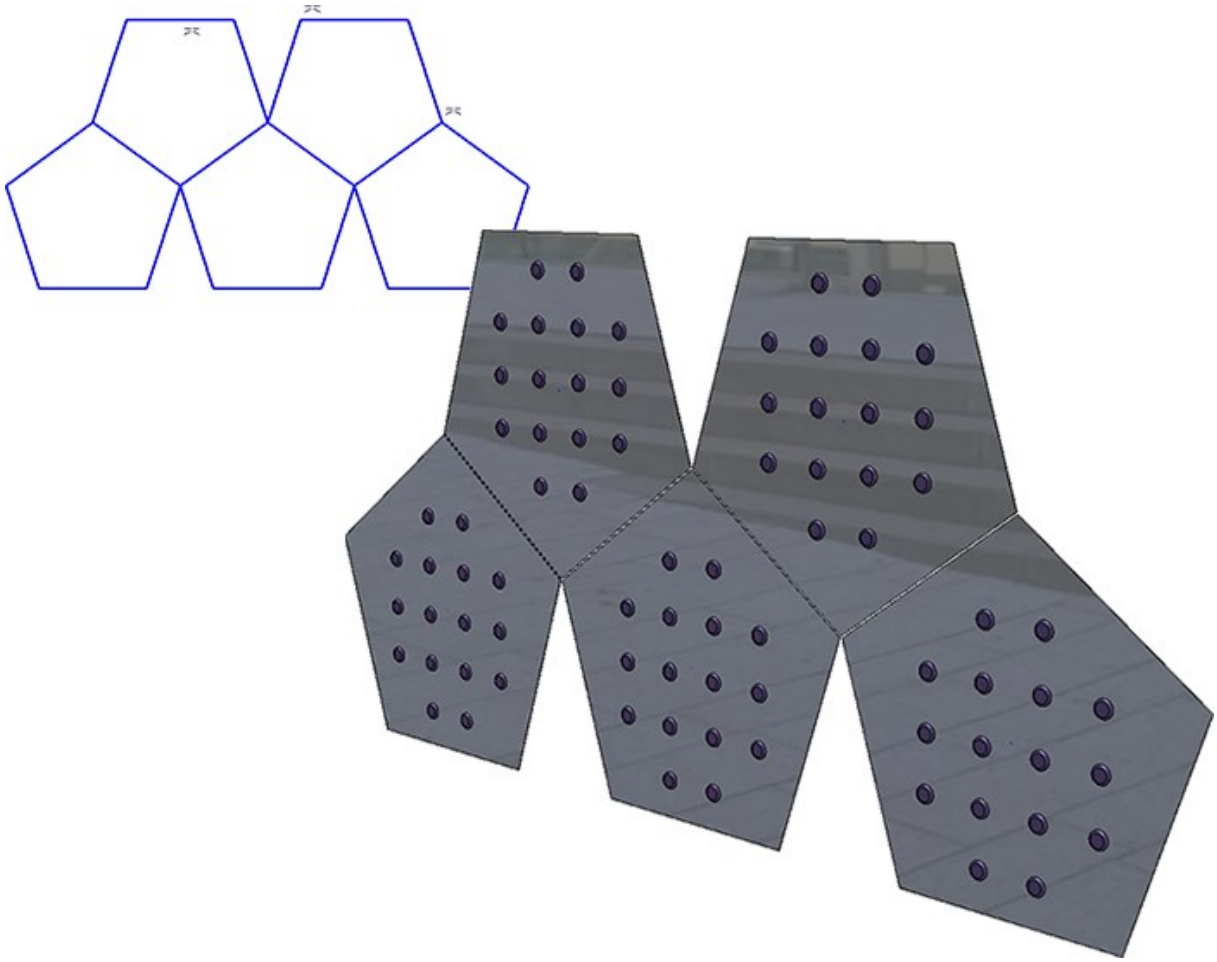


If this option is active, and the sub-structure is mounted to a matching element installation, additional reinforcing brackets will be automatically riveted to the rear attic flange:



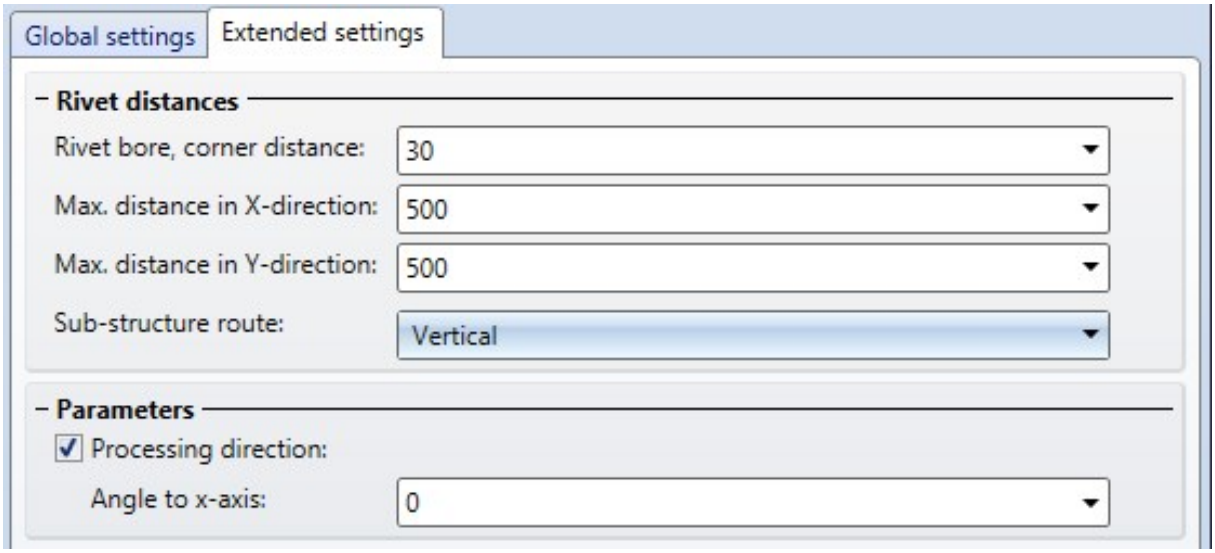
ALUCOBOND® riveted

Riveted ALUCOBOND tray panels can now be placed on any rectangular and polygonal sketch areas. However, the setting options for vertical and horizontal section are greyed out here.



On the **Global settings** tab you can specify the diameter of the fixed point bores. The corner distances of the bores for rivets can now be specified on the **Extended settings** tab.

For the visibly riveted ALUCOBOND tray panels the processings on the panels result from the mounting to the sub-structure. These processings can then be changed on the the **Extended settings** tab.

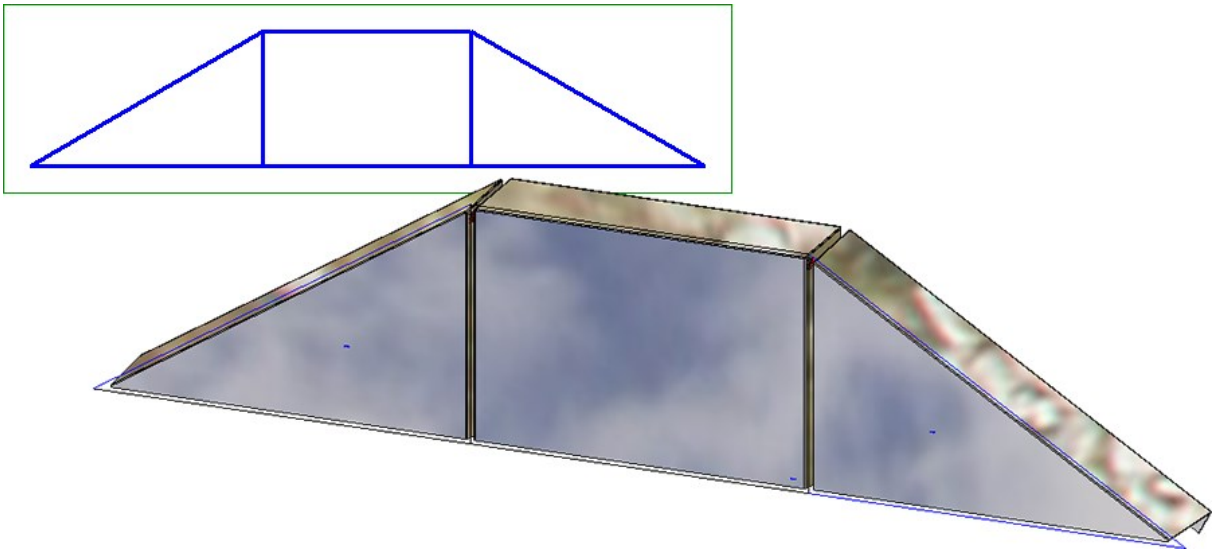


From SP2 onwards, these parameters can even be set if no sub-structure is used for the element installation.

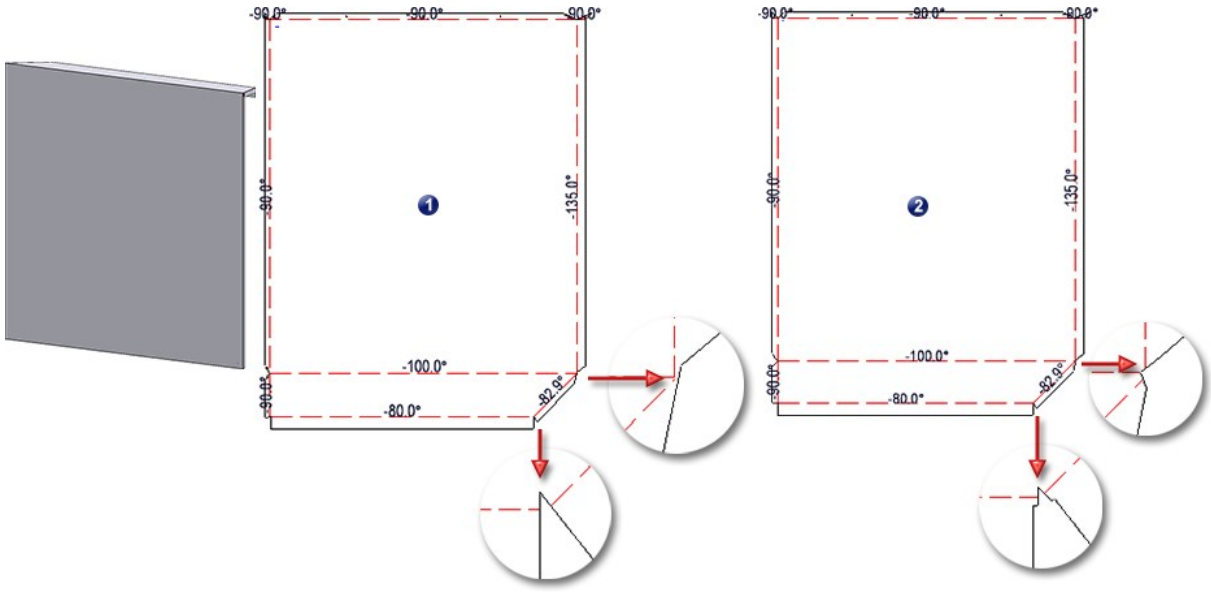
ALUCOBOND® Tray panel SZ-20

ALUCOBOND SZ 20 tray panels can now also be placed on triangular sketch areas.

Please note:
To ensure that a placing on triangular sketch areas will also work with attics and outer corners etc., HiCAD will assume that the hypotenuse at the "top" and the side on the left / right will be the shortest distance. This means that a placing on equilateral triangles is not possible.



Furthermore, milling-oriented bend zone ends are now created for ALUCOBOND SZ 20 tray panels with the help of the new Sheet Metal function **Trim corner**.



(1) Bend zone ends in HiCAD 2019 SP2, (2) Bend zone ends before HiCAD 2019 SP2

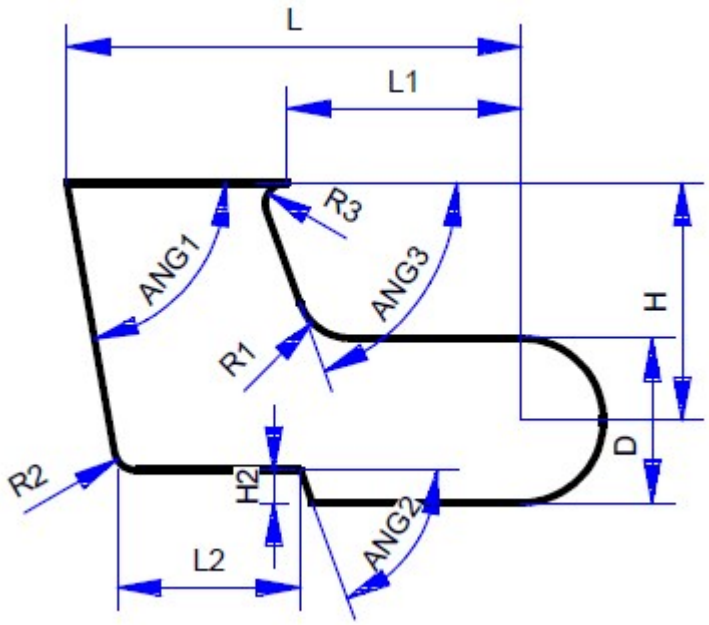
Also new is that the tray panel depth can now be set on the **Extensions for SZ-20 Standard** tab. The specified value applies to all sides.

ALUCOBOND® suspended

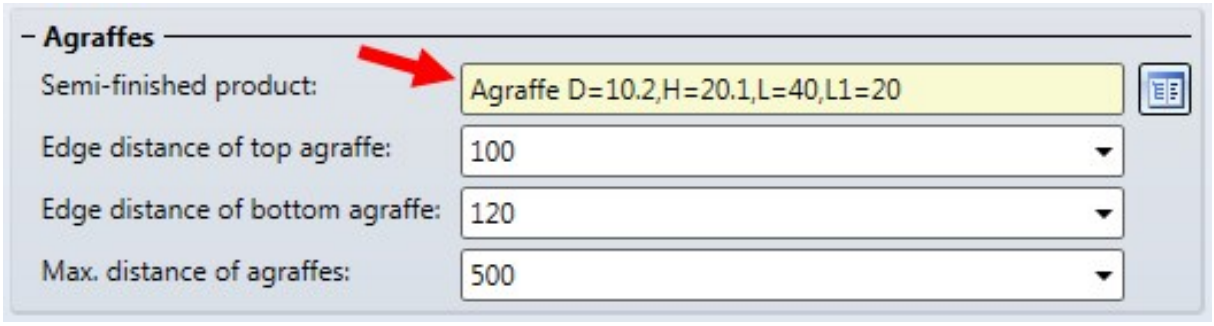
For suspended ALUCOBOND tray panels you can now specify the height and width of agraffes. This setting must take place in the table **Agraffes** of the catalogue **Factory standards > Bore patterns**. In this table two data records have been predefined that you can use as a basis for further sub-types.

	ID	MOD	STATUS	BZ	NAME	ICON	TYPE	L	H	L1	L2	H2
1	1		▸	Agraffe D=10.2,H=20.1,L=40,L1=20	MUSTERBOHRUNGEN\ISD_AGRAFFE_01+Y.KRA	ISD_AGRAFFE_01+Y.JPG	Y+	40	20.1	20	0	0
2	2		▸	Agraffe D=10.2,H=20.1,L=40,L1=20	MUSTERBOHRUNGEN\ISD_AGRAFFE_01-Y.KRA	ISD_AGRAFFE_01-Y.JPG	Y-	40	20.1	20	0	0

Please note that only agraffes based on the part ISD_AGRAFFE_01-Y.KRA, i.e. agraffes of the Type Y-, are allowed here.



The agraffes defined here will then be available on the **Extended settings** tab.



Customer-specific installation elements - Clearance

For customer-specific installation elements the clearance can be preset in the appropriate catalogue table. When choosing the corresponding installation element in the Element installation dialogue, the clearance will be set to this value.

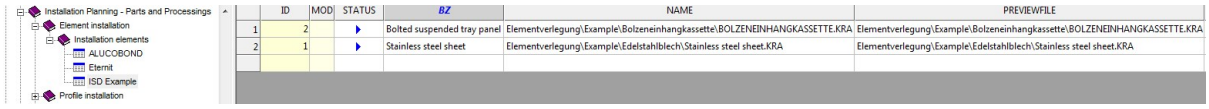
The pre-setting of the clearance takes place via the table columns

- GAPMODE and
- GAPDEFAULT.

You enter the required value for the clearance in the GAPDEFAULT column. If you want to activate this value as the default value, set the column GAPMODE to 1. When choosing the corresponding element in HiCAD for the element installation the value for the clearance will be automatically set to the value entered in the GAPDEFAULT column.

To deactivate this pre-setting, set the column GAPMODE to 0.

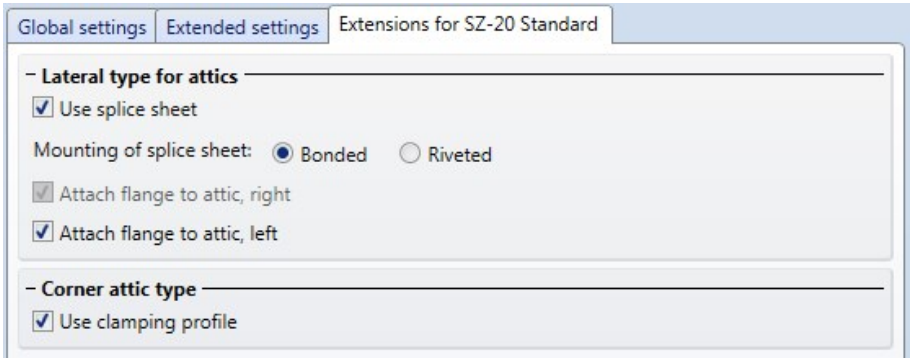
The tables in the ISD Examples catalogue that is supplied with HiCAD already contain these columns.



Service Pack 1 2019 (V 2401)

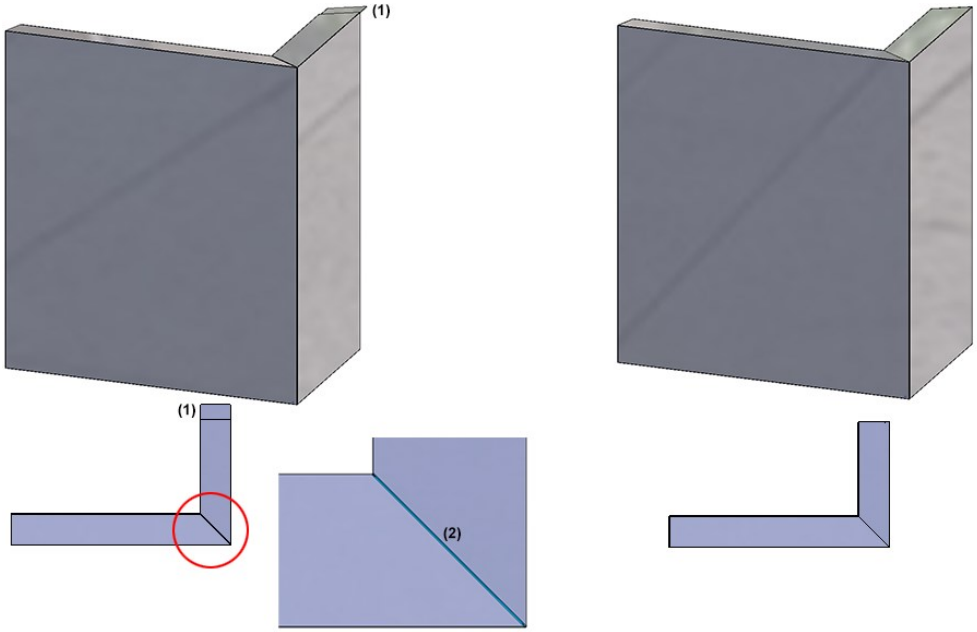
ALUCOBOND® Tray panel SZ-20

- From HiCAD 2019 SP1 onwards, all reinforcement plates will be riveted.
- For the lateral type of attic connections, HiCAD 2019 SP1 will offer additional options on the new **Extensions for SZ-20 Standard** tab. For instance, you can specify here whether flanges are to be attached and if so, on which side, and whether the corner attics are to be created with or without clamping profile.



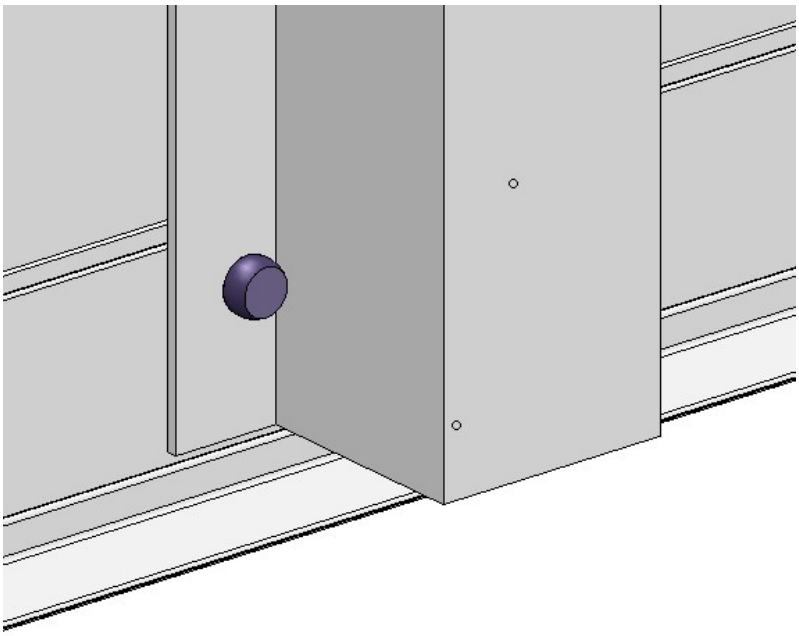
Example:

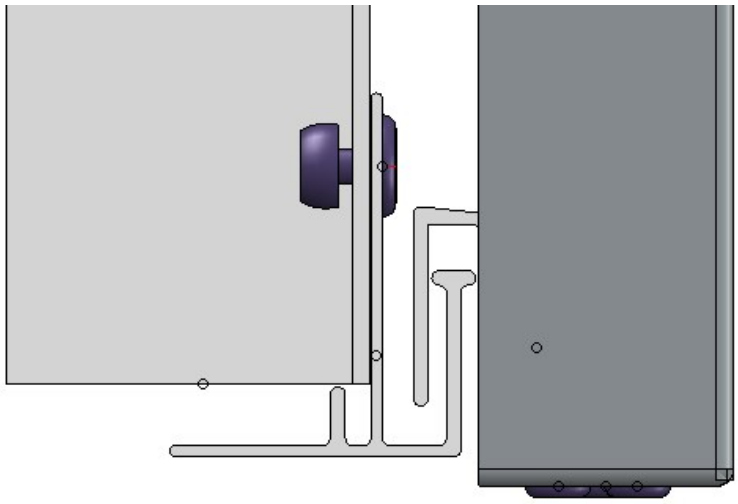
The image below shows a SZ-20 tray panel. Horizontal section: Connection, top. Attic, long; Vertical section: Connection, right. Outer corner with folded sheet. The left tray panel has been created with splice sheet and clamping profile, the right one without these elements.



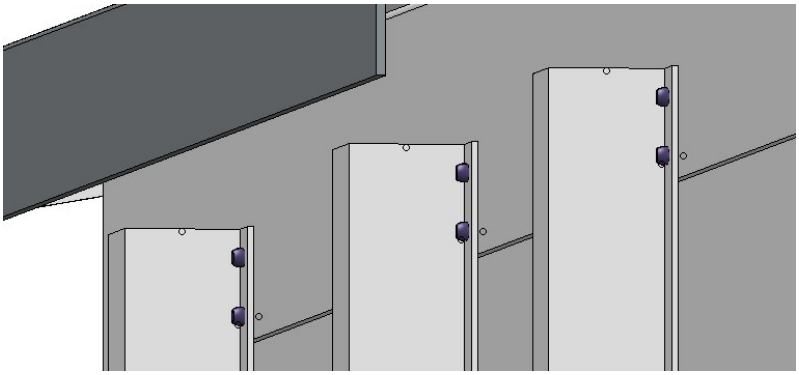
(1) Splice sheet, (2) Corner attic with clamping profile

- When connecting an element installation of the type SZ-20 to a sub-structure while the connection type **End, bottom** has been chosen, the "base profile" will be considered and displayed correctly.



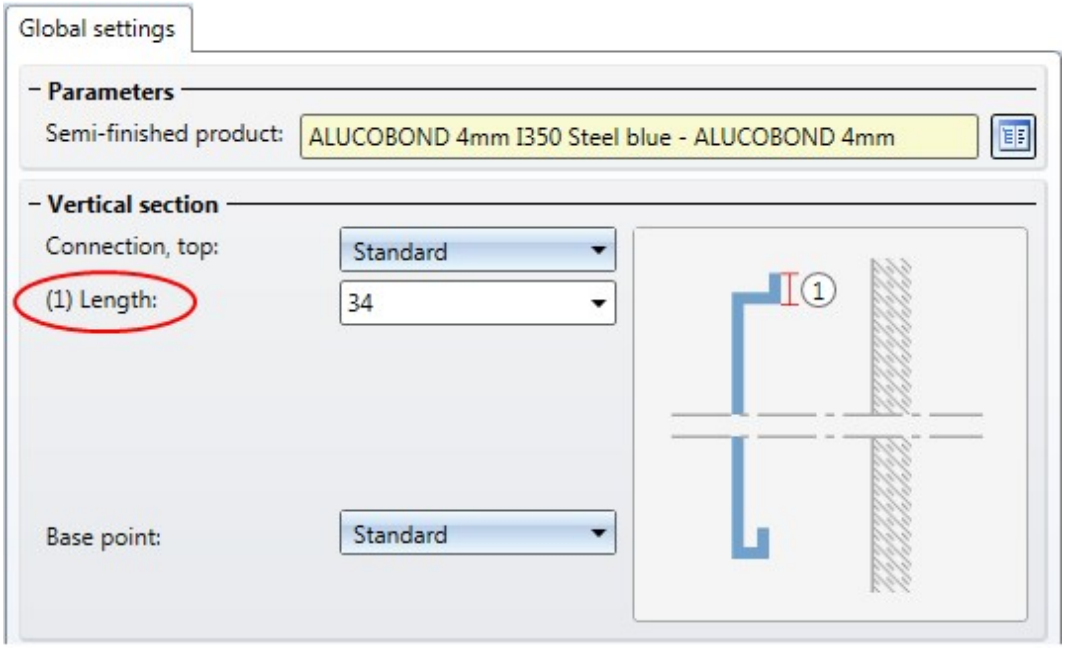


- When connecting an element installation of the type SZ-20 to a sub-structure, while the connection type **Window connection, Parapet** has been chosen, not just bores, but also rivets will be created.



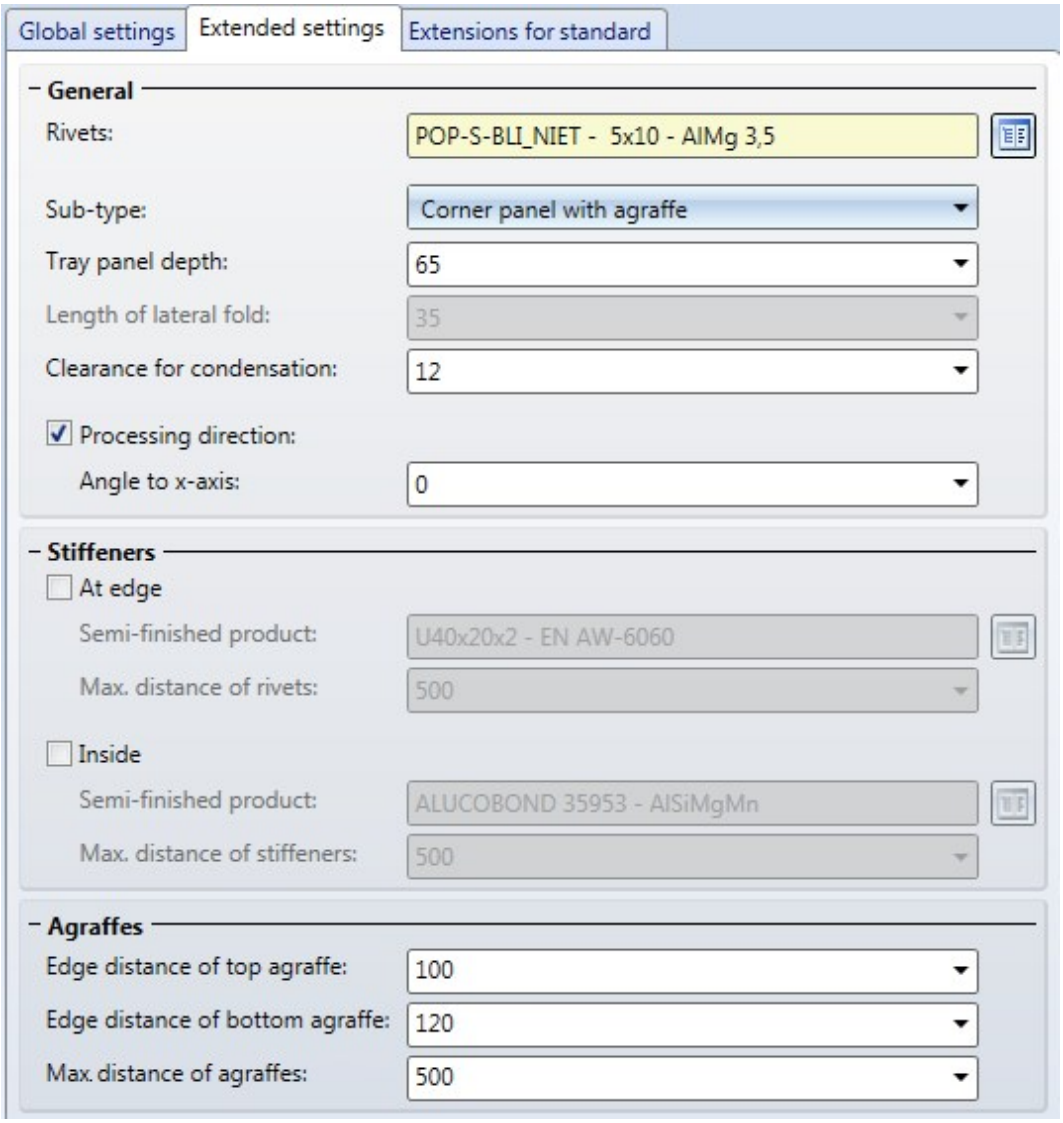
ALUCOBOND® suspended

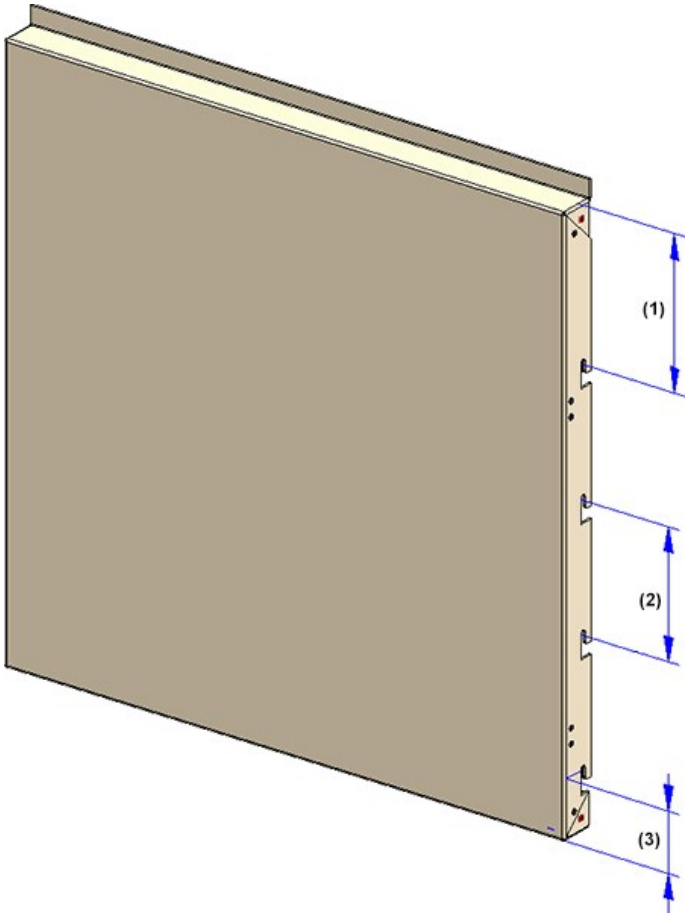
- For this panel type you could previously specify the length of the folding via the **Tray panel height** parameter on the **Extended settings** tab. Since this parameter name was misleading and applies only to the setting Connection, top: **Standard**, the setting can now be specified on the **Global settings** tab in the **Length** field:



- An additional tab called **Extensions for standard** for attic connections is now available for suspended tray panels.

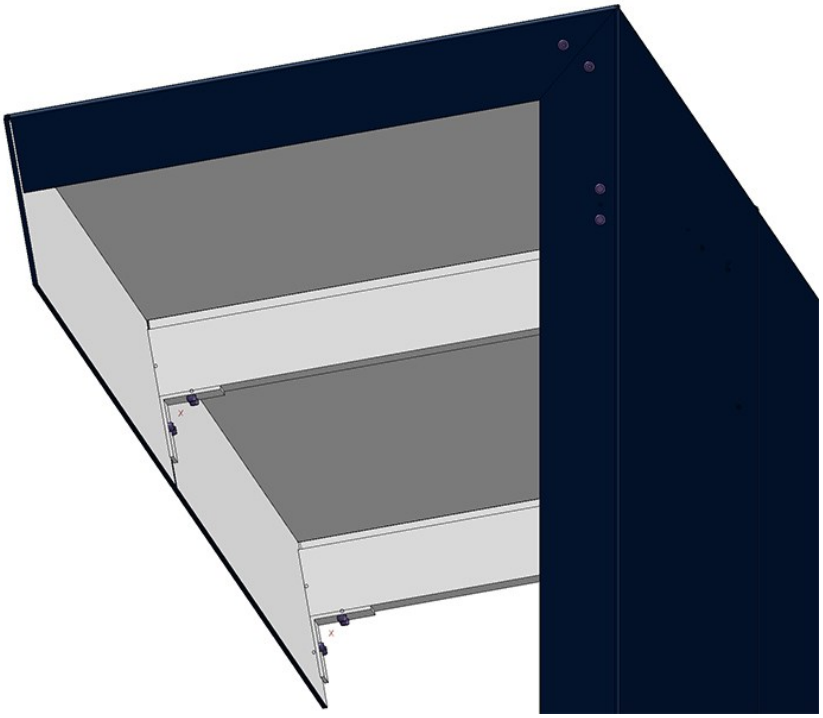
- On the **Extended settings** tab you can now specify
  - the edge distance of the top agraffe,
  - the edge distance of the bottom agraffe, and
  - the maximum distance of agraffes.





(1) Edge distance of top agraffe; (2) Distance between 2 agraffes; (3) Edge distance of bottom agraffe

- When connecting a sub-structure to elements of the connection type **Attic, long**, the corresponding connections, too, will now be correctly visualized.



New test for ALUCOBOND sheets in Design Checker

In the Design Checker a new test for Sheet Metal parts is now available. The new test **Sheet dimensions defined by semi-fin. product (developed)** checks the dimensions for the Material ALUCOBOND. For this purpose the table **Sheet dimensions (defined by semi-fin. prod.)** at **Factory standards > Sheet dimensions** in the Catalogue Editor will be evaluated. Based on this table the maximum length and width depending on the TableID of the semi-finished product will be calculated.

Renamed system variables for element installations and sub-structures

The variables for element installations and sub-structures for the creation of user-defined installation elements and profiles have bee renamed as follows:

Element installation	old	new
Width	width	i_l
Length	height	i_h
X-, Y-distance to origin of Fitting CS	x0, y0, ...	i_x0, i_y0, ...
Number of corners	pointquantity	i_nr
Sub-structure	**old**	**new**
Length	length	i_l

Variants that have been created with the old variable will still work.

Major Release 2019 (V 2400)

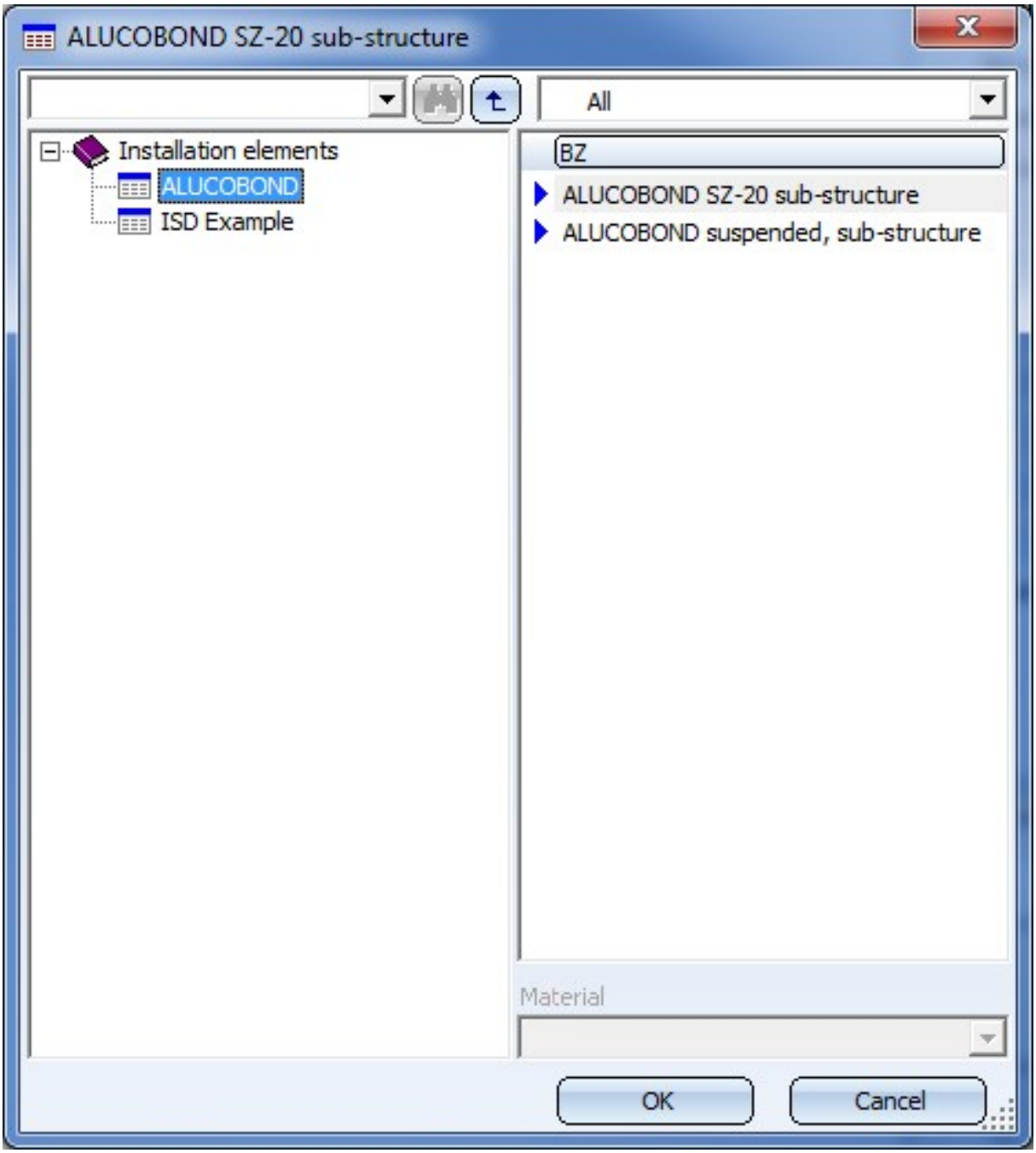
Please note the following:
If you have started an element installation with ALUCOBOND tray panels in an earlier HiCAD version and want to process this installation further in HiCAD 2019, you can only use the new connections and options for ALUCOBOND tray panels if you click on the **Create New** button in the dialogue of the function. However, please note that in this case, dimensions and further processings will be lost on these parts. If you close the dialogue window with **OK** instead, the HiCAD will continue using the "old" ALUCOBOND tray panels, even if you have used the new settings/options in the dialogue window.

Performance

The performance for insertion and processing of element installations could be significantly improved in HiCAD 2019.

ALUCOBOND® SZ20 sub-structure moved

The ALUCOBOND® SZ20 sub-structure that could be found in the **ISD Example** catalogue in previous versions has been moved to an own **ALUCOBOND** catalogue.



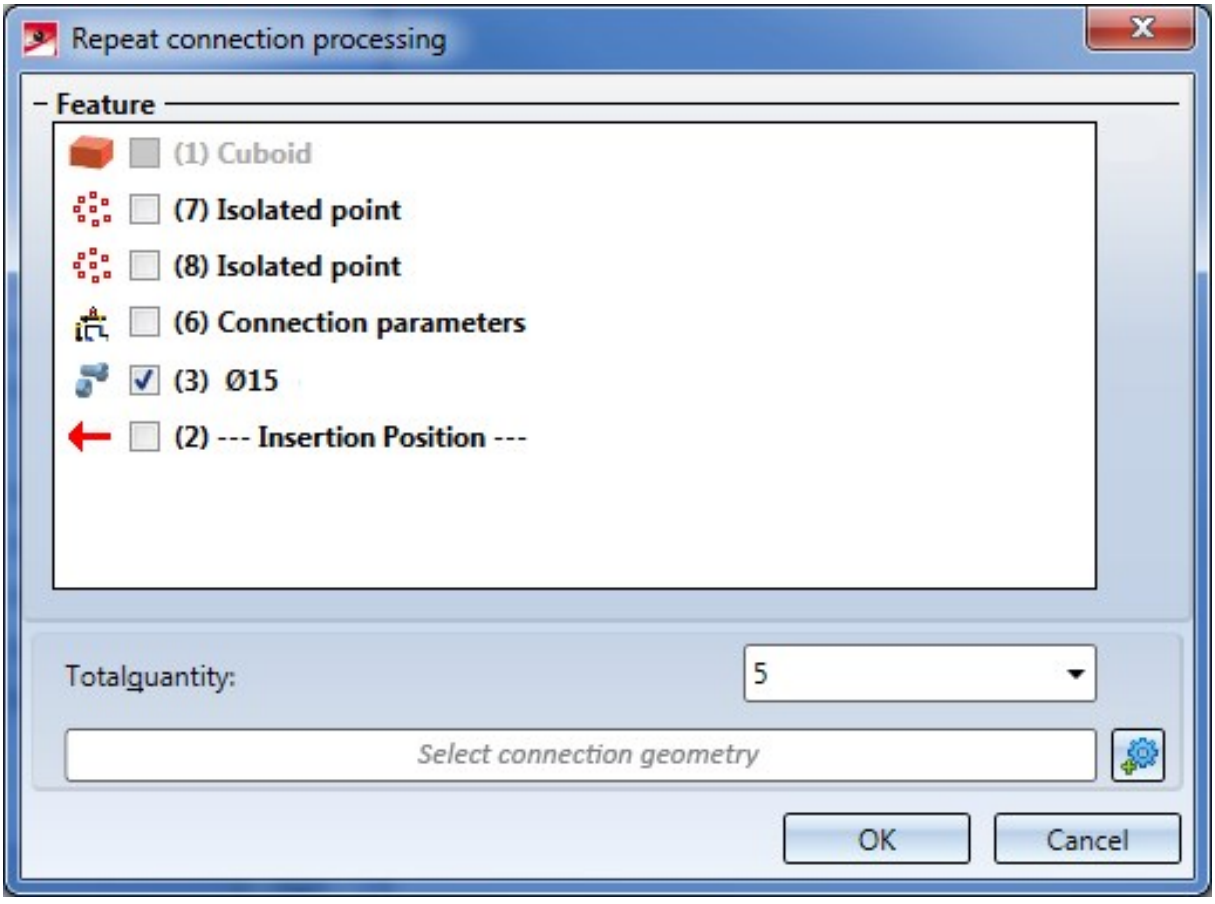
Connection information renamed to Connection parameters

The **Connection information** function which is required for preparing profiles that are to be used in a connection of sub-structures and installed elements, has been renamed to **Connection parameters**.

Nothing has changed concerning the operation of this function.

New auxiliary functions for the creation of user-defined profiles for sub-structures

The new **Repeat connection processing**  function in the **Civil Engineering functions** docking window beneath **Element installation** simplifies the adjustment of parts to enable their connection to a sub-structure or to an installed element.



After calling the function you can choose the connection parameters to be used and the feature steps to be repeated. These will then be repeated as desired; Processing plane and Condition will be adjusted accordingly.

