



# HICAD

UNLIMITED CAD PERFORMANCE DEVELOPED BY ISD

## What's new?

Version 2018

News Overview

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THE WORLD OF CAD AND PDM SOLUTIONS



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## Basics

### 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.

## Service Pack 2 2018 (V 2302)

### Installation / Start of HiCAD

#### The ISD Update Service

The ISD Update Service supports the automatic search for and installation of HiCAD updates (Hotfixes). For this you require the program ISDUpdateSvc.exe, that you can either activate during installation of HiCAD or install via the Windows Control Panel afterwards. Via this program you can activate the ISD Update Service. Further information can be found in the **Automatic Installation** chapter of the **Installation Notes** manual.

If the ISD Update Service is active, HiCAD checks upon its start whether a Hotfix is available. If this is the case, the following message appears at the bottom right of your screen:



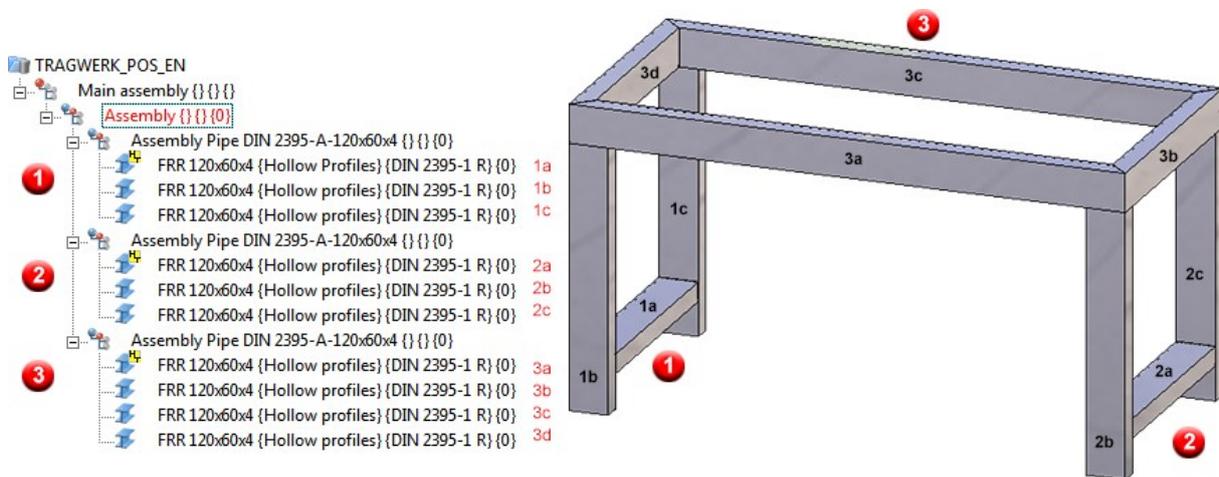
## Itemisation

### New part attribute: Item index %PIDX

Not only item numbers but also itemisation texts which consist of free text and attributes from different HiCAD/HELIOS sources. The new HiCAD part attribute **Item index %PIDX** can be used in order to create own incrementation in itemisation texts. If the itemisation text contains this attribute, each part with equal item number will be incremented on its own. This means that all n parts with common itemisation number will be numbered consecutively with an index from 1 to n. In practice, this is often used to allocate assemblies or parts with equal item number clearly to each information of the item number flag or BOM.

#### An example:

The depicted Steel Engineering drawing consists of three assemblies, which in turn consist of equal hollow profiles. The assemblies 1 and 2 are equal and consist of the parts 1b, 1c, 2b and 2c. Assembly 3 consists of four profiles, part of which are equal to 3a and 3c as well as 3b and 3d.



The drawing should be itemised. In order to set i.a. the new attribute **Item index** itemisation texts should be created automatically. For this, the function **Drawing > Itemisation > Ite... > ... with Options** will be used with the following settings:

- Load the default settings for Steel Engineering via **Favorites**
- Tab **General**: Activate the checkbox **Generate itemisation text**
- Tab **Part groups**: Definition of the itemisation text for assemblies and Steel Engineering parts as follows:

**Assemblies**

{Part type} - {Item No.} - {Item index}

Data source is the part itself.

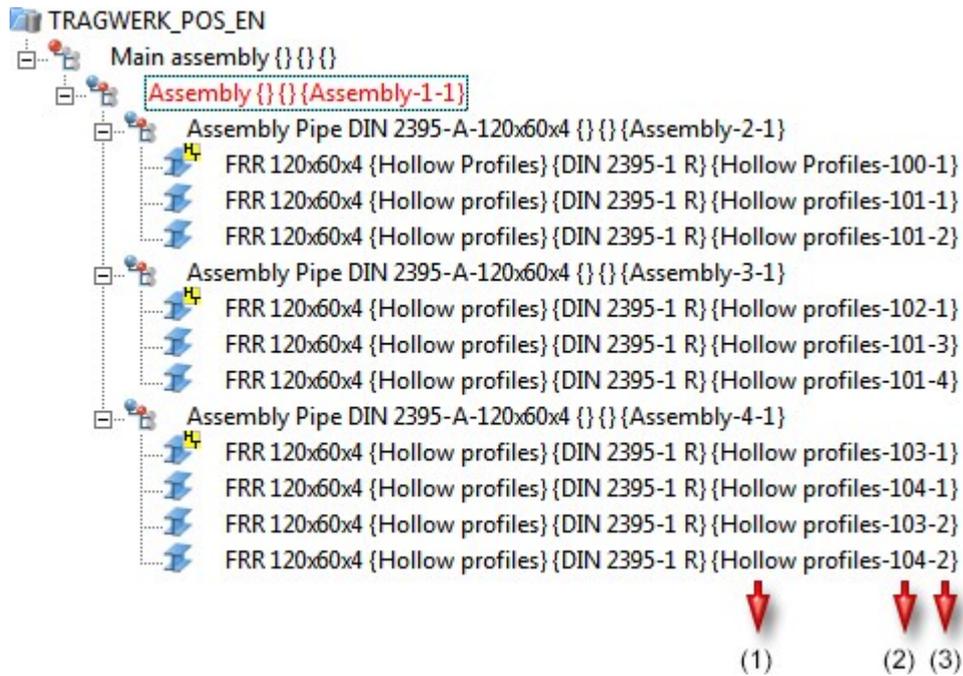
**Steel/Metal Engineering Parts**

{Part type} - {Item No.} - {Item index}

Data source is the part itself.

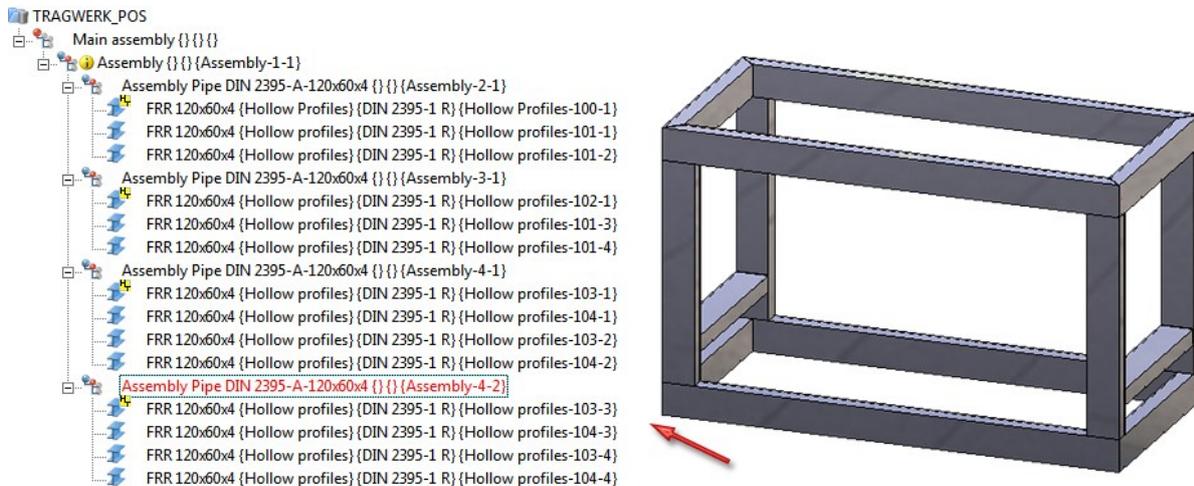
- Start itemisation.

Result



1=Part type, (2) Item number, (3) Item index

If a copy of assembly 3 would be inserted into the drawing and then itemised again, the item index continued with the next number (see example 3)



The itemisation text can also be issued via BOM.

Example - Structure list

Level	Item	Item text	Material	Qty.	Designation	Designation	W[mm]	L[mm]	DSTV part...	Total weight	Surf....
1	1	Assembly-1-1		1	Assembly		0.0	0.0		153.0	0.0
2	2	Assembly-2-1		1	Assembly Pipe DIN 2395-A-...		819.0	790.0		24.2	0.0
3	100	Hollow Profiles-100-1	S235JRH	1	FRR 120x60x4	Hollow Profiles	0.0	670.0	H	7.0	0.2
3	101	Hollow profiles-101-1	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	819.0	W	8.6	0.3
3	101	Hollow profiles-101-2	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	819.0	W	8.6	0.3
2	3	Assembly-3-1		1	Assembly Pipe DIN 2395-A-...		819.0	790.0		24.2	0.0
3	101	Hollow profiles-101-3	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	819.0	W	8.6	0.3
3	101	Hollow profiles-101-4	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	819.0	W	8.6	0.3
3	102	Hollow profiles-102-1	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	670.0	H	7.0	0.2
2	4	Assembly-4-1		1	Assembly Pipe DIN 2395-A-...		790.0	1700.0		52.3	0.0
3	103	Hollow profiles-103-1	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	1700.0	H	17.8	0.6
3	103	Hollow profiles-103-2	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	1700.0	W	17.8	0.6
3	104	Hollow profiles-104-1	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	790.0	W	8.3	0.3
3	104	Hollow profiles-104-2	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	790.0	W	8.3	0.3
2	4	Assembly-4-2		1	Assembly Pipe DIN 2395-A-...		790.0	1700.0		52.3	0.0
3	103	Hollow profiles-103-3	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	1700.0	H	17.8	0.6
3	103	Hollow profiles-103-4	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	1700.0	W	17.8	0.6
3	104	Hollow profiles-104-3	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	790.0	W	8.3	0.3
3	104	Hollow profiles-104-4	S235JRH	1	FRR 120x60x4	Hollow profiles	0.0	790.0	W	8.3	0.3
Σ										153.0	5.0



**Please note:**

When allocating the item index, it is not guaranteed whether already allocated indices still remain.

A simple example:

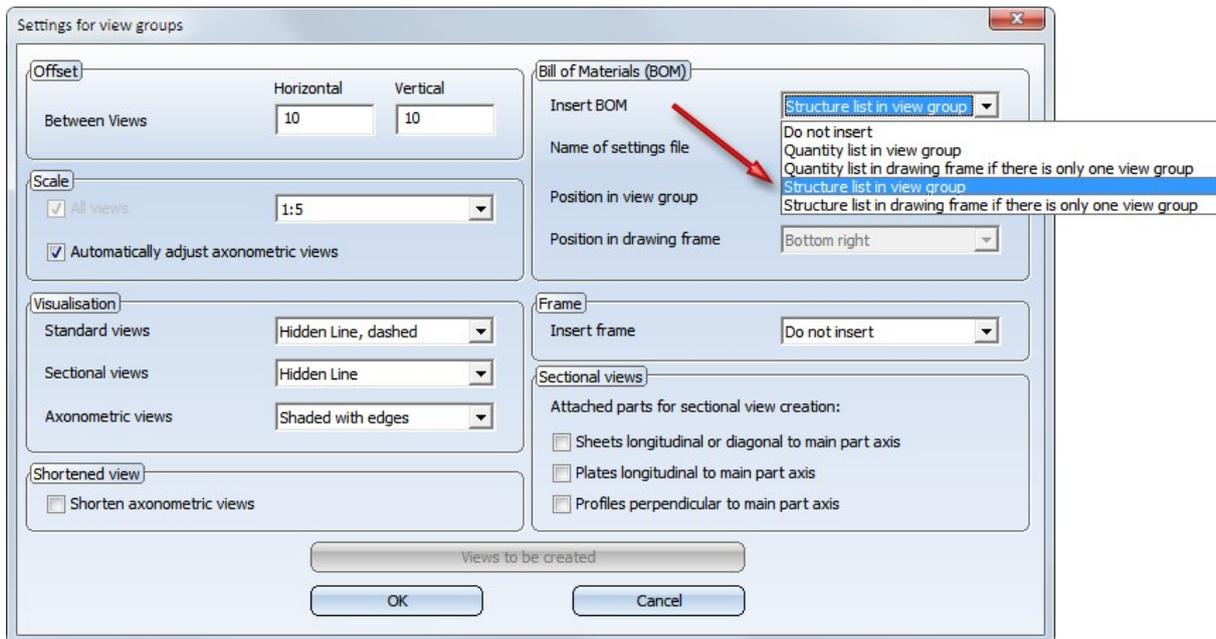
Let it be supposed that the original drawing contains three equal parts with position indices 1, 2 and 3. Now, the part with item index 2 will be deleted. The parts with index 1 and 3 remain. If the drawing is itemised automatically, two possible options can result:

- The part with former index 1 maintains it; the part with former index 3 retains index 2.
- The part with former index 1 retains index 2; the part with former index 3 retains index 1.

## Drawing Derivation

### Structure Lists

As of HiCAD 2018, besides Quantity Lists also Structure Lists can be added in a Drawing Derivation. For this, the dialogue window **Settings for view groups** and the settings in the configuration management via **Automatic drawing derivation > Production drawing > Usage-dependent > NAME > View group** (here, the NAME is the name of the respective intended use) have been enhanced.



### New / changed Dimensioning Rules for Sheet Metal

New dimensioning rules are available for sheet metal:

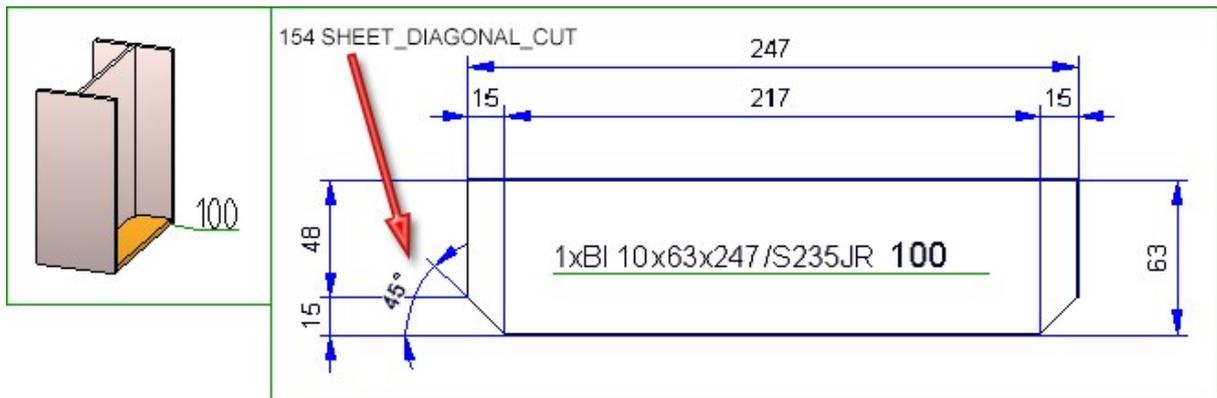
- **154 SHEET\_DIAGONAL\_CUT**

applies dimension to diagonal sheet/plate cuts

*Recommended settings Rule 154*

⬆️ **154: Diagonal sheet/plate cuts**

ID:	154: Diagonal sheet/plate cuts	All
Type of dimension:	Angular dimensions	
Position of dimension chain:	Top	
Dimension chain reference:	X-axis	
Direction of dimension chain:	Parallel to sheet/plate X-axis	



■ 155 SHEETMETAL\_FLANGE\_INSECT / 156 SHEETMETAL\_BENDINGANGLE\_INSECT

Rule 155 applies dimension to the flange length of sheets in sectional views, Rule 156 to the bend angle of sheets in sectional view. In case of flanges that are not parallel, the theoretical intersection point of the flanges will be dimensioned.

*Recommended settings Rule 155*

⤴ 155: Flange length of sheets in sectional views

ID:	155: Flange length of sheets in sectional views	All
Type of dimension:	Individual dimensions	
Position of dimension chain:	Automatic	
Dimension chain reference:	Direct, without reference	
Direction of dimension chain:	Parallel to sheet/plate X-axis	

⤴ 155: Flange length of sheets in sectional views

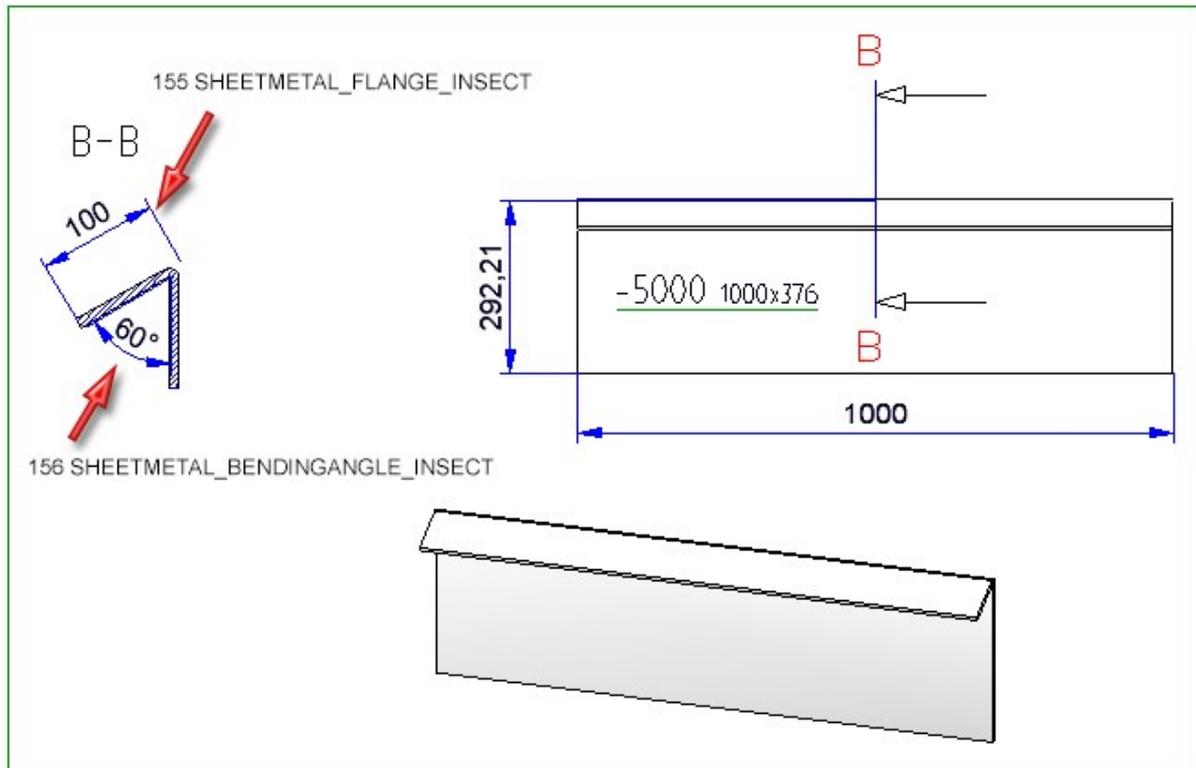
ID:	155: Flange length of sheets in sectional views	All
Type of dimension:	Individual dimensions	
Position of dimension chain:	Automatic	
Dimension chain reference:	Direct, without reference	
Direction of dimension chain:	Vertical to X-axis of sheet	

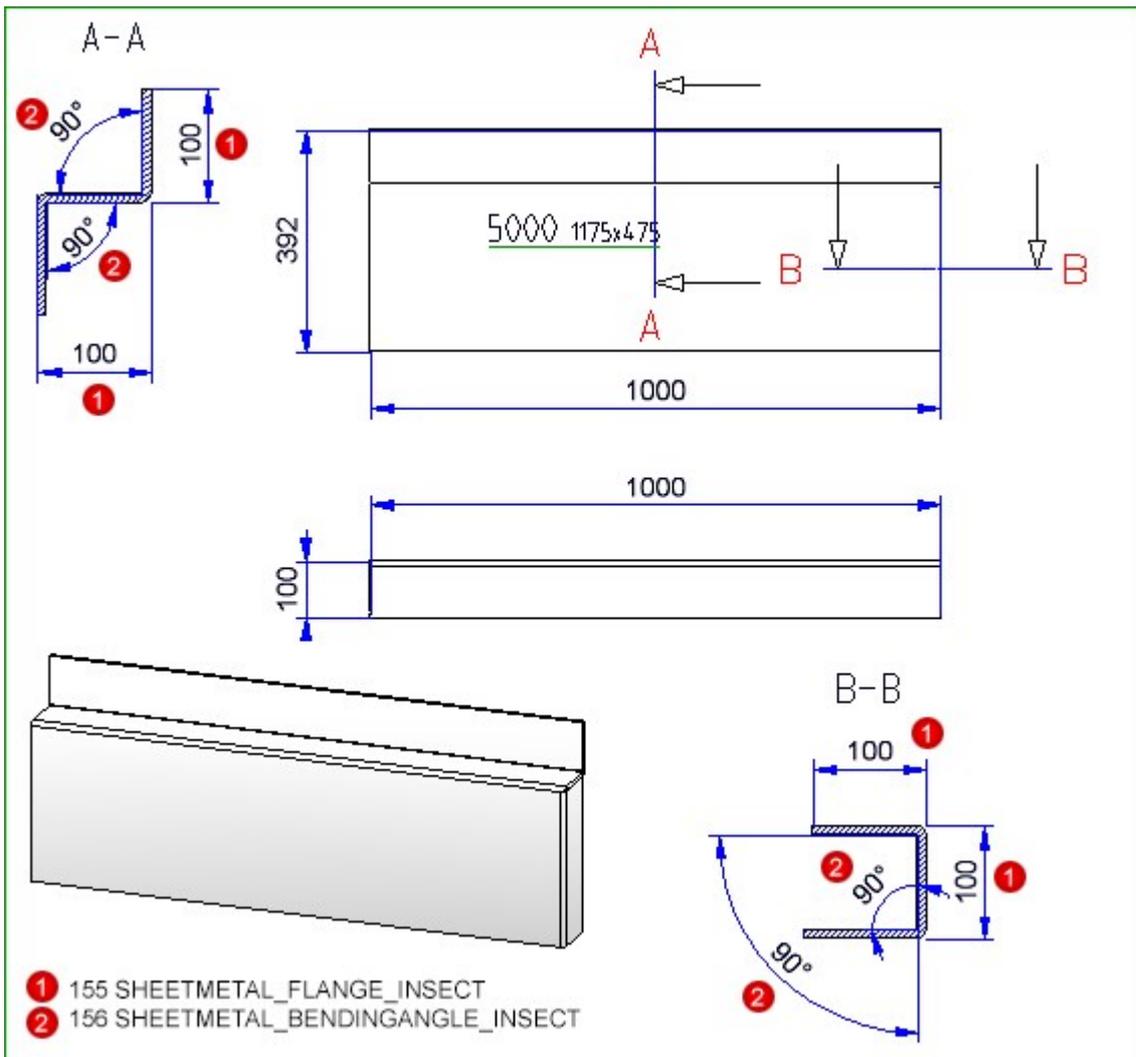
*Recommended settings Rule 156*

⤴ 156: Opening angles of bend zones of sheets

ID:	156: Opening angles of bend zones of sheets	All
Type of dimension:	Angular dimensions	
Position of dimension chain:	Deactivated	
Dimension chain reference:	Direct, without reference	
Direction of dimension chain:	Direction determined by 2 points	

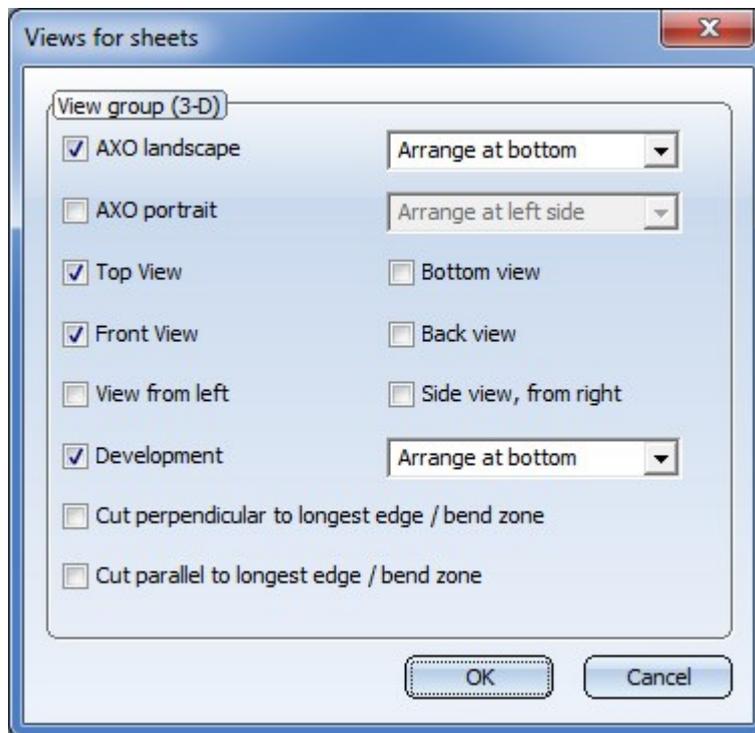
The recommended three settings for Rule 155 and 156 are simultaneously the new default settings for the sectional views of sheets in the workshop drawing (intended use: DEFAULT (SHEET METAL)).



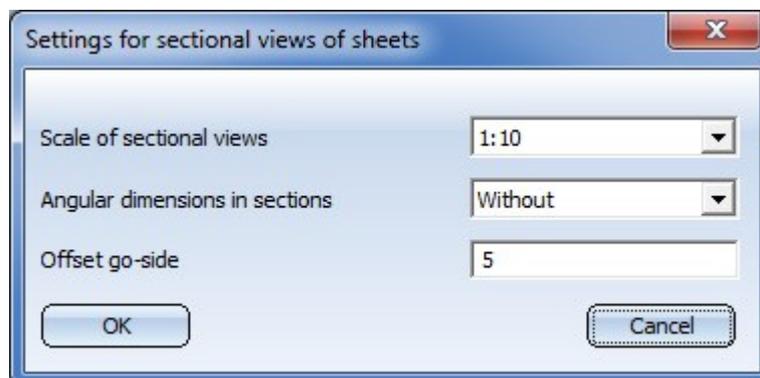


## Sectional views of sheets

As of HiCAD 2018 SP2, the Section for generating sectional views always takes places at the shortest bend zone. The sectional views are created with depth limitation and perpendicular to each bend zone.



In addition, the dialogue window Settings for sectional views of sheets has changed:



The following settings are possible:

**Scale of sectional views**

■ **Main parts / Subordinate parts**

Here you select whether and how to name Main parts/Subordinate parts in sectional views. Possible options: via this parameter you define if sections should be created either with or without angular dimensions.

- **Do not annotate**
- **Annotate, default**

The default template for Sheet metal will be used for annotation (PosNummer\_Kantblech.ftd).

- **Annotate, acc. to usage**

Annotation takes place according to the usage-specific template for Sheet metal (POSNUMMER\_VERWART...DEFAULT(KANTBLECHE).FTD). If such template does not exist, the annotation will be applied according to the default template.

■ **Boltings, Standard Parts, Weld seams**

Via the checkboxes below you can determine whether boltings, standard parts and weld seams should be annotated in sectional views or not.

The previous settings **Angular dimensions in sections** and **Offset go-side** are not available anymore. The representation of Angular dimensions can be defined via the new dimensioning rule 156 and the Offset go-side via the settings of the configuration management under **Drawing > Representation > Sectional views with coating lines**

*Example of a workshop drawing of a coated sheet metal - with sectional view and representation of coating*

Anzahl	Position	Benennung	Länge	Breite	Stärke	Winkel Ansicht	Winkel Draufsicht	Gewicht
		Projekt/Projektnummer	LOS					kg
Platinhalt						1:5:0		
1x Blechx129.634x50.000						THE WORLD OF CAD AND PDM SOLUTIONS		
in folgenden Diagrammen verwendet:						Menge: PLANART: Index Änderung Dat		
in folgenden Zeichnungsansichten enthalten:						Zeichnungsnummer: Index		
Maßstab: 1:1								

## Dimensioning 90° bend angle of sheet metal

As of SP2 it can be defined whether the dimensioning of 90° bend angle for sectional views of sheet metal can be defined in drawing derivation. The ISD default settings are that all bend angle are to be dimensioned.

In order to deactivate the dimensioning of 90° bend angle, currently the system files

- STAB3DPAR.DAT (new customers) or
- STW\_DIMSETTINGS.XML

must be adjusted manually.

### STAB3DPAR.DAT

Here, the setting is done via

Display right angle for opening angle of bend zones of sheet metal? 1:yes, 0:no

0

### STW\_DIMSETTINGS.XML

Here, deactivating the dimensioning is done via line

```
</PARAM><PARAM Name="SHOWRIGHTANGLEOFBENDZONE" Typ="INT" Value="0">
```

If this line is not available it must be added manually.

## Usage assignment for own part types

Usage assignments can now not only be defined for assembly and part types provided by the ISD, but also for freely definable part types.

This means that you now have the option to define own part types you can, for instance, use own drawing derivation configurations for Steel Engineering series beams that are based on 2-D cross-sections. For example, you could define a part type called **Series**, and a corresponding usage assignment called **Default(Series)**.

If you now create workshop drawings for model drawings with series beams and use the drawing parameters **From configuration**, the configuration **Default(Series)** will be used for all series beams with the part type **Series**.

## Changed dimensioning rules for railing segment

The automatic dimensioning for oblique railings has been improved. For this, dimensioning rules 1006 to 1009 for the stair railing segment have been changed:

- **Rule 1006: CLEARANCE\_POST**

If the clear distance between two posts is dimensioned as Individual dimension, the post thickness will not be considered. The dimensioning takes places in all fields of the railing segments. If the dimensions of the rule are not equal in all fields, only the dimension of the first field will be depicted.

*Recommended setting for Rule 1006*

⬆️ **1006: Clear distances between posts**

ID:	1006: Clear distances between posts	All
Type of dimension:	Individual dimensions	
Position of dimension chain:	Internal	
Dimension chain reference:	Direct, without reference	
Direction of dimension chain:	In X-direction of railing post	

In dimension type **Chain of dimensions** the post thickness will be taken into consideration, too. If the chain of dimensions of all fields is equal, only the chain of dimensions of the first field will be depicted.

■ **Rule 1007: CENTERDISTANCE\_POST**

The axis distance of two posts is dimensioned in all fields of the railing segment. If the dimensions of this rule are equal in each field, only the dimension of the first field will be depicted.

*Recommended setting for Rule 1007*

⬆ **1007: Distances between post axes**

ID:	1007: Distances between post axes	All
Type of dimension:	Chain dimensions	
Position of dimension chain:	Internal	
Dimension chain reference:	Direct, without reference	
Direction of dimension chain:	In X-direction of railing post	

■ **Regel 1008: CLEARANCE\_WEBMEMBER**

If the clear distance between two posts is dimensioned as **Individual dimension**, the post thickness will not be considered. The dimensioning takes places in all fields of the railing segments. If the dimensions of the rule are not equal in all fields, only the dimension of the first field will be depicted.

*Recommended setting for Rule 1008*

⬆ **1008: Clear distances between rods**

ID:	1008: Clear distances between rods	All
Type of dimension:	Individual dimensions	
Position of dimension chain:	Internal	
Dimension chain reference:	Direct, without reference	
Direction of dimension chain:	In X-direction of railing post	

In dimension type **Chain of dimensions** the post thickness will be taken into consideration, too. If the chain of dimensions of all fields is equal, only the chain of dimensions of the first field will be depicted.

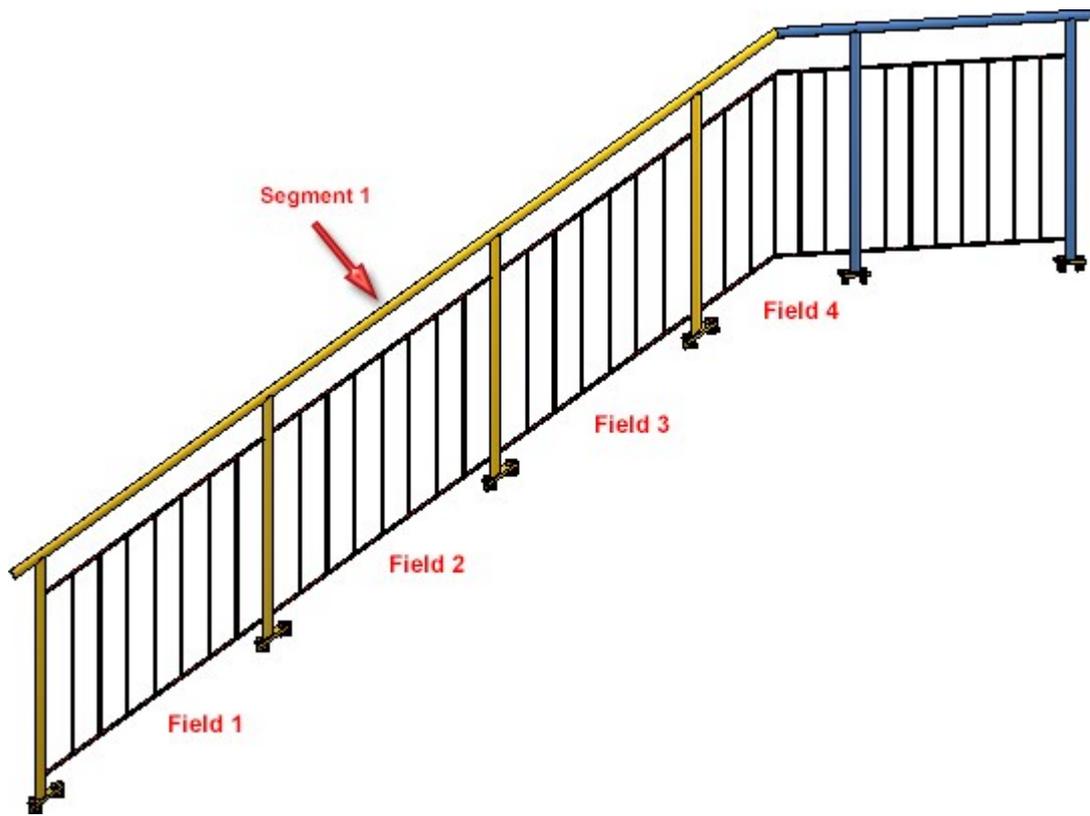
■ **Rule 1009: CENTERDISTANCE\_POS**

This rule dimensions the axis distance between posts and vertical rods as chain of dimensions. The dimensioning takes place in all fields of the railing segment. If the dimensions of the rule are not equal in all fields, only the dimension of the first field will be created.

*Recommended setting for Rule 1009*

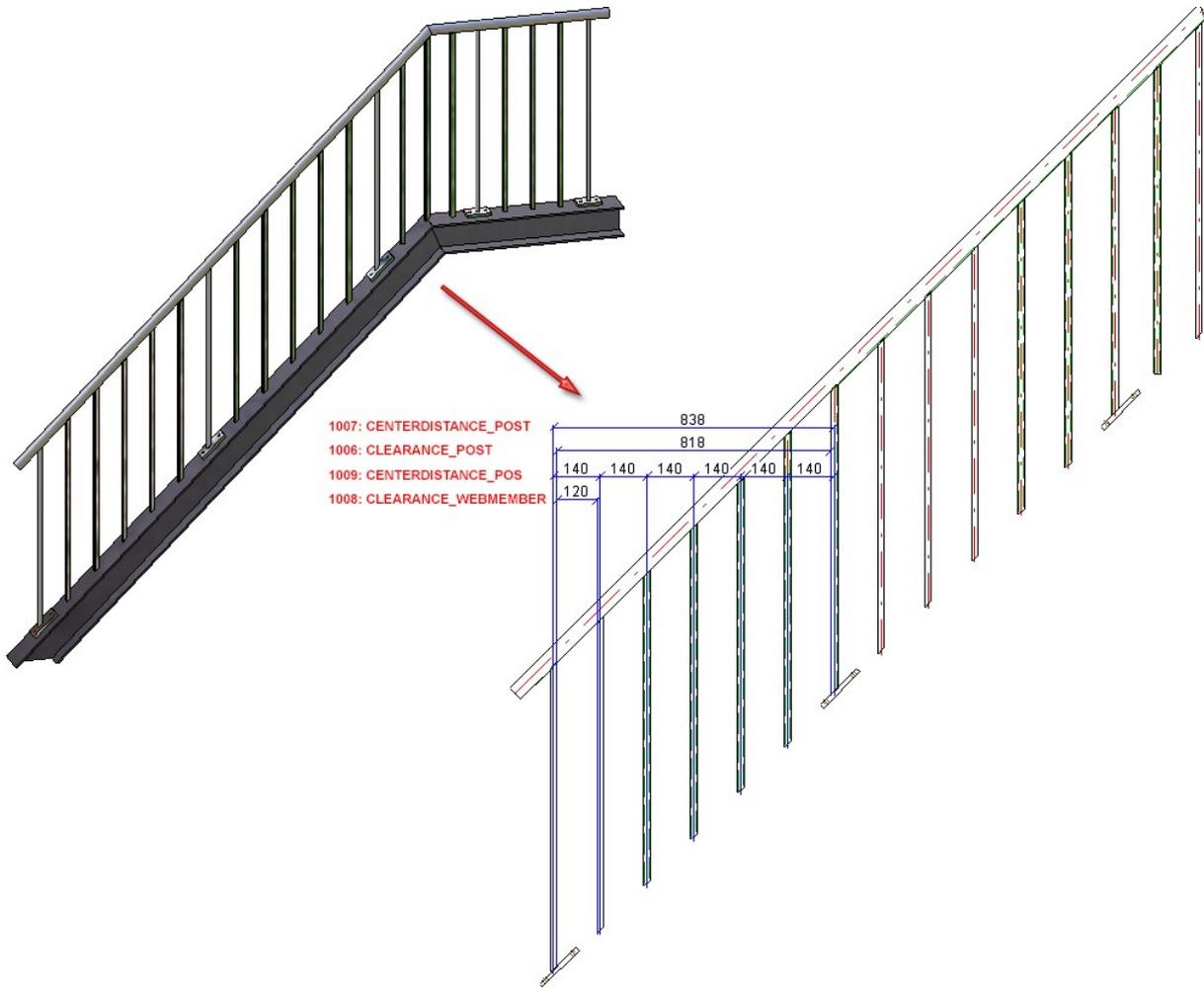
⬆ **1009: Distances between rod axes**

ID:	1009: Distances between rod axes	All
Type of dimension:	Chain dimensions	
Position of dimension chain:	Internal	
Dimension chain reference:	Direct, without reference	
Direction of dimension chain:	In X-direction of railing post	



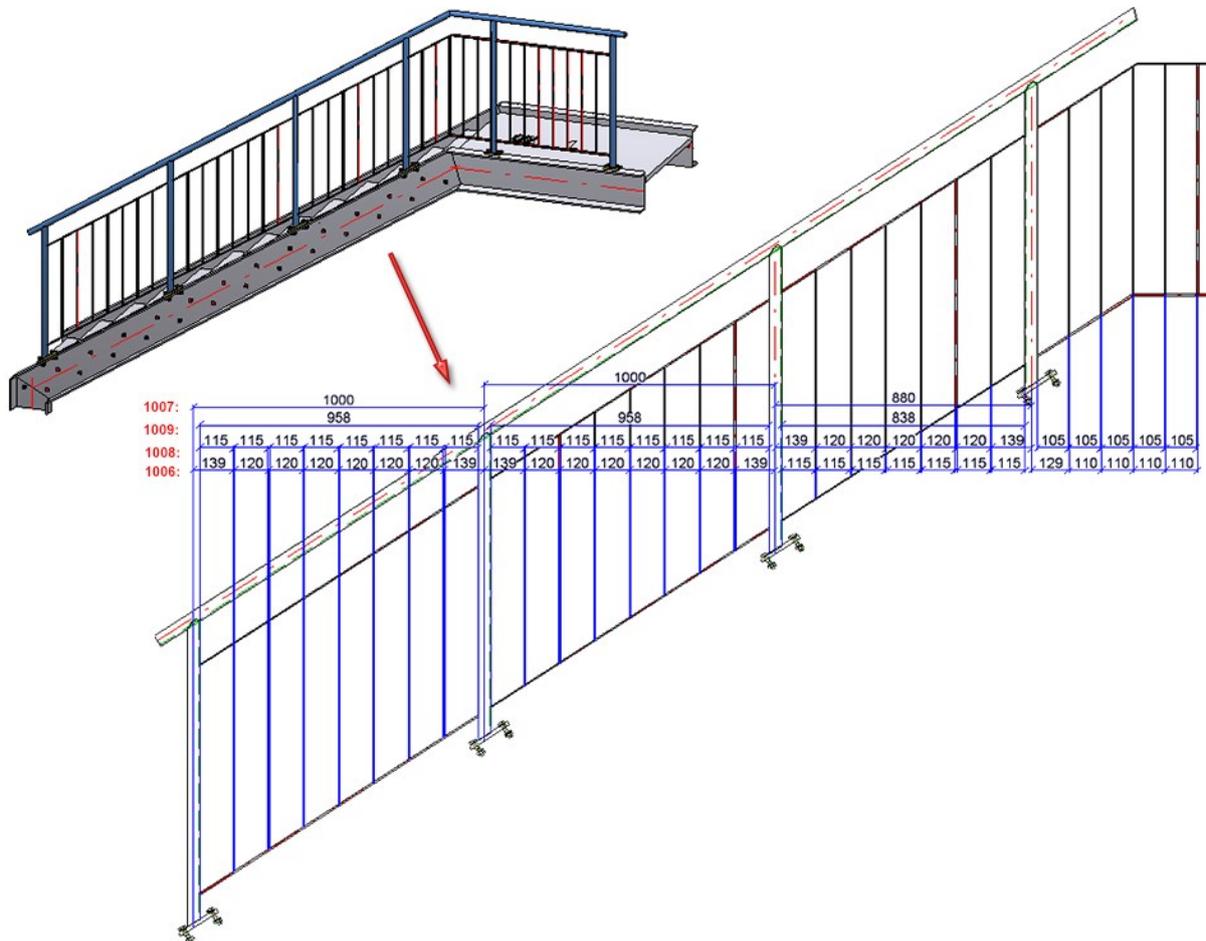
**Example 1:**

The depiction shows a staircase with equal distance between post and following vertical rod as well as between the vertical rods. The thickness of post and vertical rod are identical. The dimension and chain of dimensions of rules 1006-1009 are equal for each field.

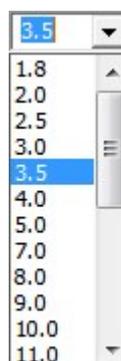


**Example 2:**

The oblique segment of the depicted staircase has individual distances for posts and vertical rods as well as different post and vertical rod thickness. Hence, dimension and chain of dimensions of rules 1006 to 1009 are not equal.

**Others****Selection of fonts**

With regards to font size (e.g. for texts, measures, captions), HiCAD offers nominal size for fonts according to DIN EN ISO 3098-0.



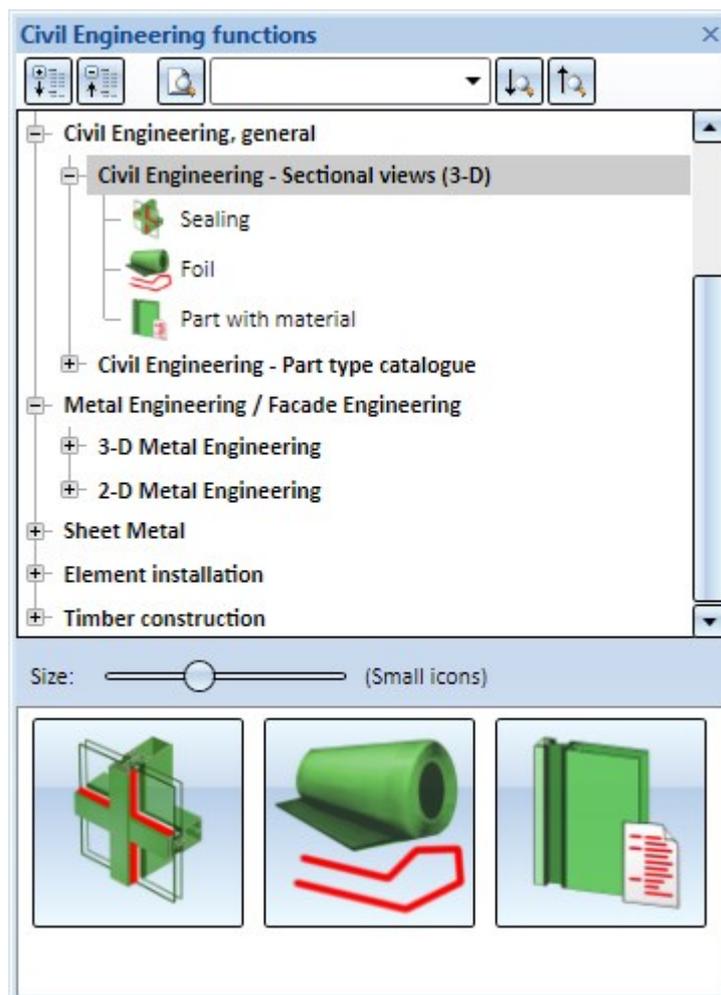
## Service Pack 1 2018 (V 2301)

### Licensing

As of HiCAD 2018, the Metal Engineering functions Sealing, Foil and at Part with Material are also available in the following modules and suites:

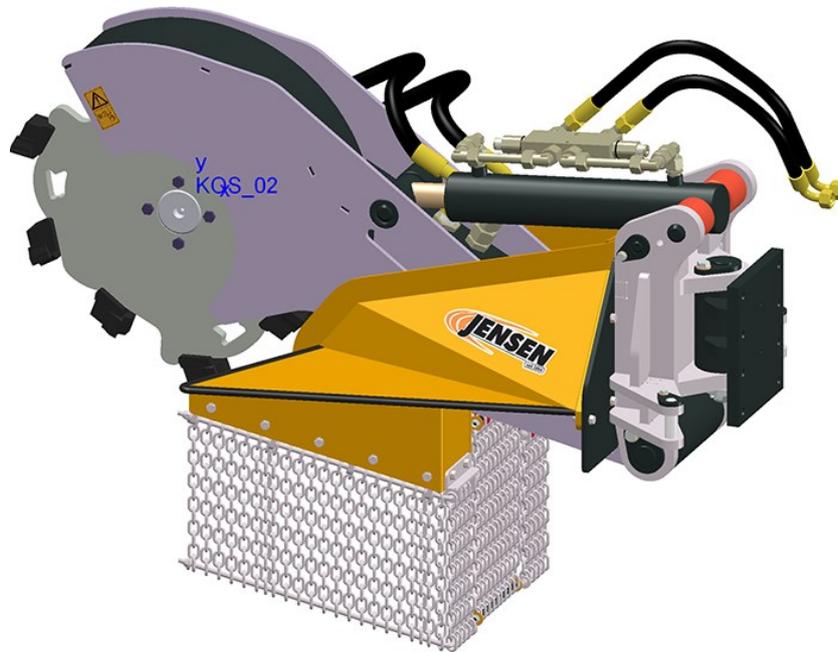
- HiCAD Beams+Profiles,
- HiCAD Metal Engineering,
- HiCAD Steel Engineering,
- HiCAD Steel Engineering - Stairs and Railings,
- HiCAD Steel Engineering suites,
- HiCAD Metal Engineering suites,
- HiCAD ALUCOBOND® suites.

Previously, the functions have been part of the **Metal Engineering**-Ribbon and could be found via **3D-Section**. As of HiCAD 2018, these functions are available via the docking window **Civil Engineering functions** under **Civil Engineering, general > Civil Engineering - Sectional views (3D)**.



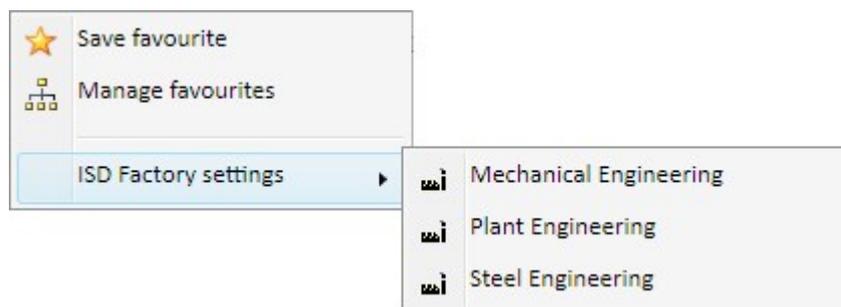
## Performance - Reduced polyhedron model

When saving SZA and KRA files, the polyhedron model will be reduced as far as possible. This applies to referenced parts and standard parts without processing in particular. By this, the file size and the time of saving and loading will be decreased. When testing this, the drawing shown below has been taken into account with approx. 1700 parts. About 900 parts of it were same parts - internally referenced. In HICAD 2018 SP1, the file size has been reduced from 33 MB to 11 MB.



## Manage favourites - Default settings

To several functions - for example Itemisation - settings predefined by ISD - **ISD default settings** and **ISD factory defaults** - are available in the scope of delivery of HiCAD. These are locked for editing and marked with  in Manage Favorites. In order to change and reuse these settings, they must be saved under a different name.



### Itemisation - Context menu for favourites

With a lock symbol, marked favourites are locked by your system admin. They are also not deleteable or changeable. If you have questions, please contact your system admin.

In case of an update, the existing default settings can be prescribed by ISD and/or further settings may be added.

## Drawing derivation - Usage-dependent configurations

For **Railings** the following three new usage types are available in the **Factory standards** catalogue at **Usage > Civil Engineering > Steel Engineering > Railing**:

- Skirting board profile,
- Handrail profile,
- Post profile.

You can assign usage-dependent configurations for drawing derivations to these new railing elements.

The following configurations have been preset for the new usage types in the Configuration Editor at **Automatic drawing derivation > Production drawing > Usage assignment**:

Usage	CONFIGKEY	Utilized configuration	Consider for drawing derivations
Post profile	POSTPROFILE	DEFAULT (STAHLROHRE)	Yes
Handrail profile	RAILINGPROFILE	DEFAULT (HOHLPROFIL)	Yes
Skirting board profile	SKIRTINGPROFILE	DEFAULT (FLACHSTAHL)	Yes

Furthermore, the default setting for the vertical rods used for infills has been changed:

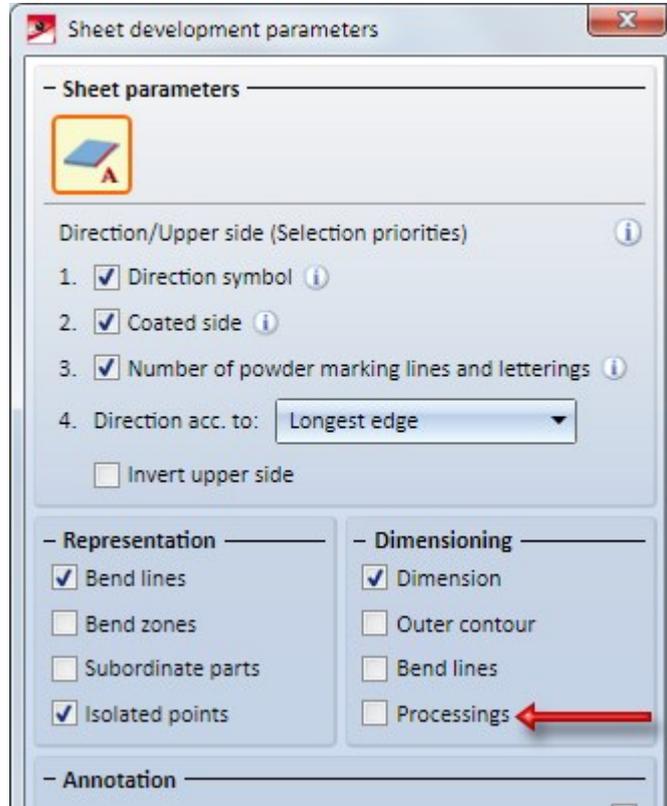
Usage	CONFIGKEY	Utilized configuration	Consider for drawing derivations
Rod	WEBMEMBER	DEFAULT	Yes

The default setting for **Steel Engineering** elements of the part type **Round steel** has also been changed:

Usage	Part type	Utilized configuration	Consider for drawing derivations
All	Round steel	DEFAULT (STAHLROHRE)	Yes

## Drawing derivation - Dimensioning of sheet developments

The parameters for sheet development have been enhanced. In the dialogue **Sheet development parameters**, the check box **Processings** is new.



By ticking this checkbox you determine that bores and subtractions will be dimensioned on blank sheets.

If you tick the **Processings** checkbox below **Dimensioning**, dimensionings will also be created for bores and subtractions. The following must be considered: If the checkbox **Separate chains of dimensions for bores and subtractions** is activated from the tab **Bores/Boltings** of the function **Drawing > Itemisation/Detailing > Dim... > Dimensioning Settings**, separated chains of dimensions for bores and subtractions will be created.

## HELiCON-specific attributes

Due to the discontinuation of HELiCON, the HELiCON-related attributes shown in the table below are no longer available after a new installation of HiCAD 2018 SP1. This change concerns the **System attributes** catalogue and the settings in the Configuration Editor at **System Settings > Referencing > Synchronization of attributes**.

Attributes for HELiCON	
%INSTANCE_ID	HELiCON instance ID
%ISCHANGED	HELiCON IsChanged flag
%ISSELECTED	HELiCON IsSelected flag
%LAST_INSTANCE_ID	Last HELiCON instance ID
\$TN_COPY	Copy of part name for HELiCON without HELiOS
CONFIGURATIONID	HELiCON configuration ID
STATIC_PART	HELiCON purchased part marking
TRANS_INFO	HELiCON transfer info

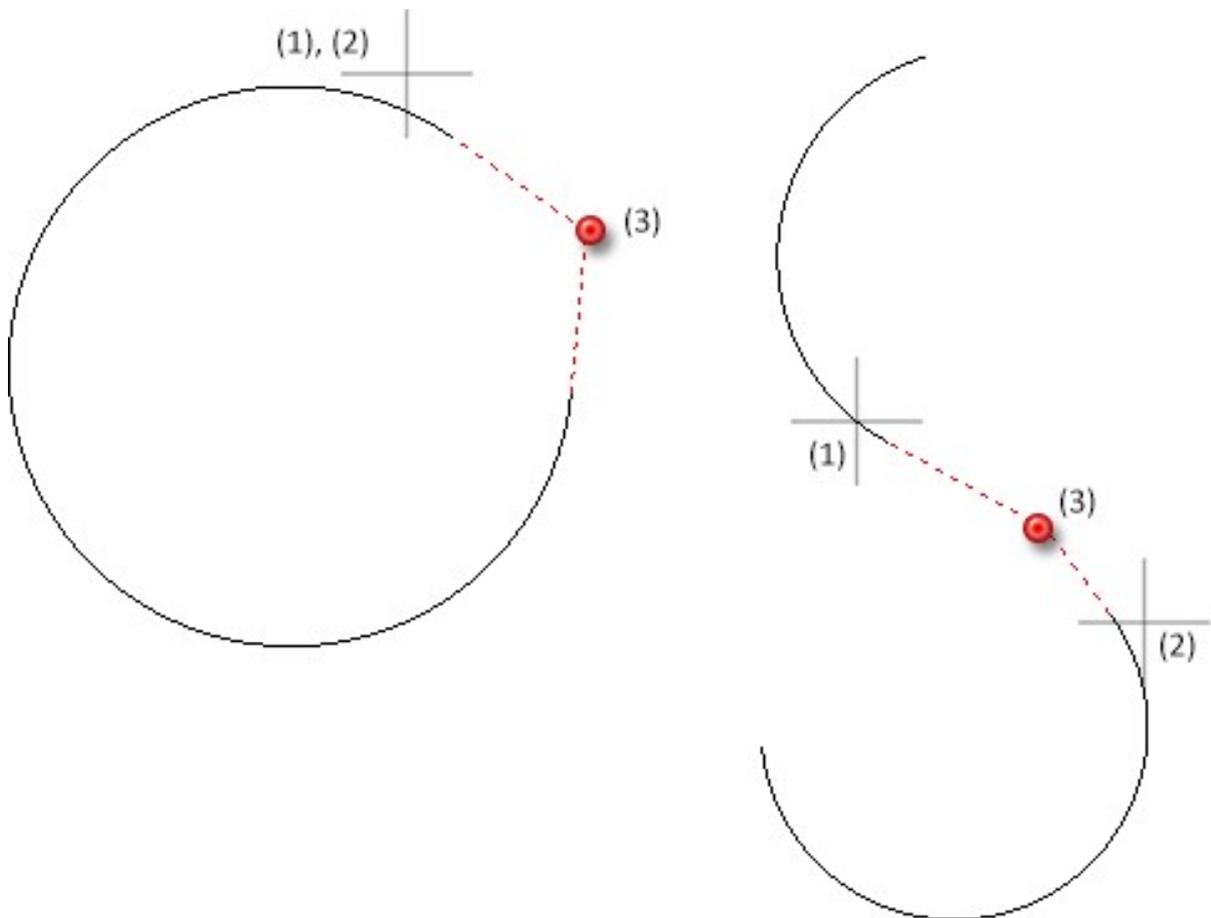
After an update installation from an older version (before HiCAD 2018) the attributes will remain available.

## Intersection of tangents

A point option is available in 2D and 3D:  Intersection of tangents

With this option you determine the intersection of two tangents. HiCAD will request you to identify two graphical elements you require. The tangent will be defined by the end point of the graphical element lying closer to the cursor position. The new point is the intersection of both tangents.

In case if a graphical element is identified twice, for instance as arc, the intersection of tangents will be defined on both end points of the graphical element.

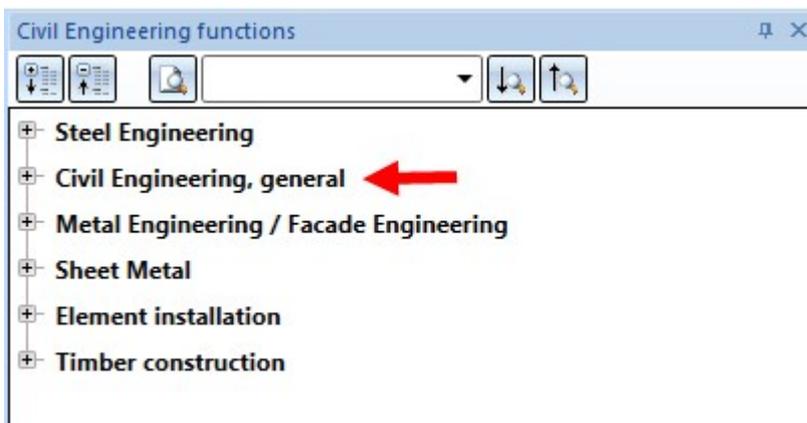


(1), (2) - selected graphical elements (here planar sketch), (3) identified point

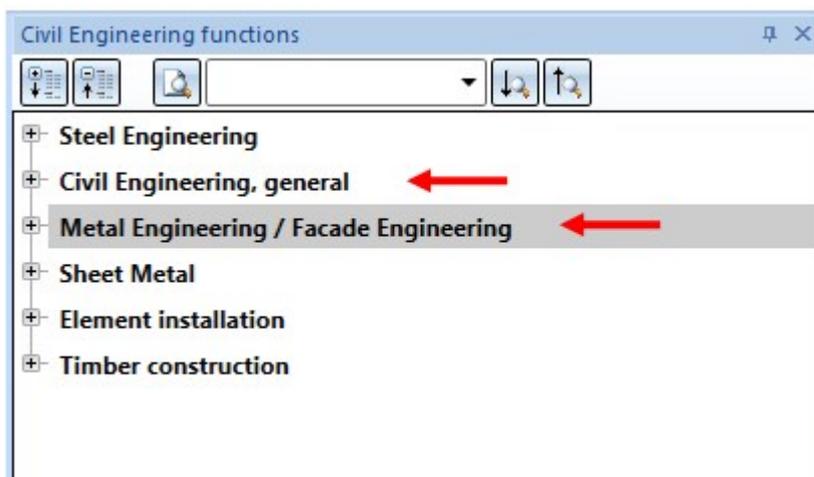
## Major Release 2018 (V 2300)

### Changed licensing

- As of HiCAD 2018 the **Steel/Metal Engineering** module is no longer available. This module will be replaced with the two modules **HiCAD Steel Engineering** and **HiCAD Metal Engineering**.
- In addition to the existing **HiCAD Education Edition**, a new HiCAD basic module, the **Education Edition Premium** is now available. This module is exclusively indented for schools, i.e. only schools with an existing customer number and license contact can have this module. The module contains all functionalities of the **Education Edition** plus all interfaces and can be supplemented by HiCAD extension modules.
- From Version 2018 onwards, the **HiCAD Steel Engineering** module will contain the connections that you can find in the **Civil Engineering functions** docking window at **Civil Engineering, general**.



- As of HiCAD 2018, the joints and connections of the **Metal Engineering** module can be found in the Civil Engineering functions docking window at **Metal Engineering / Facade Engineering** and **Civil Engineering, general**.



## Itemisation - New procedure

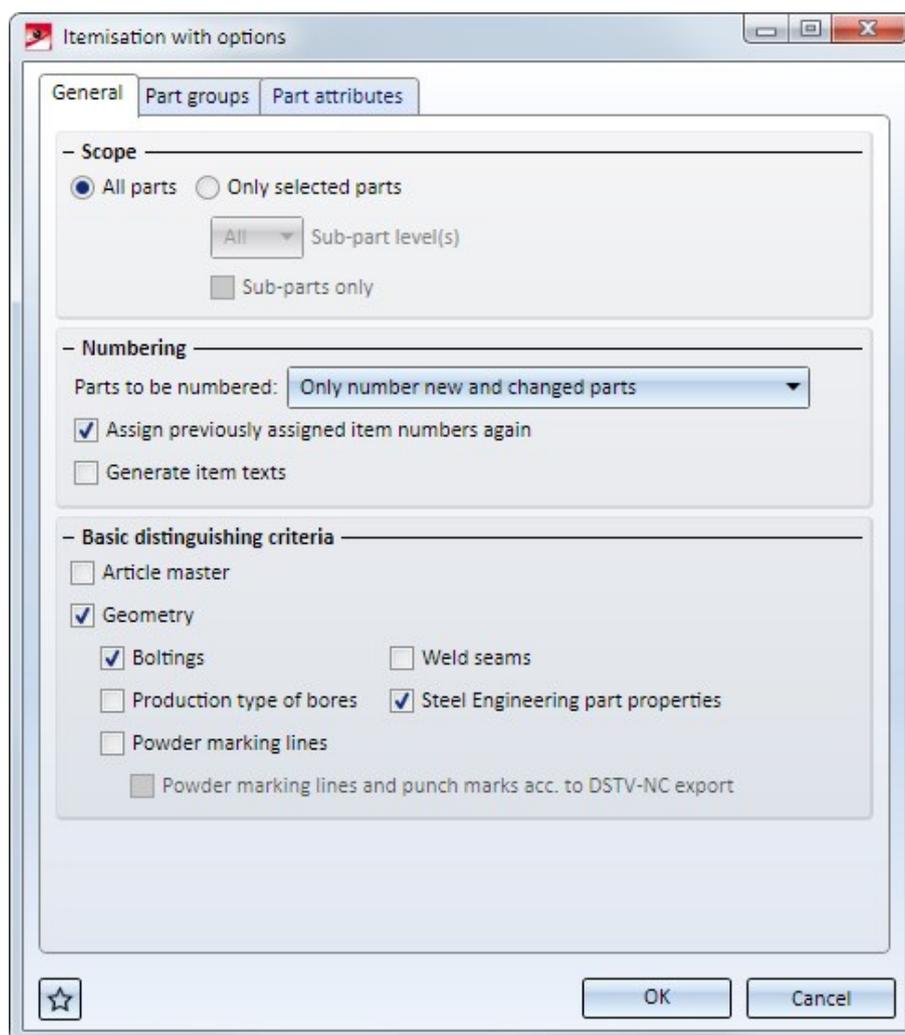
HiCAD currently supports two different itemisation modes:

- Itemisation up to HiCAD 2017 and
- Standard itemisation as of HiCAD 2018 with new parameter settings.

One property of each drawing is the question whether it is to be itemized in the old or remastered mode. In the Configuration Editor at **System settings > Itemisation, revised > Itemisation mode** you can specify the itemisation mode to be used in new drawings. For model drawings without a fixed itemisation mode, the **Itemisation up to HiCAD 2017** mode will be used if they have at least been partly itemized.

In contrast to the **Itemisation up to HiCAD 2017** mode, itemisation settings in the **Standard itemisation as of HiCAD 2018** mode will not be saved globally, but by model drawings. For a management of individual configurations you can use the **Manage favourites** option that you already know from other functions. In the Configuration Editor at **System settings > Itemisation > Default settings** you can choose a Favourite defining the pre-settings for the **Standard itemisation as of HiCAD 2018** mode. By default, settings for Mechanical Engineering, Steel Engineering and Plant Engineering have been pre-defined. Which of these Favourites will be used depends on the settings specified in the HiCAD Parameter configuration dialogue window.

The fixed part groups for the old **Itemisation up to HiCAD 2017** are replaced by freely configurable part groups when you use the new **Standard** itemisation. You can use a part filter to specify the elements belonging to a part group.



Furthermore, two options for number assignment are available:

- **Rule-compliant (automatic / manual)**

The item numbers for elements of the part group can be assigned automatically or manually and must comply with the specified equality criteria.

- **Unregulated (only manual)**

The item numbers for elements of the part group can be assigned freely (except for conflicts with item numbers according to rule-compliant number assignment). Automatic itemisation is not supported here.

#### Advantages of the new itemisation:

- ✔ The current settings of the itemization parameters are saved in the model drawing. Furthermore, the itemisation settings can be saved as Favourites, making individual settings for various industries available at the push of a button.
- ✔ The fixed part groups used for the old itemization up to 2017 are replaced with freely configurable part groups. You can individually specify via the part filter which parts are to belong to a part group.
- ✔ In addition to the item number, a freely configurable item text, e.g. for a detailed description of the type and quality of a service can be generated automatically. Item texts are composed of components of the type:
  - Text
  - HiCAD part attribute
  - HiCAD drawing attribute
  - HELIOS document attribute
  - HELIOS project attribute

HiCAD part attributes can originate from the part itself as well as from the main part of an assembly, or a superordinate element with user-specific properties. Item text settings, too, can be managed as Favourites.
- ✔ The range of application for itemisation can be limited to selected parts and their sub-parts within a given number of levels.

#### Other differences to the old **Itemisation up to HiCAD 2017**:

- The result of the itemization is influenced by the order of part groups, i.e. each part will be assigned to the first matching part group. But the order of the item number assignment depends exclusively on the chosen sort criteria. If you distinguishing / sort criteria have been defined, HiCAD will simply go through the part structure for the assigning of item numbers.
- The new itemisation does not allow consistent states, that is, in case a a rule-compliant number assignment all itemised identical parts will have the same item number.
- The **Lock given numbers** option is now available as the **Assign previously assigned item numbers again** option with a "reverse logic", so to speak. It is now possible to change the option subsequently.
- The possibility to **Reference** during itemisation is not available. Referencing and itemisation use different, incompatible methods for identical part search.
- An itemisation by assemblies is currently not supported by the **Standard Itemisation as of HiCAD 2018**.

## Settings from TXTPAR.DAT moved to Configuration Editor

In previous versions the settings for texts and annotations could be set in the TXTPAR.DAT file. As of HiCAD 2018 this file is no longer available. Also, the corresponding setting options at  **Settings > Basic settings > 2-D** have been removed.