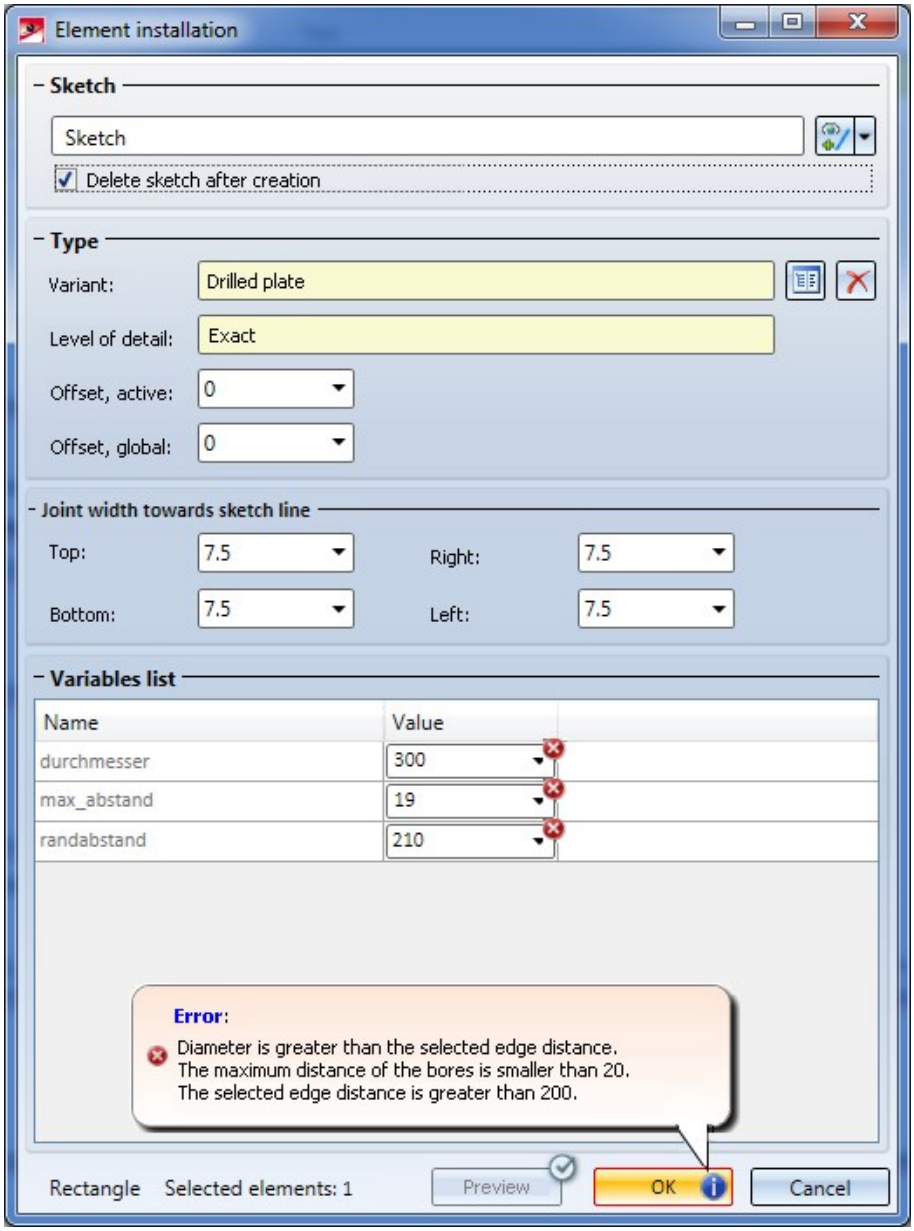
Conditions and error messages in user-defined dialogues

For installation elements you can now assign conditions for the utilized variables. In addition, error messages can be defined that are displayed if the conditions for the insertion of the element are not fulfilled. In this case the respective input field or the OK button will be marked with the or symbol. When you move the cursor over these symbols, the corresponding error message will be shown.

The conditions and error messages for the variables must be stored in a file the name of which must be entered in the CONDITIONS column.

Example:

```
1  randabstand|randabstand >= 50|The selected edge distance is smaller than 50.
2  randabstand|randabstand <= 200|The selected edge distance is greater than 200.
3  max_abstand|max_abstand >=20|Maximum distance of bores is less than 20.
4  durchmesser|durchmesser > randabstand|Diameter is greater than or equal to the selected edge distance.
```

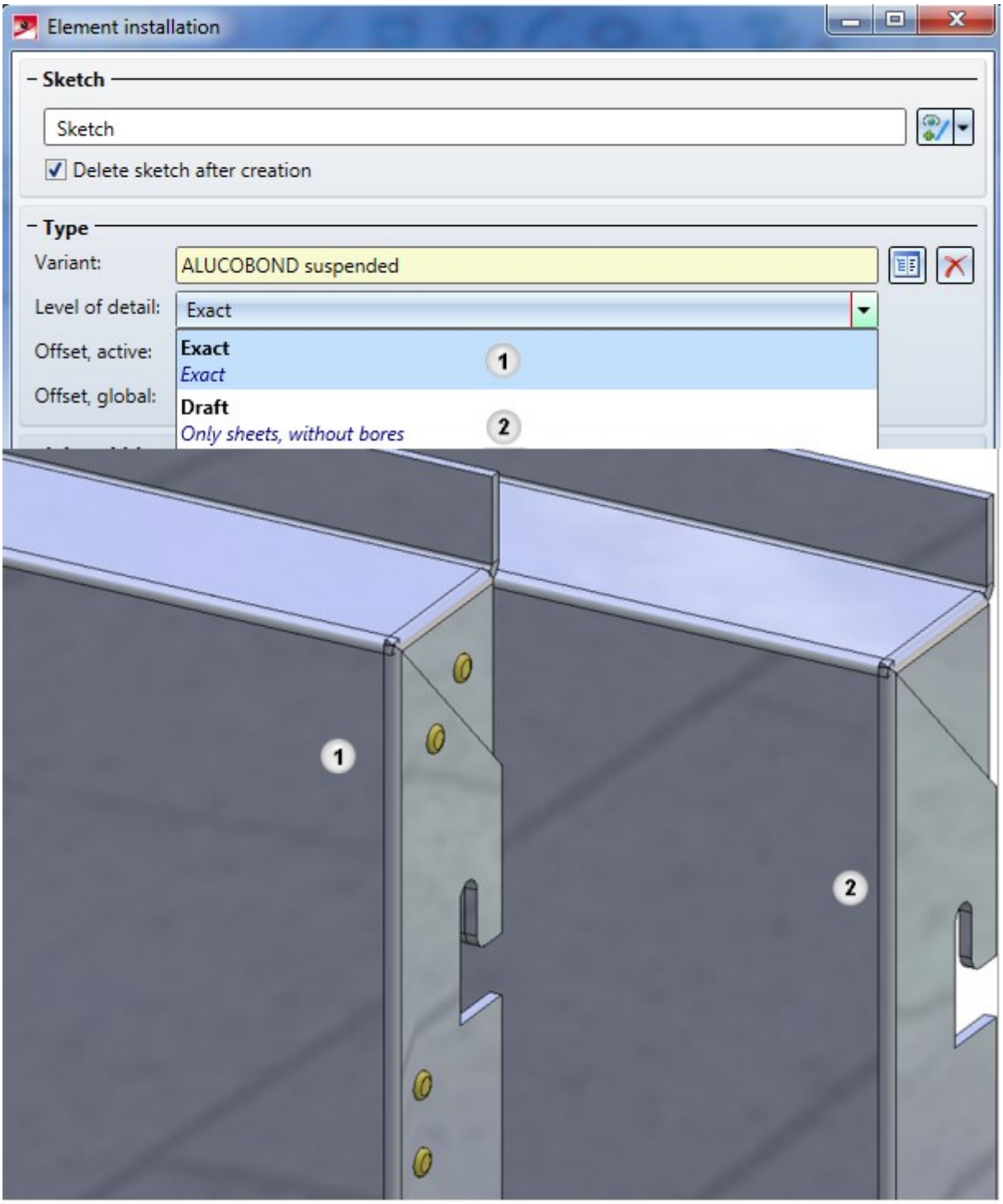


Extensive information on conditions and error messages can be found in the topic Customizing the Dialogue Window.

Suspended ALUCOBOND® panels

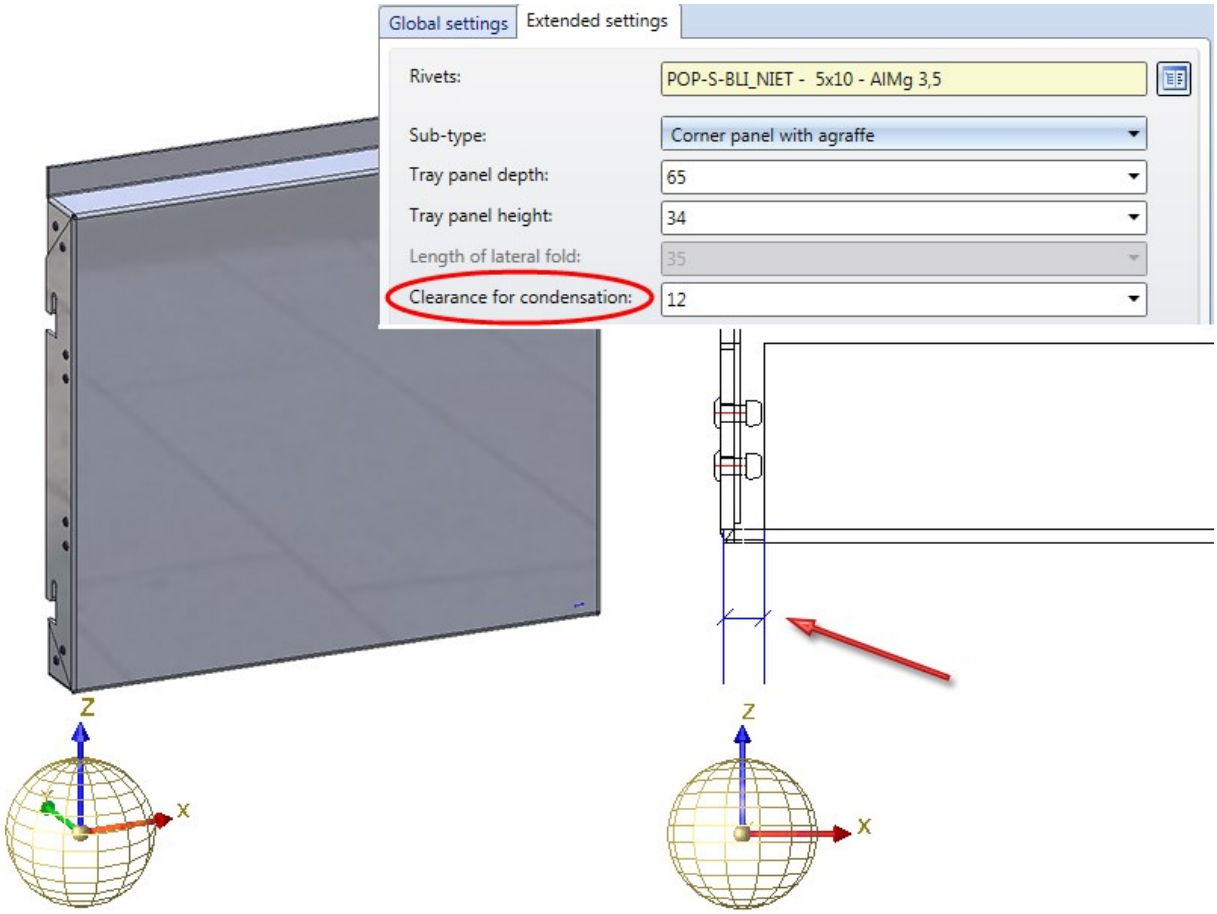
Level of detail

For suspended ALUCOBOND panels, too, you can now select the **Level of detail** can now be selected, i.e. you can choose between **Exact** or simplified representation (**Draft**). The representation as draft can be useful if you want to increase the performance in drawings with very many panels. The representation can be changed back at any time by processing of the element installation.



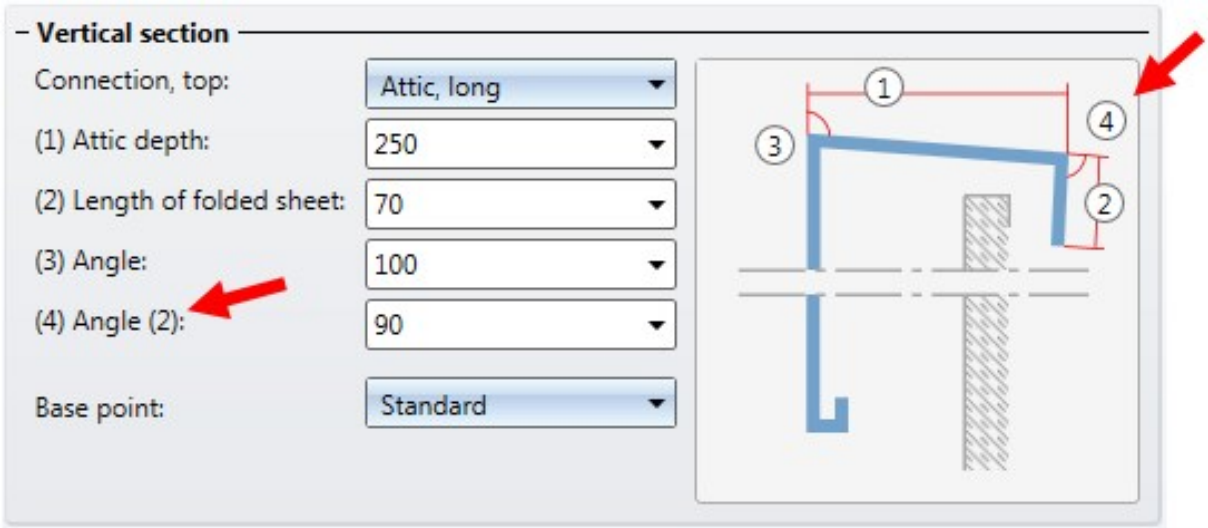
Clearance for condensation

On the **Extended settings** tab of the dialogue window you can now specify a new **Clearance for condensation** parameter. The default setting is 12 mm.



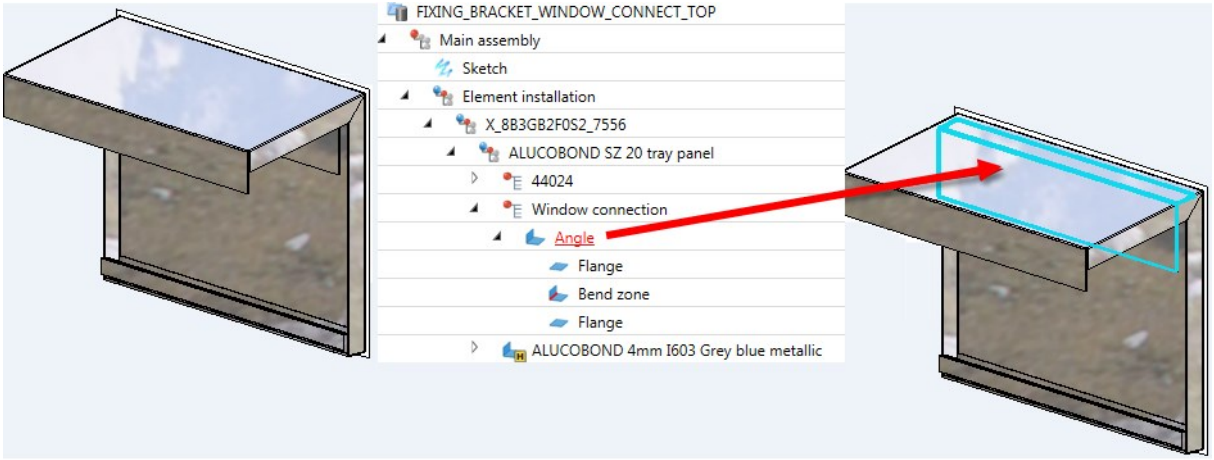
Attics - Fold angle specification

You can now specify a fold angle for attics.

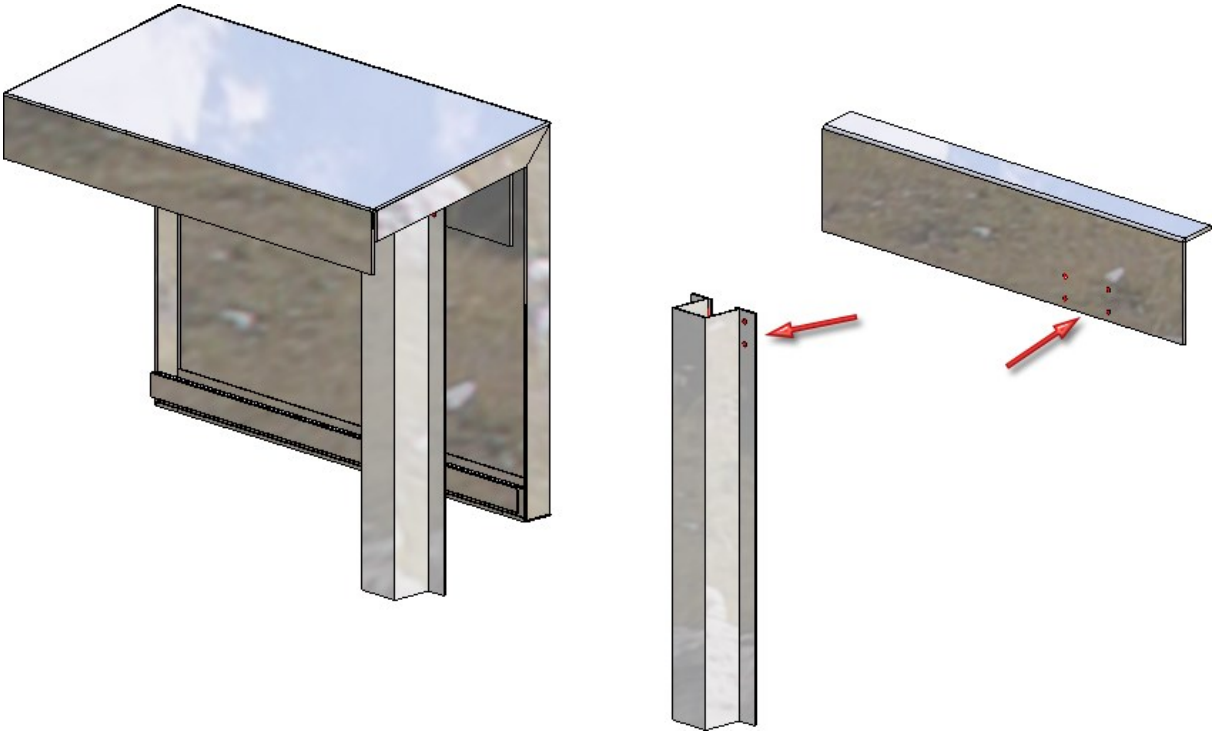


ALUCOBOND® SZ 20 tray panels

If you choose the option **Vertical section - Connection, top: Window connection, parapet** when installing ALUCOBOND SZ 20 tray panels, an additional, angular fixing bracket will now be inserted.



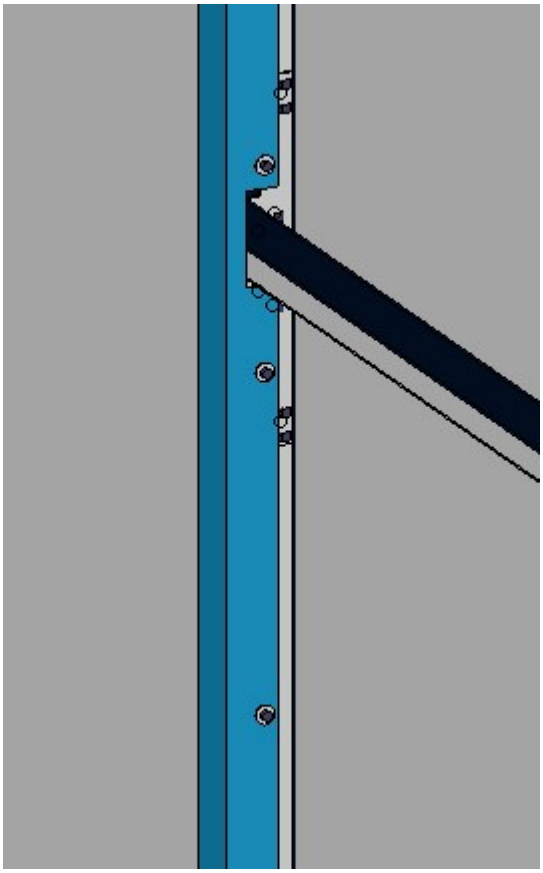
If you connect such an element installation with a sub-structure, the sub-structure will be shortened accordingly and bores will be applied to the substructure and the fixing bracket.



For **Horizontal sections, Connection left/right** you can now choose the option **Window connection with L-profile**. In this case, an additional L-profile will be inserted. This option replaces the same-named checkbox on the **Extended settings** tab.

ALUCOBOND suspended, sub-structure

A new sub-structure of the type **ALUCOBOND suspended, sub-structure** is now available. This part can be mounted to element installations of the type ALUCOBOND, suspended by means of the **Connection** function; recesses for the suspension will be created automatically in the process.

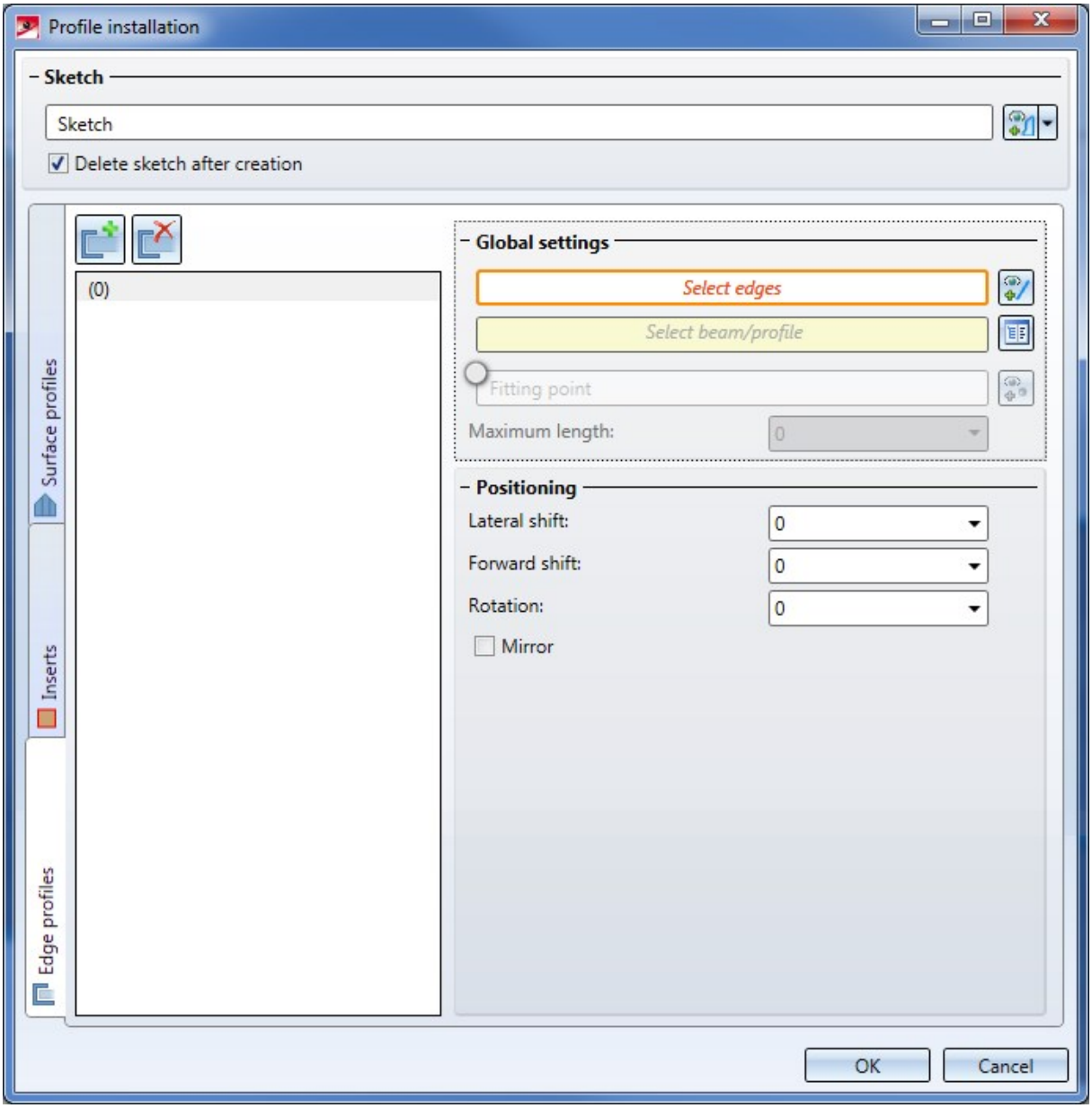


Profile Installation

Service Pack 2 2019 (V 2402)

Edge profiles

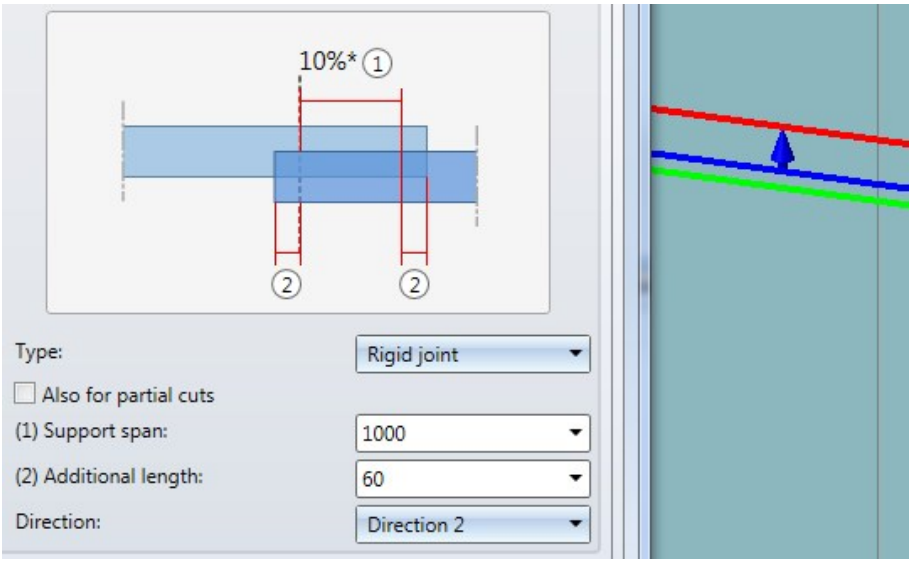
Edges of profile installations can be finished with **Edge profiles**. A same-named tab has been added to the **Profile installation** dialogue window for this purpose.



Here you can create as many edge profile installations as you want and with different settings. Besides a **Maximum length** of the profiles you can define a **Rotation** and a **Shift** in lateral and forward direction.

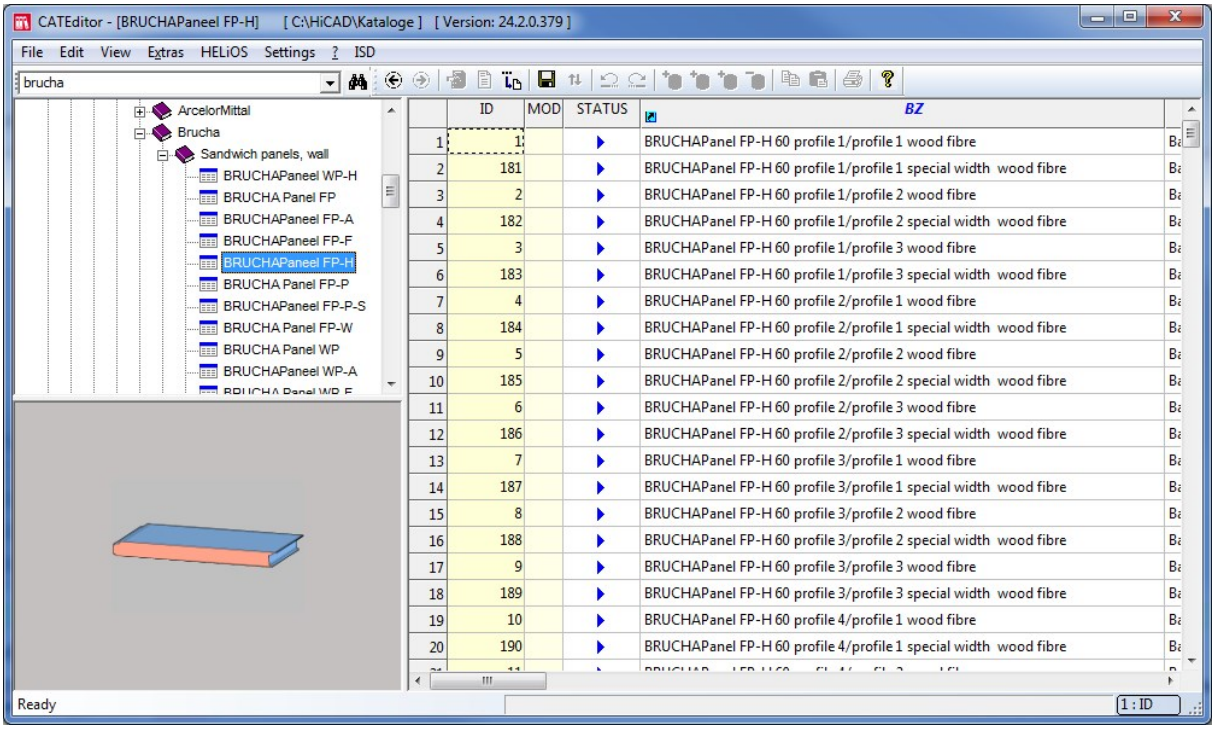
Visualization for rigid joints

If you use the joint type **Rigid joint** type for transverse joints, the selected direction and the selected distances will be visualized directly in the graphic by means of an arrow and cutting lines.



New BRUCHA panels

New BRUCHA panels with different sizes have been added to the HiCAD catalogues and can be used for Profile Installation. You can find them at **Factory standards > Series > Roof wall facade > Room-closing profiles > Brucha**.

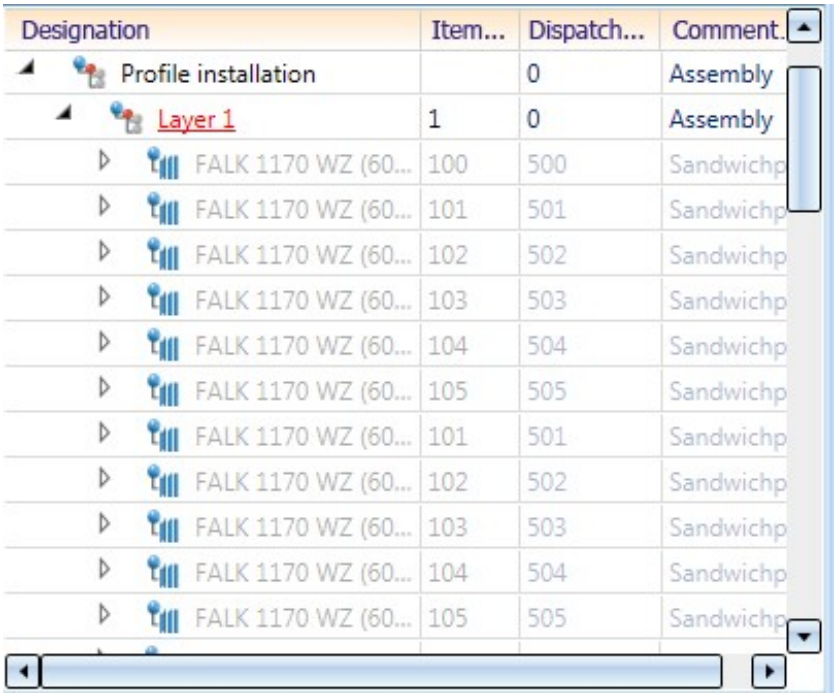


Show dispatch item numbers in ICN

Thanks to configurable user-defined columns in the ICN it is now possible to display the dispatch item numbers directly in the ICN.

The file SYS/ICN3D_USER1.HDB already contain entries for the dispatch item number, which only need to be activated by changing the comment characters as hown below:

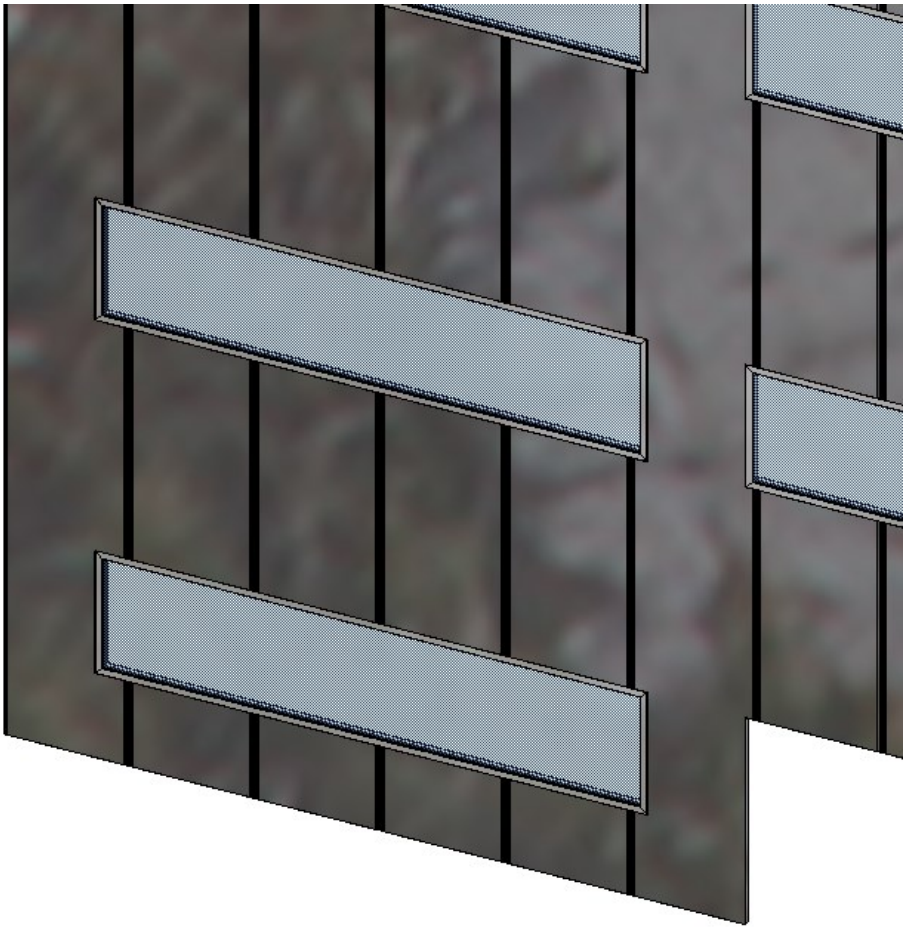
```
# HDB::HEADER="dwf_totalNumber"::SORTTYPE="INTEGER"
HDB::HEADER="Versand-Pos..."::SORTTYPE="INTEGER"
#
# ::SORTTYPE="INTEGER" or "DOUBLE" or "STRING"
#
# Configuration file for the output of any number of
# HiCAD part attributes instead of the part name in the 3-D browser.
# TEXT - Commentary ( will not be displayed )
# ATTR - Attribute to be displayed
# TYP   - STRING(default), DOUBLE
# NKS   - Decimal places
# <H> - HiCAD part attribute
# <E> - Features

# <H>::TEXT=""::ATTR="%06"::TYP="INTEGER"
<H>::TEXT=""::ATTR="%PI"::TYP="INTEGER"
# <H>::TEXT=""::ATTR="§01"::TYP="DOUBLE"::NKS="2"
```



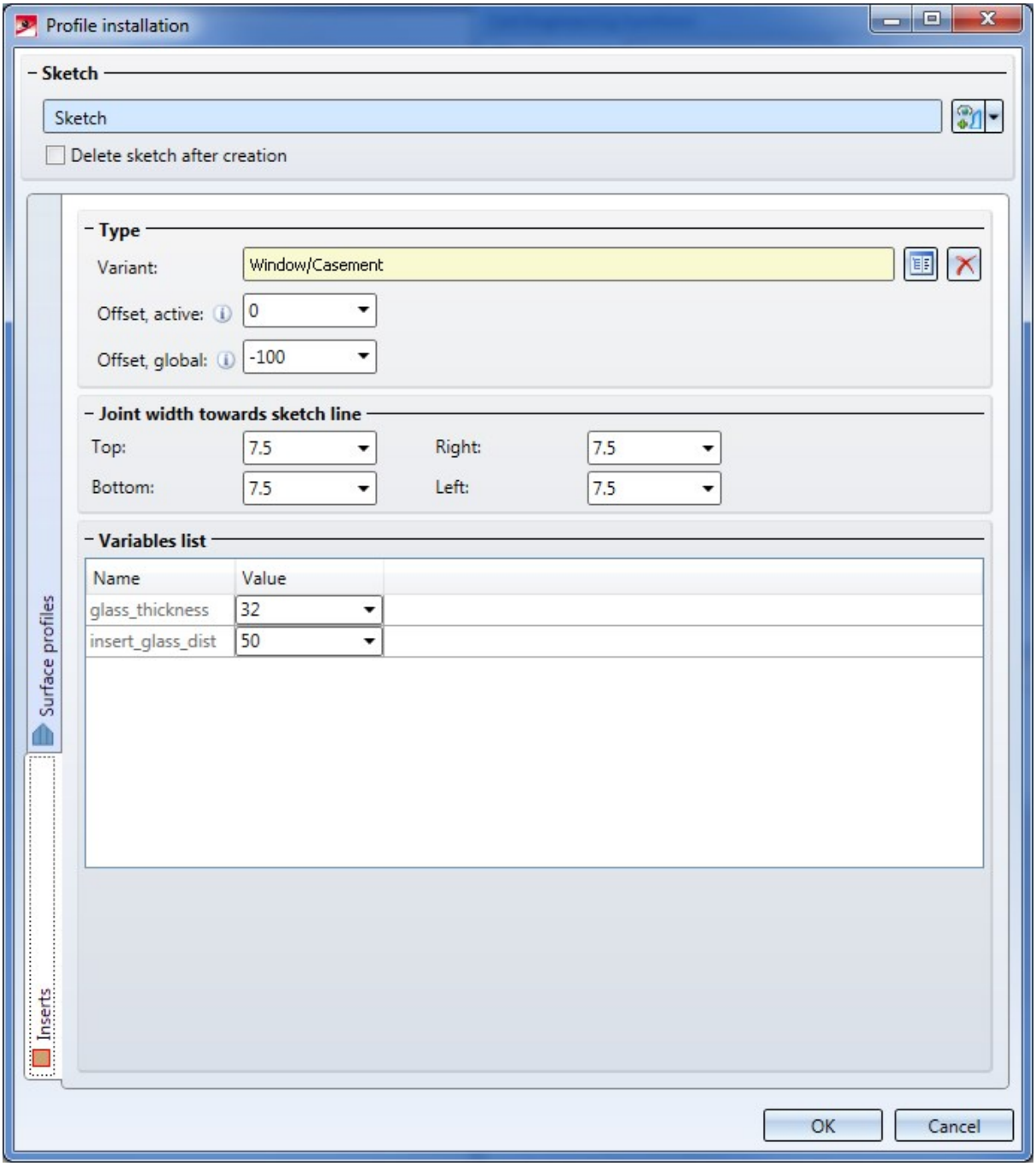
Service Pack 1 2019 (V 2401)

Automatic creation of inserts

You have now the option to automatically place inserts in openings of profile installations.

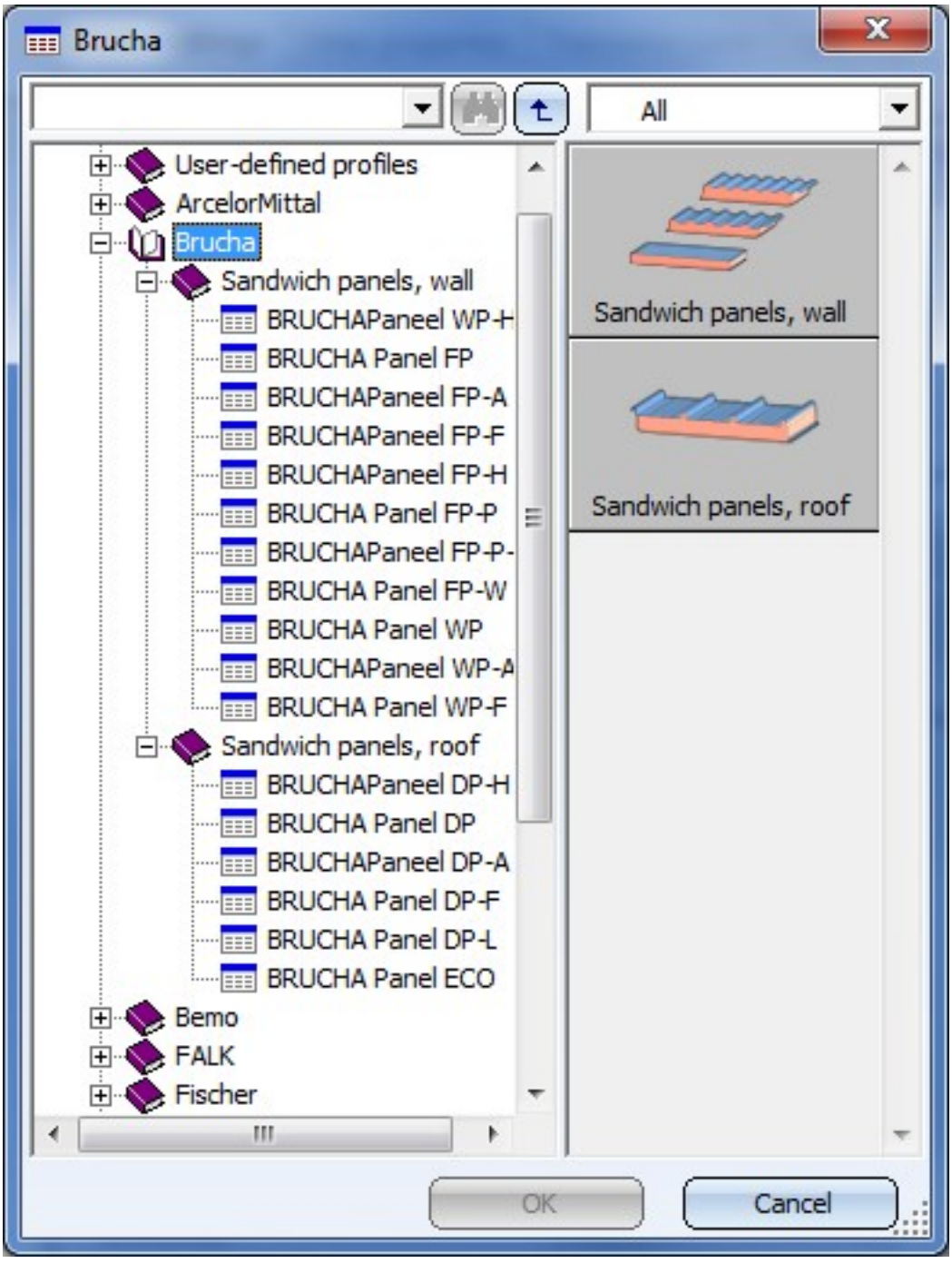


For this purpose, a new side tab called **Inserts** has been added to the **Profile installation** dialogue window, allowing you to configure the inserts.


Profile installation
Sketch
Sketch
Delete sketch after creation
Type
Variant:
Window/Casement
Offset, active:
0
Offset, global:
-100
Joint width towards sketch line
Top:
7.5
Right:
7.5
Bottom:
7.5
Left:
7.5
Variables list
Name
Value
glass_thickness
32
insert_glass_dist
50
Surface profiles
Inserts
OK
Cancel

New catalogue: BRUCHA profiles

Profiles of the company BRUCHA have been added to the catalogue.

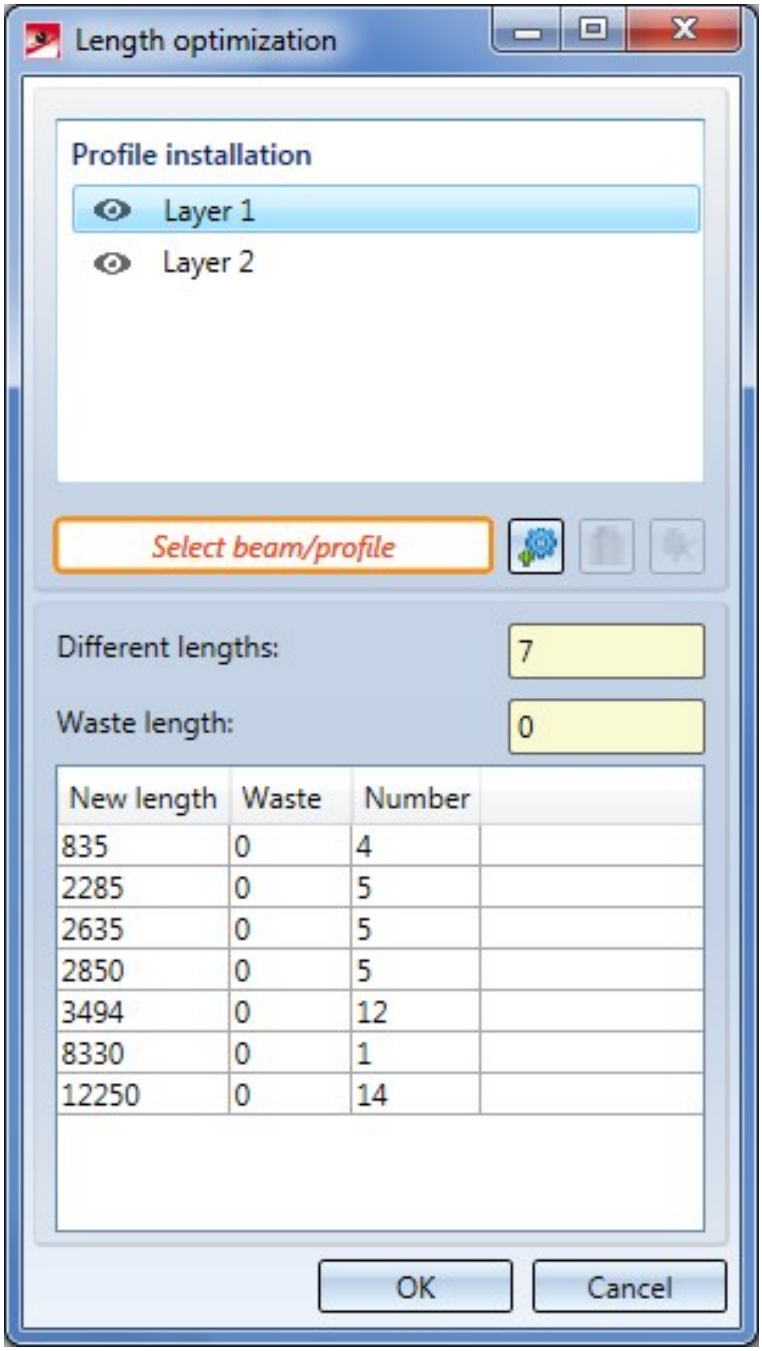


17 profile series have been added to the catalogue, and the number of default profiles has been increased from 913 to 2253.

Improved Length optimization function

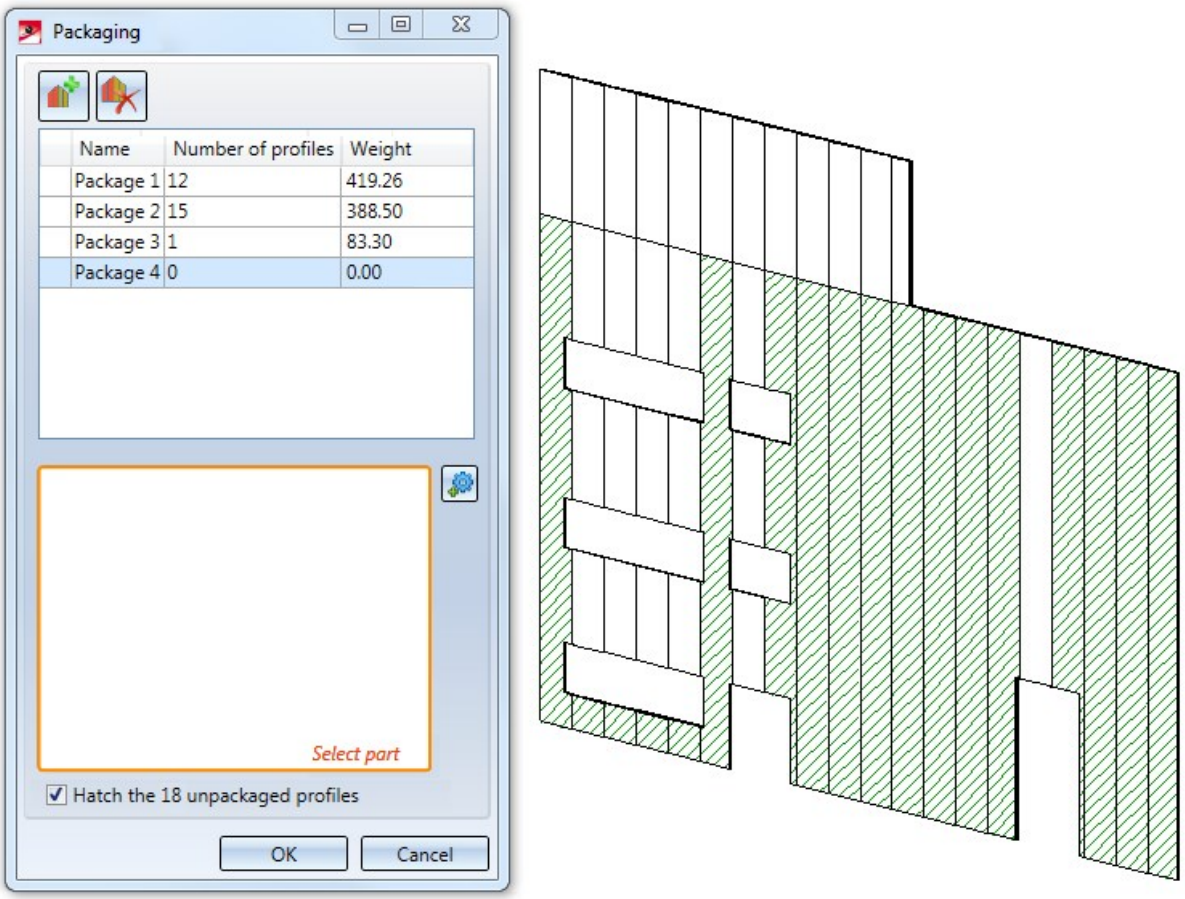
The **Length optimization** has been improved to make its operation easier. You have now the option to add all profiles of one length to the active selection via the context menu function **Add profile(s) to selection**.

Also, the number of different profile lengths and the total waste will now be indicated in the middle of the dialogue window.



Hatching of unpackaged profiles

Searching for unpackaged profiles with the **Packaging** function, too, has been made easier:At the bottom of the dialogue window you can find the **Hatch the x unpackaged profiles** checkbox, with **x** being the number of profiles in the model drawing to which no package has been assigned yet. If you activate this checkbox, all affected profiles will be hatched.



Plant Engineering

Service Pack 2 2019 (V 2402)

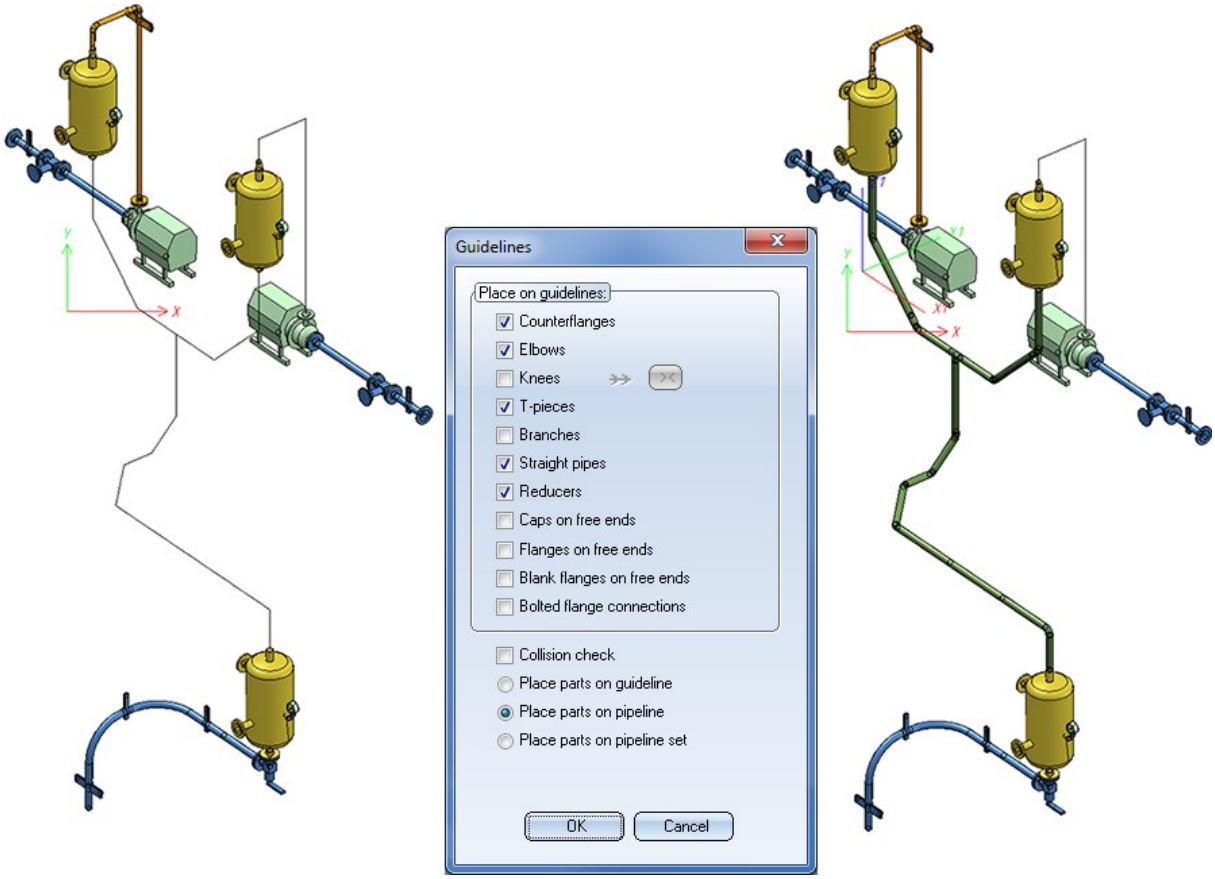
Performance

Automatic placing of parts on guidelines

The redesigned function for the automatic placing of parts on guidelines comes with a significantly increased performance.

Example 1:

Options: With weld seams, maximum pipe length 700, no connecting parts, part cache active and filled, data source: HELiOS:



Previously	SP2
12sec	2sec, i.e. **16%** of former time

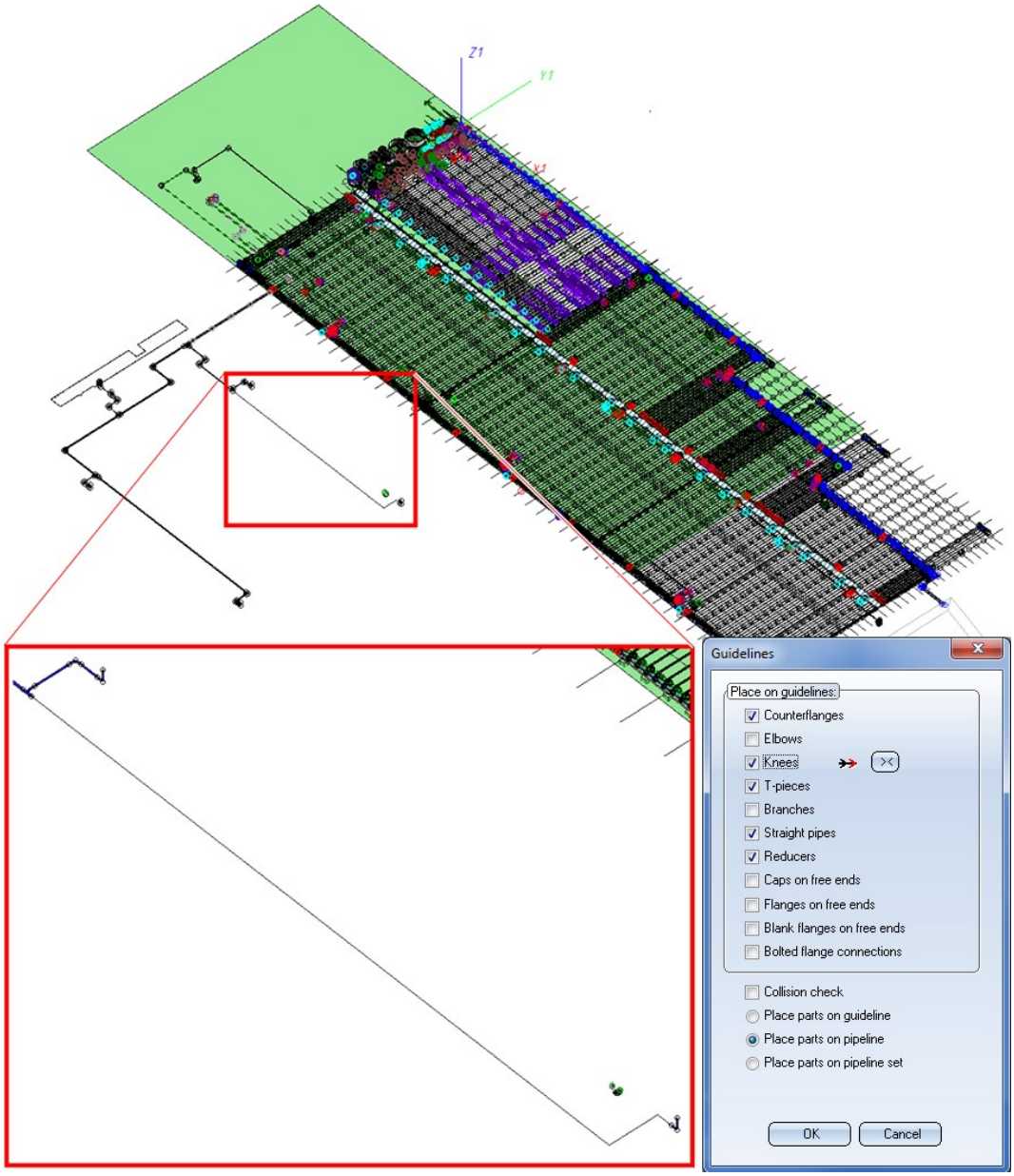
Example 2:

The image below shows a model drawing by the company Certhon Build B.V. (Poeldijk, The Netherlands). The highlighted section is to be occupied with parts.

Options: With weld seams, no connecting parts, part cache active and filled, data source: HELiOS:

The parts that are to be placed on the guidelines are also parts made by Certhon, with the utilized pipe class containing the following parts according to EN 10219:

- Pre-insulated elbows
- Pre-insulated reducers
- Pre-insulated pipes

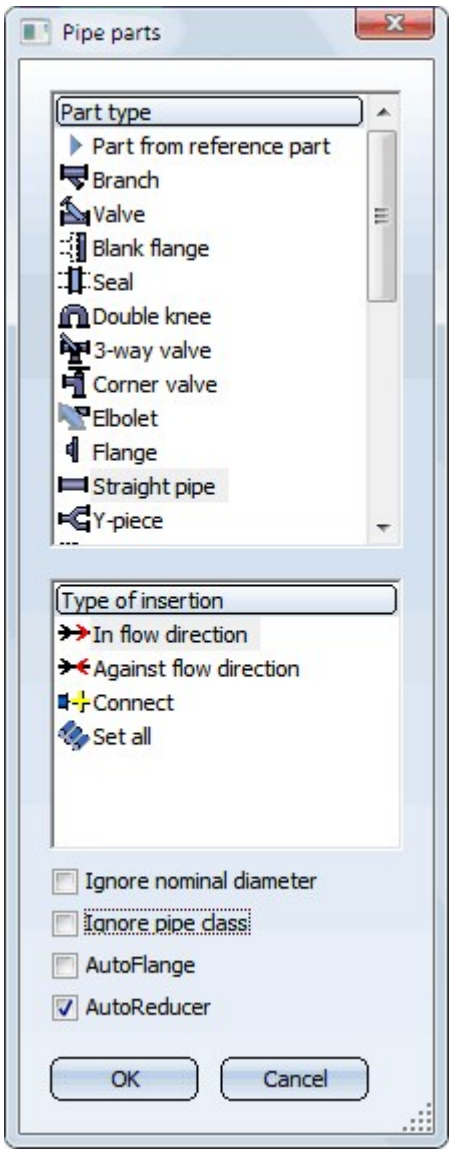


Test		Previously	SP2
1	Maximum length for straight pipes: 10m.	2min	10sec, i.e. **8%** of former time
2	Maximum length for straight pipes: 1m.	17min	15sec, i.e. **0.5%** of former time

Parts

Part insertion

When inserting parts you have now the option to temporarily ignore the pipe class. For this purpose the **Ignore pipe class** checkbox has been added in the selection window for pipe parts. The default setting of this checkbox can be set on the **Part selection** tab of the **Plant Engineering Settings** dialogue window.



The choice of the pipe class takes place according to the following principle:

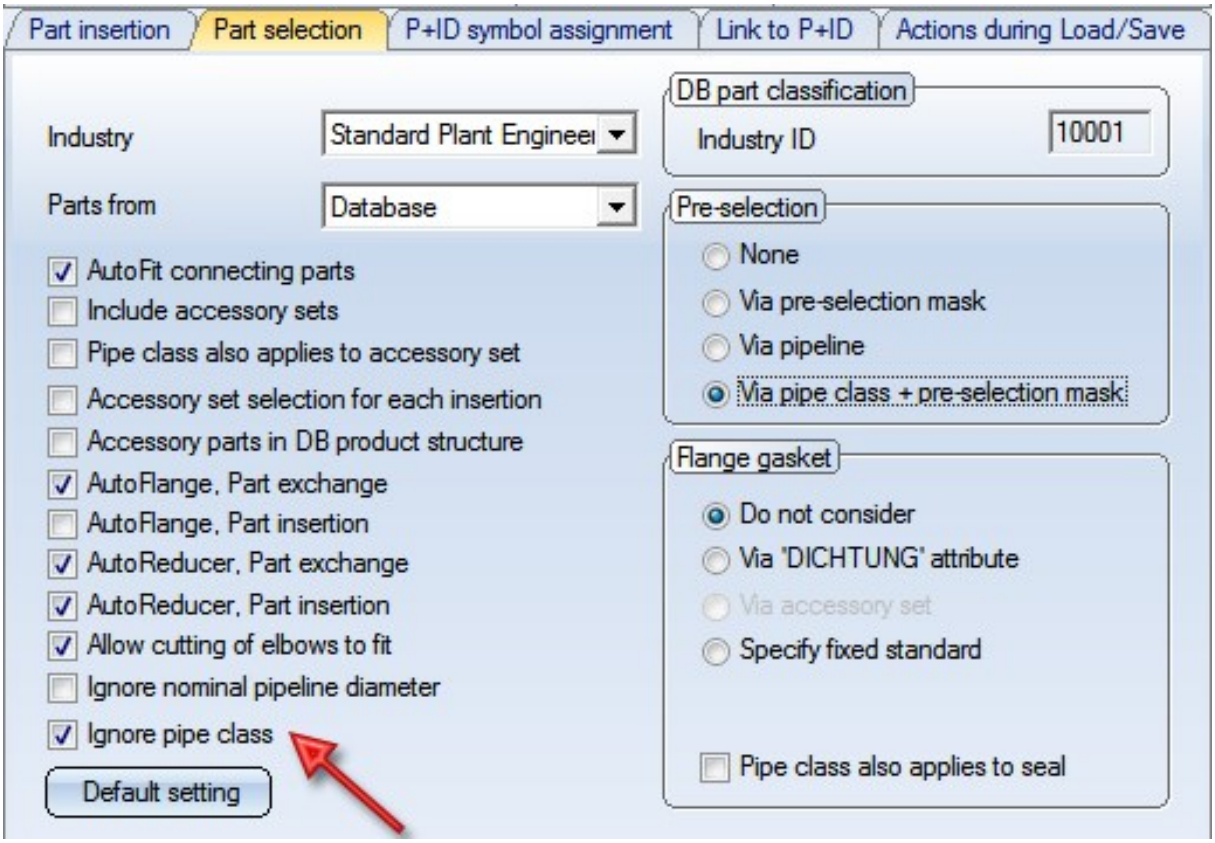
- If the **Ignore pipe class** checkbox has been activated, no pipe class will be used for the part search.
- If the **Ignore pipe class** checkbox has been deactivated, the handling of the pipe class depends on the settings in the **Pre-selection** area:
  1. If no pre-selection has been chosen (**None**) , no pipe class will be used.
  2. If the pre-selection is to be made **Via pre-selection mask**, only the pipe class from the pre-selection mask will be used.

3. If the pre-selection is to be made **Via pipeline**, only the pipe class from the pipeline will be used.
4. If the pre-selection is to be made **Via pipe class + pre-selection mask**, the pipe class from the pre-selection mask will be used. If no pipeclass has been entered here, the pipe class of the pipeline will be used instead.

Settings

Plant Engineering Settings - Part selection

New on this tab is the **Ignore pipe class** checkbox. This checkbox determines whether or not the pipe class is to be ignored when you insert parts.

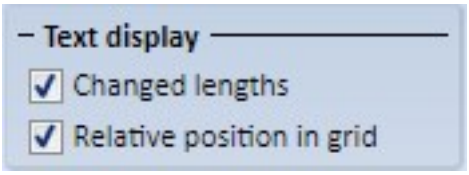


Pipeline Tools

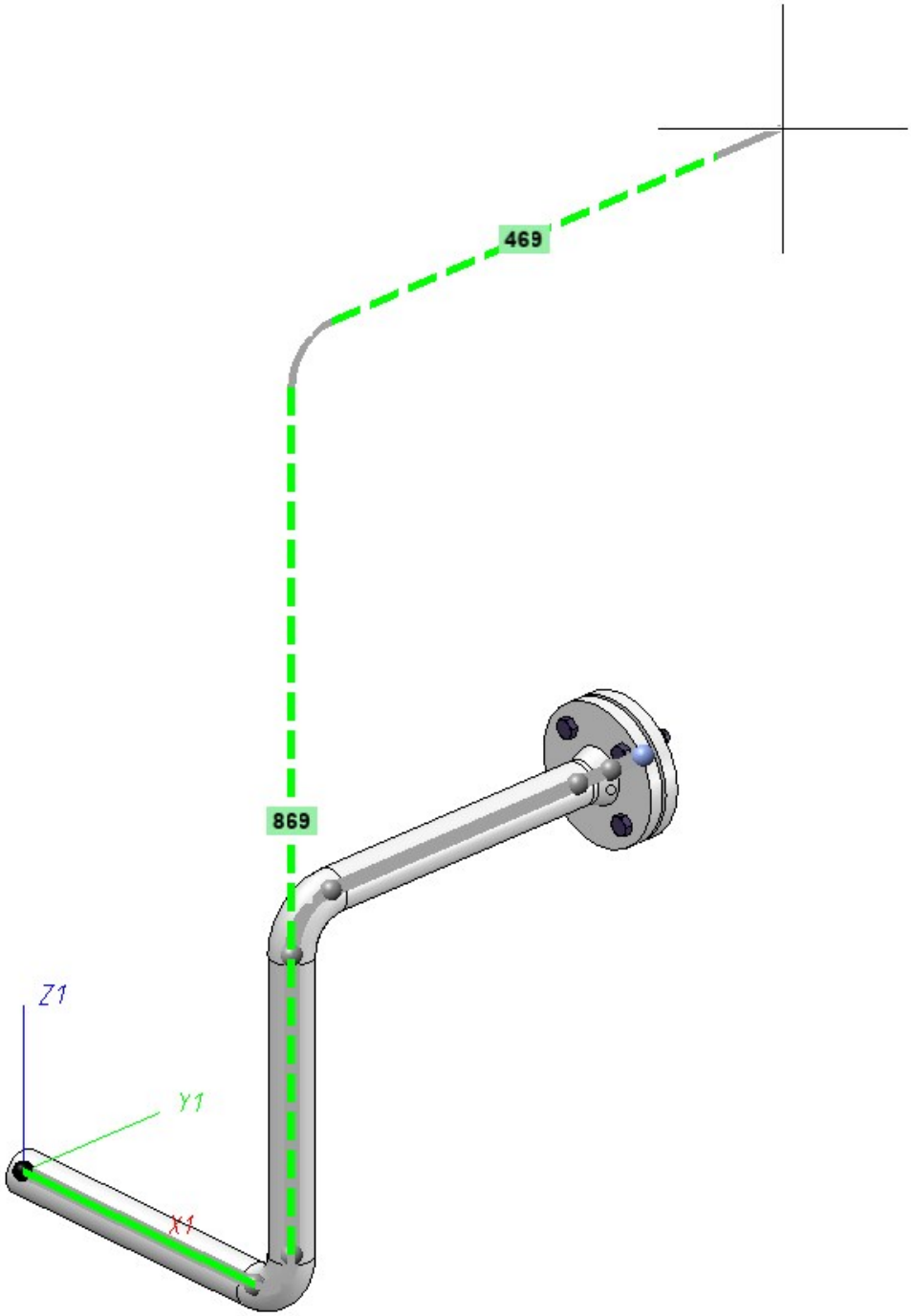
Dynamic route change - Grid and improved length control

Changed lengths

Sections the length of which have been changed by a new route will now be represented as dashed lines. Also, you have now the option to show additional information by activating the corresponding checkboxes beneath **Text display** in the **Change route** dialogue window.

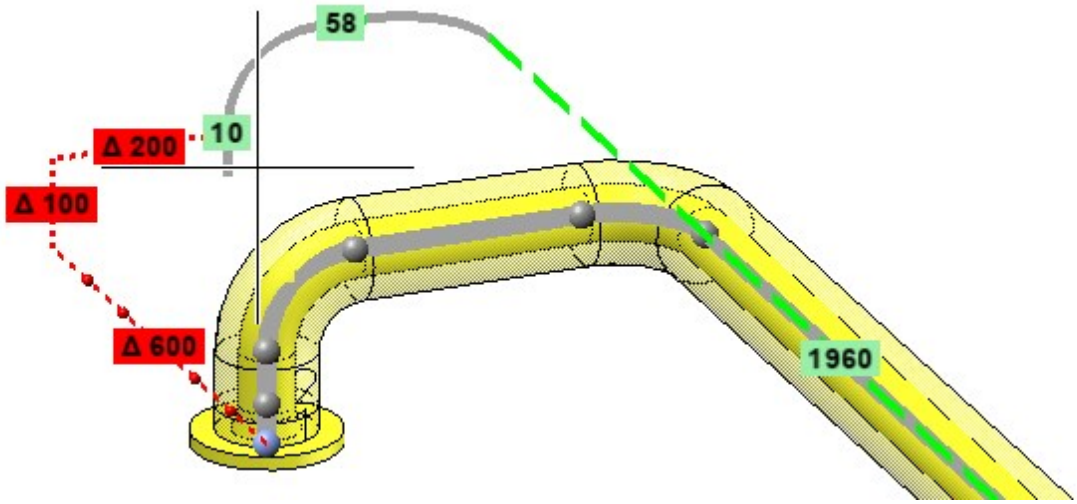


If you want to display the changed lengths as texts, activate the **Changed lengths** checkbox.


469
869
Z1
Y1
X1

Relative position in grid

If the **Use grid** checkbox is active, the grid will be visualized by dotted lines. If the **Relative position in grid** is also active, the distance to the start point will be written, along the grid axes, to the auxiliary lines. To distinguish such distances from the texts for the changed lengths, they will be preceded by a capital Δ (Delta).

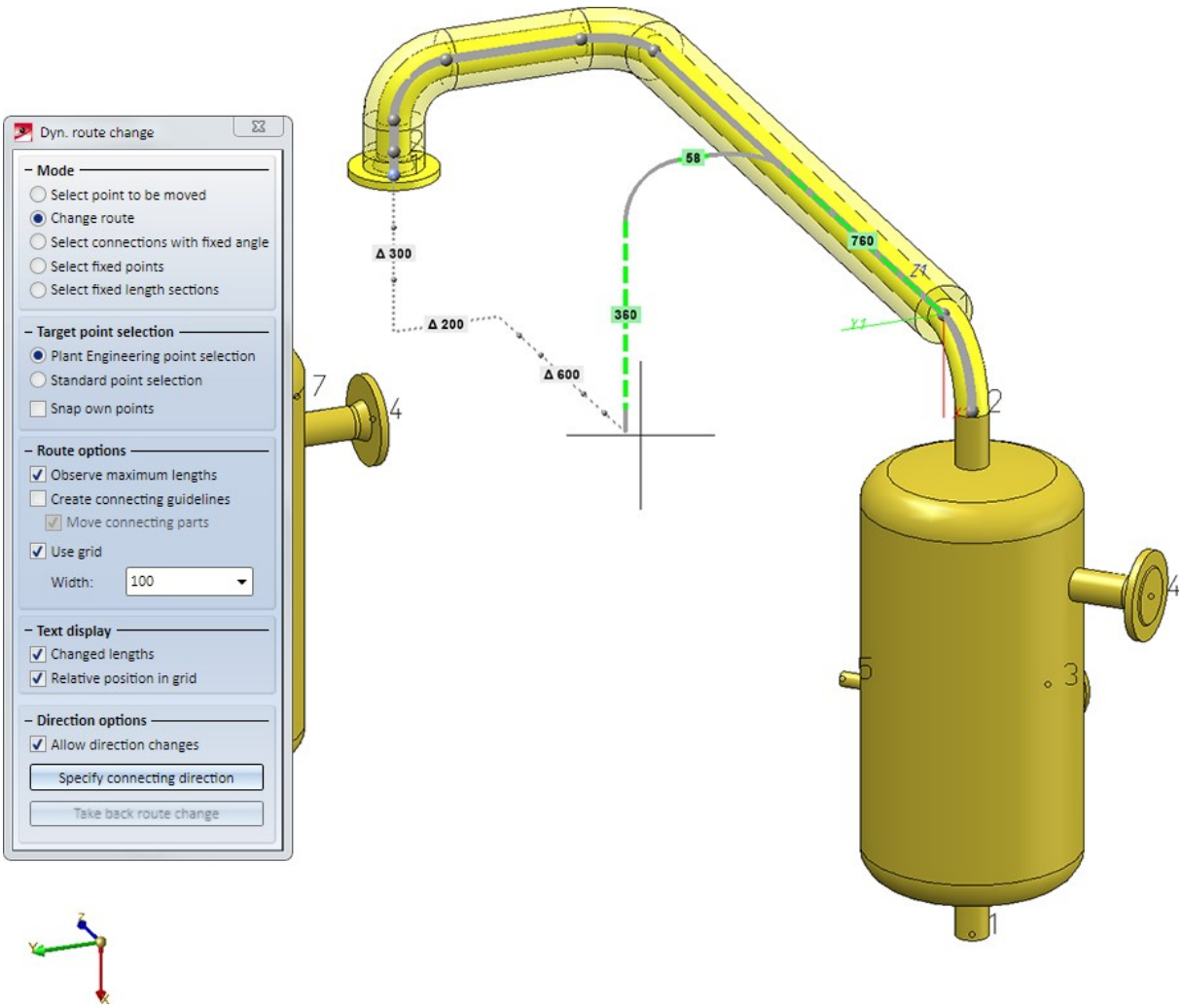


Use grid

The dynamic route change is now supported by a grid. This can be useful if you want to move the connecting point of a pipe by a defined value in the direction of a connection that does not run along one of the axes of the currently active coordinate systems.

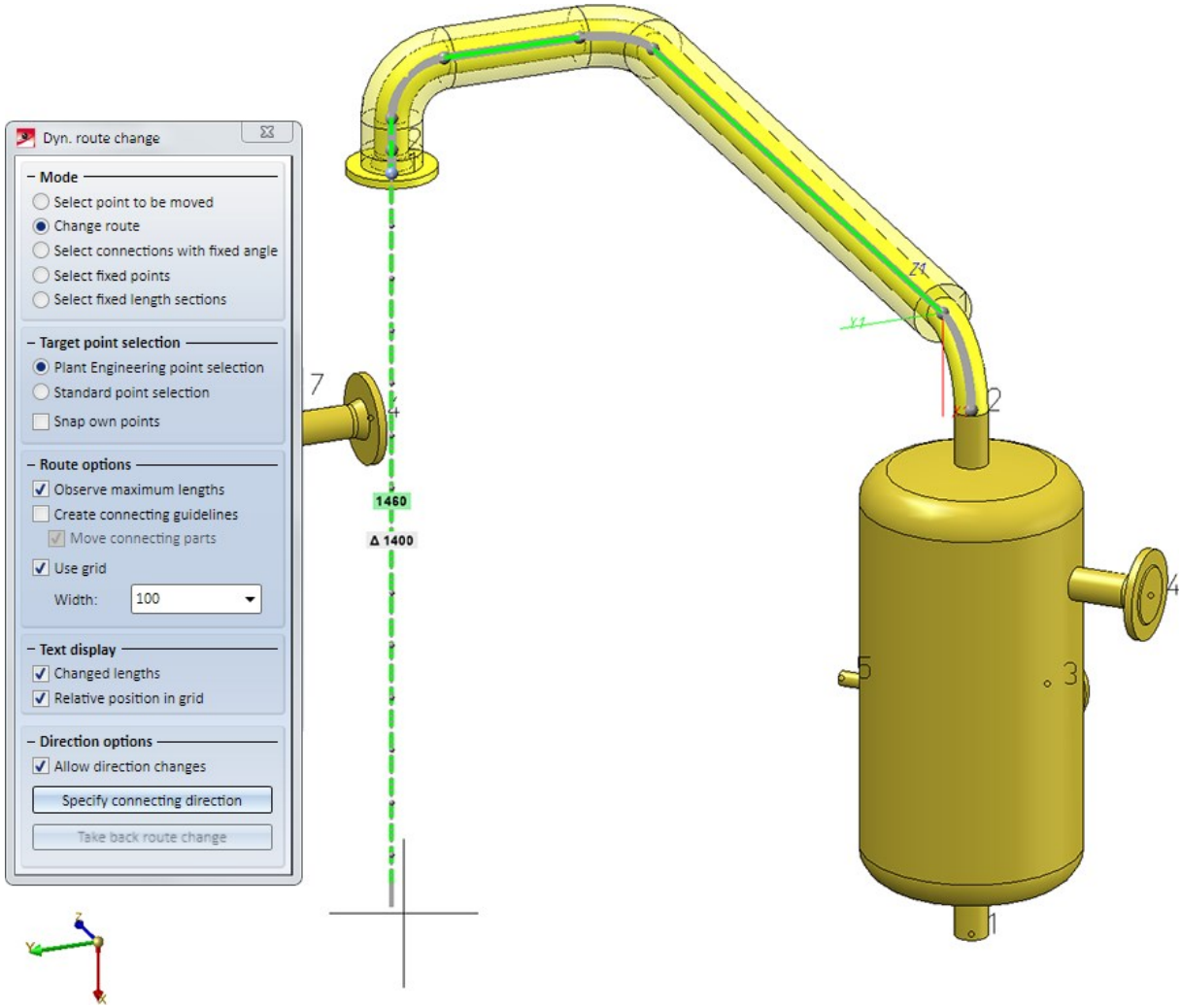
The origin of the grid will always lie in the point to be moved in the process. The orientation will be selected in such a way that one of the axes of the grid runs in connection direction, and a second axis lies in the XY-plane of the active coordinate system.

The grid is represented by dotted auxiliary lines, on which the current grid spacing is visualized by means of small spheres.



If the **Relative position in grid** checkbox beneath **Text display** has been activated in the dialogue window, the distance to the start point will be written, along the grid axes, to the auxiliary lines. To distinguish such distances from the texts with changed lengths, they will be preceded by a capital Δ (Delta).

This display is particularly useful if a length change runs along a grid axis. In the image below, for instance, the short straight pipe at the end of the pipeline is lengthened in downward direction. The new length of the pipe will be 1460, while the end point of the pipeline will be moved downwards by 1400 units:



Immediately below the **Use grid** checkbox you can specify the incremental width (spacing) of the grid. Please note that you can also pick the grid spacing from the drawing when you have chosen **Pick distance** from the context menu of the input box. If the selected grid point cannot be reached with the route change, the auxiliary grid lines and the text boxes will be highlighted in red:

Please note:

- While the grid is active, you can, of course, still snap the target point of the in the drawing. In this case the grid will be automatically hidden.
- The dropping on a grid point takes place by pressing the middle mouse button, i.e. the behaviour corresponds to the "free" dropping without grid. This prevents an accidental, unwanted dropping on a grid point when you actually wanted to select a connecting point, as the dropping at connecting points is still carried out with a left-click.

Check for Collisions of Pipelines with Rest of Model Drawing

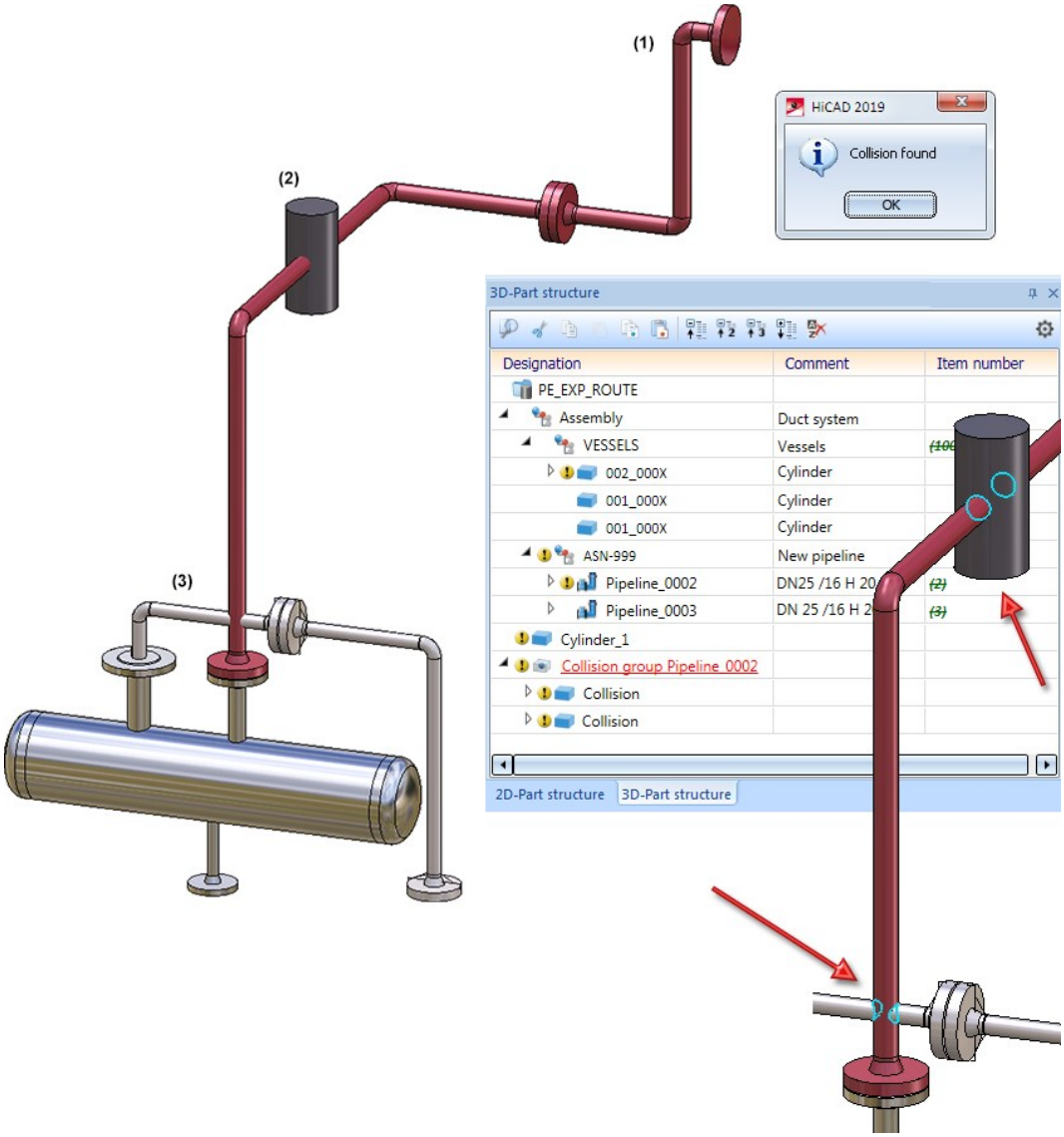
New in the **Assign** pull-down menu is the **Collision check** function. This function checks whether a pipeline collides with other objects of the model drawing. It does not check whether parts of the pipeline collide with each other

If a pipeline is active when you call the function, the collision check will be carried out for this pipeline. Otherwise, HiCAD asks you to choose a pipeline for which the collision check is to be carried out.

If collisions are found, an appropriate message will be displayed and HiCAD will generate so-called "collision parts" which can be found in the ICN, beneath the part **Collision groups** *name* . *name* is the name of the checked pipeline.

An example:

In the image below the Pipeline (1) collides with the Cylinder (2) and the Pipeline (3). If you carry out the collision check, there will be 2 collision parts in the Collision group.



Guidelines

Revised automatic placing of parts on guidelines

The **AutoPlace parts on guidelines** function has been fundamentally revised.

Detail representation for a clearer overview

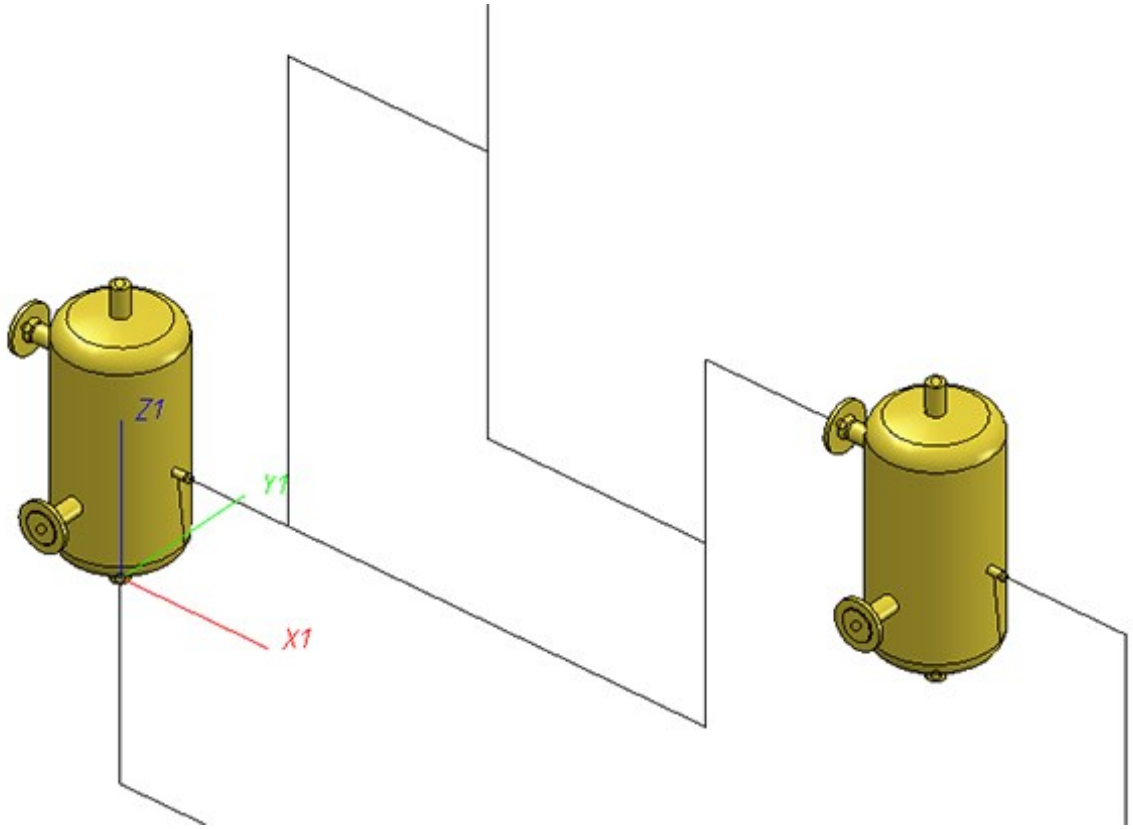
If a part for a particular fitting situation needs to be found during automatic placing of parts, an appropriate search window will be shown. This search window can belong to a different application, e.g. HELiOS. If such a window is shown, HiCAD will not allow any inputs, i.e. you cannot navigate in such a situation.