Instead, these settings are now available in the Configuration Editor, namely, at

- **Drawing > Annotations > Text > Font** and
- **System settings > Annotations > Text.**

## Macro for body rebuilding not available anymore

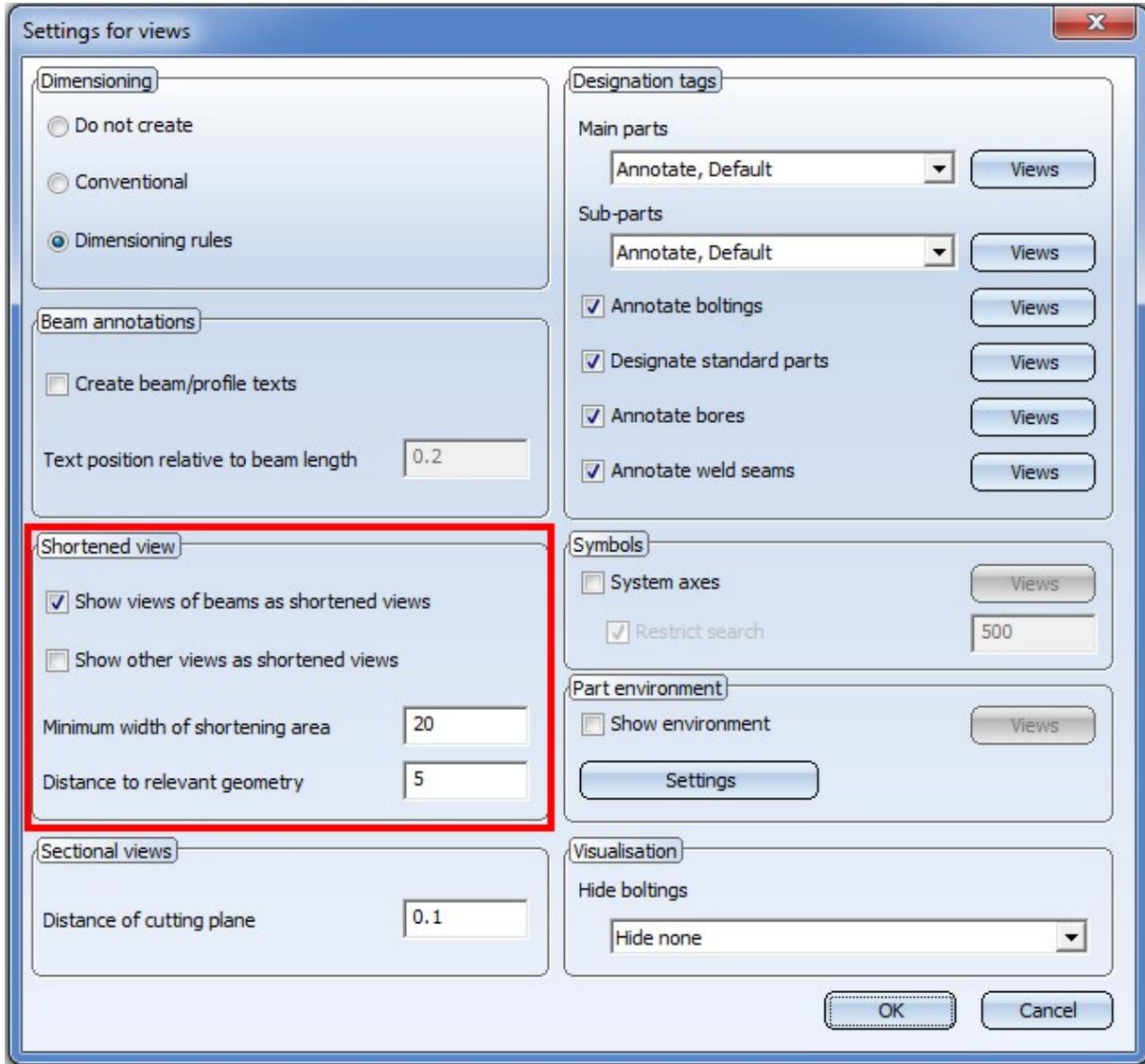
The macros

- **Body rebuilding** (2dinfo\_krpneuaufbau1)
- **Body rebuilding all parts** (3dinfo:krpneuaufbau)

are not available as of HiCAD 2018.

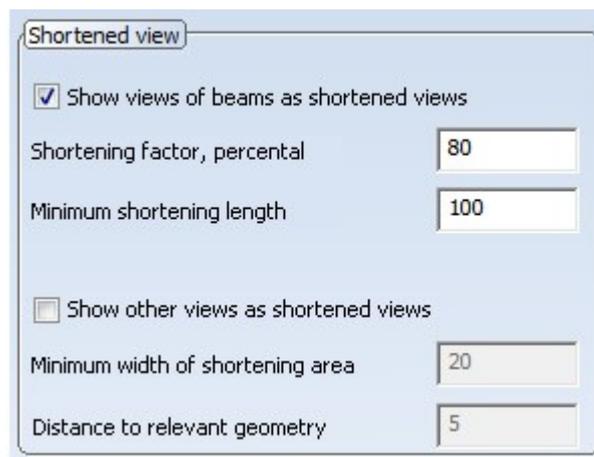
## Derived drawings - Shortened views of beams and other part types

The parameters for the shortening of beams and other part types have been combined in the **Settings for views** dialogue window.



**Please note:**

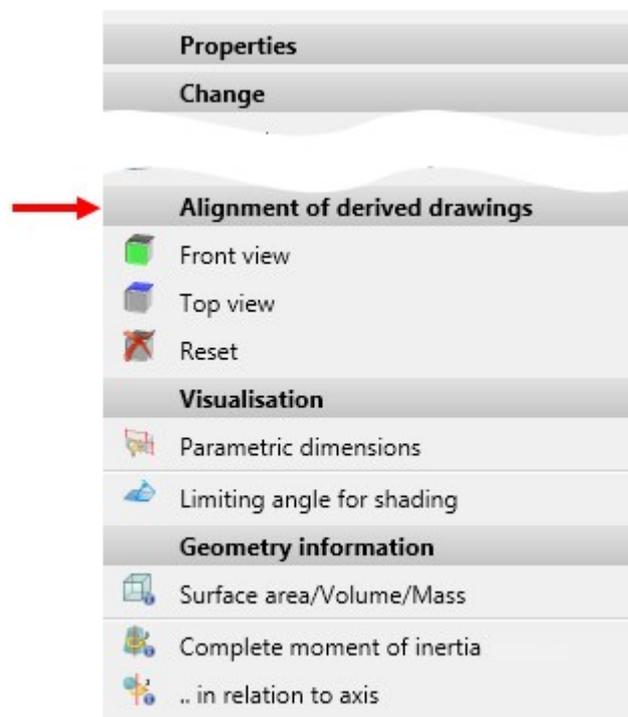
In HiCAD 2018 the Settings for views dialogue window for the Drawing derivation function has changed. If you load workshop drawings or model drawings with workshop drawings created with a version before HiCAD 2018 and then use the **Change settings - Active view** function, the **Shortened view** area of the previous version will be shown in the **Settings for views** dialogue window:



### Derived drawings - Simplified definition of front view

The definition of the front view and the top view for assemblies, general 3-D parts, beams, profiles, plates and sheets has been simplified.

For this purpose the functions in the context menu for parts (right-click part) at **Properties >Alignment of derived drawings** has been expanded accordingly. You have now the option to select an arbitrary plane for the definition of the front view or the top view. The same functions as the ones for the determining of processing planes are available to you.



## Suppress temporary preview for part placement

When selecting certain functions, a shaded polyhedron model preview will be attached to the cursor. For instance, this is the case when inserting or transforming parts, inserting new parts in the preview mode, or inserting standard parts.

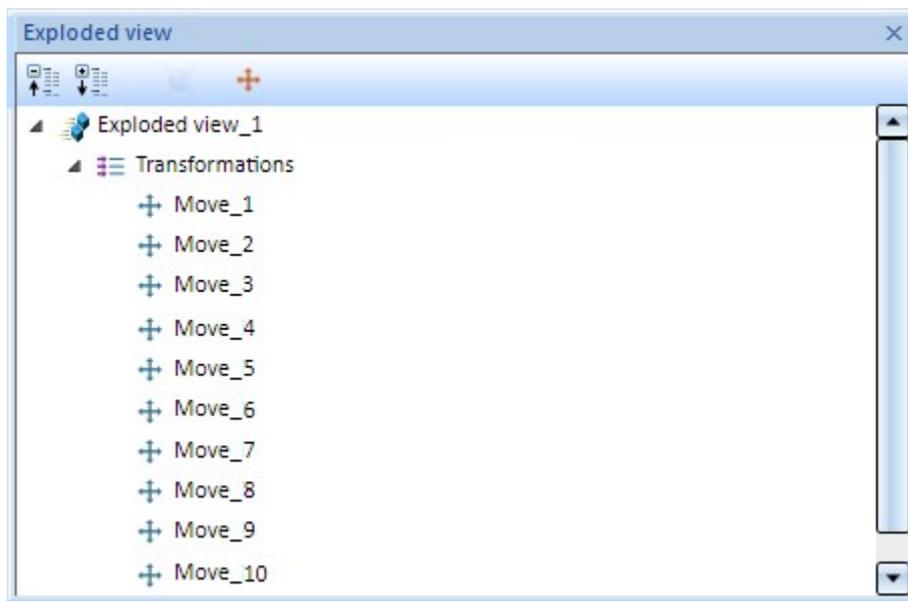
This preview can be hidden temporarily by pressing and holding down the SHIFT key. This may be useful when working with complex models due to performance reasons. When the key is released, the preview is visible again.

This behaviour can also be inverted by a corresponding setting in the Configuration Editor at **System settings > Visualisation > Temporary preview for part placement**. Possible setting options are:

- **Hide with SHIFT key**  
When a preview for part placement is shown it can be hidden by pressing the SHIFT key. This is the default setting.
- **Show with SHIFT key**  
No preview for part placement will be shown. By pressing and holding down the SHIFT key, however, it can be shown temporarily.

## New docking window: Exploded view

The 3-D function **Exploded view** has been revised. Amongst other things, all displacements of the exploded view will now be documented. The corresponding log will be, similar to the Feature log, displayed in the new **Exploded view docking window**. You can also change exploded views via this docking window.



Exploded view docking window

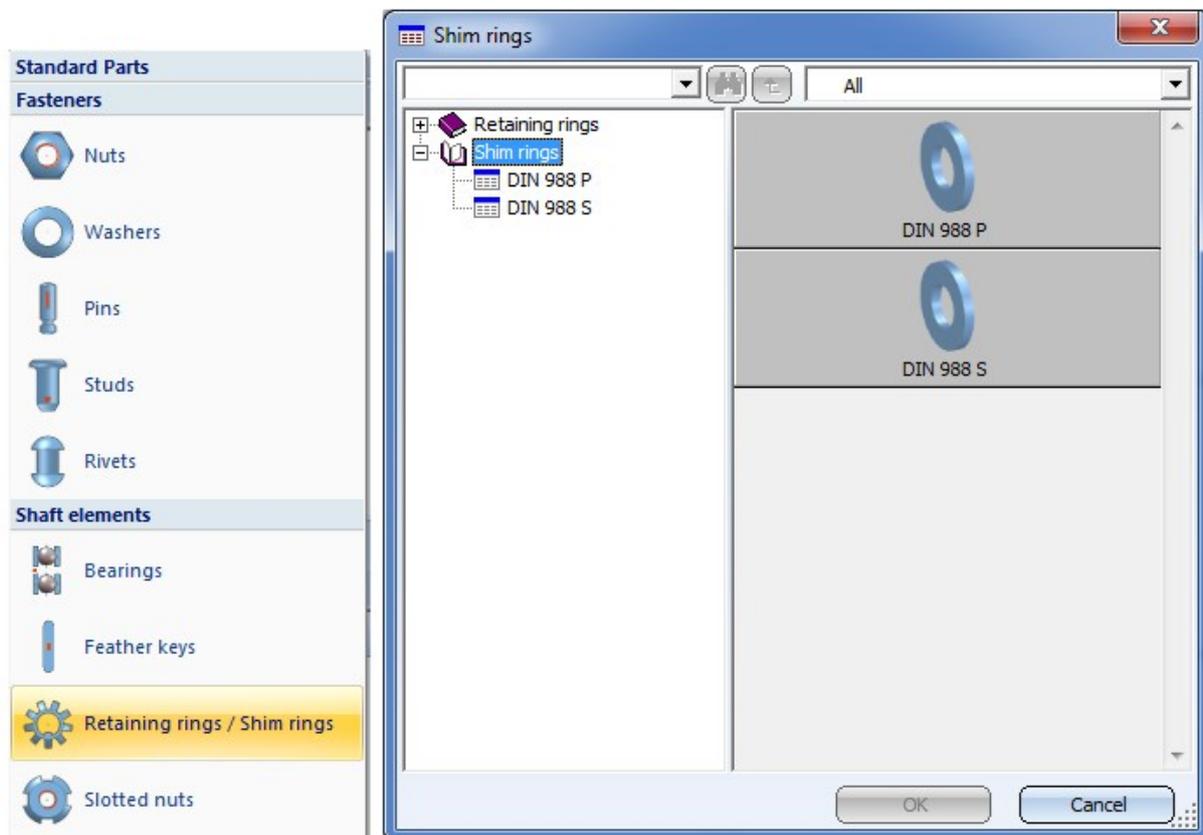
## 2-D

### Service Pack 2 2018 (V 2302)

#### New catalogue for shim rings

Shim washers could previously be found in the catalogue **Fasteners > Retaining rings**. As of HiCAD 2018 SP2 a separate catalogue exists for **Shim rings**.

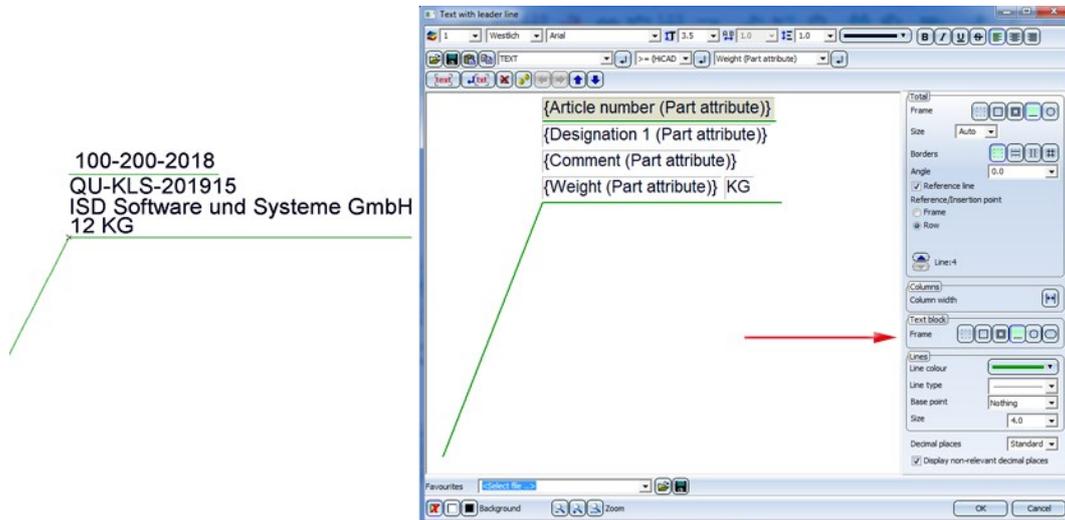
Therefore, the 3-D function **2-D Part > Standard Parts > BoltScrew** > Retaining rings has been renamed to **2-D Part > Standard Parts > BoltScrew** > Retaining rings / Shim rings .



## Dimensioning / Annotations

### Underlined text blocks

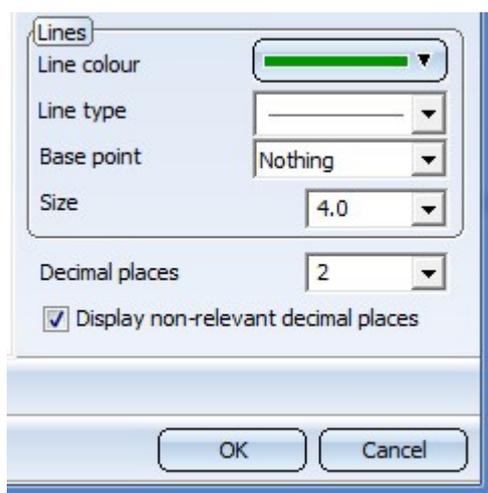
In the settings for annotations you have now the option to underline individual text blocks. This was previously only be possible for the entire annotation.



In older HiCAD versions (before HiCAD 2018 SP2) the underlining of text blocks is ignored. While the underlining is still visible and printable when loading the model drawing, the corresponding text blocks will be shown without frames after updating of the annotation.

### Decimal places

For attribute values with decimal places you have now the option to determine the number of decimal places at the bottom right of the dialogue window. You can specify whether non-relevant decimal places (e.g. final zeroes) are also to be displayed by activating or deactivating the checkbox.



### Font size selection

HiCAD now offers the nominal font sizes according to DIN EN ISO 3098-0 for the font size selection of texts, dimensions, annotations etc. in the Text Editor and Annotation Editor.

## Service Pack 1 2018 (V 2301)

### Change language of texts

The Change language of text  function and the program WEdit.exe now also consider auxiliary texts in 2-D/3-D dimensions, as well as texts instead of dimension figures.

## Major Release 2018 (V 2300)

### Settings from TXTPAR.DAT moved to Configuration Editor

In previous versions the settings for texts and annotations could be set in the TXTPAR.DAT file. As of HiCAD 2018 this file is no longer available. Also, the corresponding setting options at **Settings > Basic settings > 2-D** have been removed.

Instead, these settings are now available in the Configuration Editor, namely, at

- **Drawing > Annotations > Text > Font** and
- **System settings > Annotations > Text**.

Use the **Default parameters**  function (2-D Dimensioning + Text > Edit (Text) Settings  > ...) to load the settings from the Configuration Editor at **Drawing > Annotations > Text > Font**.

The values of the drop-down lists for font size, text width coefficient and line spacing in the Text Editor and Annotation Editor can be arbitrarily expanded in the Configuration Editor at **System settings > Annotations > Text > Preferred text sizes**. After restarting HiCAD the new default settings will be used.

### Default fonts in TTFONT.DAT

The supplied TTFONT.DAT file previously contained the 3 fonts

- Arial
- MS Mincho
- Arial Narrow

Since the second TrueType Text Font **MS Mincho** is rarely used, it was replaced with the **Arial Unicode MS** font.

From Version 2300 onwards, you will find the following fonts in the TTFONT.DAT file:

- Arial
- Arial Unicode MS
- Arial Narrow

### Translate annotation tags

Texts, text blocks and annotation tags will be shown in the selected language in the Text Editor or Annotation Editor,

respectively, after their translation  with the program WBEEdit.exe.

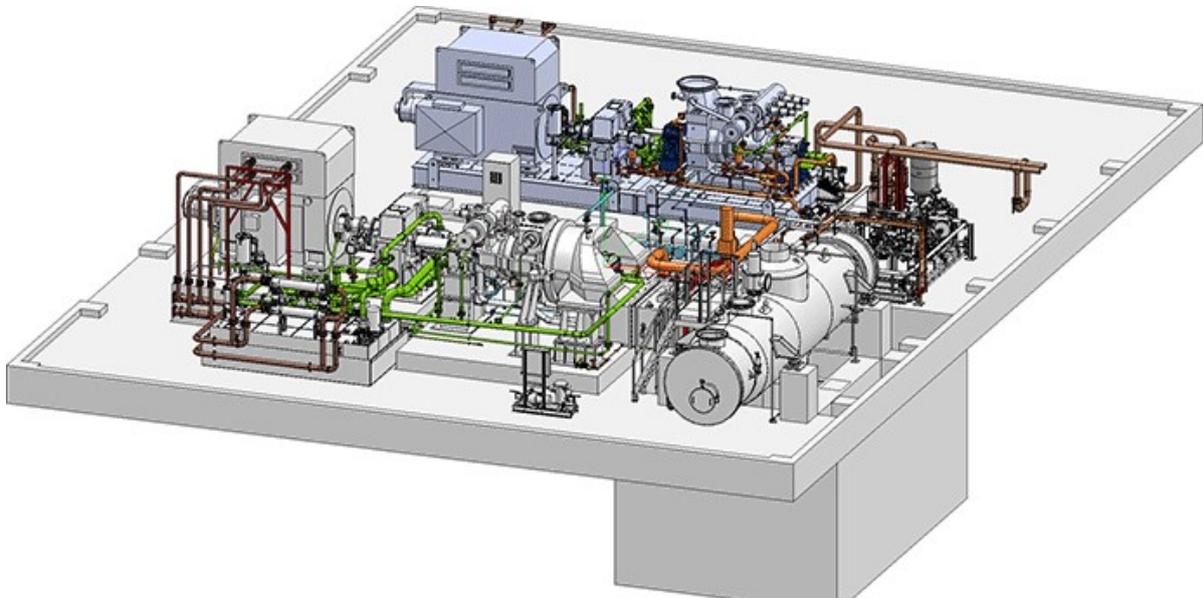
## 3-D

### Service Pack 2 2018 (V 2302)

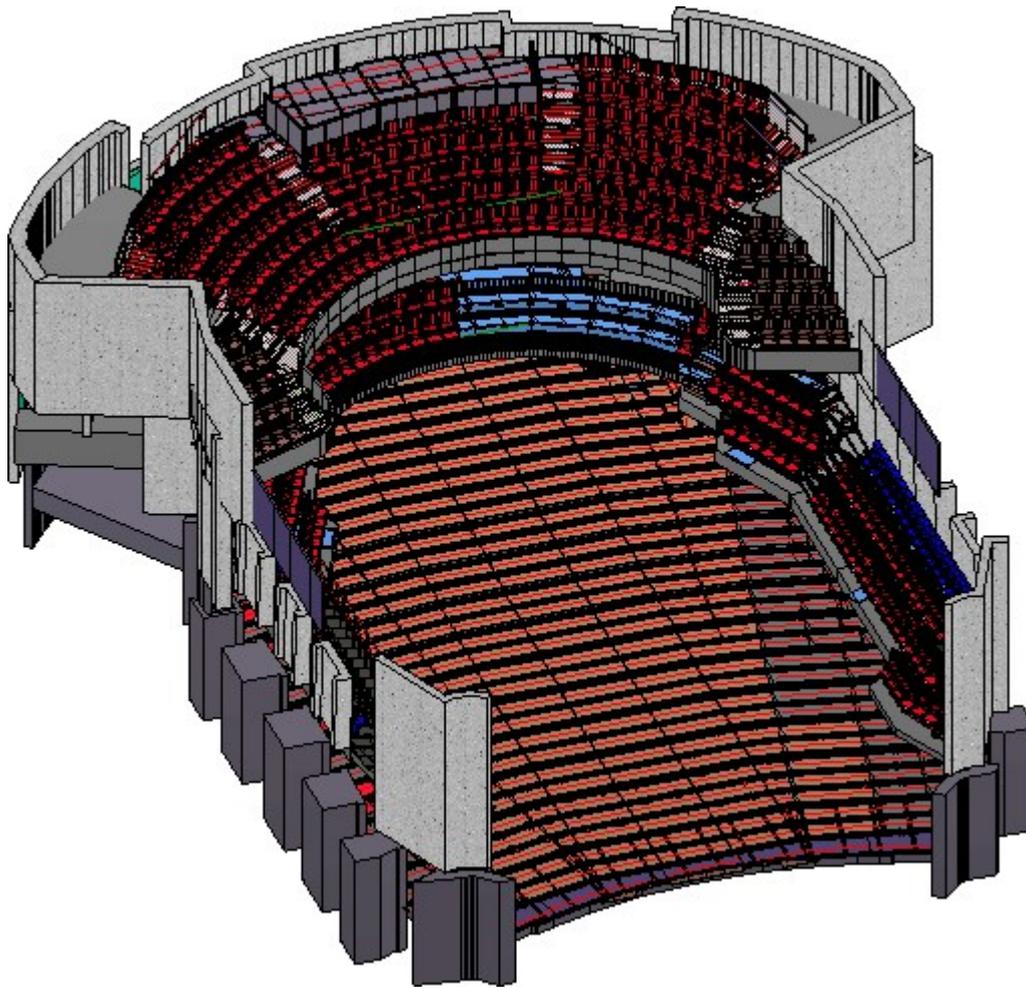
#### Views

##### Performance for rotations

Shaded views can now be rotated much faster - especially shaded views with visible edges. Three different model drawings - consisting of 20.000, 60.000 and 90.000 parts, respectively, were submitted to performance tests. Some of these parts were highly complex, imported elements. Here are some concrete examples of the tested drawings:



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



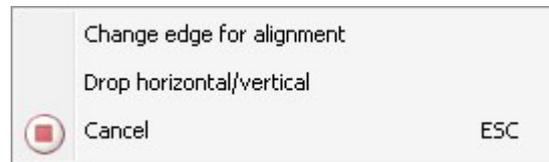
60.000 parts (Image: CAD Planung Arnold Matei, Mannheim, Germany)

Compared with HiCAD 2017, a significant increase of the frame rate (the number of images created during rotating) for the tested drawings could be achieved:

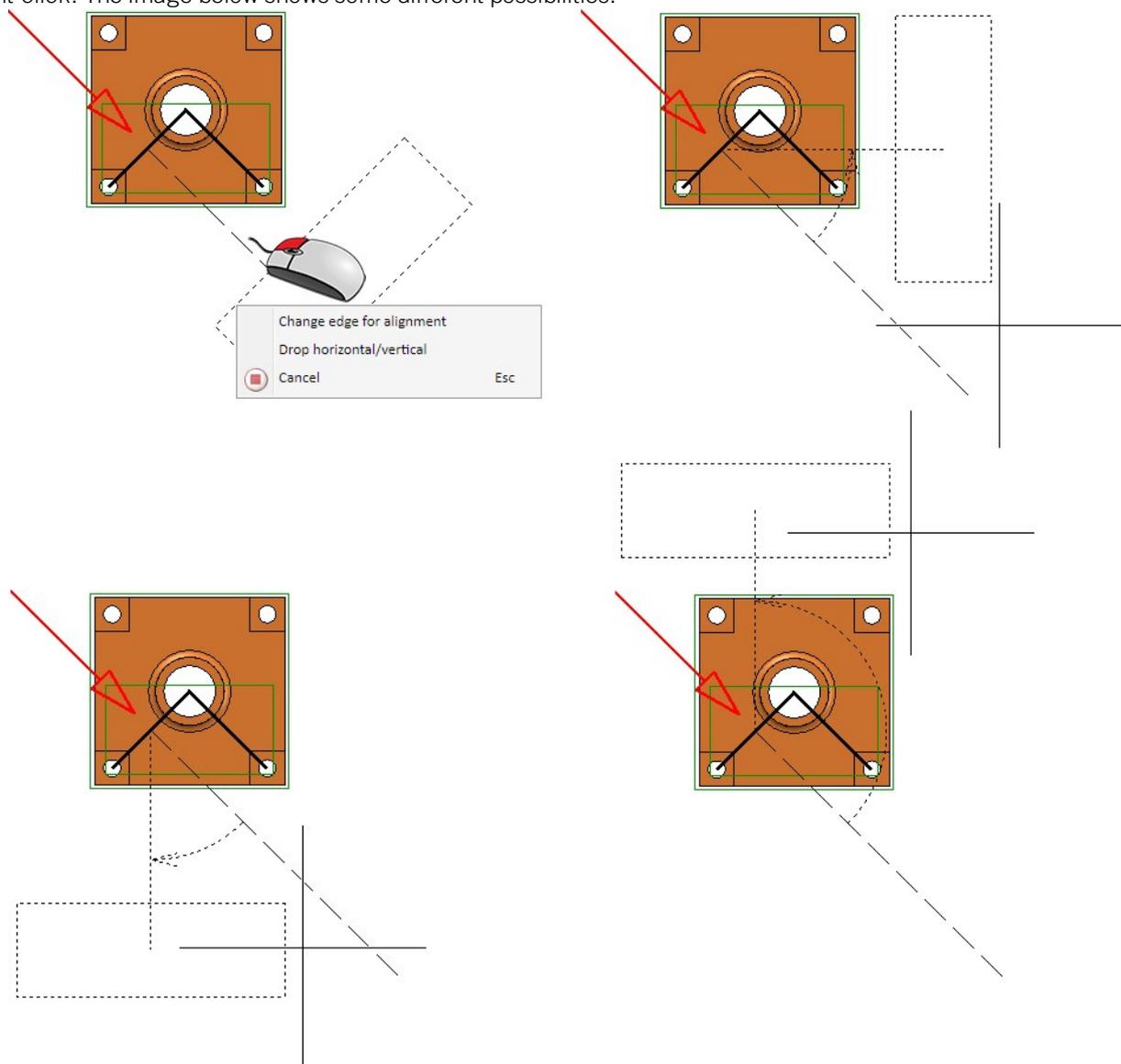
- Shaded with edges: Speed of rotation increased by a factor of 3.7 and
- Shaded without edges: 27% additional images per second.

## Horizontal/vertical alignment of sectional views

Sectional views can now also be dropped horizontally or vertically. For this purpose the context menu that you can open while the view is "attached" to the cursor has been extended accordingly.



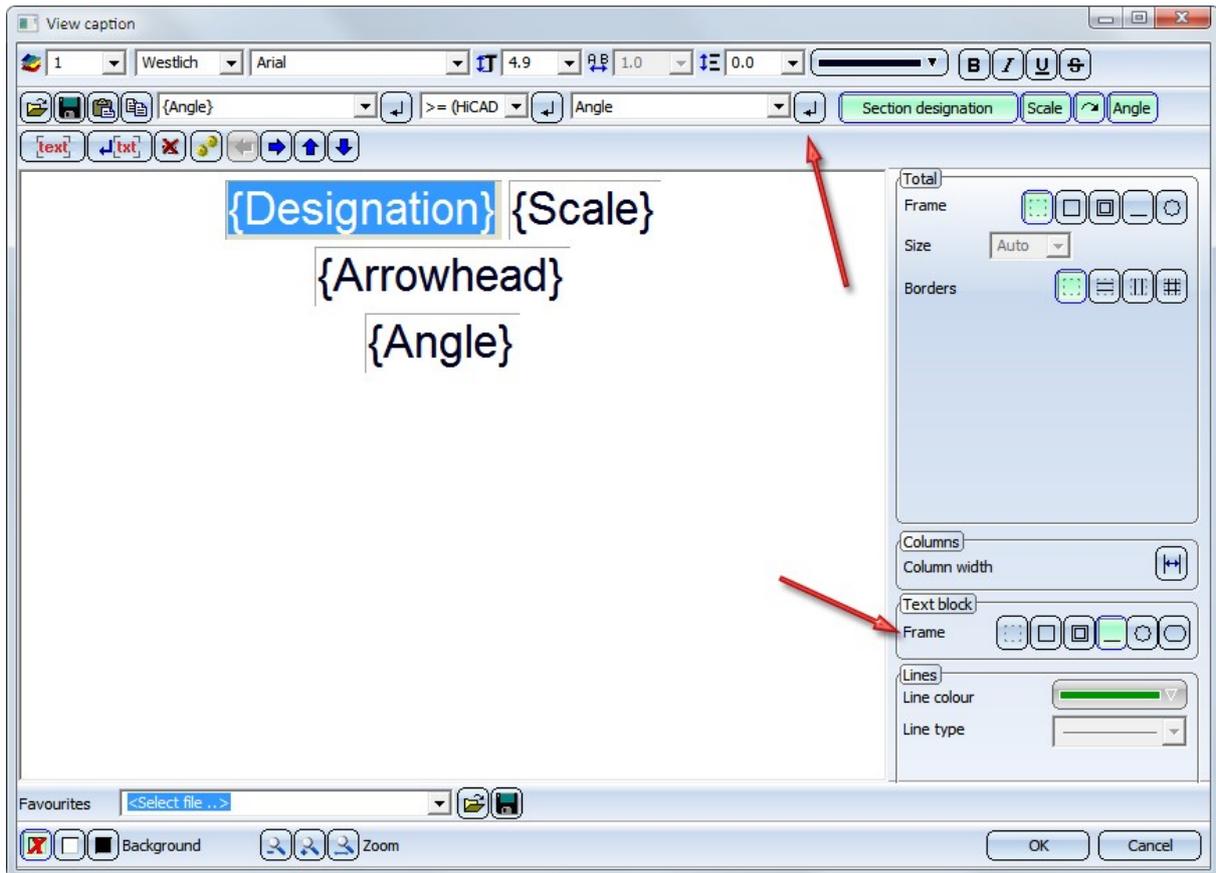
After calling the **Drop horizontal/vertical** function you can select the the rotation- upwards, downwards, to the left or right - with the cursor and drop the view. The viewing direction of the section will be determined in the moment of the function call, that is, it depends on the side of the reference edge on which you called the context menu with a right-click. The image below shows some different possibilities.



## Captions of sectional / detail views

The **View caption** dialogue window has been redesigned.

- Like annotations, the individual text blocks of view captions can now be displayed in a frame. Each text block can be shown with or without a frame or underlined. Possible frames are rectangles, rectangles with double lines, filleted rectangles or circles.
- You have now the option to use drawing, project or document master attributes for captions. Since sectional or detail views do not belong to particular parts, part and article master attributes will not be supported here.



In older HiCAD versions (before HiCAD 2018 SP2) the underlining of text blocks will be ignored. Although the underlining will still be visible when loading the model drawing, and they can also be printed. But after updating the annotation the corresponding text blocks will be shown without frames.

## Exploded view

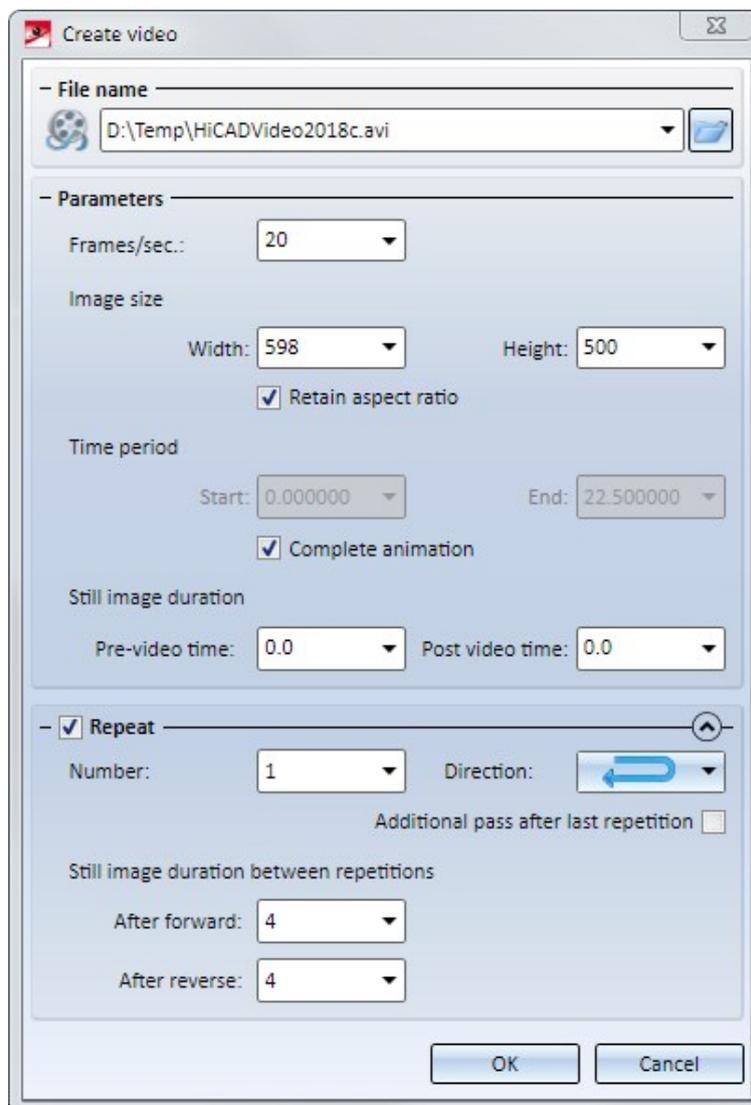
### New Ribbon

All possible functions for exploded views can now be found in a new Ribbon.



### Exploded view creation in video

Use the new **Video**  function to save the creation of exploded views as videos (similar to Simulations).



## Delete mounting tracks / Change representation

New are the Mounting tracks functions in the same-named function group of the Exploded view Ribbon:



Delete mounting track



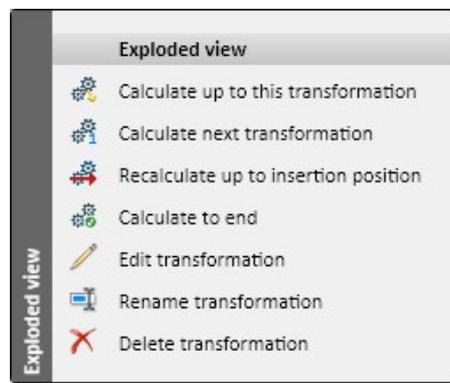
Change mounting track parameters



Default mounting track parameters

## New context menu for logs of exploded views

In the **Exploded view** docking window you can now open a new context menu. Right-clicking an item in the log of the exploded view, the following context menu will be displayed:



These functions enable you to partly or completely recalculate exploded views, e.g. after changes to the original model, or edit transformations.



### Calculate up to this transformation

The exploded view will be calculated up to the marked transformation. The insertion position will then be placed before the marked transformation.



### Calculate next transformation

Only the transformation behind the insertion position will be calculated. The insertion position will then be placed after this transformation.



### Recalculate up to insertion position

The exploded view will be calculated up to the insertion position.



### Calculate to end

The exploded view will be recalculated completely.



### Edit transformation

Use this function to edit the movements and rotations of an exploded view. When you call the function, a dialogue window will be displayed for this purpose.



### Rename transformation

Use this function to rename transformations.



### Delete transformation

This function deletes the transformation that was marked in the log. Multiple selections are also possible.

## Delete exploded views

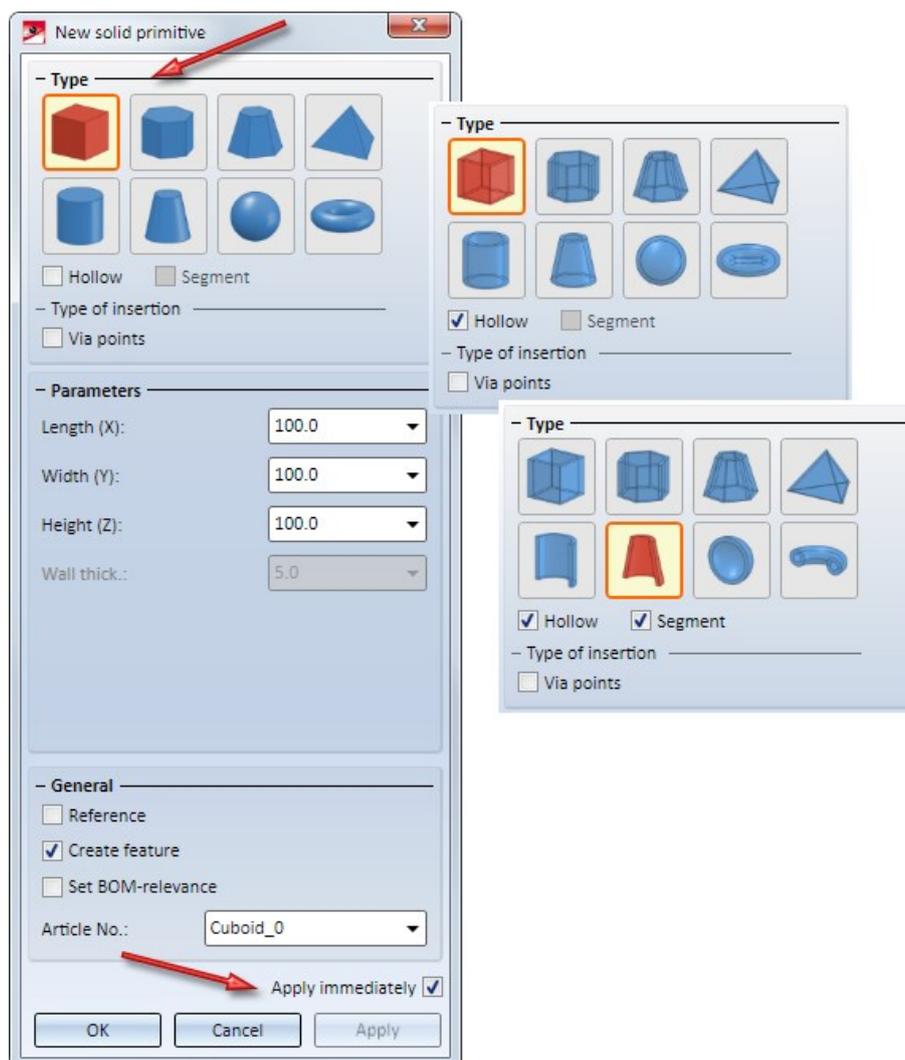
Exploded views can be restored to normal views with the new **Delete exploded layout**  function. You find the function at **Views > Representation > Shaded**  > ... and in the context menu at **Representation > Others > ...**

## Create / process parts

### New dialogue window for solid primitives

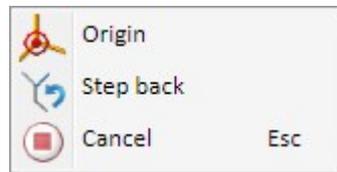
The **New solid primitive** dialogue window has been revised.

- The icons have been redesigned.
- Use the **Create as main part** checkbox to determine whether the part is to be inserted as a new main part or as sub-part to the active part. The **Create new solid primitive, as sub-part** function on the **3-D Standard** tab is therefore no longer available.
- The dialogue window will now remain open after insertion of a part, enabling you to insert more parts without having to call the function again. Click the **Close** button to close the dialogue window. If you prefer an automatic closing of the window after part insertion, activate the **Close window after creation** checkbox.



### Move+Rotate part - Changed behaviour

When using the Move+Rotate part, via fitting points function, the specification of the 3 fitting point pairs will now be graphically visualized. During fitting point specification in the model drawing, you can right-click to open a context menu with further point specification functions.

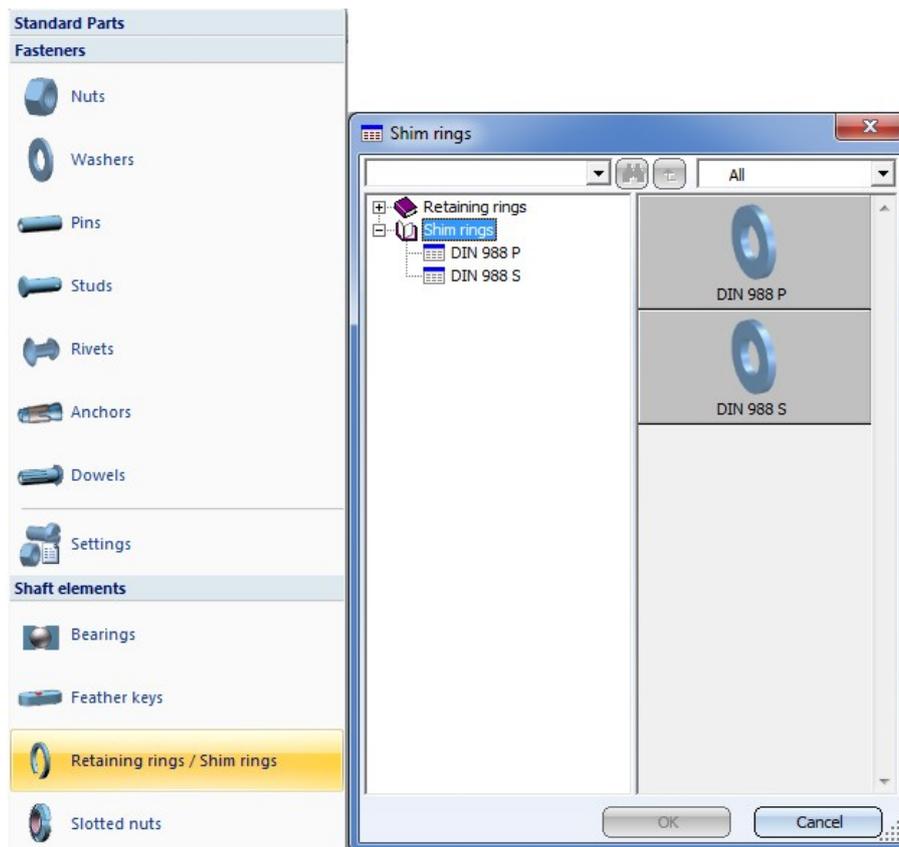


Use the **Origin** function to place the fitting point in the origin of the active coordinate system. Use the **Step back** function to select a new fitting point on the part

### Standard Parts / Standard Processings

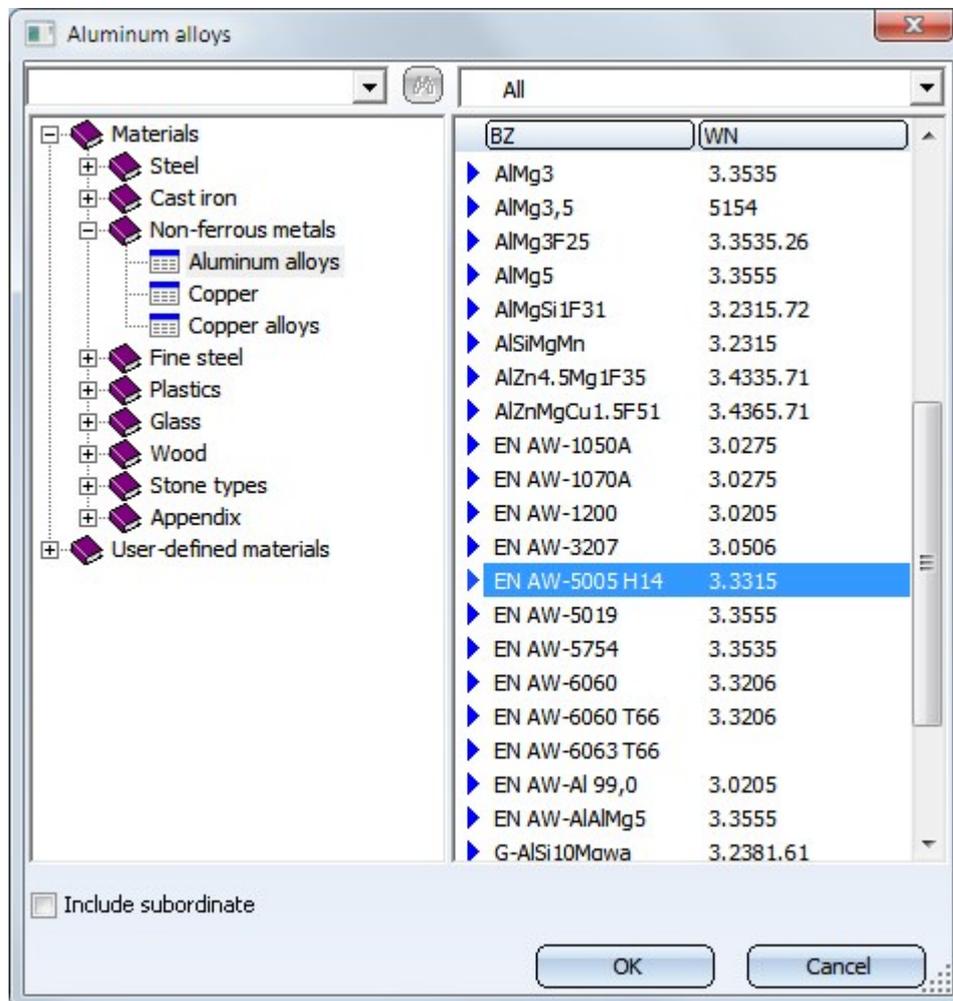
#### New catalogue for shim rings

Shim washers could previously be found in the catalogue **Fasteners > Retaining rings**. As of HiCAD 2018 SP2 a separate catalogue exists for **Shim rings**. Therefore, the 3-D function **3-D Standard > Standard Parts > BoltScrew > Retaining rings** has been renamed to **3-D Standard > Standard Parts > BoltScrew > Retaining rings / Shim rings**.



## New material EN AW-5005 H14

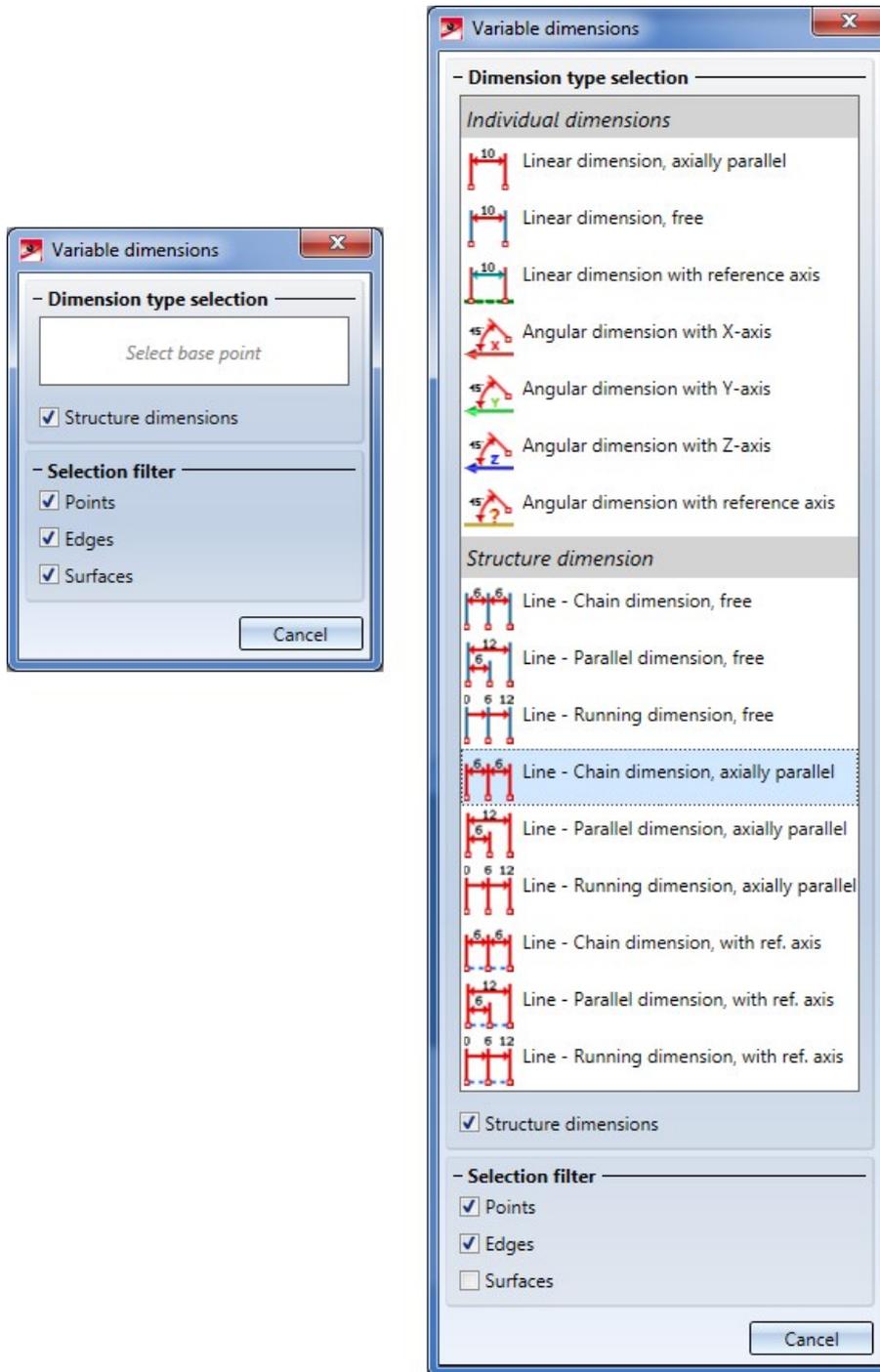
The material EN AW-5005 H14 has been added to the **Aluminium alloys** table (in the catalogue **Materials > Non-ferrous metals**).



## Dimensioning / Annotations

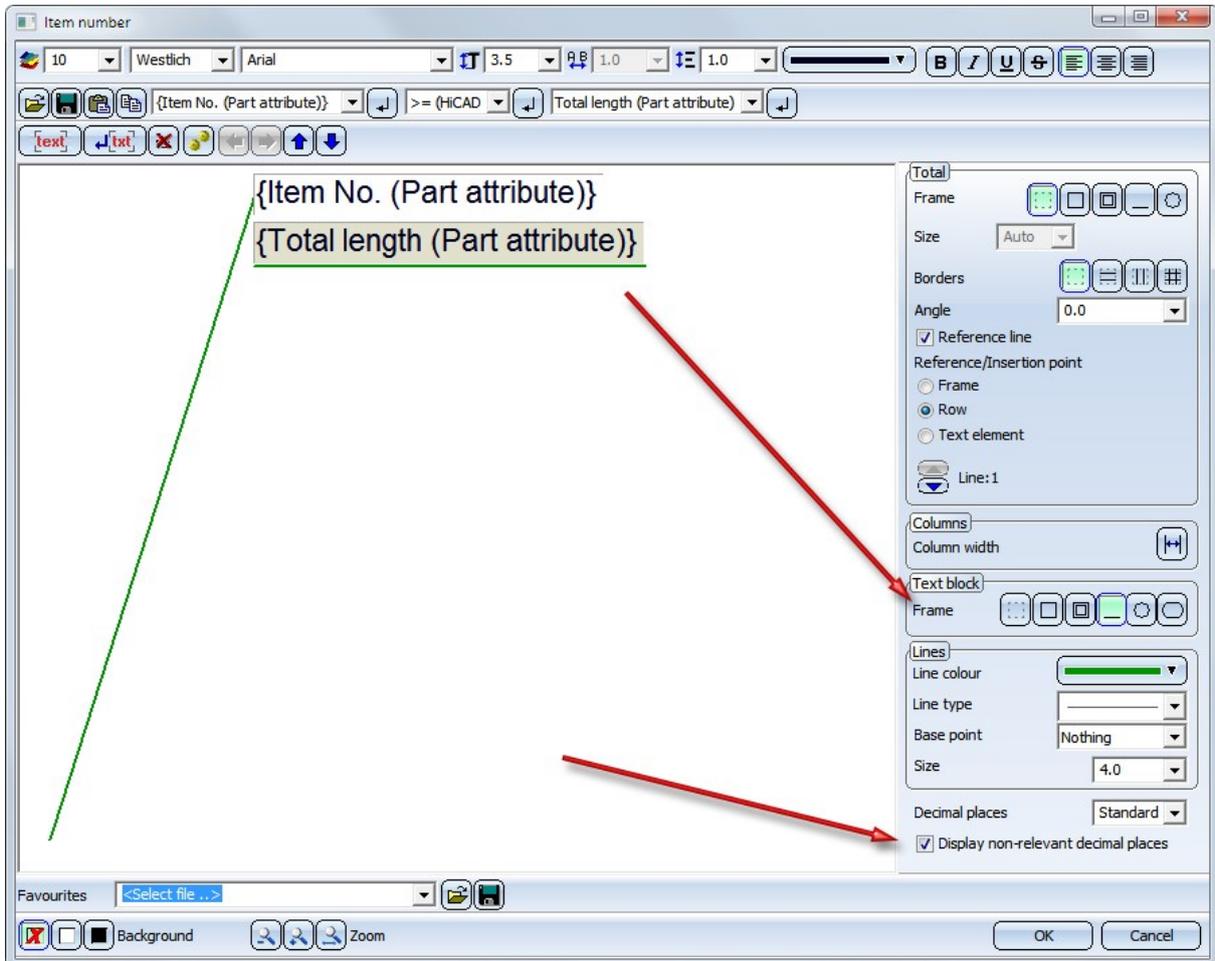
### Enhanced variable dimensioning

The variable dimensioning now also allows you to determine linear structure dimensions, i.e. chain, parallel and running dimensions. To do this, activate the **Structure dimensions** checkbox in the dialog window.



## Extended settings for annotations

- In the annotation settings you can now specify that only individual text blocks are to be displayed as underlined text. In previous version only the complete annotation could be formatted in this way.
- If you want to display attribute value with decimal places, you have the option to display non-relevant decimal places (e.g. final zeroes) as well. To do this, activate the same-named checkbox.



In older HiCAD versions (before HiCAD 2018 SP2) the underlining of text blocks will be ignored. Although the underlining will still be visible when loading the model drawing, and they can also be printed. But after updating the annotation the corresponding text blocks will be shown without frames.

## Graphic preview for polygon models

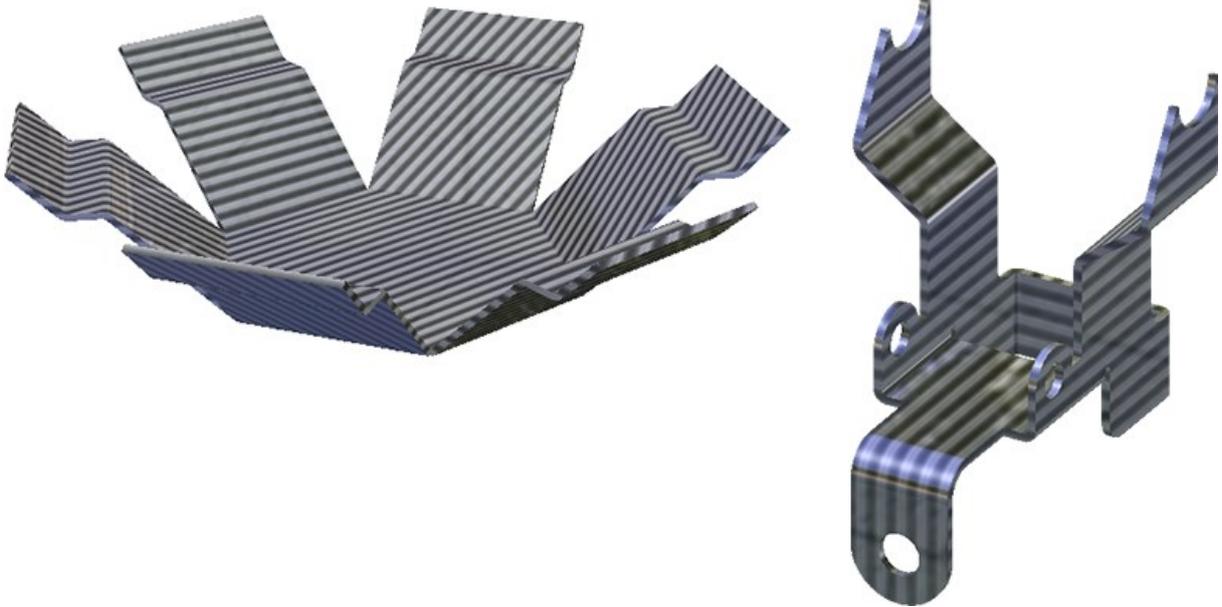
### Performance

HiCAD 2018 SP2 supports display lists for the graphic preview of polygon models. Especially if there are many polygon models, e.g. in element installation drawings, a performance increase by a factor of up to 3 can be achieved (with a suitable graphic card).

## Presentation of 3-D models

### Textures on Sheet Metal parts

The applying of textures to individual sub-parts of Sheet Metal parts has been optimized in such a way that the result will be consistent both with regard to the alignment of the texture and the transitions between neighbouring parts.



## Service Pack 1 2018 (V 2301)

### Exchange 3-D parts

#### Subsequent referencing of already linked detail drawings

This function searches, via the article master of the active part, a KRA file linked via HELIOS, and automatically exchanges the active part with the KRA file. In contrast to the functions **Exchange 3-D part, via document master** and **Exchange 3-D part, via article master**, no document or article selection and no placing of the exchanged part will be necessary here.

The function is particularly useful when you subsequently reference parts, for which a linked detail drawing already exists, in the assembly of the original model drawing. In this case it will be ensured that the part that exists in the detail drawing will be exchanged with the referenced part. HiCAD will also attempt to preserve the dimensions of the part to be exchanged.



#### Please note:

Here, "detail drawings" are not the detail drawings that are created with the **Drawing derivation** function on the **Drawing** tab, but manually created drawings which have either been created via **Drawing > Save/Reference > Part** and the **Detail drawing...** option, or by pasting of individual parts into a drawing file from the Clipboard. When using the Clipboard, please remember that dimensions will not be taken over.

When you call the function, HiCAD will check via the article master of the active part, whether a linked KRA file exists in the HELIOS database.

- If such a file is found, the active part will be automatically - without any further steps in the dialogue window being required - replaced with the part from the KRA file. In the process, HiCAD will attempt to preserve the dimensions of the active part. If this is not possible, the dialogue window for the updating of the dimension associativity will be displayed.
- If the linked document is no referenced part, the placing of the exchanged part will be requested.



#### Notes on dimension reconstruction:

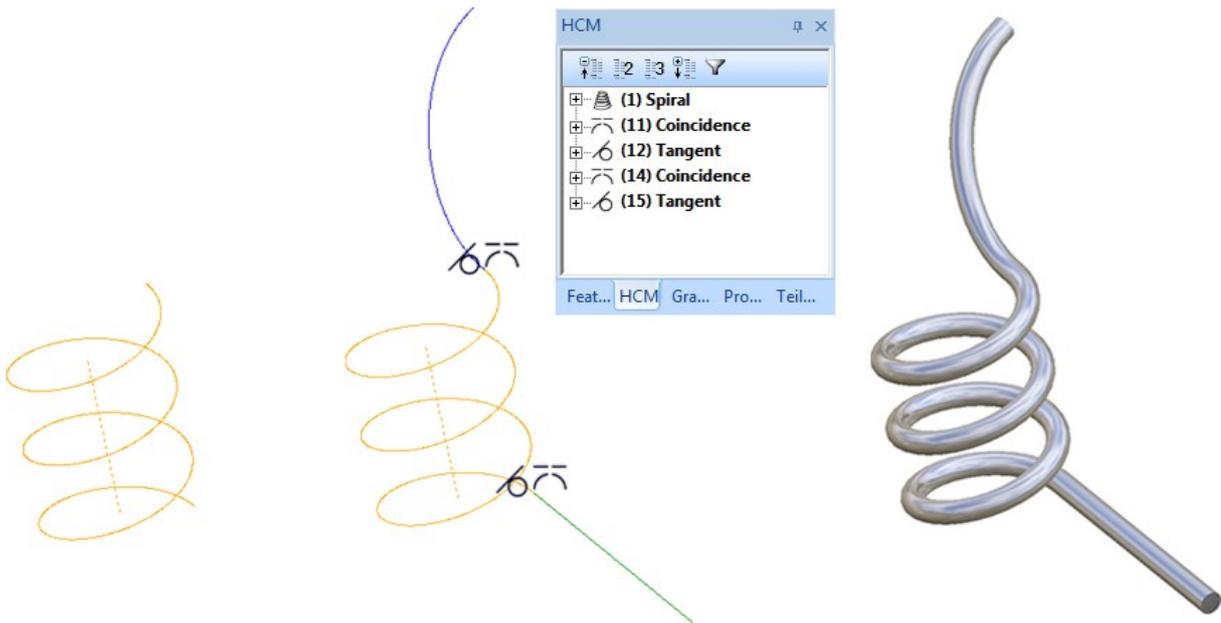
If a part has sub-parts and if the detail drawing has been created with a version older than HiCAD 2018 SP1, dimensions cannot be reconstructed, i.e. they cannot be assigned to the corresponding parts. In this case you should open the Configuration Editor, go to **System settings > Referencing** and activate the **Recover dimensions and weld symbols** before calling the **Exchange 3-D part, via DB link** function. The dimensions and weld symbols will then be assigned to the superordinate part.

## Sketches

### Sketches as guidelines for the "C-edge sweep" function

Spirals can also be used as guidelines for the **C-edge sweep** function. This is now even possible if further sketch elements have been attached to the start or end of the spiral. Please note however that in such cases these elements must have been linked to the start or end of the spiral via a **Coincidence** constraint!

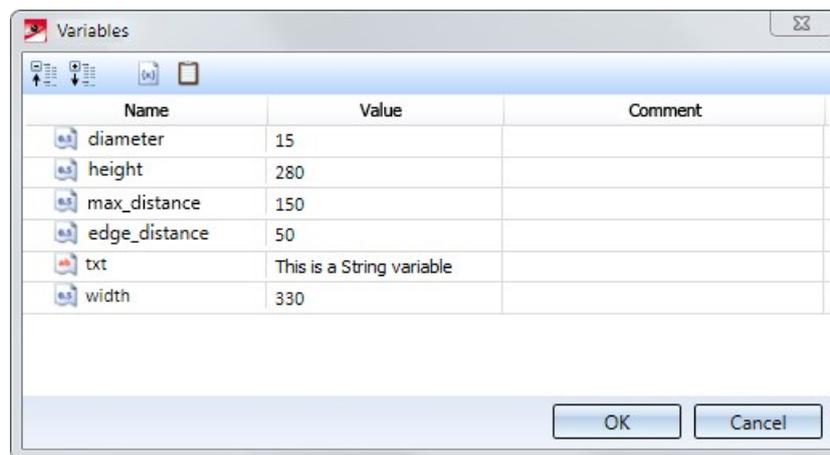
In the example shown below, further edges have been added both at the start and at the end of a spiral. The start point of the edges lies in the start (or end) point of the spiral, where **Coincidence** constraints have been assigned. Then, the **C-edge sweep** function was used to place a circular cross-section along this "expanded spiral".



## Tools

### New dialogue for part, view and feature variables

The dialogue of the Part variables  function has been largely adjusted to the one of the Part variables docking window.



The dialogues for view variables and feature variables have been adjusted accordingly, too.

### Show/Hide axes

In the **3-D Standard > Tools > Crossh.** > ... pull-down menu you can now find new functions that allow you to show and hide axes:



#### Hide axis - Individual

Hides individual axes. Select the axes you want to hide.



#### Hide axis - Group

Hides a complete group of axes that form a crosshairs. For instance, if you select one axis of a crosshairs of a sphere, all axes will be hidden.



#### Hide axis -All in active part

Hides all axes / crosshairs of the active part.



#### Show axis - All in active part

Shows all axes / crosshairs of the active part again.

The hiding of the axes is recorded in the Feature log in the form of an entry called **Hide crosshairs**.

If one of the **Hide axis** functions is applied to axes which have been automatically generated by a different processing (e.g. after insertion of a bore), this step will not be entered in the Feature log as a new feature, but as a sub-feature of the creation feature. In this way it is ensured that hidden axes will actually remain hidden after feature recalculations.

If you want to redisplay individual axes / crosshairs, right-click the **Hide crosshairs** entry in the corresponding feature and choose **Clear**. The corresponding feature entry will then be deleted.

To redisplay all axes / crosshairs of the active part in one step and simultaneously delete the corresponding feature entries, choose the **Hide axis - All in active part** function.

## Show/Hide isolated points

In the **3-D Standard > Tools > Point**  > ... pull-down menu you can now find new functions that allow you to show and hide points:



### Hide point

Hides individual isolated points.



### Hide points, active part

Hides all isolated points of the active part.



### Show points, active part

Shows all isolated points of the active part.

The hiding of isolated points will be recorded in the Feature log, in the form of an entry called **Hide isolated points**. In this way it is ensured that hidden isolated points will actually remain hidden after feature recalculations. Right-click the feature entry and choose **Clear** to remove it and redisplay the point.

To redisplay all hidden points of the active part in one step and simultaneously delete the corresponding feature entries, choose the **Show points, active part** function.

## Copy edges

The "Copy from part" function is no longer available. Instead, the selection menu of the **Copy edges**  function (open with a right-click after calling the function) now contains the new



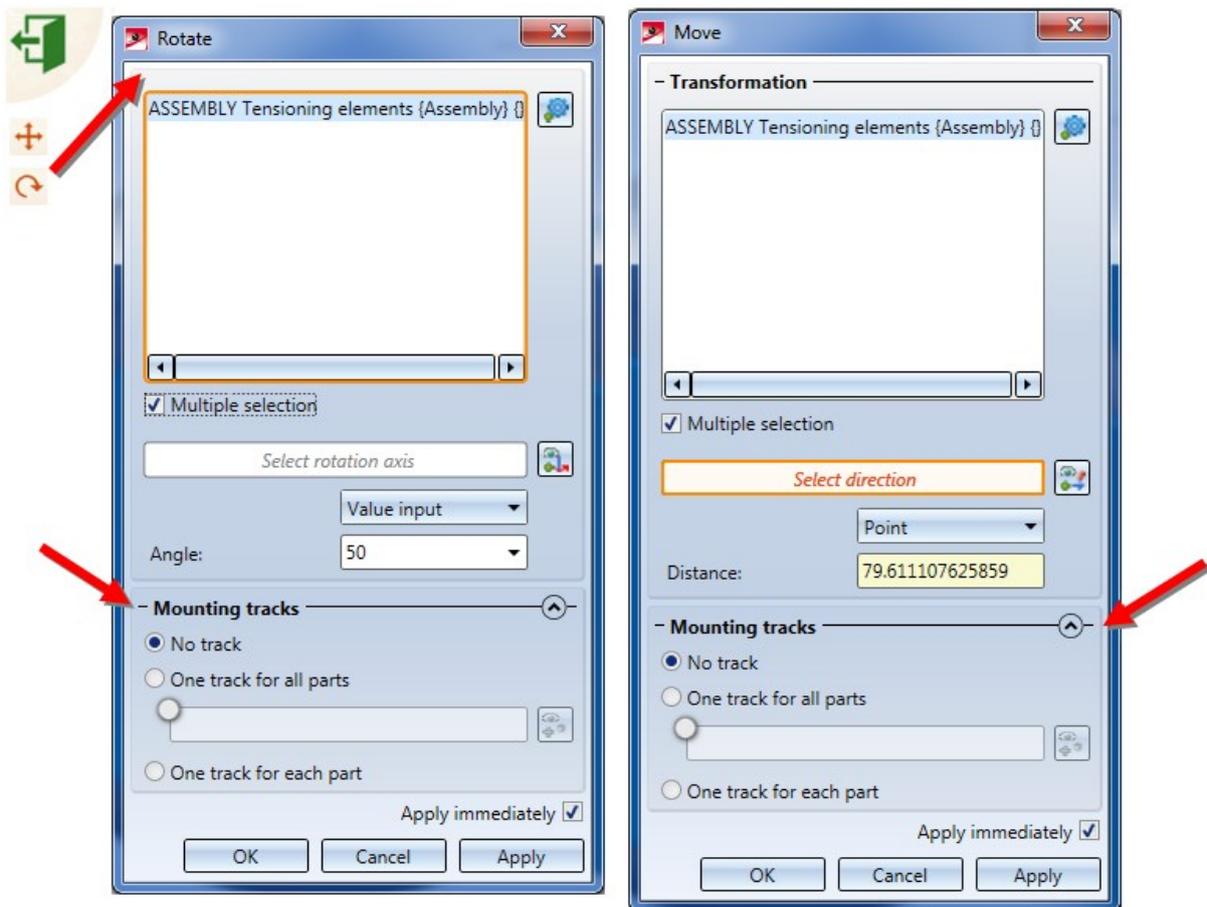
### All lines or edges of part

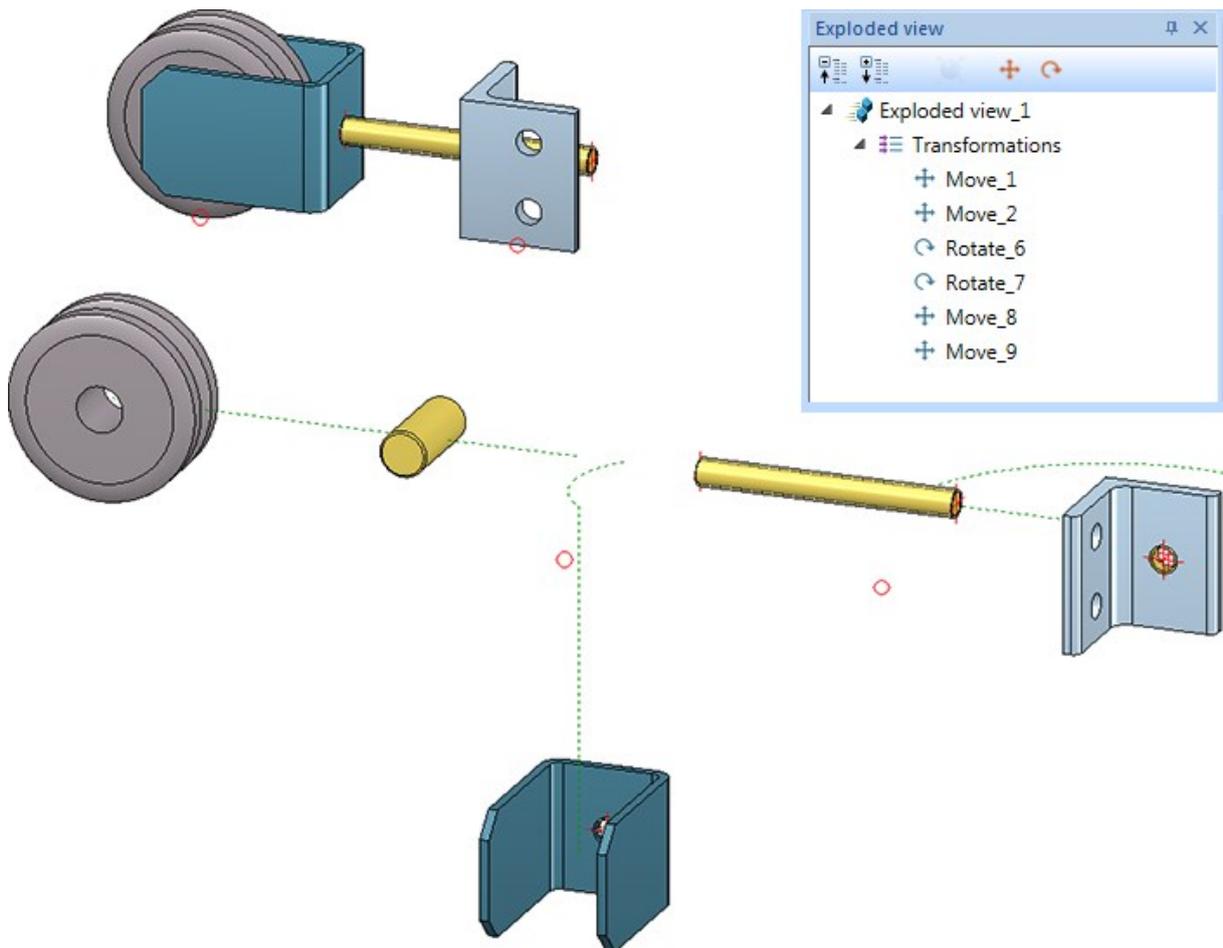
option. You can use this option to copy all lines or edges of any 3-D part into the active part. In contrast to the old "Copy from part" function, the new function enters a feature called **Isolated edges** into the Feature log that contains the inserted edges.