In previous versions the selection of a suitable parts could sometimes be difficult, if the search request referred to an "invisible" fitting situation. In many of such situations the fitting context of the part was rather unclear.

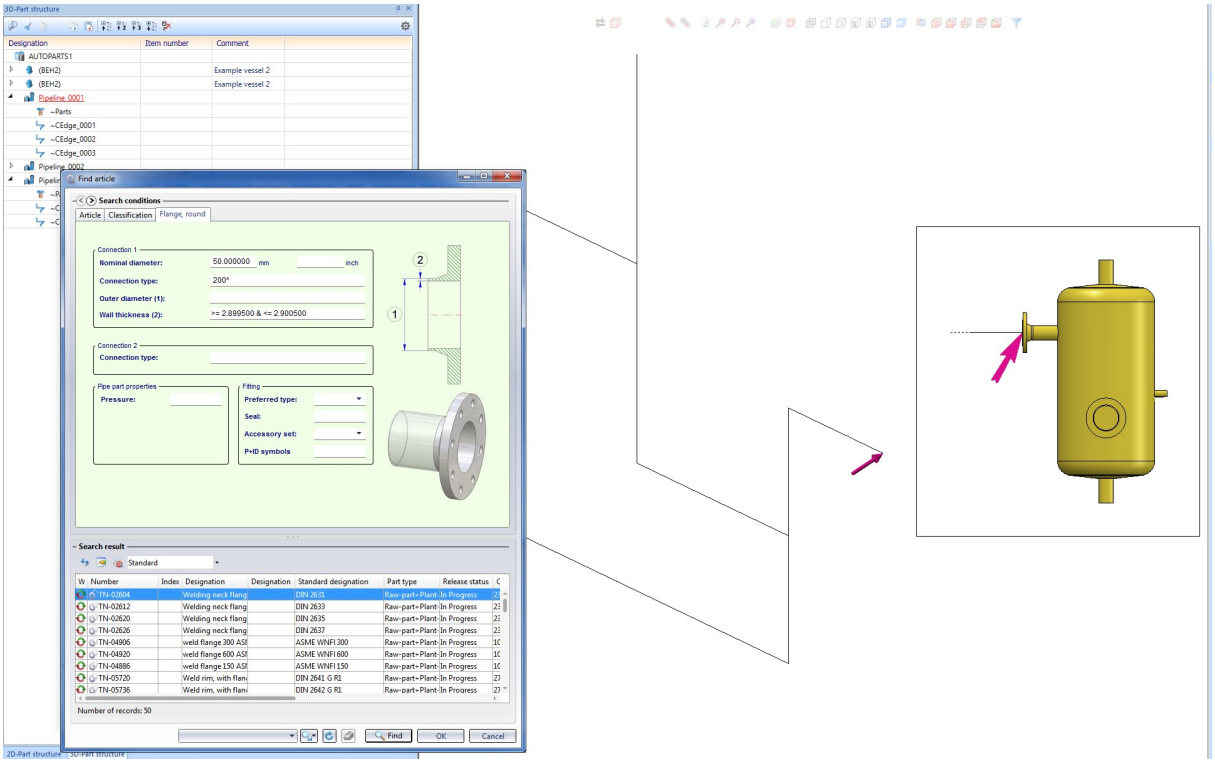
For the sake of a clearer overview, a second window with an enlarged detail view of the current fitting situation will be shown. This means that the drawing area is split into two areas during automatic part placing: On the left an overview of the complete pipeline is shown; on the right, an enlarged representation of the concrete fitting situation is shown. In both representations, an arrow pointing at the current fitting situation is displayed. The direction of the arrow corresponds to the viewing direction in the detail view.

Example:

The following example shows the original drawing, where the automatic placing of parts initially sets a counterflange:



If you now start the automatic placing of parts, the following will be displayed as soon as the search window offers concrete flanges for selection:



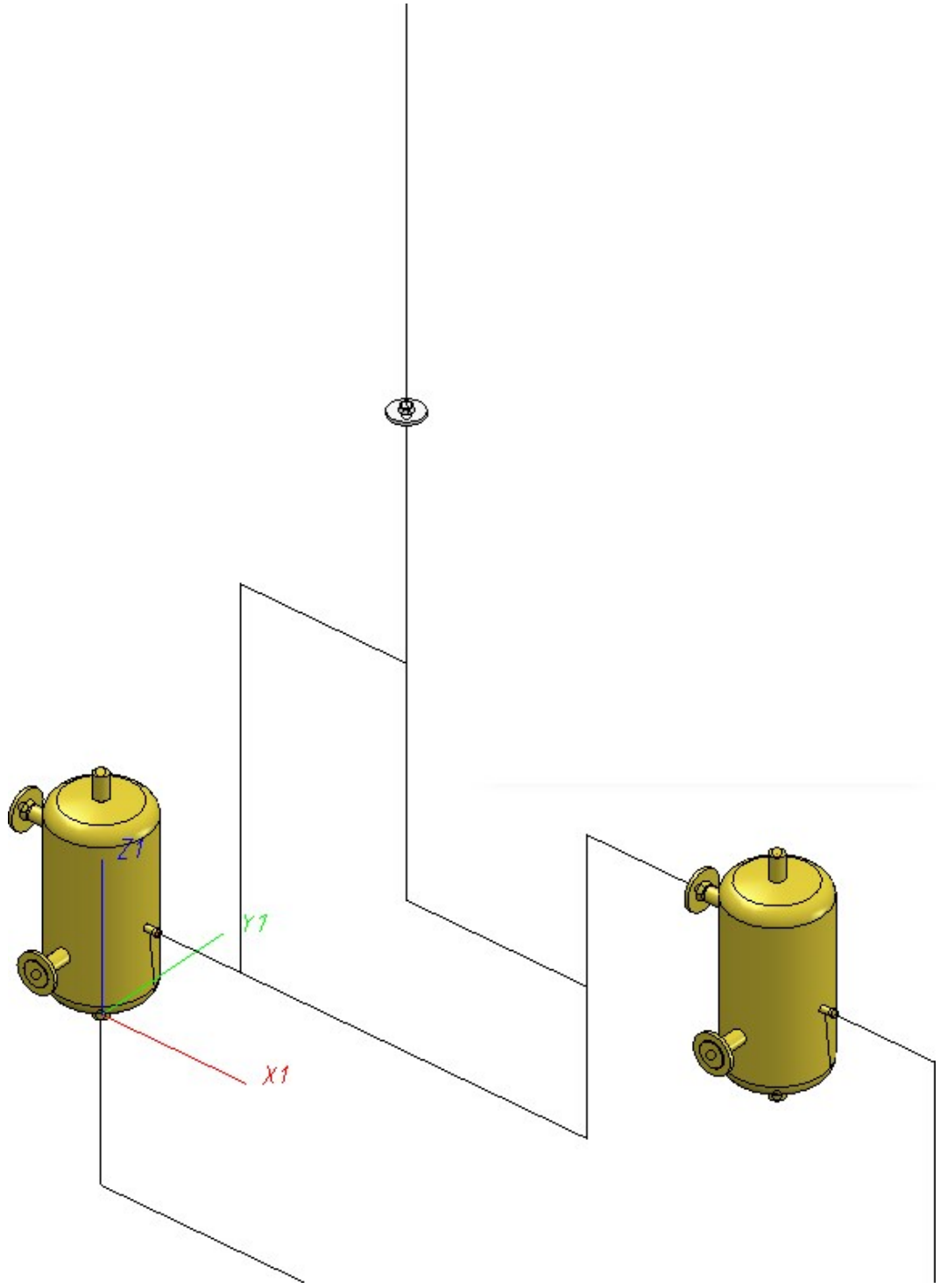
The vessel will only be displayed in the detail window, as it does not belong to the pipeline. This representation serves the purpose of a clearer overview, since vessels are normally the larger parts. In the detail view, longer parts, such as pipes and guidelines, are sometimes only implied: In the present example, a portion of the guideline is implied by means of a dashed line. This representation, too, servers the purpose of a clearer overview.

Placing of parts on pipeline sets

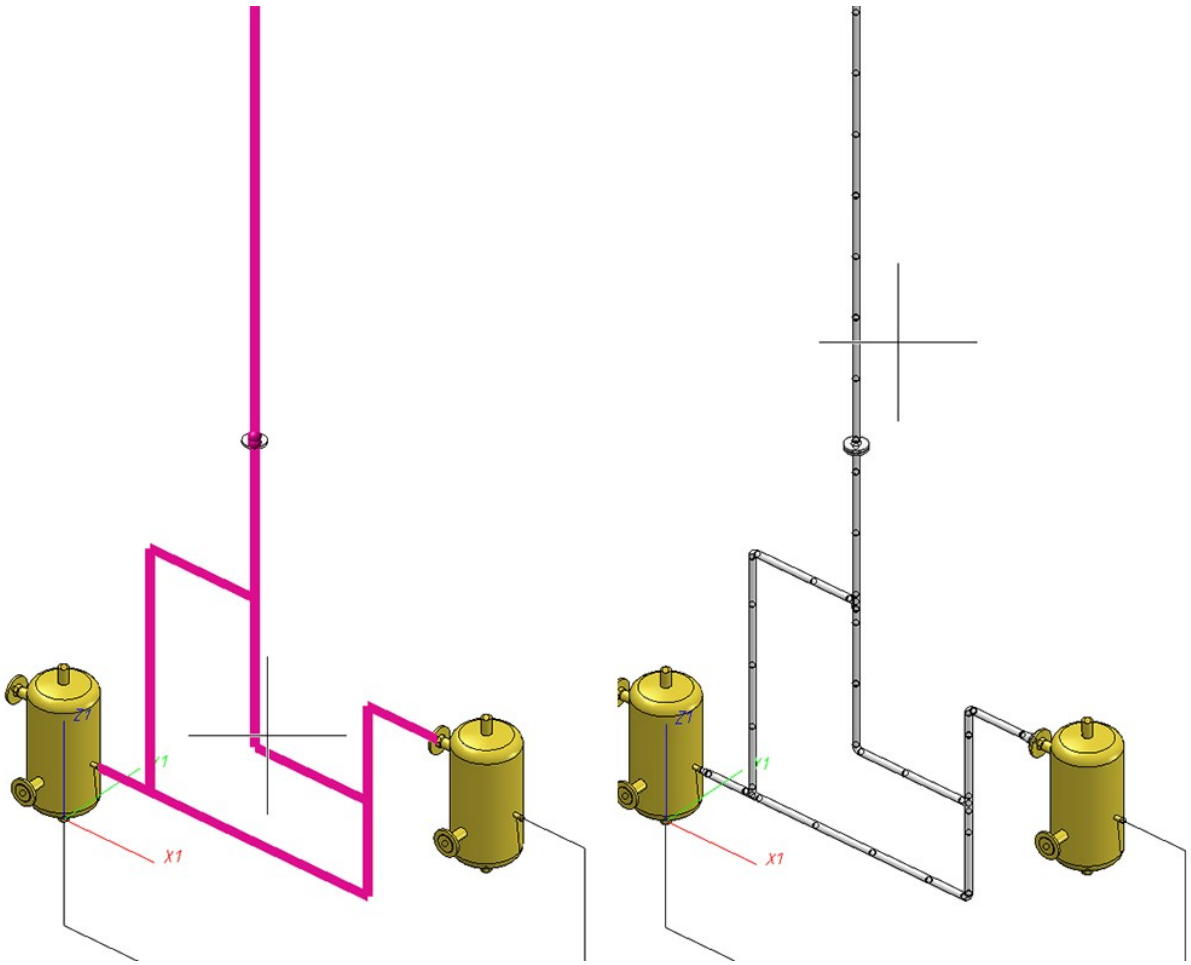
From SP2 onwards it is possible to interpret several connected pipelines as one pipeline set and place parts on them in one step. Use the new **Place parts on pipeline set** option for this.

Example:

In the image below (left), a new pipeline begins on the single flange.



If you have chosen the **Place parts on pipeline set** option, parts will be placed on the pipeline set in such a way, as if the pipeline set was one single pipeline.

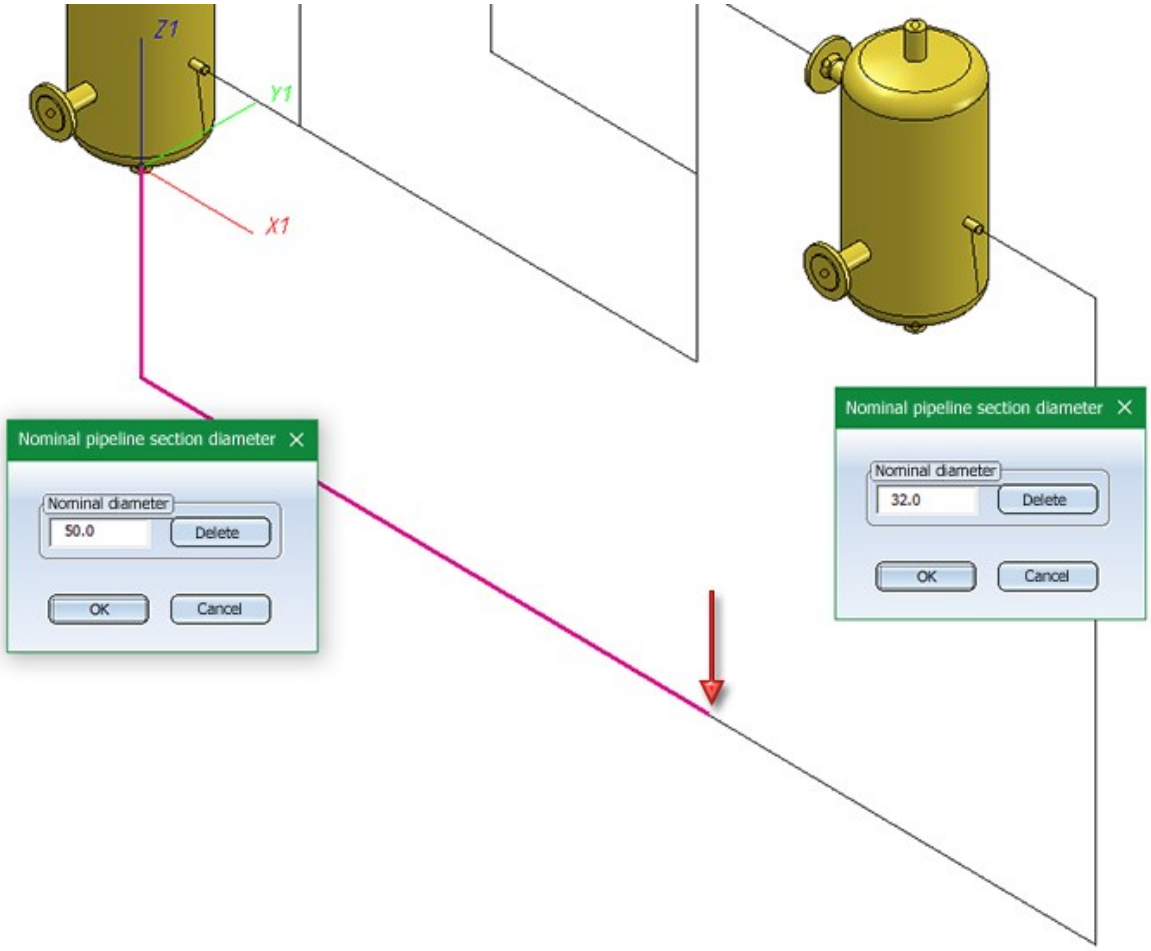


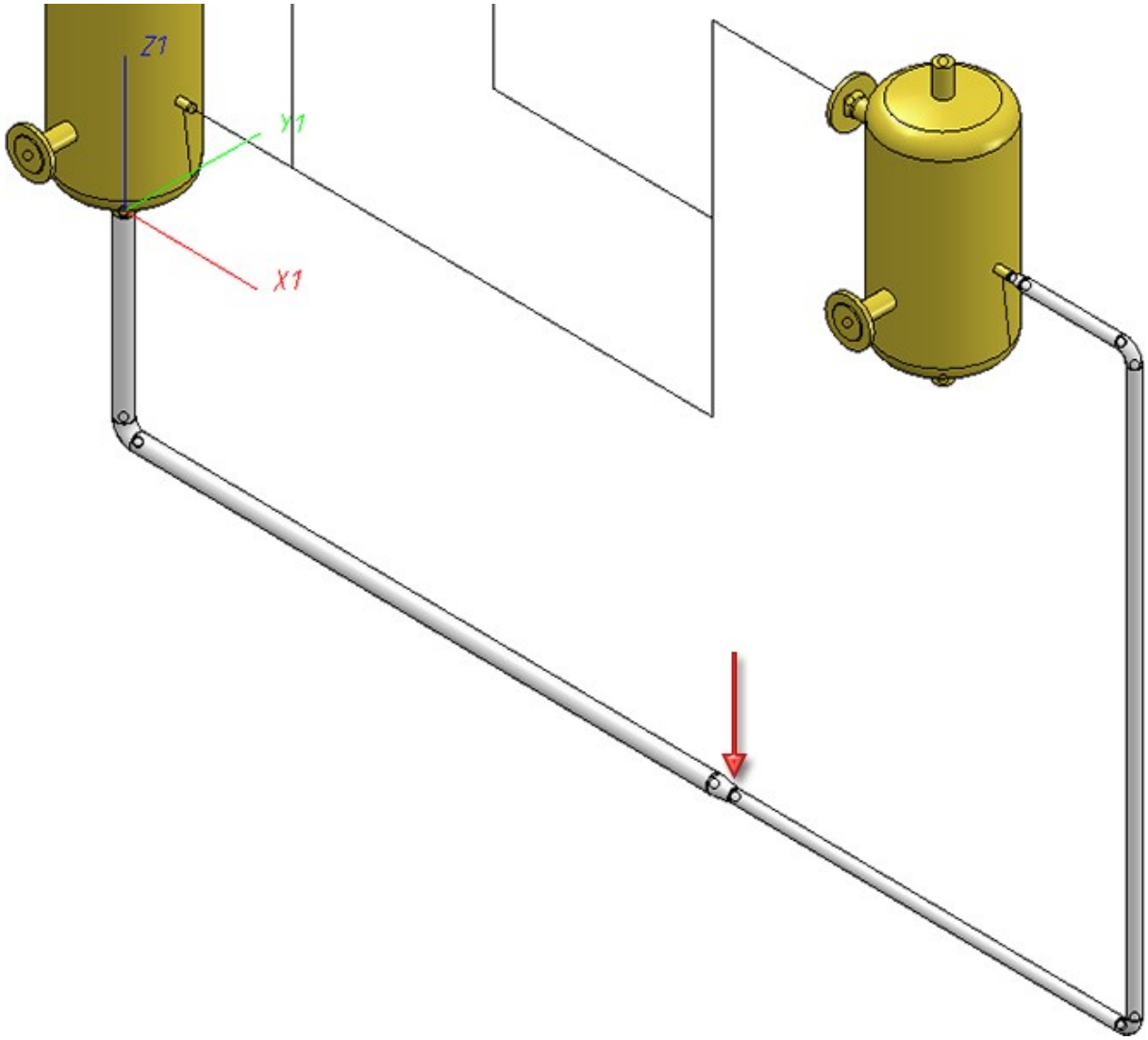
Reducers on guideline transitions

Automatic placing of parts on guidelines also supports reducers on guideline transitions. The function will then place reducers in places where there is a transition between guidelines of different nominal widths.

Such a reducer is placed midway on the transition:

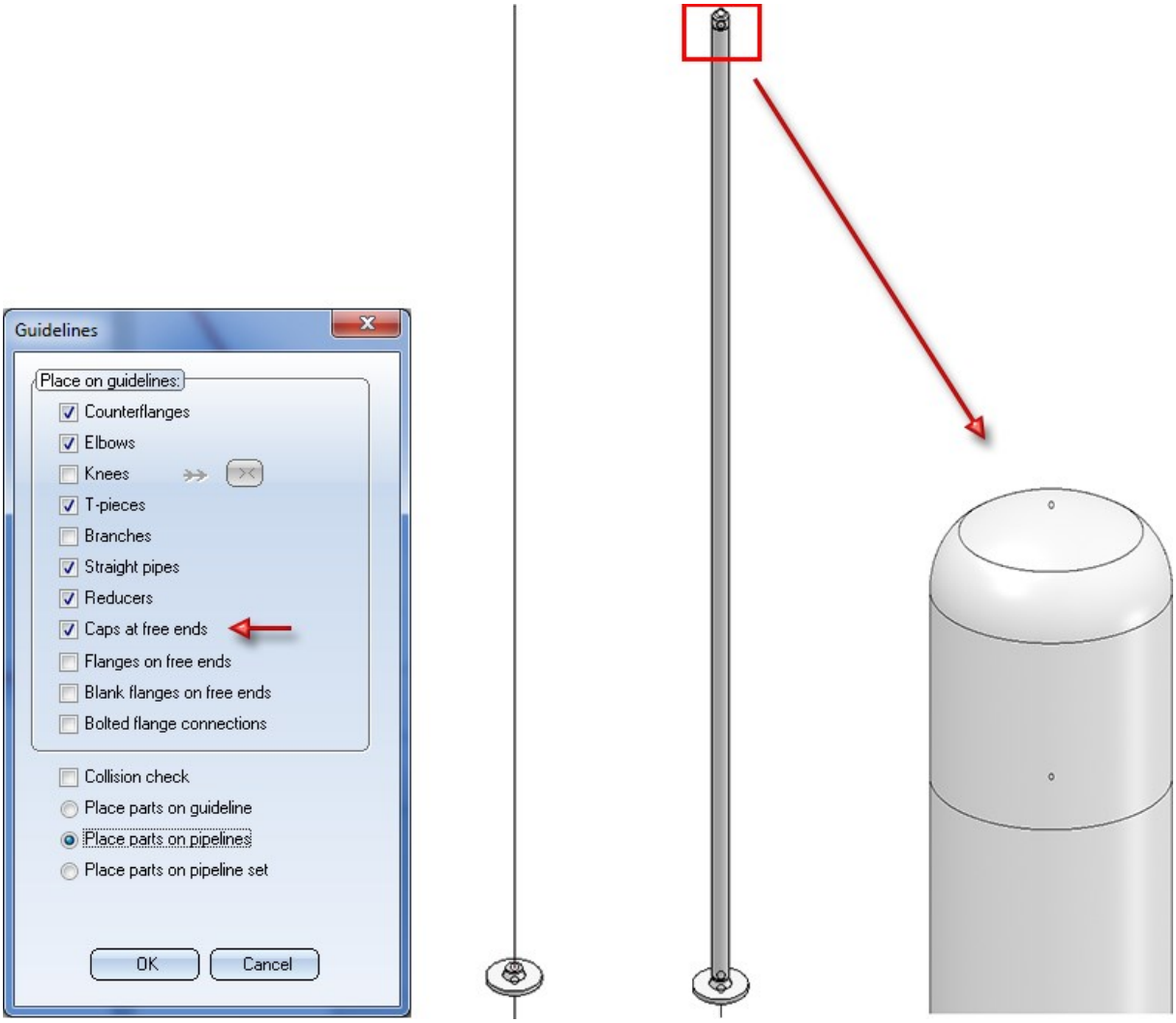
Graphic




Z1
Y1
X1

Caps on free ends

This option automatically places caps on vacant guideline ends (i.e. the start or end of a guideline). A guideline end is vacant if no part is located on the end point, and the end point is not located on the connection of a part that does not belong to the active pipeline.



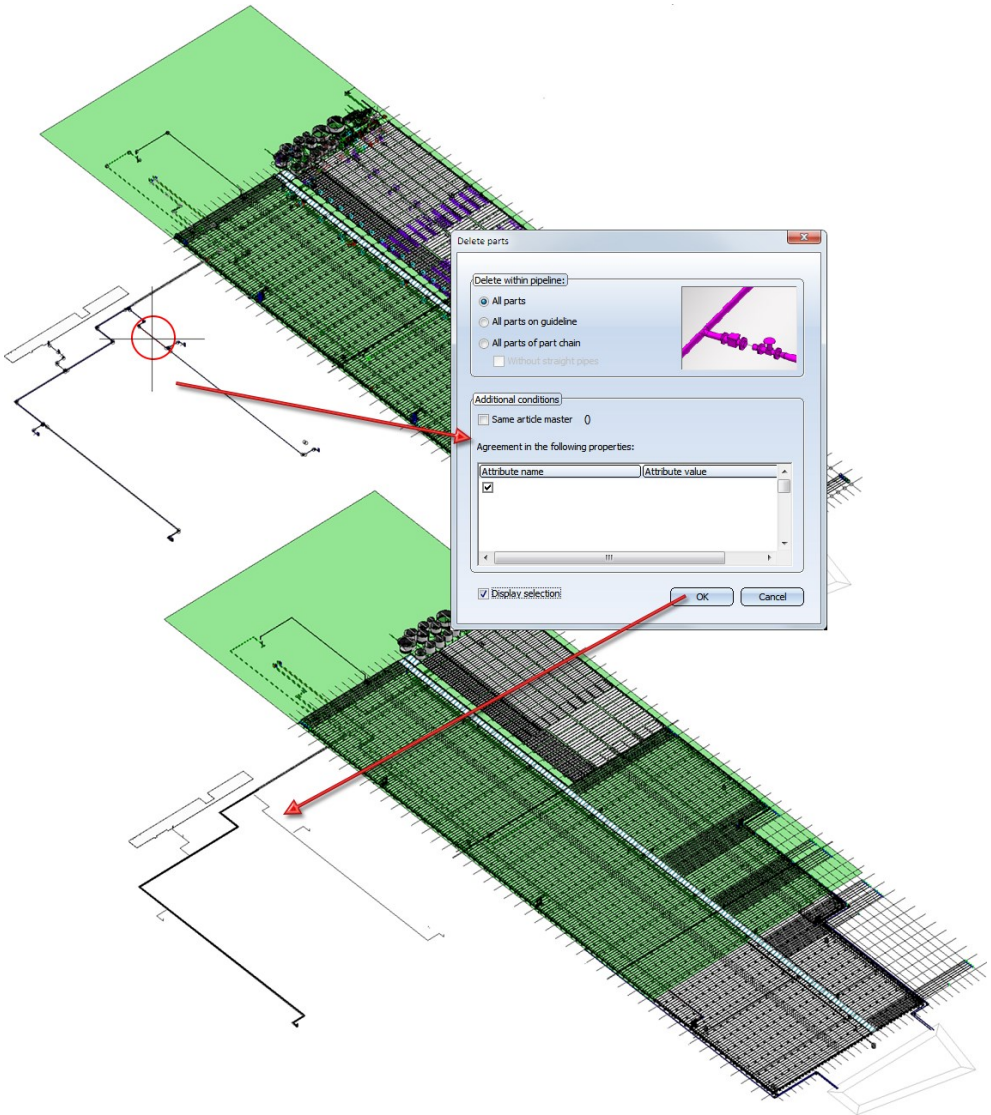
Service Pack 1 2019 (V 2401)

Performance

Delete several parts

The performance for part deletion has been increased significantly. For example, a performance increase by a factor of 10 could be achieved for the model drawing shown below:

In our test the highlighted part was chosen, and the context menu function (right-click part) **Delete several parts**  was selected, with the **All parts** option.

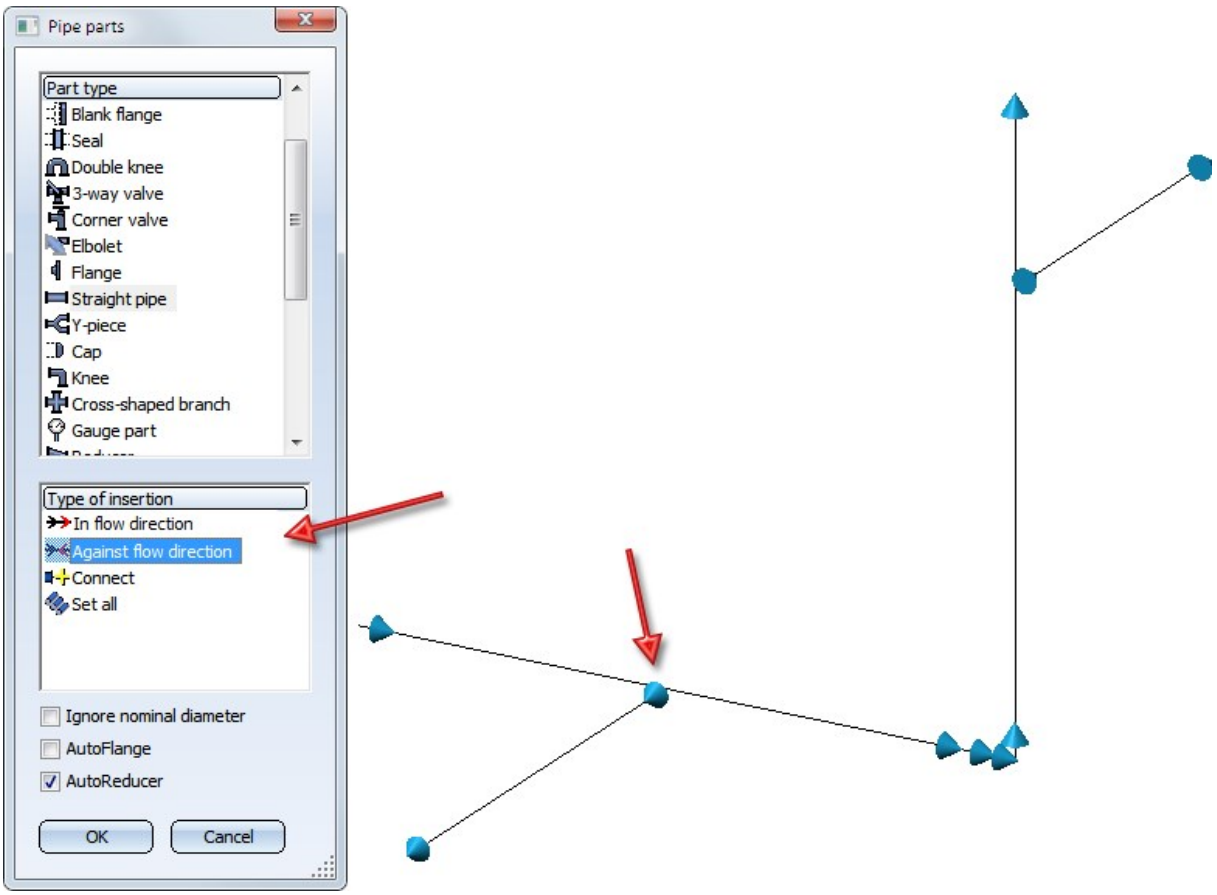


Example drawing (Certhon Build B.V., Poeldijk, The Netherlands)

Working with guidelines

Inserting parts in flow direction / against flow direction

From HiCAD 2019 SP1 onwards, the flow direction (and no longer the edge direction) will be used for the determining of the orientation of parts placed onto guidelines. During part insertion, appropriate flow direction arrows will be displayed.



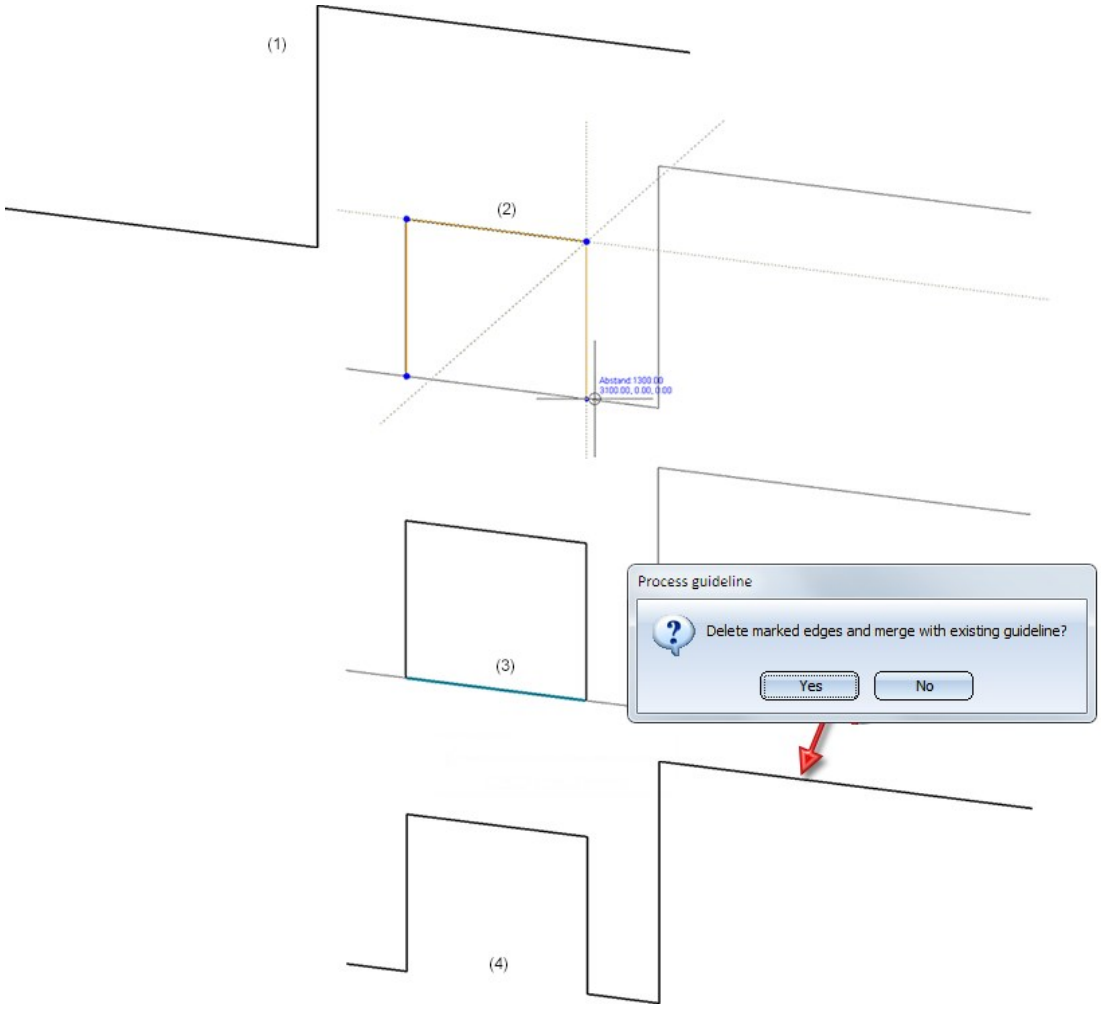
Trimming of guidelines

After insertion of a new guideline and during processing of existing guidelines HiCAD will now check whether the relevant pipeline contains a guideline on which the start point and also the end point of the new/processed guideline is located.

If this is the case, the edges that would be trimmed away will be highlighted. At the same time, an appropriate message will be displayed.

- If you confirm the trimming when inserting a new guideline, the already existing guideline will be trimmed and expanded by the new guideline. This means that no new guideline will be created, but only the existing guideline will be modified.
- If you confirm the trimming during processing of an already existing guideline, the highlighted edges will be removed and the two guideline will be merged into one.

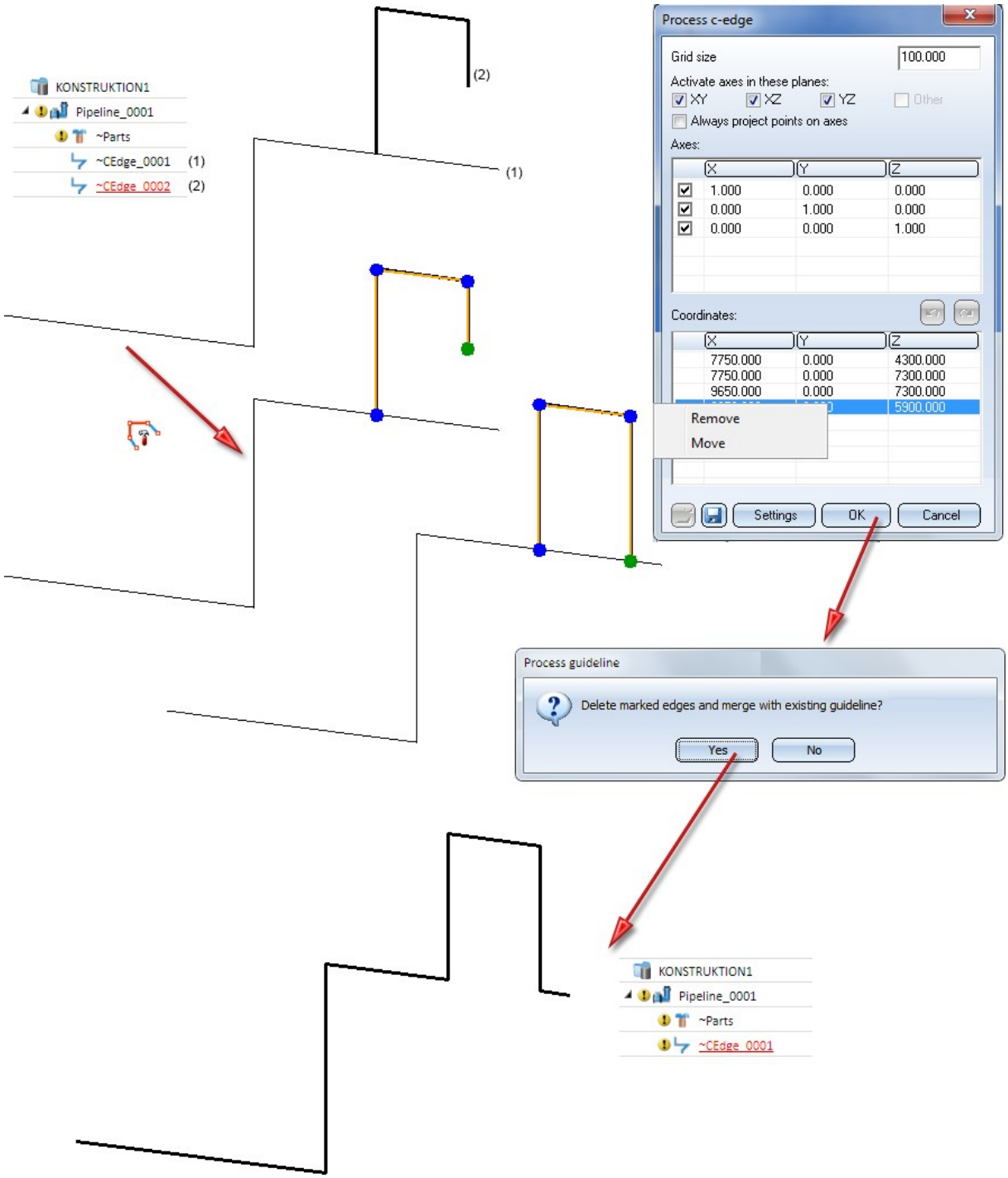
Example 1: Insertion of a new guideline



(1) Existing guideline, (2) New guideline , (3) Confirm with **Yes**, (4) Modified guideline

Example 2: Processing of an already existing guideline

The image below shows 2 guidelines of a pipeline. The start point of Guideline 2 lies on Guideline 1, but the end point does not. When inserting this guideline, no Trim option will therefore be offered. But if you not modify Guideline 2, moving, for example, its end point onto Guideline 1, you can trim. If you confirm the message with **Yes**, the highlighted edges will be removed and the two guidelines will be merged into one:

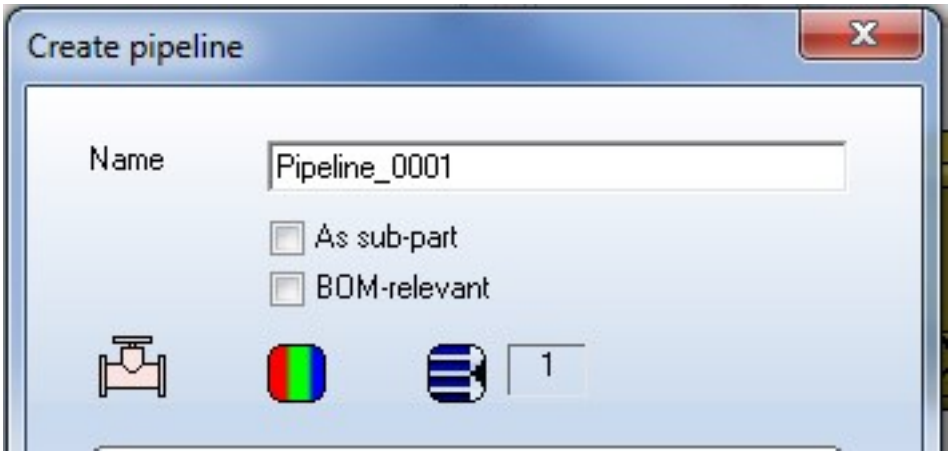


Trimming is not possible if this would remove guidelines that already contain parts.

Pipeline Tools

Create pipeline - BOM-relevance

New pipelines can be marked as BOM-relevant directly during their creation by activating the new **BOM-relevant** checkbox.