## Views

### New options for exploded views

- For exploded views you now have the additional option to rotate parts.
- You can now also add automatically created mounting tracks in an exploded view. The Move and Rotate dialogue windows contain new checkboxes, enabling you to specify if you want to create mounting tracks and, if, so specify the type of mounting tracks.
- Exploded views can now also be copied like other views. In the process, all exploded view data (transformations) will be copied as well.





Example of an exploded view with displacements, rotations and mounting tracks

### Show/hide processing planes in views

The former "Show/hide edges in view" function at **Views > Properties** has been renamed to **Show/hide elements in**



**view**. New in the dialogue window of the function is the **Processing planes** checkbox. By activating or deactivating the checkbox you can show or hide processing planes of the active view in one step.

Alternatively, you can access the functions in the context menu for views at **Show/hide parts > Others**:



**Hide processing planes**



**Show processing planes**

To open the context menu for views, right-click the pink dashed frame around the drawing object.

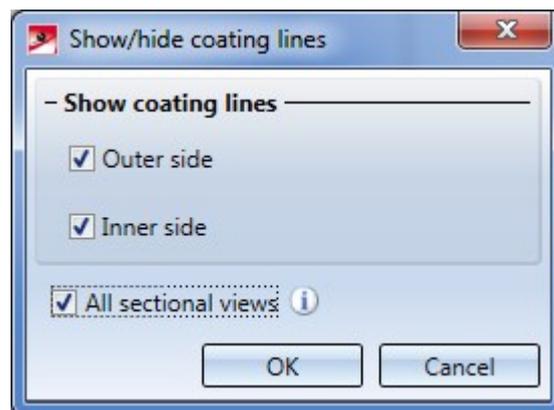


In shortened views, processing planes are not being shown - regardless of the setting in Show/Hide elements in view.

## Sectional views - Hide/Show coating lines

The "Delete coating line" function has been replaced with the **Show/Hide coating lines**  function. You find the function in the context menu for coating lines.

Coating lines can be hidden or shown in the active view or in all sectional views. One distinguishes between coating lines on the inner side and the outer side. When you call the function, the following dialogue window will be displayed:



With the settings shown above, the coating lines will be shown in all sectional views on the outer side and inner side.

If all coating lines have been hidden, the lines can be redisplayed again via the context menu for sheets: In the sectional view, right-click the sheet and choose **Properties > Coating line**.

## Dimensionings / Annotations

### Variable dimensioning - Angular dimensioning with X-, Y- and Z-axis or a reference axis

Via the variable dimensioning you can now also determine dimensions of the type

- Angular dimension with X-, Y- and Z-axis, and
- Angular dimension with reference axis

by selecting an edge or a surface.

If you choose **Angular dimension with reference axis** as dimension type, a reference axis needs to be specified. In such cases the **Select reference axis** button will be displayed in the dialogue window, and HiCAD will ask you to specify a reference axis - either by identifying an edge or by specifying 2 points. The dimension will then be attached to the crosshairs.

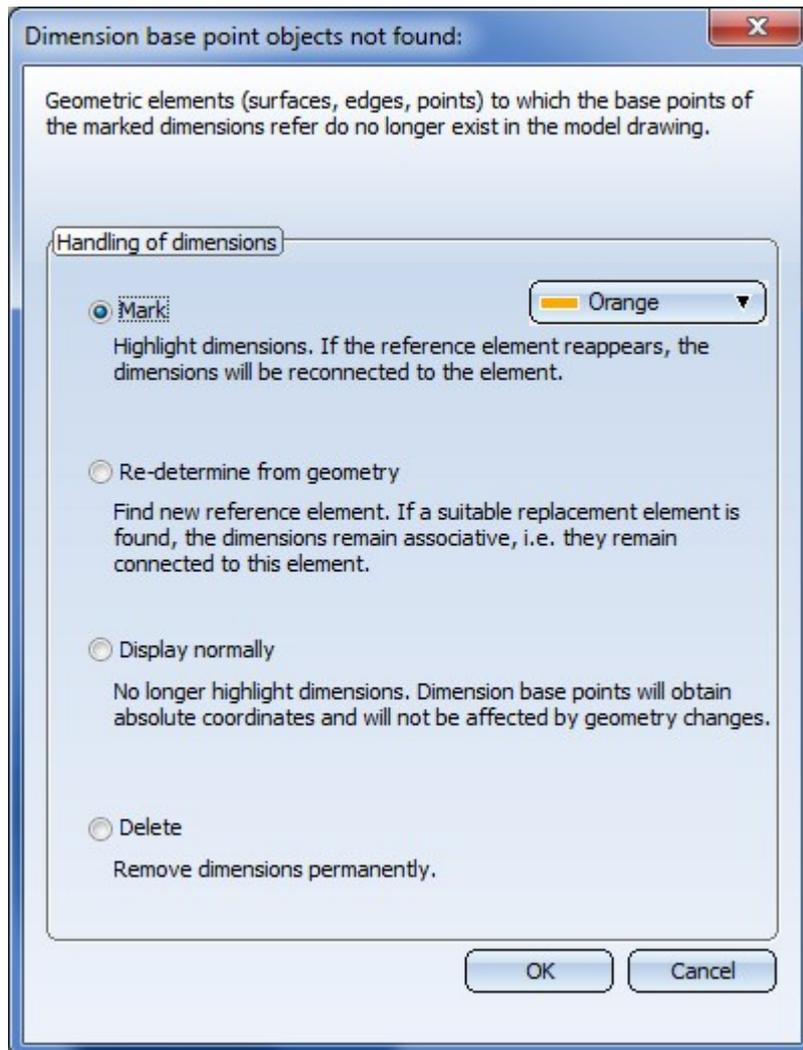
The selected reference axis will remain active until you select a different reference axis or end the function. This means that if you choose **Angular dimension with reference axis** in the dialogue again, the previously selected reference axis will be used again. If you want to use a different axis, click the **Select reference axis** button and specify a different axis.

### Auxiliary dimension texts

The **Change language of text**  function on the **2-D Dimensions + Text** tab can now also be applied to auxiliary texts in 3-D dimensions.

## Update dimensions

Dimension base points refer to geometric elements, e.g. surfaces, edges or points. If these elements are removed, the dimensions will lose their reference and will no longer be associative. In such cases the **Dimension associativity - Update** functions can no longer be carried out properly, and the **Dimension base point objects not found** dialogue window will be displayed. The dialogue window and the corresponding settings in the Configuration Editor at **System settings > Annotations > Dimensioning, 3-D > Dialogue default setting: Handling of non-associative dimension base points** have been revised in SP1.



	Description	Value	Comment
System settings	Decimal separator	Comma	Separation of decimal places
Sketch HCM	Rounding	Mathematical	Rounding of dimension figure
Itemisation	Update dimensions in views	Displayed views	In which views should dimensions be updated?
Processing plane	Dialogue default setting: Handling of non-associative dimension base points	Mark	Delete, Mark, Set to non-associative, or Re-determine from geometry
Sketches	Marking colour	6:  Orange	Marking colour for non-associative dimensions
Start configuration	Behaviour when changing scale	Scale	Behaviour of dimension line in case of scale changes
Directories	Hide background lines when printing shaded views	<input type="checkbox"/>	Hide white rectangular backgrounds behind dimension figures in shaded views when printing?
Load/Save	Correction when deleting workshop drawing dimensions	<input checked="" type="checkbox"/>	Adjust dimensioning rules of view group when deleting automatically created workshop drawing dimensions
Data save			
Identification			
Referencing			
Annotations			
Text			
Dimensioning, 2-D			
Dimensioning, 3-D			
HCM symbol			
Annotation tags			

## Itemise part

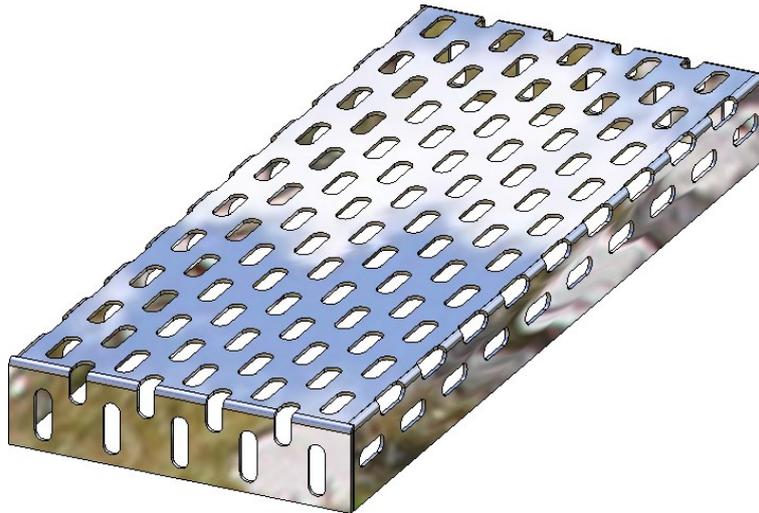
The function Itemise part  now proceeds as Itemisation, individual parts  **1 ... n** (via Drawing > Itemisation/Detailing > Itemisation). This means i.a. that conflicts with other positions can be identified.

## Standard Parts / Standard Processings

### Hole Patterns on sheets after bending simulation

Now you have the option to integrate hole patterns in a sheet after adjusting the bend zone. Thus, the hole pattern (analogically to bores) can be applied even in accordance with multiple flanges and bending zones.

In the wake of this enhancement, the requirement that the processed area must be on the whole side, became no longer necessary.

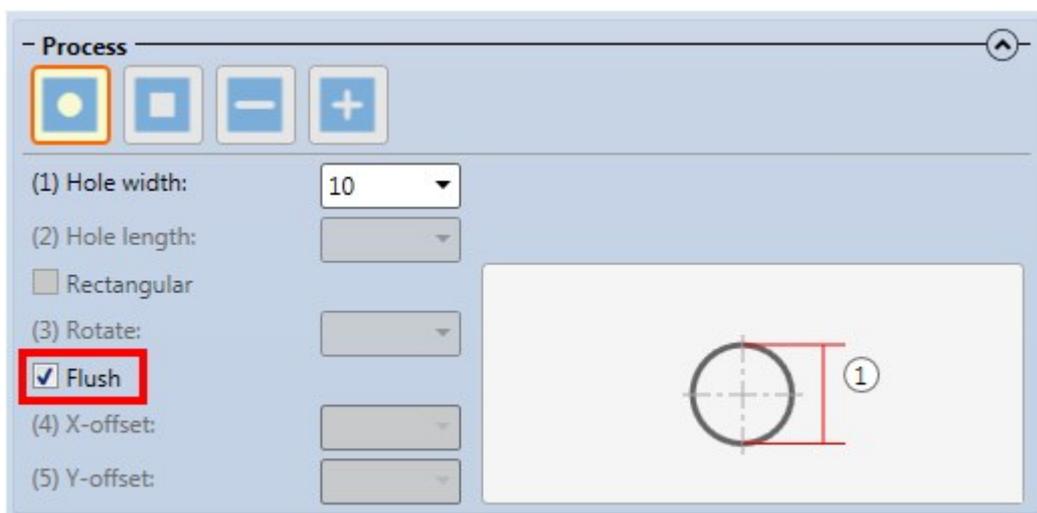


Another new enhancement is that the usual features, such as **Process sketch** and **Create new sketch in plane** are also accessible via the dialogue window **Hole pattern** for working with processed and omitted areas.

Furthermore, the function **Hole Pattern** is only accessible via the Ribbon **3D-Standard**; the entry on the ribbon **Sheet Metal** has been removed. However, the function is still applicable on sheet metal.

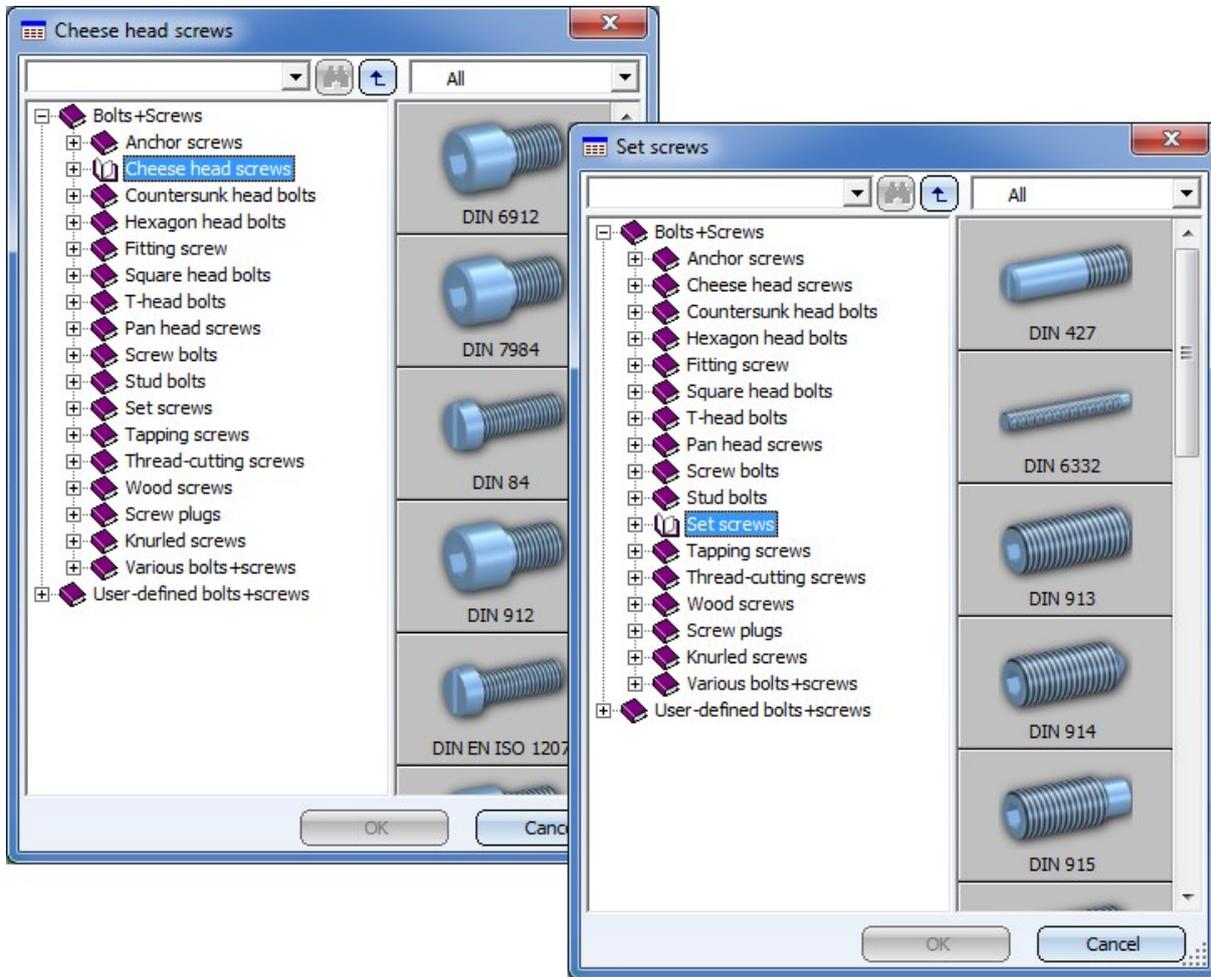
### Hole patterns - Flush vs. central placing of hole patterns

Previously the start hole for insertion of hole patterns was always placed flush on the right and above the origin of the Pattern CS. As of version HiCAD 2018 SP1 the dialogue window offers the additional **Flush** checkbox. If you deactivate this checkbox, the start hole will be placed centrally into the origin.



### New preview images

The tables of the HiCAD Standard parts catalogues are to contain new preview images and descriptive detail graphics. In the first step these were implemented for the **Fasteners** catalogue. The same which will soon be done for the other catalogues.



## Major Release 2018 (V 2300)

### Standard Processings

#### Thread representation in shaded views

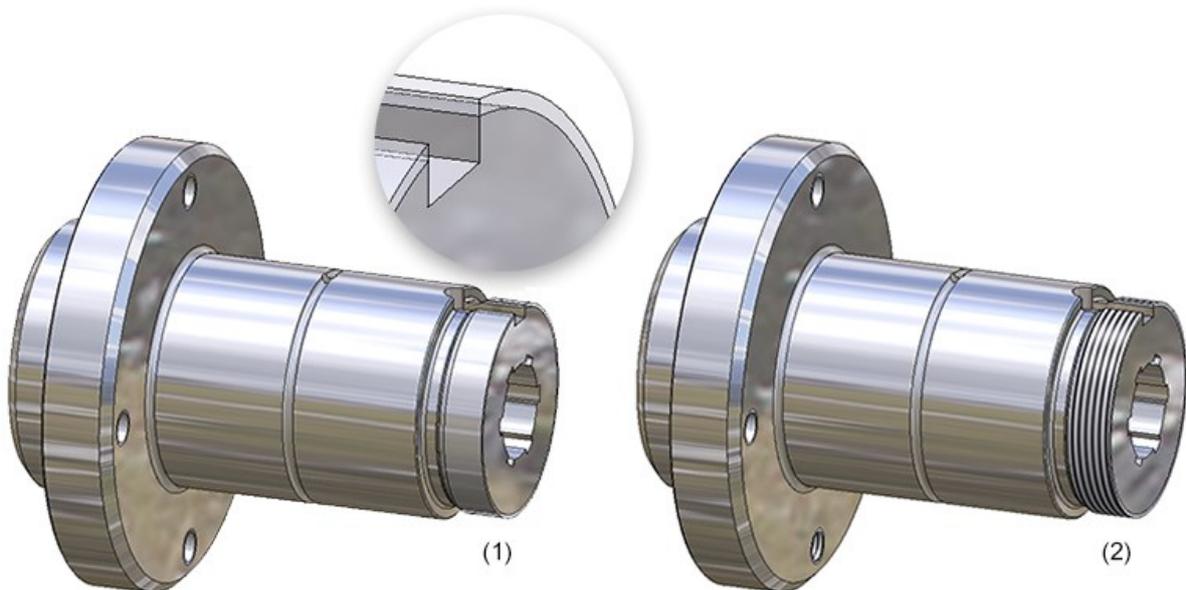
Threads in shaded views can be represented either with thread texture or transparently. The representation can be predefined via the configuration management under **Visualisation > Views** with the parameter **Thread representation in shaded views**.

- **With thread texture**

This setting - the ISD-side default setting - depicts pitch and chirality of threads realistically. However, this only concerns the shaded view and not representations such as glass model, hidden line or quick hidden line representation.

- **Transparent**

The threads will be depicted transparently as usual.



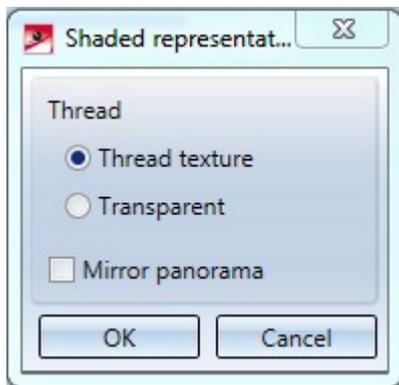
Flanged shaft - (1) Transparent view, (2) With thread texture

Irrespective of the default setting in the Configuration Editor, the thread representation in shaded views can be determined separately for each view. For this, use the function Shaded representation via **Representation > Views > Shaded repr...**



#### Shaded representation (Active view)

With this function, the active view of thread representation can be changed. For that, the dialogue window **Shaded representation** is shown:



Choose the desired representation by clicking the respective option and close the window by clicking **OK**.



#### Shaded representation (All views)

This function complies with the function for **Active view** but affects all views of the model.

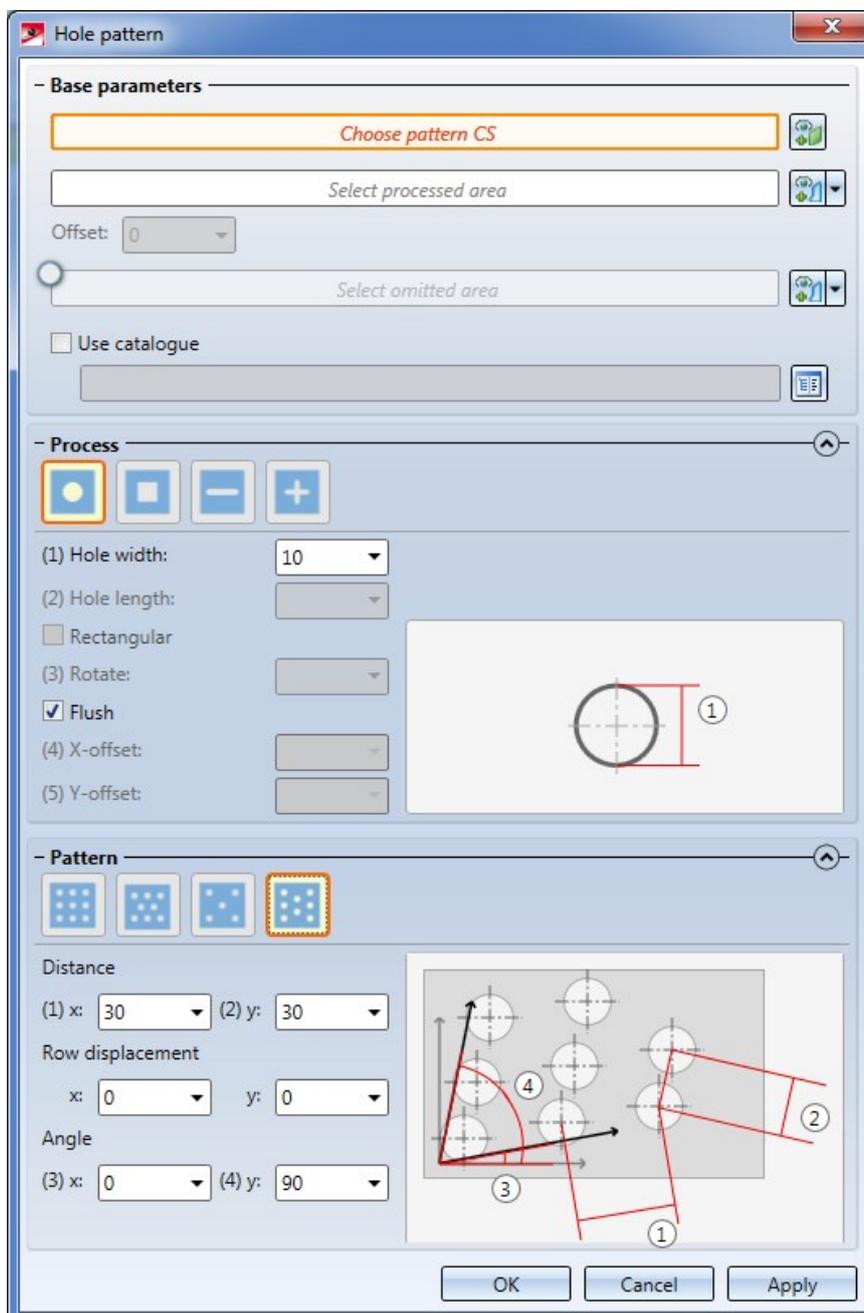


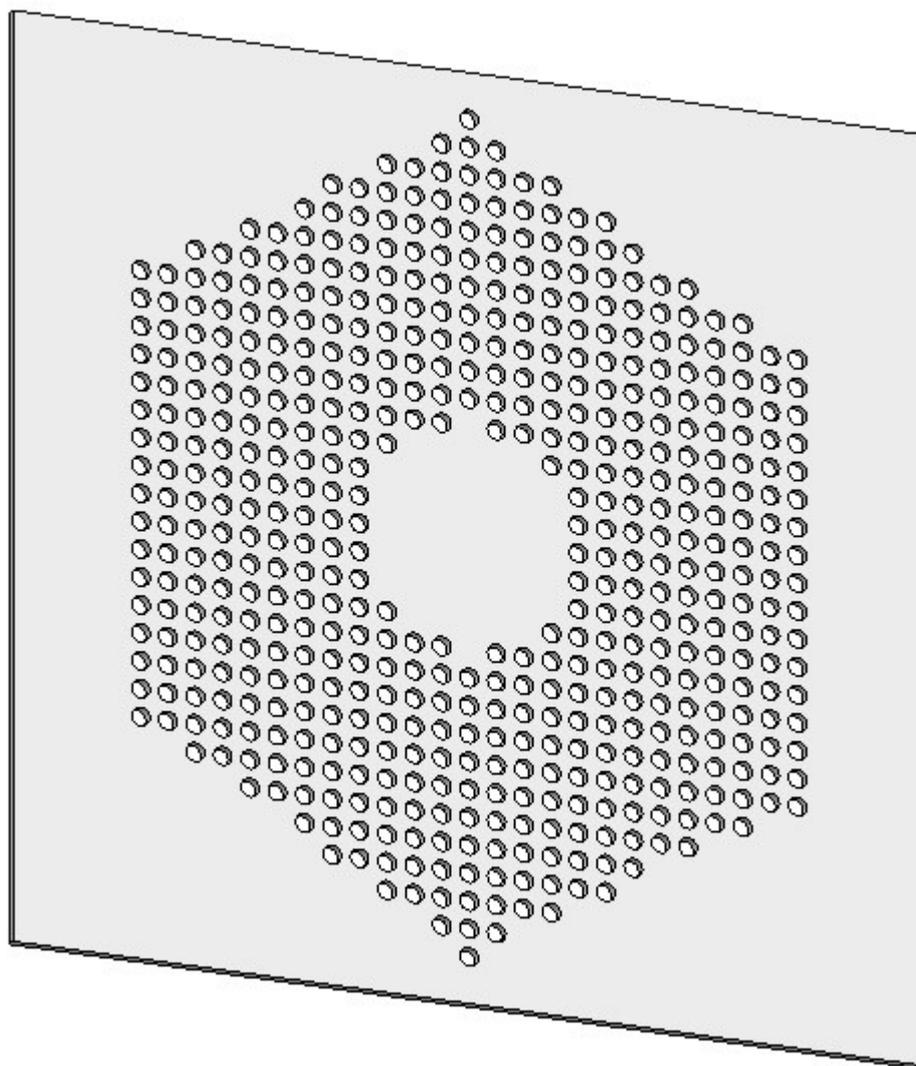
Only one thread will be equipped with texture in case of multiple insertion via grid.

## Hole patterns

HiCAD 2018 allows a generation of sheets and plates with hole patterns according to DIN 4185-2 and DIN 24041. These are sheets/plates with identical holes arranged in regular patterns, which can, for instance, be realized by punching or drilling tools. Possible hole types are round holes, square holes, slots and special holes. The holes can be arranged in straight, staggered, diagonally staggered or freely staggered rows. You have a choice between the selection of predefined holes and hole patterns according to DIN from a catalogue, or the use of self-defined holes and hole patterns. In the process, the holes will only be placed within a freely defined area if they stay within its limits. In addition, an area for the omission of holes can be defined.

You find the function at **3-D Standard > -Standard Processings > Bore**  **> Hole pattern**  and **Sheet Metal > Further Tools > HoleP** .





## Create and process parts

### Optimise part - No longer available for individual surfaces



The Optimise part function can now no longer be applied to individual surfaces, but only to the active part or to the entire model drawing.

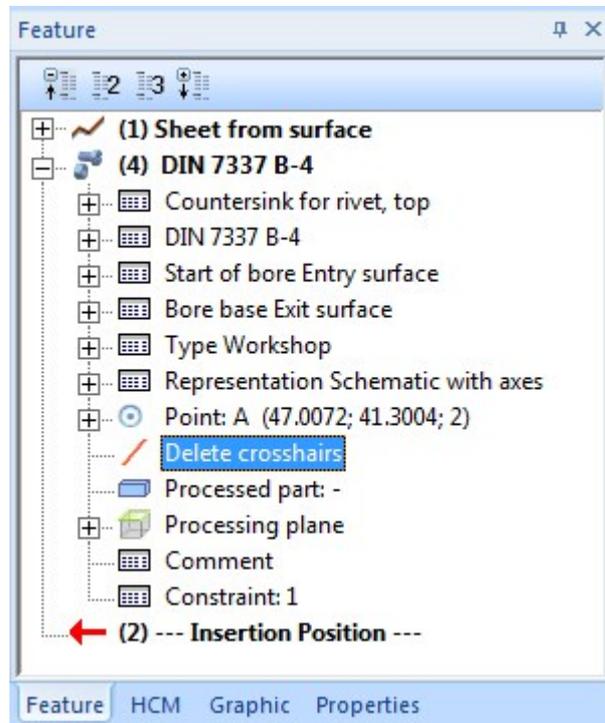
### Create parts - Article number with upper case / lower case letters

Various part creation functions could only show the article number in upper case letters. The functions shown below now also allow a spelling of the article number in upper or lower case letters:

3-D Standard		3-D FFS	
	Solid primitive		Extrude cross-section
	Extruded solid		Rotate cross-section
	Revolved solid		Move cross-section
	C-edge sweep		Transition part
			Connect cross-section
			Surface from curve mesh
			Surface from 3-D polyline frame
			Patch

## Delete crosshairs - Now in Feature log

If one of the **Delete crosshairs** functions is applied to a crosshairs that has been automatically created by a different function, e.g. the insertion of a bore, this deletion will not be entered as a new feature, but as a sub-entry called **Delete crosshairs** beneath the creation feature:



This change will also be preserved in case of a recalculation. To delete this processing step later, right-click the **Delete crosshairs** entry and choose **Clear**.

## Deleted isolated points considered in Feature log

Up to HiCAD 2017 the deletion of isolated points was not considered in the Feature log, causing an unwanted restoring of deleted points in the drawing after feature recalculation.

From HiCAD 2018 onwards the deletion of isolated points will also be considered in the log of their respective creation Feature. If an isolated point is deleted in the drawing, the corresponding entry will also be deleted in the Feature log. After a recalculation, the deleted point will remain deleted.

Please also note that you can now also directly choose during development of sheets whether you want isolated points of the Sheet Metal part to be shown or hidden. The setting concerns only the isolated 3-D points that the Sheet Metal part provides. The display of these points can be changed in the Feature of the sheet development. If an isolated point of the Sheet Metal part is deleted, this point will also be removed from the development when it is synchronized with the part. Isolated points that have been created in the development subsequently are always visible.

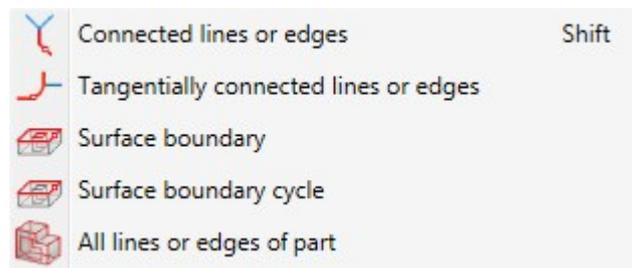
## Copy edges

The **3-D Standard > Tools > Edge** > ... pull-down menu has been revised.

The functions

- Copy, Individual
- Copy, In rectangle
- Copy group

have been combined into one new function called **Copy edges** . After calling the function, right-click to open a context menu with various options:

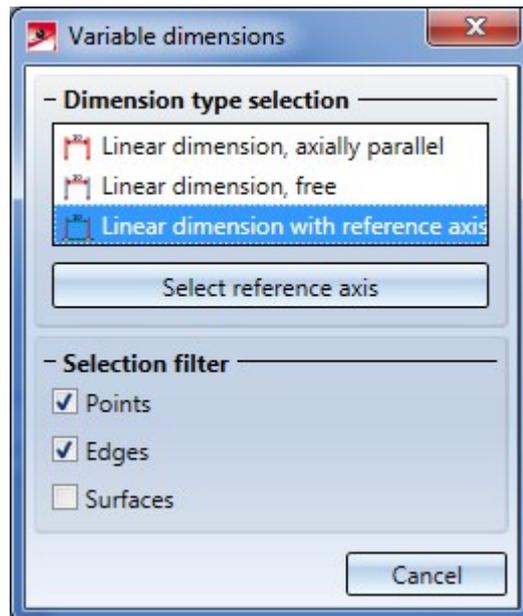


The new function is also supported by Feature Technology: In contrast to the old functions, for edges that are taken over with the new function a Feature called **Isolated edges** will be created.

## Dimensioning / Annotation

### Variable dimensioning - Linear dimension with reference axis

The Variable dimensioning function now also allows the detection of **Linear dimensions with reference axis**. If you choose this dimension type, you need to specify a reference axis. To do this, click on the **Select reference axis** button. HiCAD will then prompt you to select a reference axis, either by identifying an edge or specifying 2 points. The dimension will then be attached to the cursor.

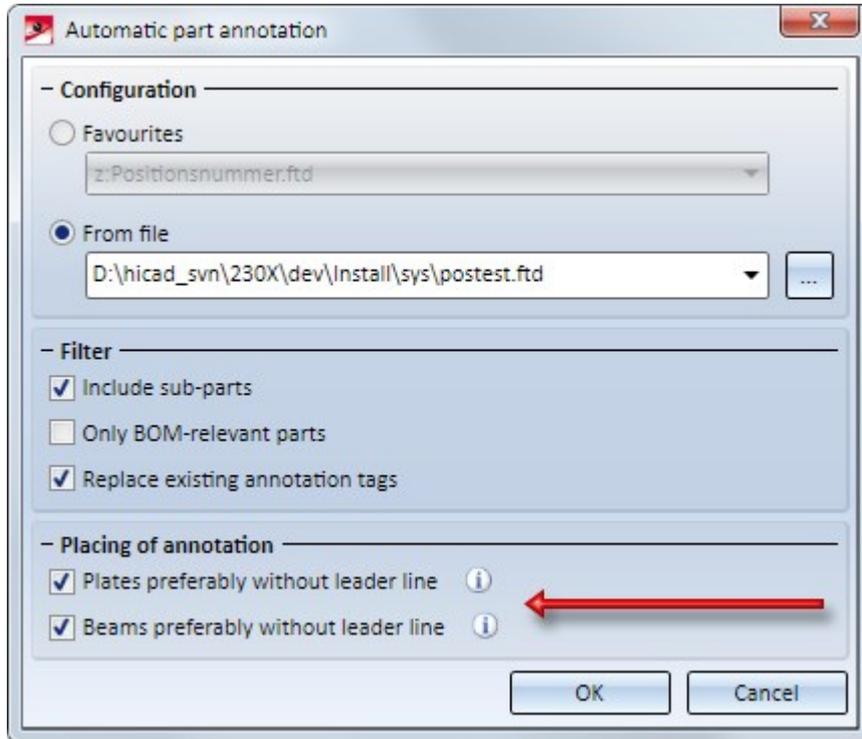


The selected reference axis will remain active until you select a different reference axis or end the function, i.e. if you select the dimension type **Linear dimension with reference axis** in the dialogue window again, the previously selected reference axis will be used. If you want to select a different axis, click on the button and define the new axis.

## Automatic part annotation - Beams and plates

The Automatic part annotation  now allows you to determine whether the annotations for beams and plates are to be applied with or without leader line.

For this purpose the dialogue window has been expanded accordingly:

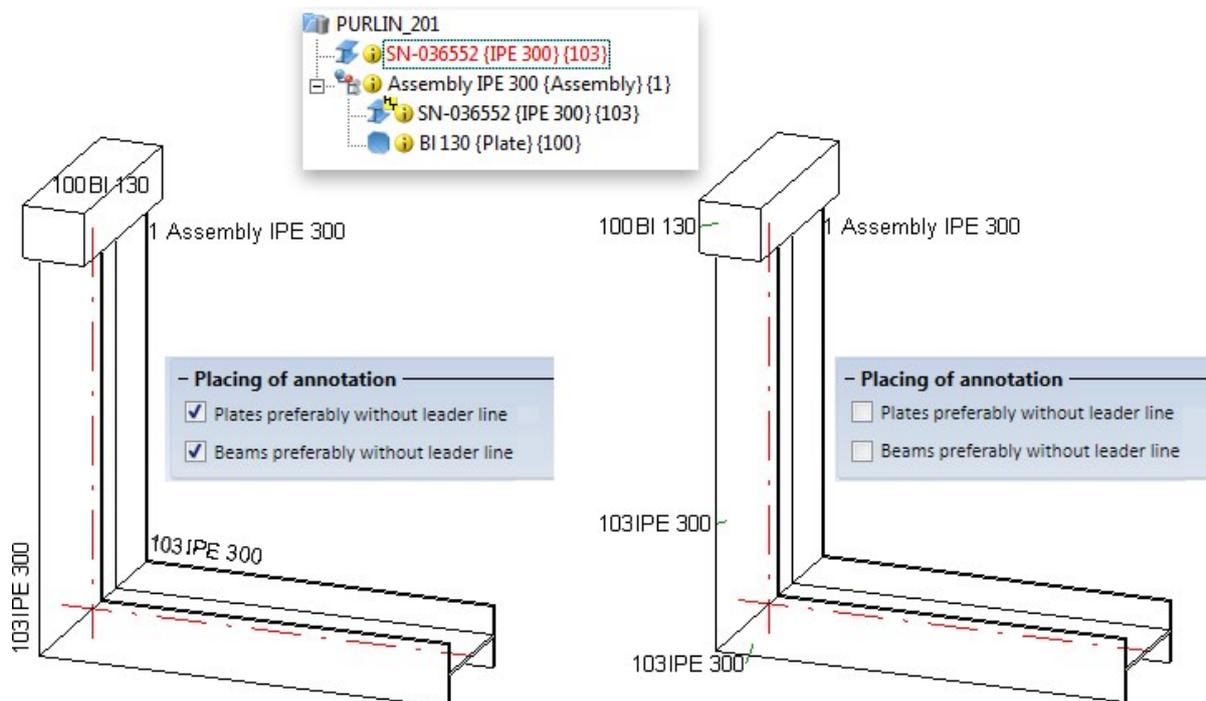


- **Plates preferably without leader line**

If this checkbox is active, HiCAD proceeds as follows: If there is enough space on the part, the annotation for the plate will be placed directly on the part. If there is not enough space, the annotation will be created with a leader line.

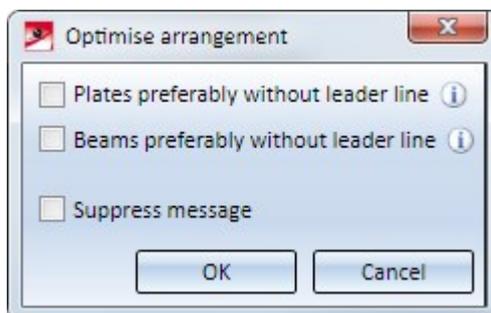
- **Beams preferably without leader line**

If this checkbox is active, HiCAD proceeds as follows: If there is enough space in the vicinity of the part, the annotation will be placed along the beam as shown below. If there is not enough space, the annotation will be created with a leader line.



## Optimize arrangement - Beams and plates

Like the **Automatic part annotation** function, the **Optimize arrangement**  function now allows you to determine whether the annotations for beams and plates are to be applied with or without leader lines.



For plates and beams you can determine (as with the automatic part annotation) whether the annotations for beams and plates are to be applied with or without leader line.

- **Plates preferably without leader line**

If this checkbox is active, HiCAD proceeds as follows: If there is enough space on the part, the annotation for the plate will be placed directly on the part. If there is not enough space, the annotation will be created with a leader line.

- **Beams preferably without leader line**

If this checkbox is active, HiCAD proceeds as follows: If there is enough space in the vicinity of the part, the annotation will be placed along the beam as shown below. If there is not enough space, the annotation will be created with a leader line.

The last specified settings that have been confirmed with **OK** will be the default settings when the function is called again.

## Views

### Exploded view

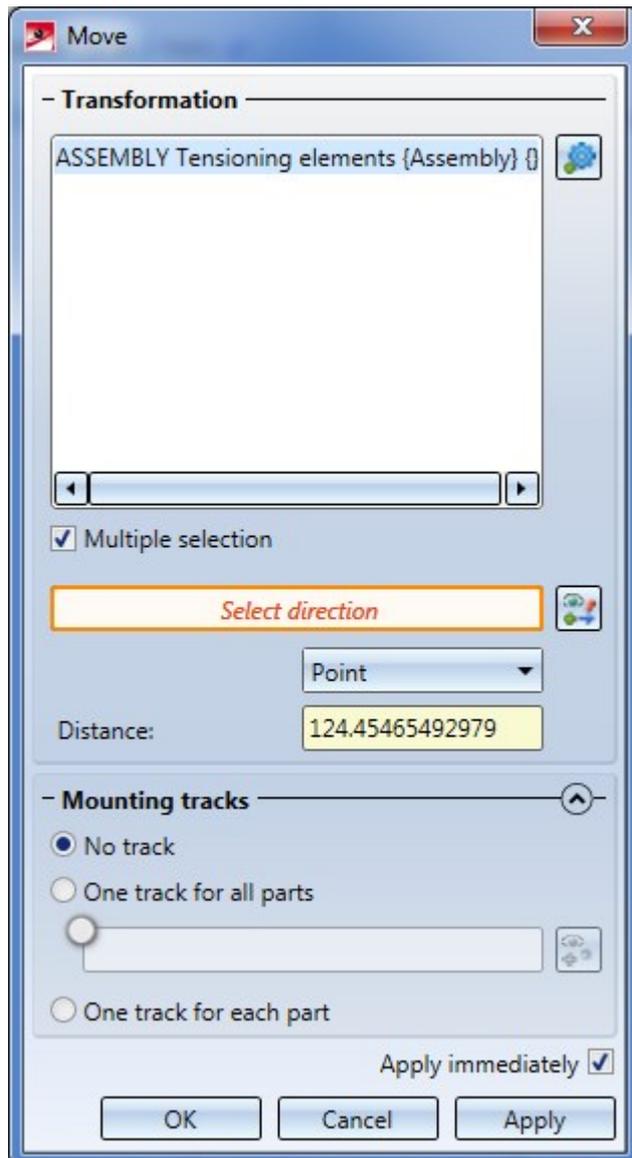


The **Exploded view** function has been revised completely.

This function enables you to display the active view as a so-called "exploded view". This type of view shows the part and assemblies of the model drawing in "dismantled" state, with the dismantling being based on a translatory displacement of the parts. Exploded views are saved together with the geometry in the .SZA file. Since all part displacements are recorded in an explosion log, they can be changed subsequently if required.

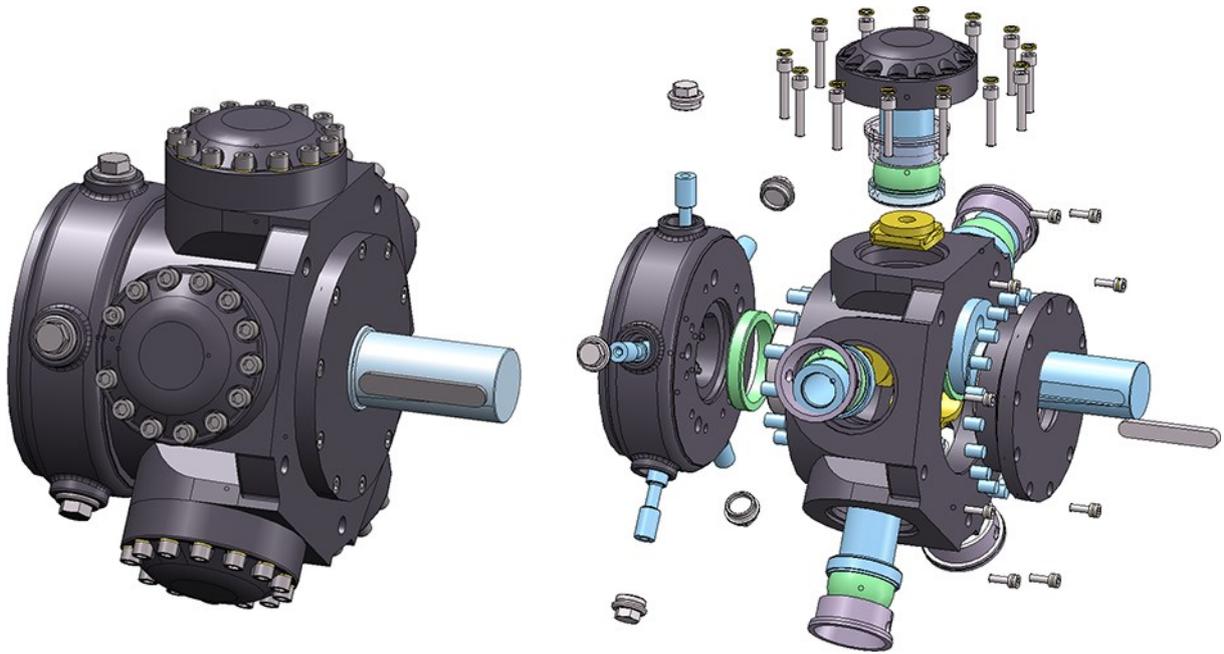
Proceed as follows to define an exploded view:

1. Load the model drawing of which you want to create an exploded view.
2. Create a new view for the exploded representation and activate this view.
3. Call the **Exploded view** function.
4. HiCAD switches into the **Exploded view** mode. In this mode you can define the required displacements. For this purpose, the **Exploded view** dialogue window will be displayed.

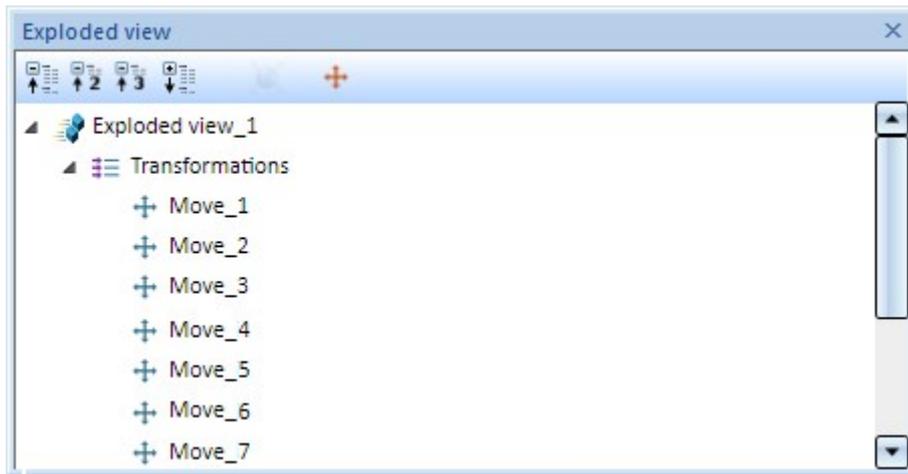


5. After defining the Exploded view, you can exit the **Exploded view** mode again.

All displacements of the exploded view will be logged. The log is shown - similar to the Feature log - in the **Exploded view** docking window. Via this docking window you can change exploded views subsequently.



Example of an exploded view (some of the lids have been excluded here)



The "Exploded view" docking window

## Display of coatings in sectional views

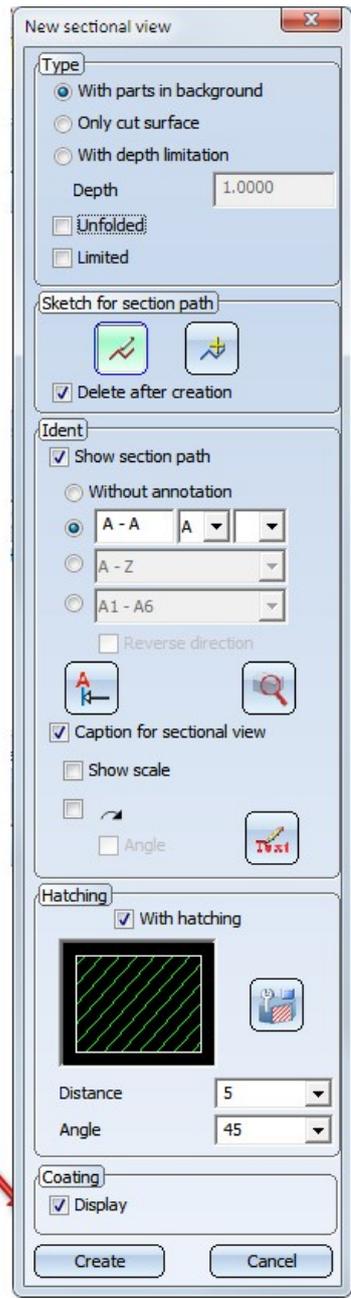
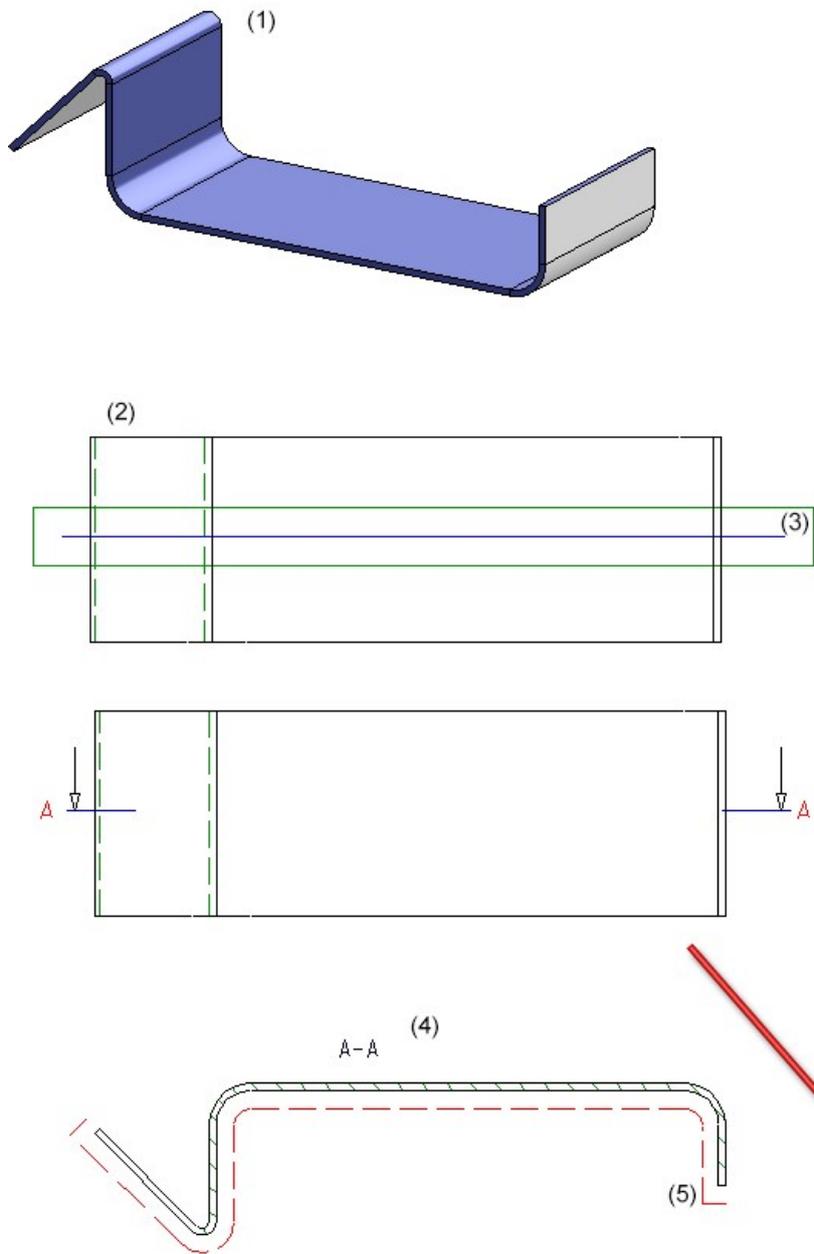
The dialogue window for Sectional views has been expanded.

For Sheet Metal parts you have now the option to specify whether the coating in the sectional view is to be shown or not, by activating or deactivating the **Display** checkbox beneath **Coating** in the dialogue window. The coating is displayed in the form of an offset edge, the so-called "coating line". The settings for the representation of the coating line can be found in the Configuration Editor at **Drawing > Annotations > Coating line in sectional view**.



The representation of the coating line can also be changed subsequently. To do this, right-click on the coating line in the required view and choose the desired function in the **Coating symbol** context menu:

Function	
	<p><b>Change coating line parameters</b></p> <p>Use this function to change the colour, line type and layer of the coating line. When you call the function, the <b>3-D edge parameters</b> dialogue will be displayed for this purpose.</p>
	<p><b>Change offset distance</b></p> <p>Use this function to change the offset distance, i.e. the distance between coating line and cut edge.</p>
	<p><b>Delete coating line</b></p> <p>Deletes the coating line.</p>



Sheet Metal part (1) with outside coating; Top view (2) with cut edge (3), Sectional view (4) with coating line (5)



**Please note:**

- The offset calculation for the coating lines is always carried out in the screen plane, i.e. the lines are no "fixed", permanent 3-D polylines, but will be updated upon rotation of the view.
- In dashed representations the dashing takes place segment-wise, but not over the entire length.

## Shaded representation of threads and panorama mirroring

At **Views > Representation > Shaded**  > ... you can find the following new functions:



### Shaded representation (Active view)

Use this function to change the representation of threads and switch panorama mirrorings on or off in active views.

- **Thread**

Here you can specify whether you want thread to be displayed with thread texture or transparent by activating the desired option.

- **Mirror panorama**

HiCAD offers the option to simulate environments by assigning a panorama to a Sheet or Model area. The surfaces of shaded models will then mirror this panorama, i.e. their environment. Activate the **Mirror panorama** checkbox to switch this effect on, deactivate it to switch it off again.



### Shaded representation (All views)

This function corresponds to the same-named function beneath **Active view**, with the difference that it affects all views of a model drawing.

In this way, panorama mirrorings can now be switched on or off separately for each view.

The formerly available functions **Mirror panorama**  and **Do not mirror panorama**  are no longer available in HiCAD 2018.

Others

**Setting options from system file ref3d\_attr\_akt.dat moved to Configuration Editor**

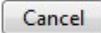
If referenced parts have part attributes which are not to be updated on a regular basis, the synchronization can also take place in a specific, targeted way. In previous versions this behaviour was determined in the system file **ref3d\_attr\_akt.dat**, and from now on in the Configuration Editor at **System settings > Referencing > Synchronization of attributes**.

 Synchronization of attributes    Table     Part attributes that are NOT always to be updated. Unlisted attributes will be updated for main parts and sub-parts.

Click the  symbol to activate a table in which you can specify for each part attribute

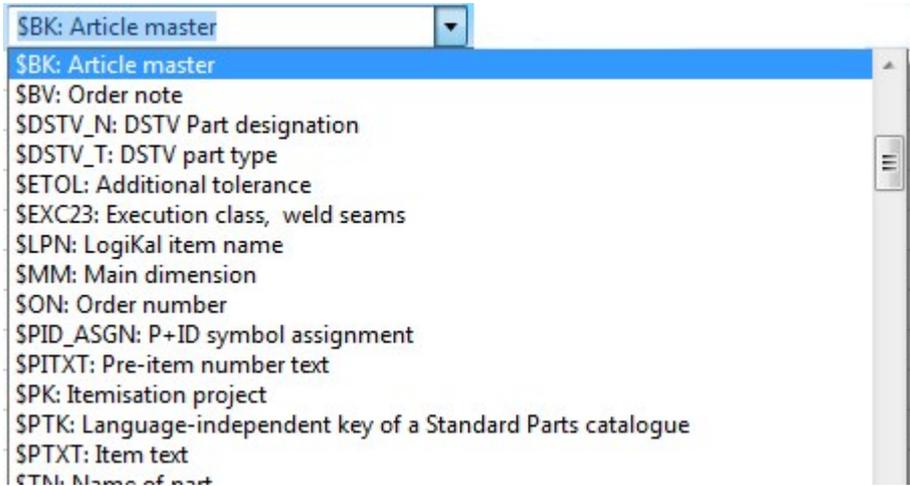
- whether you want it to be updated only for sub-parts,
- whether you want the settings to be taken from the Configuration Editor or
- whether you never want it to be updated.

Attribute synchronization options				
<input checked="" type="checkbox"/> Active	Name	Data type	Synchronization option	Comment
<input checked="" type="checkbox"/>	NAME	String	Never	
<input checked="" type="checkbox"/>	\$17: Surface per length	Double	From configuration	
<input checked="" type="checkbox"/>	%02: Item No.	Integer	From configuration	
<input checked="" type="checkbox"/>	#BR	Integer	From configuration	
<input checked="" type="checkbox"/>	\$14: Angle 1 of section schema	Double	From configuration	
<input checked="" type="checkbox"/>	%01: Qty. per part	Integer	From configuration	
<input checked="" type="checkbox"/>	#SR: BOM-relevant	Integer	Never	
<input checked="" type="checkbox"/>	\$BB: Article No.	String	From configuration	
<input checked="" type="checkbox"/>	\$23: Material length	Double	From configuration	
<input checked="" type="checkbox"/>	#PI	Integer	Sub-parts only	
<input checked="" type="checkbox"/>	\$16: Weight per length	Double	From configuration	
<input checked="" type="checkbox"/>	%03	Integer	From configuration	
<input checked="" type="checkbox"/>	\$11: Quantity 1	Double	From configuration	
<input checked="" type="checkbox"/>	%04	Integer	From configuration	
<input checked="" type="checkbox"/>	\$10: Surface	Double	From configuration	
<input checked="" type="checkbox"/>	%05: Part-Item-ID (Model ID)	Integer	From configuration	
<input checked="" type="checkbox"/>	TRANS_INFO: HELiCON Transfer info	String	Never	
<input checked="" type="checkbox"/>	\$13: Quantity 3	Double	From configuration	
<input checked="" type="checkbox"/>	%06: Total quantity	Integer	Never	
<input checked="" type="checkbox"/>	\$12: Quantity 2	Double	From configuration	
<input checked="" type="checkbox"/>	%07: Auxiliary body ID	Integer	From configuration	
<input checked="" type="checkbox"/>	\$01: Designation 1	String	From configuration	
<input checked="" type="checkbox"/>	%08: Assembling ID for bolts+screws	Integer	From configuration	
<input checked="" type="checkbox"/>	%09: Set ID for boltings	Integer	From configuration	
<input checked="" type="checkbox"/>	%10: Part type	Integer	From configuration	

ISD default settings (detail)

For this to happen, a row that is composed as follows must exist for the corresponding part attribute:

<input checked="" type="checkbox"/> Active	Name	Data type	Synchronization option	Comment
<b>Active</b>	<p>Via the checkbox of this column you determine whether the synchronization is to be carried out or not.</p> <p>If you want to activate or deactivate all check boxes in one step, then activate or deactivate the check box in the column heading, e.g. <input checked="" type="checkbox"/> <b>Active</b>.</p> <p>As an alternative, you could also use the context menu's function for activation by clicking the right button on a table entry.</p>			
<b>Name</b>	<p>Here you enter the name of the attribute. Attributes predefined by the ISD can also be chosen via a selection box. To do this, double-click on the corresponding row and choose the attribute from the list.</p> 			
<b>Data type</b>	<p>Here you select the attribute type (double-click on the row):</p> <ul style="list-style-type: none"> <li>■ Integer</li> <li>■ Double</li> <li>■ String or</li> <li>■ String (ANSI)</li> </ul>			
<b>Synchronization option</b>	<p>Here you select the type of the synchronization. The following options are possible:</p> <ul style="list-style-type: none"> <li>■ <b>From configuration</b> The setting from the Configuration Editor at <b>System settings &gt; Referencing &gt; Synchronize item numbers / part attributes when updating file</b> will be applied.</li> <li>■ <b>Main parts and sub-parts</b> The part attribute will be updated for main parts and sub-parts.</li> <li>■ <b>Sub-parts only</b> The part attribute will be updated only for sub-parts.</li> <li>■ <b>Never</b> The specified part attribute will never be updated.</li> </ul> <p> Part attributes that are not listed in the table will be updated for main parts and sub-parts.</p>			
<b>Comment</b>	<p>Here you can enter a comment.</p>			

To insert further attribute rows, scroll to the end of the table. There, click on the empty row and specify the required settings. Proceed likewise for further rows. Click **OK** to close the **Attribute synchronization options** dialogue window.

The sorting of the table can be changed by clicking on a column header.

With a right mouse click on a table entry you activate the context menu, in which you are enabled to delete the active table row as well as activate or deactivate all check boxes of the column **active**.



#### Please note:

Up to HiCAD 2017 the settings for attribute synchronization were defined in the system file **REF3D\_ATTR\_AKT.DAT**. If you want to take over settings from an existing **REF3D\_ATTR\_AKT.DAT** file, this can be done via the **Open file**  button. After selecting the file, the settings shown in the **Attribute synchronization options** dialogue window will be overwritten with those of the selected file after closing the dialogue window with **OK**.

If you have applied any changes to the file **REF3D\_ATTR\_AKT.DAT**, please save the file before making an update.

## Performance

The updating of referenced 3-D parts with many HCM dimensions has been speeded up significantly.

# Catalogue Editor

## Service Pack 2 2018 (V 2302)

When carrying out an update to Version 2302.0, the catalogues will be cleaned up. Depending on the version to be updated, this process may take up to 20 minutes.

## Improved and new preview images

The tables of the catalogue **Processing, general**, too, now have new preview images and descriptive detail graphics.

ID	MOD	STATUS	BZ	SIZE	TYPE	DN	P	KERD
1	1	▶	DIN 103-Tr 8x1,5	Tr 8x1,5	F	8	1,5	
2	2	▶	DIN 103-Tr 9x1,5	Tr 9x1,5	F	9	1,5	
3	3	▶	DIN 103-Tr 9x2	Tr 9x2	F	9	2	
4	4	▶	DIN 103-Tr 10x1,5	Tr 10x1,5	F	10	1,5	
5	5	▶	DIN 103-Tr 10x2	Tr 10x2	F	10	2	
6	6	▶	DIN 103-Tr 11x2	Tr 11x2	F	11	2	
7	7	▶	DIN 103-Tr 11x3	Tr 11x3	F	11	3	
8	8	▶	DIN 103-Tr 12x2	Tr 12x2	F	12	2	
9	9	▶	DIN 103-Tr 12x3	Tr 12x3	F	12	3	
10	10	▶	DIN 103-Tr 14x2	Tr 14x2	F	14	2	1
11	11	▶	DIN 103-Tr 14x3	Tr 14x3	F	14	3	1
12	12	▶	DIN 103-Tr 16x2	Tr 16x2	F	16	2	1
13	13	▶	DIN 103-Tr 16x4	Tr 16x4	F	16	4	1
14	14	▶	DIN 103-Tr 18x2	Tr 18x2	F	18	2	1
15	15	▶	DIN 103-Tr 18x4	Tr 18x4	F	18	4	1
16	16	▶	DIN 103-Tr 20x2	Tr 20x2	F	20	2	1
17	17	▶	DIN 103-Tr 20x4	Tr 20x4	F	20	4	1
18	18	▶	DIN 103-Tr 22x3	Tr 22x3	F	22	3	1
19	19	▶	DIN 103-Tr 22x5	Tr 22x5	F	22	5	1
20	20	▶	DIN 103-Tr 22x8	Tr 22x8	F	22	8	
21	21	▶	DIN 103-Tr 24x3	Tr 24x3	F	24	3	2
22	22	▶	DIN 103-Tr 24x5	Tr 24x5	F	24	5	1
23	23	▶	DIN 103-Tr 24x8	Tr 24x8	F	24	8	
24	24	▶	DIN 103-Tr 26x3	Tr 26x3	F	26	3	2
25	25	▶	DIN 103-Tr 26x5	Tr 26x5	F	26	5	2
26	26	▶	DIN 103-Tr 26x8	Tr 26x8	F	26	8	

## Sheets with thickness 1.5 mm / 2.5 mm

Sheet thicknesses of 1.5 mm and 2.5 mm per table and material have been added to the following catalogues at **Factory standards > Sheets**:

- Aluminium sheet/plate
- Stainless steel sheet/plate and
- Steel sheet/plate

For bulb plates, composite panels and ALUCOBOND sheets/panels such thicknesses are not available.

## New material for sheets - X6CrNiMoTi17-12-2

The material **X6CrNiMoTi17-12-2 (1.4571)** has been added to the **ISD sheet** table at **Factory standards > Sheets**.

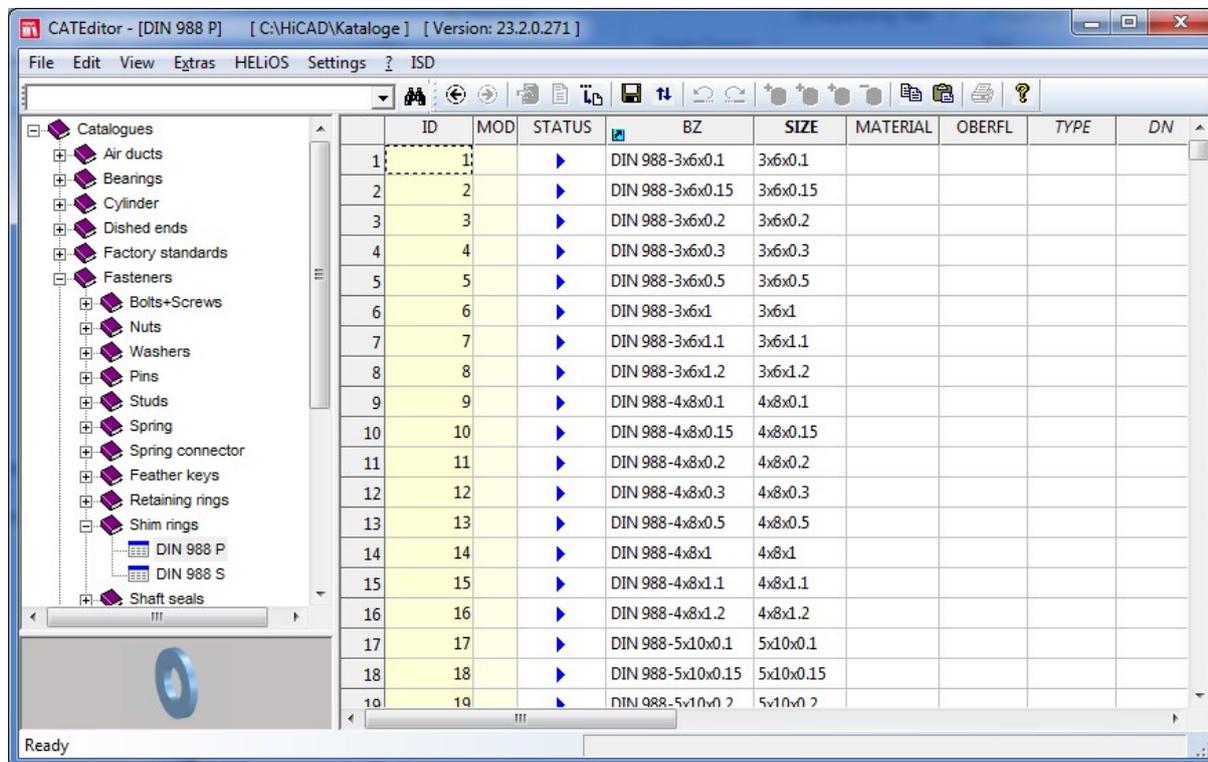
## Expanded tables for aluminium alloys / aluminium sheets/plates

- The material **EN AW-5005 H14** has been added to the **Aluminium alloys** table (in the catalogue **Materials> Non-ferrous metals**).
- The material **EN AW-5005 H14** has been added to the **Aluminium sheets/plate** table (in the catalogue **Factory standards > Sheets**).