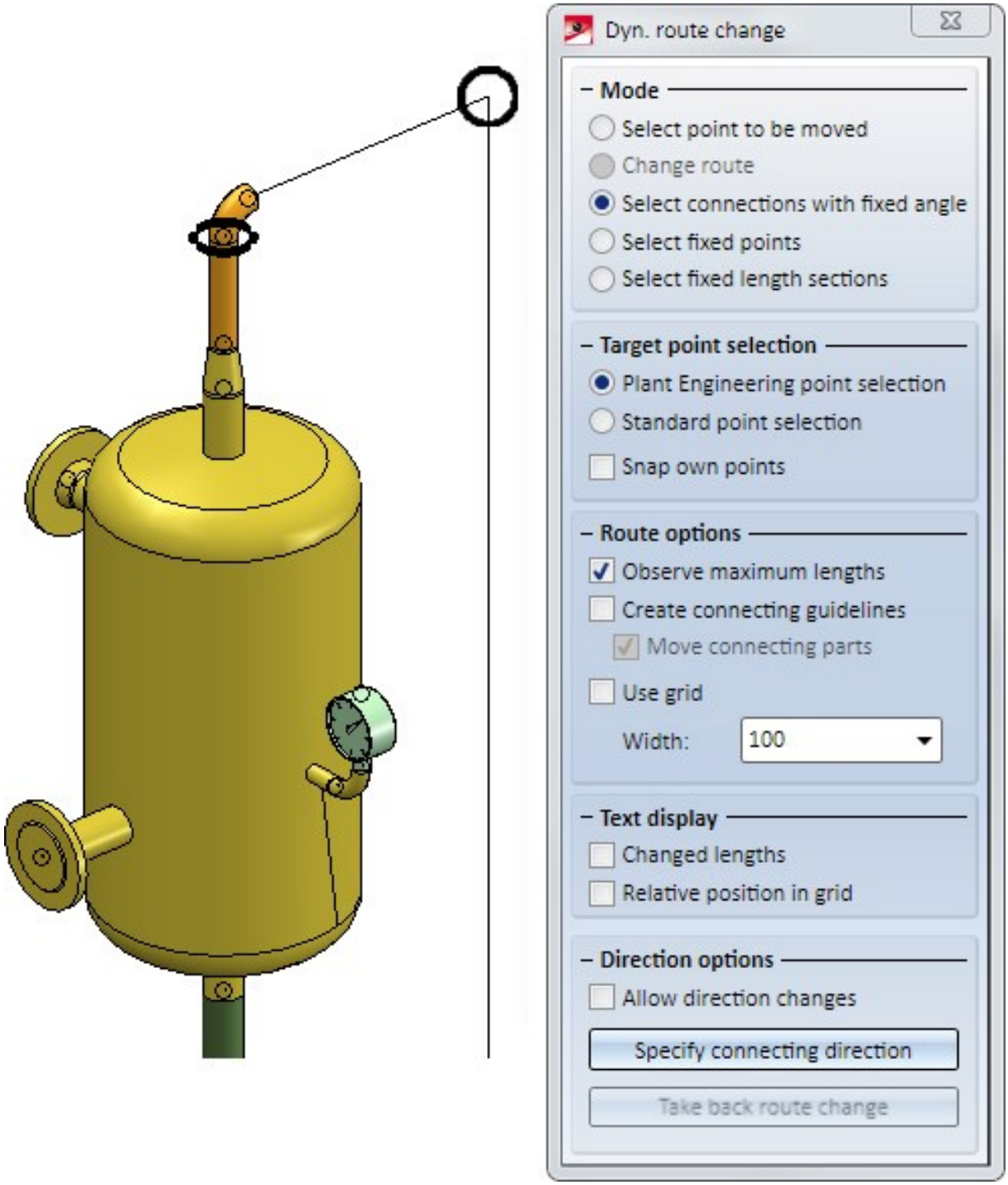


Dynamic route change

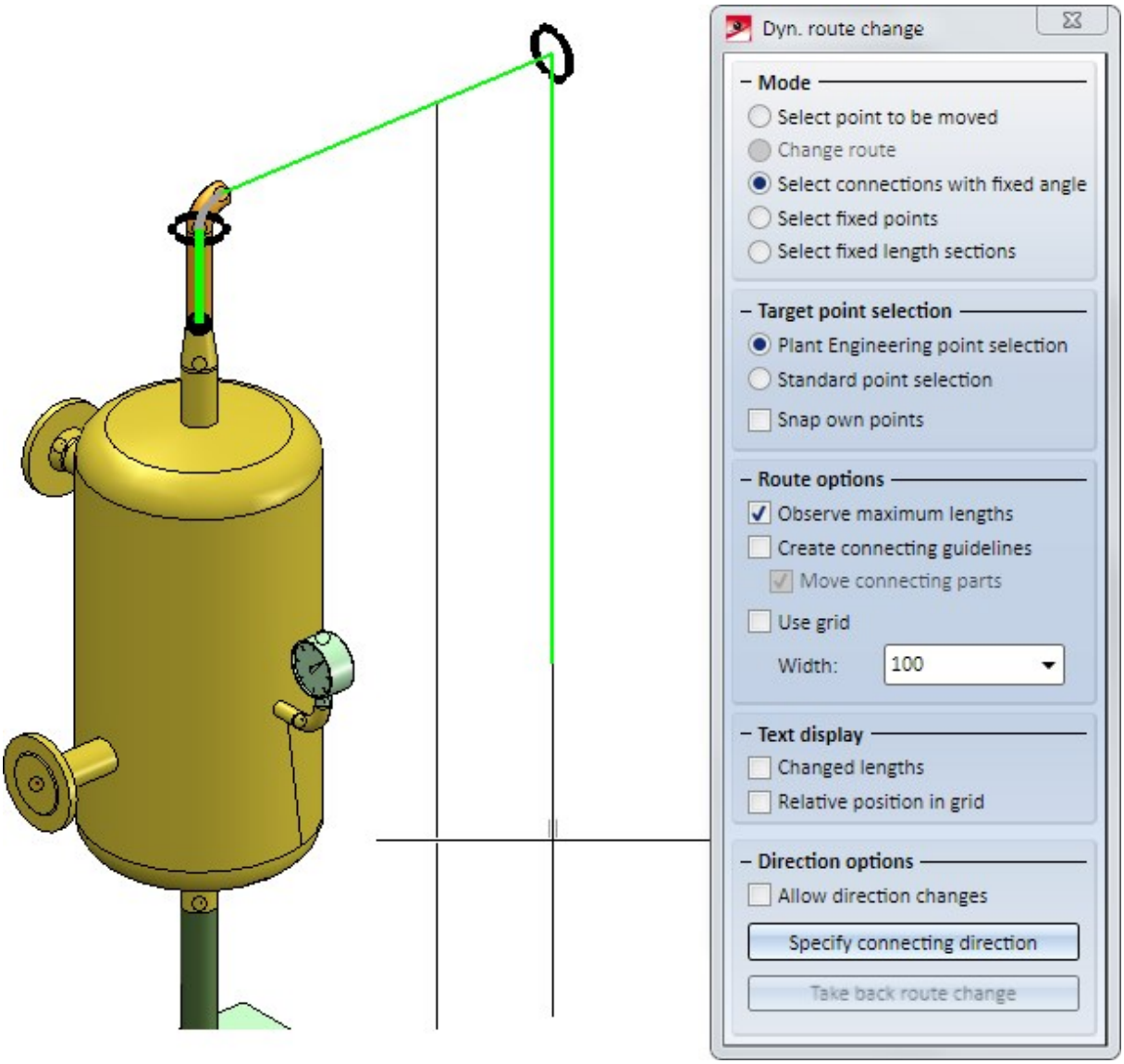
The Dynamic route change function offers a multitude of options which, however, were only active after selecting a point to be moved. In model drawings with very many connections, this could sometimes lead to confusion if the cursor had to move over a multitude of connections on its way to the dialogue window. While the **Allow direction changes** option was activated, unwanted route change calculations and their subsequent cancellation were often the result. The previous behaviour also made a local displacement of valve more difficult, since the fixed point required for this could only be specified after selecting the point to be moved.

- To improve operability in such cases, HiCAD now offers all dynamic route change options right from the start. In particular, it is now possible to specify fixed points, connections with fixed angles and fixes lengths sections before selecting the point to be moved.
- Fixed points, connections with fixed angles and fixes lengths sections can now be chosen in the entire drawing at any time. If you want to change the routes of several pipelines, you can now define degrees of freedom for all pipeline first before adjusting their routes.
- In previous versions the selected options were always assigned to the pipeline that was changed. For the next route change of the same pipeline, these options were then available again. This behaviour does now no longer make sense, since the options are no longer bound to the pipeline. Instead, the dialogue window will memorize the last settings, irrespective of the pipeline.

Also changed has the display of connections with fixed angles. These are normally displayed by a circle with the special colour **Grid** (Default: Black). The circle surrounds the axis, about which the rotation would otherwise have been performed. However, there are situations where this rotation axis will only be determined after selecting the point to be moved. In such cases, connections with fixed angles will be displayed by means of a circle that lies always in the projection plane. This situation occurs, for instance, when there are bent guidelines:



After determining the rotation axis by selecting the point to be moved, the display will be adjusted as follows:



Pipeline isometry

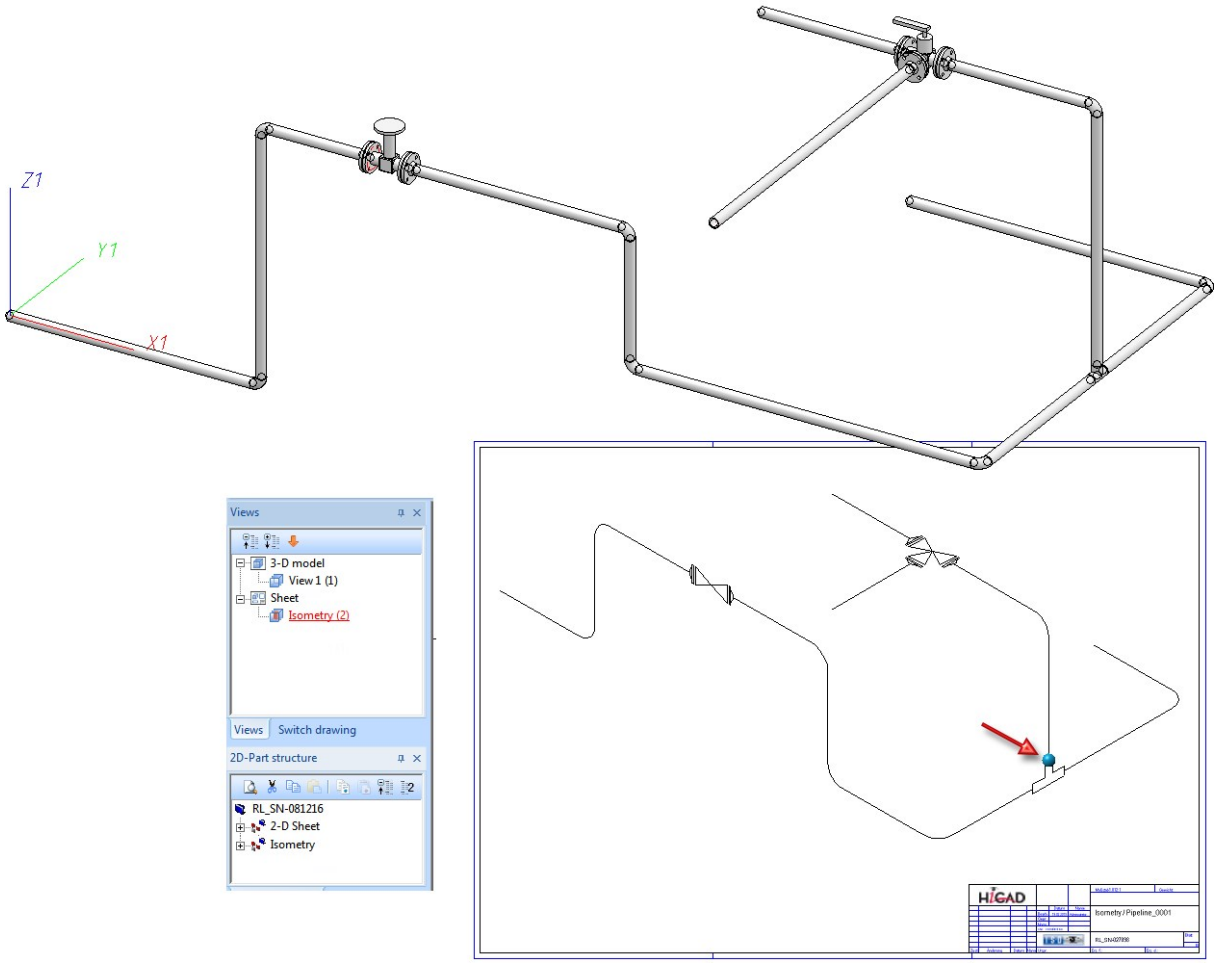
Spreading isometries over several drawing sheets

In previous versions, when you spreaded an isometry over several sheets, one new drawing was created for each section of the pipeline (and, if HELiOS is used, a new document master was created). If an isometry was subdivided several times, things could easily get confusing.

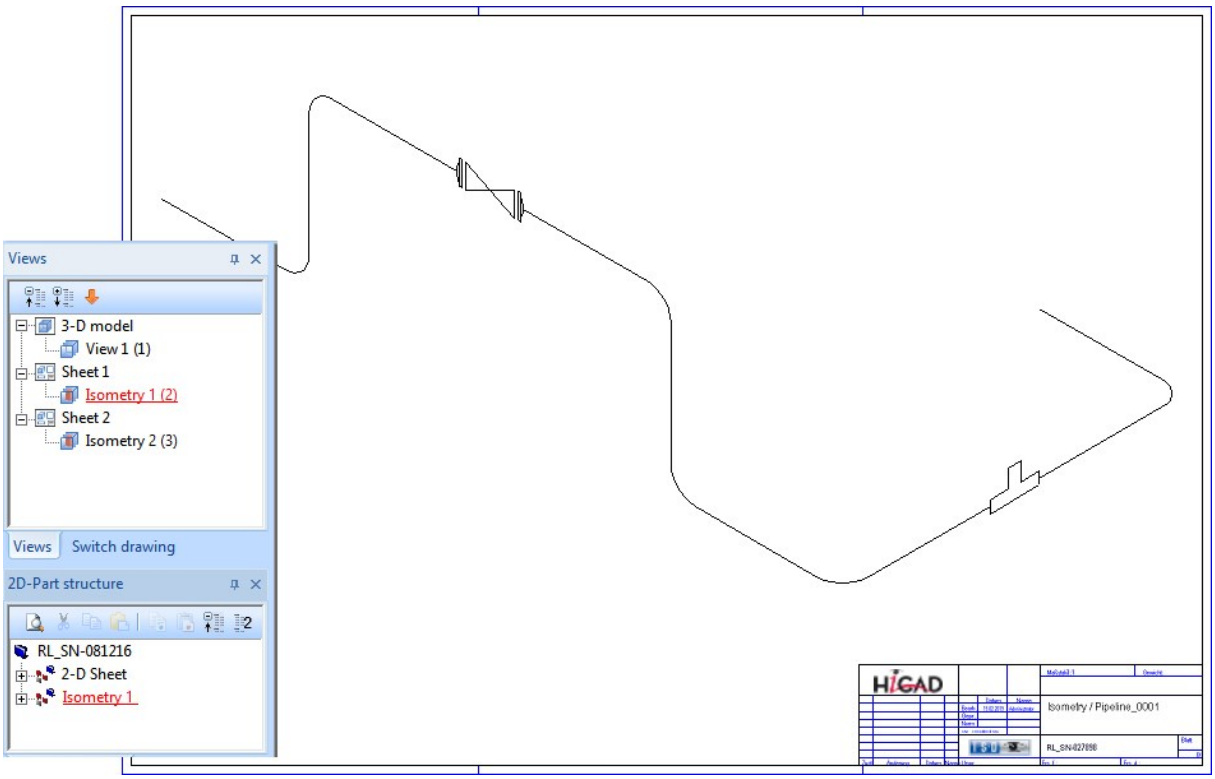
From HiCAD 2019 SP1 onwards, only one drawing will be created for each pipeline. If a pipeline isometry is subdivided, the old isometries will be deleted, and the partial isometries will be created in the same drawing. In the process, the symbols of the isometries will be arranged into assemblies and one sheet will be created for each part isometry.

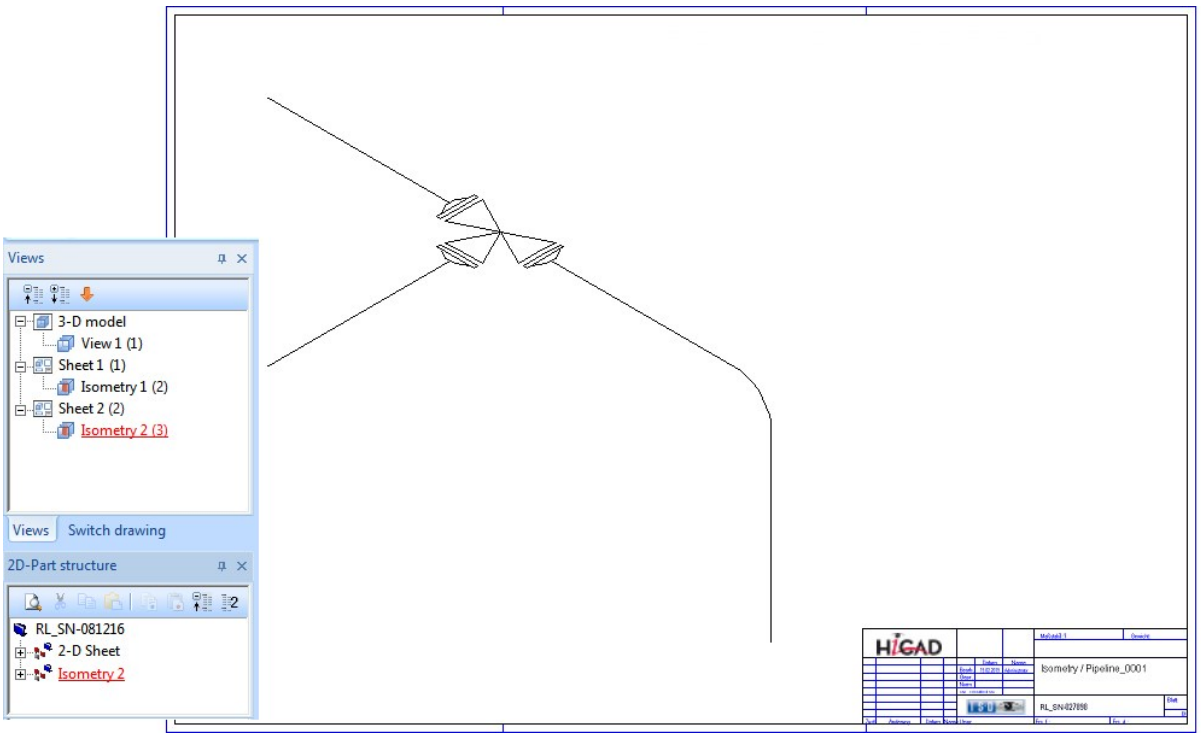
Example:

Take a look at the layout plan and its corresponding isometry shown below:



If you divide the isometry at the blue point (see red arrow) and then generate it again, the sheet with the current isometry will be deleted, and two new sheets for the partial isometries will be generated.

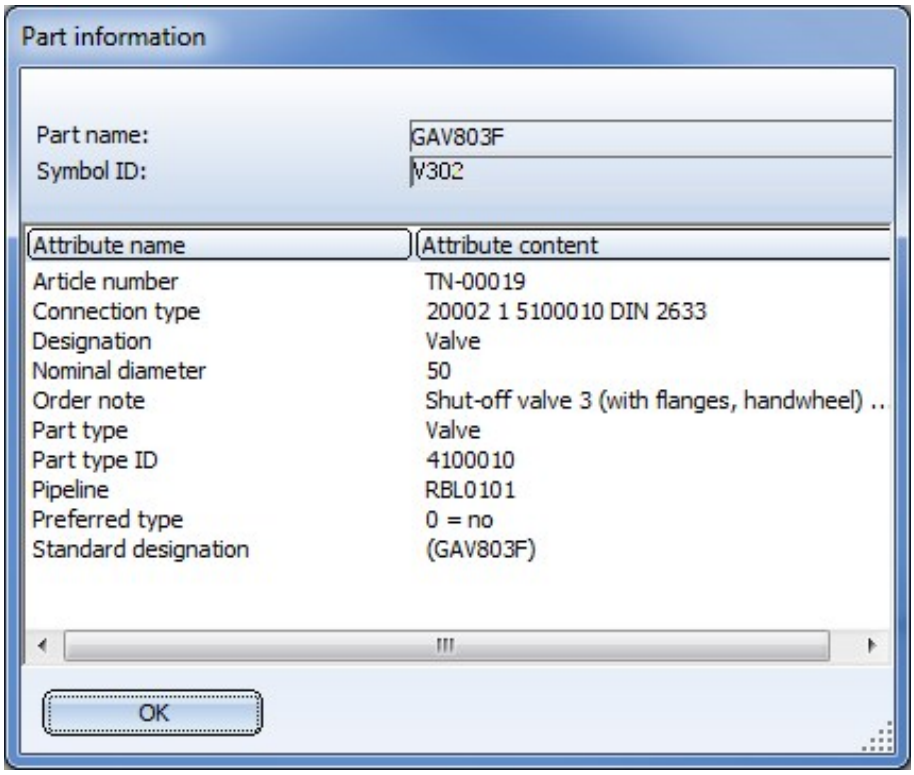




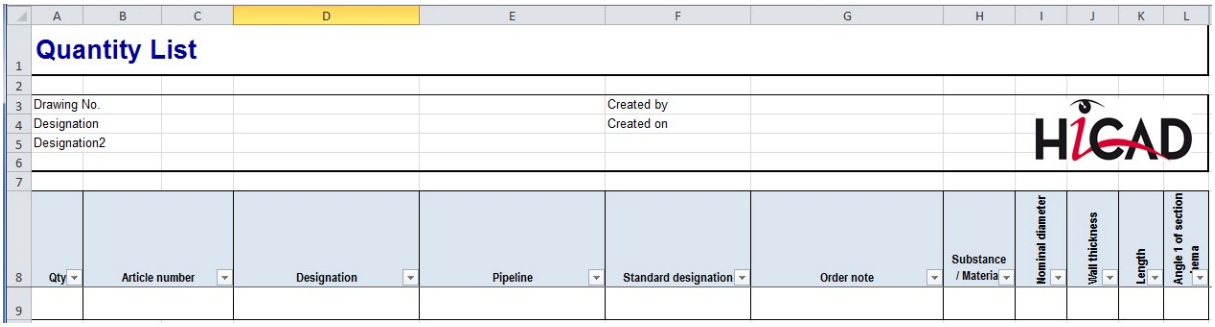
Evaluation / Information

Associated pipeline indicated

The pipeline to which a part belongs is now displayed in the Part information as well as in BOMs.



In Plant Engineering BOMs you can now find the new **Pipeline** (Attribute: **%PipeLineName**) column that indicates the pipeline to which the respective part belongs. The default template for plant Engineering BOMs, **Anlagenbau_ohne_DB_SZN.rms** (HiCAD sys directory) and the corresponding EXCEL template in the BOMTemplates folder have been adjusted accordingly.

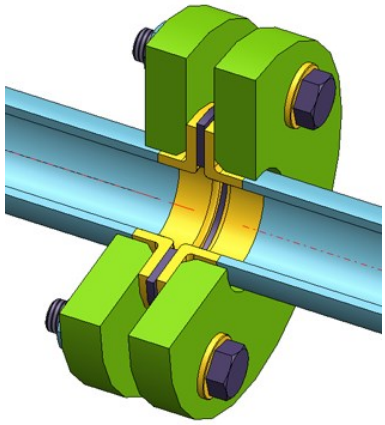


Parts / Components

Model welding necks as flanges

Since HiCAD 2200 it has been possible to insert welded necks with loose flanges. The logic in older versions interprets the welding neck as a straight pipe to which regular loose flanges are assigned. For a placement of these loose flanges the variable F1 has been introduced, which determines the distance of the loose flange from con-

necting point 1 of the welding neck. However, this logic is not optimal if you want to create an isometry of the pipeline, as the flange symbol of the loose flange will then be slightly displaced.



As of HiCAD 2019 SP1 you have an alternative to model such a combination of welding neck and loose flange. When using this procedure, the flanging must be of the type "Flange", while the loose flange is an asymmetrical fastener. The flange connection of the welding neck must have the connection type 20600. The 6 coming in the third place encodes the asymmetrical fastener with flange connection, i.e. normally a loose flange classified as fastener. For this procedure the variable **F1** will also determine the distance of the loose flange to connecting point 1 of the welding neck. For welding necks, F1 normally equals the wall thickness.

In contrast to welding necks that are modelled as straight pipes, the flange symbol is assigned to the welding neck here. This ensures that the position of the flange symbol in a generated isometry will not be affected by a possible moving of the loose flange.

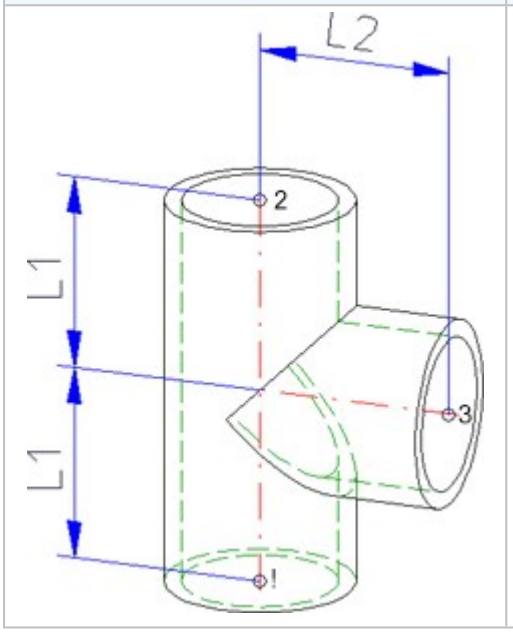
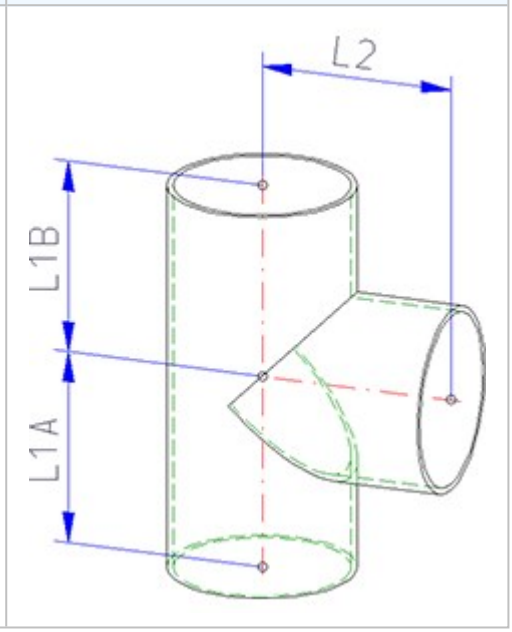
Please note:

If you want to fix the loose flange by a welding point, you should not model it as a fastener, as fasteners do not support welding points on connecting points 2. In this case you must use genuine loose flanges, i.e. such flanges that are actually classified as flanges.

Asymmetrical T-pieces

In previous versions, T-pieces were always symmetrical, i.e. the inserted pipe had always to be located in the middle between the named points 1 and 2. From now on, you can also insert asymmetrical T-pieces.

If the variable L1 exists in the variant, the insertion length will be the double amount of L1. Otherwise, the insertion length will be the sum of L1A and L1B.

Old	New
	

New standard parts

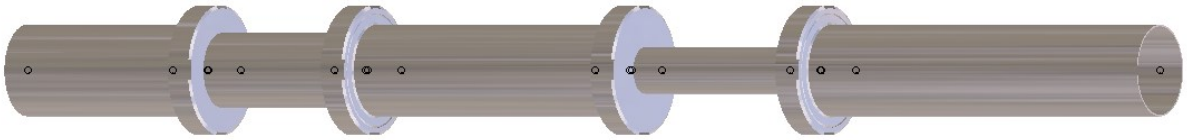
Parts for pipeline planning in the food industry

The standard part inventory has been expanded by the following parts acc. to **DIN 11851**:

Part type	Part	Variant
Blank flange	Threaded dummy socket	N11851_BGS.VAA
Blank flange	Tapered dummy socket	N11851_BKS.VAA
Sealing gasket	O-ring	N11851_OR.VAA
Double knee	Reversal pipe bend 180° KK	N11851_DOUBLEKNEE_KK.VAA
Double knee	Pipe bend 180° KK	N11851_KNEE180_KK.VAA
Flange	Threaded socket	N11851_GS.VAA
Flange	Tapered socket	N11851_KS.VAA
Straight pipe	Intermediate piece GK	N11851_STR_GK.VAA
Straight pipe	Intermediate piece KK	N11851_STR_KK.VAA
Straight pipe	Intermediate piece GG	N11851_STR_GG.VAA
Knie	Pipe bend GG	N11851_KNEE_GG.VAA
Knee	Pipe bend GK	N11851_KNEE_GK.VAA
Knee	Pipe bend GS	N11851_KNEE_GS.VAA
Knee	Pipe bend KK	N11851_KNEE_KK.VAA
Knee	Pipe bend KS	N11851_KNEE_KS.VAA
Cross	Cross piece GGGG	N11851_CROSS_GGGG.VAA
Cross	Cross piece KGGG	N11851_CROSS_KGGG.VAA
Cross	Cross piece KGKG	N11851_CROSS_KGKG.VAA
Cross	Cross piece KKKG	N11851_CROSS_KKKG.VAA
Cross	Cross piece KKKK	N11851_CROSS_KKKK.VAA
Reducer, symm.	Reducer GG	N11851_RED_GG.VAA
Reducer, symm.	Reducer GK	N11851_RED_GK.VAA
Reducer, symm.	Reducer KG	N11851_RED_KG.VAA
Reducer, symm.	Reducer KK	N11851_RED_KK.VAA
Reducer, symm.	Threaded reducer socket	N11851_RGS.VAA
Reducer, symm.	Tapered reducer socket	N11851_RKS.VAA
Other pipe part	Threaded screw-on socket	N11851_GAS.VAA
Other pipe part	Threaded screw-in socket	N11851_GES.VAA
Other pipe part	Tapered screw-on socket	N11851_KAS.VAA
Other pipe part	Tapered screw-in socket	N11851_KES.VAA
Other pipe part	Compensator GG	N11851_KOMP_GG.VAA
Other pipe part	Compensator GK	N11851_KOMP_GK.VAA
T-piece	T-piece GGG	N11851_TBEND_GGG.VAA
T-piece	T-piece GGG	N11851_TEE_GGG.VAA

Part type	Part	Variant
T-piece	T-piece GGK	N11851_TEE_GGK.VAA
T-piece	T-piece GKG	N11851_TEE_GKG.VAA
T-piece	T-piece GKK	N11851_TEE_GKK.VAA
T-piece	T-piece KKG	N11851_TEE_KKG.VAA
T-piece	T-piece KKK	N11851_TEE_KKK.VAA
T-piece	T-piece SGK	N11851_TEE_SGK.VAA
Fastener, asymm.	Slotted nut	N11851_NUT.VAA

For an easy transfer to the database these parts are listed in the **foodline11851.lst** file in the PlantParts directory.



Also new are the following parts from the catalogue of the company **Kieselmann**:

Part type	Part	Variant	Standard designation
Blank flange	Blind plug - Sheet	KM11851_BKB.VAA	Kieselmann 11851
Blank flange	Tapered dummy socket	KM11851_BKS.VAA	Kieselmann 11851
Seal	Sealing ring	KM10357_OR.VAA	Kieselmann 10357
Flange	S-flange PN10	KM10357_SF_ PN10A.VAA	Kieselmann 10357
Flange	Small flange	KM10357_SF.VAA	Kieselmann 10357
Flange	Small flange, slot	KM10357_SF_SLOT.VAA	Kieselmann 10357
Flange	Welding neck, DIN	KM10357_WN_DIN.VAA	Kieselmann 10357
Flange	Threaded socket	KM11851_GS_L.VAA	Kieselmann 11851
Flange	Tapered socket	KM11851_KS_L.VAA	Kieselmann 11851
Flange	Small blank flange	KM11853-2_BBF-2.VAA	Kieselmann 11853
Flange	Small blank flange 1	KM11853-2_BBF.VAA	Kieselmann 11853
Flange	Clamped welded fusion spigot	KM32676_CSS.VAA	Kieselmann 32676
Reducer, symm.	Reducer, concentric	KM11852_RK.VAA	Kieselmann 11852
Reducer, asymm.	Reducer, excentric	KM11852_RE.VAA	Kieselmann 11852
Fastener, asymm.	Slotted nut	KM11851_NUT.VAA	Kieselmann 11851

These parts can be found in the list file **foodline_kieselmann.lst**.

Welding necks and loose flanges

With the variants **KM10357_WN_DIN.VAA** and **ROFI10357_WN_ISO.VAA**, two part families have been added to the standard part inventory that, although classified as flanges, are in fact welding necks. In total, the following welding neck variants are available:

Variant	Standard designation	
KM10357_WN_ DIN.VAA	Kieselmann 10357	
ROFI10357_WN_ ISO.VAA	RO-FI 10357	Contains all sub-types of the modelled welding neck
ROFI10357_WN_ISO_ R1.VAA	RO-FI 10357 R1	Each variant contains a portion of the sub-types from ROFI10357_WN_ISO.VAA
ROFI10357_WN_ISO_ R2.VAA	RO-FI 10357 R2	

The variants refer with regard to their connection type to the new loose flanges of the standard DIN 2642, which are classified as asymmetrical fasteners:

Series	Variant	Standard designation
1	N2642_LF_R1.VAA	N2642 R1
2	N2642_LF_R2.VAA	N2642 R2

The loose flanges of Series1 match the Variant ROFI10357_WN_ISO_R1.VAA. The loose flanges of Series 2 match the Variants ROFI10357_WN_ISO_R2.VAA und KM10357_WN_DIN.VAA.

Attributes for the fourth connection

Previously, the HiCAD standard parts inventory only contained such parts with 4 connections that had identical third and fourth connections. HiCAD 2019 SP1 offers new cross-pieces ("Crosses") for which this is different. Therefore, the following system attributes have been added:

CNT4	Connection type 4
DA4	Outer diameter 4
DN4	Nominal diameter 4
DNI4	NPS inch 4
WTH4	Wall thickness 4

The corresponding attributes are available in HELiOS. To assign these attributes to those of the HiCAD catalogue, the following system files in the HiCAD folder PlantParts/CatSearch have been expanded accordingly:

File	Expansion
3110010.CatSearchAtt.txt	ANSCHLUSSART2
4300010.CatSearchAtt.txt	ANSCHLUSSART2
3300010.CatSearchAtt.txt	ANSCHLUSSART2 und ANSCHLUSSART4
4400010.CatSearchAtt.txt	ANSCHLUSSART2 und ANSCHLUSSART4

Expanded database

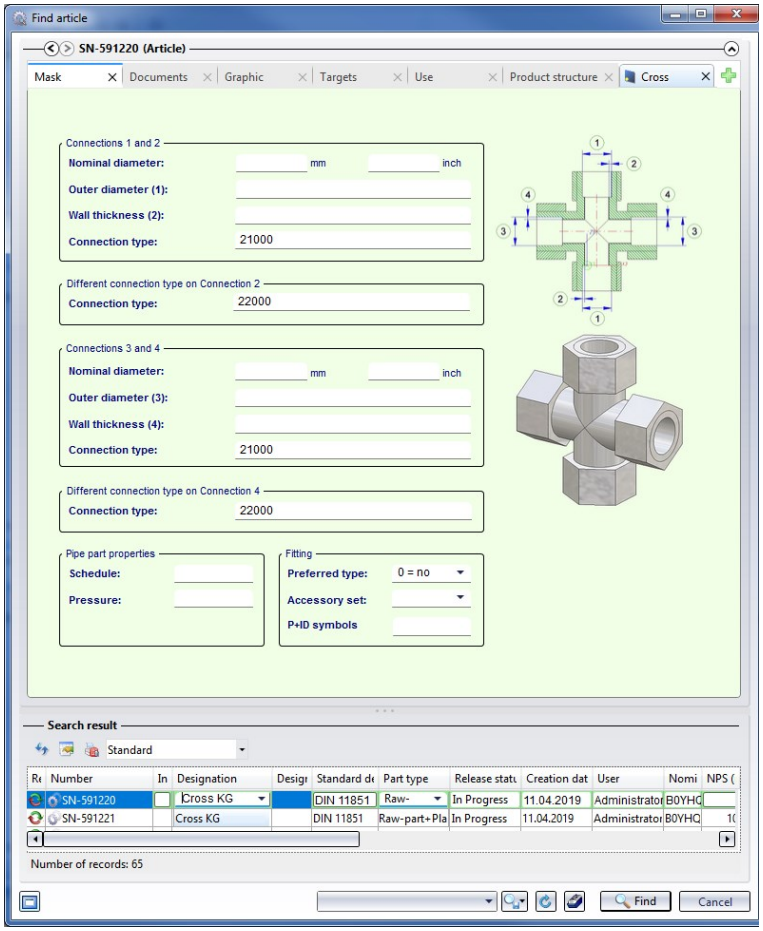
Attributes for fourth connections

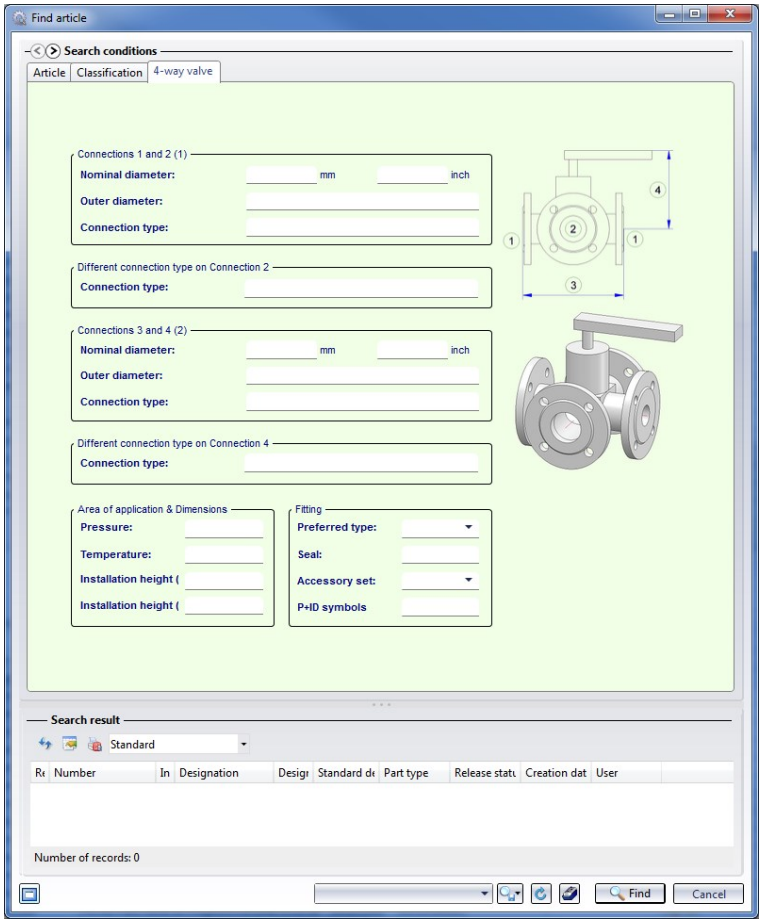
When preparing a HELiOS database for Plant Engineering tasks with the DBPlantDataImport tool, the following additional attributes will be created for a better support of parts with 4 connections:

- NENNWEITE4 (Nominal diameter 4)
- WANDDICKE4 (Wall thickness 4)
- ANSCHLUSSART4 (Connection type 4)
- N4_INCH
- NPS4_INCH
- D4_AUSSEN (D4 Outer)

Enhanced search masks for Cross and 4-way valve

The new parts acc. to DIN 11851 for food ducts used in the food industry include crosses and valves the connection types for Connections 3 and 4 differ from each other. From HiCAD 2019 SP1 onwards, the HELiOS search masks will therefore contain corresponding new input fields, so that separate input fields now exist for deviating connection types for the Connections 2 and 4.

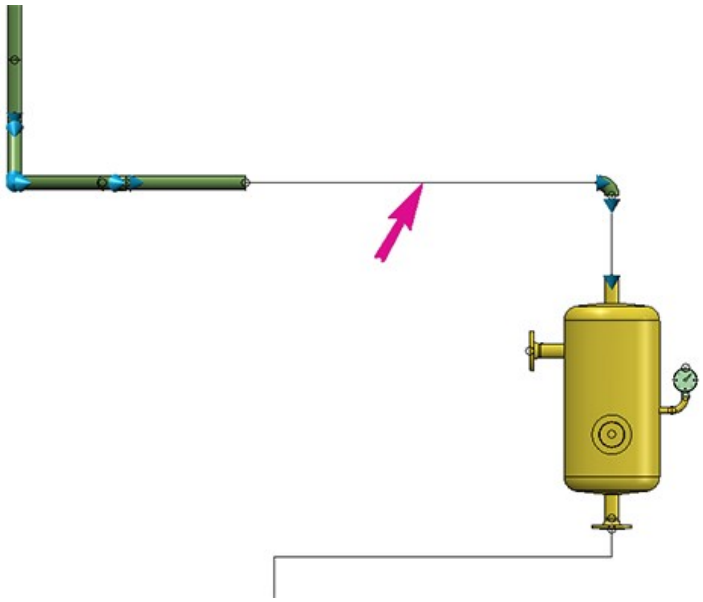




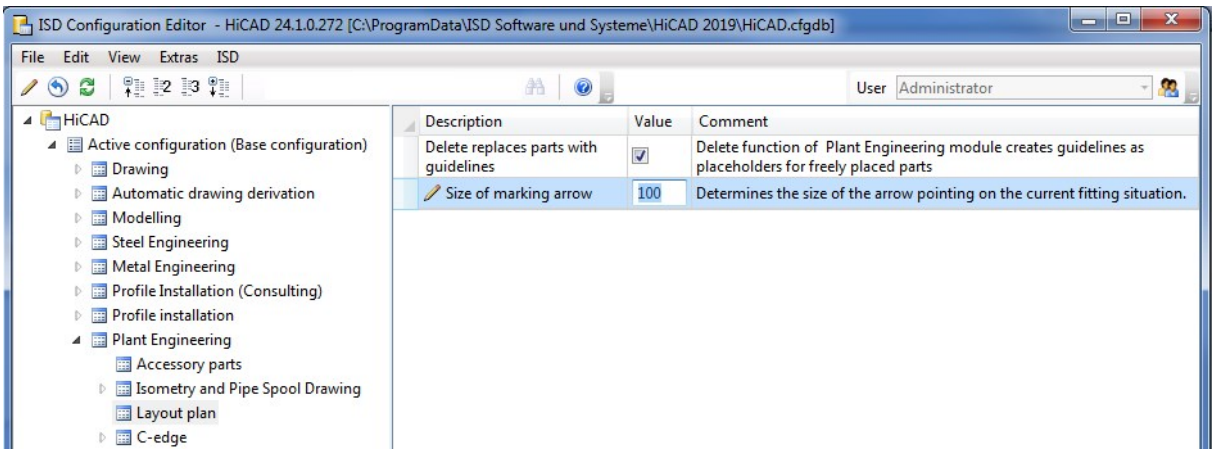
Other new features

Specify size of marking arrow

For various functions, e.g. the automatic placing of parts onto guidelines or the exchanging of parts on guidelines, an arrow pointing on the currently processed part or guideline will be displayed. The representation of the marking arrow has been changed in HiCAD 2019 SP1.



Also, the size of that arrow can be specified in the Configuration Editor at **Plant Engineering > Layout plan**. The value must be between 10 and 300 pixels, the default value is 80.



Changing the part type ID during automatic deriving of variants

From HiCAD 2019 SP1 onwards, you can also change the part type ID, and thus also the classification of a VAA and a PAA file when automatically deriving variants.

The attribute assignment file (attribute_assignment.csv) supports the pseudo attribute **VAREDIT_KEEP_DB_IDS** for this purpose.

If this attribute has the value 1, the previous HELiOS IDs will be preserved during automatic deriving. i.e. no "actual" deriving takes place. From a HELiOS perspective, the resulting VAA or PAA file refers to the same parts. This is helpful for the adjustment of existing VAAs or PAAS, especially in conjunction with VAREDIT_MOVE_ON_SUCCESS.

Also, it is now possible to map the attribute ARTSCHLUESSEL (Type key) to a column in the file customer_list.csv. In this column the part type ID of the VAA or PAA file must then exist in the format AAAAA_BBBBBBB, with AAAAA being the coding for the industry according to **anbtlken_top.dat** and BBBBBBB the ID according to anbtlken.dat.

If the format should not be correct, the message

Classification not in the format AAAAA_BBBBBBB, with A = Industry and B = ID

will be issued in the output column.

PartDataAutoSync with check for part type ID

If you use the PartDataAutoSync.exe tool for part data synchronization with the HELiOS database, you can now check whether the part type ID of a VAA or PAA file has changed. If this is the case, the classification of the article masters in HELiOS will be adjusted accordingly.

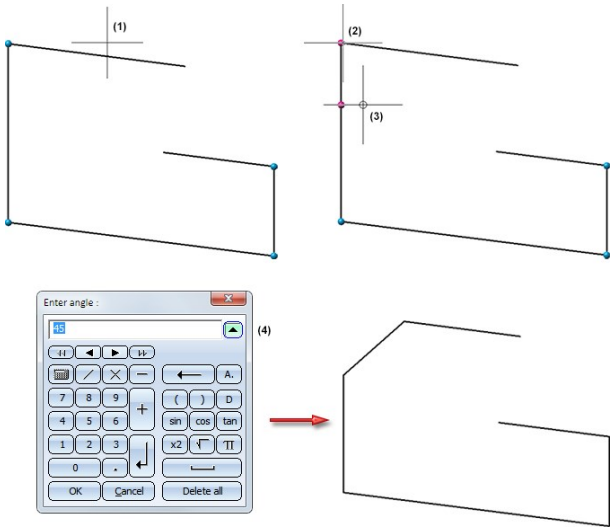
Major Release 2019 (V 2400)

Processing of guidelines

Cut off corner

The **Cut off corner** function has been revised completely. As of HiCAD 2019 the identification of the cut takes place in 4 steps:

1. Select the guideline.
   All corners where a cutting off is possible will be marked.
2. Choose the corner to be cut off.
   If there is only one corner, this corner will be selected immediately.
3. Determine the leg start.
   The leg start can be located on one of the two edges that converge in the selected corner point.
4. Enter the angle to the leg.



During selection of the corner point or the leg start, respectively, you can go back to the previous selection step via the corresponding context menu function. When determining the angle, this happens by entering an empty angle.

Place pipe parts on guidelines

Change insertion direction

In previous versions the insertion direction of pipe parts on guidelines - **In edge direction** or **In opposite edge direction** - needed to be specified during part selection: After part selection the direction could not be changed any more. As of HiCAD 2019 it is possible to change the insertion direction of a part during its placing in the drawing.

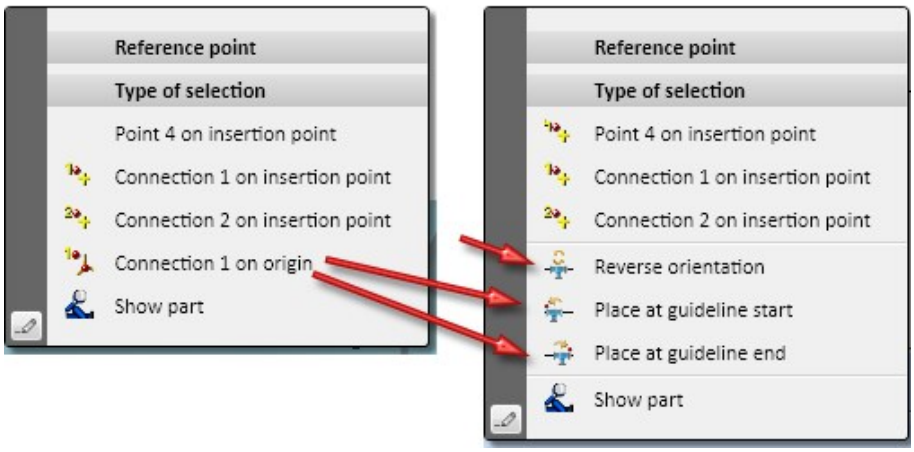
For this purpose the new **Reverse orientation**  function has been added to the **Reference point** context menu.