ID	MOD	STATUS	BZ	T	MATERIAL
11	21	▶	Aluminium sheet 0.5mm	0.5	EN AW-1050A
12	22	▶	Aluminium sheet 0.75mm	0.75	EN AW-1050A
13	23	▶	Aluminium sheet 1mm	1	EN AW-1050A
14	24	▶	Aluminium sheet 2mm	2	EN AW-1050A
15	25	▶	Aluminium sheet 3mm	3	EN AW-1050A
16	26	▶	Aluminium sheet 4mm	4	EN AW-1050A
17	27	▶	Aluminium sheet 5mm	5	EN AW-1050A
18	28	▶	Aluminium plate 6mm	6	EN AW-1050A
19	29	▶	Aluminium plate 7mm	7	EN AW-1050A
20	30	▶	Aluminium plate 8mm	8	EN AW-1050A
21	31	▶	Aluminium sheet 1mm	1	EN AW-5005 H14
22	32	▶	Aluminium sheet 1.5 mm	1.5	EN AW-5005 H14
23	33	▶	Aluminium sheet 2 mm	2	EN AW-5005 H14
24	34	▶	Aluminium sheet 3 mm	3	EN AW-5005 H14
25	11	▶	Aluminium sheet 0.5mm	0.5	EN AW-6060 T66
26	12	▶	Aluminium sheet 0.75mm	0.75	EN AW-6060 T66
27	13	▶	Aluminium sheet 1mm	1	EN AW-6060 T66
28	14	▶	Aluminium sheet 2mm	2	EN AW-6060 T66
29	3	▶	Aluminium sheet 3mm	3	EN AW-6060 T66
30	16	▶	Aluminium sheet 4mm	4	EN AW-6060 T66
31	5	▶	Aluminium sheet 5mm	5	EN AW-6060 T66
32	18	▶	Aluminium plate 6mm	6	EN AW-6060 T66

## New catalogue for shim rings

Shim washers could previously be found in the catalogue **Fasteners > Retaining rings**. As of HiCAD 2018 SP2 a separate catalogue exists for **Shim rings**.



Therefore, the 3-D function **3-D Standard > Standard Parts > BoltScrew > Retaining rings** has been renamed to **3-D Standard > Standard Parts > BoltScrew > Retaining rings / Shim rings**.

## Catalogues/Tables for Element Installation / Substructures

Tables can, depending on the catalogue to which they belong, be assigned to specific categories. For Element Installation and Substructures the categories

- ELEMENT\_INSTALLATION\_ELEMENTS bzw.
- SUBSTRUCTURE\_ELEMENTS

are available.

If a new table is created for these categories, the corresponding table columns will be created automatically.

The tables at **Factory standards > Installation Planning - Parts and Processings > Element installation** and **Factory standards > Installation Planning - Parts and Processings > Sub-structure**, respectively, that are supplied with HiCAD are already assigned to the corresponding categories.

## Service Pack 1 2018 (V 2301)

### New functions in the "View" menu

The **View** menu has been expanded:



- **AutoFit column width**

The width of table columns can be manually changed by moving the borders of the columns with the cursor while holding down the left mouse button. If you activate the **AutoFit column width** option, the column widths will be automatically adjusted to the widths of the column cell contents. This is the default setting.

- **Hide deleted data records**

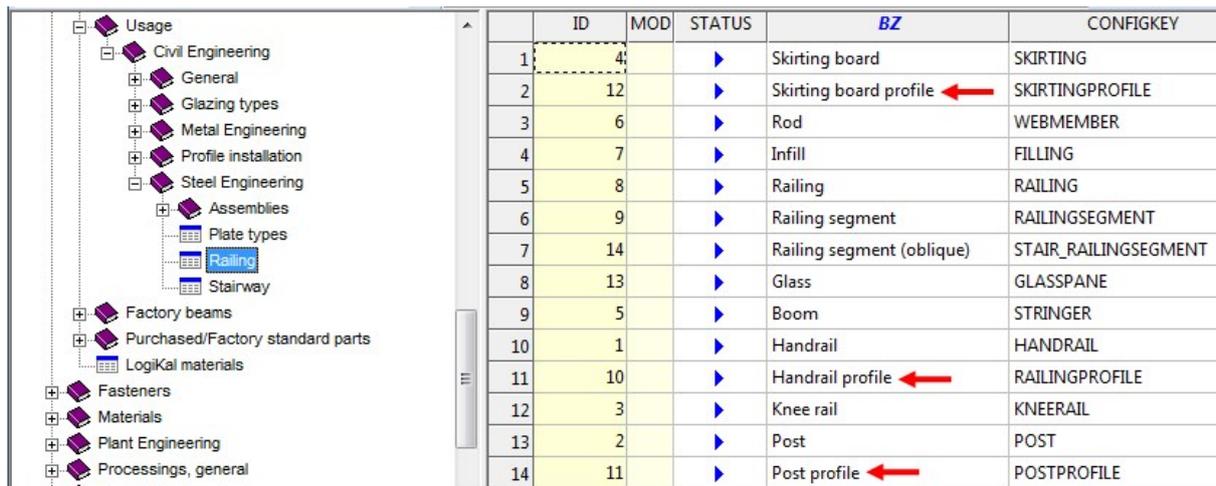
Data records with the status **Deleted**  are not offered for selection in HiCAD. If you want to hide these data records in the Catalogue Editor, too, activate the **Hide deleted data records** option. This is the default setting. If this option has been activated, the message **!!! Caution! Hidden contents: Data records !!!** will be displayed in the title bar of the Catalogue Editor when you open a table with "deleted" data records.

## New usages for railings

Three new usages are available at **Factory standards > Usage > Civil Engineering > Steel Engineering > Railing** namely,

- Skirting board profile,
- Handrail profile, and
- Post profile.

You can assign usage-dependent configurations to these railing elements for drawing derivations.



	ID	MOD	STATUS	BZ	CONFIGKEY
	1	4	▶	Skirting board	SKIRTING
	2	12	▶	Skirting board profile ←	SKIRTINGPROFILE
	3	6	▶	Rod	WEBMEMBER
	4	7	▶	Infill	FILLING
	5	8	▶	Railing	RAILING
	6	9	▶	Railing segment	RAILINGSEGMENT
	7	14	▶	Railing segment (oblique)	STAIR_RAILINGSEGMENT
	8	13	▶	Glass	GLASSPANE
	9	5	▶	Boom	STRINGER
	10	1	▶	Handrail	HANDRAIL
	11	10	▶	Handrail profile ←	RAILINGPROFILE
	12	3	▶	Knee rail	KNEERAIL
	13	2	▶	Post	POST
	14	11	▶	Post profile ←	POSTPROFILE

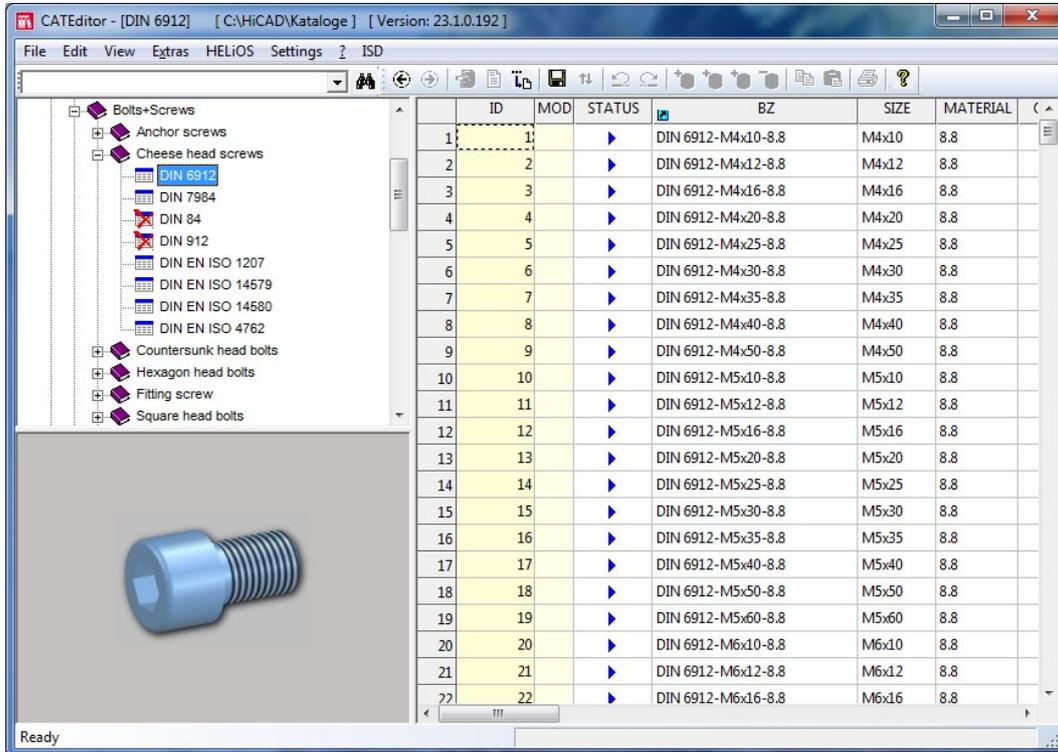
## Changed alias names

The alias names of the following catalogues have been changed:

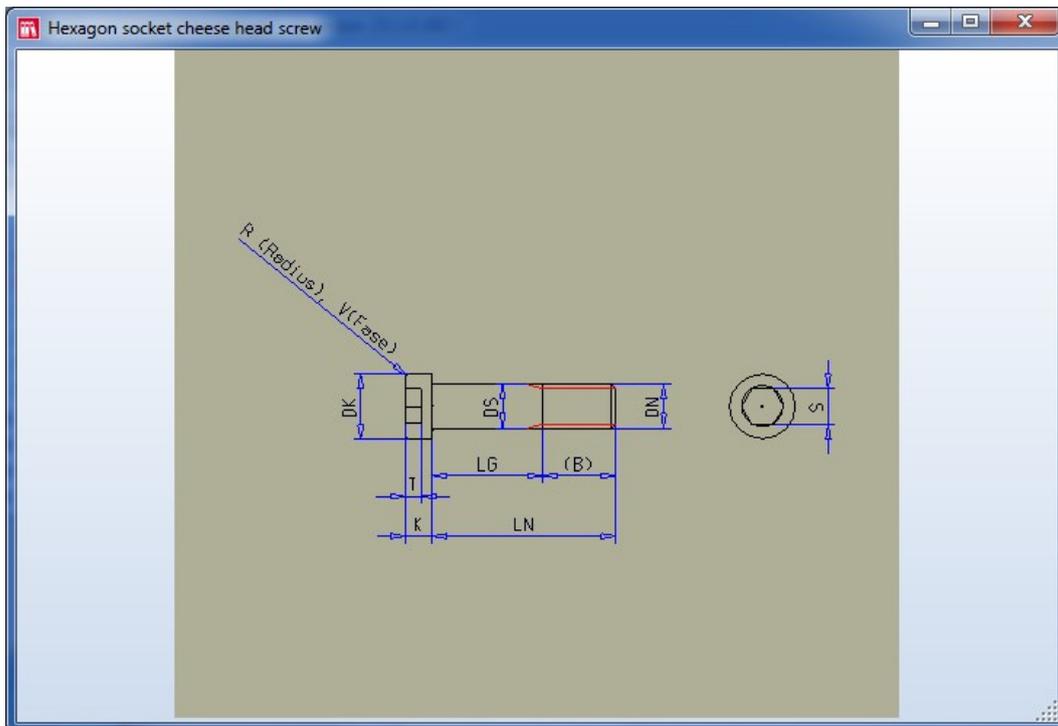
Old	New
System Settings / Colours	System Settings / System colours
Factory standards / Surface treatment / Profile installation	Factory standards / Surface treatment / Profile and Element installation

### Improved and new preview images

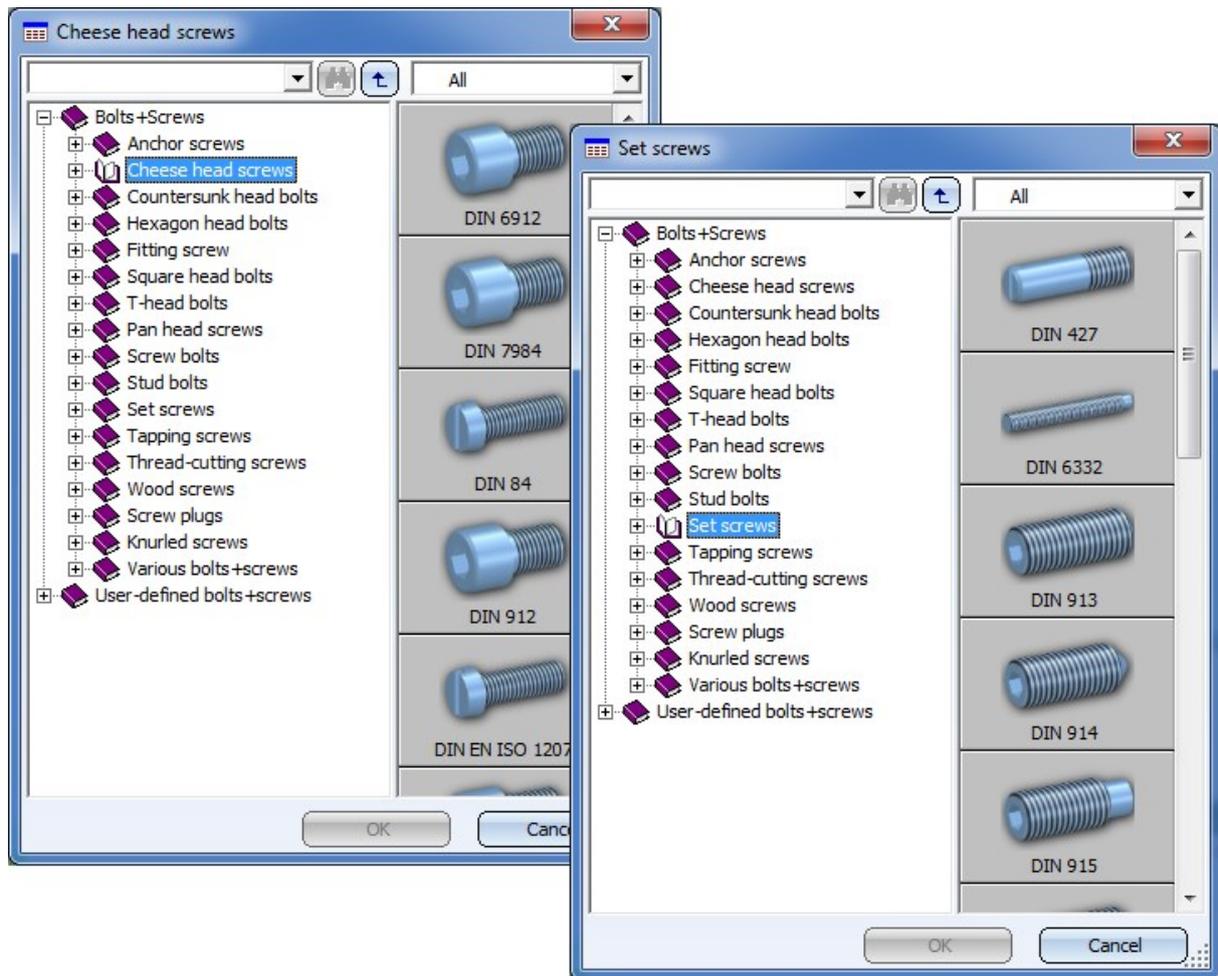
In the future, the tables are to contain new preview images and descriptive detail graphics. In the first step these were implemented for the **Fasteners** catalogue. The same will soon be done for the other catalogues.



Double-click the preview image to vie a detail graphic:



The new preview images will also be shown when inserting parts in HiCAD:



## DIN EN 10255 - Medium-weight and heavy threaded pipes

At **Semi-finished products > Beams+Profiles > Steel pipes**, medium-weight and heavy threaded pipes according to DIN EN 10255 are now available.

## New fasteners with technical approval for building construction

The catalogue **Factory standards > User-defined fasteners** has been expanded by the following elements:

- New table SFS GESIPA Blind rivet, large head at User-defined rivet > SFS
- New table MBE Blind rivet, large head at User-defined rivet > MBE
- New tables JT4-FR and JT9-FR Hexagon socket tapping screws at User-defined bolts+screws > Ejot
- Expansion of table SLA54-6-S and renaming to SFS Drilling screw SLA at User-defined bolts+screws > SFS



## New table: Text alignment symbol

New in the **Factory standards** catalogue, at **Symbol > Textalignment**, is the **Textalignment symbol** table.

The symbols defined here will be used for sheet development. The relevant HiCAD function here is the new Text pos-

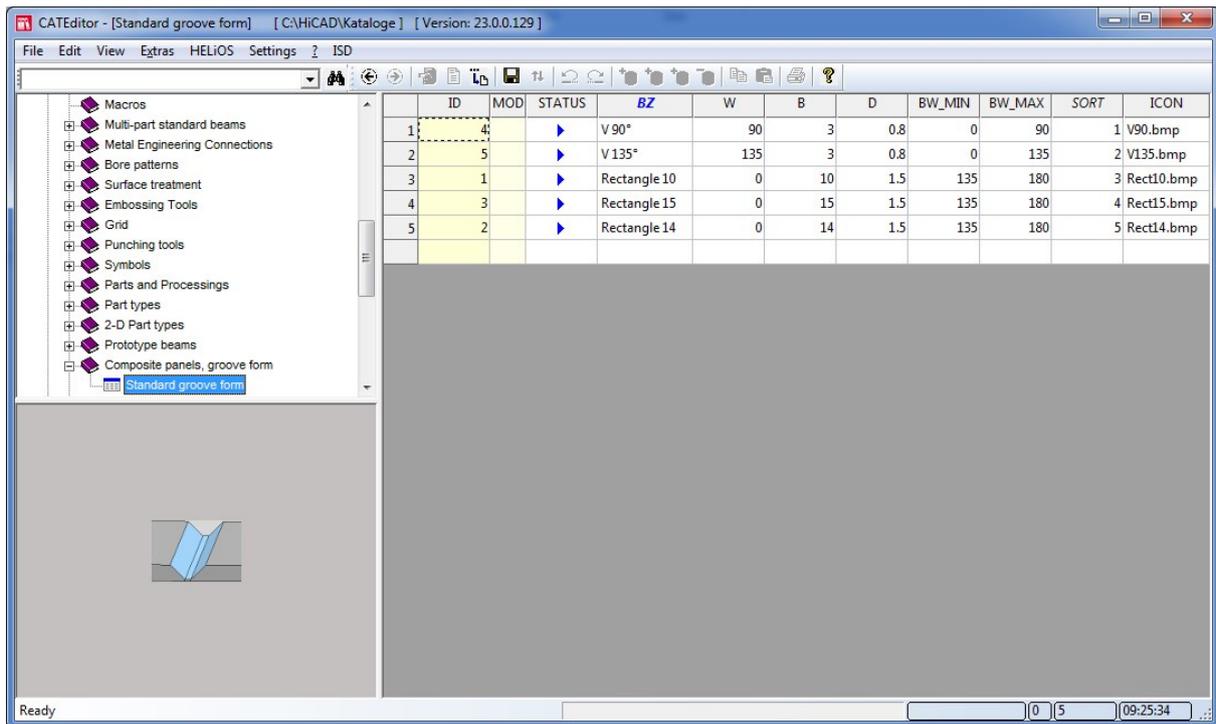
ition on development  function that you can find at **Sheet Metal > Process > Extras**  **> ...** .

The table is individually expandable.

## Major Release 2018 (V 2300)

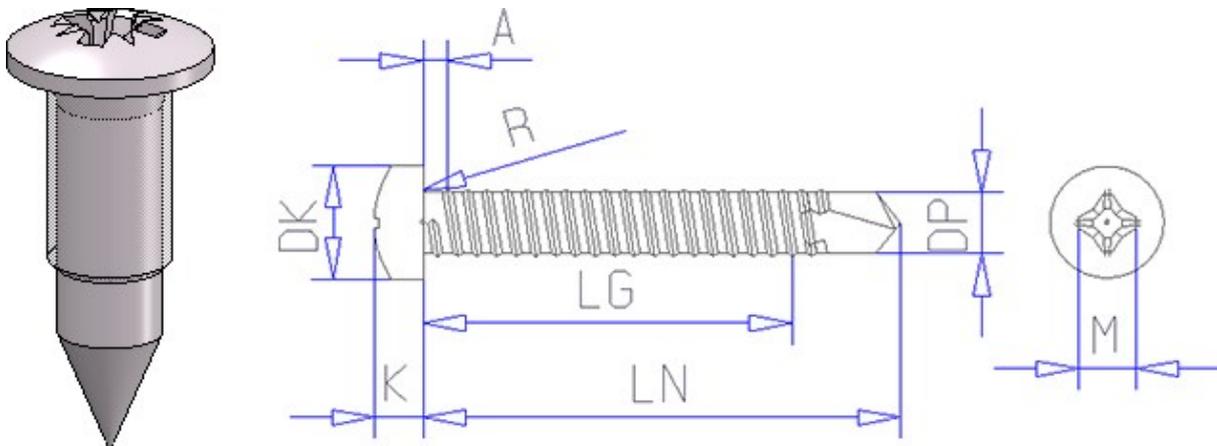
### Milling tools for processing of composite panels

New in the **Factory standards** catalogue are the milling tools for processing of composite panels. You can find them at **Factory standards > Composite panels, Groove form**.



### HILTI Metal Engineering screws - Self-tapping screws S-AD01LHS

New in the **Factory standards** catalogue are the HILTI Self-tapping screws S-AD01LHS without sealing washers. You can find them at **Factory standards > User-defined fasteners > User-defined bolts+screws > Hilti**.



## Symbols for processing of Sheet Metal parts

The **Direction arrows** table at WERKSNORMEN/SYMBOLS/PFEILE now contains the new symbols **Direction symbol Composite panel** and **Direction symbol One-sided**. With the help of these symbols the texturing or coating directions of panels can be specified. When creating sheet developments, the direction can be used for the alignment of the development.

# Feature Technology

## Service Pack 2 2018 (V 2302)

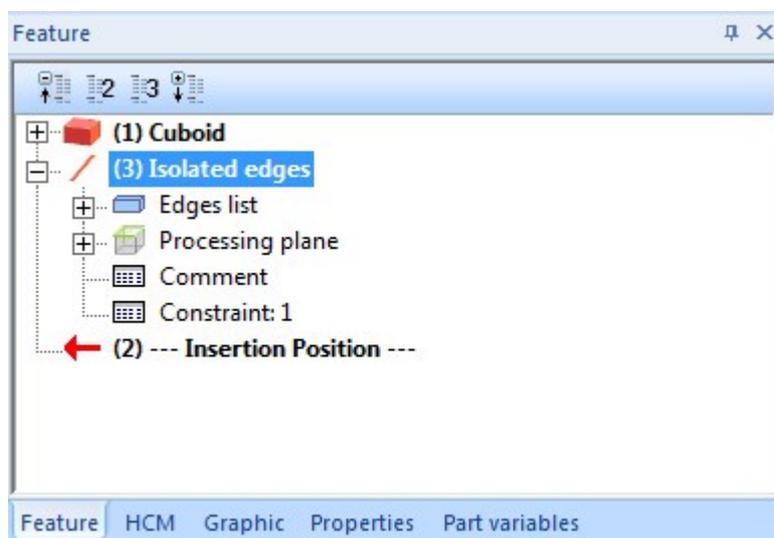
### Performance

Due to the further optimization of the **Subtract...** feature, the calculation of variants containing many of those features in the deactivated state could be speeded up significantly.

## Service Pack 1 2018 (V 2301)

### Copy edges

The previous function **3-D Standard > Tools > Edge > Copy from part** has been removed and replaced with the new and enhanced **Copy edges** function: The context menu of this function offers the new **All edges or lines of part** option, which enables you to copy all edges or lines of any 3-D part into the active part. This function is also recorded in the feature log, so that edges and lines inserted in this way will remain after feature recalculations.



### Performance

Thanks to extensive code revisions model drawings with large Feature logs can now be loaded much faster.

### Hide points in Feature log

Isolated points can now also be hidden. To do this, use the functions **3-D Standard > Tools > Point**  **> Hide -**

**Active part**  , and **3-D Standard > Tools > Point**  **> Show - Active part** .

Points that have been hidden via these functions will be entered in the Feature log of their creation Feature. In this way they will be preserved when Feature recalculations are performed.

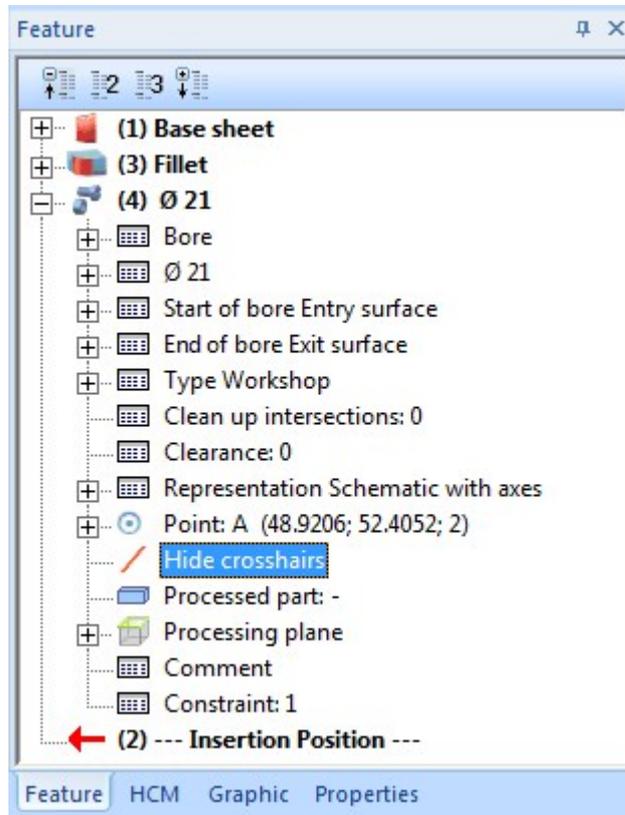
### Hide crosshairs in Feature log

In addition to the deletion of crosshairs, you have now also the option to hide crosshairs. At **3-D Standard > Tools**

**> Crossh.**  **> ...** you can now find the functions **Hide axis - Individual** , **Group** , **All in active part**  and **Show axis - All in active part** .



Hidden crosshairs will be entered as parameters of their creation Feature and will be preserved after Feature recalculations.



## Major Release 2018 (V 2300)

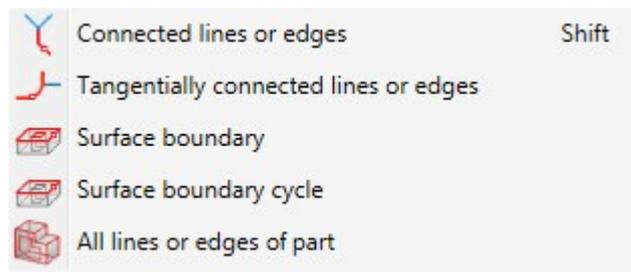
### Insertion of edges

The pull-down menu at **3-D Standard > Tools > Edge**  > ... has been revised.

The functions

- Copy, Individual
- Copy, In rectangle
- Copy group

have been combined into a new function called **Copy edges** . When you call this function, the following options will be available to you in a context menu:

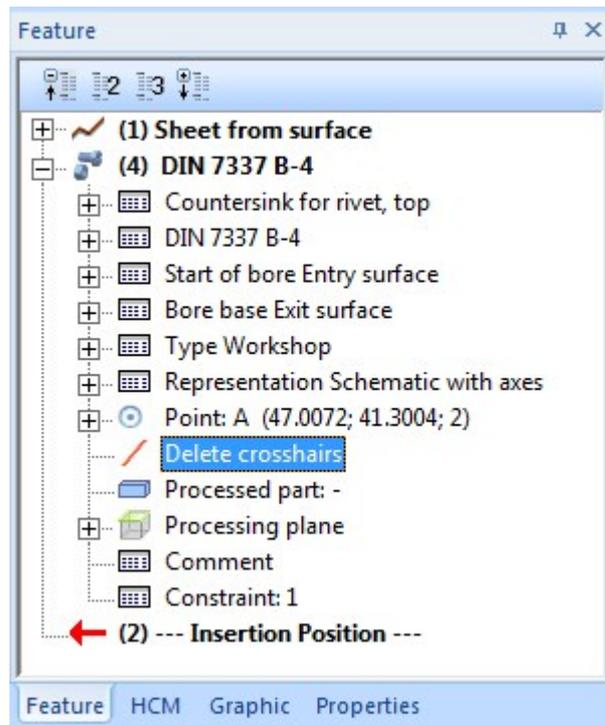


In contrast to the old functions, the new function is also supported by Feature Technology: Edges that have been created with this new function will have a Feature called **Isolated edges**.

## Permanent deletion of crosshairs in Feature log

If you delete crosshairs that have been automatically created by a processing, e.g. **3-D Standard > Standard Processings > Bore/Thr.** this deletion will now be saved in the Feature log. This ensures that this change will also be preserved in case of a recalculation.

In the process, the deletion will not be entered as an independent step in the Feature log, but as a sub-point to the feature that originally created the crosshairs.



To delete this processing step later, right-click the **Delete crosshairs** entry and choose **Clear**.

## Permanent deletion of isolated points in Feature log

Up to HiCAD 2017 the deletion of isolated points was not considered in the Feature log, causing an unwanted restoring of deleted points in the drawing after feature recalculation.

From HiCAD 2018 onwards the deletion of isolated points will also be considered in the log of their respective creation Feature. If an isolated point is deleted in the drawing, the corresponding entry will also be deleted in the Feature log. This means that isolated points deleted isolated points actually remain deleted after a feature recalculation.

# Parametrics (HCM)

## Service Pack 2 2018 (V 2302)

### Display of positional and dimensional constraints

In the **HCM** function group of the **3-D Standard** and the **Sketch** tab you can now find the functions **3-D Standard**

> **HCM > Toggle visibility of dimensions**  and **Sketch > HCM > Toggle visibility of constraints** , respectively.

For sketches you have the option to toggle between the following display modes:

- Show positional and dimensional constraints
- Show only dimensional constraints
- Show neither positional nor dimensional constraints

The "Show positional constraints" option that could formerly be found at **Sketch > HCM Tools**  > **Settings**, has been removed.

For 3-D parts this function activates or deactivates the display of dimensional constraints.

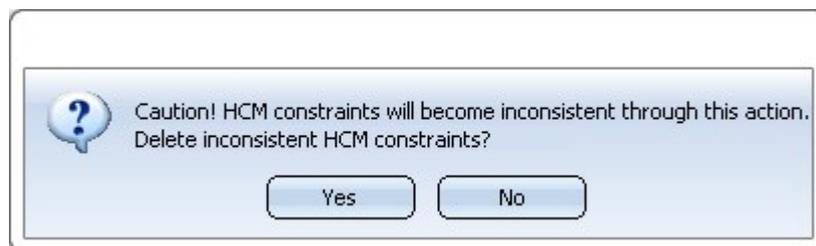
The settings of the display mode affect the entire drawing, and not just the active sketch or assembly.

## Service Pack 1 2018 (V 2301)

### Forced fixing

A new positional constraint called **Forced fixing**  is now available. A force-fixed part can generally not be transformed (moved, rotated, scaled or mirrored) any more - in contrast to parts that have been fixed with the normal **Fix/Unfix** function that only prevents transformations of parts when trying to drag them.

If you attempt to transform a force-fixed part, the following message will be displayed:

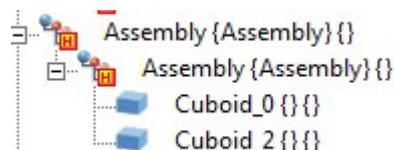


If you click on **Yes**, the forced fixing will be removed and the part will be transformed. If you click on **No** the transformation will not be executed and the forced fixing remains.

Please note however that the forced fixing refers only to one part within an assembly, i.e. the transformation of the complete assembly will still be possible. If the assembly is transformed, the force-fixed part will be transformed together with the assembly.

### Indication of inconsistent HCM models in the 3-D part structure

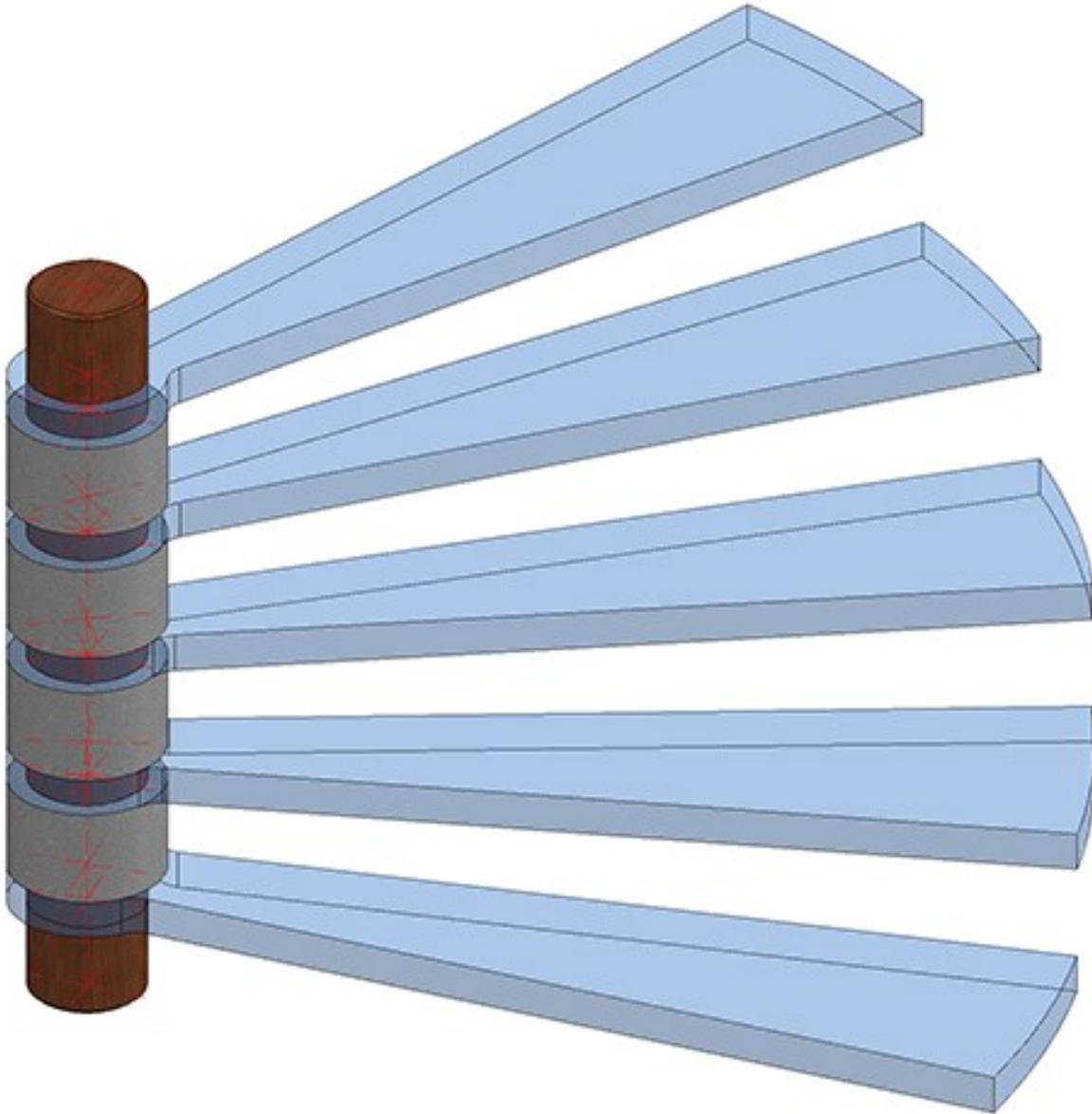
In the Configuration Editor at **System settings > Assembly HCM** and **System settings > Sketch HCM**, respectively, you find the new setting option **Show HCM errors in part structure?**. If you activate this option, inconsistent HCM models in sketches or assemblies will be marked in the 3-D structure with a corresponding symbol. This marking will also be applied to superordinate parts and assemblies to indicate the inconsistencies up to the top level of the affected element.



### Positional constraint "Equal angle" also in 3-D Part HCM

The positional constraint **Equal angle**  can now be used not just in sketches, but also in 3-D parts. You find this constraint at **3-D Standard > HCM > Fix**  **> Equal angle**.

After calling the function hiCAD will asks you to identify two or more angles by specifying two legs for each of them. You can the apply the positional constraint **Equal angle** to these legs. HiCAD will then apply the **Equal angle** constraint between them.



Example of the positional constraint **Equal angle** applied to a part

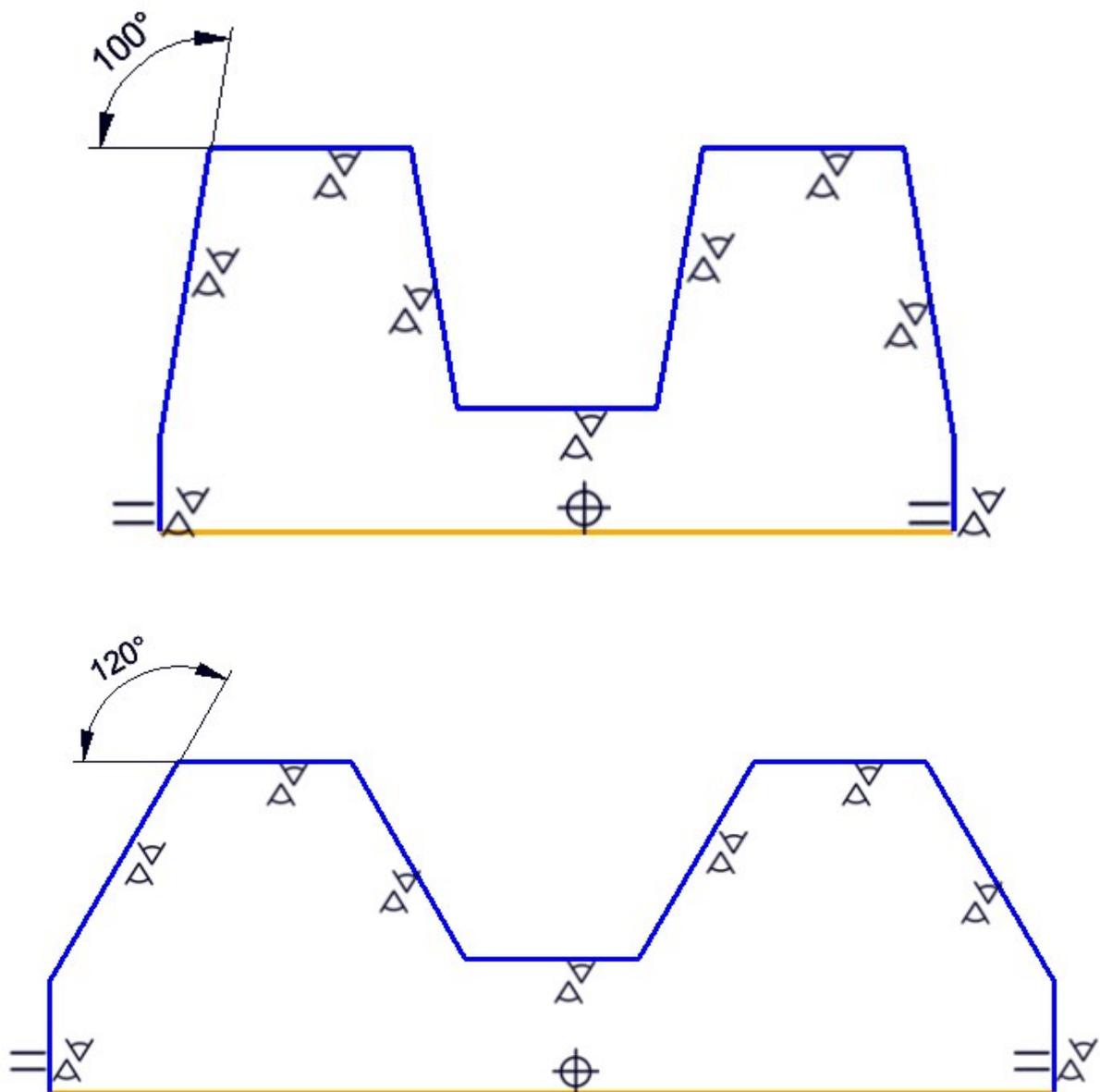
## Major Release 2018 (V 2300)

### New positional constraint: Equal angle

A new positional constraint for sketches and 3-D sketches - **Equal angle** - is now available.

When you select Sketch > HCM > Fix  > Equal angle, HiCAD will ask you to identify two angles, either by selecting an arc or by selecting two straight lines.

The positional constraint will then be applied and displayed both in the drawing area and in the ICN.



# Configuration Management

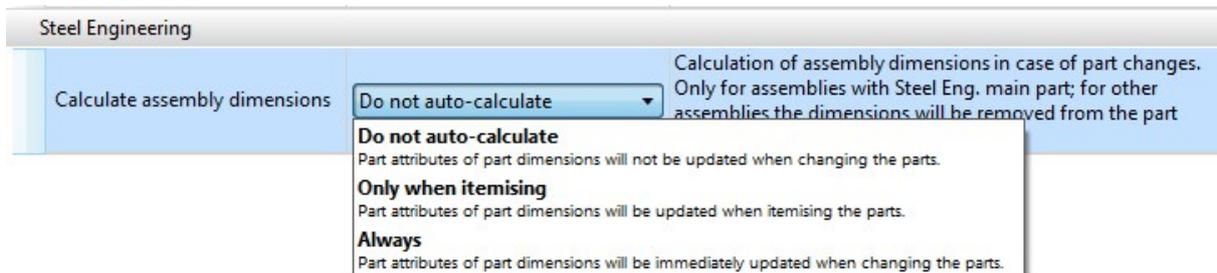
## Service Pack 2 2018 (V 2302)

### UNDO setting for 32 Bit versions removed

The setting **UNDO memory allocation for HiCAD 2-D** has been removed. Since it refers only to 32 Bit versions it is no longer relevant.

### Texts adjusted

The texts at **Modelling > Part Properties > Calculate assembly dimensions** have been optimised and adjusted to other, similar entries in this area.



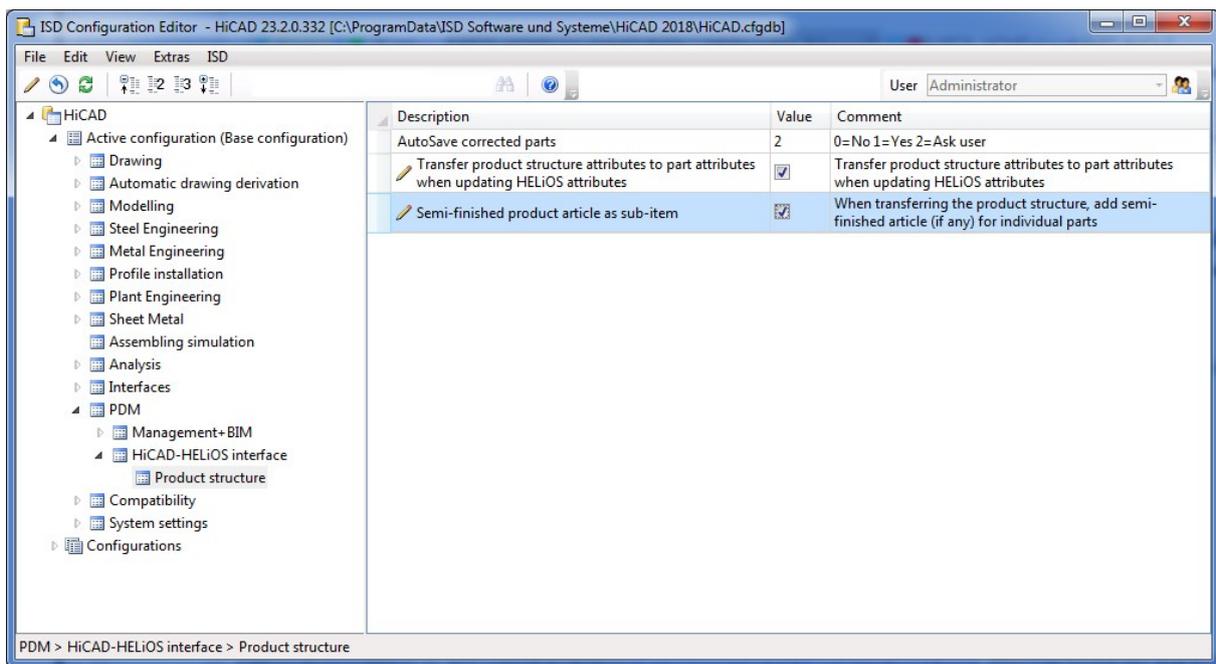
### Default settings fro mounting tracks

The ISD default settings for mounting tracks can be found in the Configuration Editor at **Drawing > Views > Exploded view**.

### Semi-finished Product Article as Sub-item

Semi-finished products can be automatically transferred as sub-items of a part to HELIOS.

This can be set in the Configuration Editor at **Active configuration > Base configuration > PDM > HiCAD-HELIOS interface > Product structure**. Here, activate the **Semi-finished product article as sub-item** checkbox.



If a HiCAD model drawing (.SZA document) or an individual geometry (.KRA document) is saved in HELiOs while the option has been activated, it will be checked whether a semi-finished product needs to be saved as a structure sub-item of the corresponding part (=HELiOs article).

The relevant semi-finished product article masters will then be added as sub-item, or updated accordingly in the product structure of the part.

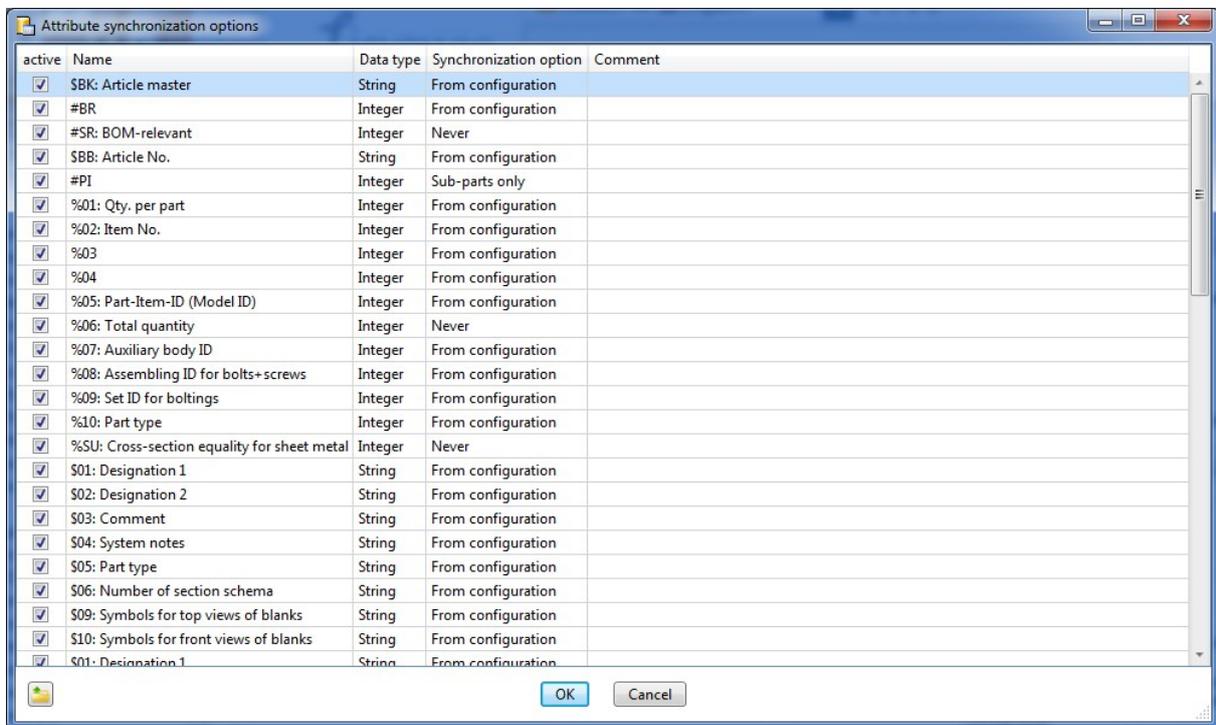
## Major Release 2018 (V 2300)

### Settings from REF3D\_ATTR\_AKT.DAT moved to Configuration Editor

The settings from the file REF3D\_ATTR\_AKT.DAT for the synchronization of part attributes for referenced parts have been moved to the Configuration Editor (ISDConfigEditor.exe). At **System settings > Referencing** you can find the setting option **Synchronization of attributes** (SYSTEM > Referencing > RefPartAttrUpdate).

If you click on the  symbol, a window with a table will be opened, enabling you to specify for each part attribute:

- if it is to be updated for sub-parts only,
- whether the settings from the Configuration Editor are to be applied, or
- whether it is never to be updated.



active	Name	Data type	Synchronization option	Comment
<input checked="" type="checkbox"/>	SBK: Article master	String	From configuration	
<input checked="" type="checkbox"/>	#BR	Integer	From configuration	
<input checked="" type="checkbox"/>	#SR: BOM-relevant	Integer	Never	
<input checked="" type="checkbox"/>	SBB: Article No.	String	From configuration	
<input checked="" type="checkbox"/>	#PI	Integer	Sub-parts only	
<input checked="" type="checkbox"/>	%01: Qty. per part	Integer	From configuration	
<input checked="" type="checkbox"/>	%02: Item No.	Integer	From configuration	
<input checked="" type="checkbox"/>	%03	Integer	From configuration	
<input checked="" type="checkbox"/>	%04	Integer	From configuration	
<input checked="" type="checkbox"/>	%05: Part-Item-ID (Model ID)	Integer	From configuration	
<input checked="" type="checkbox"/>	%06: Total quantity	Integer	Never	
<input checked="" type="checkbox"/>	%07: Auxiliary body ID	Integer	From configuration	
<input checked="" type="checkbox"/>	%08: Assembling ID for bolts+screws	Integer	From configuration	
<input checked="" type="checkbox"/>	%09: Set ID for boltings	Integer	From configuration	
<input checked="" type="checkbox"/>	%10: Part type	Integer	From configuration	
<input checked="" type="checkbox"/>	%SU: Cross-section equality for sheet metal	Integer	Never	
<input checked="" type="checkbox"/>	\$01: Designation 1	String	From configuration	
<input checked="" type="checkbox"/>	\$02: Designation 2	String	From configuration	
<input checked="" type="checkbox"/>	\$03: Comment	String	From configuration	
<input checked="" type="checkbox"/>	\$04: System notes	String	From configuration	
<input checked="" type="checkbox"/>	\$05: Part type	String	From configuration	
<input checked="" type="checkbox"/>	\$06: Number of section schema	String	From configuration	
<input checked="" type="checkbox"/>	\$09: Symbols for top views of blanks	String	From configuration	
<input checked="" type="checkbox"/>	\$10: Symbols for front views of blanks	String	From configuration	
<input checked="" type="checkbox"/>	\$01: Designation 1	String	From configuration	



**Please note:**

Up to HiCAD 2017 the settings for attribute synchronization could be found in the system file REF3D\_ATTR\_AKT.DAT.

If you want to take over the settings from an existing REF3D\_ATTR\_AKT.DAT file, this can be done via the **Open file**  button. After selecting the file, the settings shown in the **Attribute synchronization options** dialogue window will be overwritten with those of the selected file. Confirm with **OK**.

If you have customized the REF3D\_ATTR\_AKT.DAT file, save the file before performing an update!

## Settings from TXTPAR.DAT.DAT moved to Configuration Editor

The settings from the file TXTPAR.DAT have been moved to the Configuration Editor. There, you can find the settings at:

- Drawing > Annotations > Text > Font
- System settings > Annotations > Text
- System settings > Annotations > Text > Preferred text sizes

# Bill of Materials / Report Manager

## Service Pack 2 2018 (V 2302)

### Sheet development attributes

The system attributes

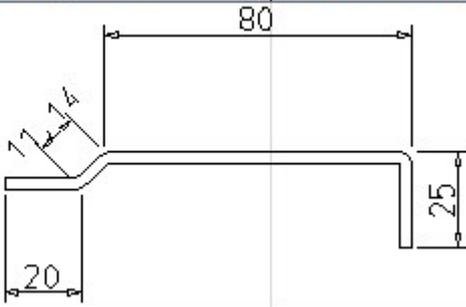
- \$L2D** Development length
- \$B2D** Development width
- \$T2D** Sheet thickness

will now be considered for transfer of part attributes to the Report Manager. However, this does not apply to data-base BOMs.

## Service Pack 1 2018 (V 2301)

### Sheet Metal parts with images in Excel BOMs

The creation of images for Excel BOMs for Sheet Metal parts with images has been optimized. The created images are better scaled, the dimensioning is clearly visible and the images are rotated where required to allow a better fitting.

Item	Qty.	Designation	Dev. Width (mm)	Dev. Length (m)
				

# Variant Editor

## Major Release 2018 (V 2300)

### Automatic derivation of variants with varying sub-types

The variant editor can be used to derive variants automatically. VAA files can be created, whose attributes are being modified according to a template in form of a CSV file. So far, the automatic derivation has been tied to fit attributes which are constant via all sub-types.

As of HiCAD 2018 it is also possible to modify sub-types with regards to the automatic derivation. The variants call and procedure take place analogously to automatic derivation of variants with constant sub-types. However, additional attributes are available for varying sub-types:

- **VAREEDIT\_CLUSTERING\_KEY**

One list of columns of the CSV-file is being assigned to the attribute, which is checked for equality.

- **VAREEDIT\_SUBTYPE\_KEY**

This attribute can be assigned as value to a series of HELiOS attributes, which determine the columns to be compared. By contrast to VAREEDIT\_CLUSTERING\_KEY, the CSV file's names of columns are not considered here.

# Interfaces

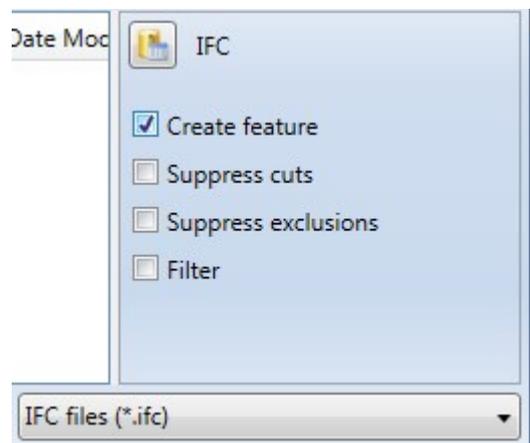
## Service Pack 2 2018 (V 2302)

### STEP export via selection list

The internal structures for the STEP export of entire drawings and the export via selection list have been standardized. Inconsistencies that occasionally occurred in older versions have been eliminated.

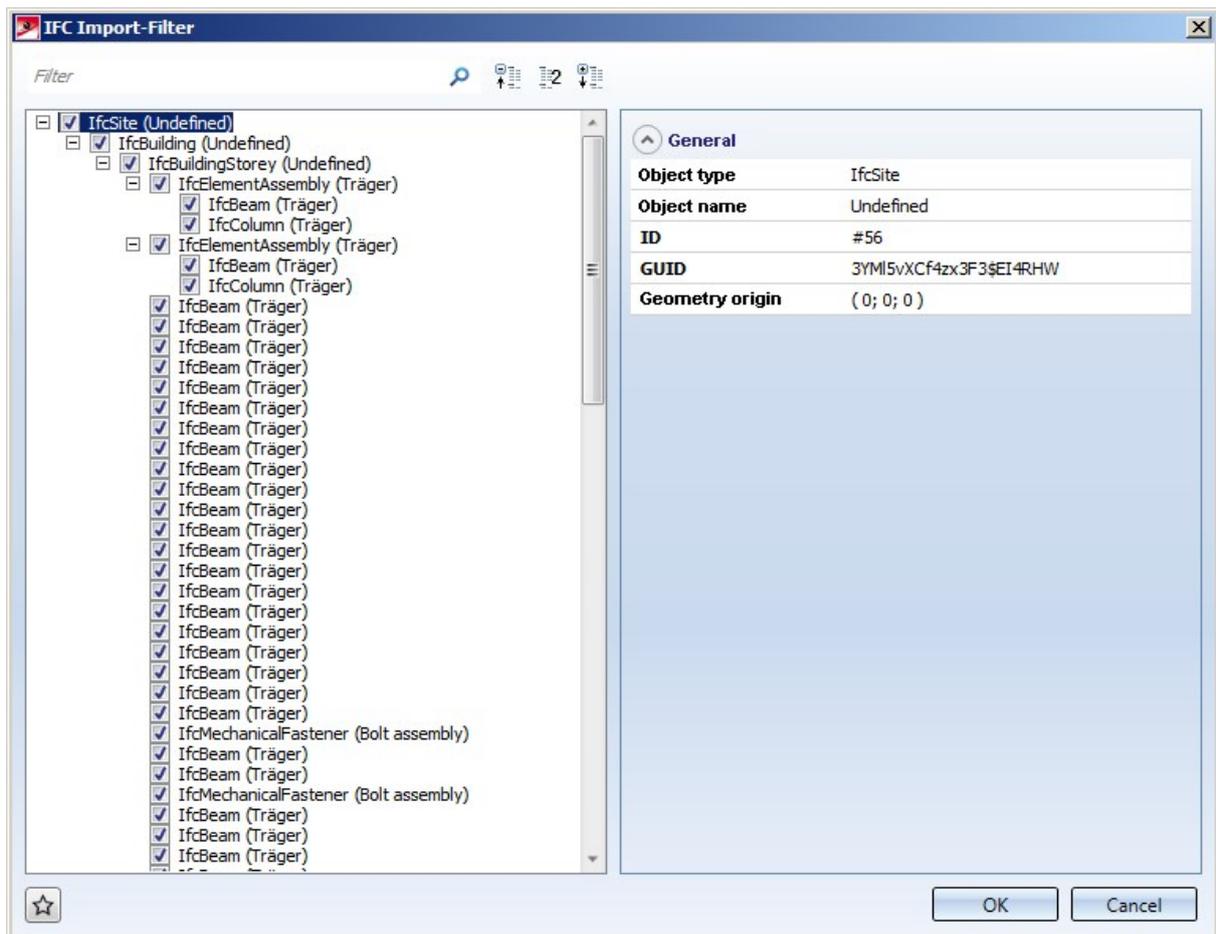
### IFC import: Class filter

The new **Filter** option has been added to the IFC import options.



If this option has been activated, another dialogue for deselection of IFC classes will be opened after choosing **Open**.

In the tree structure you can deselect all classes that you want to exclude from import.



At the top left of the window you can find an input field where you can restrict the displayed classes by an explicit search for particular classes.

## DSTV-NC export via Spooler

The DSTV-NC format can now be exported from the HELiOS Desktop via HELiOS Spooler.

The format-specific export options that were last specified and saved in HiCAD will be used in the process.

## DXF export for sheet developments

The DXF export functions for sheet developments have been revised and rearranged. The following three functions are now available:

- 

 ■ Export development as DXF
- 

 ■ Develop sheets and export as DXF
- 

 ■ Export sheet developments as DXF

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

Furthermore, an interface between HiCAD and Lantek Flex3d is created by the export of the sheet metal blank as DXF file.

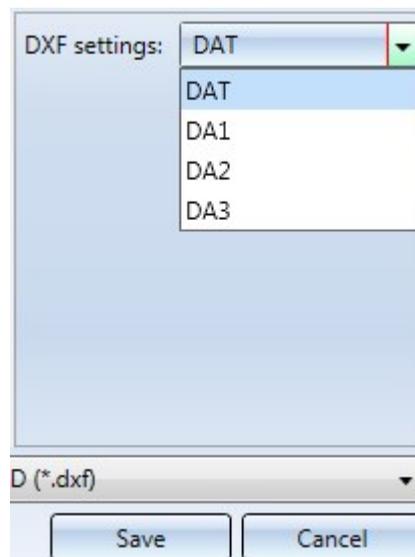
Lantek Flex3d is a software solution for the automating the programming of sheet metal cutting machines.

## Save 2-D part as DXF file

2-D parts can now be saved in DXF format by choosing **Drawing > Save/Print > Save as** > 2-D part DXF.



In the export dialogue you can choose a file for the settings of the 2-D DXF export:



## Service Pack 1 2018 (V 2301)

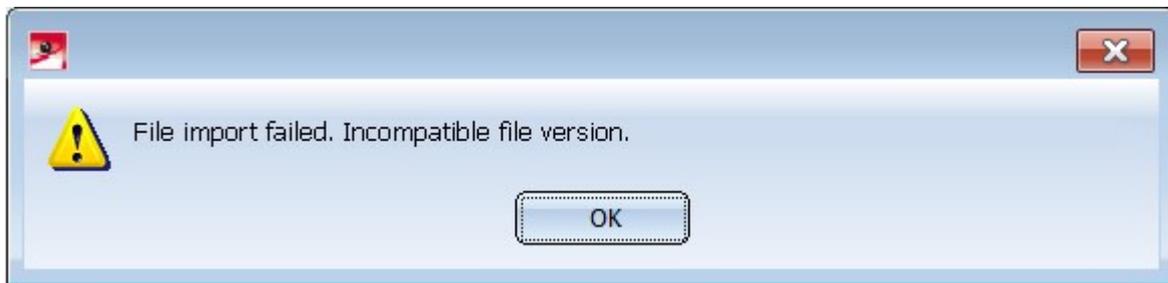
### AutoCAD 2018

HiCAD now also supports the import and export of 2-D DWG files from AutoCAD 2018.

### Recognition of incompatible file versions during 3-D foreign data import

When importing foreign data to 3-D formats that are internally converted via the CADfix interface (DXF,DWG, SolidWorks, Inventor etc.) the file to be imported might exist in a version that cannot be processed by CADfix yet. In such cases HiCAD will stop the conversion and display the error message *An error occurred during import*.

In HiCAD 2018 SP1 this general message has been replaced by the following message indicating that there is a compatibility issue:



**Please note:**

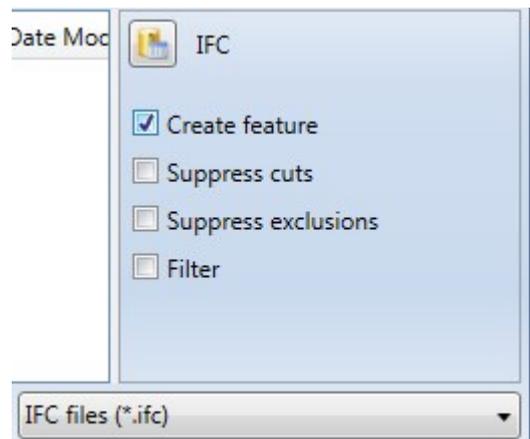
An overview of all interfaces with additional information (e.g. supported versions etc.) can be found online in the ISD-Wiki at [Product know-how / FAQ/ FAQ - Frequently asked questions](#).

## IFC import - Optional suppressing of processings

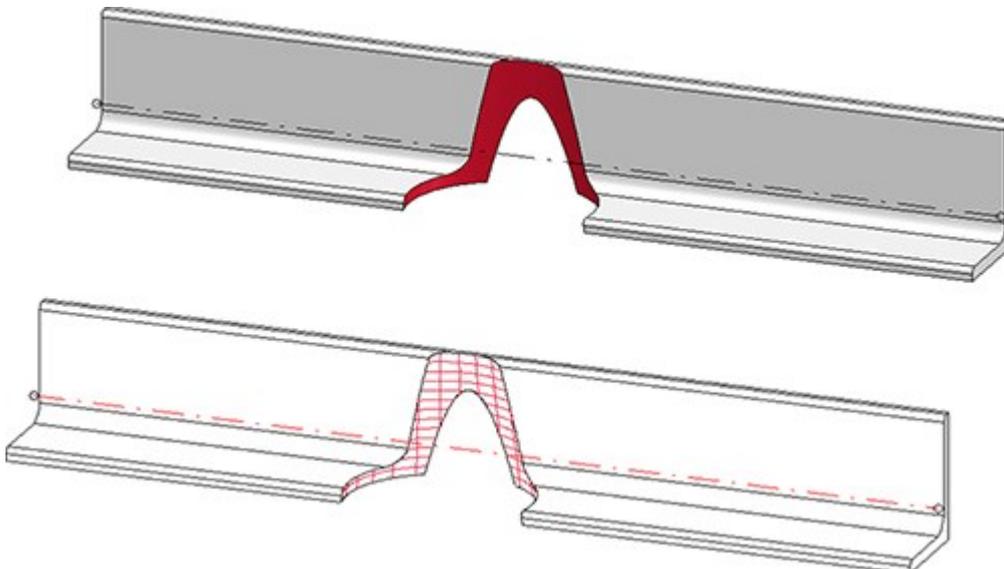
You have now the option to suppress processings when carrying out IFC imports, namely,

- Cuts (trimmings, mitres, notches etc.) and
- Exclusions (subtractions, bores etc.)

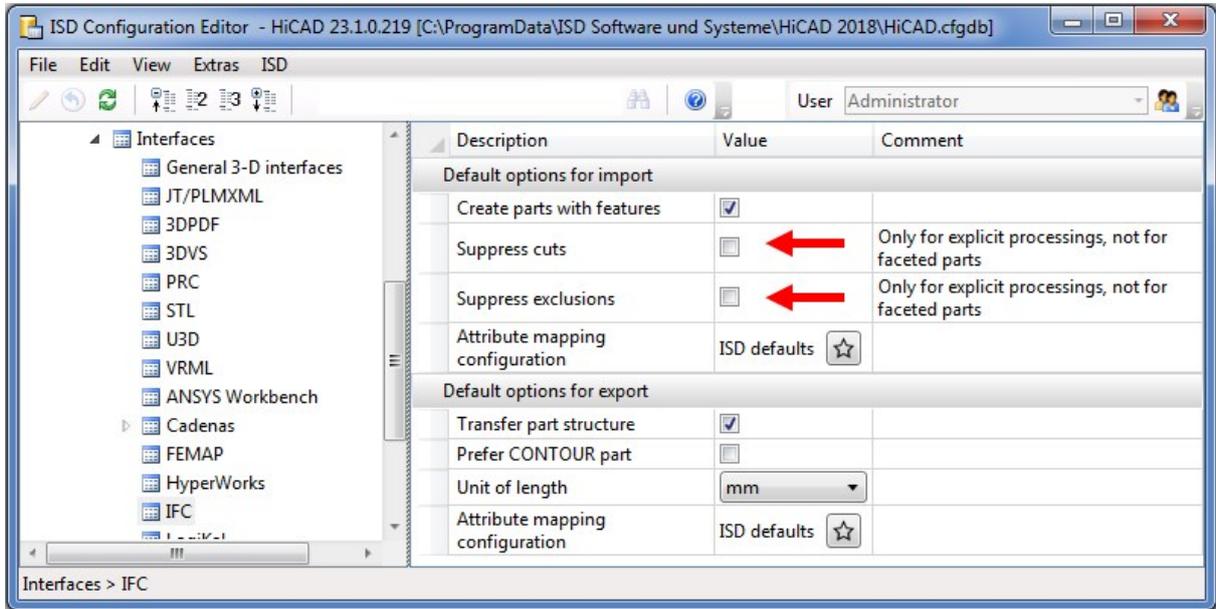
Activate or deactivate the corresponding checkbox(es) in the IFC import dialogue window.



Please note that this suppression is only possible for specific processings, and not for faceted parts. For instance, the processing shown in the image below will always be imported:



The handling of cuts and exclusions can be preset in the Configuration Editor at **Active configuration > Interfaces > IFC**:



### 2-D DXF export - CLOSED flag for export of (LW)POLYLINES

To enhance compatibility with CAM system, the CLOSED flag (Group code 70, also see <https://www.autodesk.com/techpubs/autocad/acadr14/dxf/>) has been appropriately set for the export of POLYLINES and LWPOLYLINES.

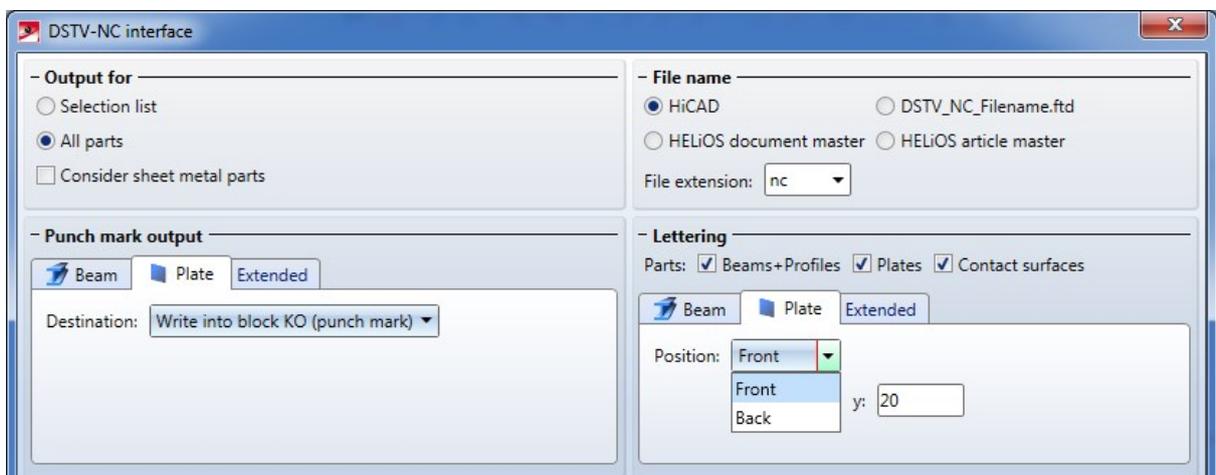
## Major Release 2018 (V 2300)

### Update to CADfix 11 SP2

With the CADfix 11 SP2 update the following format versions will be available in HiCAD: CATIA V5-6R2017, Inventor 2017, JT v8.5, Parasolid 29, ACIS 2018.

### DSTV-NC export - Lettering for plates

In the **DSTV-NC interface** dialogue, you can now not only specify for Beams + Profiles, but also for **Plates** whether the **Lettering** is to be applied at the front or back of the plate for DSTV-NC exports.



# Sheet Metal

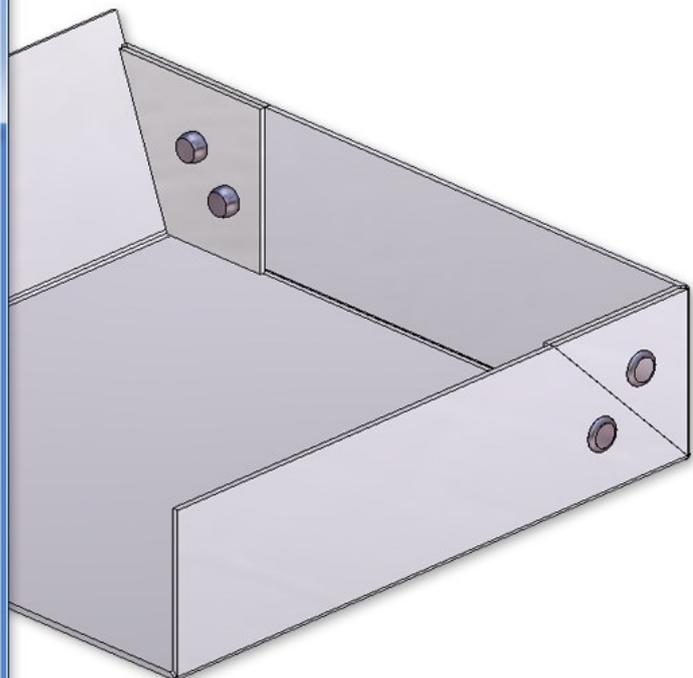
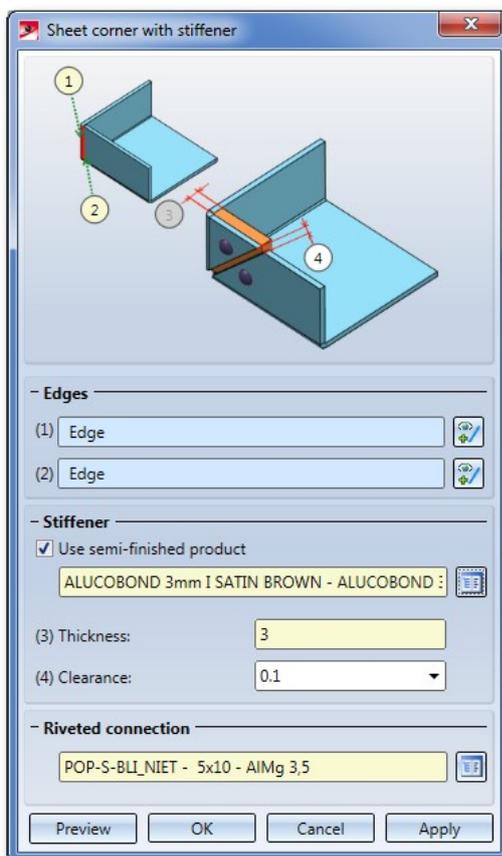
Service Pack 2 2018 (V 2302)

## Riveted sheet corner



Use the new **Sheet corner** function to create a riveted corner, reinforced with a stiffener, between 2 sheet flanges. The flanges can have different lengths and different bend angles. You can apply this connection type to milling edge zones and cylindrical bend zones.

When you call the function the following dialogue window will be displayed:

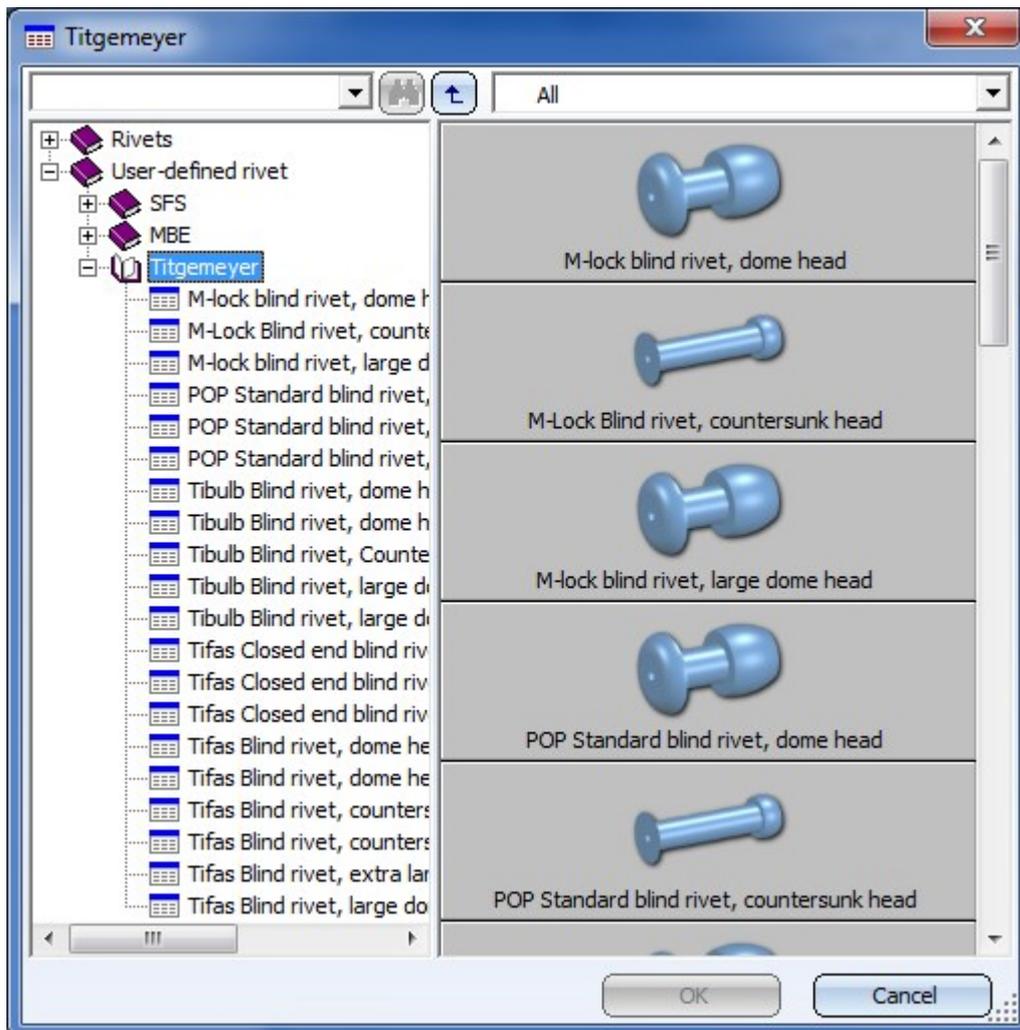


You determine the position of the connection by identifying two flanges on the longitudinal edge of the front side. If you want to identify a different flange, click the  icon beneath **Edges** and then identify the desired edge.

The connection will be applied to the last identified flange. If the flanges have different lengths, these will remain unchanged.

If you select a semi-finished product here, the Thickness (3) of the sheet will be automatically entered here. The standard for semi-finished products can be taken from the Catalogue Editor with a click on the  icon. With a double-click on the **Sheet corner** feature you can activate or deactivate the semi-finished product later.

Select the riveted connection from the catalogue. You can choose between SFS and Titgemeyer.



If you have entered 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 apply the data to other sheet corners. If you click **OK**, the sheet corner will be created, and the window will be closed. Clicking **Cancel** closes the dialogue window and discards the data without creating the sheet corner.

## New DXF export for sheet developments

The DXF export function for developments has been revised completely. You can choose between the following 3 options:

### Export development as DXF

When exporting a development as DXF file  (Sheet Metal > Sheet development) you have the option

- to save only the section contour (i.e. the drawn bend zones and angles will not be considered), or
- to save the sheet development as it is shown in the drawing (i.e. with the drawn bend zones and angles).

When you have identified an edge of the development, the standard dialogue window for the saving of a file will be opened. On the right hand side you can choose the export settings - either **Only section contour** or **Complete** development.

For the DXF export you need the file HCADACAD.DAT. It contains, for instance, the assignment of the HiCAD elements for specific AUTOCAD layers. This file can be customized and is located in the HiCAD subdirectory SYS. Do not save the file with the file name extension .DAT, but with .DA1, .DA2 or .DA3 after its customisation. In this context, please also read this information.

## Develop sheets and export as DXF

The **Develop sheets and export as DXF**  (Sheet Metal > Sheet development  > Extras) function exports the section contour or the complete development of Steel Engineering plates. In the process, one temporarily generated development will be output for the export. You can choose between various sheet development parameters.

For the selection of the sheet development parameters the following options are available:

- From drawing** Use this option to load the parameters of the default settings for the drawing. The default setting for new drawings can be specified at **Sheet Metal > Sheet development > Default setting**.
- Manual settings** Clicking the  icon opens the **Sheet development parameters** dialogue window.
- Favourite** The Favourites  saved for the development parameters, e.g. LVD, Bytronic or COBUS are offered for selection here.
- According to filter list from configuration** Use this option to load the assignment of the development favourites to the part filters (Sheet Metal sheets or Steel Engineering plates) from the Configuration Editor (at **Sheet Metal > Sheet development > Filter-Development parameter assignment**). Part filters are filters that have been predefined by the ISD and also the manual filters for part search that have been stored as Favourites.

The orientation is only unambiguous if it has been determined with the Processing direction function beforehand.

The file name can be configured in the Annotation Editor with a click on the  icon. To do this, apply, for instance, the part attribute "Item number" from the Sheet Metal part. If the part attributes set in the Editor cannot be applied to the sheets, you have the option to enter your own file name.

For the DXF export you need the file HCADACAD.DAT. It contains, for instance, the assignment of the HiCAD elements for specific AUTOCAD layers. This file can be customized and then saved with the file name extension .DA1, .DA2 or .DA3.

The selection of sheets can either take place in the drawing  or by adding of all sheets .

When you click on the **Export** button, temporary developments will be generated and output of all sheets in the export list, according to the specified parameters. The status icon  will indicate problems that may occur. The files will then be located in the specified directory.

The **Close & Report** button activates the Report Manager and creates a BOM. If you did not itemize the Sheet Metal part in HiCAD, quantities will be set on **0** here.

## Export sheet developments as DXF

When exporting multiple sheet developments as DXF files  (Sheet Metal > Sheet development  > Extras) you have the option

- to save only the section contours, i.e. (i.e. the drawn bend zones and angles will not be considered), or
- to save the sheet development as it is shown in the drawing (i.e. with the drawn bend zones and angles).

Use this function to export several developments with different parameter settings of one Sheet Metal part as DXF, e.g. LVD and Bystronic.

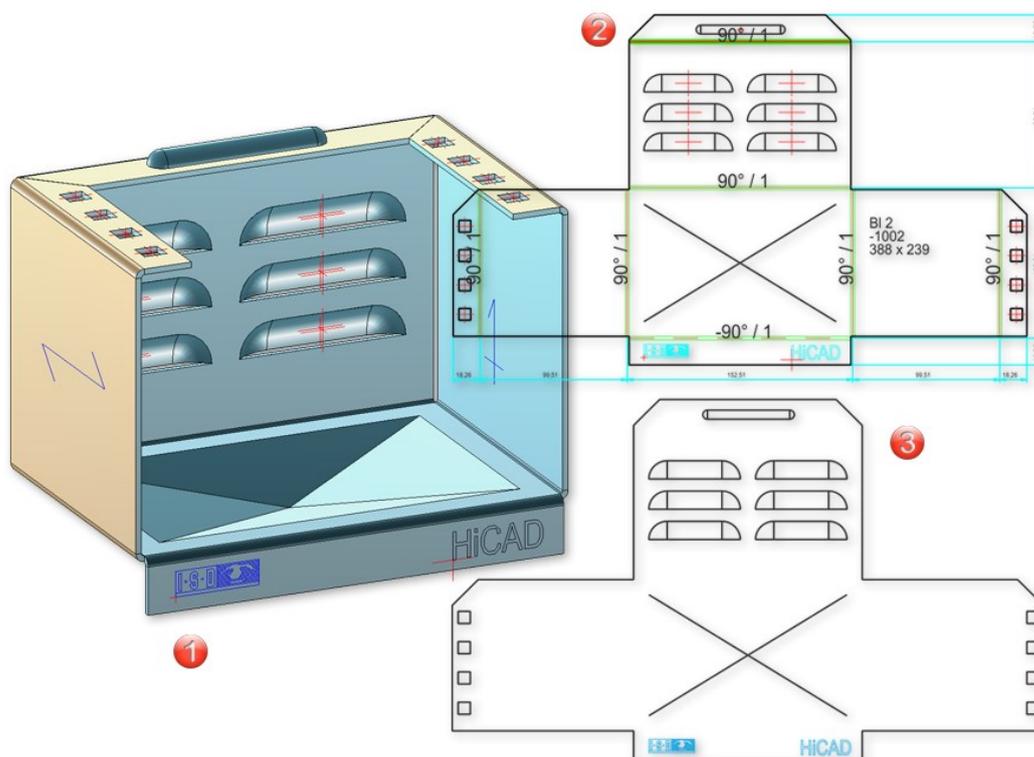
The file name can be configured in the Annotation Editor with a click on the  icon. To do this, apply, for instance, the part attribute "Item number" from the Sheet Metal part. If the part attributes set in the Editor cannot be applied to the sheets, you have the option to enter your own file name.

For the DXF export you need the file HCADACAD.DAT. It contains, for instance, the assignment of the HiCAD elements for specific AUTOCAD layers. This file can be customized and then saved with the file name extension .DA1, .DA2 or .DA3.

The selection of developments can either take place in the drawing  or by adding of all developments .

When you click on the **Export** button, all developments in the export list will be output as DXF files, according to the specified parameters.

The **Close & Report** button activates the Report Manager and creates a BOM. If you did not itemize the Sheet Metal part of the development in HiCAD, quantities will be set on 0 here



- (1) Sheet Metal part
- (2) DXF of the entire development
- (3) Section contour

### Export to COBUS NCAD

For DXF export you will need the file HCADACAD.DAT. This file contains:

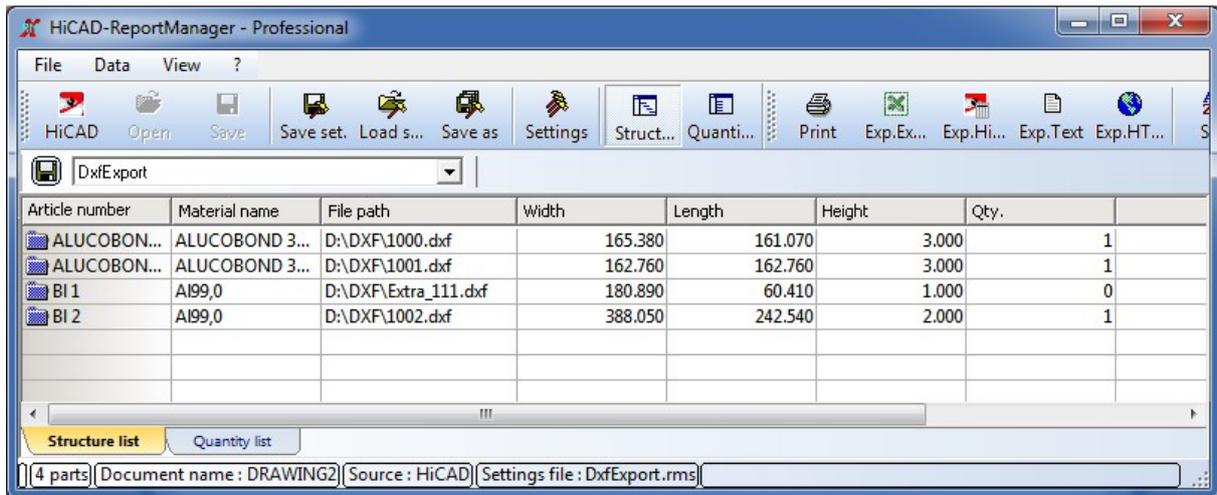
- the assignment of the HiCAD elements to specific AUTOCAD layers,
- the assignment of the HiCAD line types to the AUTOCAD line type,
- the correspondence between the symbols in HiCAD and the blocks in AUTOCAD, and
- the setting determining whether a complete DXF file including the HEADER section, TABLES section etc. is to be transferred, or only an ENTITIES section.

The file can be customized and is located in the HiCAD SYS directory. After customization, do not save the file with the extension .DAT, but with .DA1, .DA2 or .DA3.

For the export of developments to COBUS NCAD use the file HCADACAD\_COBUS.DAT in the HiCAD SYS directory. If you want to use this file, save it with the matching file extension (e.g. HCADACAD.DA1). Click here for more information.

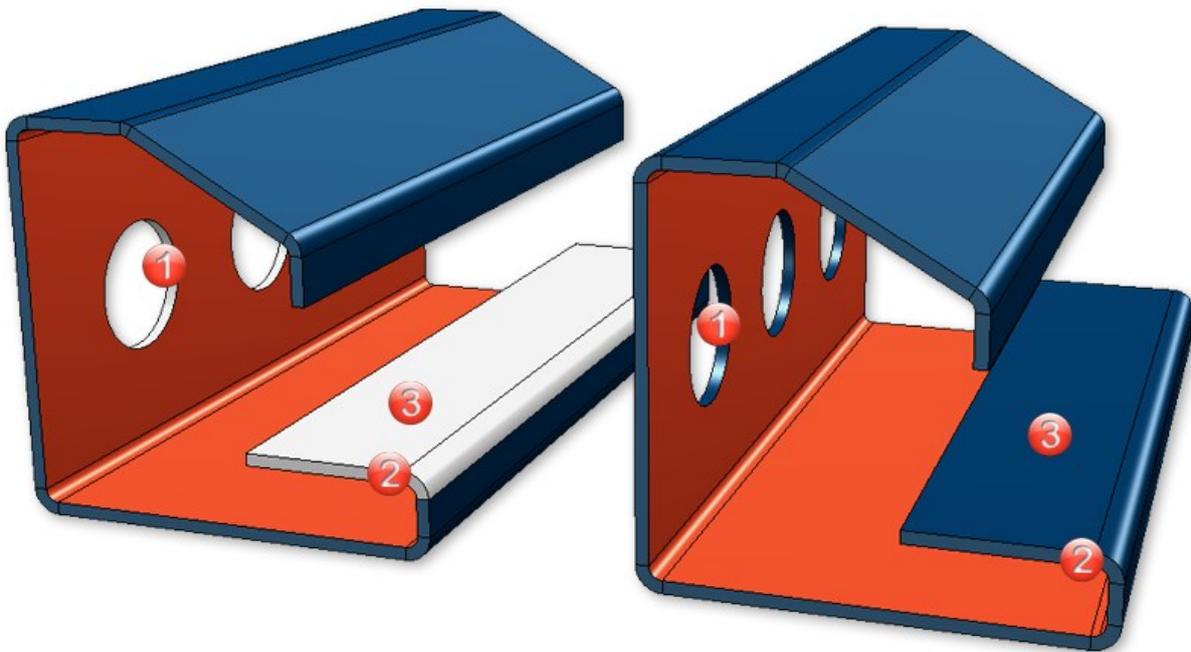
### Export of CSV files

To export a CSV file in the Report Manager, call the **Settings**  function and go to the **Export - Text** tab. Activate the option **Query file name before saving** and deactivate **Transfer header information**. Enter a comma or a semi-colon for **Separator for column**. Confirm with **OK**. Then, call the **Exp.Text**  function and choose the file type **\*.CSV**.



## Extend coating

When processing a coated Sheet Metal part, the coating will not be automatically applied to the respective processing (e.g. to new flanges, bores or corner processings). You can now use the new **Extend coating**  function to apply the coating to the new or processed elements. To change the coating colour, simply call the **Coating** feature.

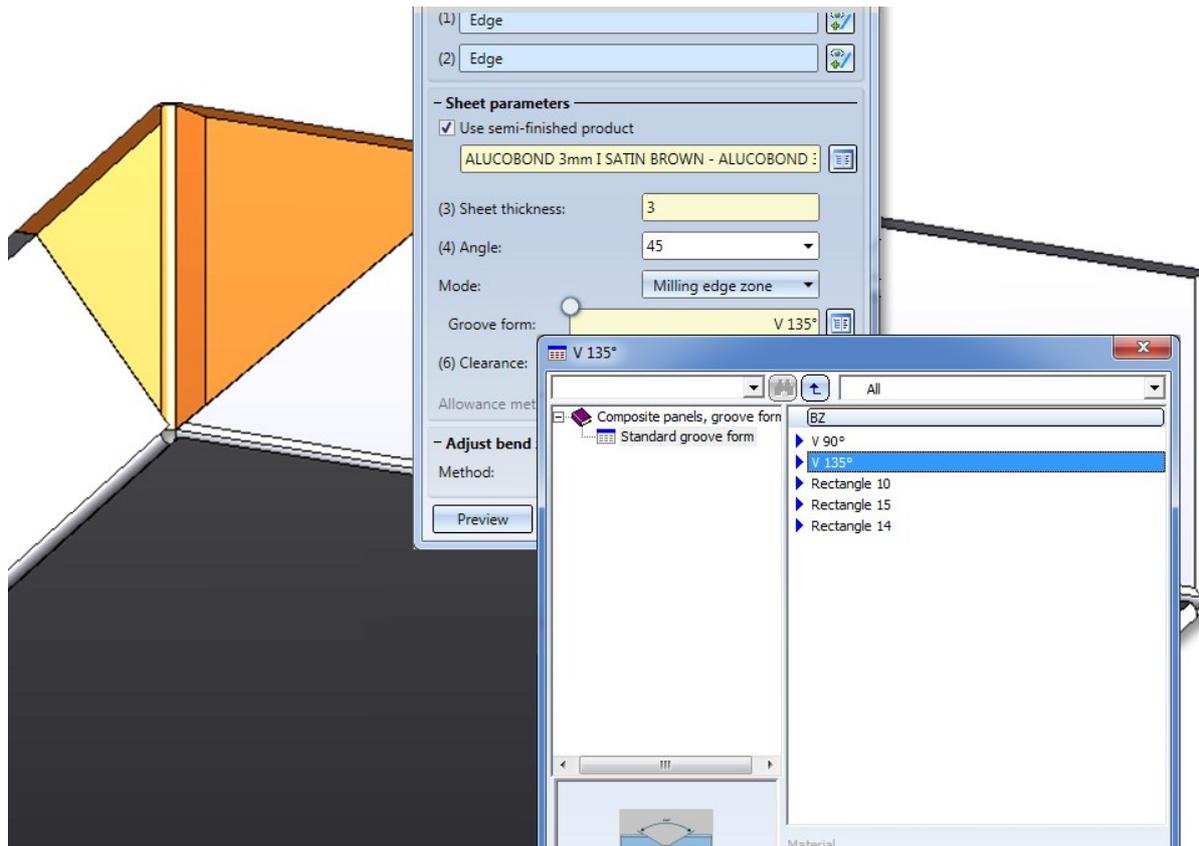


- (1) Front surface of bore (front surface like outer side)
- (2) Front surface of flange, bend zone (front surface like outer side)
- (3) Outer side of flange

## Welded corner for composite panels

Composite panels such as ALUCOBOND are deformed by means of milling edges. Therefore, the new parameters **Mode** and **Milling form** have been added to the **Welded corner** dialogue window.

In the **Mode** field you can choose between **Bend zone**, **Milling edge zone** and **Milling edge zone, inverted** options. For the milling edges you can then select, in the **Groove form** field, the matching milling tool from the catalogue **Factory standards > Composite panels, groove form**.



## Sheet development attributes

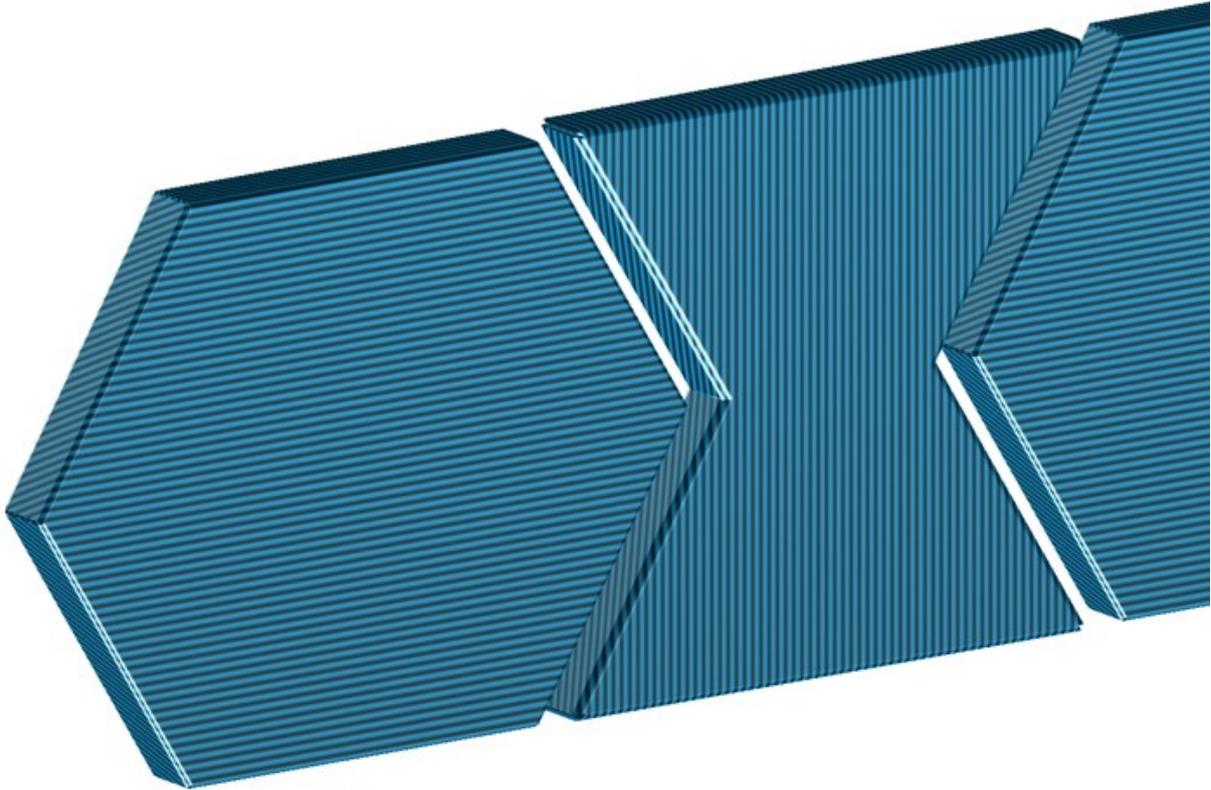
The system attributes

- §L2D Development length
- §B2D Development width
- §T2D Sheet thickness

will be considered for the transfer of the part attributes to the Report Manager. This does however not apply to data-base BOMs.

## Textures on sheets

The placing of textures on the individual sub-parts of a Sheet Metal part has been changed in such a way that the appearance of the assigned texture will be consistent - both with regard to the alignment of the texture and the transitions between neighbouring sub-parts:



Texture on composite panels

## Service Pack 1 2018 (V 2301)

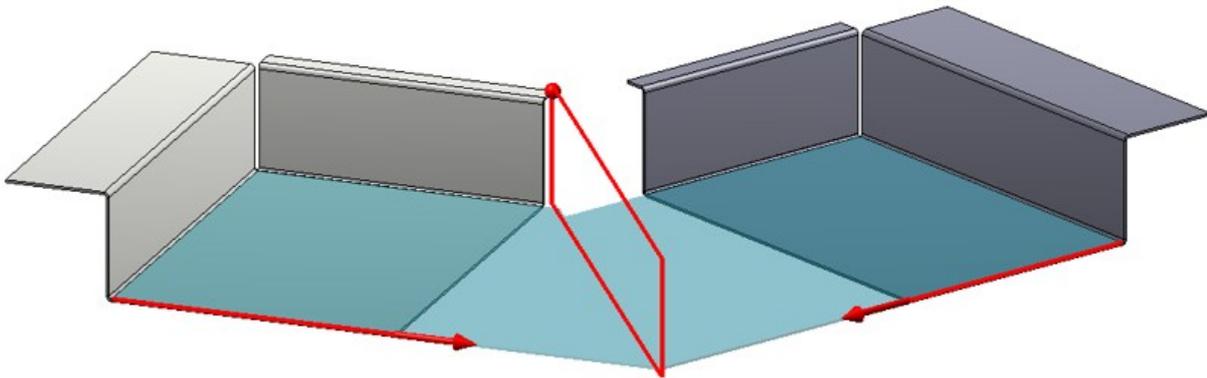
### Corner/Mitre

You can now also use the functions of the **Corner/Mitre** menu to process different Sheet Metal main parts. The functions

- **Corner processing/Mitre, betw. Sheets**  and
- **Corner processing/Mitre, Individual - betw. Sheets** 

have therefore be removed.

In the **Through point**  mode you can switch the start point of the cut plane from the inner side to the outer side. This means that the first point of the cut plane is located in the theoretical lengthening of the inner or outer edges of the flanges. Also, all point options are available to you.

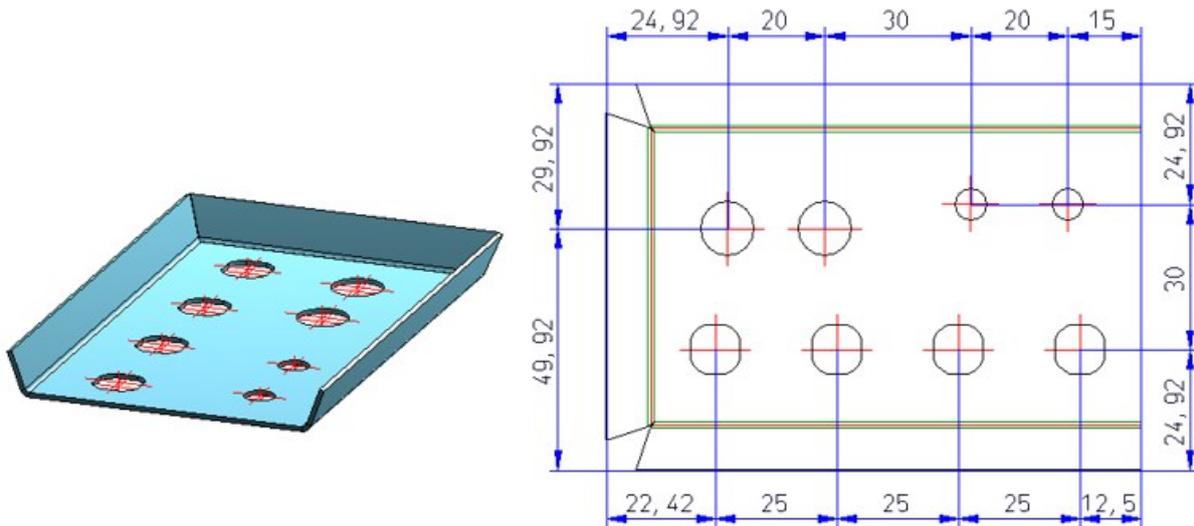


Mitre cut between different sheets  
First intersection point through outer edge, changed via right-click

## Development

### Dimensioning of processings

dialogue window for the sheet development parameters you can find the new **Processings** checkbox beneath **Dimensioning**. Activate this checkbox if you want bores and subtractions to be dimensioned.



Dimensionings for processings

### Adjust milling tool paths for production

The **Adjust milling tool paths for production** option refers to sheet corners where three milling edge zones converge. In the development, the lines do normally not converge in a common intersection point.

Therefore you have now the option to adjust the milling lines in such a way that they end in one common point.

It is important that only the milling lines are adjusted, and not the 3-D geometry.

If there are two converging milling lines, the milling lines will be lengthened / shortened, so that they converge in a common intersection point.

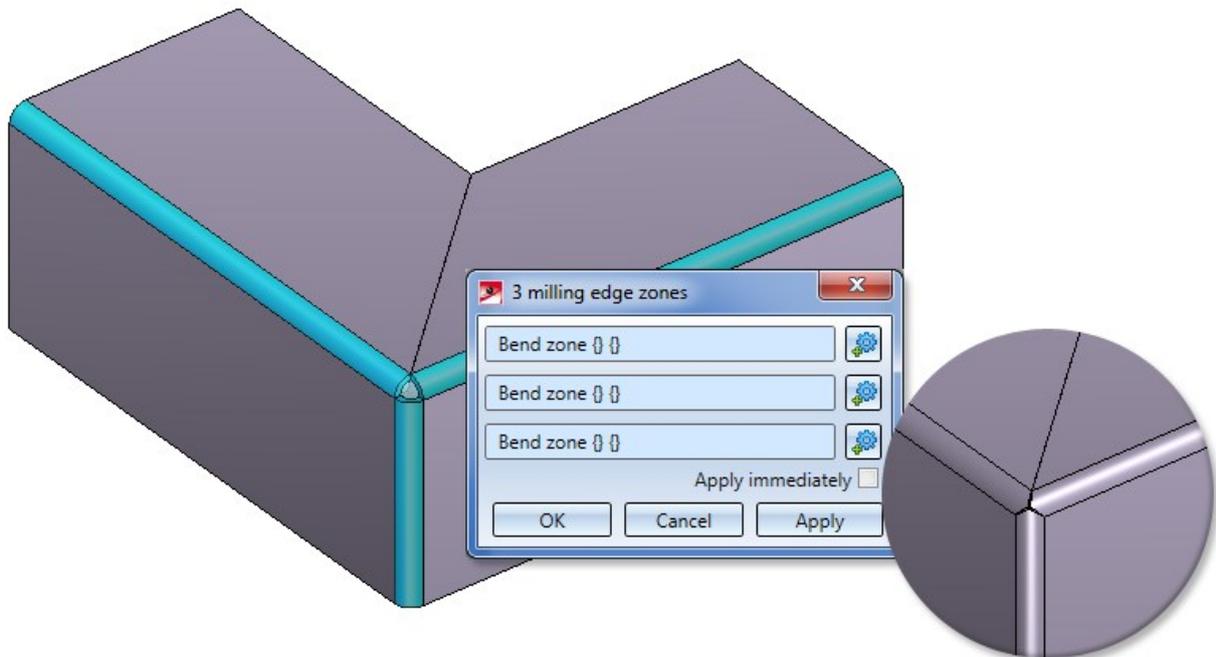
### Powder marking lines in developments

The parameters (Colour, Line type, Layer) of the powder marking lines can be specified via **Sheet development >**

**Update > Default setting**  and, in the dialogue window, **Extended settings** . The settings apply only to the 3-D sheet development.

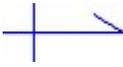
### 3 milling edge zones

Use this new 3 Milling edge zones  function to close a corner in your Sheet Metal construction with 3 milling edge zones. Just identify the milling edge zones and confirm the processing with **OK**.



### Text position on development

Use this the **Text position on development**  function to determine the position and direction of auxiliary texts on a sheet development. To evaluate the text position, you need to activate the **Auxiliary text** checkbox in the dialogue window for the sheet development parameters and choose the setting **Positioning: Inside**. If you select "Inside" but set no positioning symbol, the annotation will be placed as usual, i.e. HiCAD will search for a free space on the development.

In the catalogue you find the text position symbol  in various sizes in the **Text alignment symbol** table at **Factory standards > Symbols > Text alignment**. If desired, you can add your own sizes to this table.

The text position symbol is visible on the 3-D sheet, but invisible on the development on Layer 0.

The text position symbol is also considered for identical part search and referencing: Two sheets with different text position symbols are considered as "unequal".

### Enhanced "Attach flange" function

The **Attach flange** function now has two new reference and fitting modes. The Reference drop down list now offers the option to attach the flange angle-dependent to the inside or outside. The Fitting mode drop-down list offers the same options for connecting flanges.

## Source of article number determinable via semi-finished products

The **Article number** is editable, even if the created sheet is generated from a semi-finished product from the catalogue. The catalogue column BZ is not necessarily linked to the article number in the part attributes; BZ can be linked to any part attribute (e.g. \$01 Designation 1).

## Sheet from solid

The **Sheet from solid** function now also copies the article number of the original part.

## Hole patterns on sheets after bending simulation

You have now the option to insert hole patterns on developed sheets, i.e. after a bending simulation. In the process, the hole pattern (like bores) can be applied across multiple flanges and bend zones.

You can access this function at **3-D Standard > Standard processings > Bore**  **> Hole pattern**.

## Sectional views - Show/hide coating lines

The **Delete coating line** function for sectional views of sheets has been replaced with the new **Hide/show coating**

**lines**  function. You access the function by right-clicking the coating line in the sectional view and choosing the function in the context menu.

Coating lines can be hidden or shown in the active view or in all sectional views. One distinguishes between coating lines on the inner side and the outer side.

If all coating lines have been hidden, the lines can be redisplayed again via the context menu for sheets: In the sectional view, right-click the sheet and choose **Properties > Coating line**.

## Show/hide edges

The **Show/hide edges** function (at Views > Properties) has been enhanced (new **Processing planes** checkbox) and renamed to **Hide/show elements in view**.

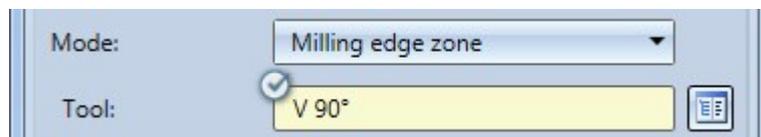
## Major Release 2018 (V 2300)

### Milling edge zone

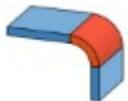
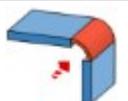
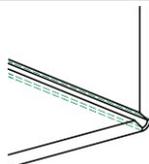
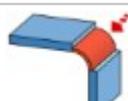
Composite panels, e.g. ALUCOBOND panels, can be deformed by means of milling and folding techniques. The resulting bend zone geometrically differ from the cylindrical bend zones. Therefore, HiCAD also offers the option to choose (besides bend zones) milling edge zones.

### Attach flange

When choosing the **Attach flange** function, you can now choose between bend zone and milling edge modes.



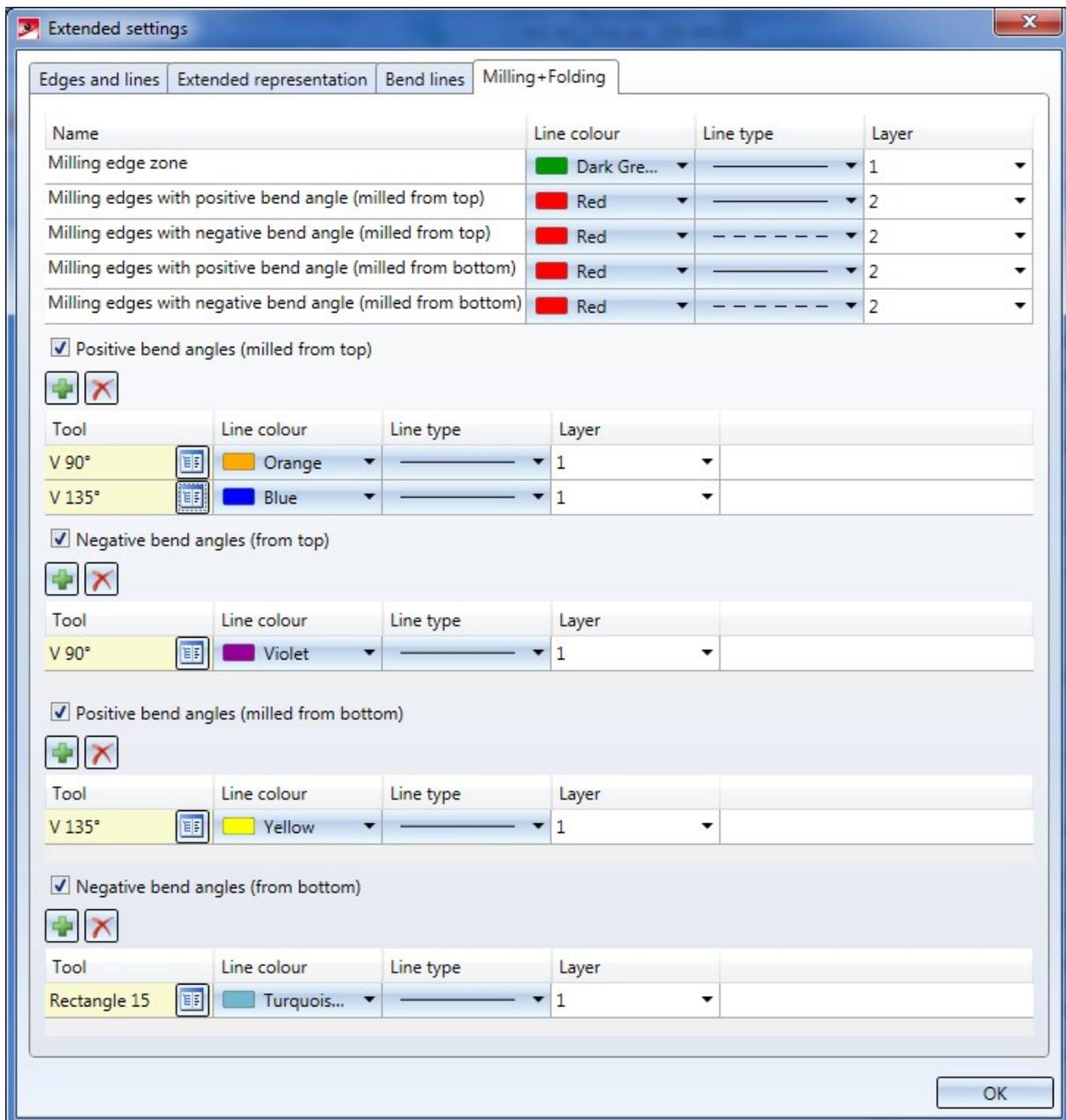
The following options are available:

	<p><b>Bend zone</b> The bend zone has the specified bend radius.</p>	
	<p><b>Milling edge zone</b> Bend zone with chosen tool (milled inside).</p>	
	<p><b>Milling edge zone, inverted</b> Bend zone with chosen tool (milled outside).</p>	

For milling edge zones you have the option to click the **Via catalogue** button next to **Tool** and choose the desired milling tool from the **Composite panels, groove form** catalogue.

## Milling + Folding

The milling tool path will be displayed if you have activated **Bend zone** checkbox beneath **Representation** in the Sheet development default settings dialogue. The appearance of the tool path is determined on the **Milling + Folding** tab of the **Extended settings** dialogue window that you open with a click on the  symbol.

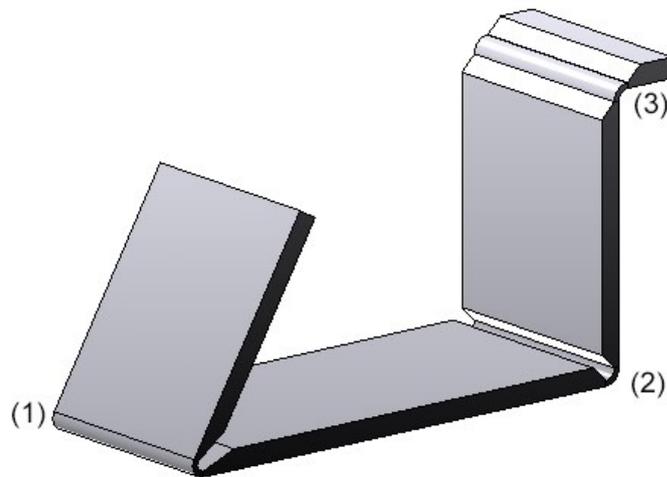


In the upper part of the dialogue you can specify the settings for **Line colour**, **Line type** and **Layer** of the different tool paths.

You can then specify tool-dependent line parameters for positive and negative bend angles; "milled from top" refers to the edge that you chose for Sheet alignment: Automatic (see above).

By the specification of tool-dependent settings the milling tool paths in the upper part of the dialogue may be overwritten.

The  icon adds a row for a tool from the catalogue. Click the  icon to delete a marked row again.

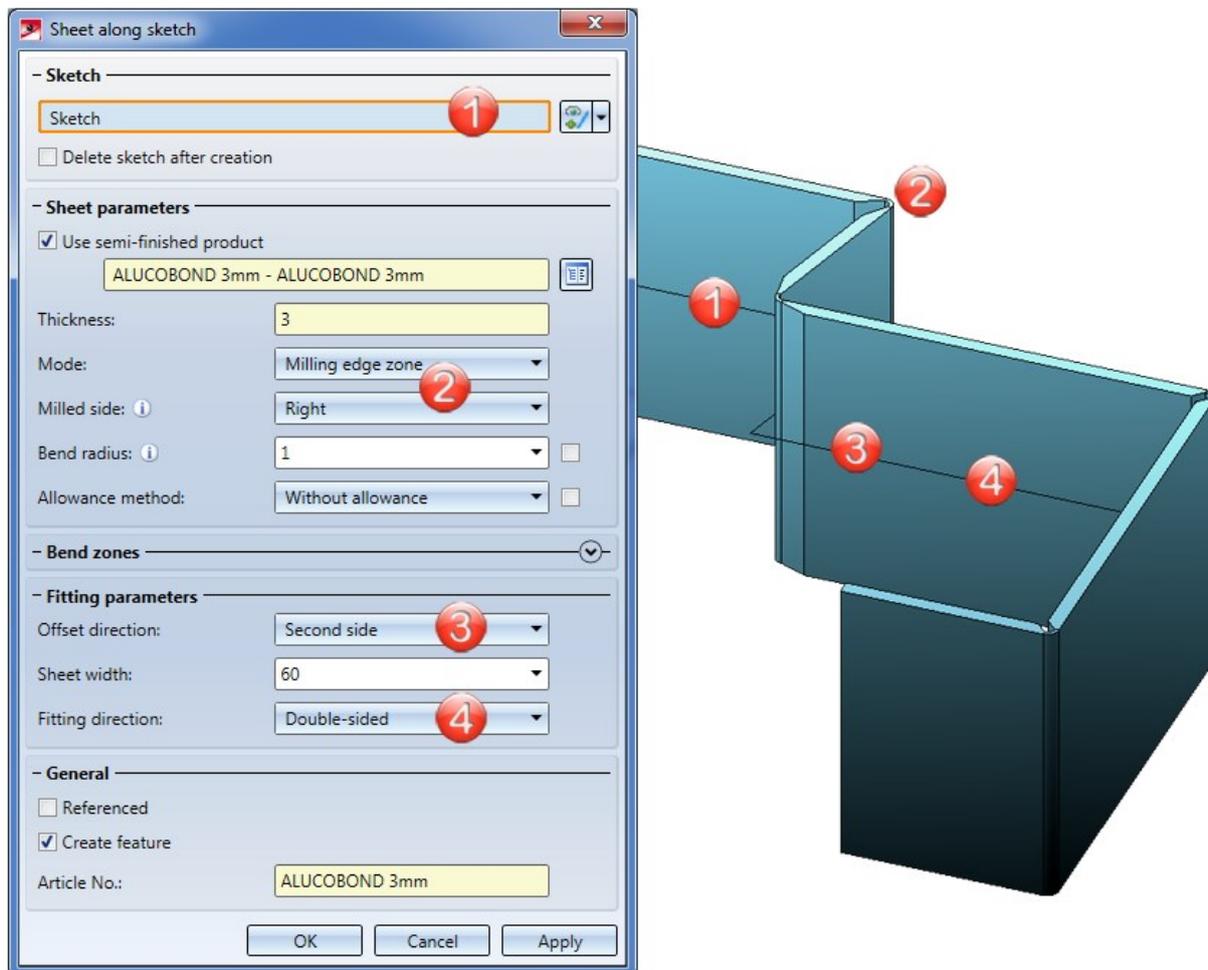


- (1) Positive bend angle, milled from top, Tool V 135°
- (2) Positive bend angle, milled from top, Tool V 90°
- (3) Negative bend angle, milled from top, Tool V 90°

The bend line text was expanded by the **Milling tool** attribute by clicking the  icon next to the corresponding option.

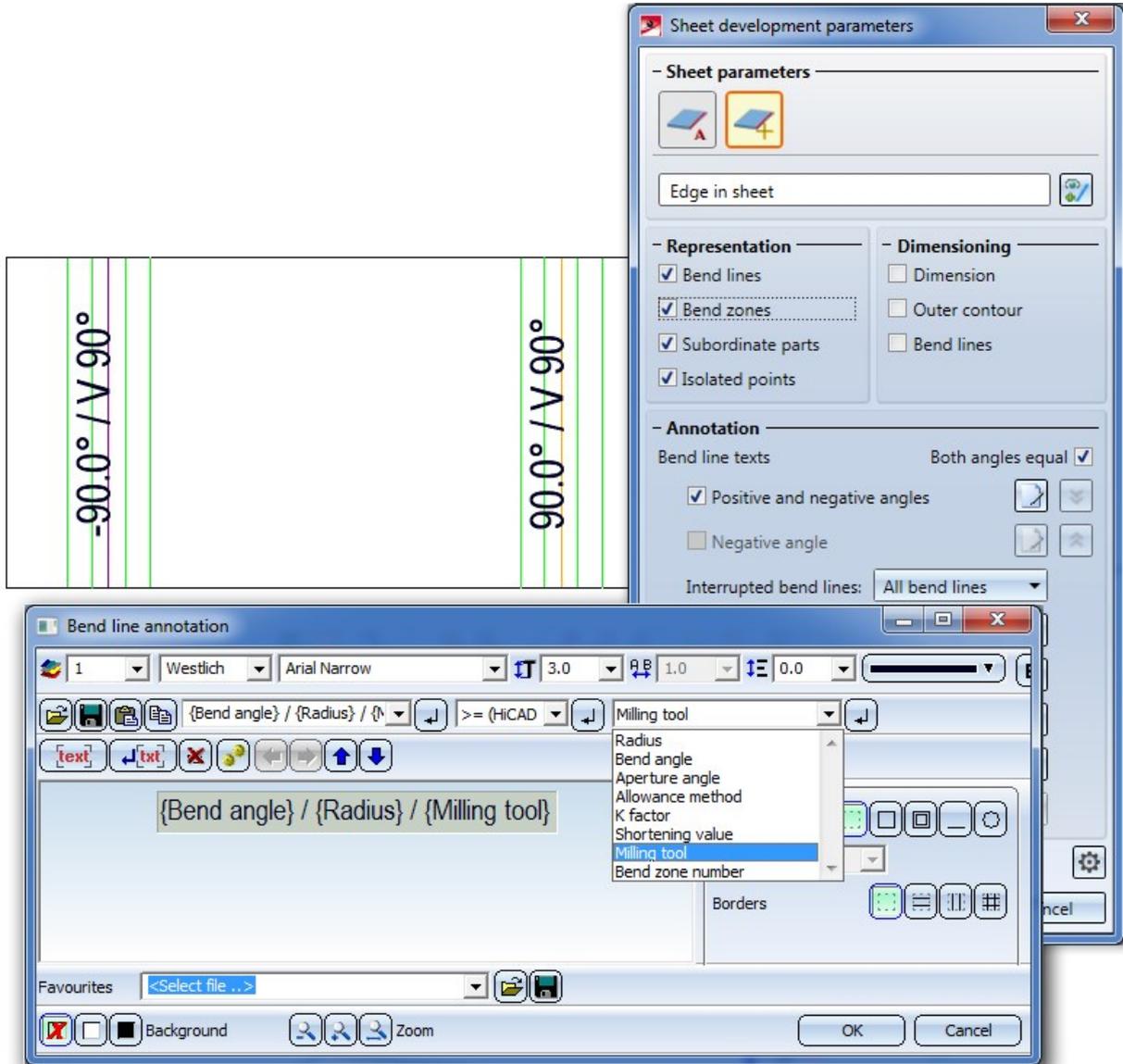
## Sheet along sketch

The **Sheet along sketch** now allows you to insert milling edge zones and/or bend zones. The pre-settings for the complete Sheet Metal part are entered in the **Sheet parameters** area. In the **Bend zones** area or (later) via the Feature, you can specify a tool from the catalogue for each milling edge zone, as well as an individual setting for the milled side.



### Milling tool for bend zone

The output of the milling tool can be selected in the Annotation Editor via the attributes. The tool that was chosen for milling edge zone creation will be output at the bend line.



### Milling tools for processing of composite panels

New in the catalogue are milling tools for processing of composite panels. These can be found at **Factory standards > Composite panels, groove form**.

## Adjust bend zone



The **Adjust bend zone** function also adjusts the new milling edge zones.

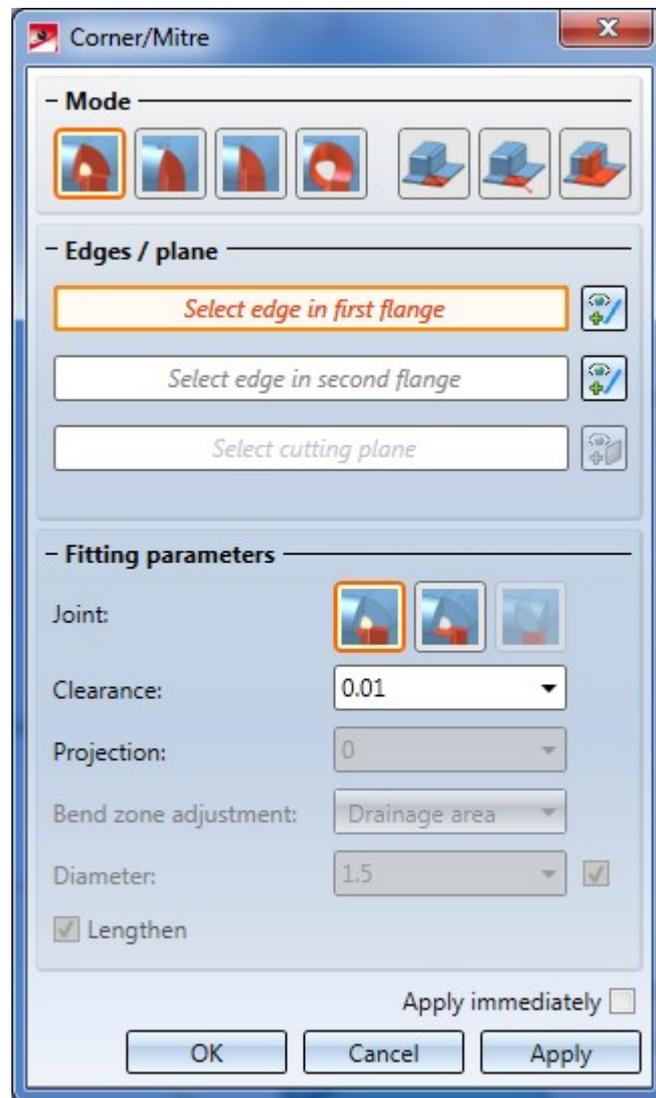
## Corner/Mitre function group

The "Process corner" and "Mitre" function groups have been combined into one function group called **Corner/Mitre**.

Corner/Mitre	
	Corner processing/Mitre, Sheet
Pull-down menu 	
	Narrow side
	Corner processing/Mitre, betw. Sheets (moved to the Corner/Mitre menu as of SP1)
Pull-down menu 	
	Corner processing/Mitre, Individual - betw. Sheets (moved to the Corner/Mitre menu as of SP1)

## Corner processing / Mitre - individual sheet

When you choose **Sheet Metal > Corner/Mitre > Sheet**  a revised dialogue window containing various functions for mitre cuts and corner processings will be displayed:



Proceed as follows:

1. First select a **Mode**.

Depending on the selected mode, some further options in the dialogue may be greyed out.

- If you choose the **Close corner, free** mode, you have, beneath **Fitting parameters** and next to **Joint**, the option to close the inner edges flush or according to a specified percentage. You enter the percentage in the **Projection** input field.
- If you choose the **Close corner, closed** mode, you have, beneath **Fitting parameters** and next to **Joint**, the option to close the inner edges flush or according to a specified percentage. Furthermore, you can close the outer edges flush.
- If you choose the **Mitre** or **Mitre, with neighbours** mode, you can activate or deactivate the **Plane, automatic** checkbox, specify a setting for **Bend zone adjustment** (Hollow (outside), Bulgy (inside), Linear, etc.), and **Lengthen** the flanges.

2. Select the two flanges by identifying an edge (right-click front edge).

If you want to identify a different edge, click the  icon beneath **Edges/plane** to identify the edge.

3. Specify the **Clearance**.

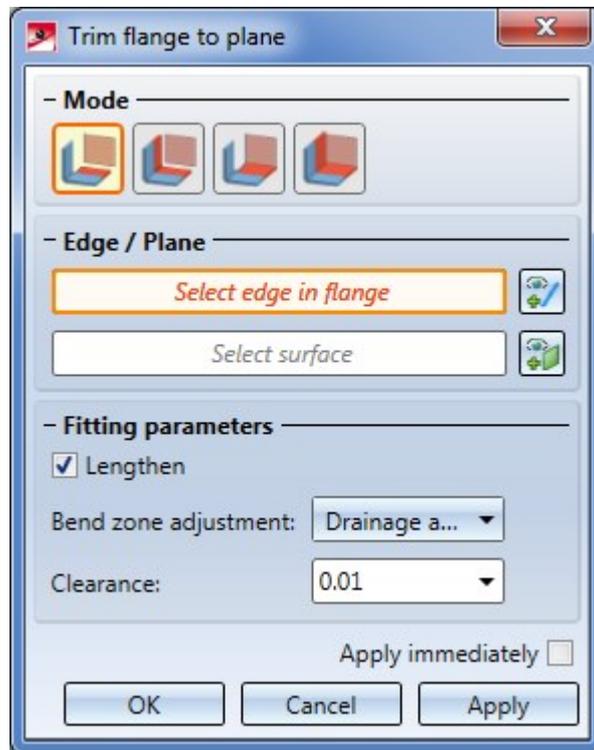
As soon as you have entered all required data, a preview will be displayed.

If you click **Apply**, the processing will be applied while the dialogue window remains open, enabling you to make further adjustments if required. If you click **OK**, the processing will also be applied, but the window will be closed.

Corner/Mitre dialogue window - Possible options	
	Mode: <b>Close corner, free</b> bend zone
Fitting parameter <b>Joint</b>	
	Inner edge, flush
	Projection, percent
	Mode: <b>Close corner, closed</b> bend zone
Fitting parameter <b>Joint</b>	
	Inner edge, flush
	Projection, percent
	Outer edge, flush
	Mode: <b>Close corner, drainage area</b>
	Mode: <b>Close corner, round</b>
	Mode: <b>Mitre</b>
	Mode: <b>Mitre, Through point</b>
	Mode: <b>Mitre, with neighbours</b>

## Trim sheets

On the **Change length** function group of the **Sheet Metal** tab you find the **Trim**  function. When you call the function, a new, revised dialogue window that offers different trimming options will be displayed:



Proceed as follows:

1. Select a mode.

Depending on the selected mode, some options may be greyed out.

- The modes **Individual, with clearance** or **Individual, flush** offer the option to trim the identified sheet flange with lengthening if desired.
- If you have chosen the mode **Plane, with clearance** or **Plane, flush**, all sheet parts of a plane will be trimmed to a cut surface.

2. Identify the longitudinal edge of the flange to be trimmed, or right-click and select the front side.

If you want to identify a different flange, click on the  icon beneath **Edge / Plane** and identify the desired edge.

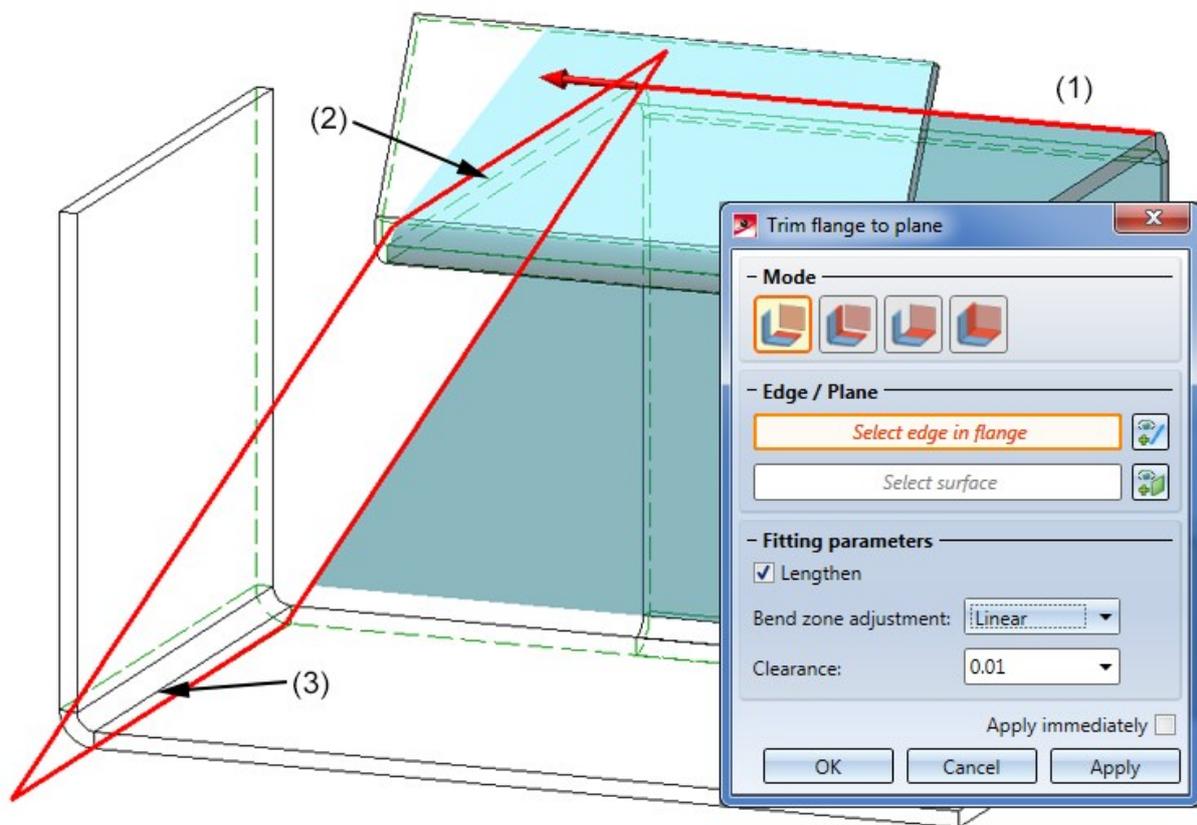
3. Now identify the cut plane, e.g. by specifying 3 points, 2 edges or 1 surface.

Here, too, you can select a different plane by clicking on the  icon.

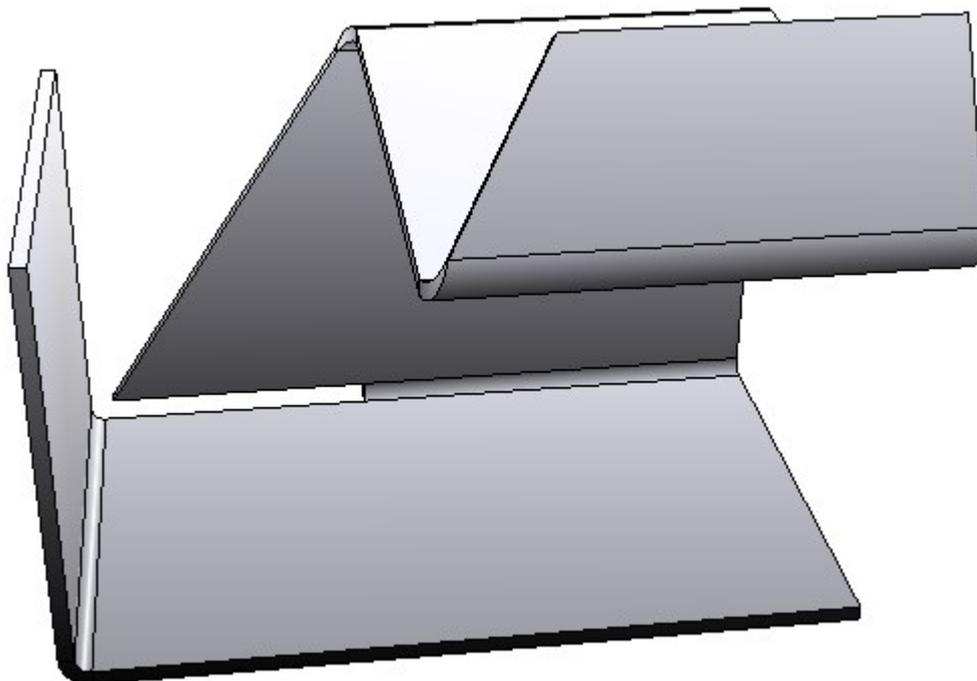
4. Activate the **Lengthen**  checkbox if there are flanges that do not reach up to the cut plane.
5. Choose a **Bend zone adjustment**
6. If required, enter a Clearance.
7. Apply the preview with **OK**.

If you click **Apply**, the change will also be inserted, but the window will remain open, allowing you to trim further sheets. If you select **Cancel**, the window will be closed, and no change will be applied. If you have activated the **Apply immediately**  checkbox, the entered data will be applied instantly.

Subsequent changes can be made via the Feature log: Double-click on the **Trim to surface** feature and change the data in the displayed dialogue window.



- (1) Edge of the sheets to be trimmed
- (2) First edge of cut plane
- (3) Second edge of cut plane

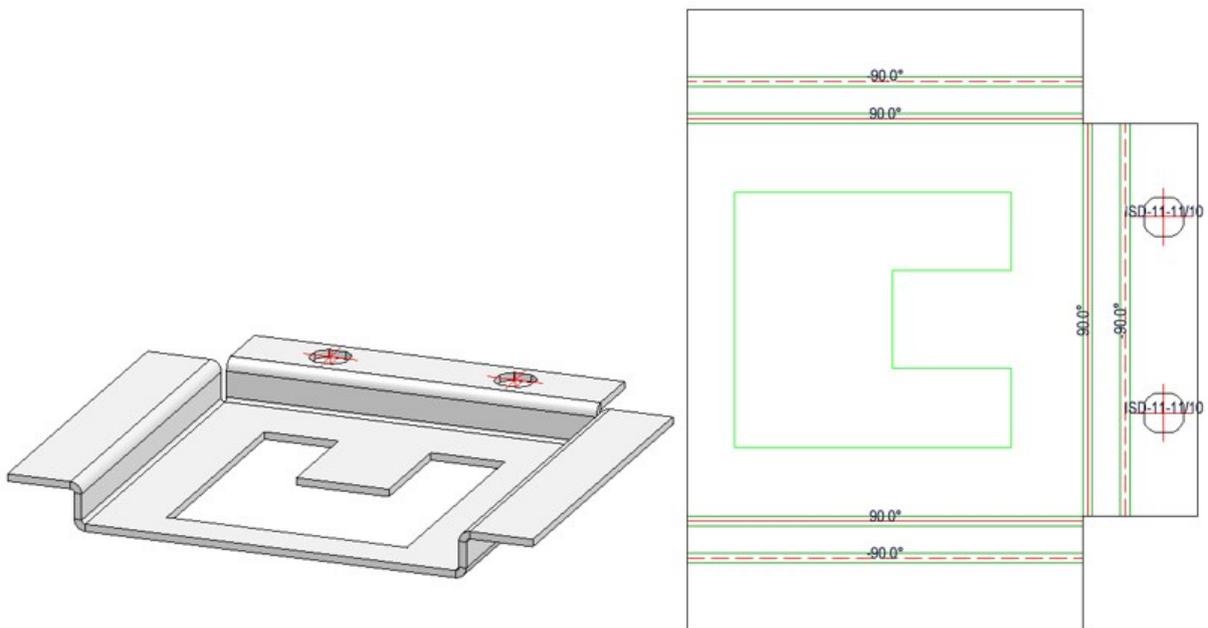
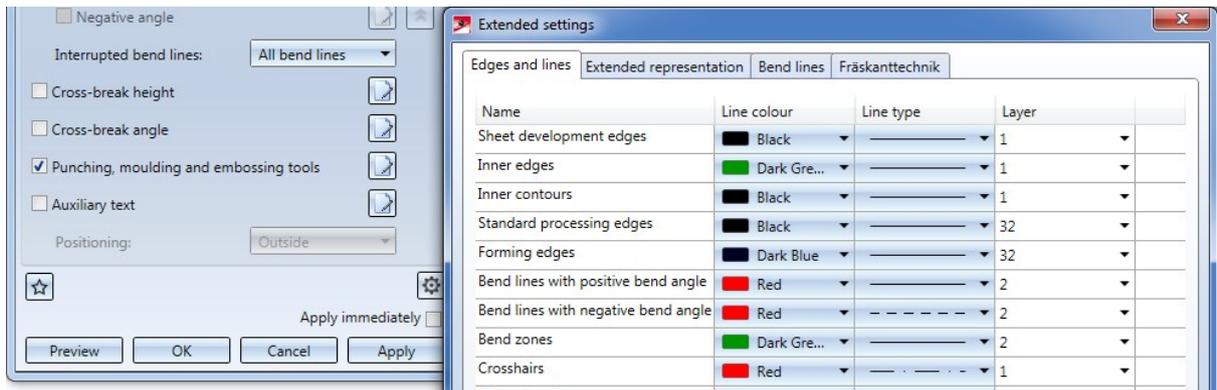


## Inner contours in sheet developments

It is now possible to distinguish between the lines of outer and inner contours in developments. Inner contours are all contours that can be cut out of the sheet and are no standard processings.