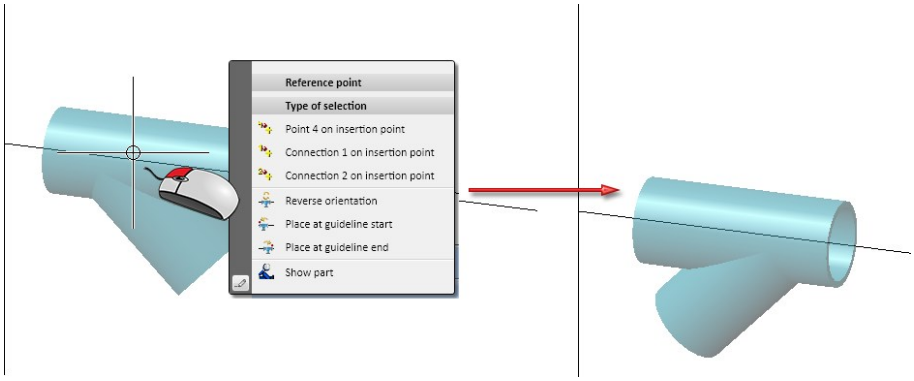
In addition, the part can now also be placed directly at the start or the end of a guideline. For this purpose the previous option **Connection 1 on origin** has been replaced with the options

 **Place at guideline start** and

 **Place at guideline edge**

The **Reference point** menu can be opened with a right-click whenever HiCAD requests the fitting point specification in the drawing.



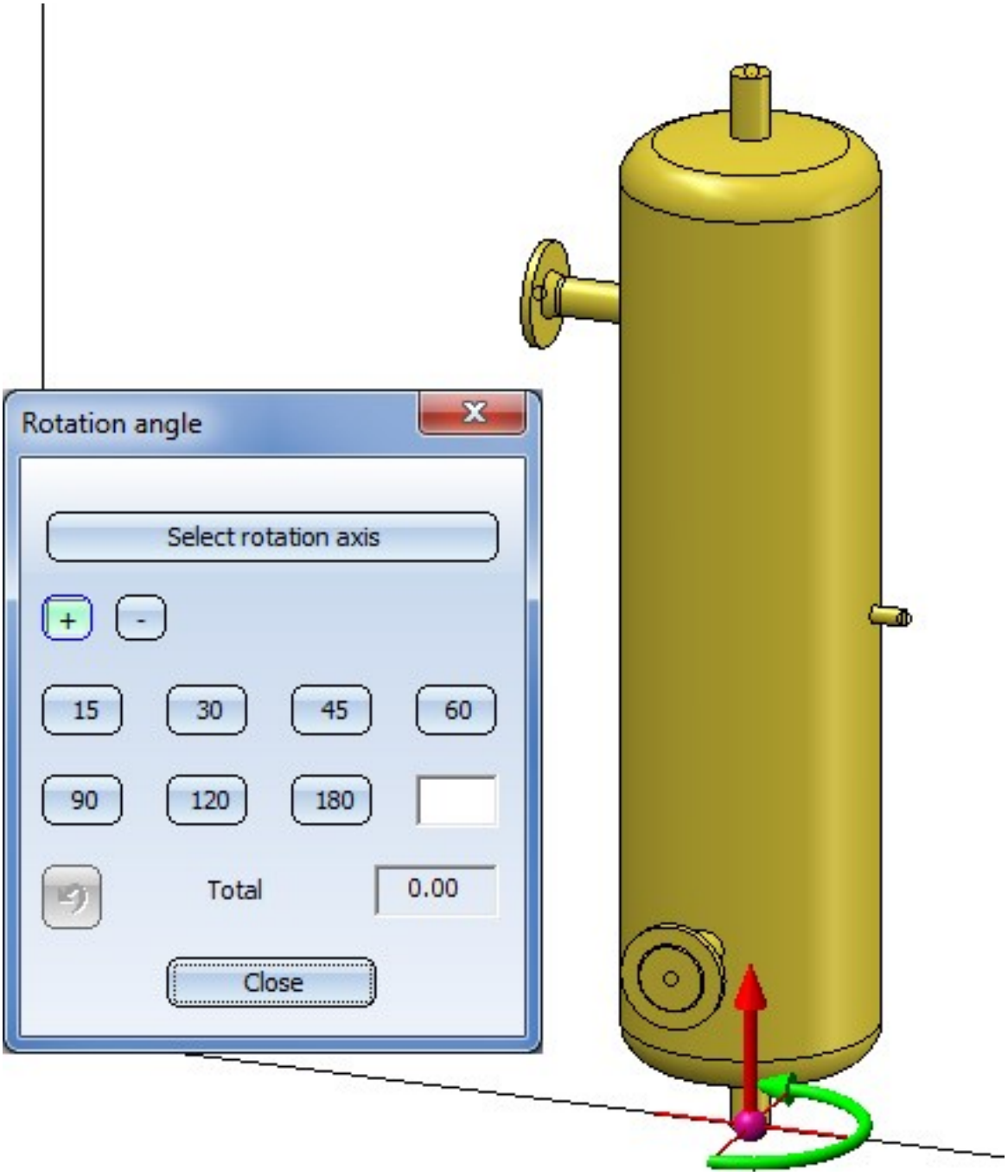


The new options are also available during exchanging of parts.

Rotate parts

Free selection of rotation axis

The dialogue for rotating of parts after their insertion has been revised. Up to HiCAD 2018 you could only select one connection number here. As of HiCAD 2019 rotation axis and rotation direction will be graphically visualized and the rotation axis can be freely selected.



This is also true for the rotating and exchanging of parts.

Extended UNDO

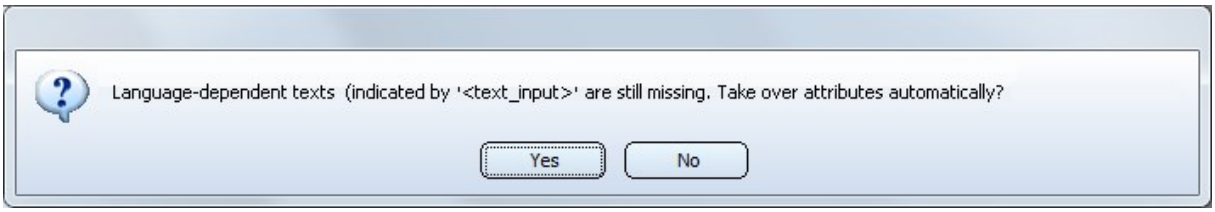
The functionality of the UNDO button in the Rotation angle menu has been enhanced. Previously, the angles had been added here and then, all rotations were revoked with one UNDO action. In some situations this had led to errors, e.g. in situations where a part was first rotated, then the connecting point was changed and, finally, the part was rotated again. As of HiCAD 2019, the UNDO button revokes the rotation about the last selected rotation axis. If the rotation is 0°, the last change of the rotation axis will be revoked. In this way, it is possible to revoke several axis changes and several rotations independently from each other.

PAA Editor and VarToCat

Take over attributes for all languages

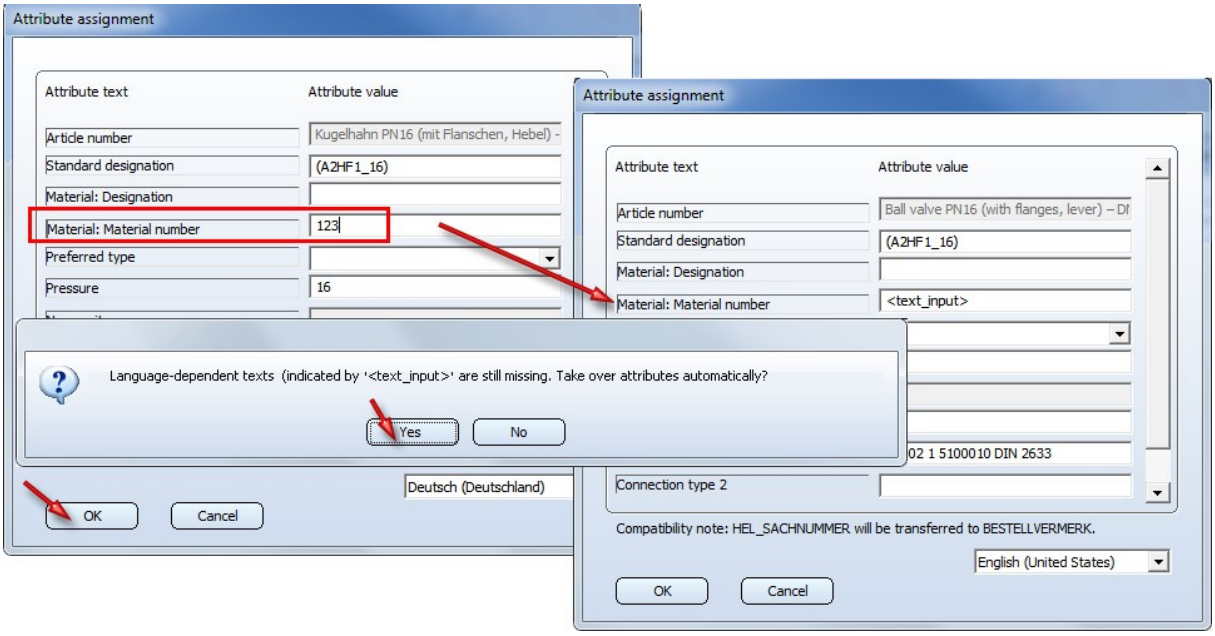
If you have used the PAA Editor or the VarToCat tool in HiCAD 2018 (or older versions) to change attribute assignments and confirmed them with **OK** without adjusting the attributes in all languages, the corresponding Editor issued a warning, and entered the text **<text_input>** into the concerned input fields.

As of HiCAD 2019, changed attribute assignments can be automatically taken over into the other languages. If attributes are missing, the following message will be displayed:



If you choose **Yes**, the attribute from the first language, in which has a value, will be transferred to the corresponding fields of the other language, if these are empty; otherwise, the current content of the field will remain unchanged.

If you choose **No**, the text **<text_input>** will be entered in the corresponding fields of the other languages.



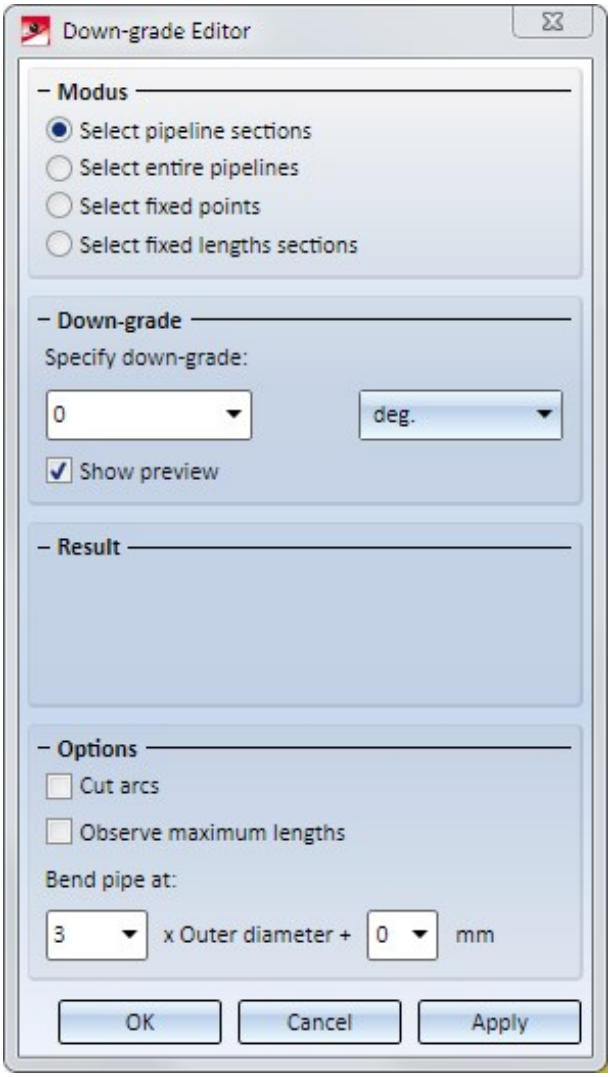
Pipeline Tools

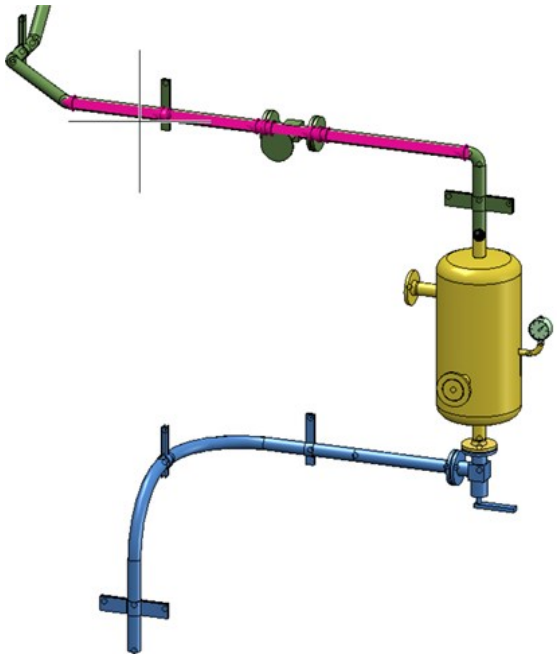
Down-grade Editor

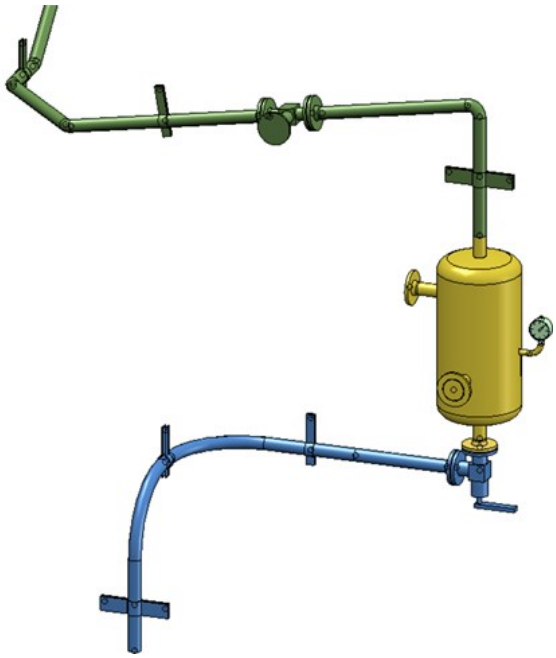
With the new **Down-grade Editor**  you can assign down-grades to individual sections of a pipeline. You find the Down-grade Editor at **Plant Engineering > Pipeline Tools > Change** **> ...**. The Down-grade Editor works according to the following principles:

- The down-grade always refers to the XY-plane of the active coordinate system.
- Down-grades can only be assigned to such sections that run approximately horizontal. "Approximately" here means: less than 45° or less tan 100% down-grade.
- The routes of sections the down-grades of which exceeds 45°, i.e. especially vertical sections, will not be changed. Such sections are not selectable either.
- Sections end at elbows and knees.
- Branches and T-pieces are assigned to one section, unless the flow direction changes within a part (see below).
- Existing lengths will be preserved if possible. If this is not possible, the changes will be distributed evenly among the pipeline.

When you choose the function the **Down-grade Editor** dialogue window will be displayed:


Down-grade Editor
Modus
Select pipeline sections
Select entire pipelines
Select fixed points
Select fixed lengths sections
Down-grade
Specify down-grade:
0
deg.
Show preview
Result
Options
Cut arcs
Observe maximum lengths
Bend pipe at:
3
x Outer diameter +
0
mm
OK
Cancel
Apply





Changed menus

The new Down-grade Editor refers to the flow direction when assigning down-grades, i.e. as of HiCAD 2019 the relevance of the flow direction concerns not just isometries. The functions

 **Edit flow** and

 **Check flow**

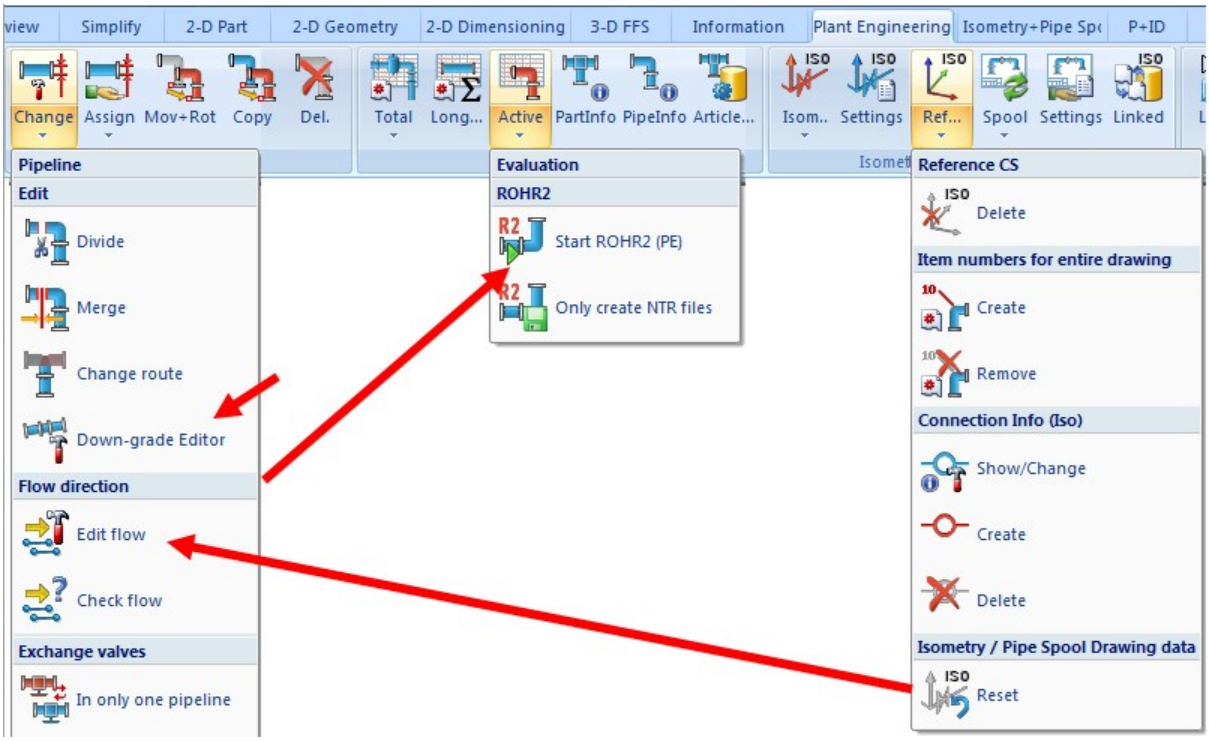
have therefore been moved from the menu **Plant Engineering > Isometry / Pipe Spool Drawing** > **Ref.** > **...** to the menu **Plant Engineering > Pipeline Tools > Change** > ....

Also, the functions

 **Start ROHR2** and

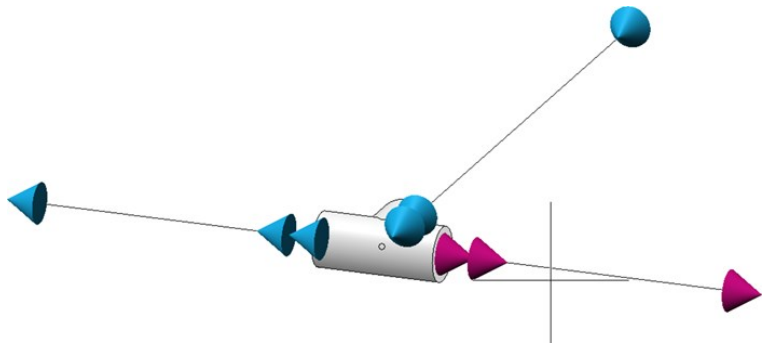
 **Only create NTR file**

from the menu **Plant Engineering Pipeline Tools > Change** > **...** have been moved to the menu **Plant Engineering > Evaluation > Active** > ....



Flow Editor

In previous versions the Flow Editor could only be used to assign flows to parts. From HiCAD 2019 onwards the Flow Editor can also be used on guidelines.



Configuration of the HELiOS Databases - DBPlantDataImport.exe

Using existing article masters for automatic deriving of variants

If Plant Engineering variants with existing HELiOS article masters are to be used, you had only the option to enter a matching HELiOS URL for each sub-type directly during part synchronization. On the one hand, this procedure is very tiresome, and on the other hand, it requires at least the explicit permission to create article masters in HELiOS. Especially the latter may be problematic in configurations where HELiOS obtains the article masters downstream from other ERP systems. To support such cases, the mechanism for automatic variant derivation has been enhanced.

To transfer VAA files to HELiOS that were generated with the new mechanism, the tool **DBPlantDataImport.exe** must be called with the parameter **/X**. This will enable you to adjust existing article masters in a Plant Engineering compliant manner.

Further information on this procedure can be found in the Online Help of the Variant Editor, in the topic Automatic Creation of Derived Variants.

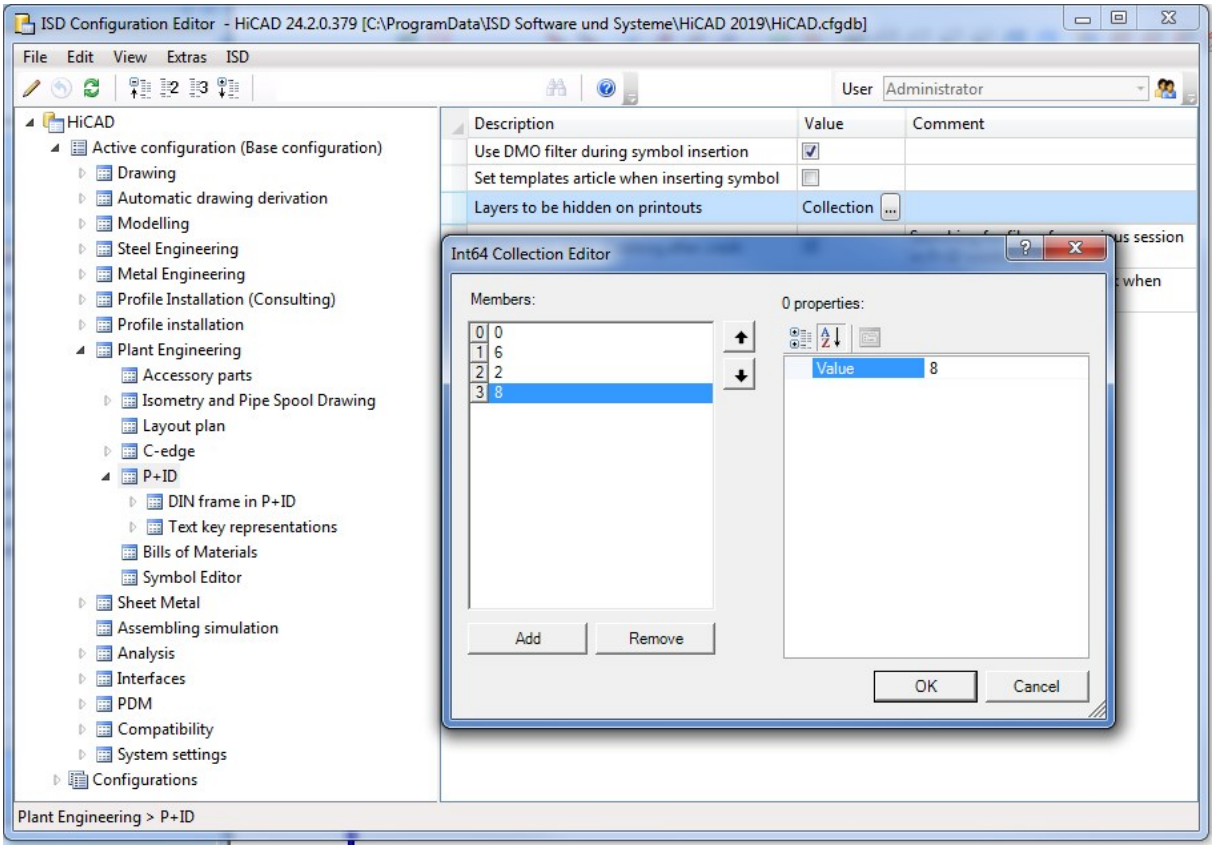
Example: Let us assume that you choose the dialogue type **Code** for the Plant Engineering attribute 1 and, in the P+ID, change the Code for a Vessel symbol from B1 to B2. When switching to the 3-D layout plan, the PE attribute 1 will be changed for the affected Vessel from B1 to B2. This will also be confirmed by a corresponding info message.

P+ID

Service Pack 2 2019 (V 2402)

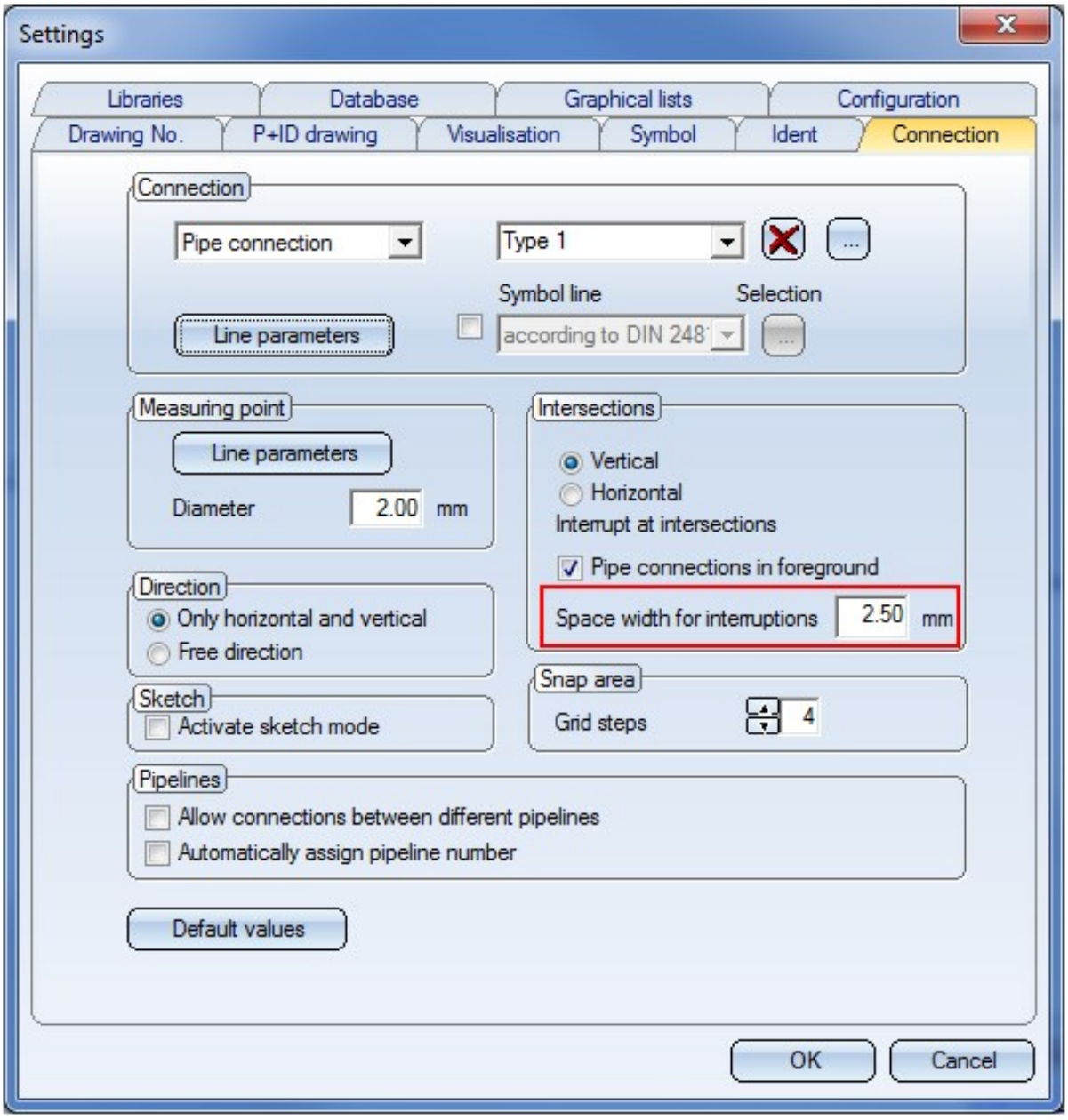
Automatically hide layers for printouts

You have now the option to specify layer-specific printing settings in the P+ID module via the Configuration Editor (at Plant Engineering > P+ID). For this purpose you can define a list with layers that are to be excluded for printouts. Click the **Add** button to expand the list. Beneath **Properties** you can then enter the number of the layer.



Configurable space widths for connection interruptions

You have now the option to define the space widths of connection interruptions. For this purpose you can find an editing box on the **Connections** tab of the P+ID **Settings** dialogue window, where you can enter the desired space width for interruptions in mm.



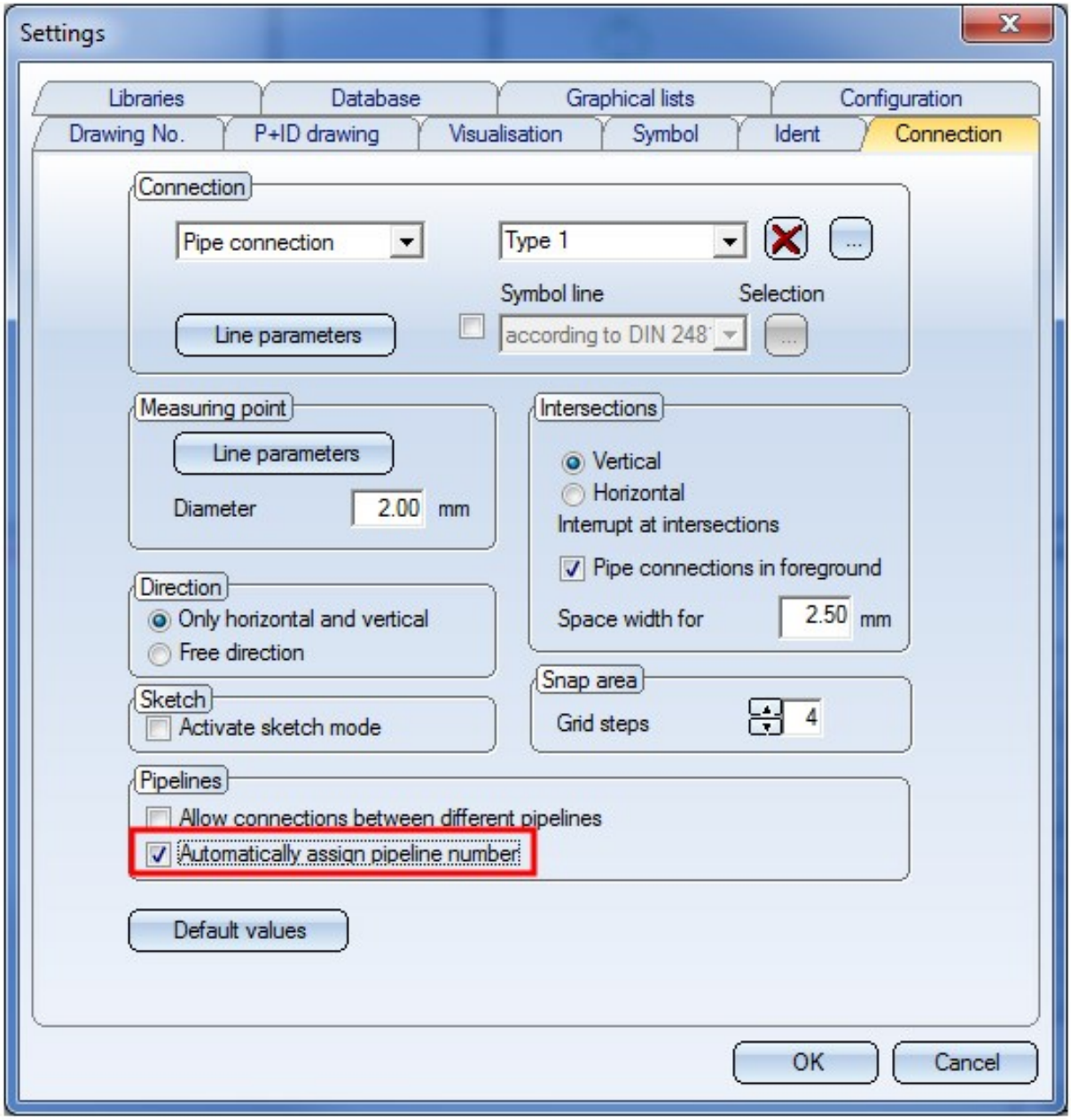
New procedure for creating new pipelines

The procedure for creating new pipelines has been changed as follows:

1. When a new pipeline is created, no message appears any more.
2. The pipeline symbol is automatically inserted at a particular place.
3. The flow direction is set from start to destination.
4. You have the option to have pipeline numbers generated automatically.

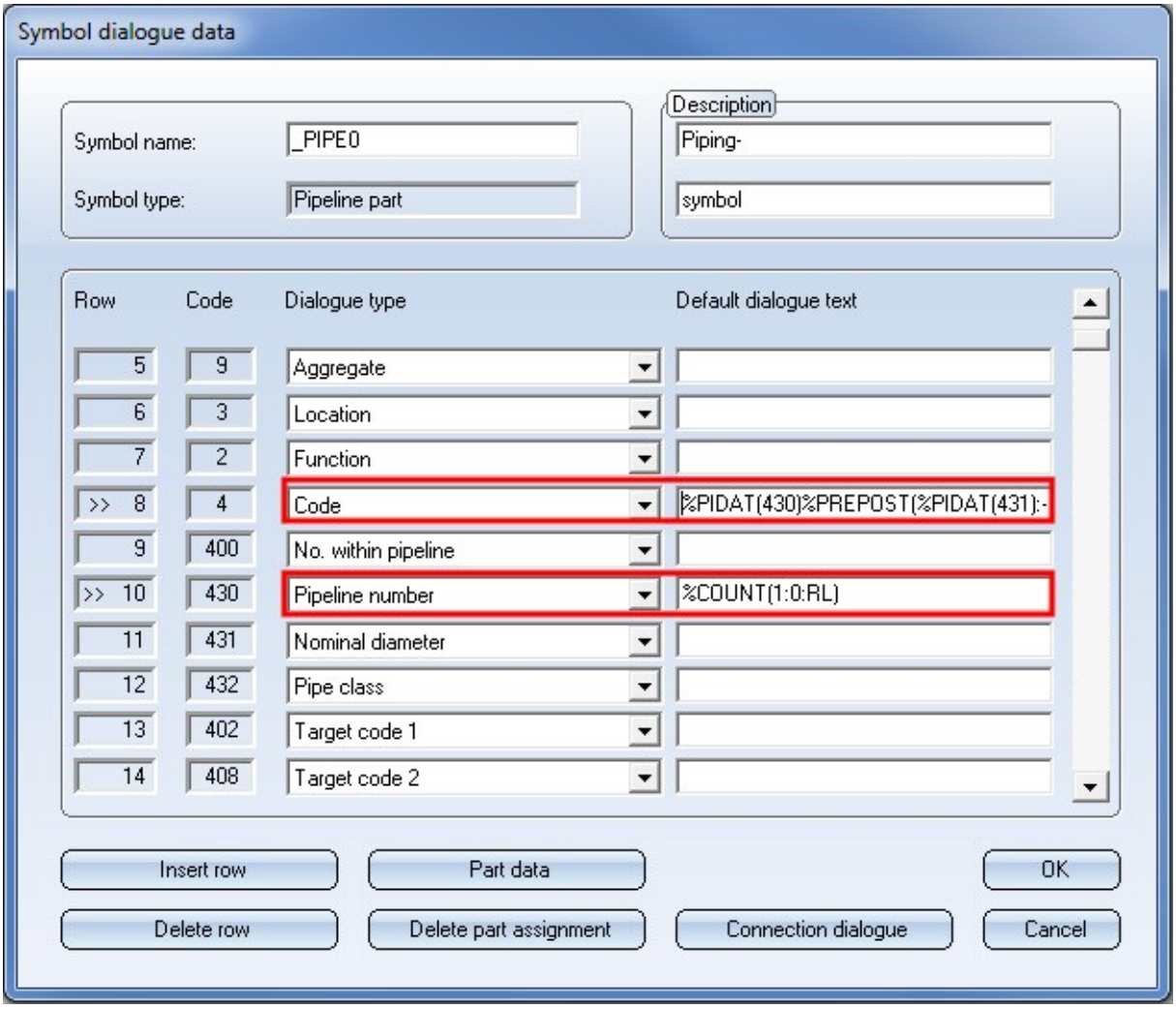
Automatically assign pipeline numbers

First, open the **Connection** tab of the P+ID **Settings** dialogue window and activate the ☑ **Automatically assign pipeline number** checkbox.



Then, you need to specify various settings in the symbol mask for the pipeline symbol, as the related entry for the attribute **Pipeline number** plays a decisive role for the generation. The entry %COUNT(1:0:RL) , for example, ensures that natural numbers beginning with 1 will be used as pipeline numbers. The entry %PIDAT(430)%PREPOST(%PIDAT(431):-)%PREPOST(%PIDAT(432):-) for the **Code** ensures that codes in P+ID are generated in the way would expect it.

Further useful information about generation instructions can be found in the topic Take Over Database Attributes as Dialogue Text.



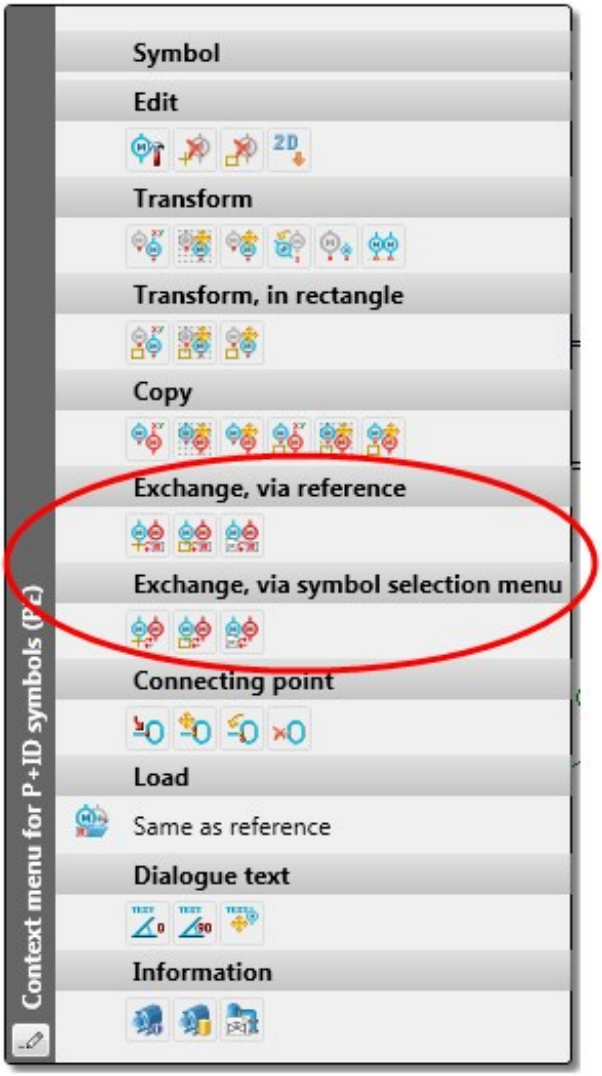
The mask as shown in the image above is the default mask in the P+ID libraries ISDDINSYM1_GER and ISDDINSYM1_ENG, respectively.

Service Pack 1 2019 (V 2401)

Exchanging symbols

You can now choose whether you want to exchange P+ID symbols via a reference symbol (as in previous versions) or determine a matching symbol via the symbol selection dialogue (new).

The context menu of symbols (right-click) now looks as follows:



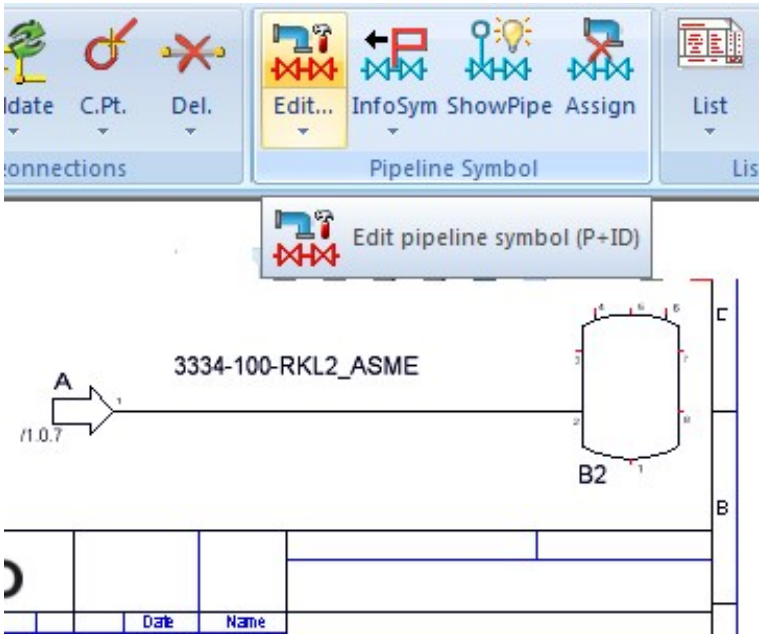
On the toolbar the following editing options are now available:

	Exchange, via reference - Individual Exchange, via symbol selection menu - Individual	If the exchange is valid, the selected symbol is replaced.
	Exchange, via reference - In rectangle Exchange, via symbol selection menu - In rectangle	This option enables you to draw a rectangular box by specifying two points with the cross-hairs. All symbols within this box matching the selected symbol are replaced. HiCAD then highlights these symbols. If required, the highlighting can be retained until the next time the screen content is redrawn.
	Exchange, via reference - Sheet Exchange, via symbol selection menu - Sheet	This option exchanges all symbols on the current drawing sheet that match the currently selected symbol. HiCAD tells you how many symbols have been replaced and then highlights them. If required, the highlighting can be retained until the next time the screen content is redrawn.

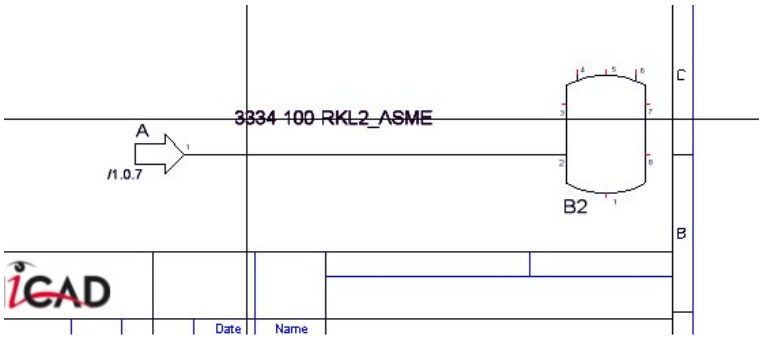
Edit pipeline symbols from a different sheet

You can now even edit a pipeline symbol that does not exist on the current drawing sheet, but to which pipeline parts or pipeline info symbols have been subordinated on the current sheet. To do this, edit the subordinated parts.

To do this, activate the **Edit pipeline symbol** function on the **P+ID** Ribbon tab and choose a symbol that has been subordinated to the pipeline symbol.



Then, choose a symbol that is subordinated to the pipeline symbol.



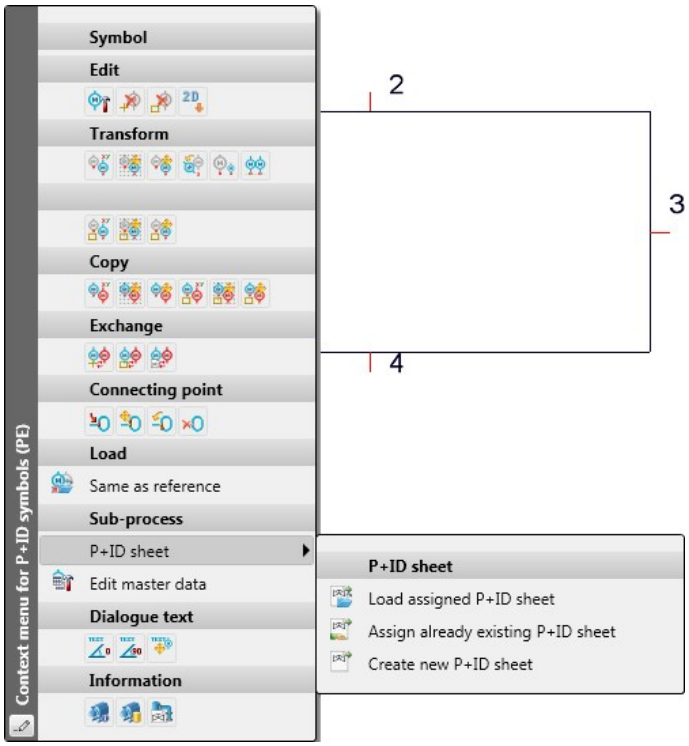
The Edit symbol mask will then be displayed for the pipeline symbol.