This allows you, for instance, to define the adjustment of the layers for the DXF export more accurately.

Colour, type and layer can be specified in the **Extended settings**  of the development parameters.



Inner contours have the line colour Green.

## Analytical development

The analytical development with the functions

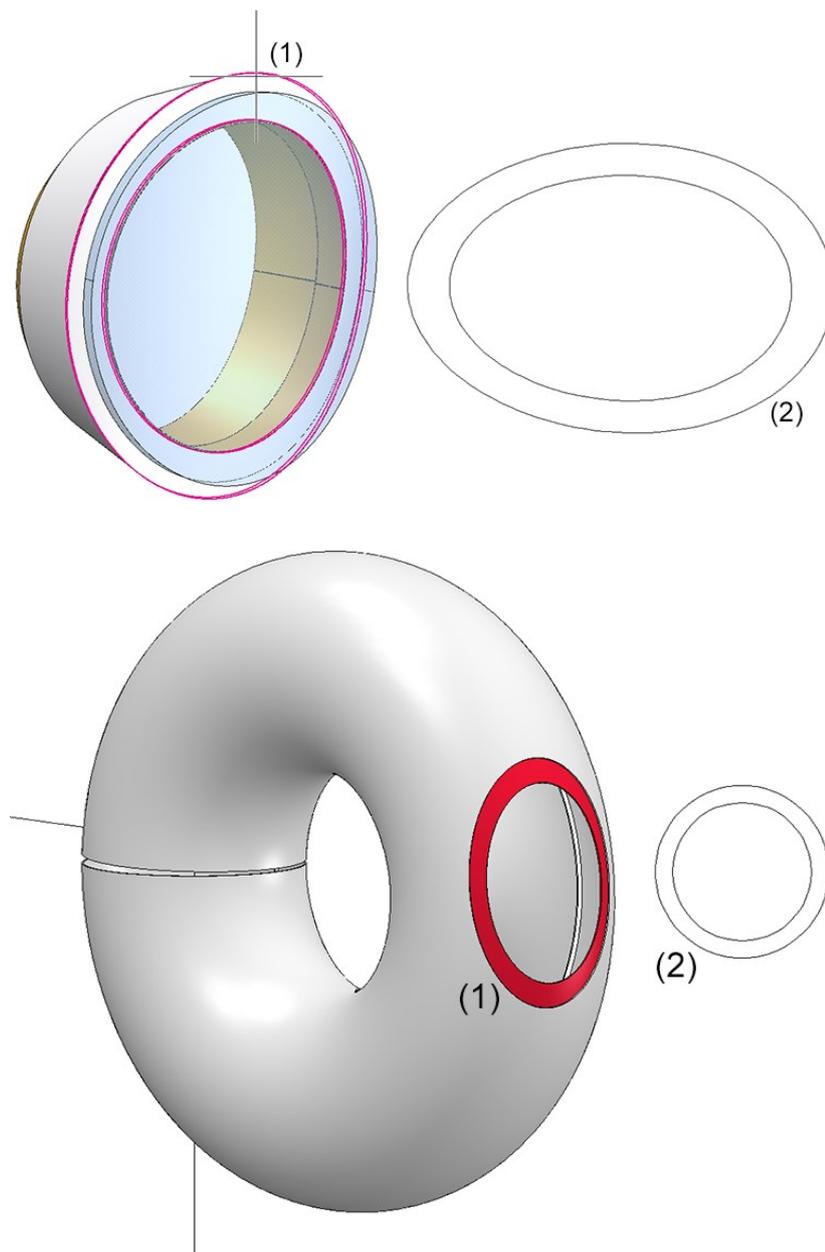
- **Surface (analytical)**
- **Surface on neutral axis (analytical)**

now offers the option to develop facets lying on a torus.

This takes place approximatively by means of a double development, first with regard to the torus axes and then, once more, with regard to the cylinder axis resulting from this development.

Therefore, the following warning message will be displayed:

The development contains a torus that might not have been developed correctly. Please check the result!

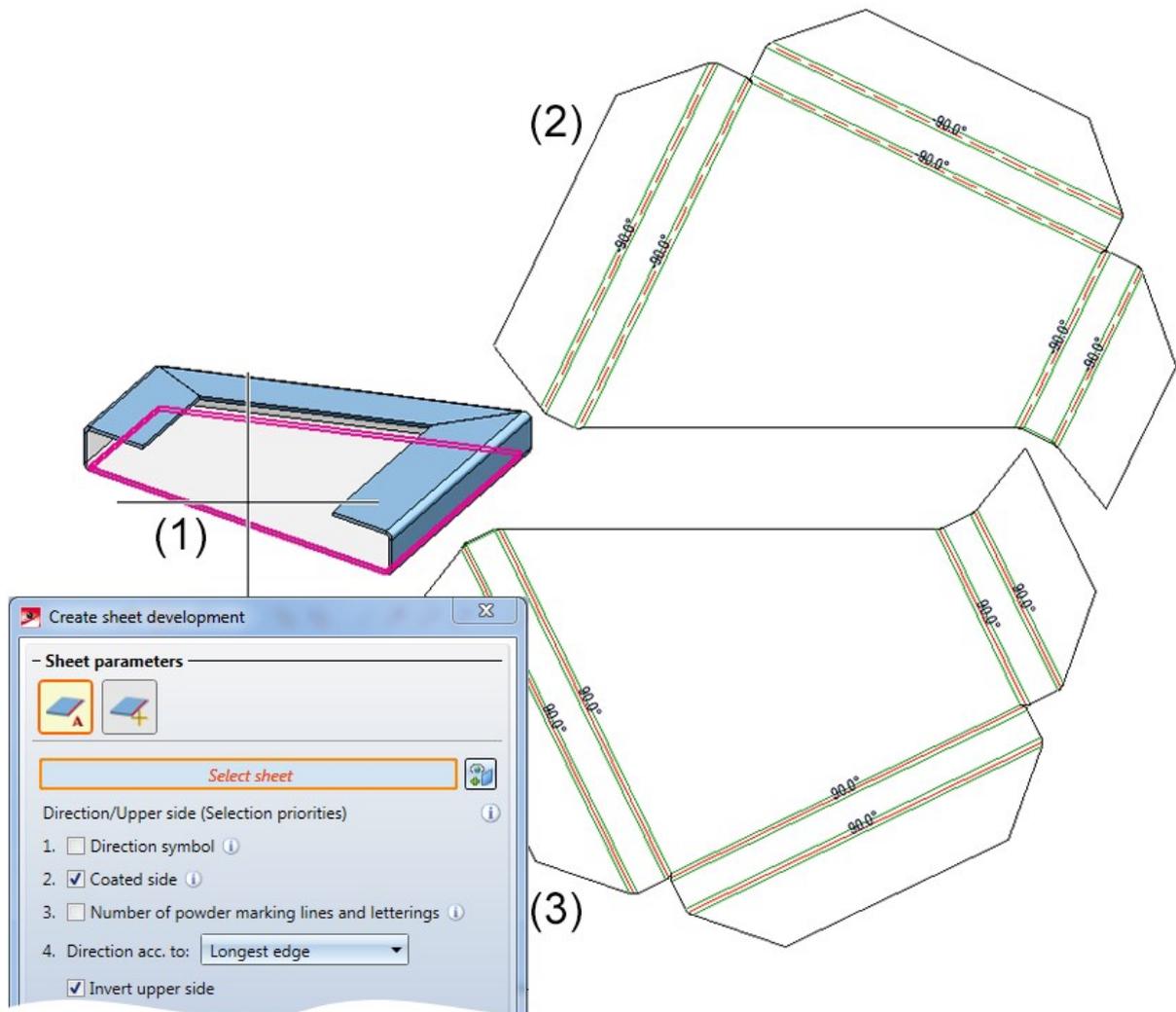


(1) Facet  
(2) Development

## Develop bottom side

In the **Create sheet development** dialogue window you can specify in the **Sheet alignment: Automatic** mode the criteria according to which the upper sheet part is to be determined for development creation. For instance, you can generate the development in such a way that the coated side will be the upper side.

When milling composite panels the coated side is normally located at the bottom. With the **Invert upper side** option you can flip the upper side to the bottom.

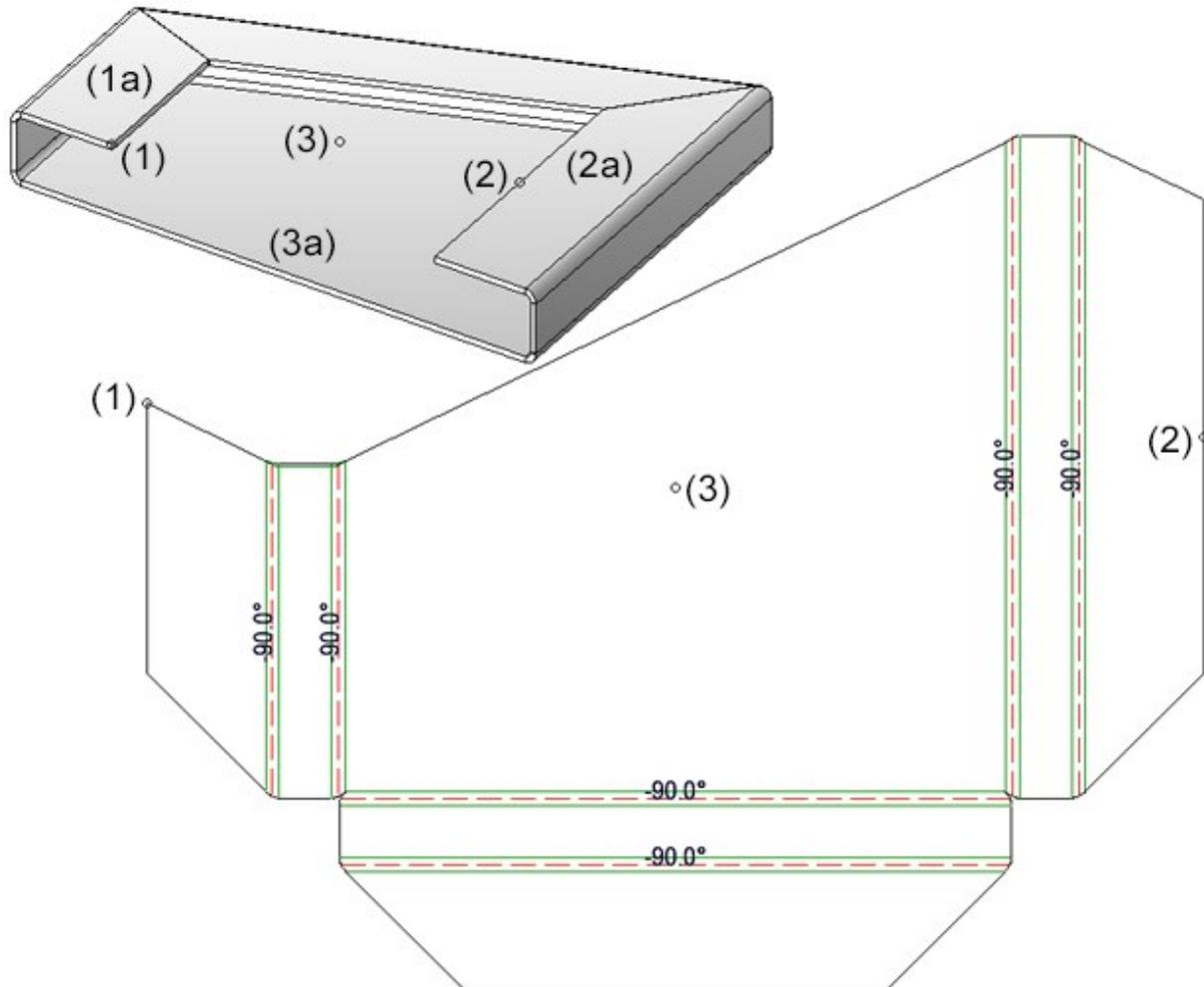


- (1) Sheet Metal part
- (2) Development of coated side
- (3) Development of coated side, inverted

## Isolated points in sheet developments

In sheet developments you can now define whether isolated points of the Sheet Metal part are to be hidden or shown. The position of the points (in the development) refer to the flange or the bend zone that was active during creation. If the main part is active, no points will be developed.

The setting affects only the **Isolated 3-D points**  supplied by the Sheet Metal part. Isolated points in the development that have been created subsequently are always visible.



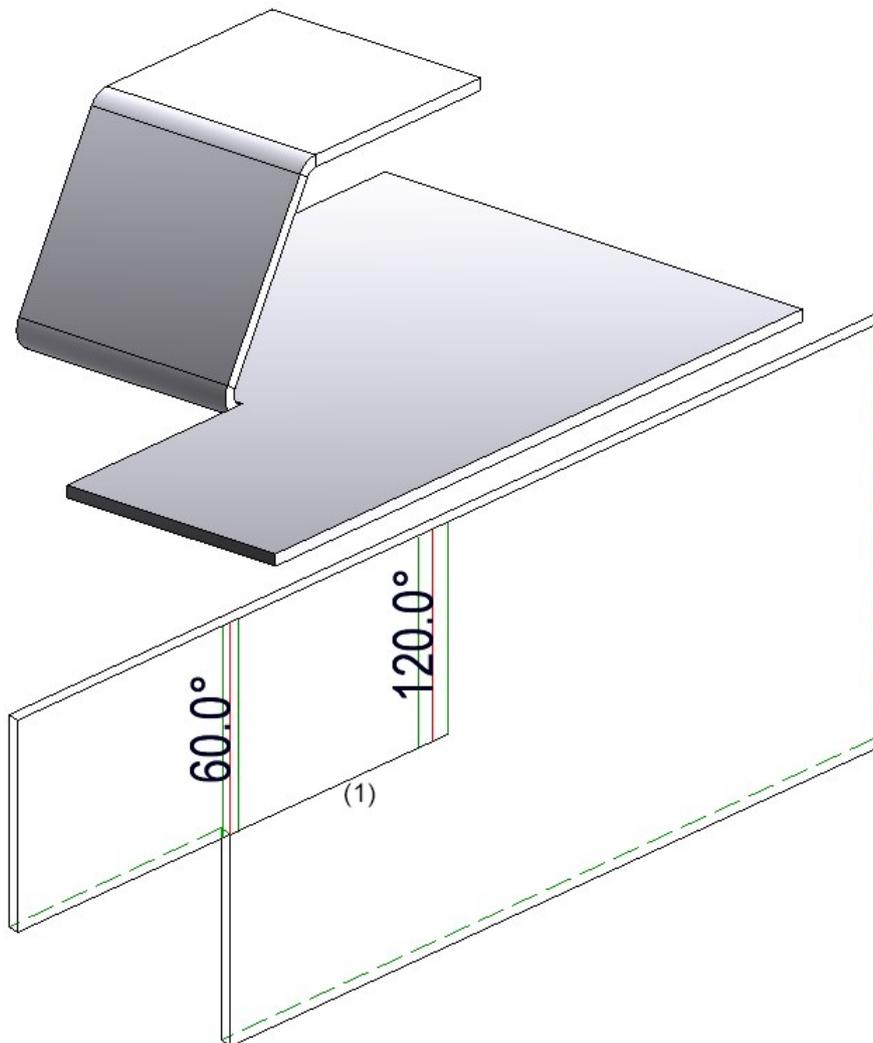
- (1) Isolated point, (1a), active sheet flange
- (2) Isolated point, (2a), active sheet flange
- (3) Isolated point (3a), base sheet active

## Separating lines in developments

If two flanges touch one another (without clearance) during development, a line representing a "separation cut" will be inserted into the development (see image below). If the front face of a bend zone directly touches a flange, such a separating line will be inserted here, too.

Through the merging of flanges and bend zones during a sheet development, edges that belong to the outer contour and that will be needed for blank creation can be lost. Therefore, those separating lines will be inserted, as free edges, at their position in the development.

The separating lines are inserted at positions where contour edges are identical before the merging, and where at least one of these identical edges originates from a flange.



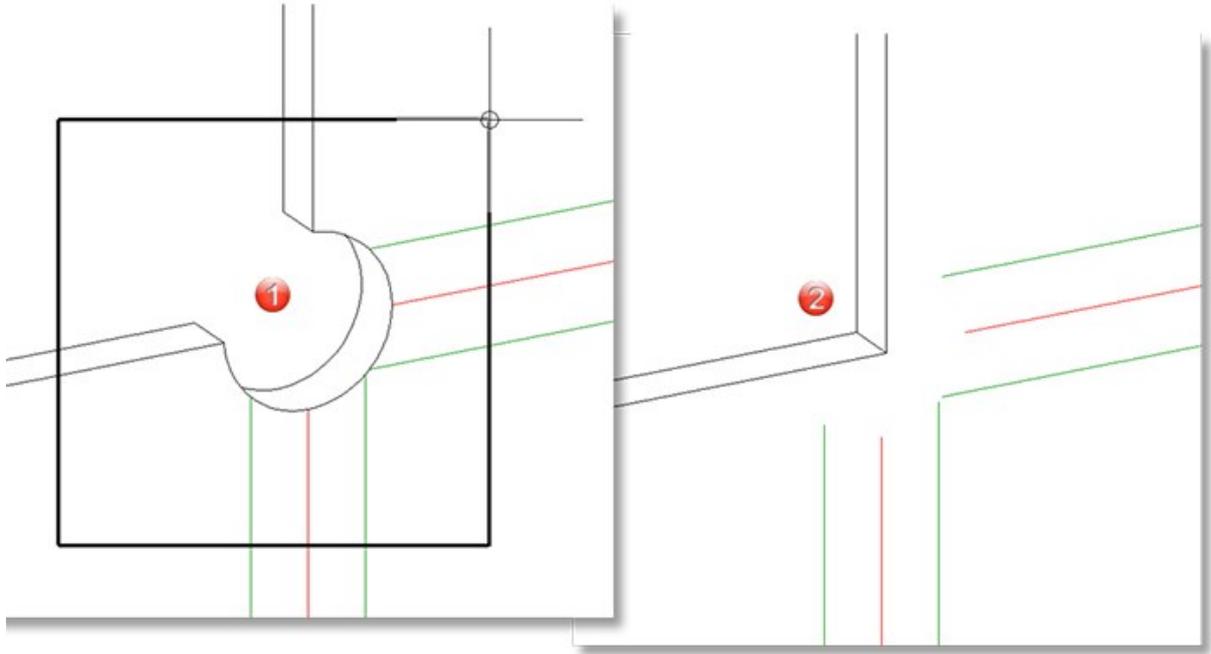
(1) Separating line

## Acute corner

You can use the **Acute corner** function to simplify developments.

1. Specify 2 diagonal points to define a rectangle around the corners of the development to be simplified.

The corners will then be simplified. You can then process further corners of the development or end the function by pressing the middle mouse button.



## Processing functions for sheet developments

Processings of developments by the functions

- Delete crosshairs 
- Delete point 
- Line: Vector menu 

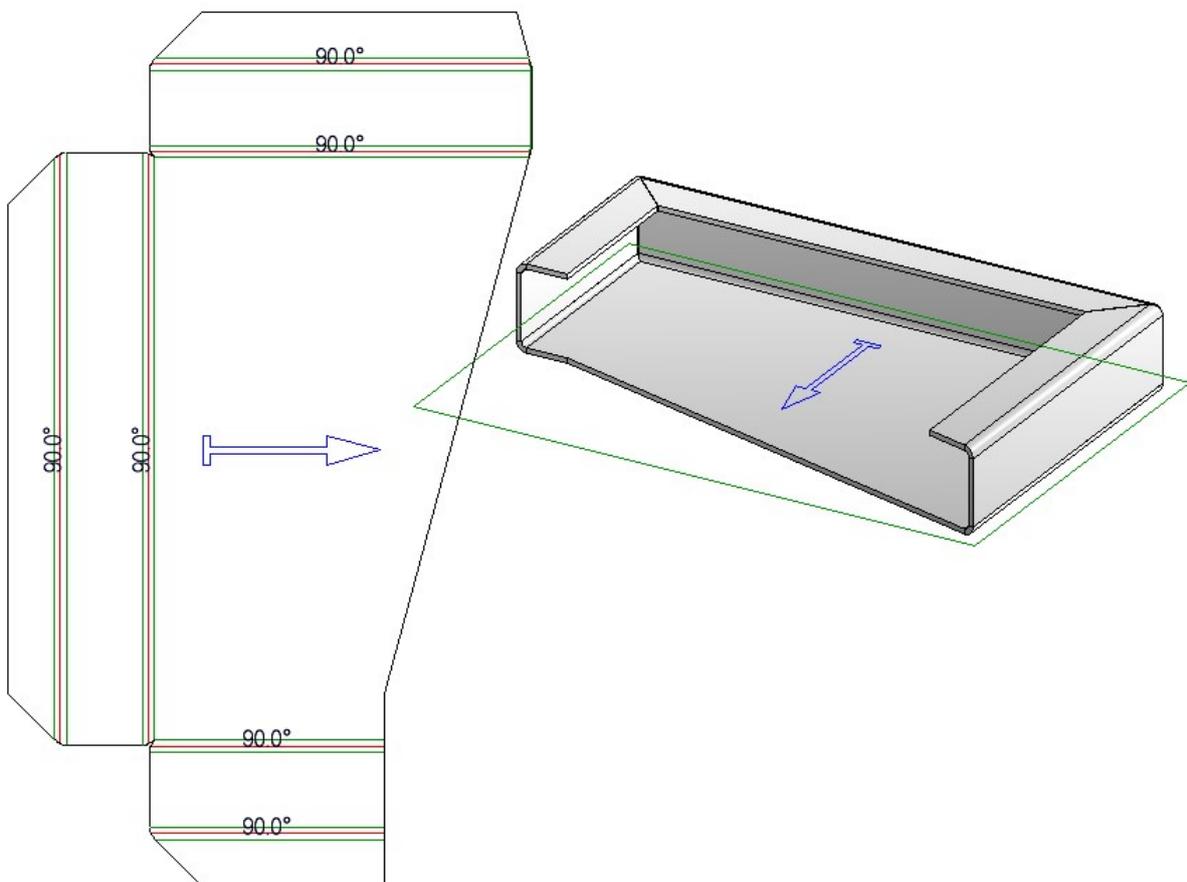
that are offered to you when you right-click a sheet development and choose **Process development** will now be retained even after a Feature recalculations and development updates.

## Additional symbols for processing direction

The **Direction arrows** table (atWERKSNORMEN/SYMBOL/BOLE/PFEILE) now contains the new symbols **Direction symbol Composite panel** and **Direction symbol, one-sided**. With these symbols the direction of coatings or textures can be determined. When creating sheet developments the direction can be optionally used for the alignment of the development.

The following **Direction arrows** can now be inserted:

- Direction symbol 
- Direction symbol Composite panel  and
- Direction symbol, one-sided .



## Hole patterns

The new **Hole pattern** function allows you to insert regular patterns of defined holes into parts or sheets.

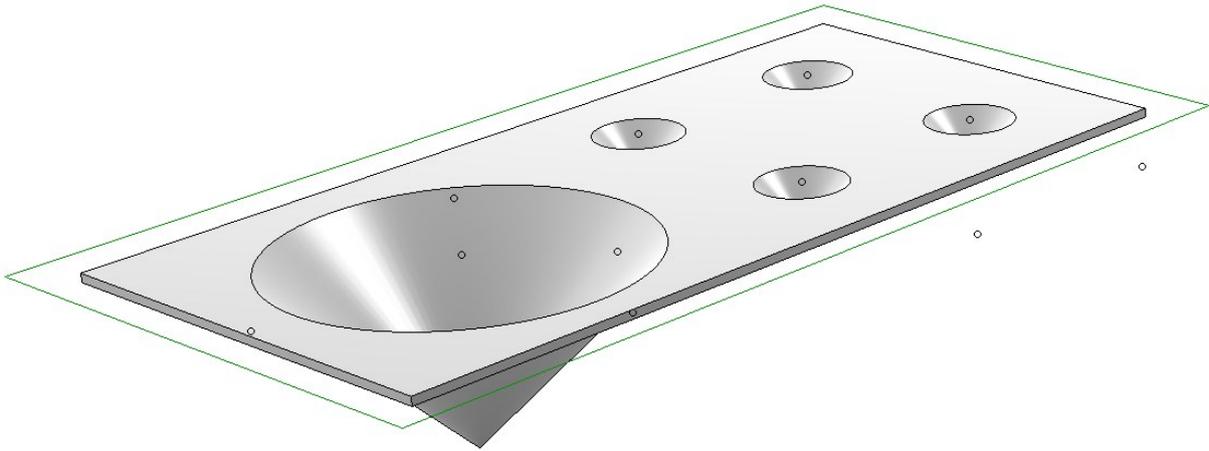
Possible hole types are round holes, square holes, slots and special holes that can be defined with the help of sketches. The holes can be arranged in straight, staggered, diagonally staggered or freely staggered rows according to DIN.

## Create own moulding tools

You can add your own moulding tools to the library. For this you need at least 2 elements: One sketch used by HiCAD to create a material subtraction in the sheet to be processed, and a part containing the actual processing.

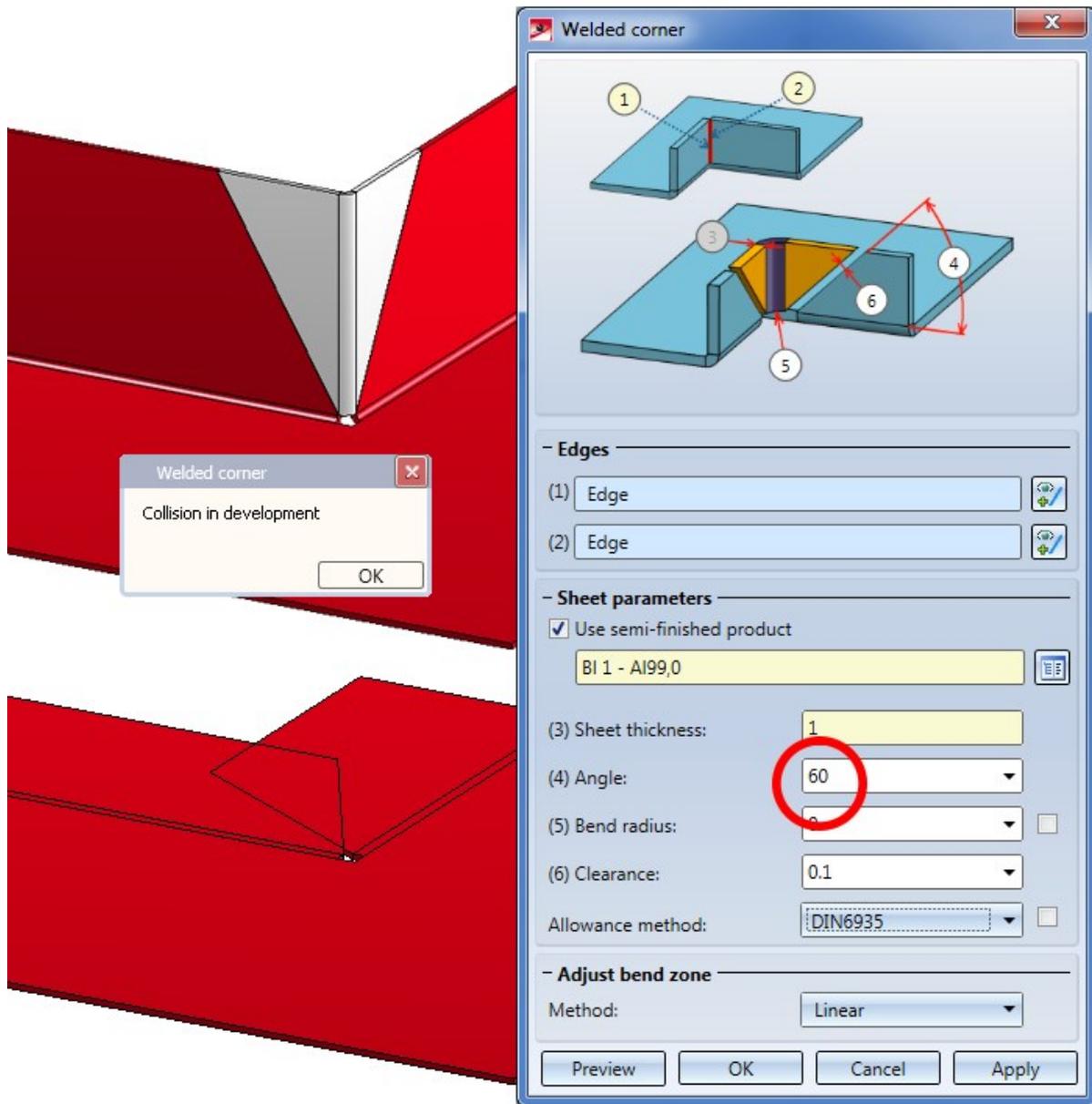
When applying the moulding tool, HiCAD initially uses the sketch to create a subtraction in the sheet; then, the part will be positioned and added to the sheet.

Both parts can be parameterized. The thickness of the sheet to be processed will be used as parameter **d**; other parameters for the size of the processing can be stored in the catalogue in order to be able to apply a moulding tool with many different sizes (see Example).



### Collision check when creating welding corners

The welded corner will also be inserted in cases where the affected flanges of the original sheet overlap in developed state. In such cases an error message will now be issued.



### Bending simulation

The **Bending simulation**  function can now be found in the **Process** function group of the **Sheet Metal** tab.

## 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 2018 (V 2302)

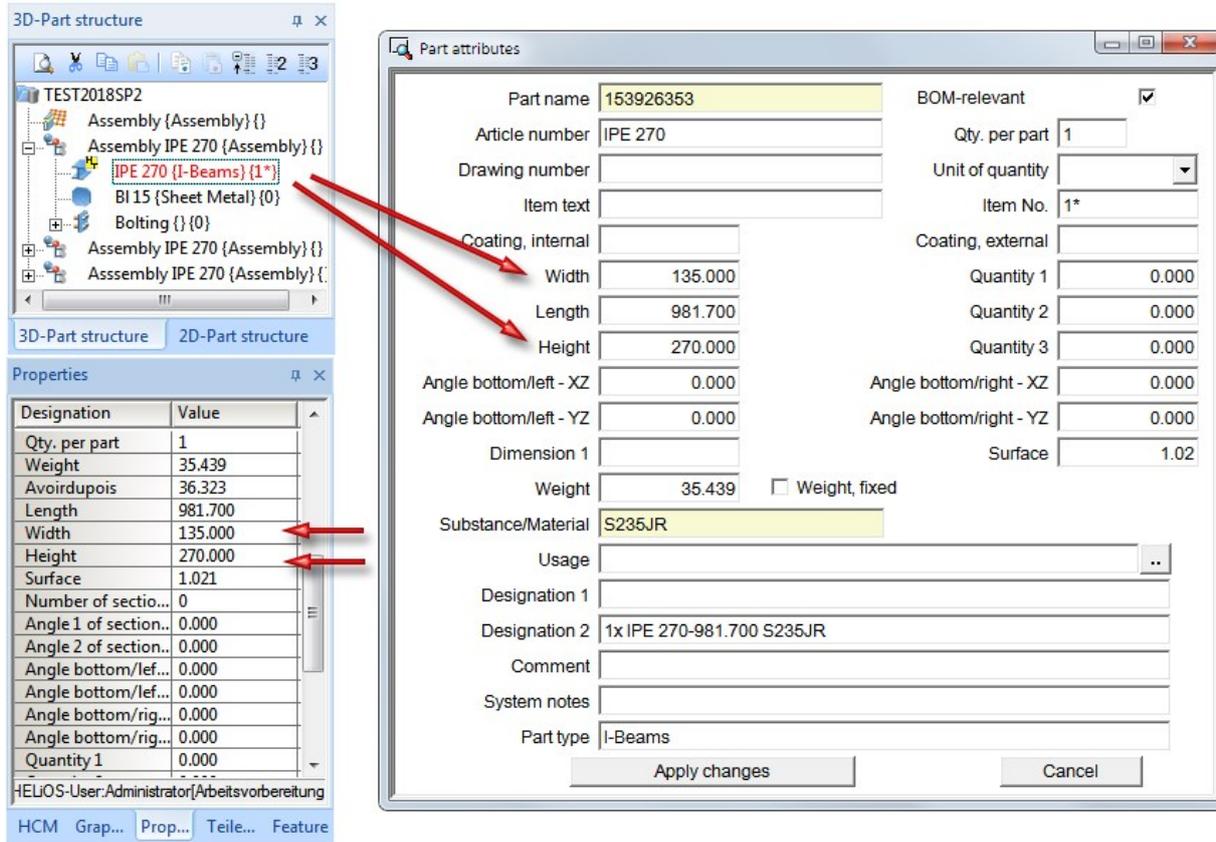
An example can be found here.

### Folded elongated plates - Calculation of attributes

As of HiCAD 2018 SP1, dimensions of elongated plates that have been placed along a 3-D composite edge are calculated with the help of an (internally) developed plate (blank). This also means that, for instance, the width shown in the part attributes mask, is not necessarily identical to the width shown in the model drawing. As of SP2, this also applies to folded elongated plates.

### Width / height of beams and profiles

The widths and heights of profiles can now also be managed in the part attribute masks.



### New / changed dimensioning rules

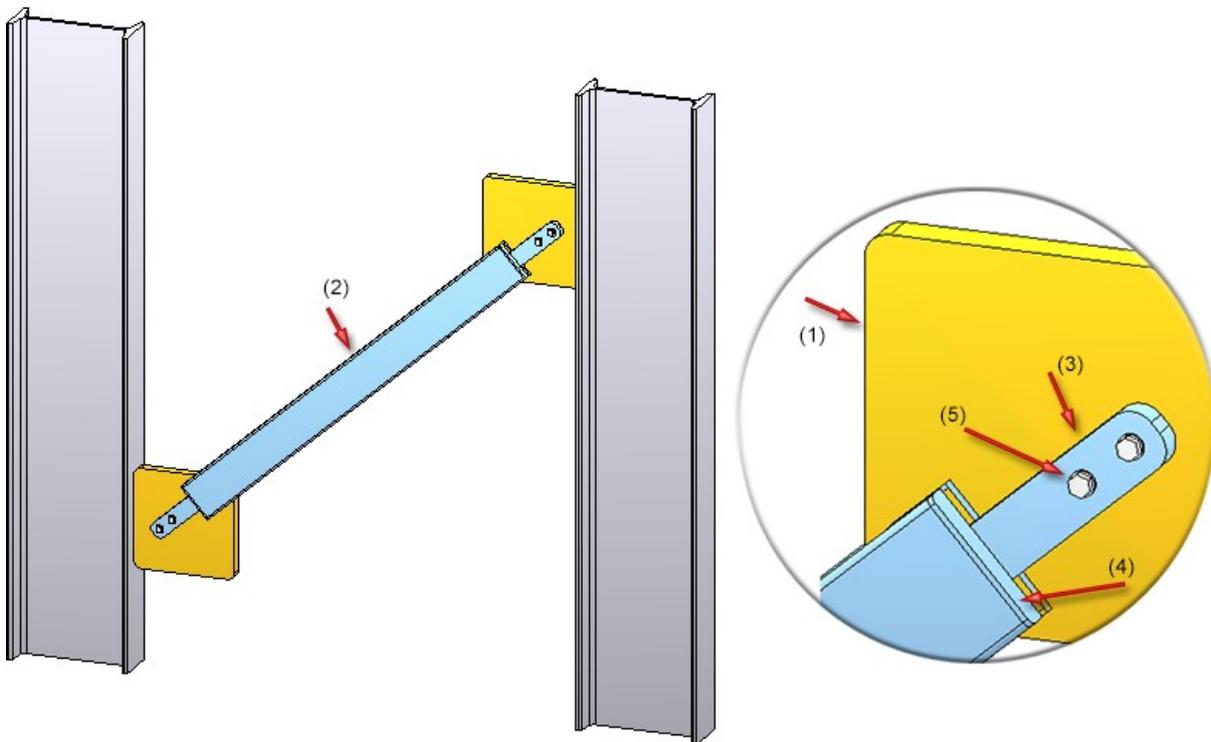
Please also read the information on the new and changed dimensioning rules for plates and railing segments in the News of the **HiCAD Basics** Help.

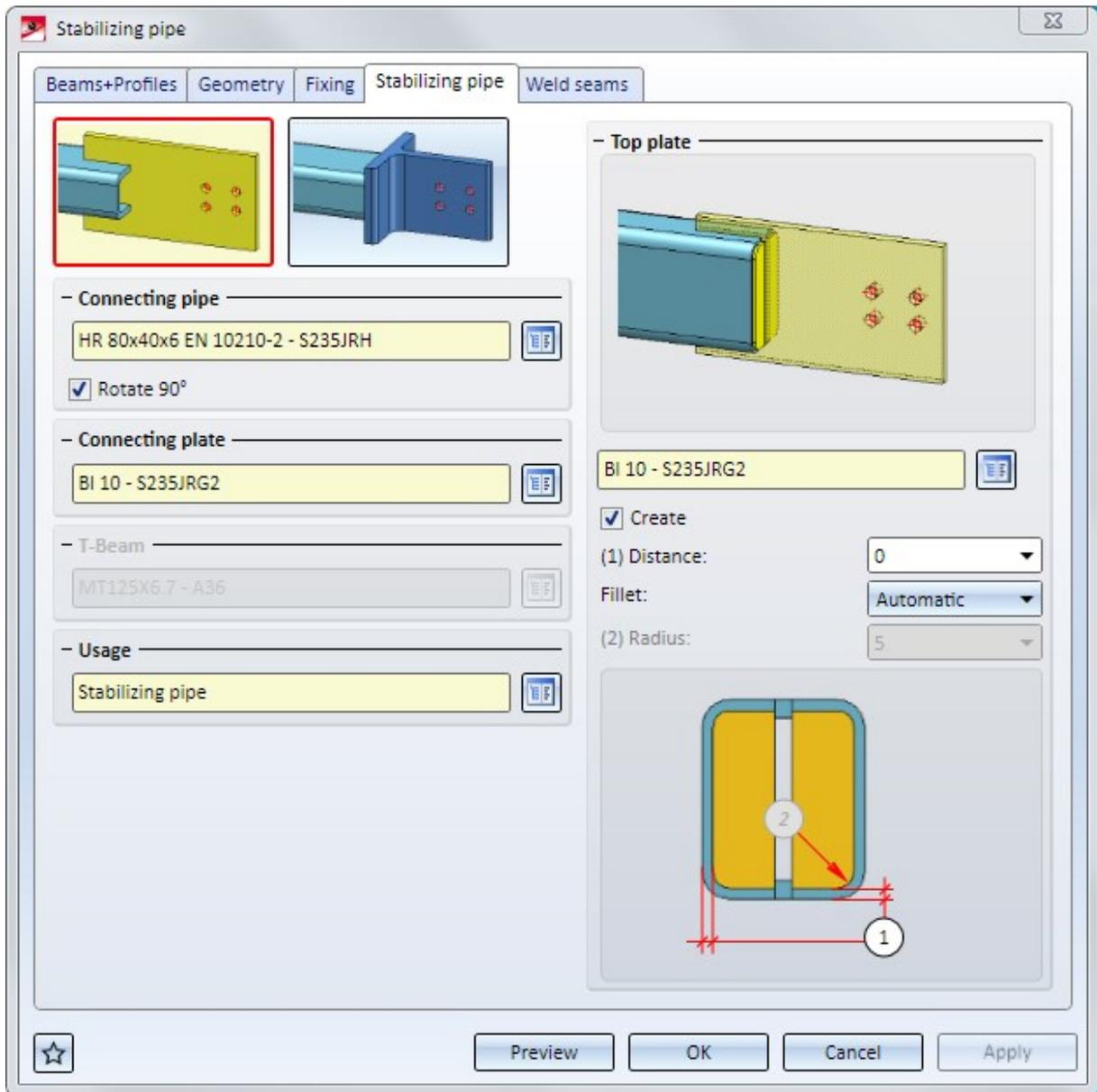
## Design Variant for Stabilizing pipe connection

The Design Variant **Stabilizing pipe connection** has been redesigned.

The created connection consists of

- one gusset plate on each beam (1),
- the connecting pipe with notch (2) and, if required, top plates (4).
- one connecting plate (3) on the stabilizing pipe or a T-profile or
- one (optional) bolting on the gusset plate or the connecting plate (5).





### Glass from sketch: Taking over of chamfer and offset parameters when exchanging or processing sketches

When using the functions **Exchange sketch** or **Process sketch** in the Feature log of a glass that has been created

with the **Glass from sketch**  function, the parameters for chamfers and offsets will now be taken over.

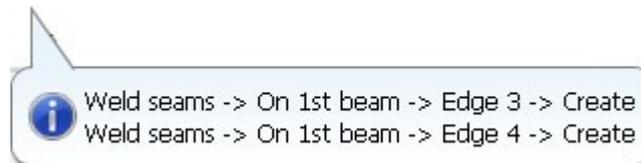
Excluded are offsets that were assigned to the individual line elements of the sketch with the **Individual** option. In this case, all offsets will be set to 0.

## DAST connections - Guidelines for weld seams

The DAST guidelines for weld seams have been implemented for the following Design Variants:

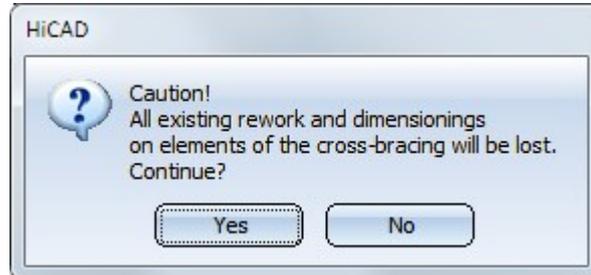
- Front plate connection to web/flange (2320),
- Front plate connection to flange (2330),
- Double-sided front plate connection to web/flange (2322) or
- Purlin joint, 2 plates with mitre cut acc. to DAST IH (2201)

If the weld seam settings are changed in such a way that they are no longer in accordance with the DAST guidelines, the DAST designation will be marked with a  symbol. When you move the cursor over the symbol, a message will be displayed, e.g.:



## Cross-bracing 2602 - Rework

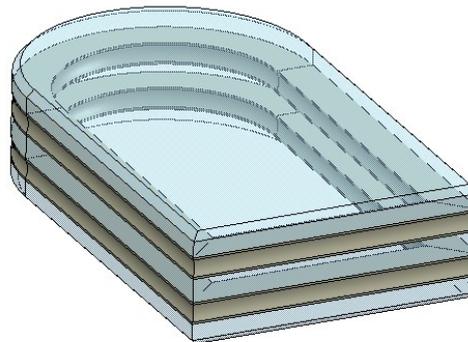
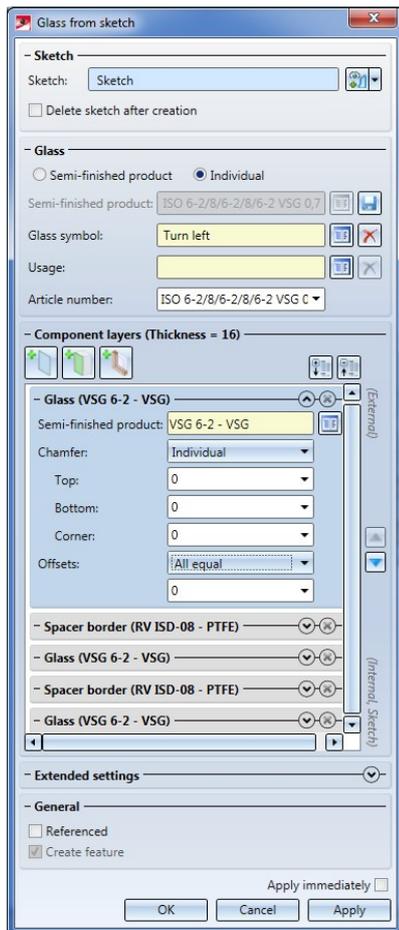
If you rework a cross-bracing subsequently (double-click on Feature) and then choose a different number of beams, reworks and dimensionings that may exist on elements of the cross-bracing may be lost. In such cases HiCAD will display the following warning message:



## Service Pack 1 2018 (V 2301)

### Glass from sketch

The **Glass from sketch**  function has been revised. You have now the option to build glass panes with as many layers as you like.

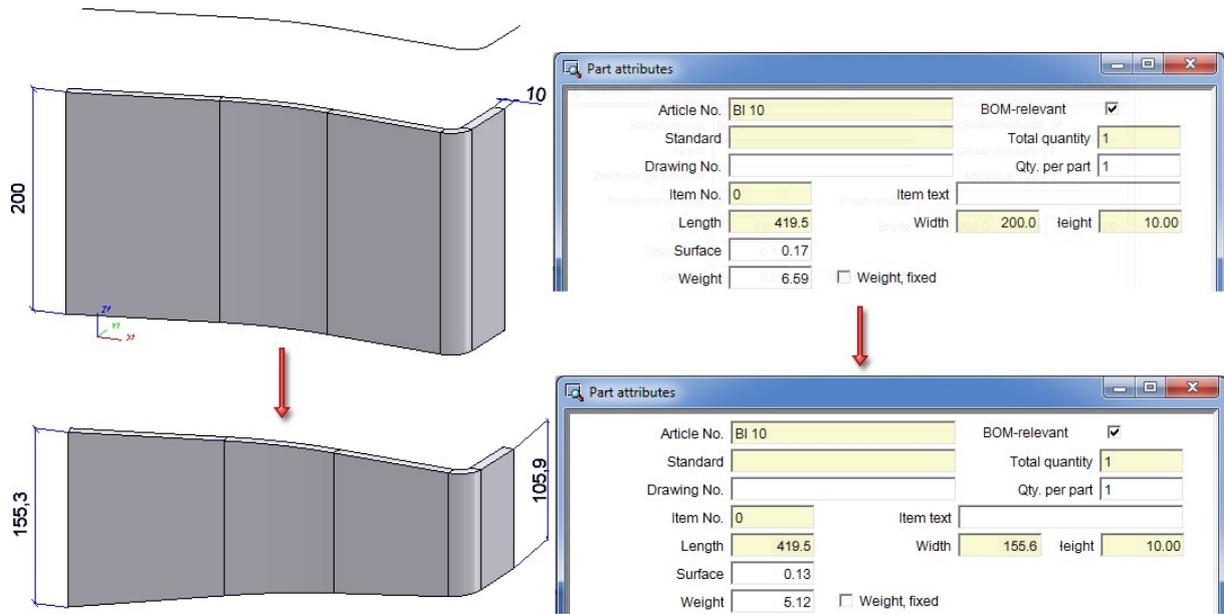


Glass panes can either be individually configured or used in the form of semi-finished products from the catalogue. An individual configuration can be directly taken from the dialogue window as a new semi-finished product stored in the catalogue.

The previous function **Glass from sketch** is currently still available as the "Glass from sketch (old)" function.

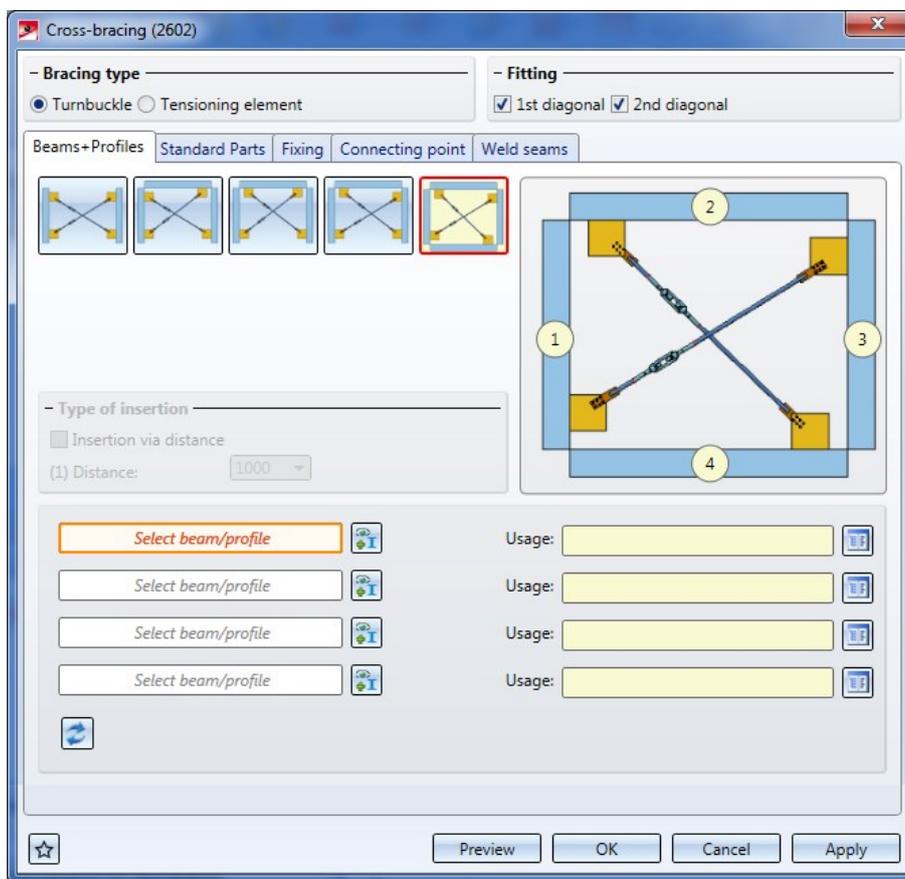
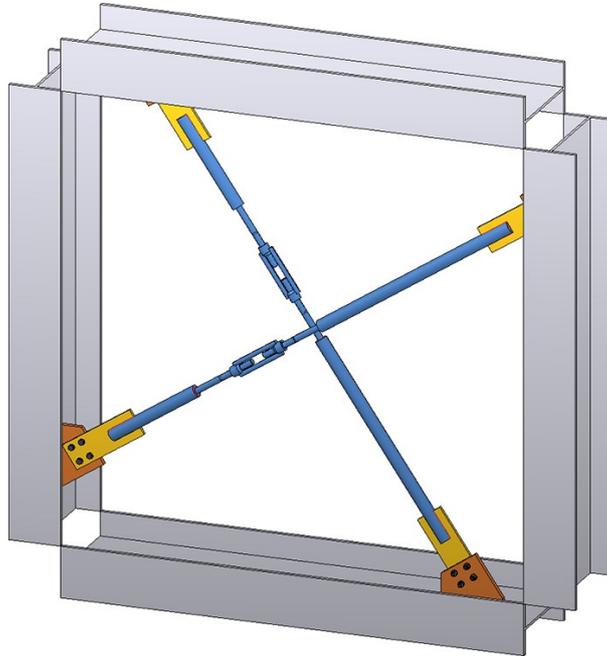
### Elongated plates - Attribute calculation

For elongated plates that have been inserted along a 3-D composite edge, the dimensions are now determined by means of the (internally) developed part (blank). This also means that, for example, the width shown in the Part attributes mask needs not necessarily be identical with the width of the part in the model drawing. The image below shows such a case: An elongated plate with a width of 200 was initially placed along a C-edge. A subtraction via translation was then applied to the part, which changed the width of the part.



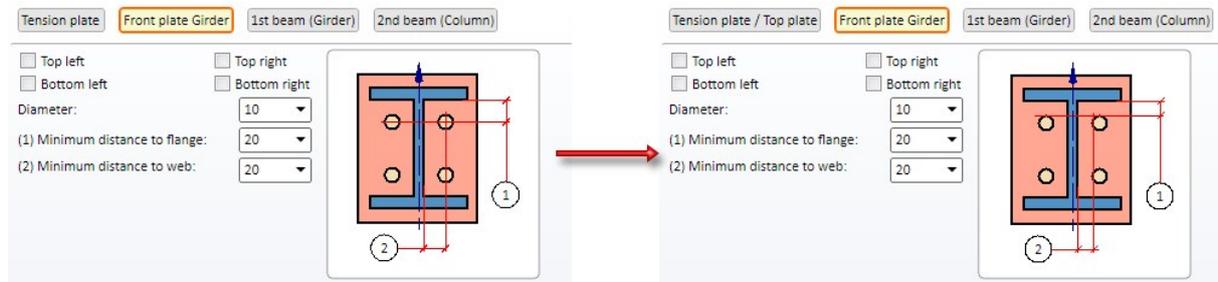
### New Design Variante - Cross-bracing (2602)

The new **Cross-bracing (2602)** design variant connects two, three or four beams by means of a cross-bracing. The cross-bracing can be created with a turnbuckle or a tensioning element, and with one or two diagonals.



## Changed dialogue windows for Design Variants

In those Design Variants that offer the option to insert galvanization holes, the minimum distance to the flange or web required in the dialogue is now always the distance between bore edge and flange (or web). In some of these variants this distance used to be interpreted as distance between the centre of the bore and the flange (or web).



Left: Before HiCAD 2018 SP1; Right: As of HiCAD 2018 SP1

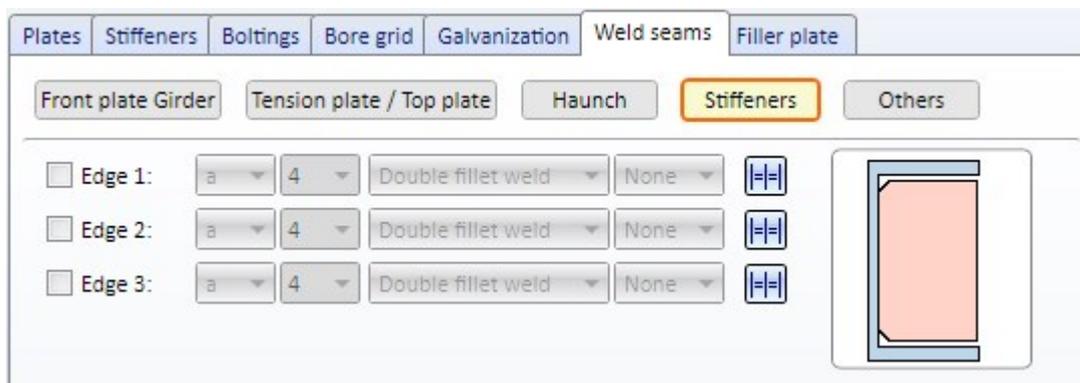
This change concerns the following variants:

- Base plate + anchor plate (2101),
- End plate (2102),
- Purlin joint, 2 plates with mitre cut acc. to DAST IH (2201) and
- Column connection, Frame corner (2203) and (2204).

### Please note:

When you load a model drawing containing one of the above Design Variants created with a HiCAD Version before 2018 SP1, nothing will change. But when you process such a Design Variant, the distances to centres of bores will be shown in the dialogue, and will also still be interpreted as such- even if the dialogue window shows a different distance!

The dialogue windows for the Column connection, Frame corner (2203) and (2204) offer an additional tab for the settings of weld seams on stiffeners:



## Parameter configuration for Steel / Metal Engineering

The following default settings of parameter configurations for Steel and Metal Engineering have been changed in the Configuration Editor at **Automatic drawing derivation > Production drawing**:

Parameter	Old	New
Usage-dependent > name* > View group > Insert BOM	In view group	In drawing frame
Usage-dependent > name* > View group > BOM: Position in view group	Bottom left	Bottom right
Usage-dependent > name* > View group > BOM: Position in drawing frame	Bottom left	Bottom right

\* *name* is the name of the usage, or part, respectively, e.g. ASSEMBLY\_BEAM or DEFAULT(L\_PROFILE).

Furthermore, the BENENNUNG (Designation) attribute for quantity lists has been removed in the BOM configuration file **Hicad\_Stahlbau.RMS**.

## Drawing derivation - Usage-dependent configurations

For **Railings** the following three new usage types are available in the **Factory standards** catalogue at **Usage > Civil Engineering > Steel Engineering > Railing**:

- Skirting board profile,
- Handrail profile,
- Post profile.

You can assign usage-dependent configurations for drawing derivations to these new railing elements.

The following configurations have been preset for the new usage types in the Configuration Editor at **Automatic drawing derivation > Production drawing > Usage assignment**:

Usage	CONFIGKEY	Utilized configuration	Consider for drawing derivations
Post profile	POSTPROFILE	DEFAULT (STAHLROHRE)	Yes
Handrail profile	RAILINGPROFILE	DEFAULT (HOHLPROFILE)	Yes
Skirting board profile	SKIRTINGPROFILE	DEFAULT (FLACHSTAHL)	Yes

Furthermore, the default setting for the vertical rods used for infills has been changed:

Usage	CONFIGKEY	Utilized configuration	Consider for drawing derivations
Rod	WEBMEMBER	DEFAULT	Yes

The default setting for **Steel Engineering** elements of the part type **Round steel** has also been changed:

Usage	Part type	Utilized configuration	Consider for drawing derivations
All	Round steel	DEFAULT (STAHLROHRE)	Yes

## Bar lists - Configuration of image texts

The long texts and short texts in the "images" of bar lists are now configurable. The ISD default setting is as follows:

- Long text = Item number and length in brackets, e.g. 102 (500)
- Short text = Item number

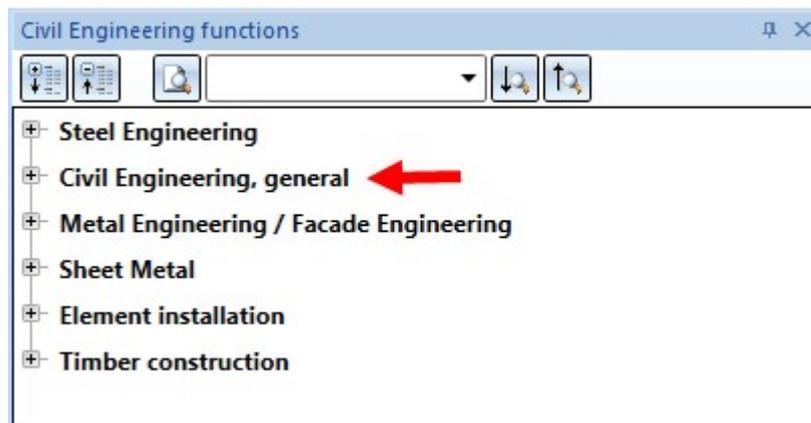
The short text will be used if there is not enough space for the long text in an image.

## Major Release 2018 (V 2300)

### Licensing

#### Enhanced Steel Engineering module

From Version 2018 onwards, the HiCAD **Steel Engineering** module will contain the connections that you can find in the **Civil Engineering functions** docking window at **Civil Engineering, general**.



#### Steel/Metal Engineering module no longer available

- As of HiCAD 2018 the **Steel/Metal Engineering** module is no longer available. This module can be replaced with the two modules **HiCAD Steel Engineering** and **HiCAD Metal Engineering**.

## Design variants

### Column Connection, Frame Corner (2204)

With this variant two I-beams are connected with a bolted or welded beam frame corner. You can create the frame corner with or without stiffeners, haunched plates or other components, e.g. haunched flanges, web reinforcements and reinforcement plates or filler plates. It is also possible to insert stiffeners at the tension plate and ribs at the front plate, and also weld seams as well as to apply galvanization holes.

In contrast to Frame Corner (2203), the connection is made with a square haunch and the tension plate can also be conical here.

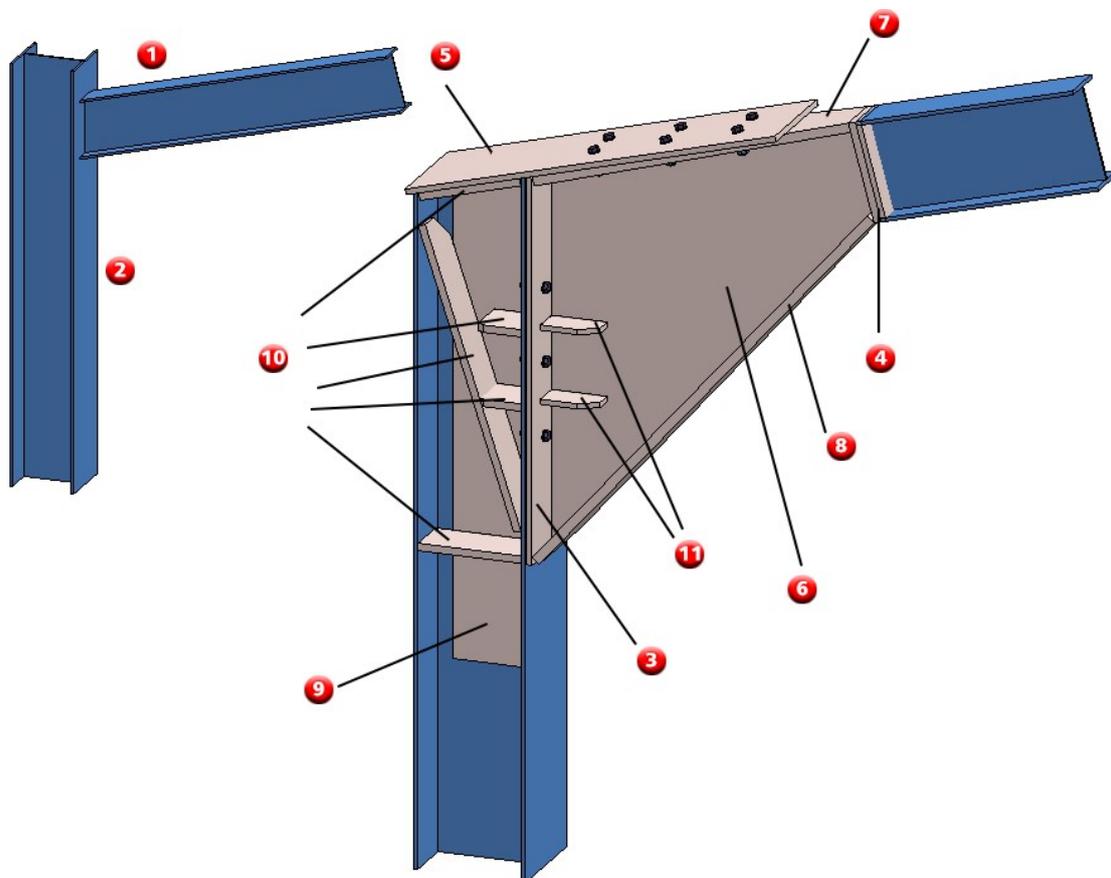
The connection to the girder can be either made with or without front plates.

- with two front plates (bolted),
- with one front plate (welded)
- without front plate (welded)

Requirements for the installation of a frame corner are:

- Both beams must be the type I-beam.
- The webs of the beams must lie in one plane.
- The beam axes must not be parallel and must not lie in one straight line.

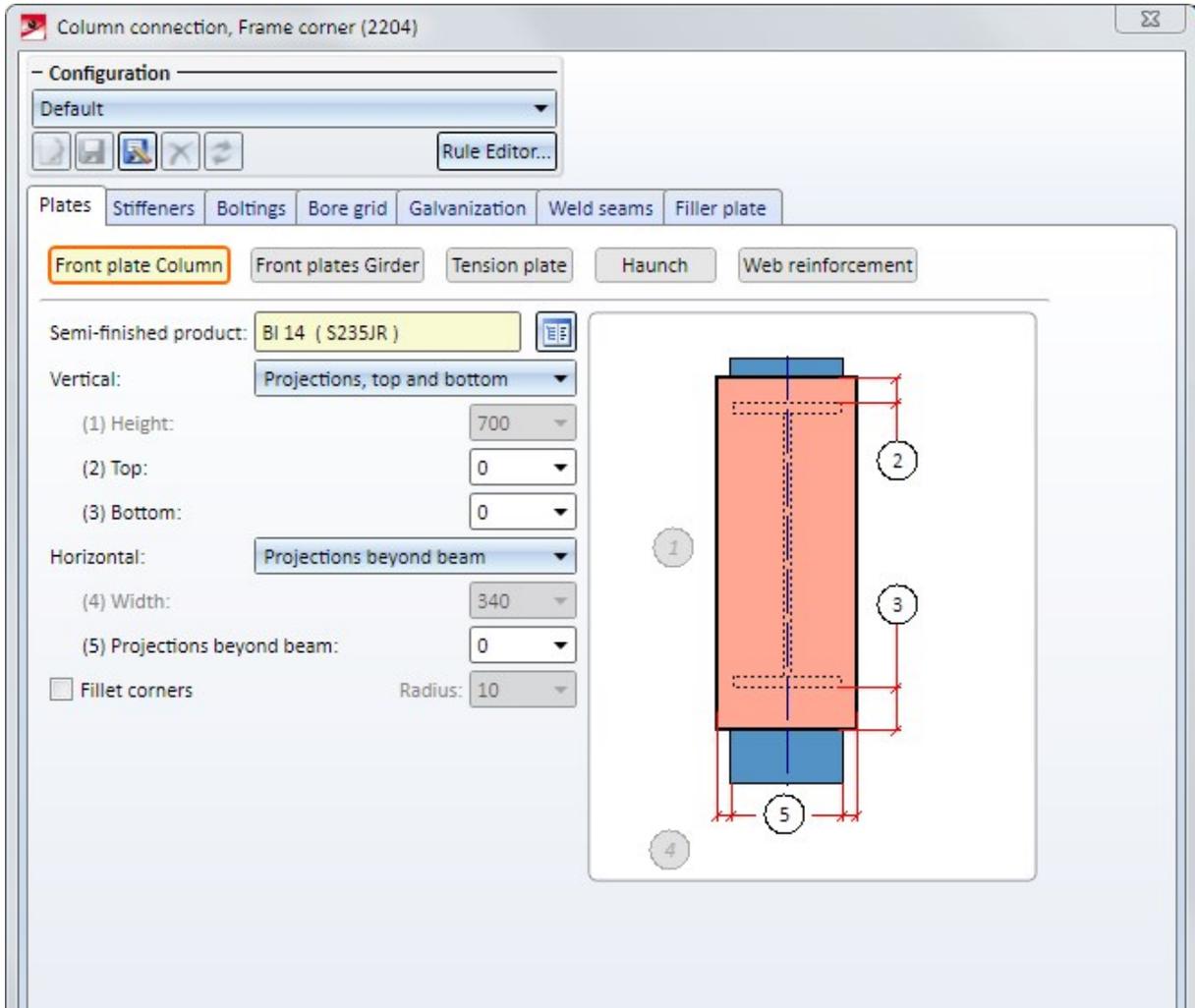
#### Example of a frame corner with an added tension plate



(1) girder, (2) column, (3) front plate at column, (4) two front plates at girder, (5) tension palte, (6) haunch, (7) haunched flange top, (8) haunched flange bottom, (9) web reinforcement, (10) stiffeners, (11) ribs

1. Identify the beam (girder) in the vicinity of the reference end.
2. Identify the beam to which the first one should be connected (column).

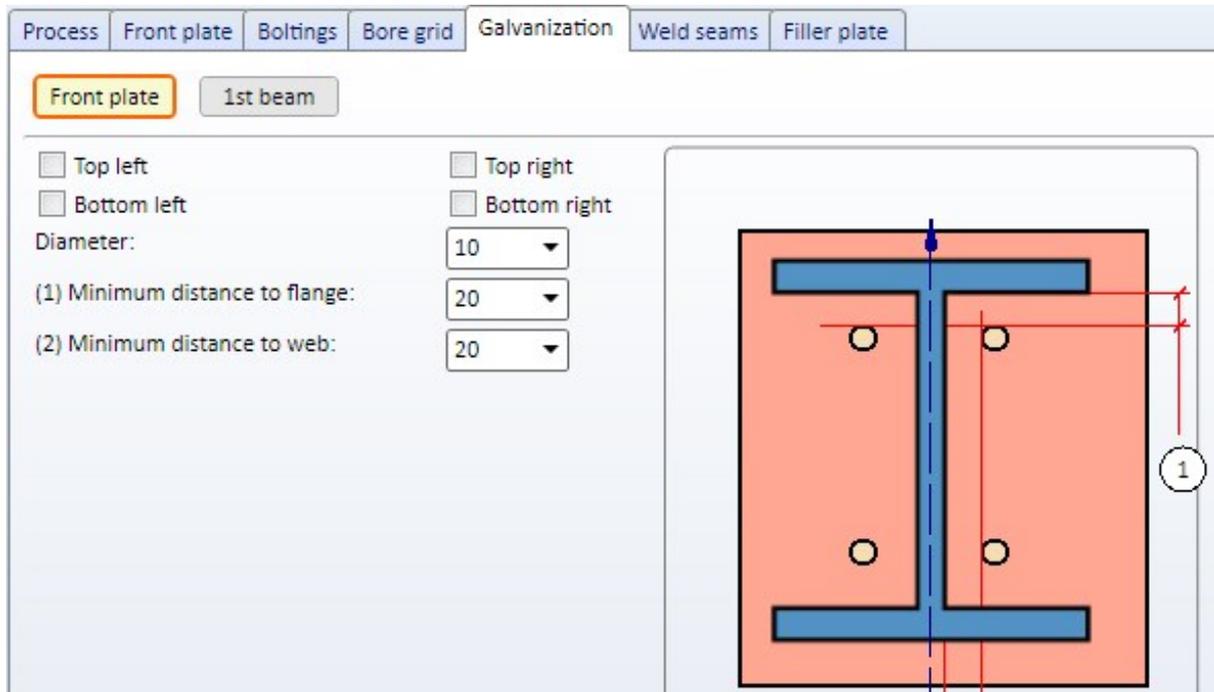
The **Column connection, frame corner (2204)** dialogue window will be displayed.