Major Release 2019 (V 2400)

Block symbols

HiCAD P+ID R+I now supports so-called block symbols. These are symbols that represent a P+ID sheet.

In the ISD library ISD_PROCESSFLOW.SZA such a block symbol is already available. Users can create such symbols using the **Symbol type 18 = Schema block**. After inserting a block symbol, you can call the editing functions for the block symbol with a right-click on the symbol.



Load assigned P+ID sheet

If a P+ID sheet has been assigned to a block symbol, you can open the assigned sheet via the context menu of the symbol.

Assign already existing P+ID sheet

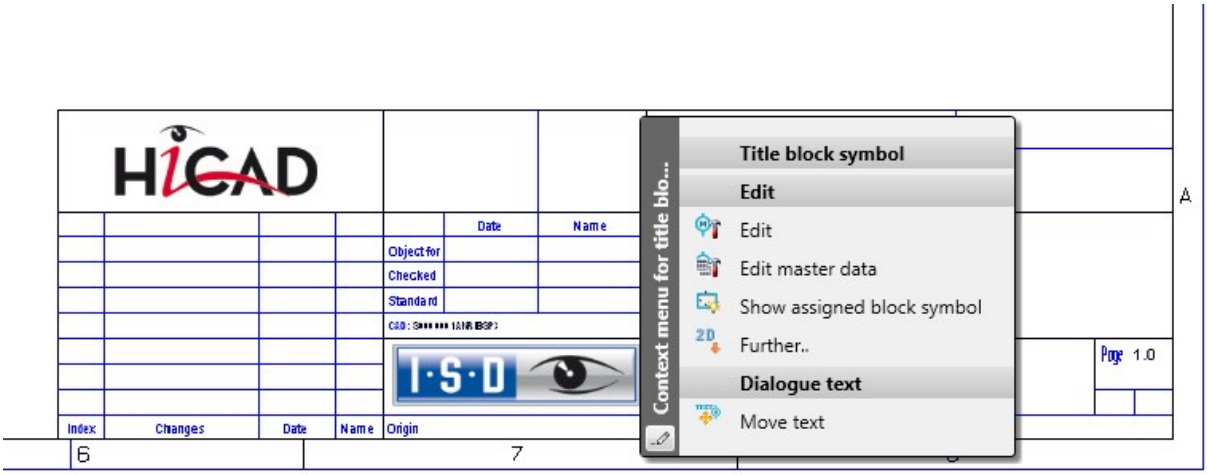
Use this function to open a dialogue window with a list of already existing, selectable P+ID sheets that have not been assigned to a a block symbol yet. If no P+ID sheets are available for selection, a corresponding message will be displayed.

Create new P+ID sheet

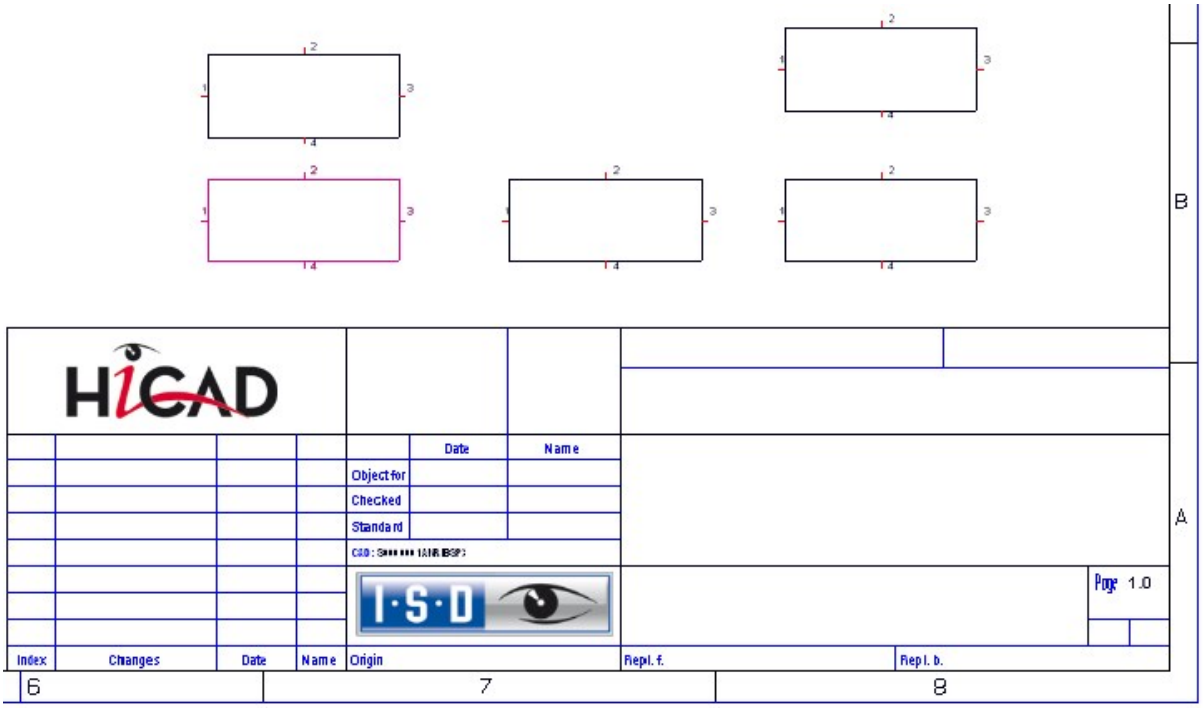
This function opens the known master data dialogue window when creating a new sheet.

Edit master data

Use this function to edit the master data of the assigned P+ID.

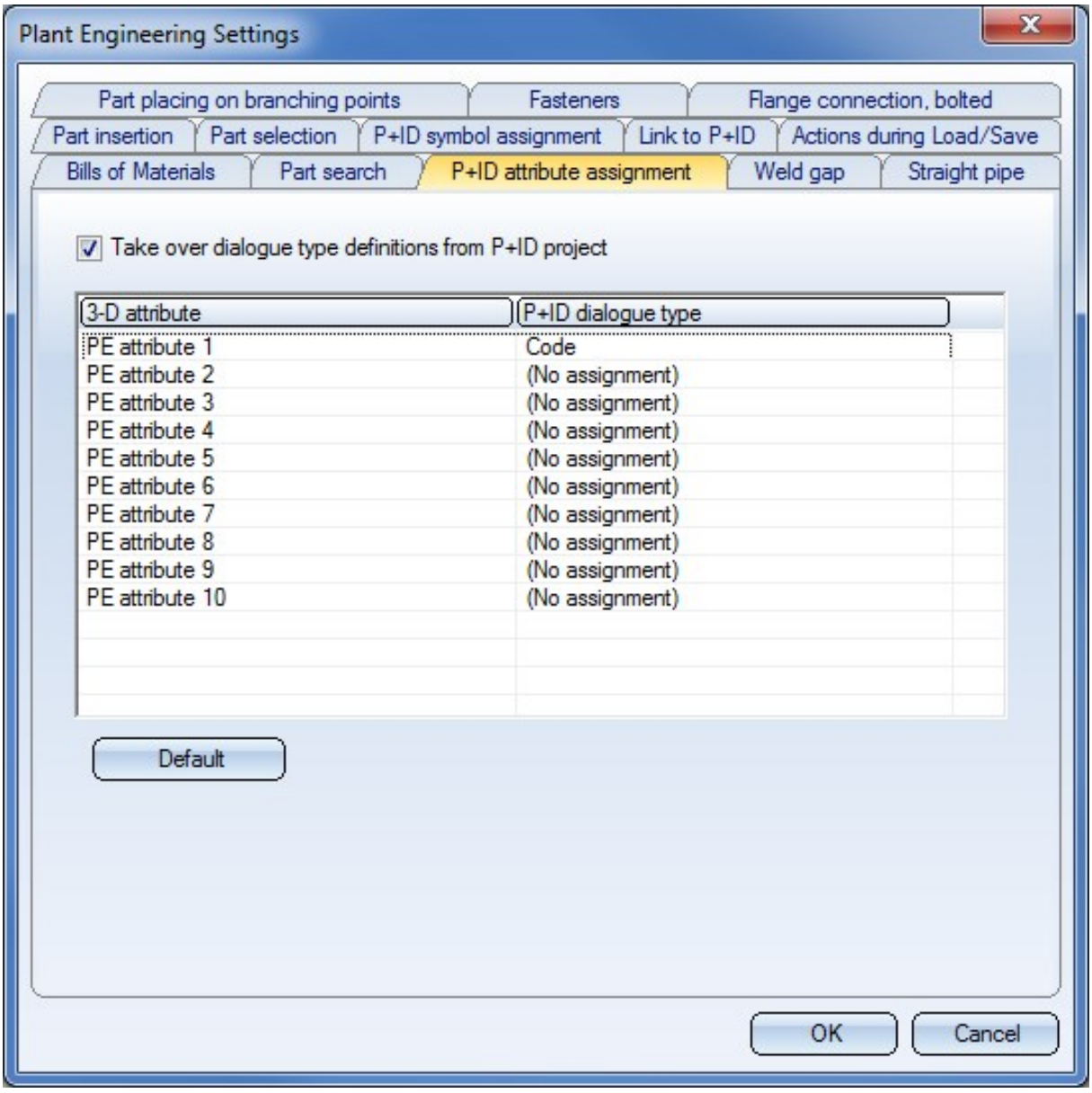


When you right-click on the title block of a sheet and choose **Show assigned block symbol**  , the corresponding P+ID sheet containing the block symbol will be loaded, and the block symbol will be highlighted:



Take over P+ID attributes from P+ID

P+ID dialogue types can be assigned to user-defined attributes can be assigned to on the p+ID attribute assignment tab of the **Plant Engineering Settings** dialogue window that you open by choosing **P+ID > Settings**  .



When you change one of the assigned dialogue types for a symbol in a P+ID, and then switch to the 3-D layout plan via **P+ID > Link to 3-D > Assigned 3-D layout plan**  , the new value of the dialogue field will be applied to the attributes of the part.

Example: Let us assume that you choose the dialogue type **Code** for the Plant Engineering attribute 1 and, in the P+ID, change the Code for a Vessel symbol from B1 to B2. When switching to the 3-D layout plan, the PE attribute 1 will be changed for the affected Vessel from B1 to B2. This will also be confirmed by a corresponding info message.

Notes on HELiOS Updates

Service Pack 2 (V 2402)

Important Notes on Updates for Administrators

For updates to Service Pack 2 of HELiOS 2019 a central update of the supplied HELiOS database is required (from sub-schema 166 to 194).

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

1.) Make a 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.

2.) If required, use the log file for the 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**).

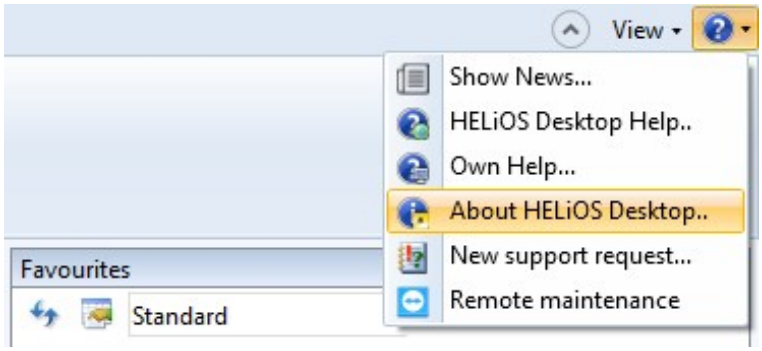
Have this file ready when contacting the ISD Hotline in case of an unsuccessful update, so that they can help you solve the problem and make a successful update.

HELiOS Desktop

Service Pack 2 (V 2402)

About HELiOS Desktop window: Installed Hotfixes

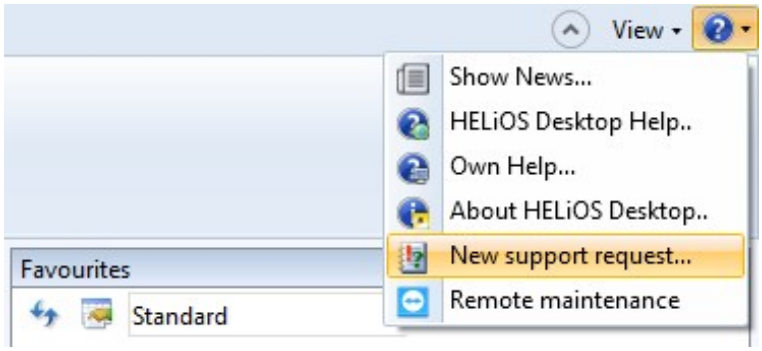
The **About HELiOS Desktop** window now also shows, besides the current Version and Build number, the numbers of installed Hotfixes. To open this window, choose > **About HELiOS Desktop** at the top right of the HELiOS Desktop.



Support request

The new  > **New support request**  function takes you directly to the support portal of the ISD Group, where you can create a support ticket.

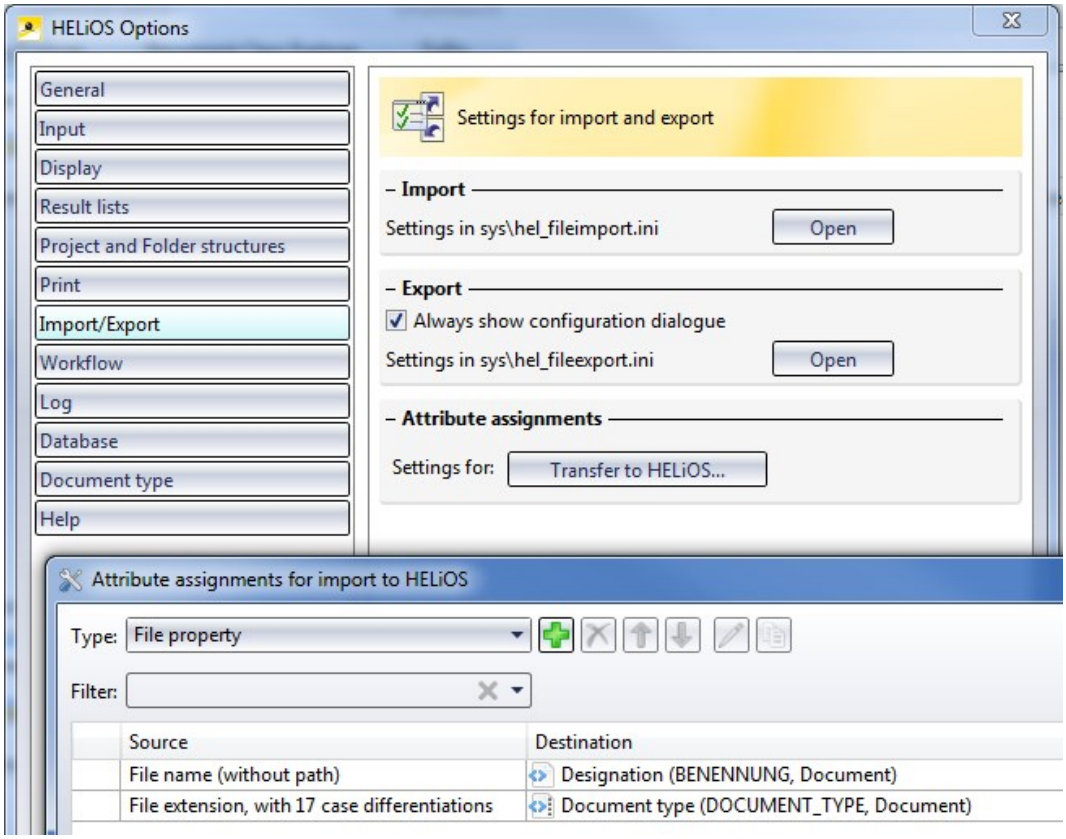
After calling the function, use your login data for my-ISD to access the support portal of the ISD. Fill in the form, attach files if required and click on **Create request**.



The new ISD support portal will be available for customers in Germany from 1 October 2019. For customers in other countries, the launch is scheduled for January 2020.

Attribute mapping for file import via Drag & Drop and serial input

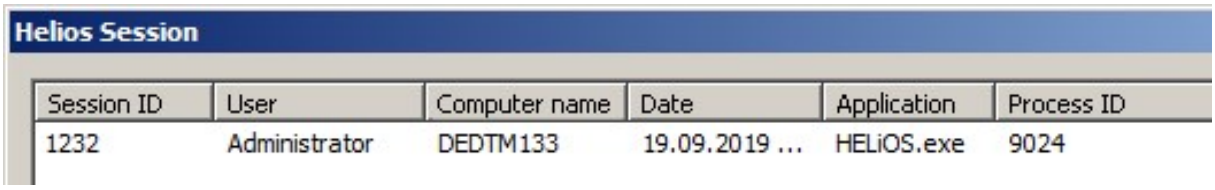
On the **Import/Export** tab of the **HELiOS Options** dialogue window you can find, beneath **Attribute assignments**, setting options for the **Transfer to HELiOS**, which will apply for file import via Drag & Drop.



The procedure is largely similar to the attribute mapping known from the Multi-CAD interfaces and also allows the definition of different templates, which can be used, via menu selection, for the import.

Automatic removing of old sessions

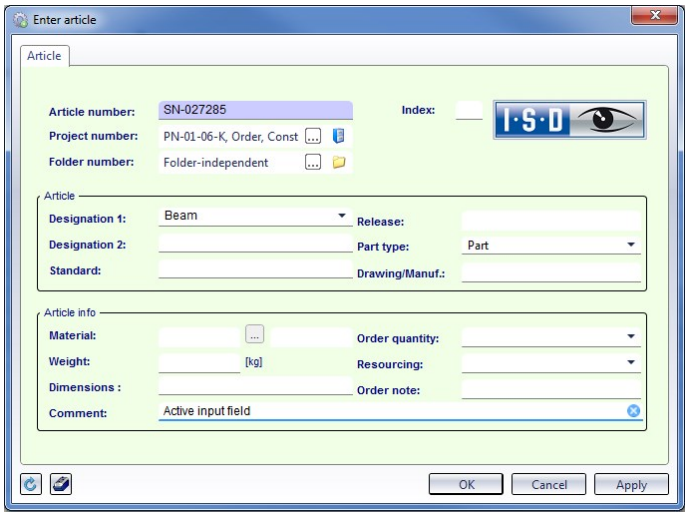
In case of a HELiOS Desktop crash or the crash of a linked application such as HiCAD, it is now ensured that “old” sessions are automatically removed, and thus unlocked, on the relevant computer system when HELiOS is restarted. This means that you do not have to manually unlock any sessions any more.



Service Pack 1 (V 2401)

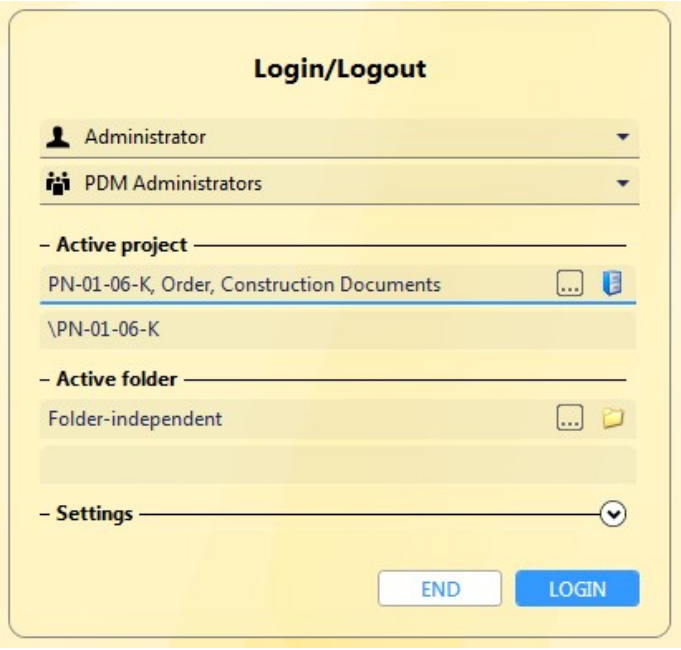
Redesigned user interface

HELiOS 2401 SP1 comes with redesigned and modernized windows and input masks for the HELiOS Desktop and interfaced applications (CAD programs, Office...). Fields with input options are underlined, currently active fields are underlined in blue.

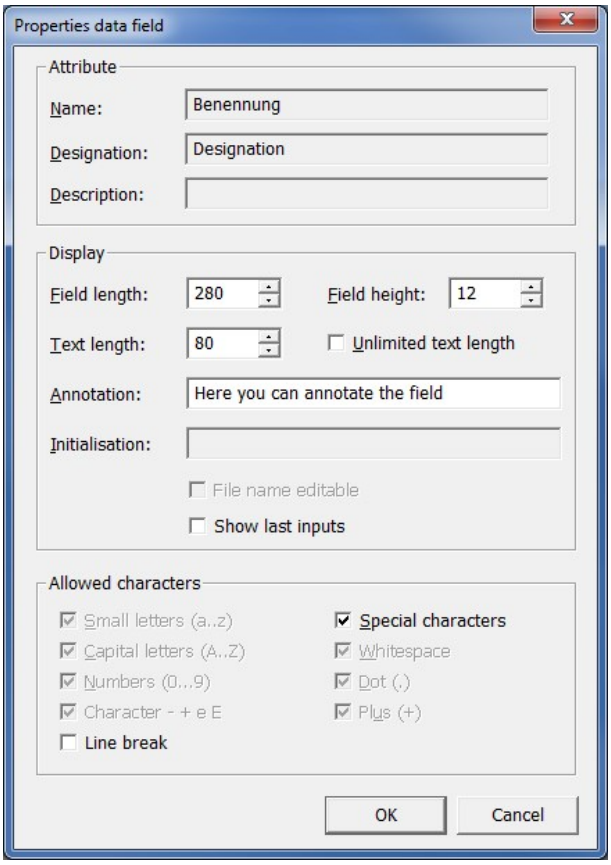


Clicking on the symbol at the end of an active input field clears the field (as an alternative to the Delete key).

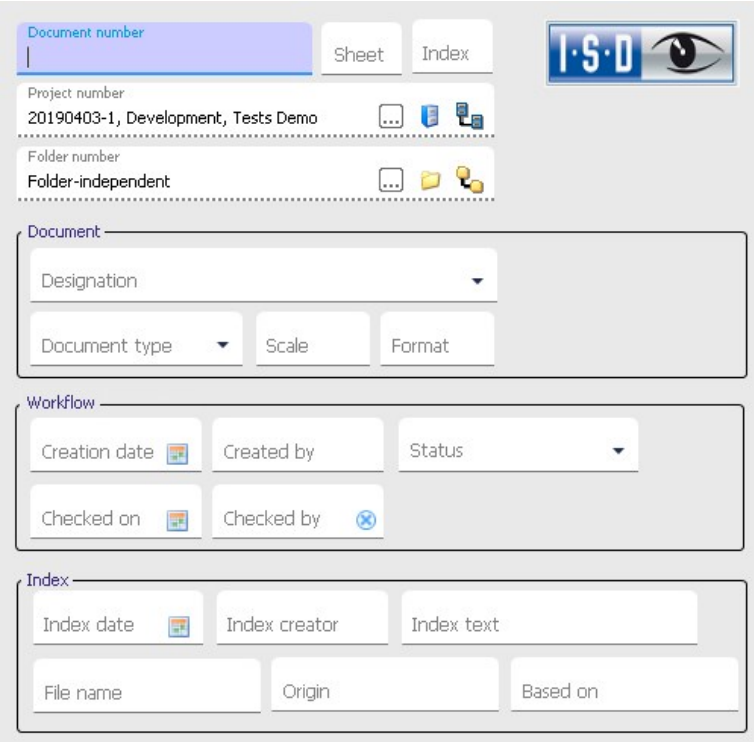
The **Login** dialogue, too, has had a redesign.



The **Mask Editor** now offers the option to add an **Annotation** to an attribute field.



Masks that have been adjusted with the Mask Editor will then appear with the new layout in the HELiOS Desktop.



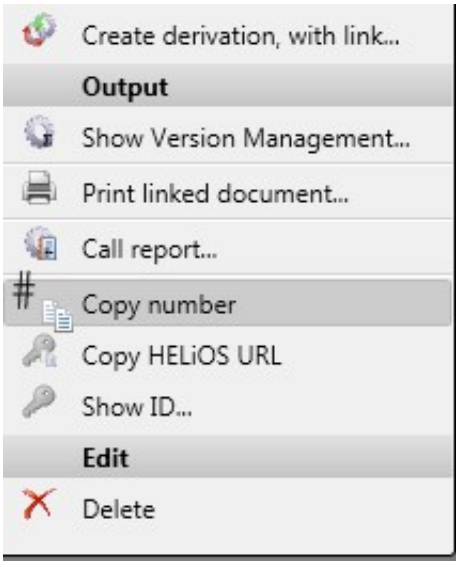
Where-used lists for documents

Like the Article detail masks, the Document detail masks now have a **Use** tab that contains a documents where-used list based on the HELiOS result list technology.

Choose > **Use** to activate this tab.

Copy number

The new context menu function **Copy number** enables you to copy the designation of a HELiOS object (Document number, Article number, Folder name, Project, Material designation) to the Clipboard, e.g. to paste it from there to other applications with only a few mouse clicks.



Major Release 2019 (V 2400)

Support of SQL Server 2019

HELiOS 2019 supports the new SQL Server 2019.

Integrated preview for standard image formats

Image formats such as JPEG, TIF, BMP, PNG, etc. which have been saved as documents to HELiOS can now be displayed by the new, integrated graphic viewer of HELiOS.

You can zoom into graphics and move the selected detail at any time. This allows an easy representation of neutral formats and reduces the number of external viewers that may be required.

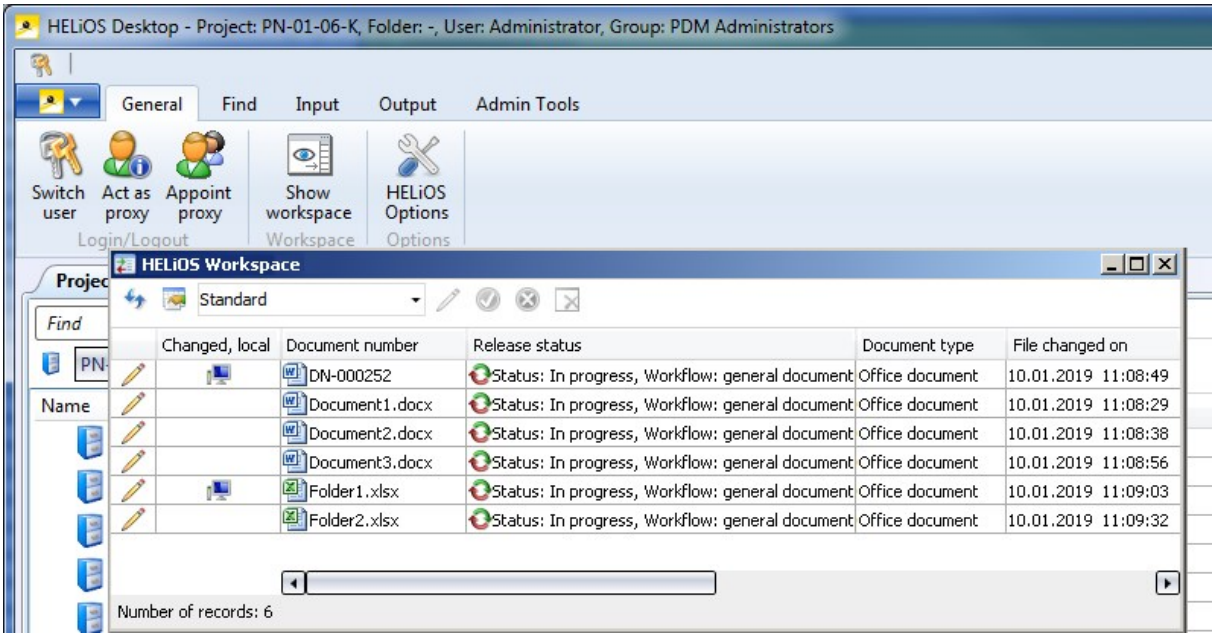
Workspace

As of HELiOS 2019 the **Workspace** that you already know from the Multi CAD and Office interfaces will be available to you in the HELiOS Desktop, too.

Click on the  **Show workspace** icon on the **General** Ribbon tab to open the **HELiOS Workspace** dialogue window.

Here, all documents of the session will be listed. Documents that are opened and/or edited via the HELiOS Desktop will be stored in the workspace directory.

This will constantly provide you with a clear overview of the corresponding HELiOS objects, so that you can check them into the database when required. When closing the HELiOS Desktop a question regarding the handling of objects that are still in the Workspace will be displayed.

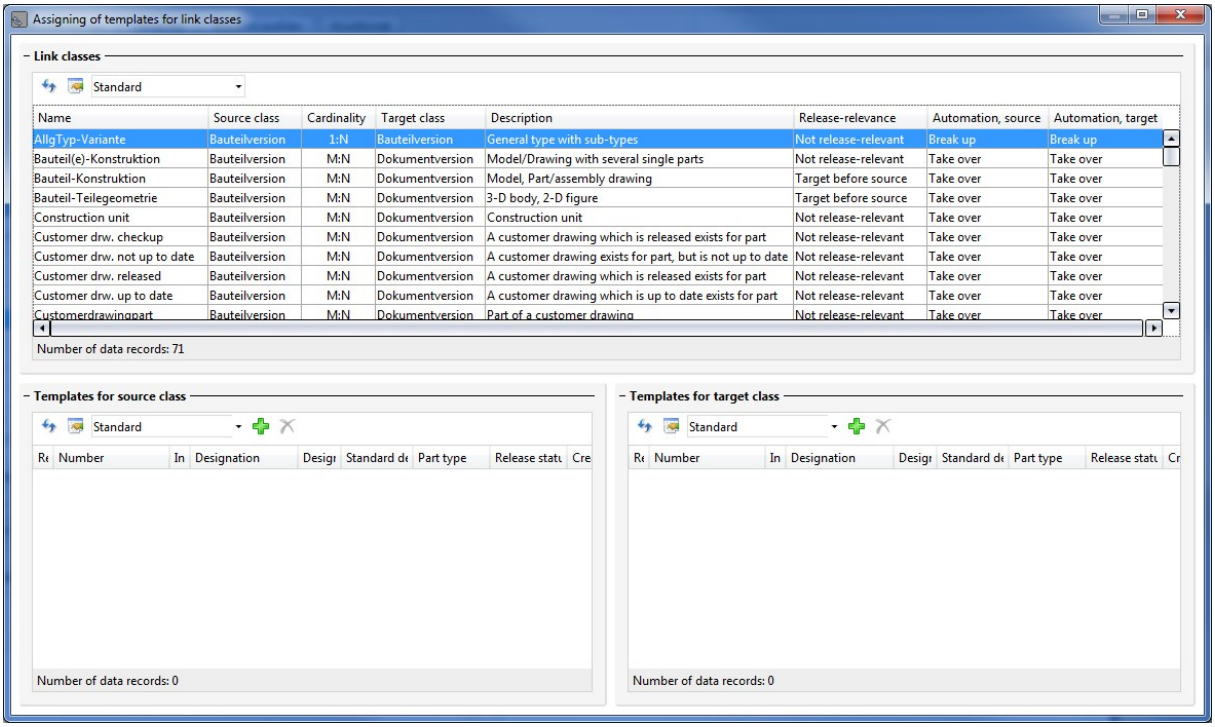


In the new **Changed, local** column the  symbol indicates that a document has been changed locally and saved since its last synchronization with HELiOS.

New dialogue windows

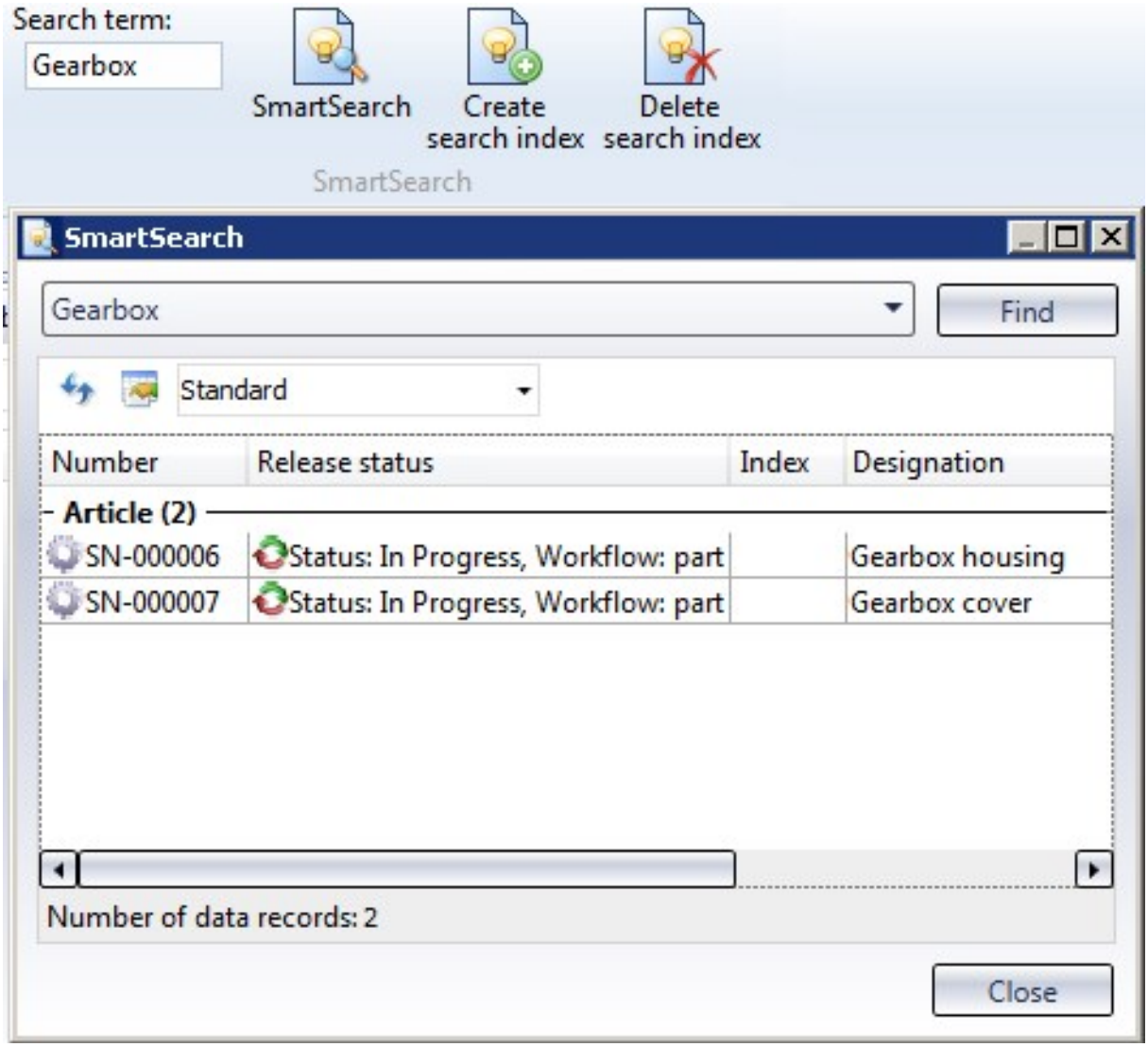
HELiOS 2019 comes with many modernized dialogue windows.

This concerns in particular the **Workflow** -related dialogues and sub-functions (e.g. the **Check list**, the **History** and the **Sub-workflow**) and the range of functions for **Links**, e.g. the assigning of templates for link classes.



Revised SmartSearch dialogue window

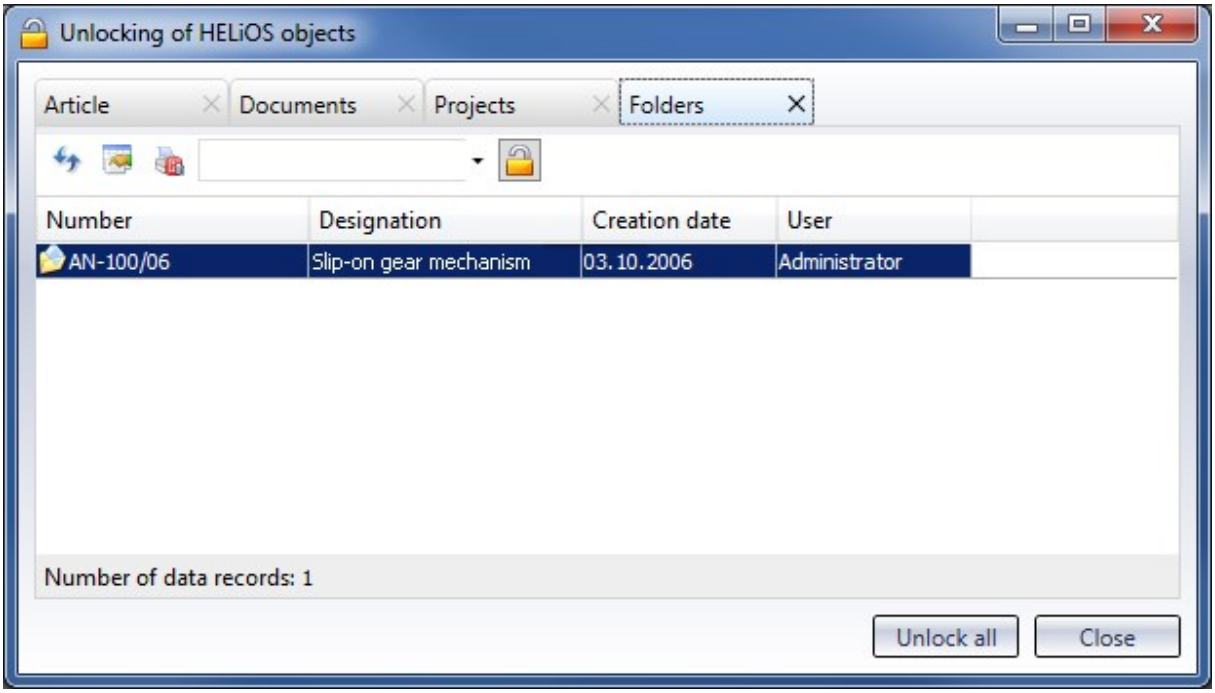
The dialogue window of the HELiOS SmartSearch has had a redesign and was adjusted to current standards.



New dialogue for unlocking of HELiOS objects

For unlocking of objects that are currently edited by other instances, the "ReleasedNext" tool was used in previous versions. This tool has now been replaced with a revised, integrated dialogue window that opens after starting HELiOS (or HiCAD) if such objects have been found.

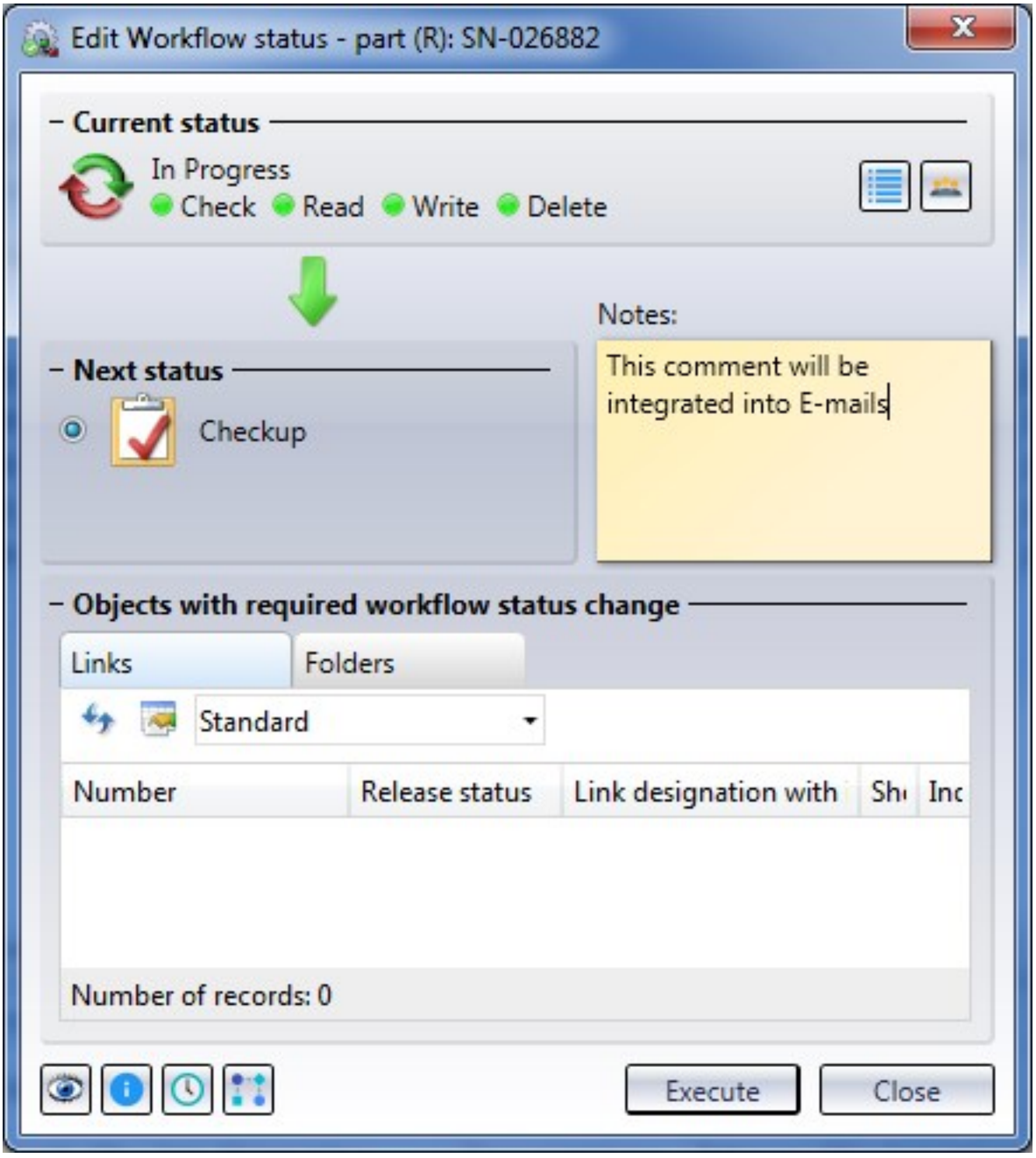
The dialogue window can also be called by choosing**Admin Tools > ReleaseNext** (Administrator rights required).