### Advancement of design variants 1211 and 2320 - galvanisation hole in beam

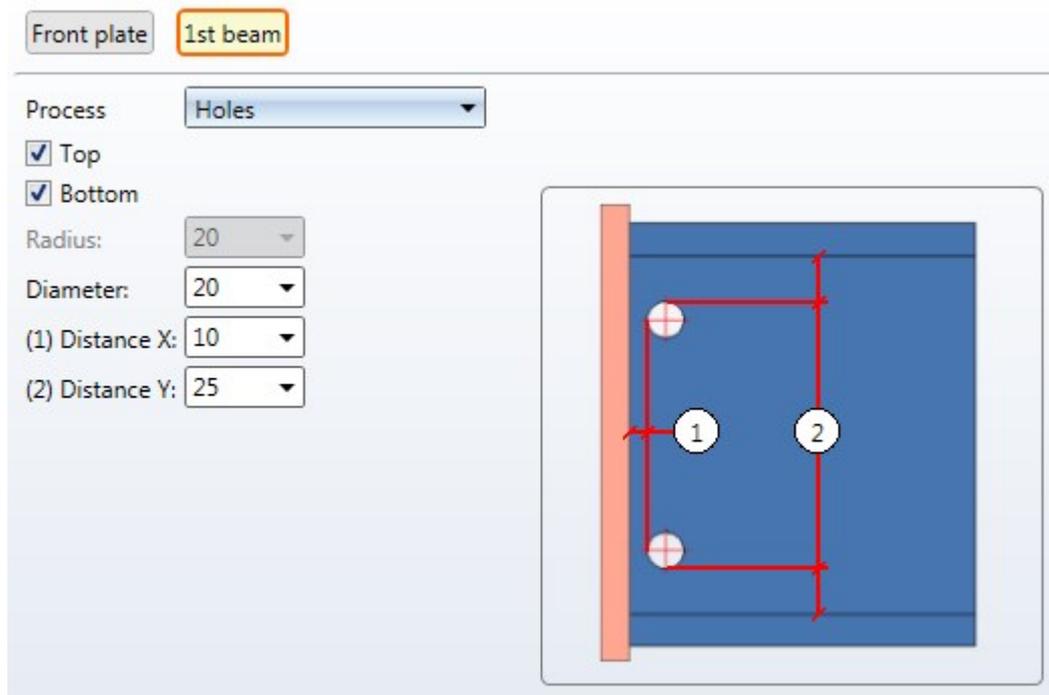
The Design Variants Front plate connection to Web/Flange Side (2320) and Beam to Web with 2 Plates and Stiffener (1211) allow to apply galvanisation holes in beams. For this, the tab **Galvanisation** has been enhanced accordingly within the dialogue boxes.

Tab Galvanisation

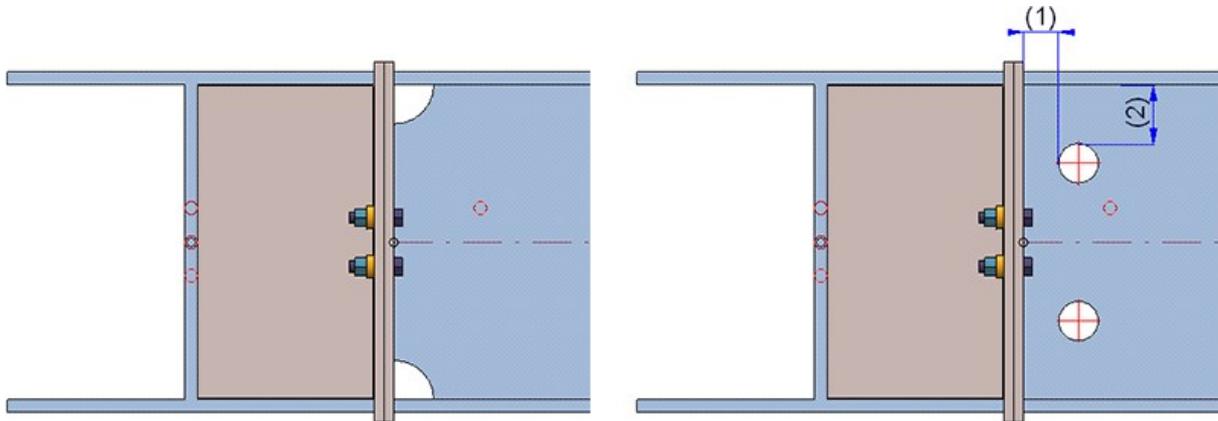


### Settings for the 1st beam

Here, you can choose via the checkbox **Edit** if either holes or web cuts should be integrated.



For integrating the web cuts, you determine the radius and the fitting position (top and/or bottom). For installing the holes, you determine the installation position (top and/or bottom), the bore as well as the distance to the plate (1) and beam edge (2).

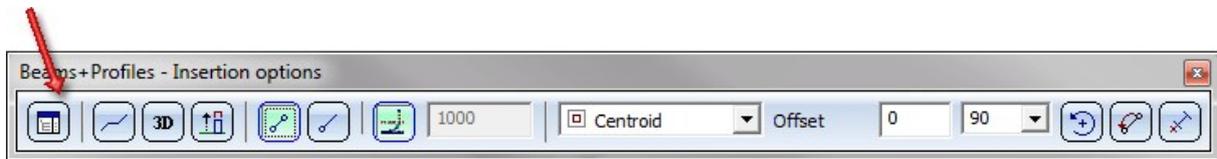


On the left: web cuts, on the right: holes

## Beams

### Insert series beams from catalogue

As far as the insertion of series beams is concerned, it has not been possible so far to insert an additional divergent beam via the function dialogue option **Beam**.



This problem is fixed in HiCAD 2018 and upwards, i.e., you can use this option to choose another standard part type before defining the next fitting point.

### Representation of beams

So far, it has not been possible to manage the display type of beams (exactly or simplified) via parameter configuration options for steel engineering. From HiCAD 2018 and higher this is not the case anymore.

The default settings have to be defined via the configuration editor under **Steel engineering > Representation**. The default settings are **Simplified**.

## Dimensioning and annotation

### Dimensioning and annotation for hidden bores and processings

Dimensioning and annotations for beams, plates and sheets in workshop drawings can (if desired) be placed only on the visible outer side, i.e. bores/processings that are displayed with dashed lines in the **HiddenLine dashed** representation will not be dimensioned in the corresponding view.

To suppress the dimensioning for hidden bores/processings, the following requirements must be met:

1. In the **STWDimSettings.xml** file in the HiCAD sys directory the switch **IGNOREHIDDENSUBPARTANDBORES** must be set to **1**.

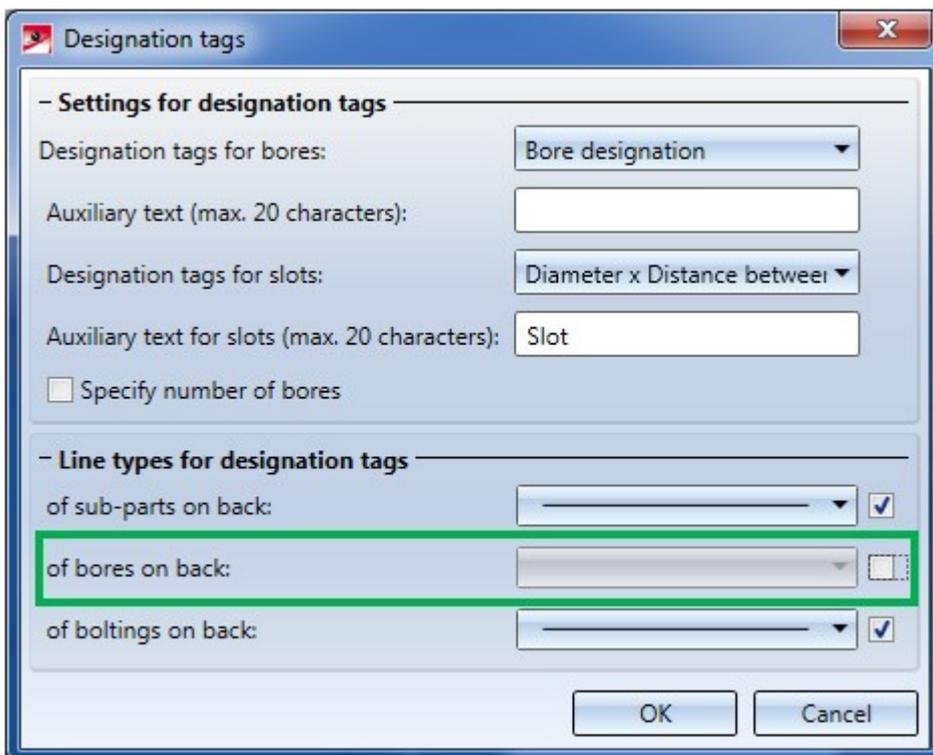
```
</PARAM><PARAM Name="IGNOREHIDDENSUBPARTANDBORES" Typ="INT" Value="1">
```

2. The HiddenLine or HiddenLine dashed representation must have been chosen.
3. The bore/processing must be entirely hidden. If it is partly visible it will be dimensioned!

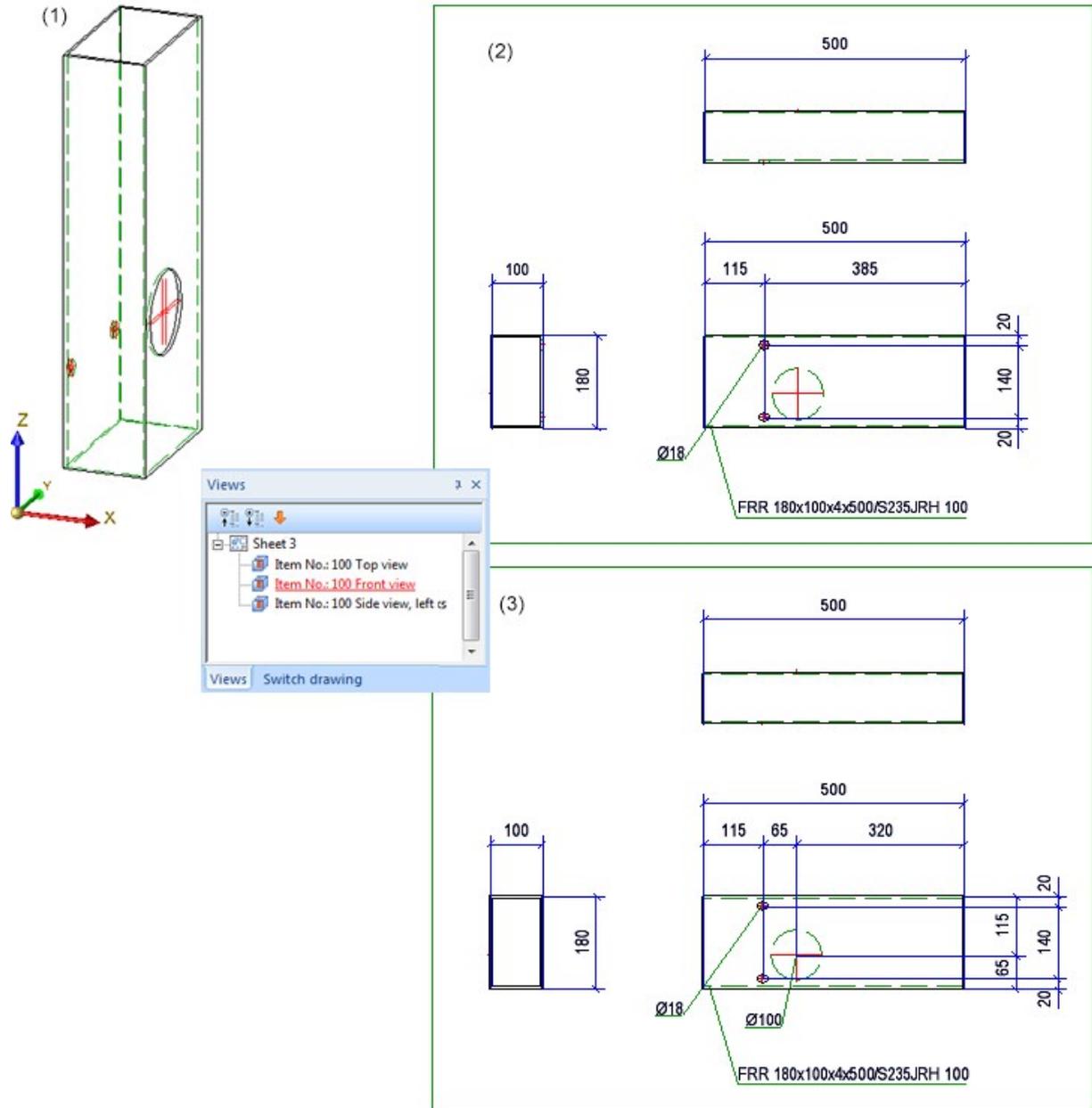
The settings for the dimensioning only apply to dimensionings via dimensioning rules.

To suppress the annotations for hidden bores/processings, the following requirements must be met:

1. In the Dimensioning Settings the line type for annotation tags of bores located on the back must have been switched off.



2. The HiddenLine or HiddenLine dashed representation must have been chosen.
3. The bore/processing must be entirely hidden. If it is partly visible it will be annotated!



(1) Original beam; (2) Workshop drawing with suppressed dimensioning + annotation of hidden bores; (3) Without dimensioning + annotation of hidden bores

## Others

### Weight calculation for bulb plates

When calculating the weights of bulb plates, the values in the columns GEW (Weight) and HGEW (Avoirdupois weight) of the catalogue will now be evaluated.

### Avoid multiple sectional views in workshop drawings

In the Configuration Editor at **Automatic drawing derivation > Production drawing > Avoid multiple sectional views** you determine that sectional views with the same content will only be created once in the workshop drawing. Sectional views are considered equal if

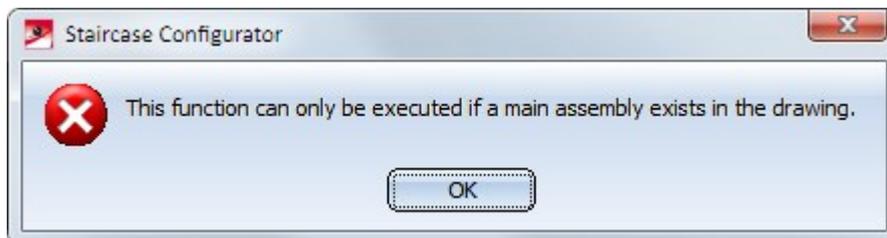
- the parts visible (displayed) in the sectional views have the same valid item number and
- the sectional views contain the same geometries.

Sectional views for plates outside of the plate end (front plates, top plates, base plates mean the same) are generally unequal.

If you have chosen the **Steel/Metal Engineering** template in the **HiCAD Parameter configuration** dialogue window (double-click ParKonfigComp.exe in the exe directory of your HiCAD installation to open this window), the **Avoid multiple sectional views** checkbox will be active by default.

### Staircase/Railing Configurator - Message displayed if main assembly is missing

A prerequisite for the start of the Staircase or Railing Configurator is that the current drawing contains a main assembly. If this is not the case, the following message will now be displayed.



Click **OK** and create a new main assembly and call the function again.

No automatic creation of a main assembly will take place anymore, as was still the case in older versions up to HiCAD 2017.

# Steel Engineering - Management + BIM

## Service Pack 2 2018 (V 2302)

### New attributes for the article master

When deriving drawings with the Management + BIM module, the document attributes

- HEL\_DOKUNUMMER ,
- DRAWINGNUMBER and
- DRAWINGNUMBER\_TEXT

will be generated automatically.

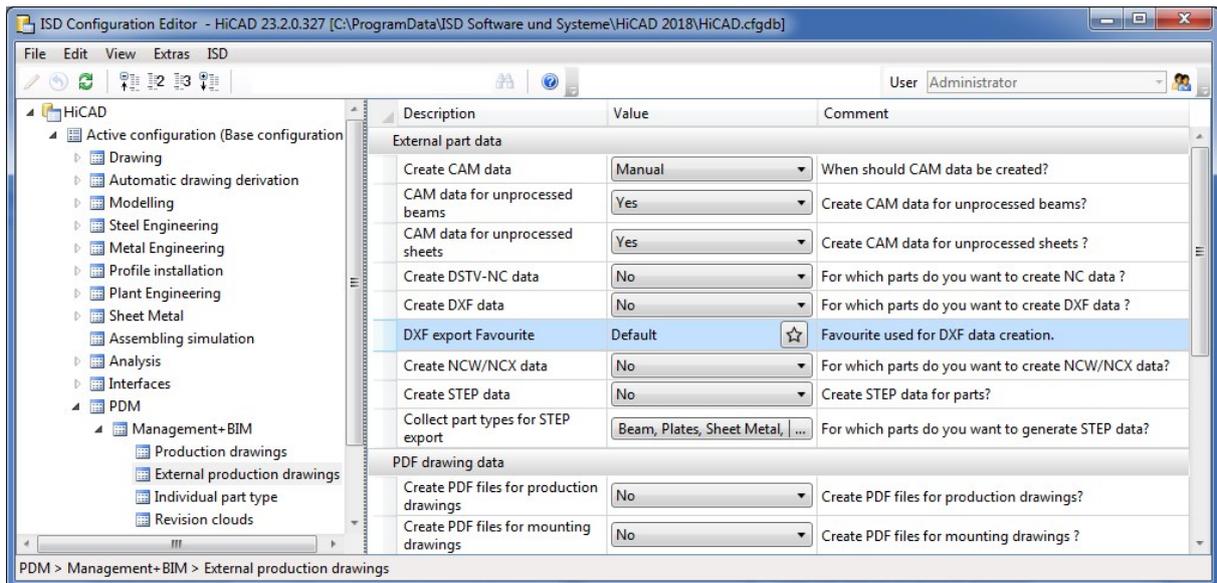
As of HiCAD 2018 SP2 these attributes will be transferred automatically to the corresponding article master attributes of the part when the production/detail drawing is generated. For this purpose the following new article master attributes have been introduced:

- COMPONENT\_WSDDOCUNUMBER,
- COMPONENT\_DRAWINGNUMBER and
- COMPONENT\_DRAWINGNUMBERTEXT

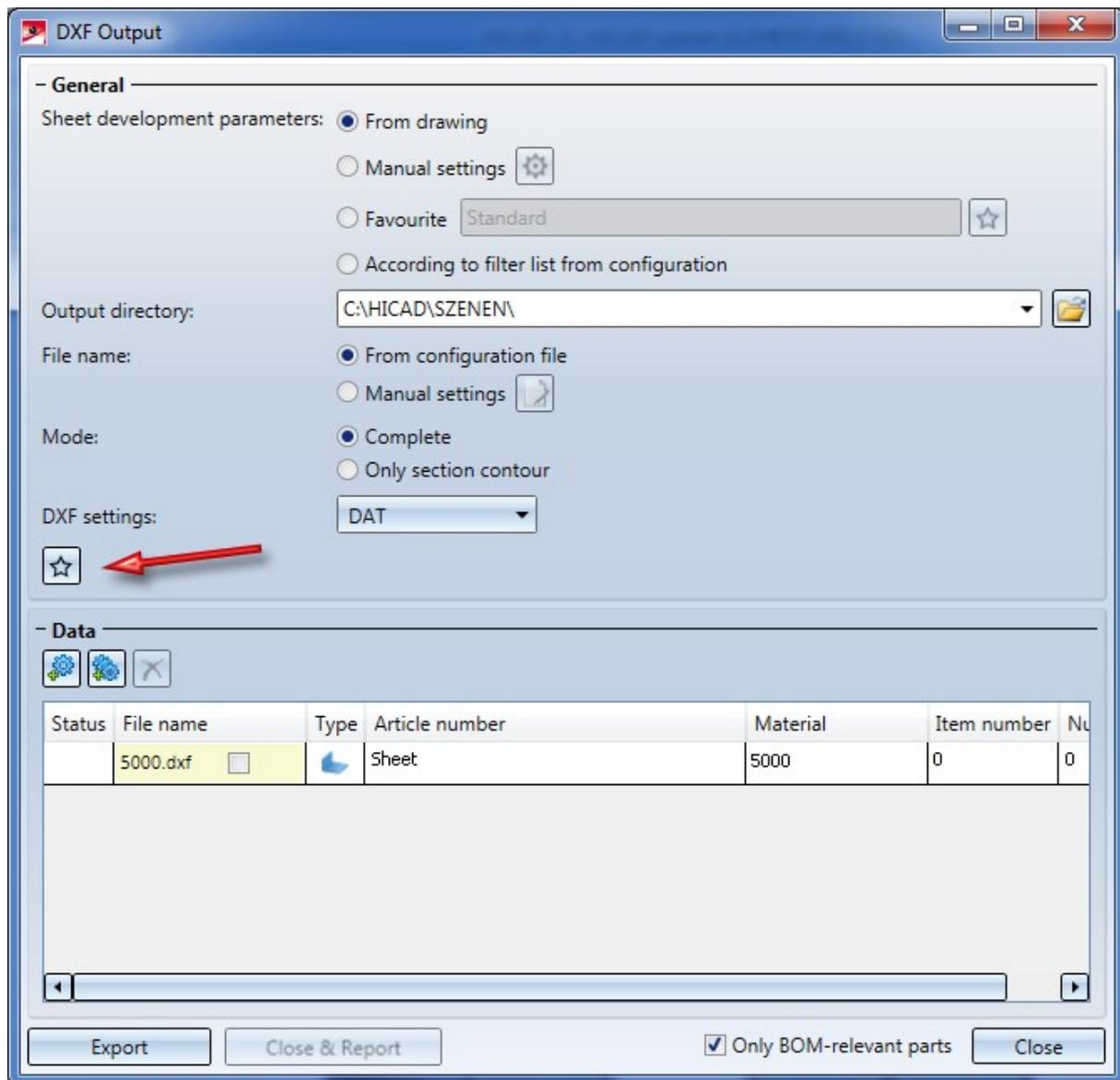
A prerequisite for the automatic transfer is that the setting **Create only one production drawing per part** has been chosen in the Configuration Editor.

### Favourites for DXF export

For the export of DXF data for Sheet Metal parts you can now choose in the Configuration Editor which Favourites file is to be used for the export.



By default, the ISD defaults have been set here. Choose **Sheet Metal > Sheet development > Extras** > Sheets to define further Favourites that will then also be available in the Configuration Editor.



## Management + BIM Help

The Management + BIM Help has been revised and restructured.



### Important:

- To ensure a smooth and trouble-free management of your model drawings with the **Management+ BIM** module, the working environment needs to be set up properly, and certain rules need to be observed. It is therefore very important that you read the relevant information given in the **Requirements for a Smooth Operation** topic!
- Plan the construction process for your project carefully. Decide whether you want to create your model drawing in one single model drawing, or - due to the size of the model or in order to save time - on several workstations. It is therefore very important that you read the relevant information given in the **Pre-planning of the Construction Process** topic!

## Semi-finished Product Article as Sub-item

Semi-finished products can be automatically transferred as sub-items of a part to HELIOS. This can be set in the Configuration Editor at **Active configuration > Base configuration > PDM > HiCAD-HELiOS interface > Product structure**. Here, activate the **Semi-finished product article as sub-item** checkbox.

**Halbzeug-Artikel als Unterposition.**

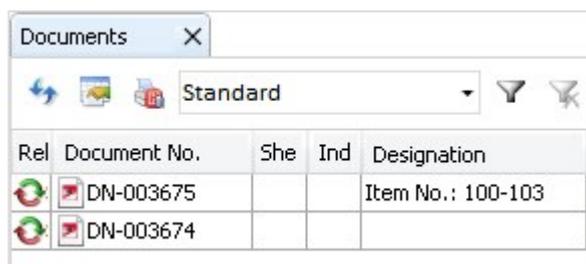
## Major Release 2018 (V 2300)

### Templates for attribute assignment - More flexible configuration of document attribute BENENNUNG (Designation)

If you are using Document and Drawing Management for Civil Engineering (Management + BIM), you have the option to create the HELIOS attributes BENENNUNG (Designation) and SACHNUMMER (Article number) of the part drawings from templates files. For this to happen, open the Configuration Editor, choose **PDM > Management + BIM > Production drawings** and set the **Create HELIOS attributes from FTD file** to **Yes** (Default). These templates can be adjusted individually with the **Templates, Attribute assignment** function.

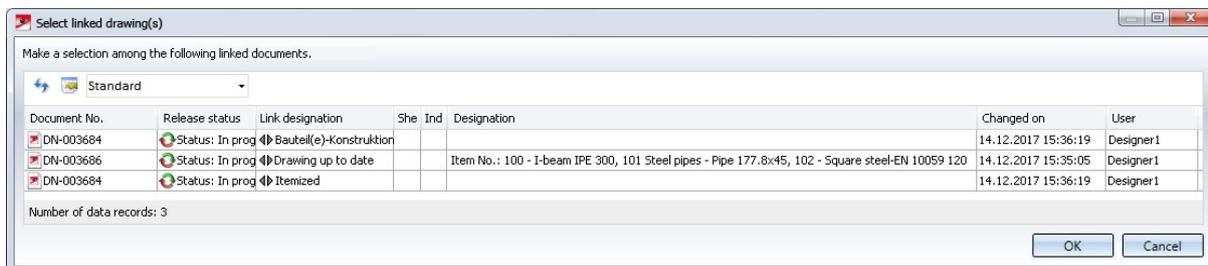
The actual configuration takes place via Drawing > Itemisation/Detailing > Attr.  > Templates, Attribute assignment  .

For the configuration of the HELIOS document attribute BENENNUNG (Designation) for a multiple selection of beams to be output on one drawing sheet, the file BIM\_PDM\_WSD\_Multi\_Designation.ftd was previously available. The default setting in this file has the following effect: If multiple parts/beams exist in the drawing, a list of the item numbers with reasonably combined items will be output, e.g.:



Rel	Document No.	She	Ind	Designation
	DN-003675			Item No.: 100-103
	DN-003674			

To make this configuration more flexible, the additional FTD file **BIM\_PDM\_WSD\_Multi\_Itemnumber.ftd** is now available. If several parts are to be output on one drawing sheet, this file for the configuration of the document attribute BENENNUNG (Designation) allows you to use a different attribute instead of the item number, e.g. the item text.

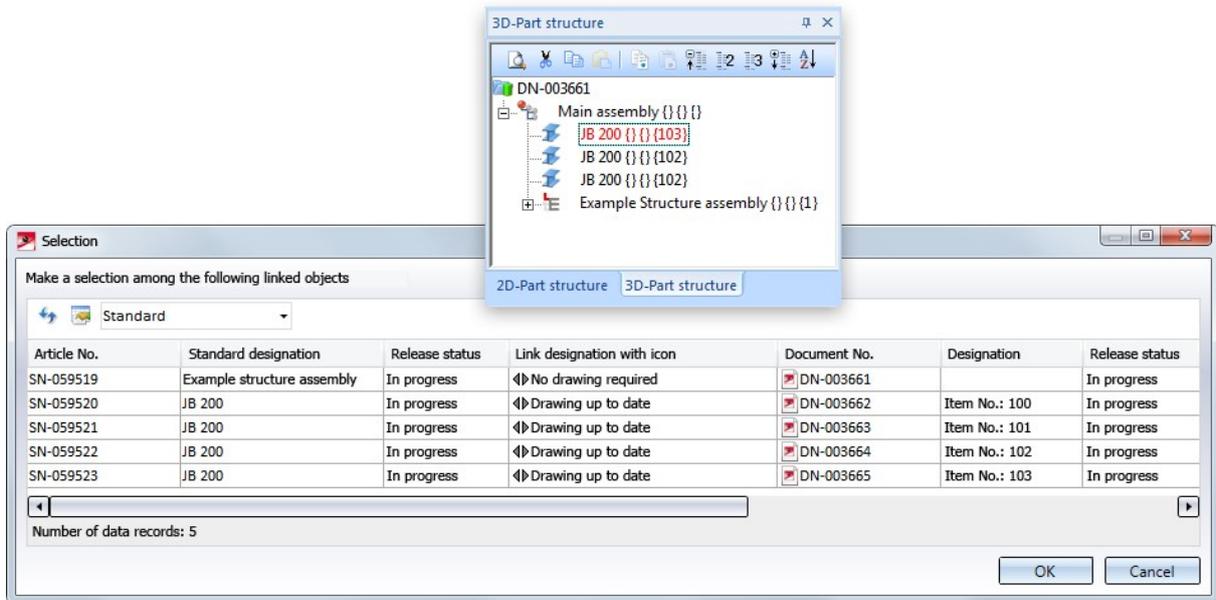


Document No.	Release status	Link designation	She	Ind	Designation	Changed on	User
DN-003684	 Status: In prog	Bauteil(e)-Konstruktion				14.12.2017 15:36:19	Designer1
DN-003686	 Status: In prog	Drawing up to date			Item No.: 100 - I-beam IPE 300, 101 Steel pipes - Pipe 177.8x45, 102 - Square steel-EN 10059 120	14.12.2017 15:35:05	Designer1
DN-003684	 Status: In prog	Itemized				14.12.2017 15:36:19	Designer1

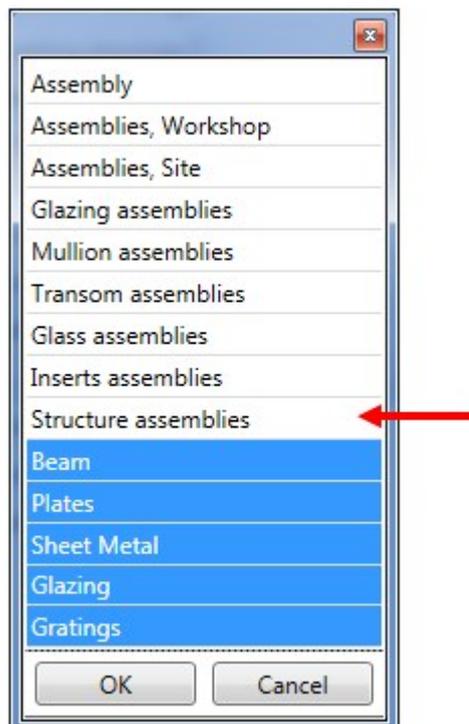
Number of data records: 3

## Management of structure assemblies

Structure assemblies serve the purpose of structuring HiCAD model drawings into construction sections, such as framework, platform, column, platform girders etc. The advantage of a structure assembly is that it behaves like a non-BOM-relevant assembly, even if it is actually BOM-relevant. These assemblies will not be considered for the deriving of production drawings. In previous version the Workflow status of structure assemblies needed to be changed manually via HELIOS. As of HiCAD 2018 structure assemblies will have the link **No drawing required**. Structure assemblies with this link will also be considered for Checkup and Release.



Furthermore, it is possible to automatically create the STEP data of structure assemblies, manage them in HELIOS and link them accordingly. For this purpose the settings in the Configuration Editor at **PDM > Management + BIM > External production drawings > Collect part types for STEP export** have been appropriately expanded.



## Save without HELiOS BIM update

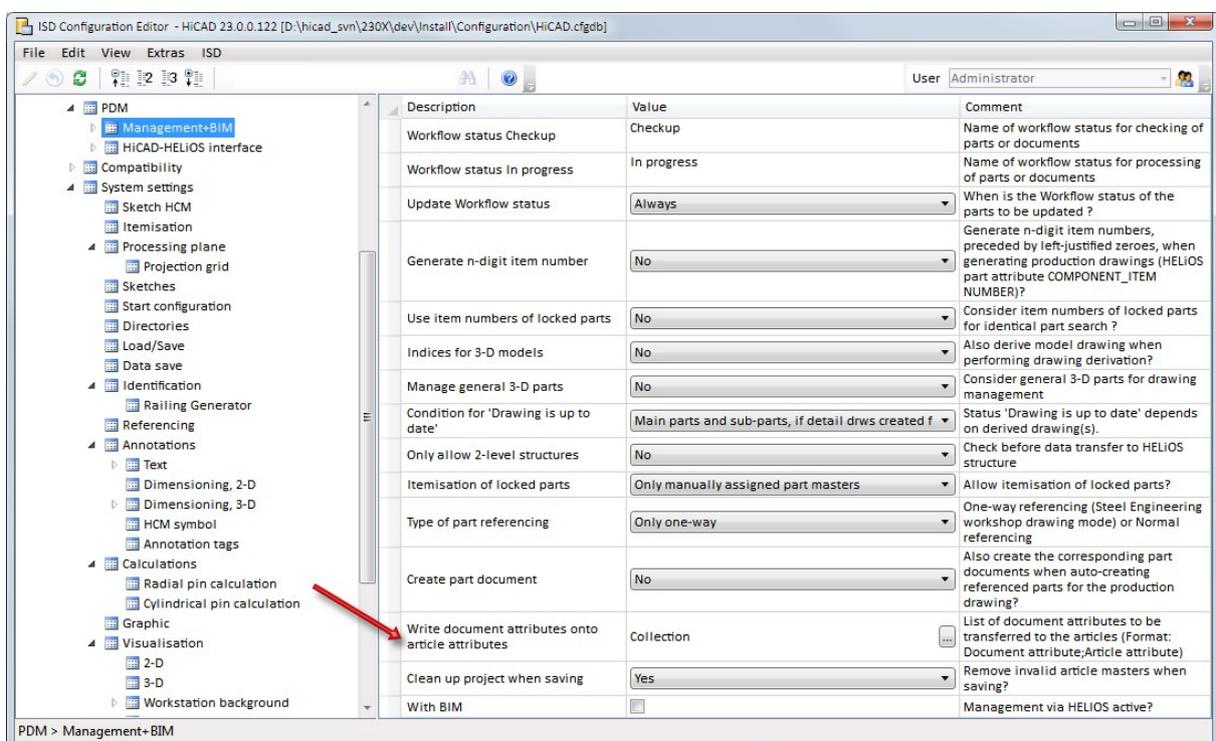


In the **Save** function group you will find the new  Save without HELiOS BIM update function. Use this function to save the current model drawing to HELiOS, but without updating the BIM links.

For this function an itemization of the parts is initially not required. The itemisation must only be carried out when you want to create or update drawings. After this, use the **Save drawing** function to update the links.

## Write document attributes on article attributes

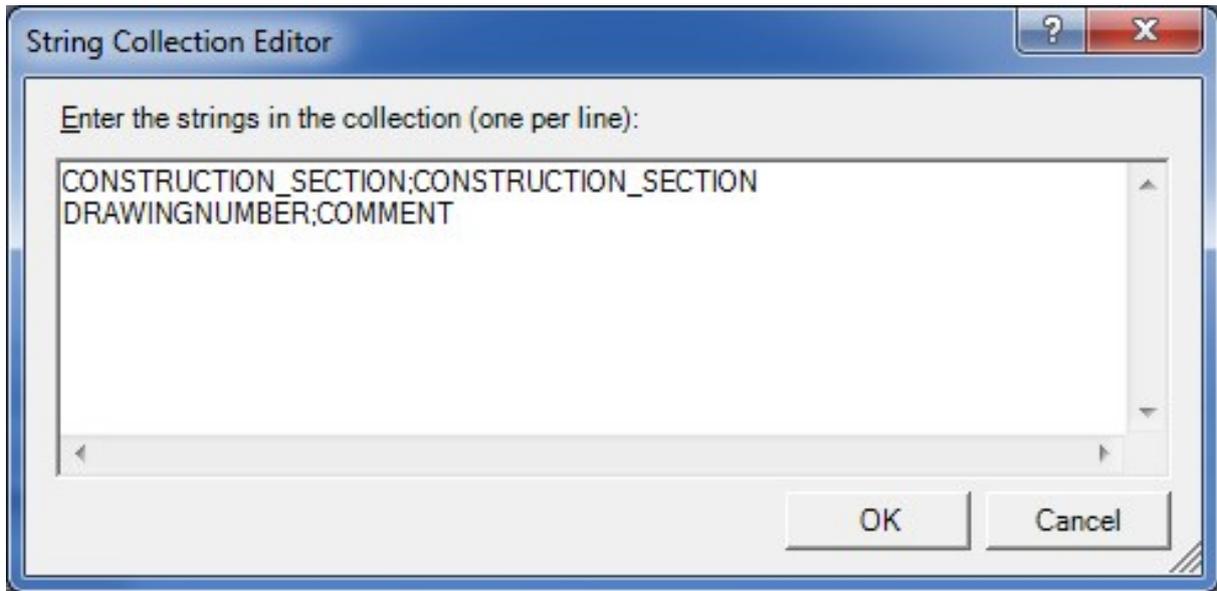
If desired, the document attributes of models that are managed with the Management + BIM module can be assigned to the article attributes of the parts. This can be done via the Configuration Editor at **PDM > Management + BIM > Write document attributes on article attributes**.



The assigning takes place via the String Collection Editor. Click on the  symbol in the Collection field and enter the desired assignments in the following order:

**Document attribute;Article attribute**

Each assignment must be defined in one line, e.g.:



Preset here is the assignment for construction sections (aka construction units)

CONSTRUCTION\_SECTION;CONSTRUCTION\_SECTION

# Metal Engineering

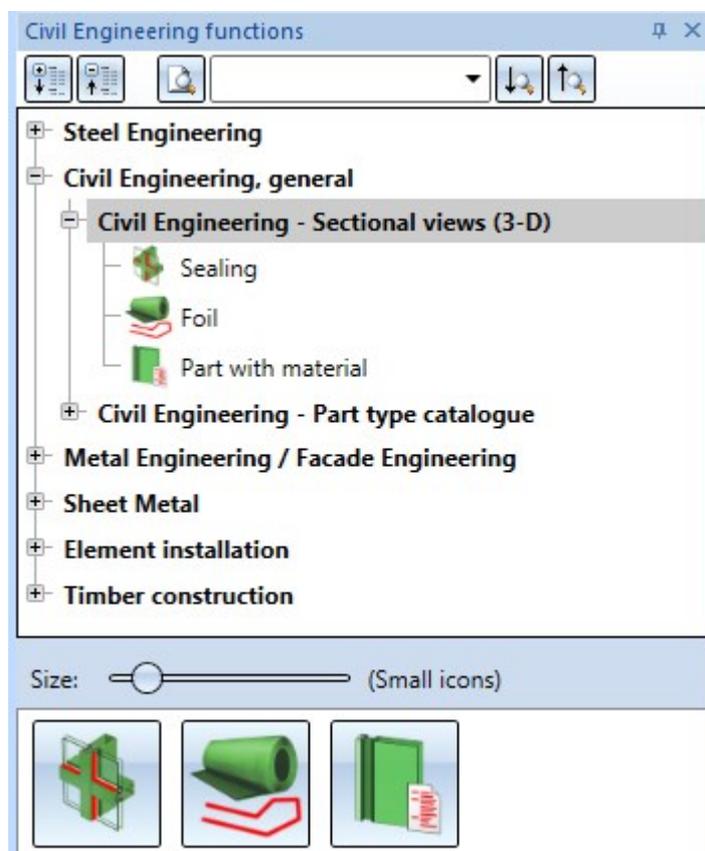
## Service Pack 1 2018 (V 2301)

### Functions for sectional views moved

As of HiCAD 2018 SP1 the functions **Sealing**, **Foil** and **Part with material** will be available in the following suites and modules:

- HiCAD Beams + Profiles,
- HiCAD Metal Engineering,
- HiCAD Steel Engineering,
- HiCAD Steel Engineering - Stairs + Railings,
- HiCAD Steel Engineering suites,
- HiCAD Metal Engineering suites, and
- HiCAD ALUCOBOND® suites.

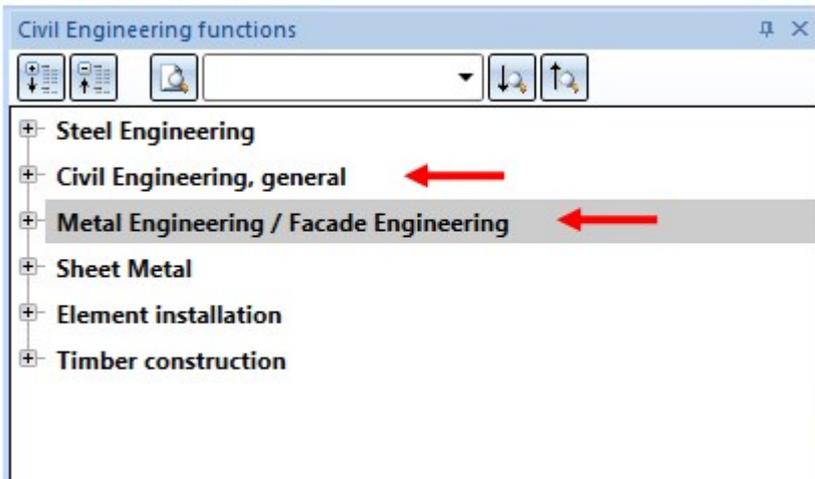
Therefore, these function can now no longer be found in the 3-D section function group of the Metal Engineering Ribbon tab, but in the **Civil Engineering functions** docking window at **Civil Engineering, general > Civil Engineering - Sectional views (3-D)**.



## Major Release 2018 (V 2300)

### Licensing

- As of HiCAD 2018, the joints and connections of the **Metal Engineering** module can be found in the **Civil Engineering functions** docking window at Metal Engineering / Facade Engineering and Civil Engineering, general.



- As of HiCAD 2018 the **Steel/Metal Engineering** module is no longer available. This module can be replaced with the two modules **HiCAD Steel Engineering** and **HiCAD Metal Engineering**.

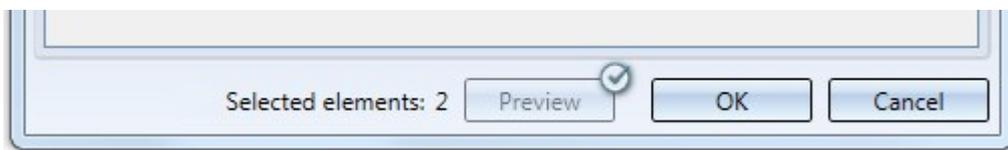
# Element Installation

## Service Pack 2 2018 (V 2302)

### Preview for element installation and sub-structure

Both in the **Element installation** and in the **Sub-structure** dialogue window you have now the option to activate or deactivate the generation of previews. By default, the checkbox is active, that is, a new preview will be generated each time you change an entry in the dialogue window.

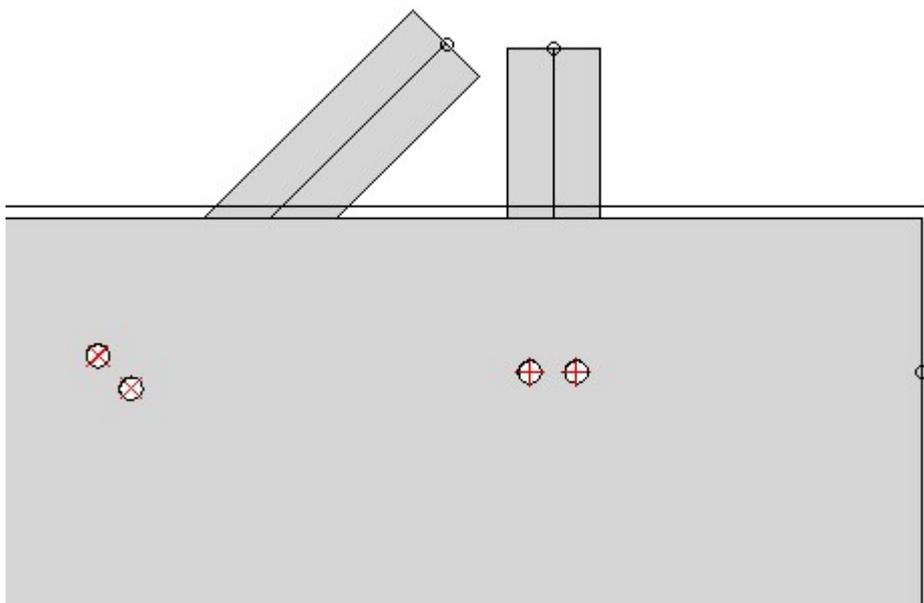
For a more fluent working with large sub-structures you can remove the checkmark from the button. After this, a preview will only be displayed if you explicitly click on the button (i.e. previews will no longer be generated automatically). Of course, you can activate the automatic preview generation again at any time.



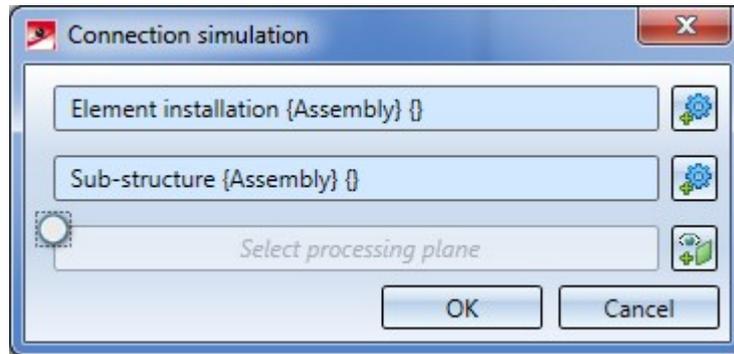
When working with large installations, removing the checkmark may therefore significantly speed up the process - especially if several variables need to be changed.

### Connecting element installation and sub-structure

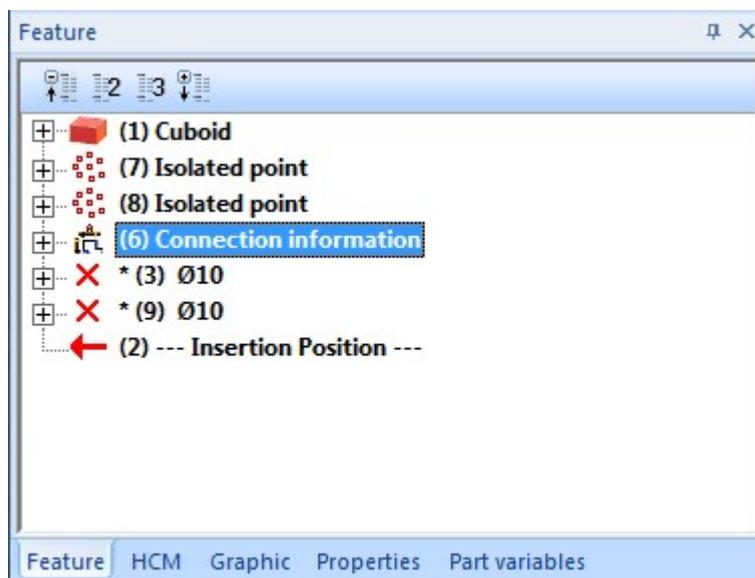
Use the **Connection** function at **Civil Engineering functions > Element installation** to automatically connect element installations and sub-structures. Bores and boltings can also be created automatically in the process.



After calling the function, simply select the element installation and the sub-structure that you want to connect.



Own variants for the element installation or sub-structure can be prepared by assigning **Connection information** for the utilization of this function.



In addition, the following two functions are available to you:

The **Connection simulation** function simulates the applying of a connection to a part that has not been mounted in an element installation or sub-structure. This can be helpful when creating own variants.

The **Connection geometries** function displays the connecting edges of one or several parts or installations. This function, too, can be helpful for error detection in own variants.

## Catalogues/Tables for Element Installation / Substructures

Tables can, depending on the catalogue to which they belong, be assigned to specific categories. For Element Installation and Substructures the categories

- ELEMENT\_INSTALLATION\_ELEMENTS bzw.
- SUBSTRUCTURE\_ELEMENTS

are available.

If a new table is created for these categories, the corresponding table columns will be created automatically.

The tables at **Factory standards > Installation Planning - Parts and Processings > Element installation** and **Factory standards > Installation Planning - Parts and Processings > Sub-structure**, respectively, that are supplied with HiCAD are already assigned to the corresponding categories.

## ALUCOBOND® tray panels SZ-20

- Connection type "Attic"

For the connection type **Attic, long** you can now specify the angle of the folded sheet.

Global settings
Extended settings

**Parameters**

Semi-finished product: ALUCOBOND 4mm I603 Grey blue metallic - ALUCOBONI

**Vertical section**

Connection, top: Attic, long

(1) Attic depth: 250

(2) Length of folded sheet: 70

(3) Angle: 100

(4) Angle-2: 90

Base point: Standard

**Vertical section**

Connection, top: Attic, long

(1) Attic depth: 400

(2) Length of folded sheet: 100

(3) Angle: 100

(4) Angle-2: 90

**Vertical section**

Connection, top: Attic, long

(1) Attic depth: 400

(2) Length of folded sheet: 100

(3) Angle: 100

(4) Angle-2: 135

For the connection types **Attic, long / Attic, short** you can now specify the type of the lateral flanges on the **Extensions for SZ-20 Standard** tab.

Global settings
Extended settings
Extensions for SZ-20 Standard

Lateral type for attics: Without side flanges

Mounting of splice sheet:  Bonded  Riveted

**Standard type**

*Standard type*

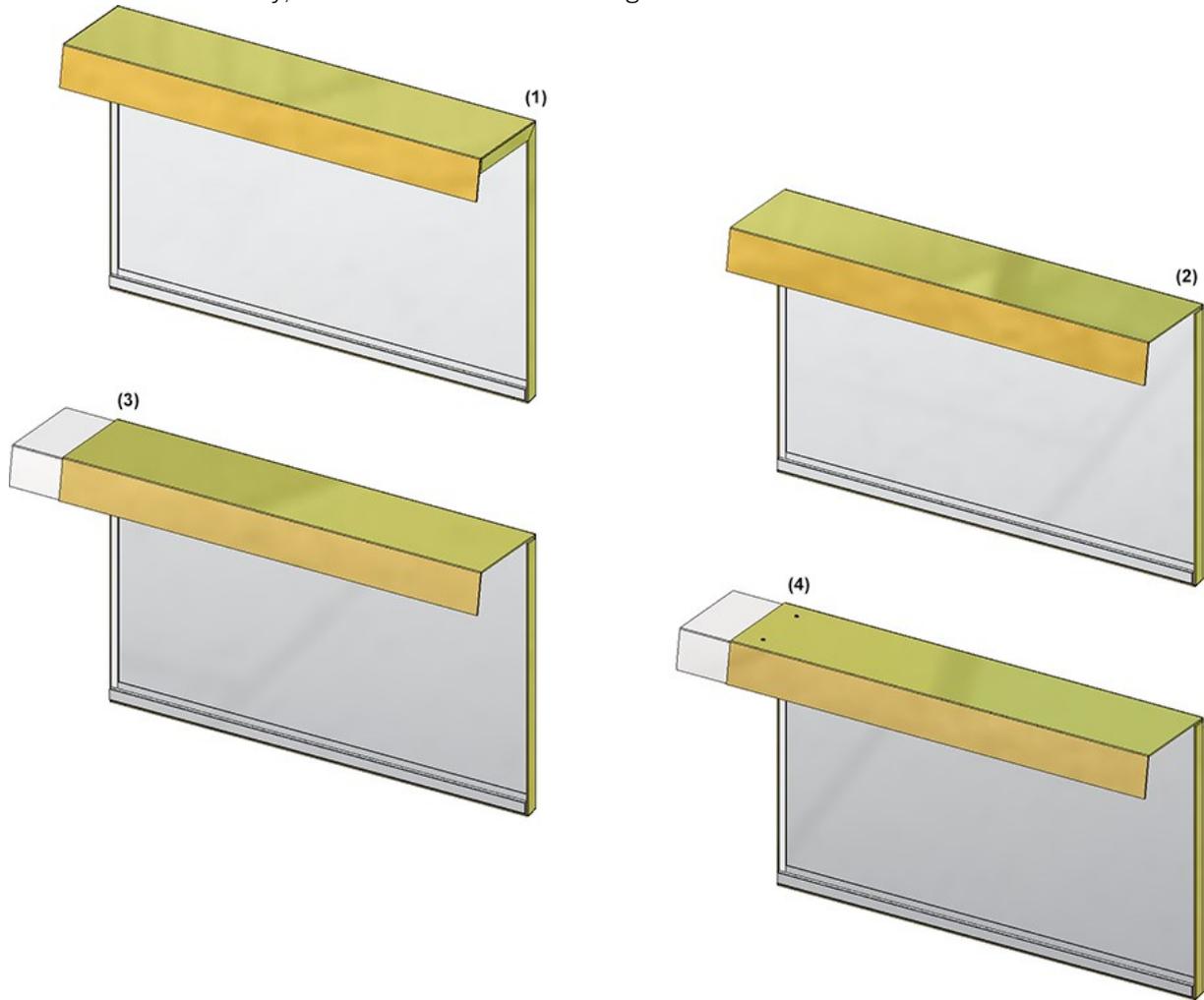
**Without side flanges**

*Without side flanges*

**With splice sheet**

*With splice sheet*

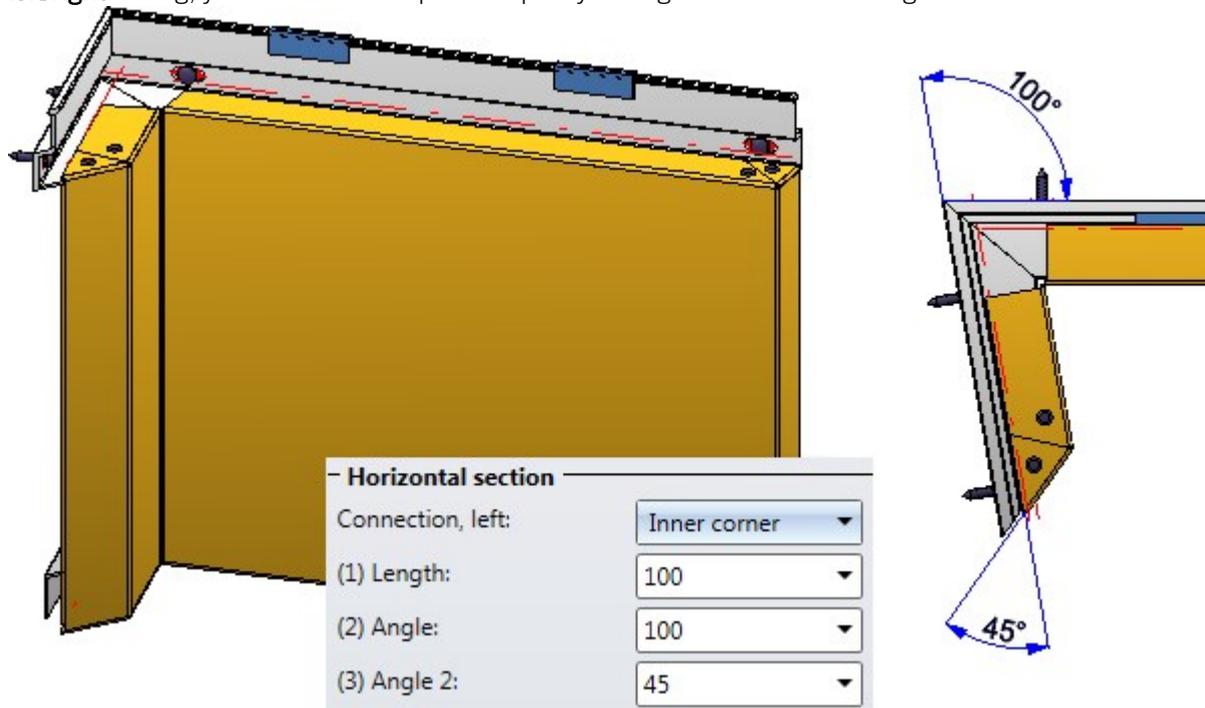
The connection between splice sheet and tray panel can be a bonded or a riveted one. The rivets will be inserted on one attic side only, at a 50 mm distance to the edge.



(1) Standard; (2) Without side flanges; (3) Splice sheet, bonded; (4) Splice sheet, riveted

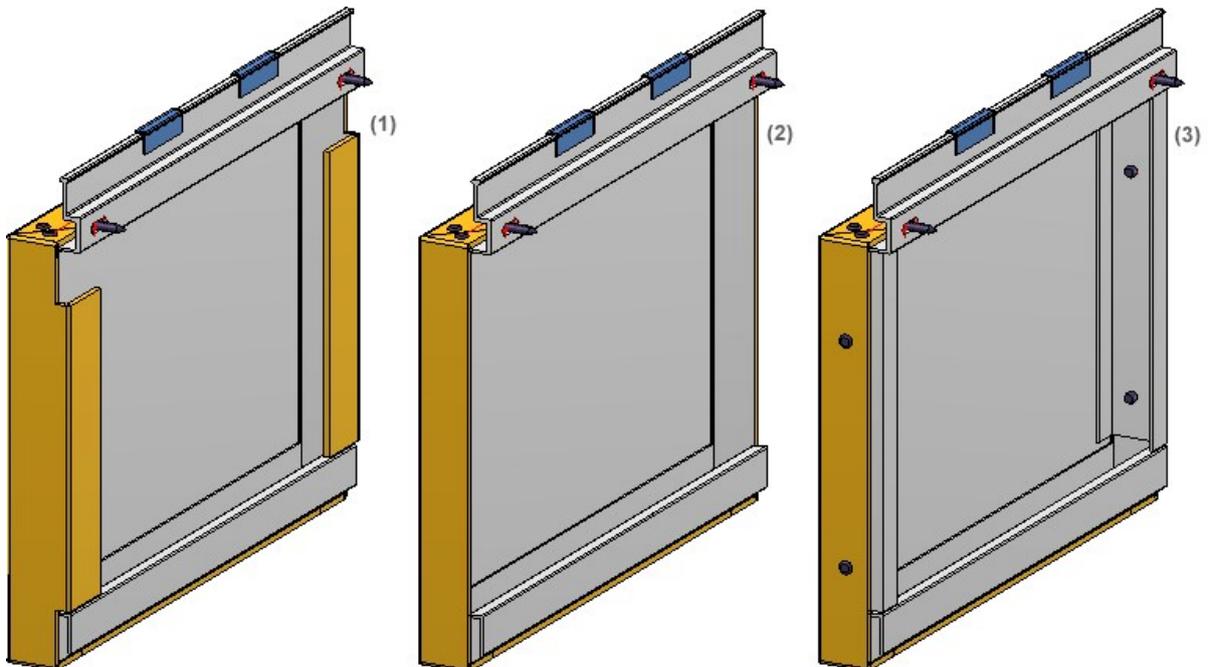
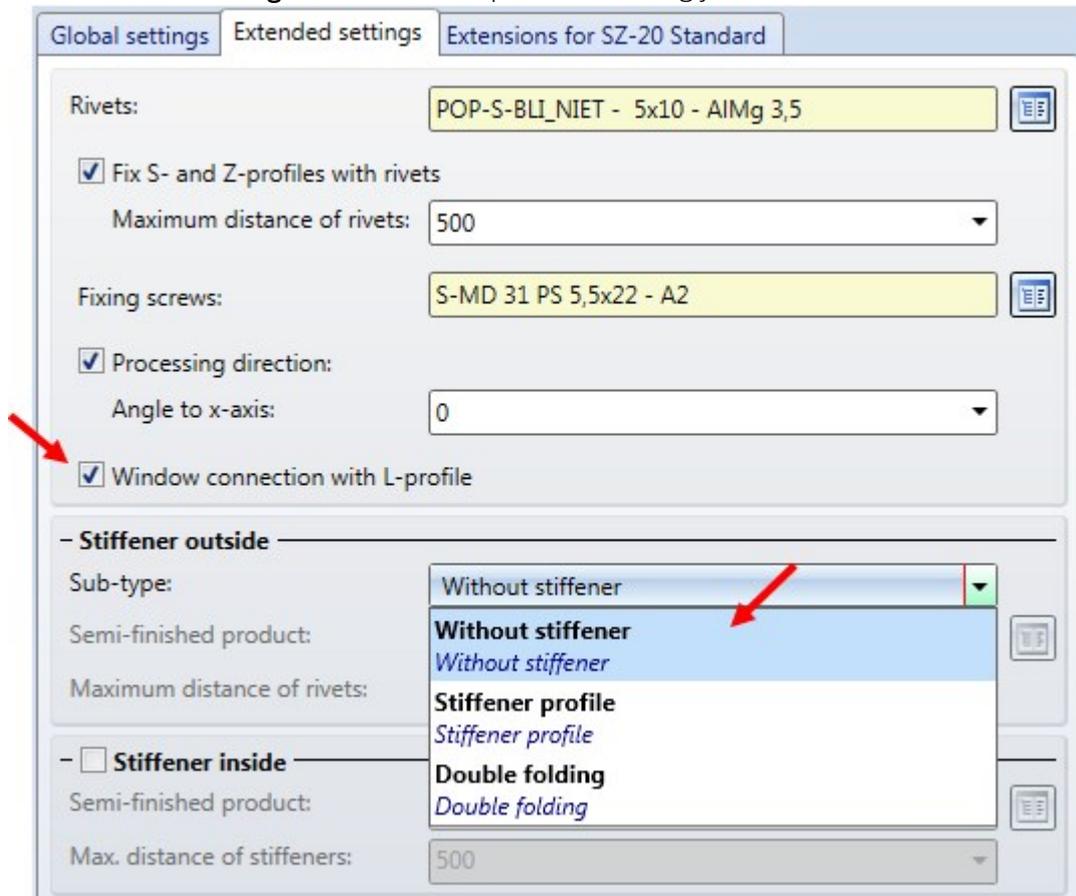
- Inner corner / Outer corner with joint - Additional angle specification

If you choose **Inner corner** or **Outer corner**, with **folded sheet** for the **Horizontal section - Connection left/right** setting, you have now the option to specify an angle for the 35 mm flange.



■ Double sheet folding

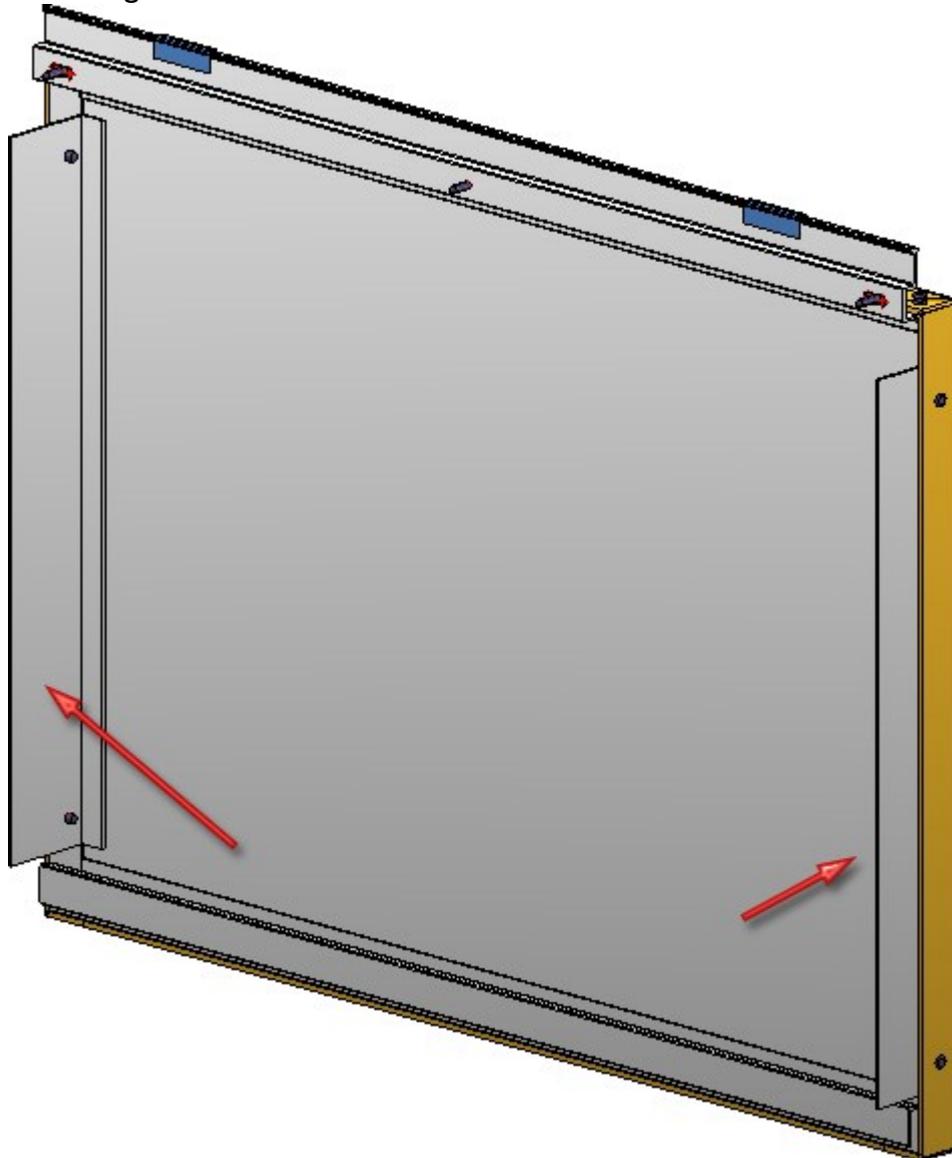
Instead of using the stiffeners at the edge you can also apply double sheet foldings. For this purpose the options on the **Extended settings** tab have been expanded accordingly.



(1) Double sheet folding; (2) without stiffener, (3) with stiffener

- **Horizontal section - Window connection, Jamb**

If you choose **Window connection, Jamb** for the **Horizontal section - Connection left/right** setting, you can insert an additional L-profile. To do this, activate the **Window connection with L-profile** checkbox on the **Extended settings** tab.



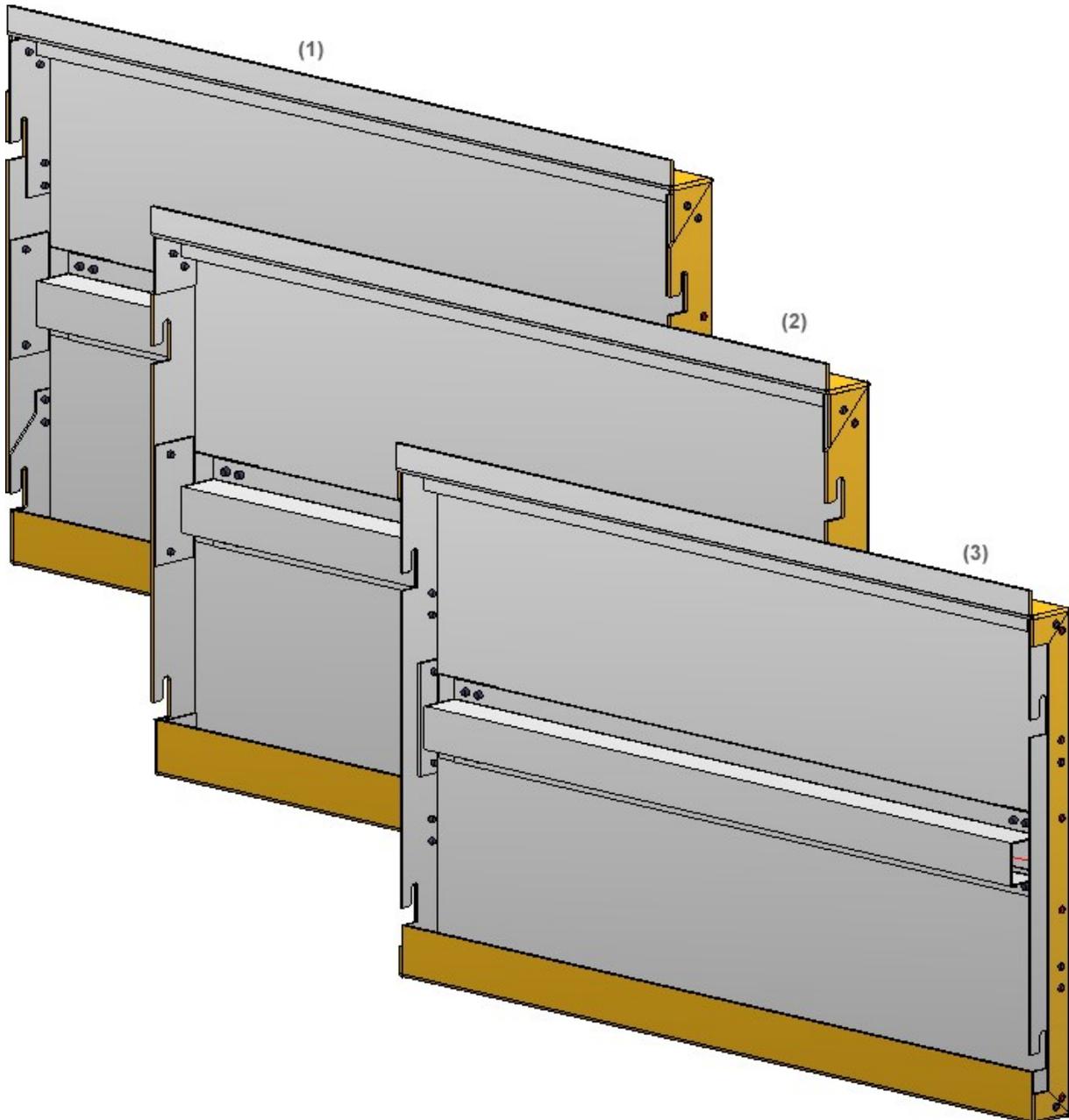
### Bonded ALUCOBOND® tray panels

Bonded ALUCOBOND tray panels can now be placed on any rectangular or polygonal sketch areas.