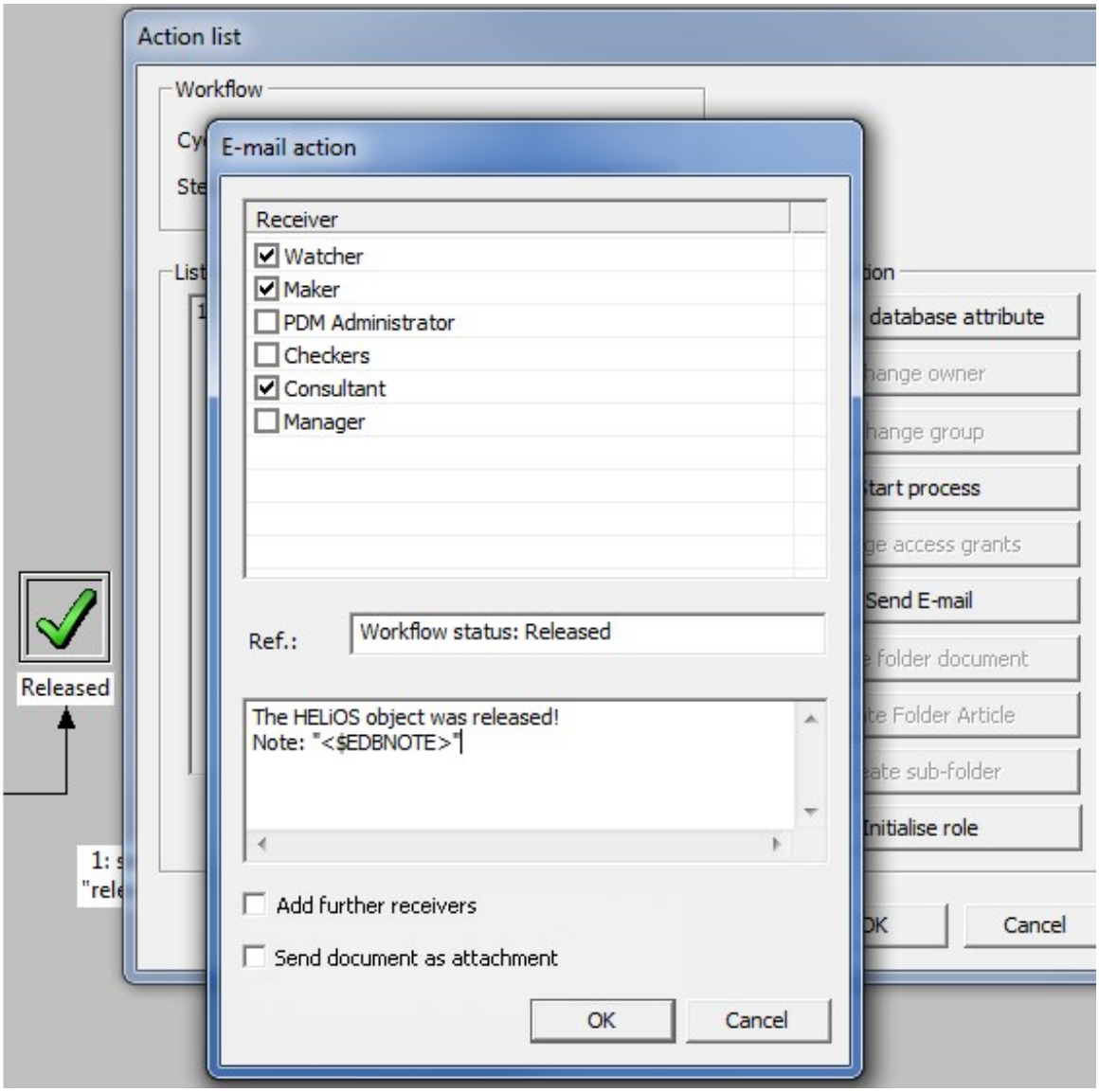
Workflow - Send E-mails with notes

In the action list for Role Workflow steps you can now determine that in the case of certain Workflow steps (e.g. Request checkup or Release), E-mails are automatically sent to relevant users.

The note commenting on a Workflow status can be sent together with the E-mail in the process.



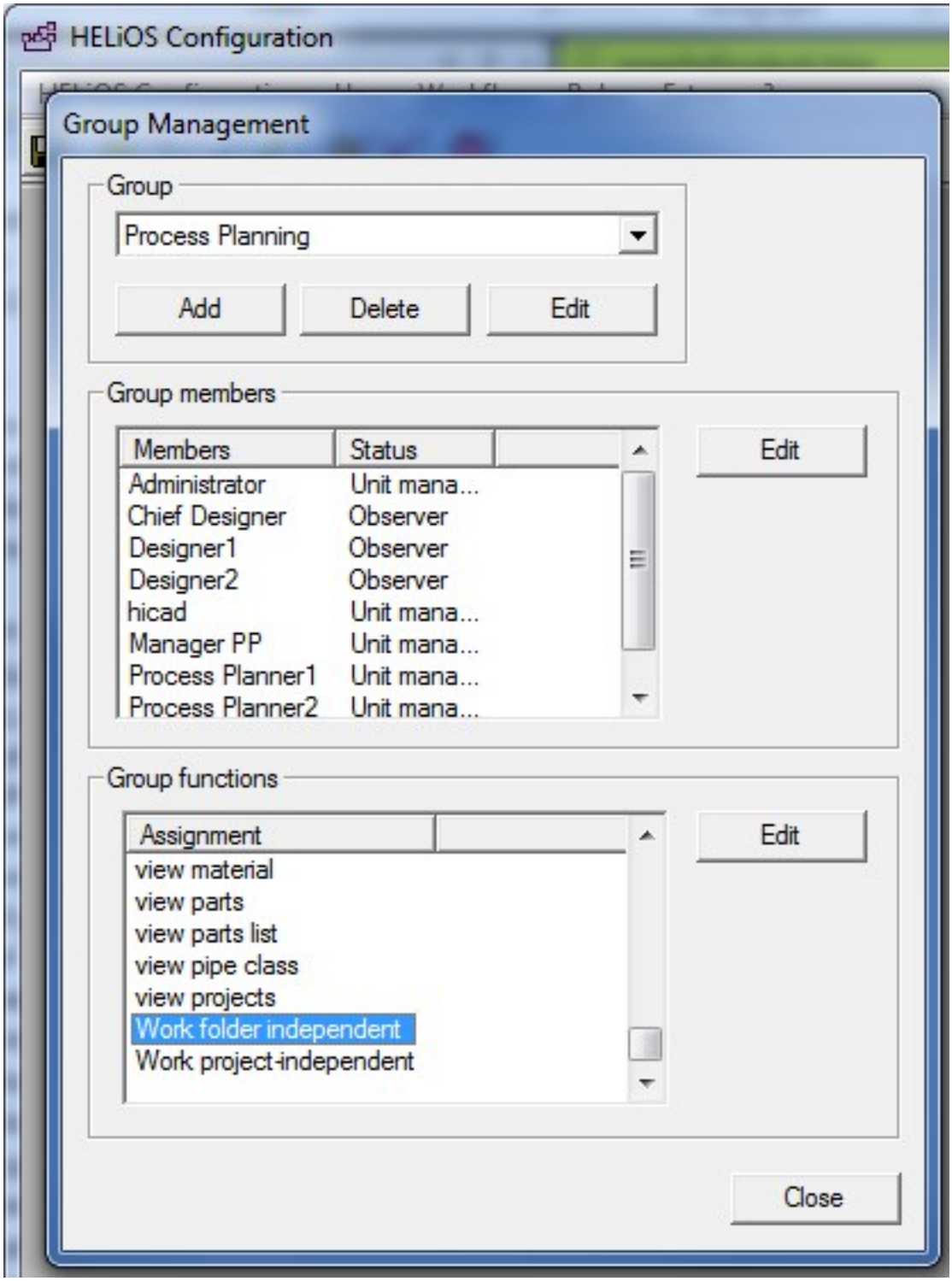
In the EDBSETUP action list of the corresponding Workflow step, enter the placeholder **<$EDBNOTE>** for the into the comment field of the **E-mail action** dialogue window that will open when you click on the Send E-mail button. The note will then appear in the E-mail.



Option for mandatory project/folder selection

In some cases it makes sense to restrict access to particular projects and/or folders of the HELiOS database for particular users or user groups - e.g. external suppliers that have been integrated into the system.

This can be achieved by deactivating the new permissions function**Work folder-independent** or **Work project-independent** in the Management of the **EDBSETUP** for particular users.



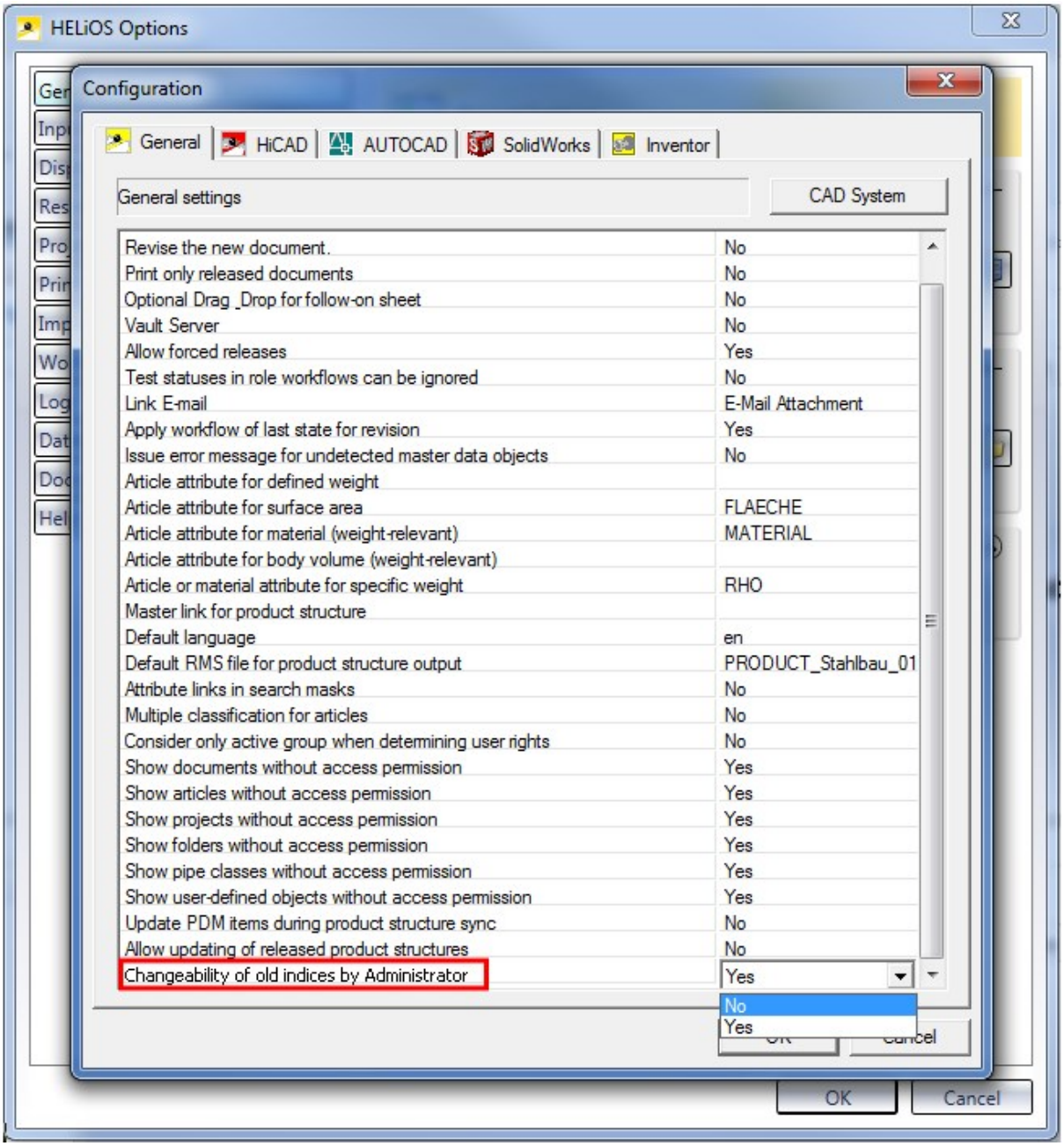
Users who do not have that permission will not have the possibility of project-independent or folder-independent views, searches or inputs any more , since the corresponding buttons or selection options do no longer exist in the HELiOS user interface.

Please note:

Already existing users and user groups will automatically receive the new permission after an update of the existing database.

Changeability of old indices by Administrator

The new database option (**HELiOS Options > Database > General > Changeability of old indices by Administrator** enables Adminsistrators to edit old indices. This concerns both meta data and the relevant files, and will also take effect when saving documents in interfaced CAD systems and importing files in the HELiOS Desktop.

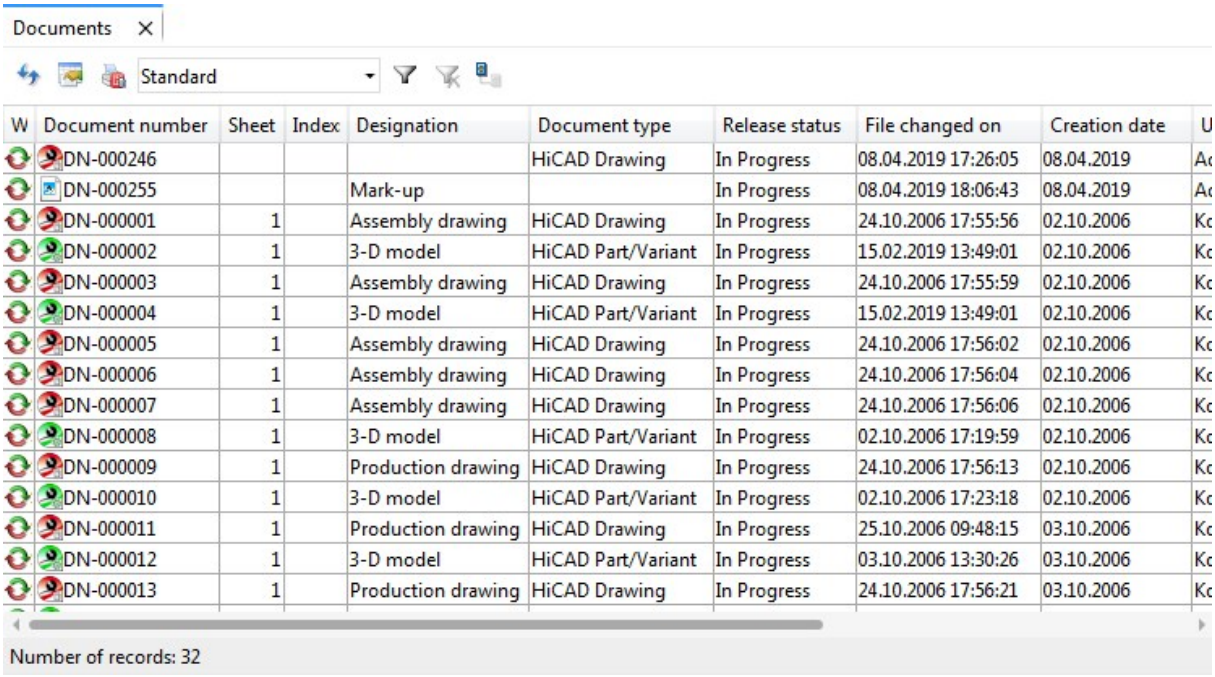


HELiOS in HiCAD

Service Pack 2 2019 (V 2402)

New icons for HiCAD file formats

From SP2 onwards, new icons for the different HiCAD file formats e.g. SZA, FGA, KRA, VAA, PAA are available. These icons will be shown in the HELiOS result lists.



	File format	File type
	.SZA	HiCAD model drawing
	.KRA	HiCAD 3-D part
	.FGA	HiCAD 2-D part
	.VAA	HiCAD variant
	.PAA	HiCAD part archive
	.RPA	HiCAD P+ID

Service Pack 1 2019 (V 2401)

Article master sync when saving

The **Article master sync when saving**  that you use to specify which HiCAD part attributes are to be synchronized with which HELiOS article master data has been moved from the **Drawing** tab to **HELiOS PDM** at **Others > Link > ...**.

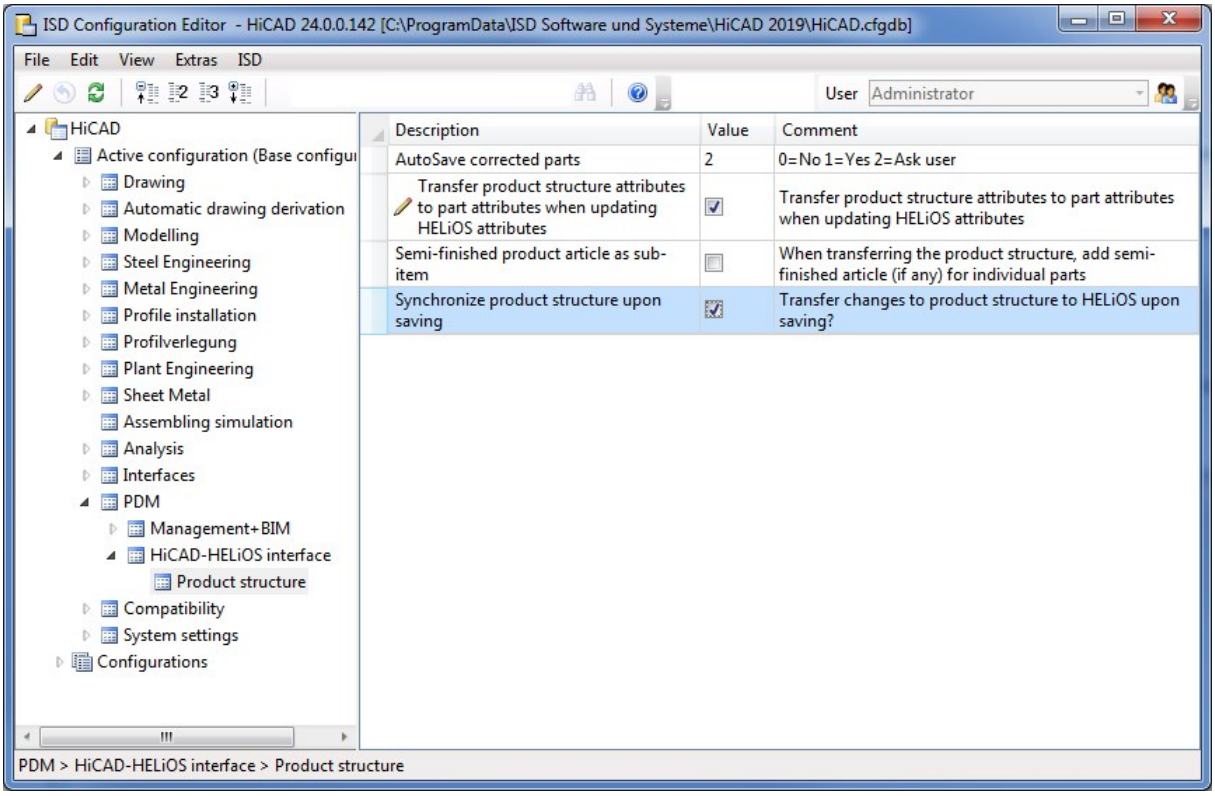
Major Release 2019 (V 2400)

Automatic transfer of product structure and article master attributes upon saving

In the **Configuration Editor** you can specify whether 1.) Changes to the product structure and 2.) HiCAD part attributes in the HELiOS article master are to be synchronized upon each saving in HiCAD.

1.) Auto-sync product structure upon saving:

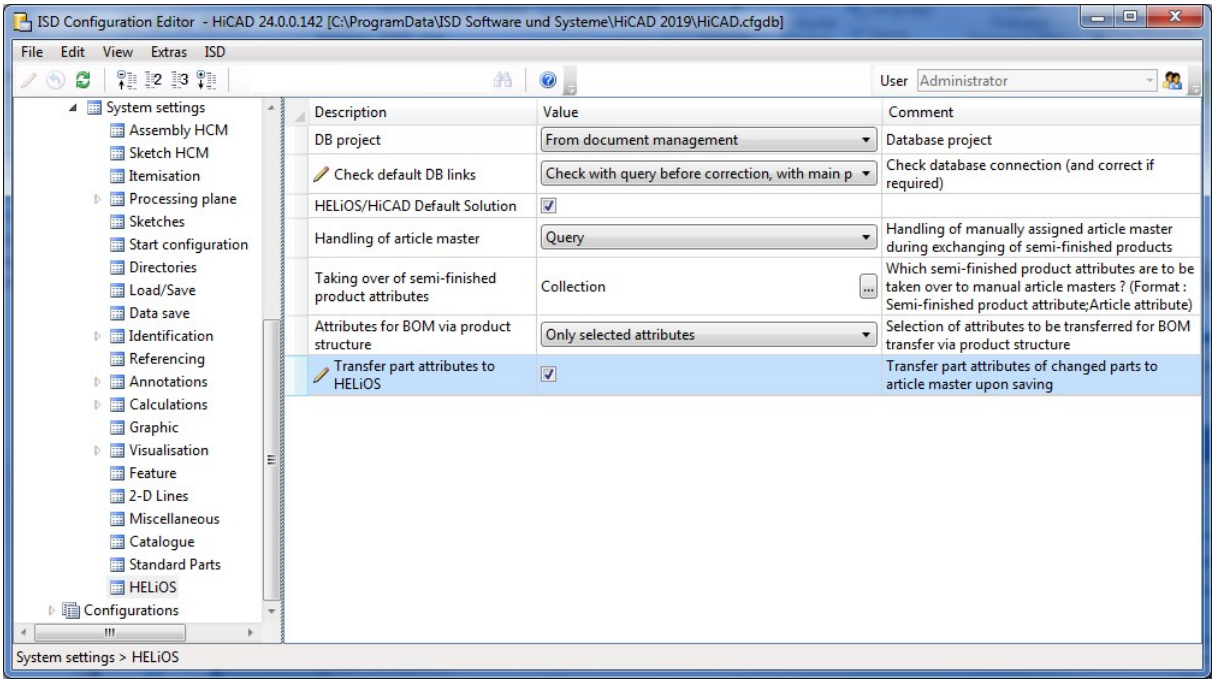
Go to **PDM > HiCAD-HELiOS interface > Product structure** and activate the **Synchronize product structure upon saving** checkbox:



After this, any changes to the product structure will be automatically transferred to HELiOS after each saving in HiCAD. This spares you the additional step of a manual transfer.

2.) Auto-sync part attributes and article master data:

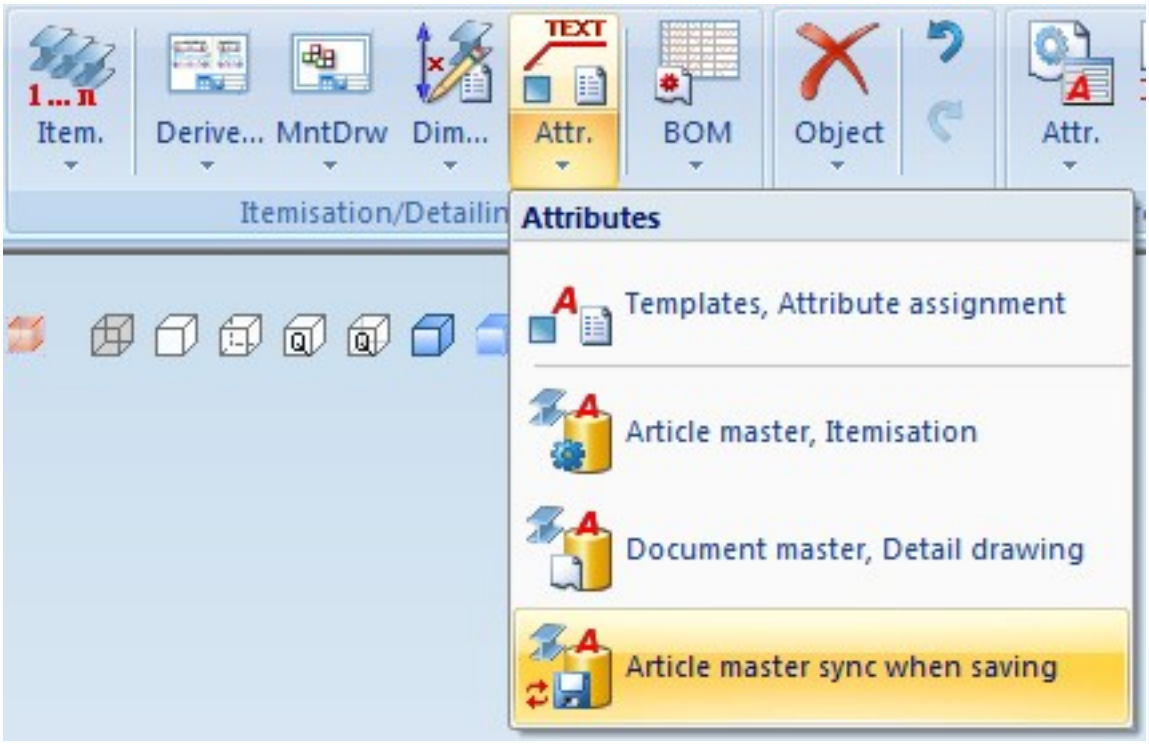
In the Configuration Editor, go to **System settings > HELiOS** and activate the **Transfer part attributes to HELiOS**:

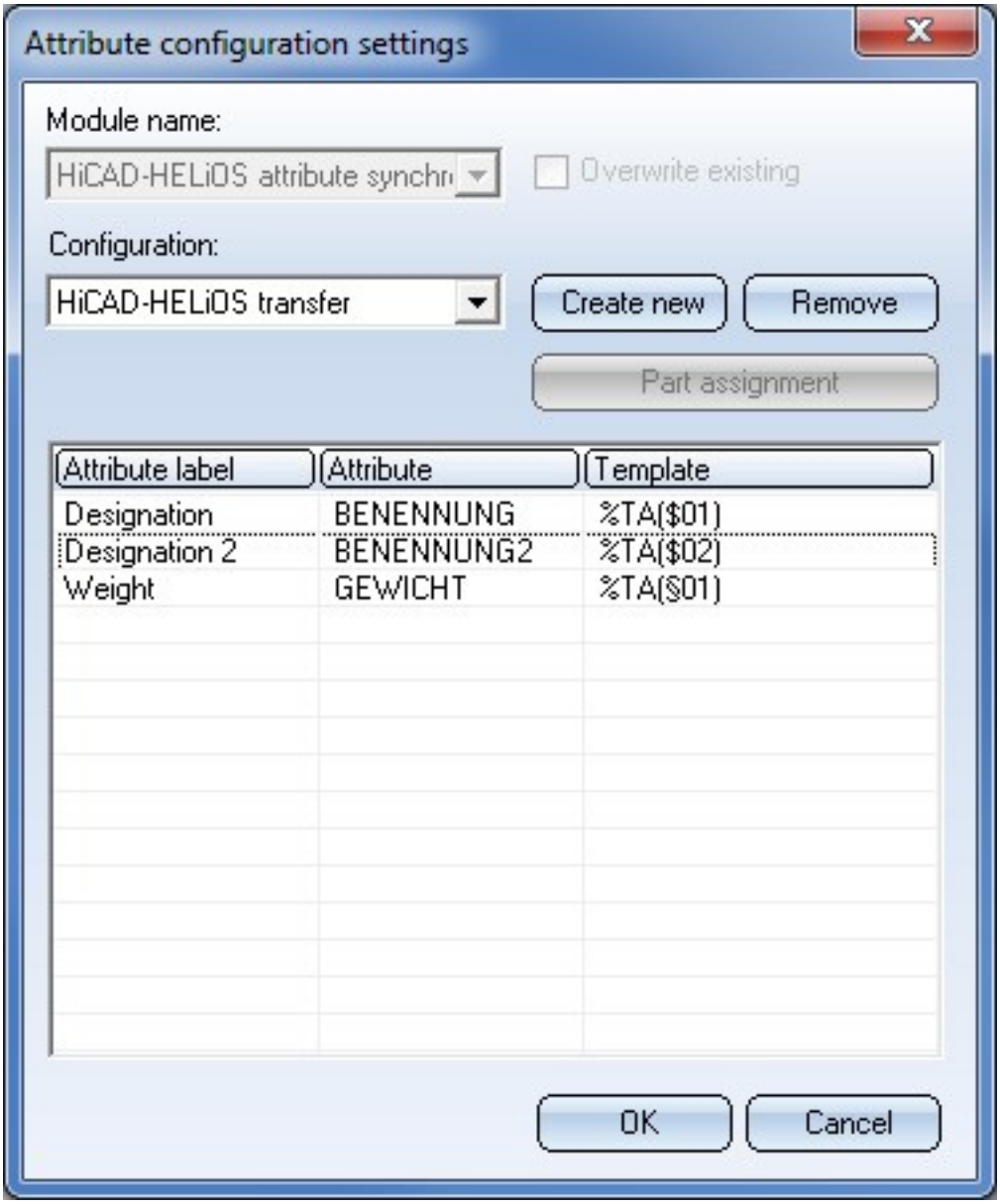


Changes to article master-relevant part attributes, will then be automatically taken over to HELiOS upon each saving in HiCAD.

Please note:

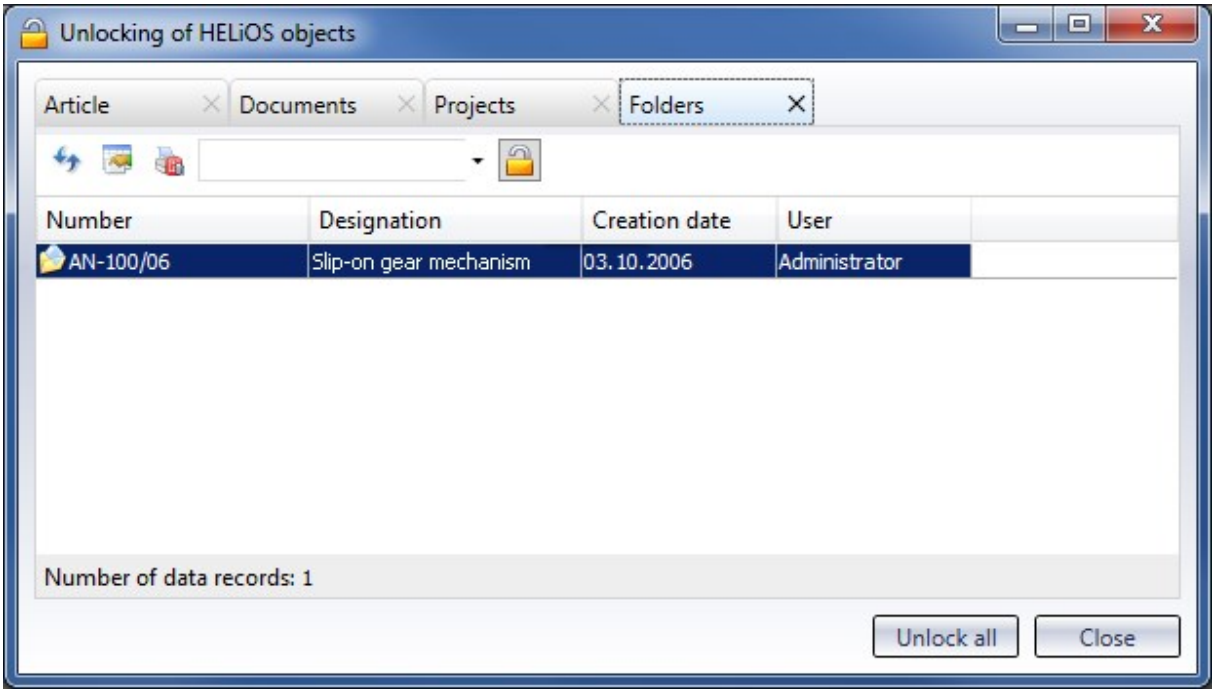
Choose **Drawing > Itemisation/Detailing > Attr. > Article master sync when saving** to specify which part attributes are to be synchronized with which article master data.


Item.
Derive...
MntDrw
Dim...
Attr.
BOM
Object
Attr.
Itemisation/Detailin
Attributes
Templates, Attribute assignment
Article master, Itemisation
Document master, Detail drawing
Article master sync when saving



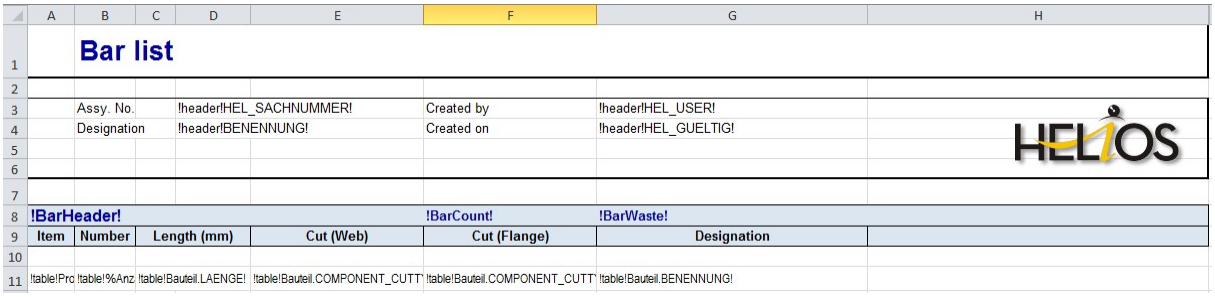
New dialogue for unlocking of HELiOS objects

For unlocking of objects that are currently edited by other instances, the "ReleasedNext" tool was used in previous versions. This tool has now been replaced with a revised, integrated dialogue window that opens after starting HELiOS (or HiCAD) if such objects have been found.


Unlocking of HELiOS objects
Article
Documents
Projects
Folders
Number
Designation
Creation date
User
AN-100/06
Slip-on gear mechanism
03.10.2006
Administrator
Number of data records: 1
Unlock all
Close

Excel BOM with bar optimisation

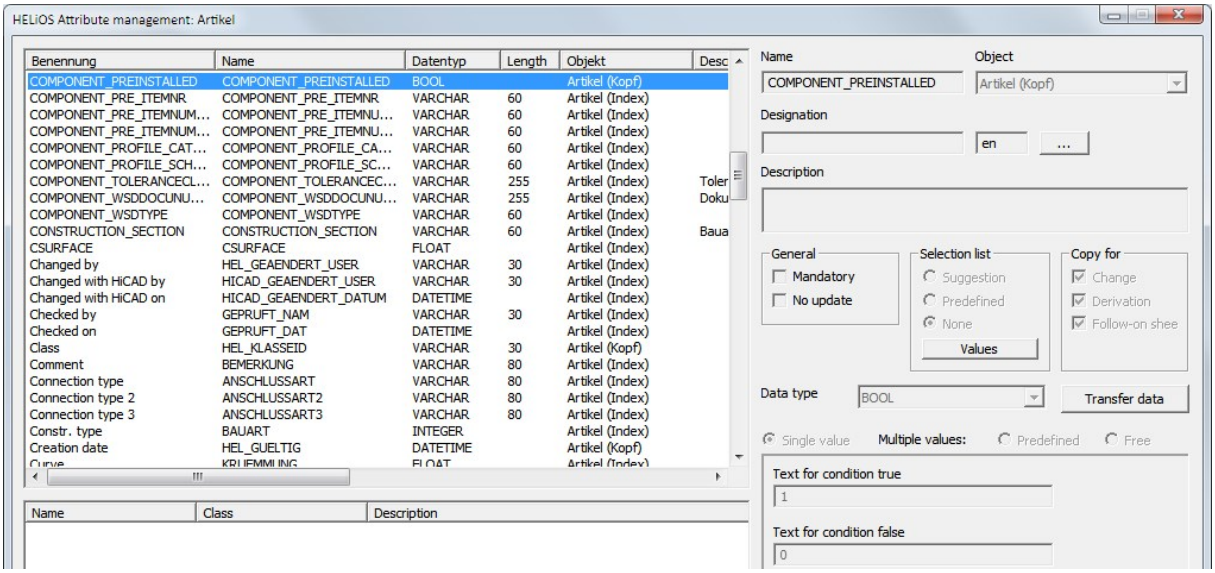
Similar to the Excel BOM output from HiCAD, a new BOM template called **PRODUCT_ISD_HiCAD** is now available, which allows the output of bar optimisations for BOM exports via product structure.



	A	B	C	D	E	F	G	H
1		**Bar list**						
2								
3		Assy. No.		!header!HEL_SACHNUMMER!		Created by	!header!HEL_USER!	
4		Designation		!header!BENENNUNG!		Created on	!header!HEL_GUELTIG!	
5								
6								
7								
8	**!BarHeader!**					**!BarCount!**	**!BarWaste!**	
9	**Item**	**Number**	**Length (mm)**	**Cut (Web)**		**Cut (Flange)**	**Designation**	
10								
11	!table!Pro	!table!%Anz	!table!Bauteil.LAENGE!	!table!Bauteil.COMPONENT_CUTT		!table!Bauteil.COMPONENT_CUTT	!table!Bauteil.BENENNUNG!	

Pre-mounted assemblies in BOMs

When working with pre-mounted assemblies in practice, you sometimes want only the assemblies, but not their sub-parts to be shown in BOMs. This can be achieved via the article attribute **COMPONENT_PREINSTALLED**, which needs to be assigned to the article master of the pre-mounted assembly.

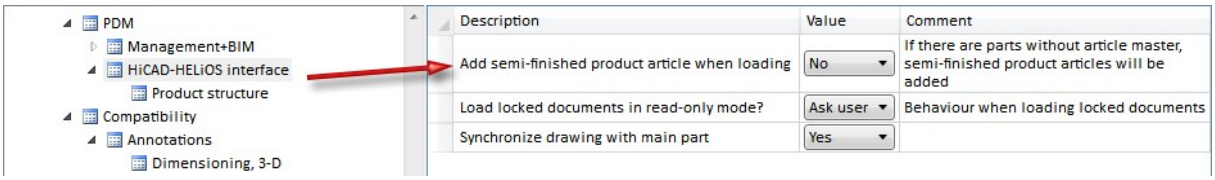


To ensure that the attribute will be shown in the HELiOS article data mask, you need to adjust the masks in HELiOS with the Mask Editor accordingly; that is, you need to add the **COMPONENT_PREINSTALLED** attribute as a new data field and, in so doing, create a new checkbox. If this checkbox is activated in the article master of an assembly, this assembly will be regarded as a "pre-mounted" assembly.

Loading of the current article master of semi-finished products in attached parts

In previous versions, existing article masters of semi-finished products were not transferred to HiCAD when inserting attached parts such as Steel Engineering connections, stairs, railings, element installations, profile installations etc.

From HiCAD 2019 onwards this behaviour can be changed via a setting in the Configuration Editor: At **PDM > HiCAD-HELiOS Interface** you can find the parameter **Add semi-finished product article when loading**. The ISD default setting is **No**, i.e. the article master of semi-finished products will not be transferred.



For example, when transferring article masters of semi-finished products from user-defined catalogues, set the parameter to **Yes**.

HELiOS Spooler

Service Pack 2 (V 2402)

Model area of AutoCAD drawings

When printing AutoCAD drawings (older versions) the model area of the drawing can be printed automatically if the layout area is empty.

You can configure this behaviour via the configuration file **AddIn.AutoCad.Config.xml** (this file will be automatically updated to the new configuration format by the Spooler).

Please also read the information given in the **Notes on File Conversion via the Spooler** topic.

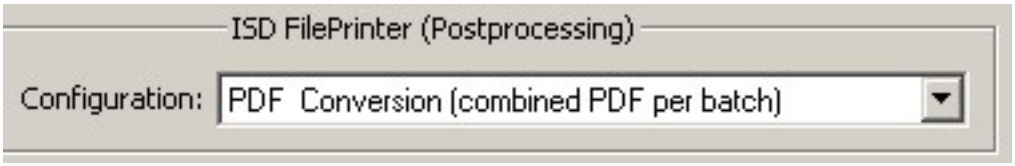
Configurable restart of CAD application

In previous versions, the appropriate CAD applications were restarted for each print or conversion job. This behaviour can now be configured, resulting in shorter processing times and a faster printing of the desired printout or file.

Service Pack 1 2019 (V 2401)

Combining of different PDFs into one document

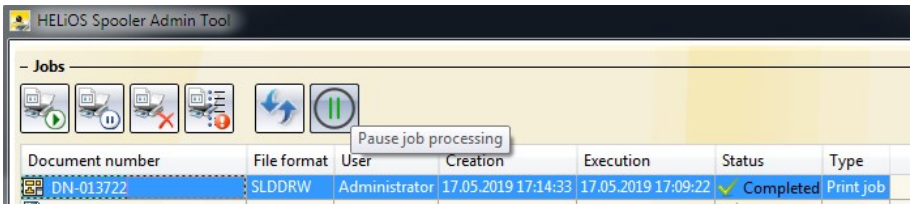
The HELiOS Print Client now contains a new option for the **Postprocessing** of print jobs, which allows you to combine individual PDF pages into one document during automated print job processing.



Pausing the Spooler

You can now pause and continue the Spooler, either interactively via the Admin Tool or via command prompt. New jobs will also be accepted while the Spooler is pausing (however they will not be processed during that period - their processing starts as soon as the Spooler is continued again).

The control via command prompt can be used for a time-controlled pausing and continuation of the Spooler, e.g. via Windows task planning.



Major Release 2019 (V 2400)

Inventor: Export of image formats and optional database connection

Image formats such as JPEG or BMP can now also be (automatically) output from Autodesk Inventor via the HELiOS Spooler by means of suitable conversion jobs.

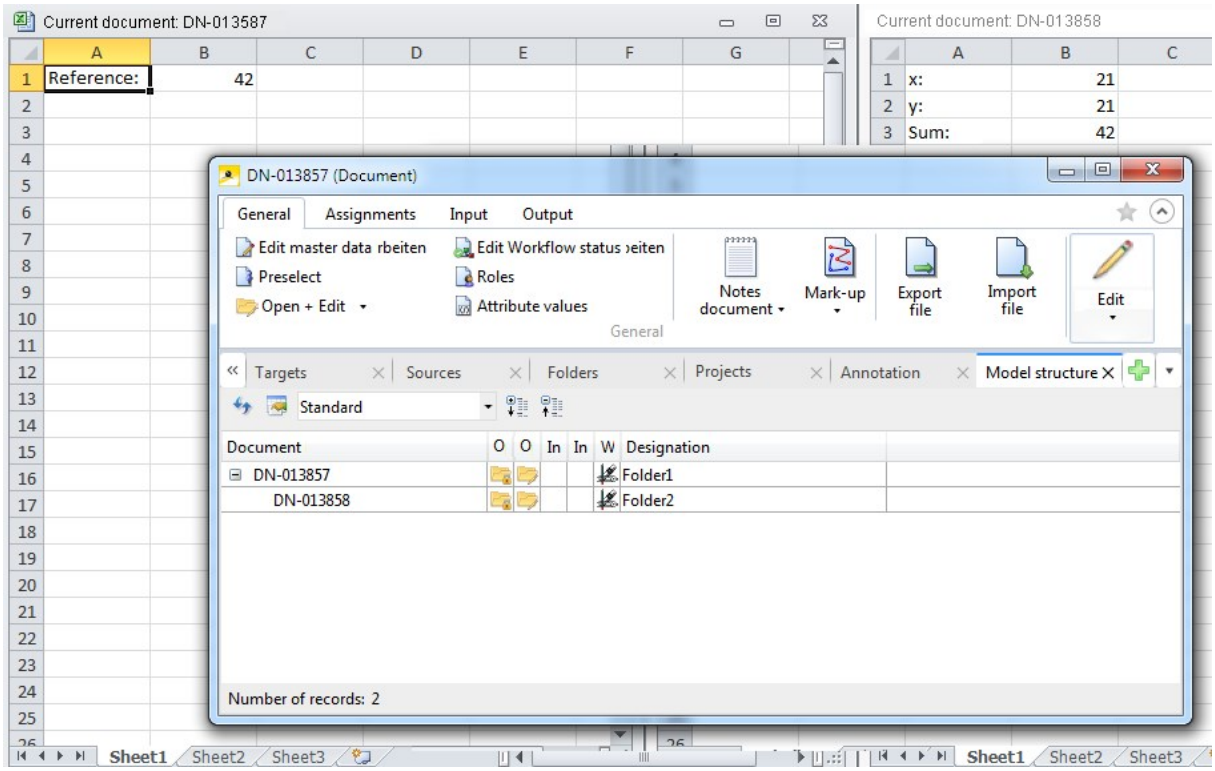
To be able to use the new attribute mapping for the filling in of title blocks in Inventor during the output via Spooler, you have also the option to configure a database connection for the HELiOS Spooler.

HELiOS MS Office Interface

Service Pack 2 (V 2402)

Model structure: Display of document structure

If only a document structure instead of a model structure is available for a document, the HELiOS-MS Office /Excel) interface displays the document structure, according to the document master data, on the **Model structure** tab.



Documents in the Workspace

The HELiOS Workspace window is now available in both the HELiOS Desktop and in the CAD or MS Office interfaces.

Irrespective of all filter options you can apply, the following rules apply for the display behaviour:

All documents that have been opened/loaded in the active application will be displayed, as well as all non-loaded documents that have been saved to the application with which the documents were originally created.

This means: As soon as a document is opened in an interfaced application, the corresponding Workspace will memorize this until the document will be removed from the Workspace again. The corresponding document will therefore be constantly displayed in the Workspace, also after closing and restarting the application.

When opening a particular document in two different applications (e.g. a DWG file in AutoCAD and Inventor), the HELiOS Workspace memorizes both applications and displays the file in the Workspaces of both applications.

Also, the following rule applies: For an Inventor assembly that references an Excel document, both the Inventor documents and the Excel file will be displayed in the Workspace after opening the assembly in Inventor. If the Excel file has previously been saved in the Office application (setting the document attribute "Related application" (HEL_APPLICATION) accordingly), this file will be additionally visible in the Workspace of Excel, even if it has not been explicitly loaded in this application by the user.

An exception to this rule is the Workspace of the HELiOS Desktop that always displays all documents.

Service Pack 1 2019 (V 2400)

Office 2019 and support of multiple versions

HELiOS 2019 SP1 supports offers an interface to MS Office 2019.

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