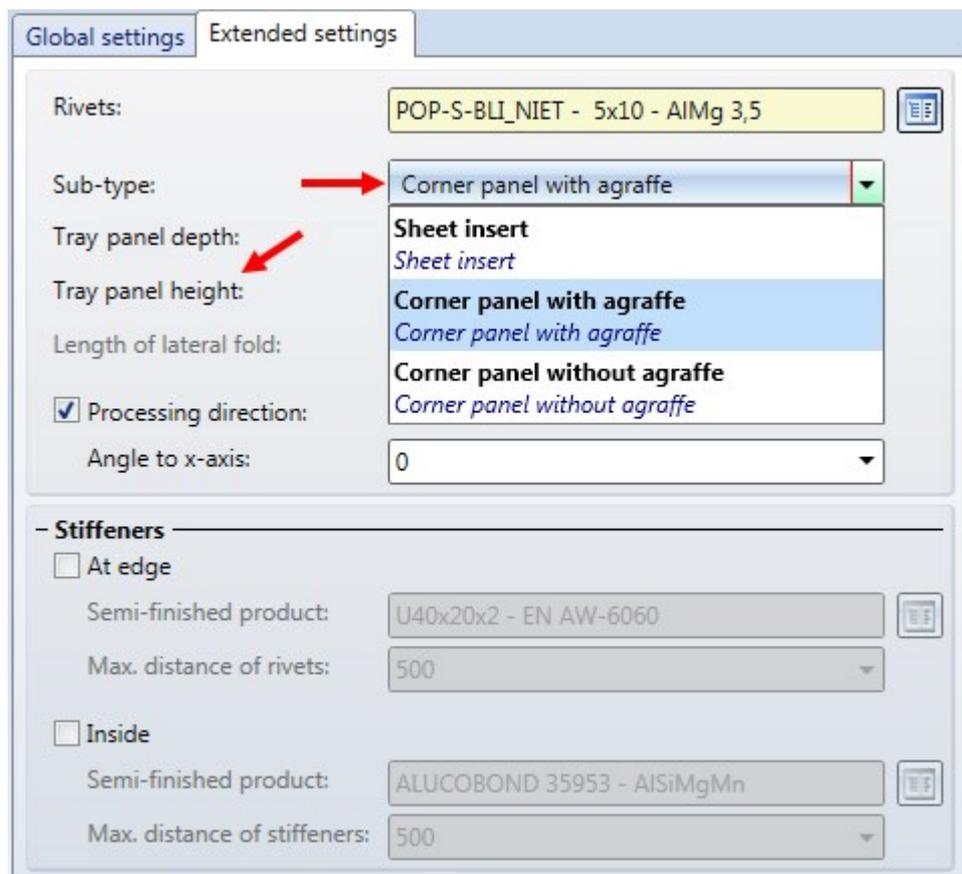
## Suspended ALUCOBOND® tray panels

For suspended ALUCOBOND tray panels you can now, besides the tray panel depth, also freely configure the tray panel height. The options for sub-types have been expanded as well - you can now choose between:

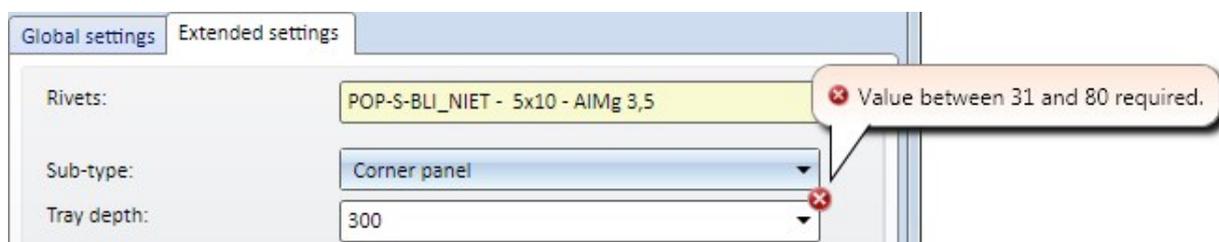
- Sheet insert,
- Corner panel with agraffe, and
- Corner panel without agraffe



Sub-type: (1) Corner panel with agraffe; (2) Corner panel without agraffe; (3) Sheet insert - with stiffeners at edges and in the middle



If you choose **Corner panel.....**, a message will be displayed if an invalid tray panel depth has been entered. e.g.:



The selection list for Connection types, left / right now offers the new **Outer corner with joint** option:

- Standard**  
*Standard*
- Inner corner**  
*Inner corner*
- Outer corner with folded sheet**  
*Outer corner with folded sheet*
- Outer corner with joint**  
*Outer corner with joint*
- Window connection, Jamb**  
*Window connection, Jamb*
- Without flanges**  
*Without flanges*

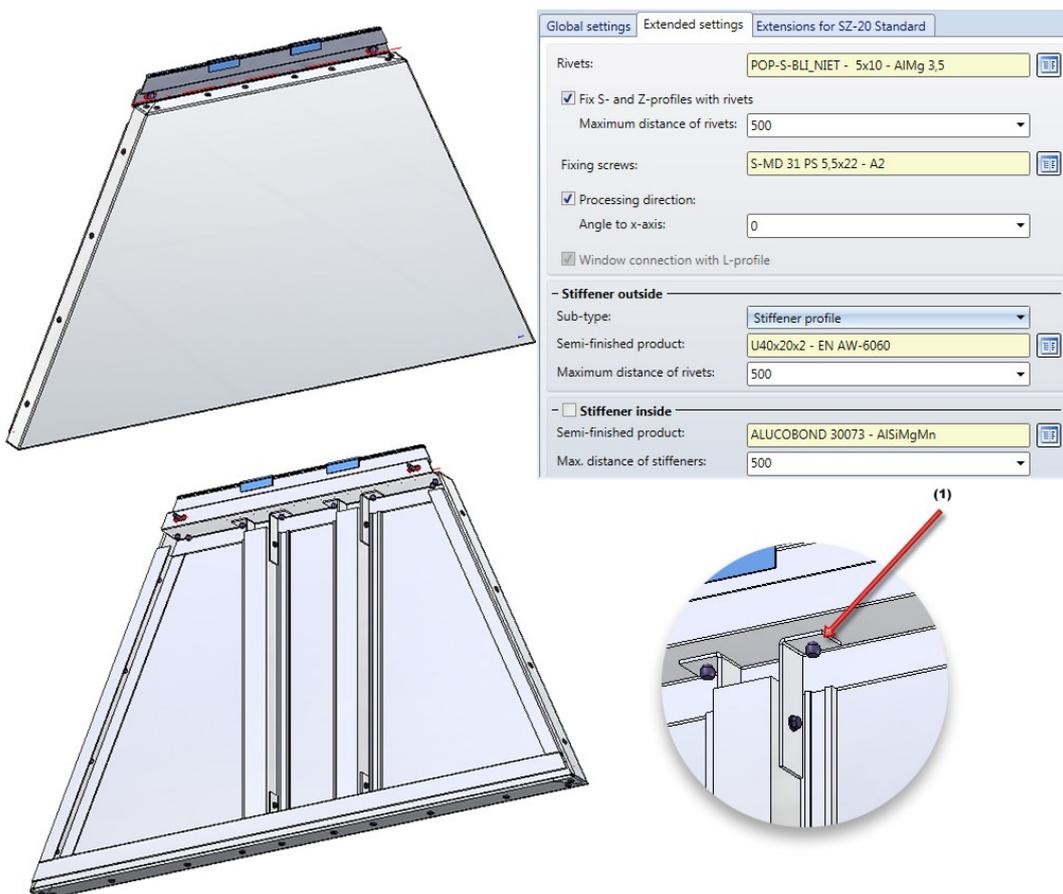
## Service Pack 1 2018 (V 2301)

### ALUCOBOND® Tray panel SZ-20 with diagonal sides

ALUCOBOND Tray panels SZ-20 could previously only be installed on rectangular sketch areas. As of HiCAD 2018 SP1 four-sided sketch areas of different shapes, e.g. parallelograms or trapezes, are also possible. This allows you to create ALUCOBOND® Tray panel SZ-20 with diagonal sides.

Please note the following for ALUCOBOND® Tray panels SZ-20 - depending on the settings during insertion:

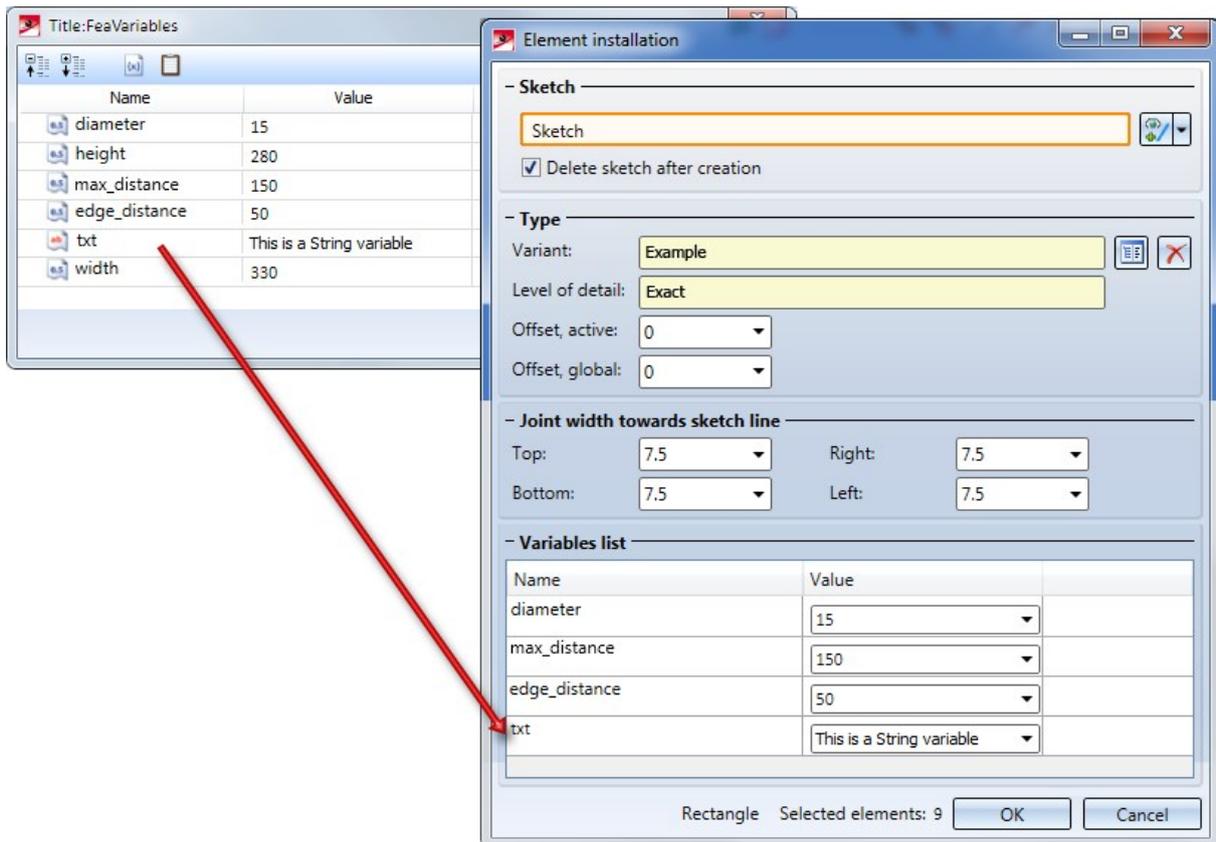
- The S- and Z-profiles are trimmed.
- The stiffener profiles are trimmed to the S- and Z-profiles.
- The stiffener profiles are bonded and fixed with angular brackets (Sheet Metal) at the top and the bottom.
- The bores for these brackets are not generated in HiCAD. Reason: They are not created by the machine, but by a worker after bonding the stiffener profile to the sheet. This bonding cannot be performed accurately enough to exactly match the previously drilled bores. (1)



Example of a trapezoidal ALUCOBOND®Tray panel SZ-20, (1) No bores are created here!

### Management of String variables

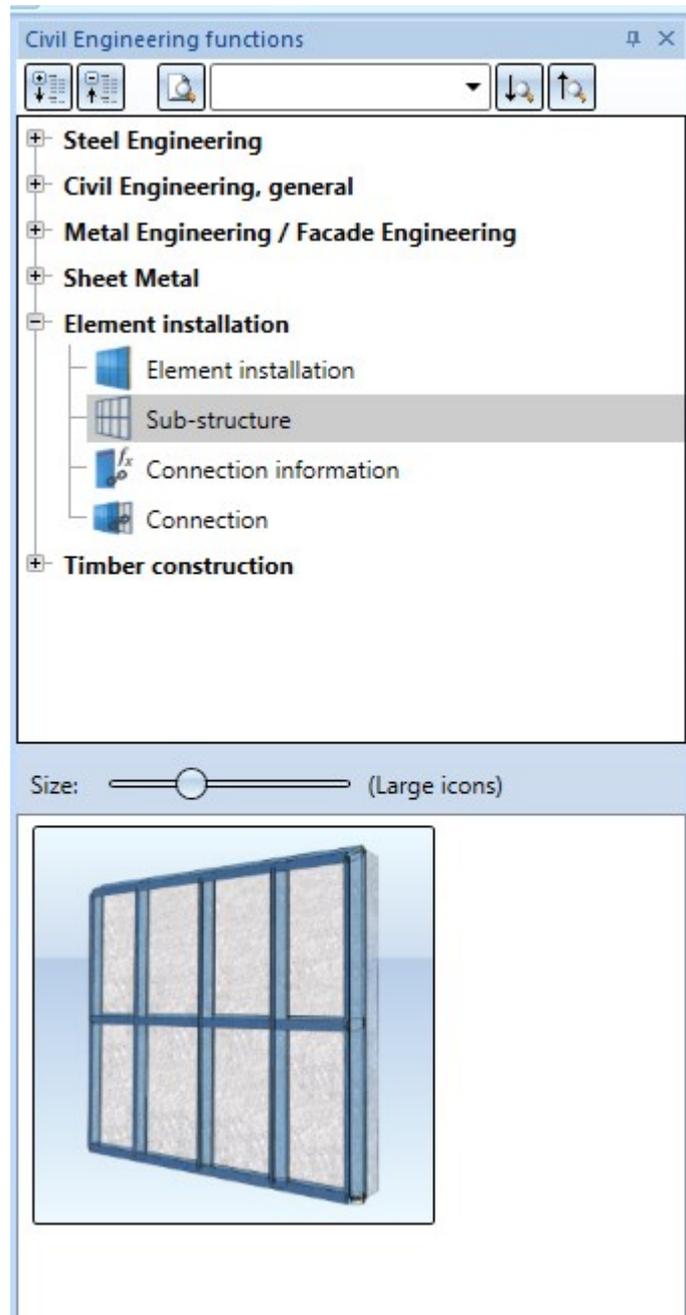
As of HiCAD 2018 SP1 the Element Installation also allows a management of variables of the type "String". This means that if the variant of an installation element contains a String variable, it will also be shown in the Element Installation dialogue window.

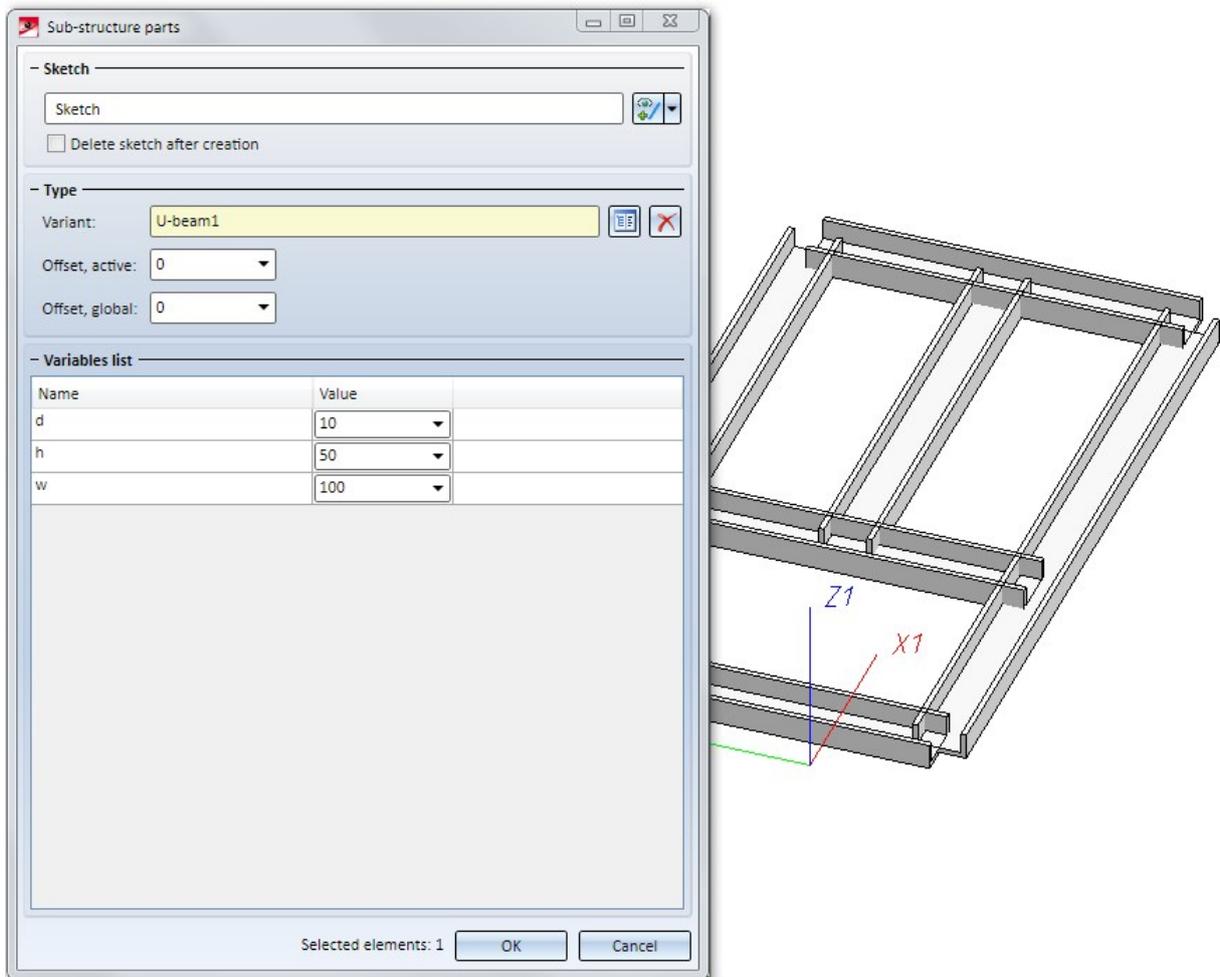


## Major Release 2018 (V 2300)

### Sub-structures

Elements that are installed with the help of the **Element installation** function must often be hanged on a sub-structure or be fixed to it. In addition to the **Element installation** function that allows a placing of tray panels onto surfaces of sketches, the **Sub-structure parts** function is now available in the Civil Engineering functions docking window. This function enables you to place profiles onto edges of sketches and, in doing so, to rapidly create sub-structures matching the installed elements.



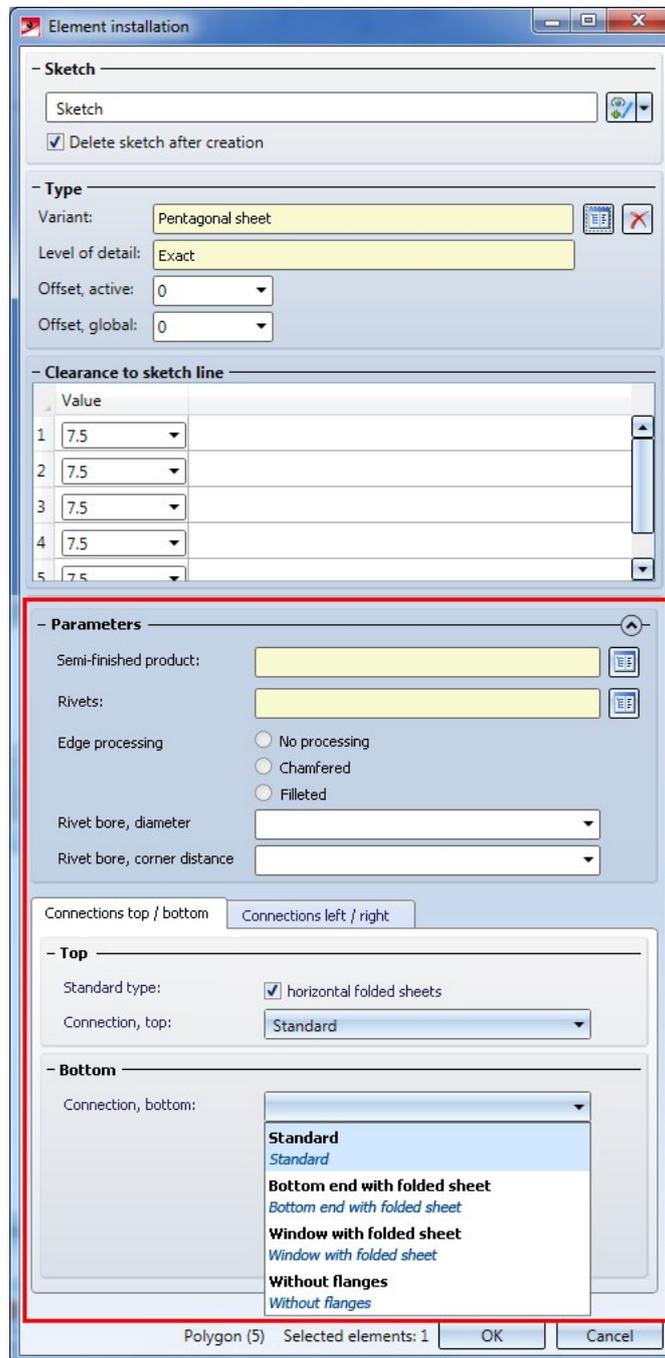


You have also the option to create your own profiles to use them for sub-structures.

## New options for customer-specific dialogue windows

New options for customizing the parameters area of the dialogue window are now available:

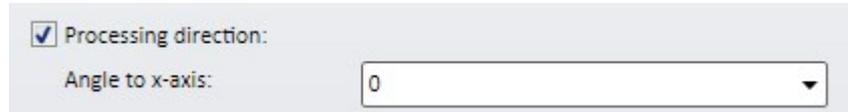
You have now the option to distribute parameters among several tabs. Furthermore, it is now possible to expand and collapse defined areas.



Customizations of the dialogue window should only be applied by ISD Consultants or users with good programming skills!

## Specify sheet texture direction

For all ALUCOBOND tray panels you can now specify the direction of the sheet textures on the Extended settings tab.



If you want to specify a particular direction of the textures, activate the Processing direction checkbox and enter the angle to the X-axis.

The processing direction has an effect on the position of the trays in the workshop drawing and the later nesting ( Cobus Ncad ) by the NC machine systems (Sheet Metal > Sheet development > Extras  > All sheets).

## General changes in the dialogue windows

- The term **Clearance to sketch line** has been replaced with the term **Joint width towards sketch line**.
- In the dialogue windows for the ALUCOBOND tray panels the **Connection, bottom** has been renamed to **Base point**.

## ALUCOBOND® tray panels

- You have now the option to mount stiffeners directly onto ("within") ALUCOBOND® Tray panels SZ-20. For this purpose the **Extended settings** tab has been expanded accordingly. In addition, you can specify the processing types for the suspension (S- and Z-)profiles - with or without riveted connection. Stiffeners are always riveted to the plate.

Global settings
Extended settings
Extensions for SZ-20 Standard

Rivets: POP-S-BLI\_NIET - 5x10 - AlMg 3,5 ☰

Fix S- and Z-profiles with rivets

Maximum distance of rivets: 500 ▼

Fixing screws: S-MD 31 PS 5,5x22 - A2 ☰

Processing direction:

Angle to x-axis: 0 ▼

Window connection with L-profile

---

**- Stiffener outside**

Sub-type: Double folding ▼

Semi-finished product: Without stiffener ☰

Maximum distance of rivets: Stiffener profile ☰

*Without stiffener*

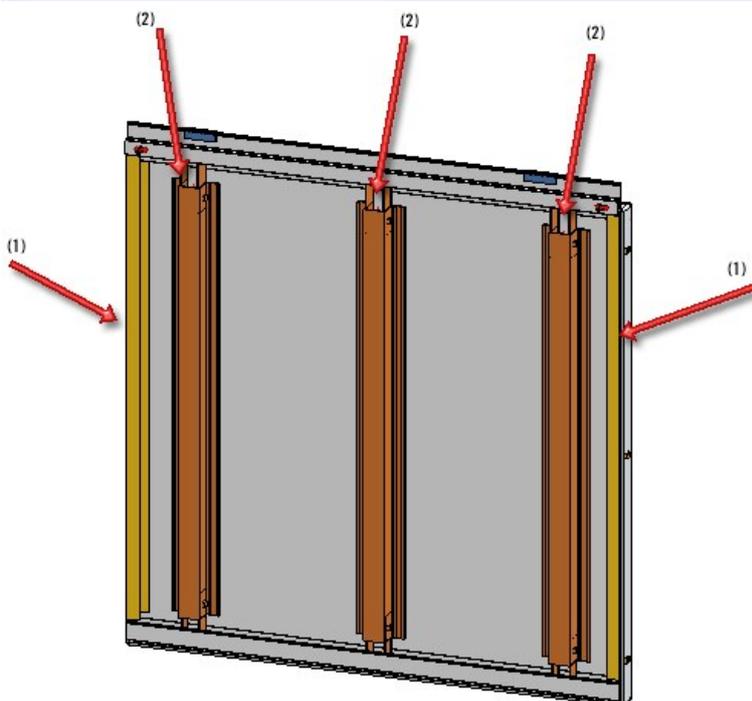
*Stiffener profile*

**-  Stiffener inside**

Semi-finished product: Double folding ☰

*Double folding*

Max. distance of stiffeners: 500 ▼



ALUCOBOND® Tray panel SZ-20 - (1) Stiffener profiles at edges of panel; (2) Stiffener profiles on ("inside") panel

- Stiffeners "within" can also be applied to suspended ALUCOBOND tray panels.
- The selection lists for connection types have been modified:

#### Top

<b>Standard</b>
<i>Standard</i>
<b>Attic, long</b>
<i>Attic, long</i>
<b>Attic, short</b>
<i>Attic, short</i>
<b>Window connection, Parapet</b>
<i>Window connection, Parapet</i>
<b>Without flanges</b>
<i>Without flanges</i>

#### Base point

<b>Standard</b>
<i>Standard</i>
<b>Ending, bottom</b>
<i>Ending, bottom</i>
<b>Window connection, Lintel</b>
<i>Window connection, Lintel</i>
<b>Without flanges</b>
<i>Without flanges</i>

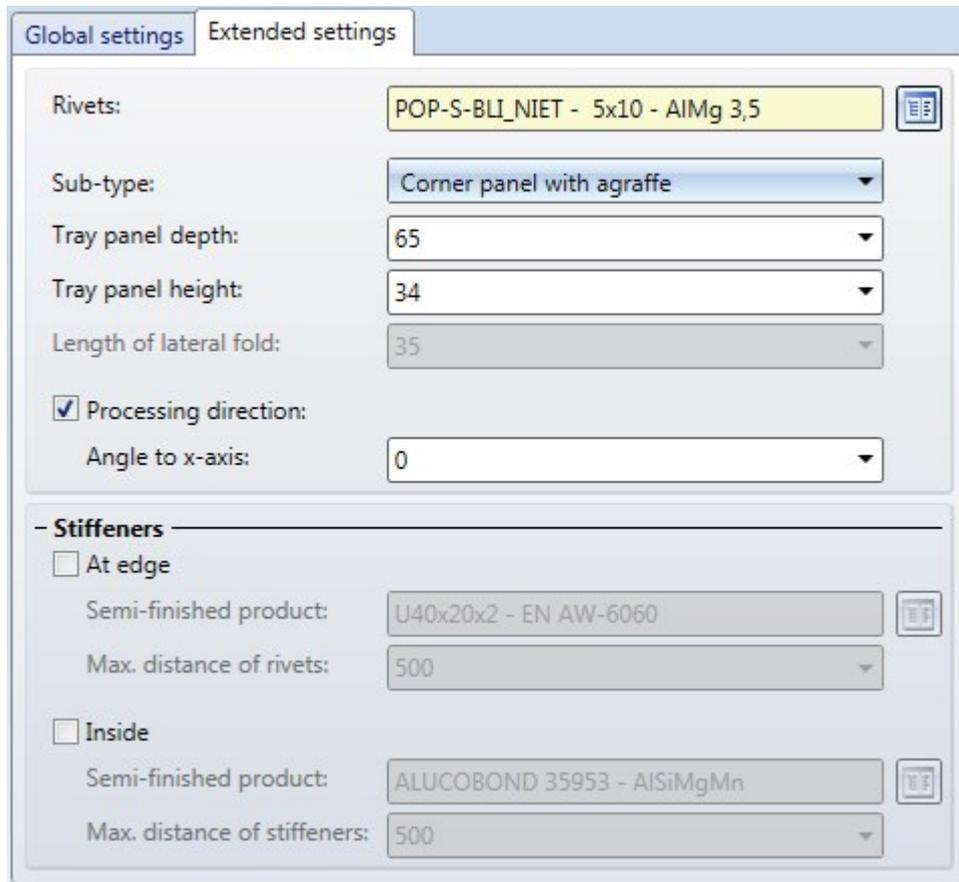
#### Left / Right

<b>Standard</b>
<i>Standard</i>
<b>Outer corner with folded sheet</b>
<i>Outer corner with folded sheet</i>
<b>Outer corner with joint</b>
<i>Outer corner with joint</i>
<b>Inner corner</b>
<i>Inner corner</i>
<b>Window connection, Jamb</b>
<i>Window connection, Jamb</i>
<b>Without flanges</b>
<i>Without flanges</i>

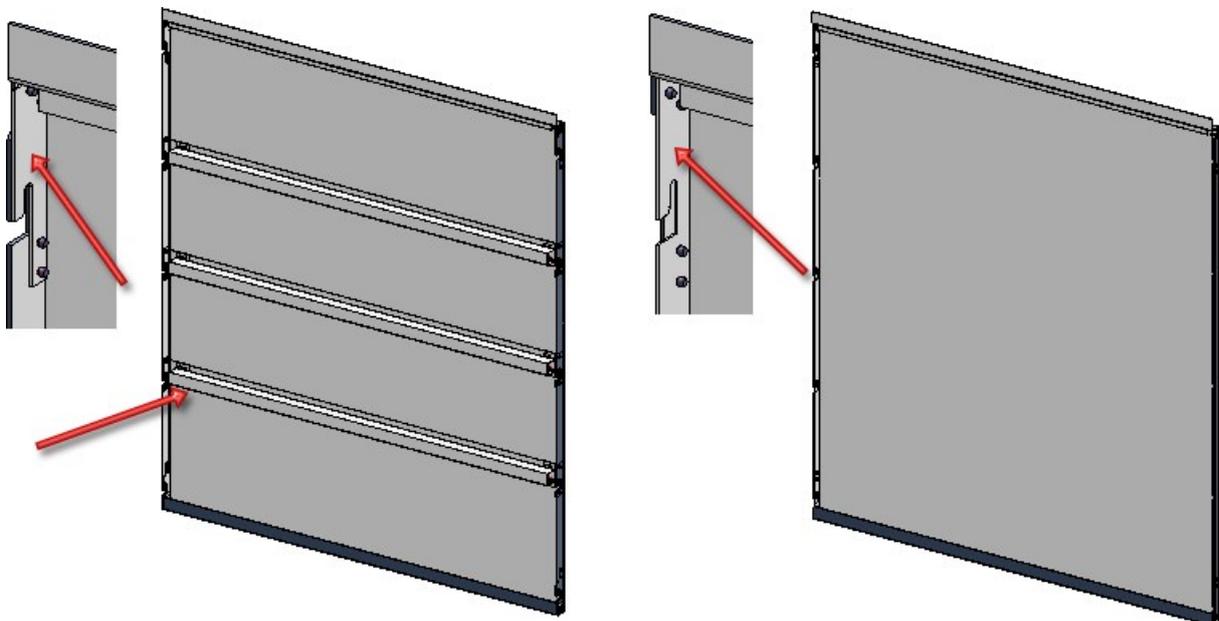
- For lateral connections and the connection type **Outer corner with joint**, HiCAD now also inserts the bores for the joints and the rivets of the ALUCOBOND® plates for the SZ panels in the "corners".

### ALUCOBOND® panels, suspended

As with the ALUCOBOND tray panels SZ 20, it is now possible to insert stiffeners at the edge or directly on suspended ALUCOBOND® panels. The **Extended settings** tab has been expanded accordingly for this purpose. Furthermore, you can now specify the processings of the profiles for suspension (with or without rivets), choose a sub-type (corner panel or sheet insert), as well as the tray depth.



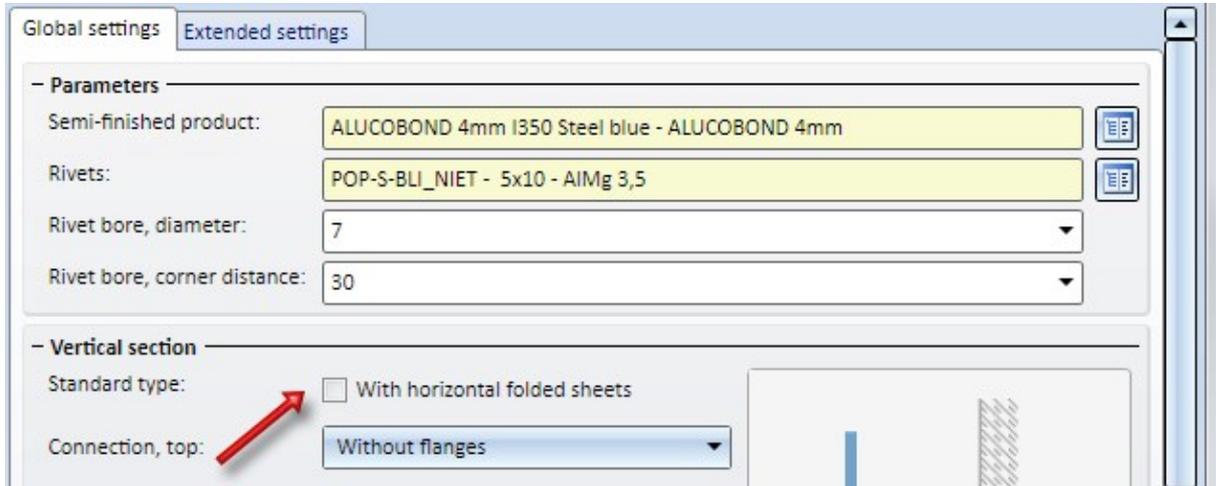
Stiffeners are always riveted to the panel.



Left: With corner panels and stiffeners; Right: With sheet insert, without stiffeners

## ALUCOBOND® panels, riveted

The default setting for riveted ALUCOBOND panes has been changed. The **With horizontal folded sheets** is now deactivated by default.



# Plant Engineering

## Service Pack 2 2018 (V 2302)

### Isometry and Pipe Spool Drawing

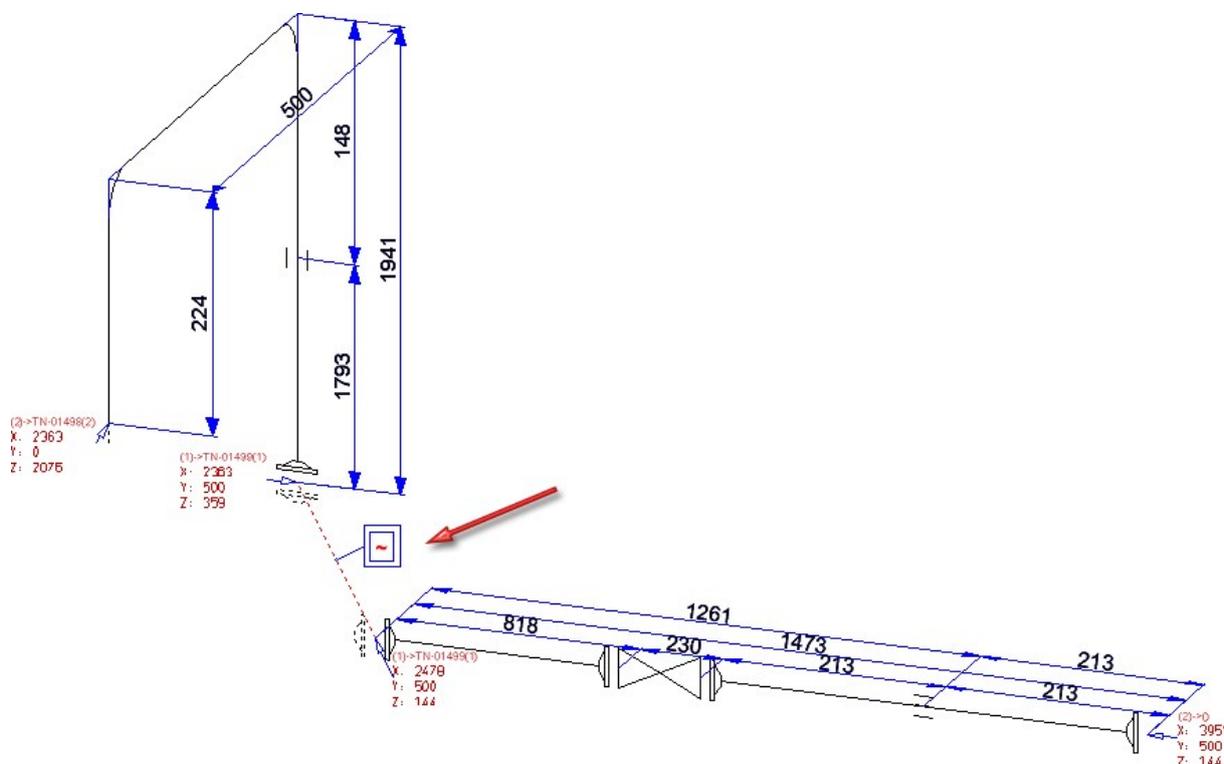
#### Drawing for pipeline set

The functions **Isometry** and **Pipe spool drawing** create workshop drawings for pipelines. Both functions are operated in a similar way. In older versions, both functions could create a drawing for only one pipeline, though.

As of SP2, HiCAD 2018 offers the option to combine several pipelines into a set of pipelines, which will then be regarded as "one" pipeline when generating the isometry or the pipe spool drawing. Since you can freely define such pipeline sets, you are enabled to separate your model drawing into pipelines in a more flexible way and without having to have the drawing generation already in the back of your mind.

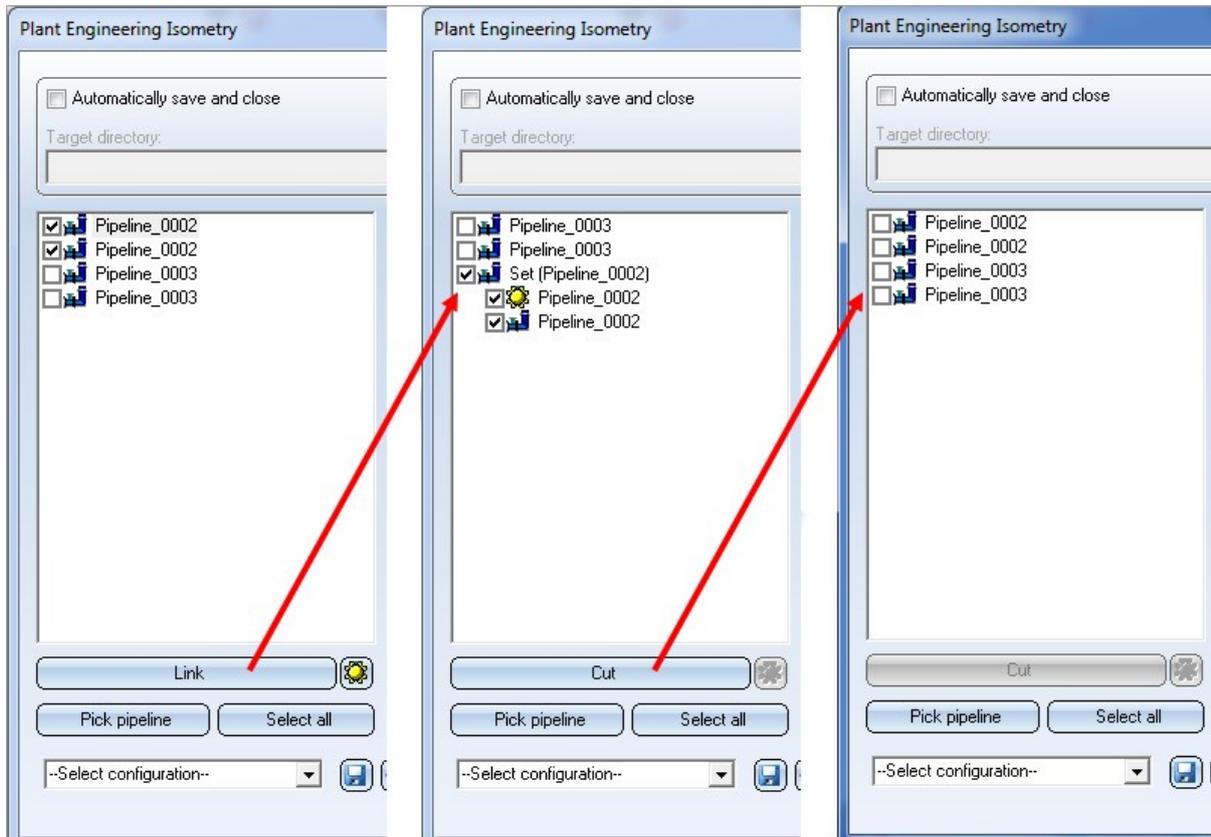
The decisive difference between the drawing of a pipeline set and the drawing of a separate pipeline is that the points where the pipelines of a set converge will be interpreted as inner points of a pipeline. Chain dimensions span these points and connecting points are marked accordingly; no annotation tags with connection coordinates will be created.

In an isometry, attention is paid to the question whether the set is continuous. If required, distance elements will be created and marked with by a tilde ~:



To define a pipeline set, simply select the pipelines you want to combine into a set and click on the **Link** button. The selected pipelines will then be merged into a set in the list, and will automatically obtain the name **Set**, followed by the name of the main pipeline in brackets, e.g. **Set (Pipeline\_0002)**.

The **Link** button is context-sensitive. When you select a pipeline set, the name of the "Link" button will change to **Cut**. Click the **Cut** button to split the set up into the individual pipelines again.



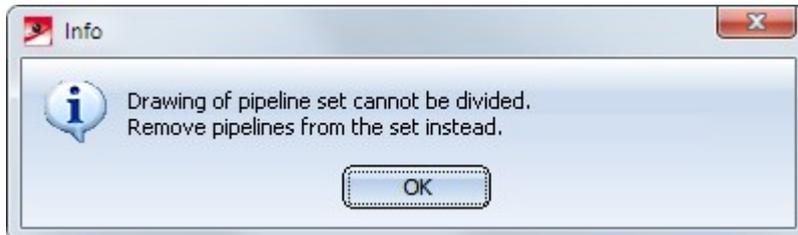
The main pipeline is marked with the  symbol. The purposes of the main pipeline in the isometry and pipe spool drawing are as follows:

- The dimensioning is assigned to the main pipeline (or, in the isometry, to the copy of the main pipeline).
- The coordinate system that was defined in the layout plan with the **Alignment of derived drawings** function will be taken over by the main pipeline.
- The optionally defined Reference Coordinate system will also be taken over by the main pipeline.
- If the drawing is created in a new model drawing, the name of the created new model drawing will be based on the name of the main pipeline. This allows you to create the same drawing multiple times with the parameters of different main pipelines, preventing any conflicts between names.
- The link between layout plan and isometry/pipe spool drawing that is evaluated by the **Linked documents** function, is also established via the main pipeline.

You can use the  icon to make a different pipeline the main pipeline of the set.

**i Please note:**

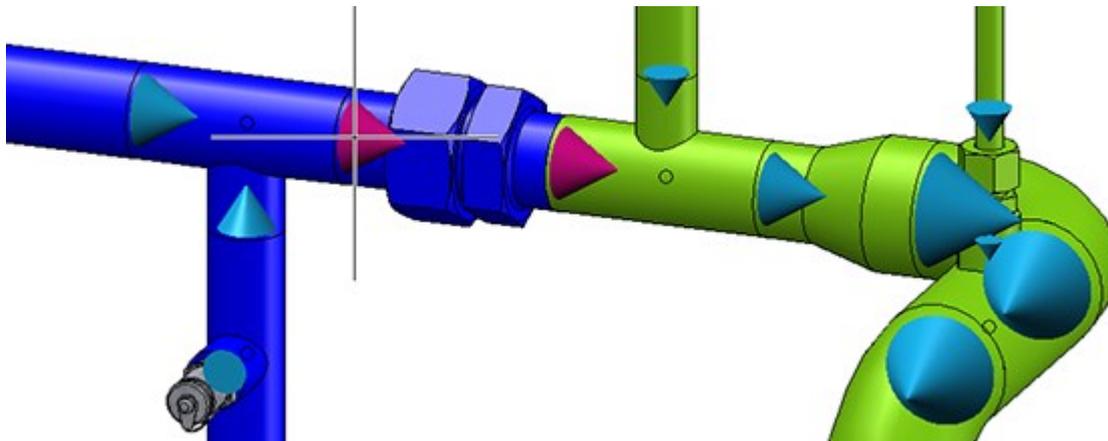
- After creating the isometry of the pipeline set, it will not be possible to spread this isometry over several pages subsequently. When you call the function **Isometry+Pipe Spool Drawing > Settings > Specify division** , the following message will be displayed:



- Together with the **Divide pipeline**  function the use of pipeline sets is a flexible possibility to arrange isometries or pipe spool drawings appropriately.

### Enhanced Flow Editor

The Flow Editor does not only consider individual pipelines, but interprets connected, continuous pipelines as one, jointly editable pipeline set, as shown by the blue and the green pipeline in the image below:



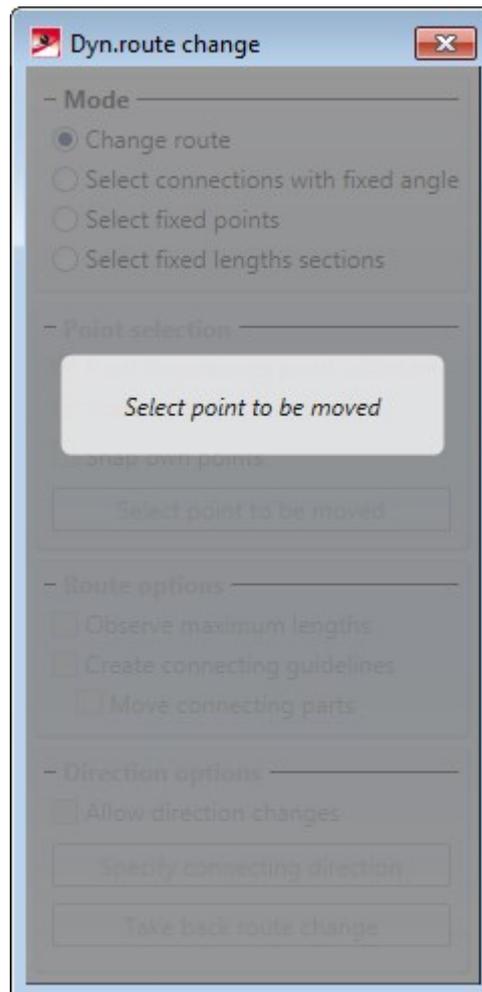
## Pipeline Tools

### Change route via new dialogue

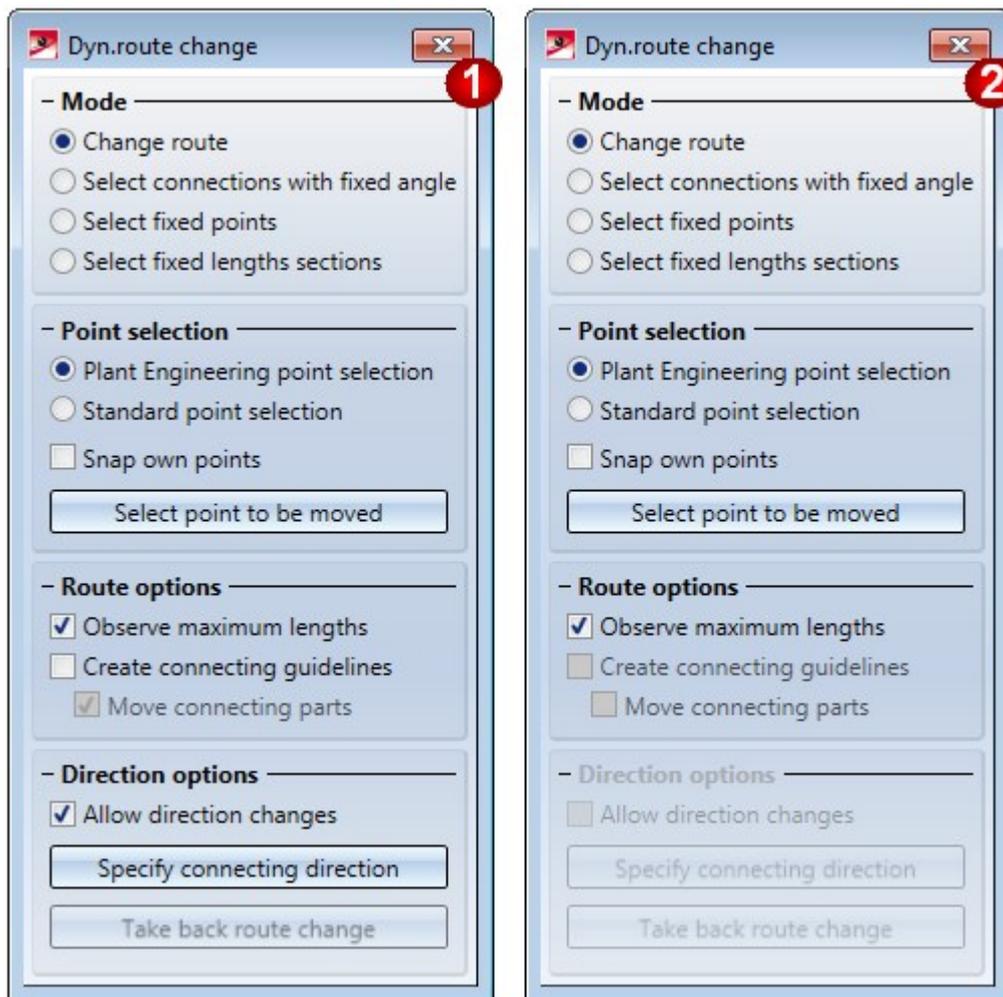


In previous versions, the **Dynamic route change** function was available via a context menu. In most cases, multiple function calls were required to achieve the desired result. As of SP2 this context menu will be replaced with a new, separate dialogue that significantly facilitates the selection of the required settings.

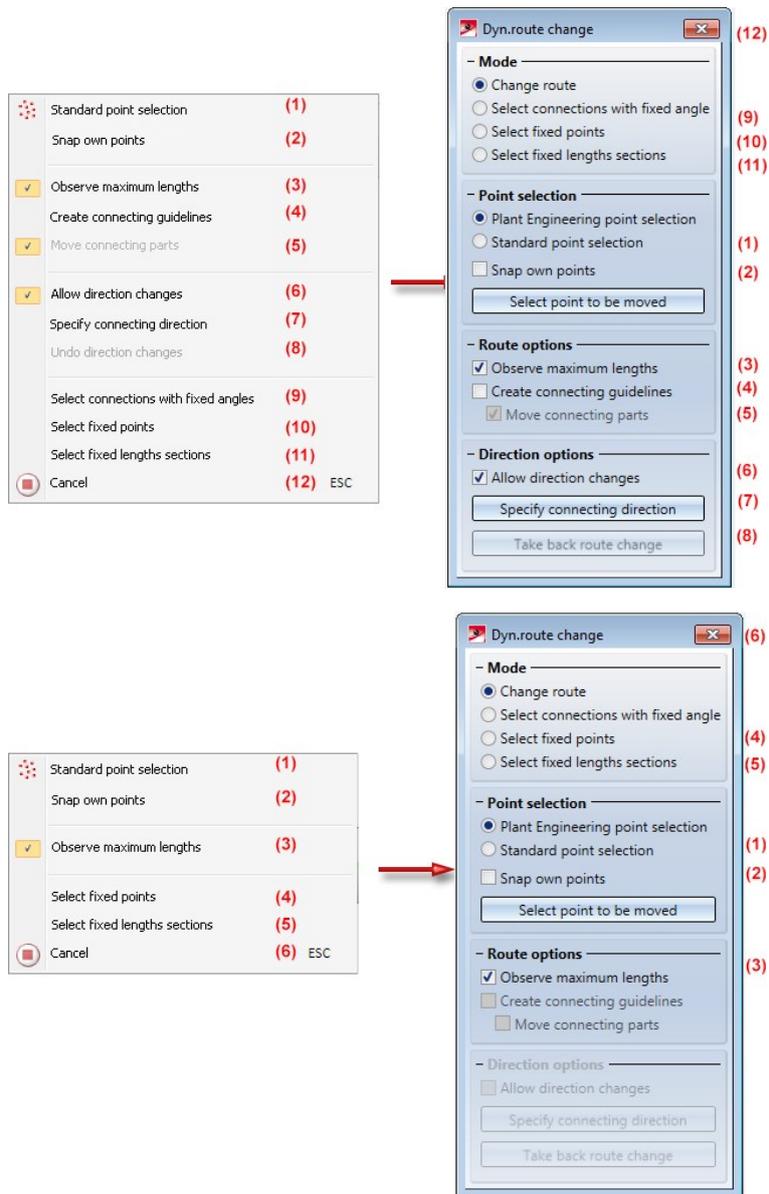
When you call the function, HiCAD will prompt you to select the point to be moved:



After selecting the point you need to specify the target point. Now you can right-click to open the dialogue window with the different setting options for the route change. The availability of options in the dialogue depends on whether you selected a pipeline end point (1) or a point in the middle of the pipeline (2) as the point to be moved.



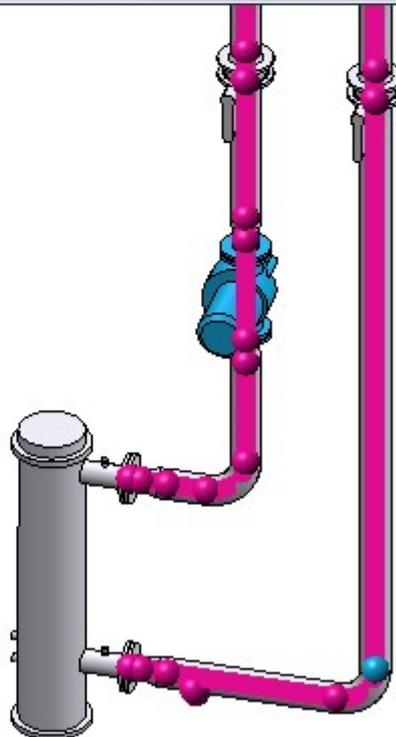
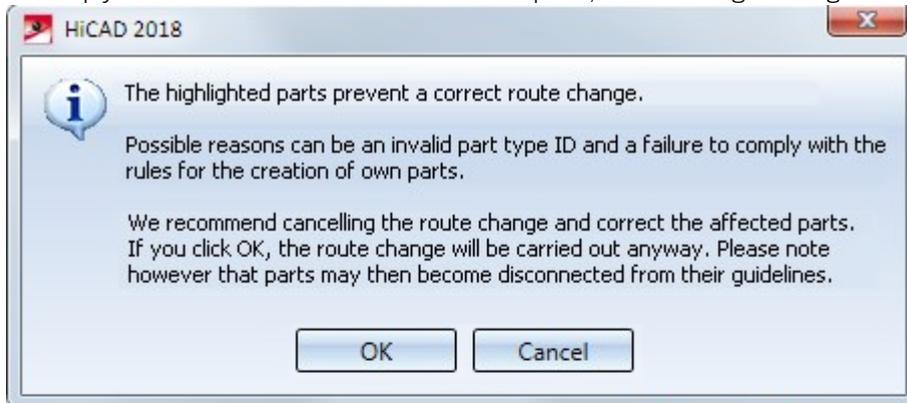
The images below show where the functions from the old versions can be found in the new dialogue window of SP2:



Also new is the option to correct the selection of the point to be moved. For this purpose you can use the **Select point to be moved** function. This function is helpful in cases where the selection of the correct point is difficult, e.g. if the pipeline ends in narrow parts such as seals and several points are very close to each other.

## Correct route change not possible

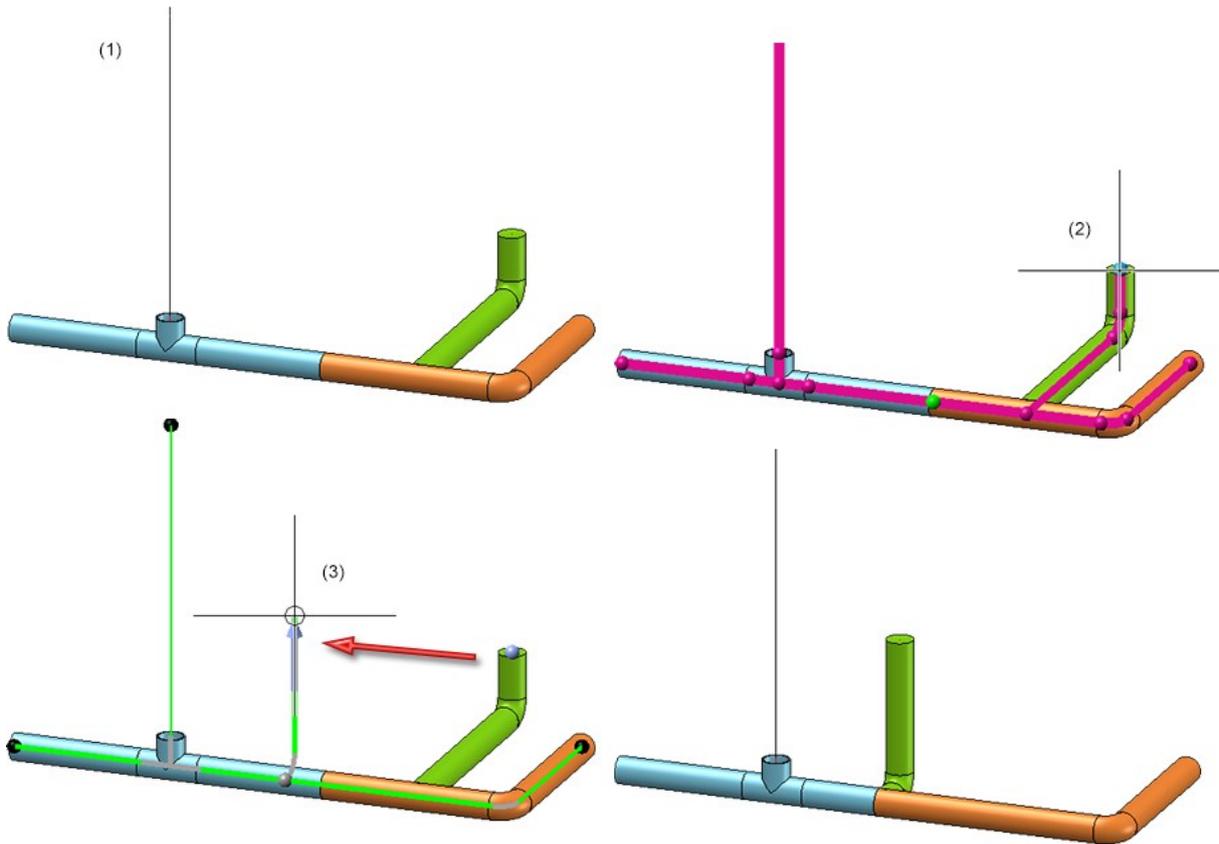
If the pipeline contains parts that prevent a correct route change, e.g. because of invalid Part type IDs, or due to failure to comply with the rules for the creation of own parts, the following warning message will be displayed:



If you click **OK** you can still carry out the route change - **However, we recommend cancelling the route change and correct the highlighted parts instead.**

### Change route of pipeline sets

The dynamic route change now interprets several continuous pipelines as a whole. This means that if the route of one pipeline is changed, the route of the connected pipelines will also be adjusted accordingly. The behaviour here is basically that of plugged-in pipelines.



(1) Pipeline set consisting of 3 pipelines; (2) Point to be moved; (3) Target point; (4) Result

Some functions use the route change internally to adjust neighbouring parts, e.g. the Close gap, the Connect pipe, or the Delete functions for certain parts (e.g. Eccentric reducer). These functions now support positional changes beyond the limits of pipelines.

## Sealing gaskets

### Pressure ranges

In previous versions the nominal pressure was of no significance for the search of matching sealing gaskets. To take pressure ranges into account, the attribute **DRUCK\_MIN** (Minimum pressure) is now available.

When you now search a sealing gasket for a flange, the following, additional search condition will be generated from the nominal pressure **PN** of the flange:

**(MIN\_DRUCK ist unbelegt oder MIN\_DRUCK <= PN) und (DRUCK ist unbelegt oder DRUCK >= PN)**  
(Minimum pressure not specified or Minimum pressure <=PN) and (Minimum pressure not specified or Minimum pressure >=PN)

Sealing gaskets without pressure specifications will thus be handled as if they were suitable for any nominal pressure.

The standard parts that are by default supplied with HiCAD do not include sealing gaskets with a defined pressure range. Therefore, this new feature is currently only relevant for gaskets that have been created by the user. Accordingly, the attribute **DRUCK\_MIN** (Min. pressure) will not be available in the search masks that are by default supplied with HELIOS. If desired, you can add this attribute with the HELIOS Mask Editor.

If you prepare the database for Plant Engineering by clicking the corresponding button in DBPlantDataImport.exe, the attribute **DRUCK\_MIN** will be entered, with the designation **Minimum pressure**, into the database.

### An example from practice:

If you do not want gaskets with an own article number for each pressure level to be created, you can avoid this by means of the **DRUCK\_MIN** attribute.

Furthermore, you have now the option to narrow search results for sealing gaskets by specifying a value for the attribute **DRUCK** (Pressure), e.g. by including only gaskets with a defined pressure in your pipe class.

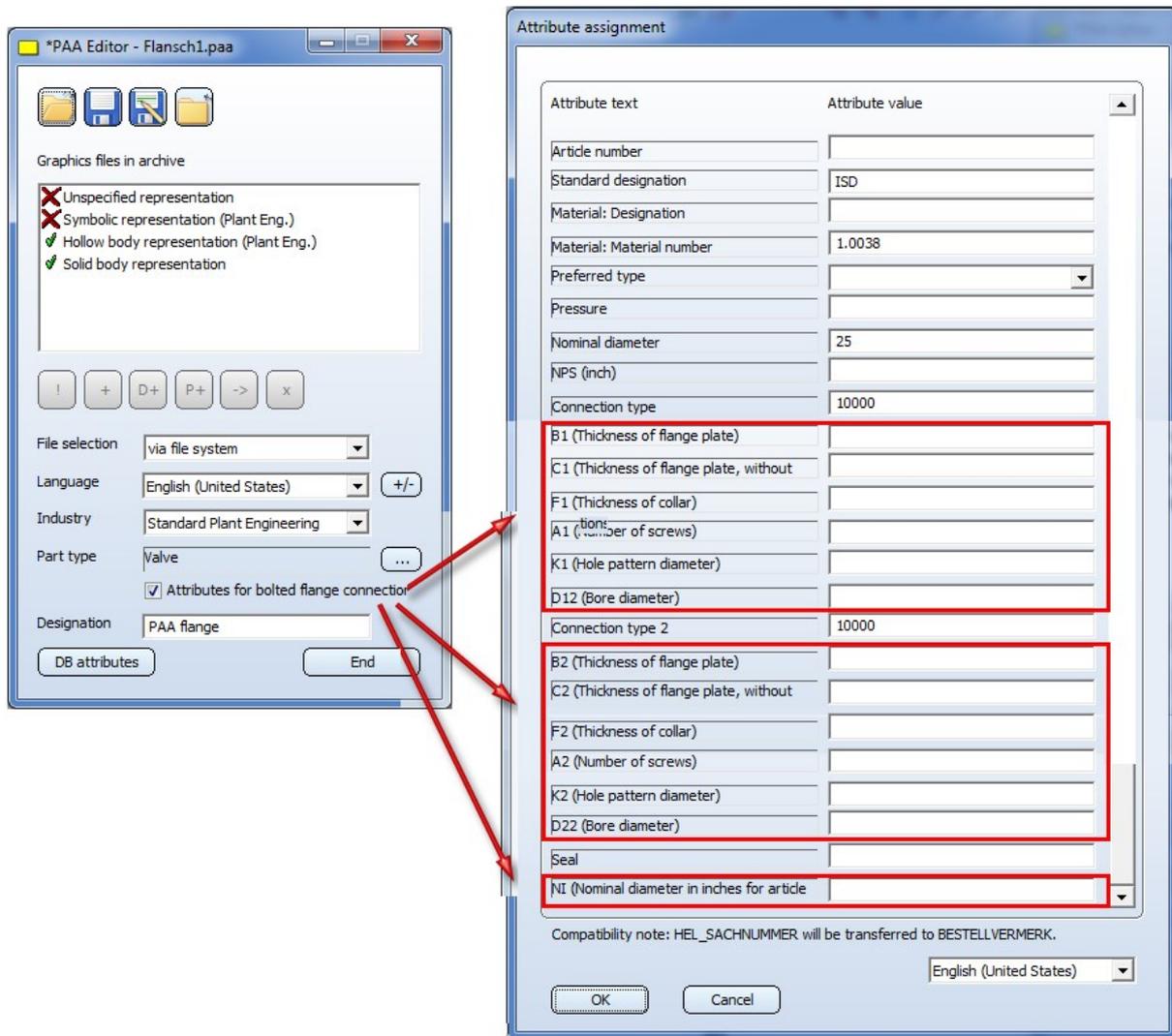
## PAA Editor

### Attributes for bolted flange connections

PAA files are part archives which are used for Plant Engineering parts with fixed geometries (that is, such parts will not just be parameterized when they are inserted). PAA files can be imagined as simplified variants without any subtypes. As such, PAA files support all what you already know from variants. In particular, you have the option to equip the connections of such parts with the connection IDs for Plant Engineering.

When you wanted to assign the parameters of a flange connection (Connection ID:20000) to a PAA file in previous versions, there were limitations to the automatic functions for bolted flange connections. These could only be applied successfully if the flange connections were defined as sub-parts referring to an own part archive (e.g. via HELIOS link). For instance, this can be seen by the example vessels (BEHALB...) that are supplied with HiCAD.

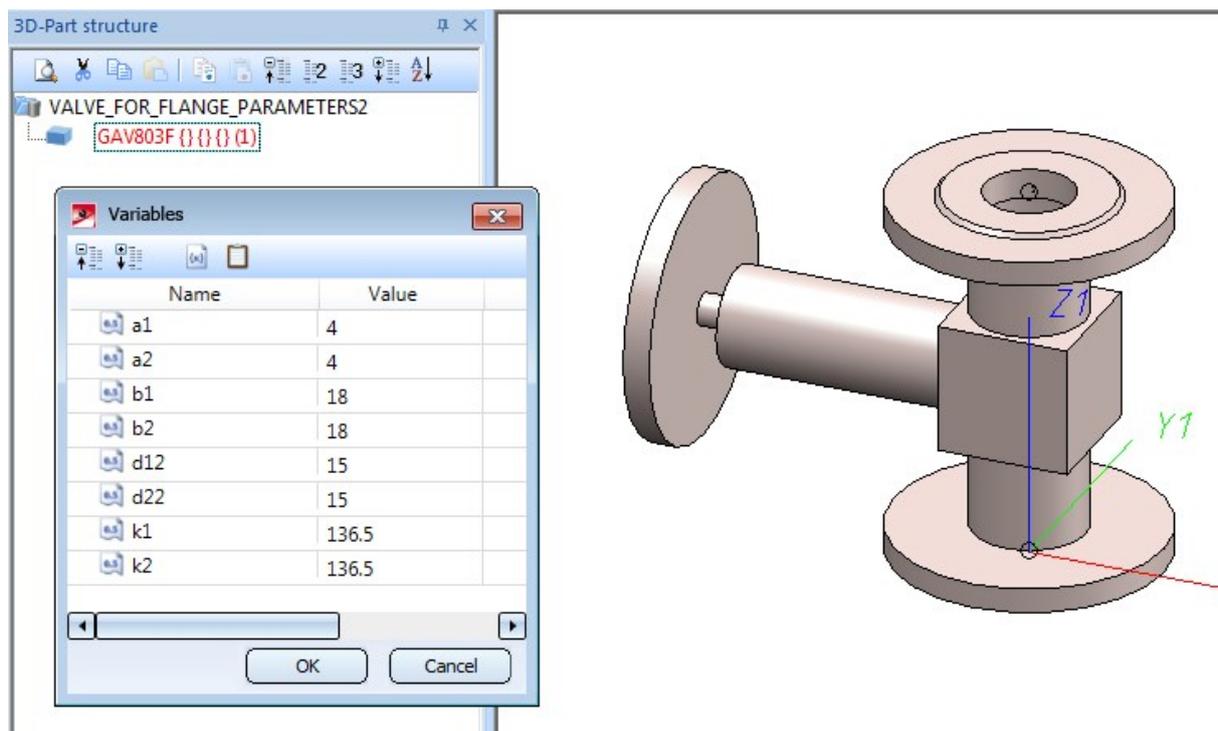
From HiCAD 2018 SP2 onwards, the PAA Editor allows you to store the parameters of a flange connection (Connection ID: 20000) directly in a PAA file. For this purpose, the **Attributes for bolted flange connections** checkbox has been added in the dialogue window of the PAA Editor. If this checkbox is active, the attributes for bolted flange connections will be added to the attributes list that is displayed when you click on the **DB attributes** button.



Connection-specific attributes (x stands for the connecting point number) for bolted flange connections are:

- Bx Thickness of flange plate
- Cx Thickness of flange plate, without collar
- Fx Thickness of collar
- Ax Number of bolts
- Kx Hole pattern diameter
- Dx2 Bore diameter
- NI Nominal diameter in inches for article master (applies to entire bolted flange connection)
- L Thickness of flange plate  
This attribute is only available for blank flanges and overwrites the attribute B1.

You can also specify the attribute values directly in HiCAD while constructing a part:



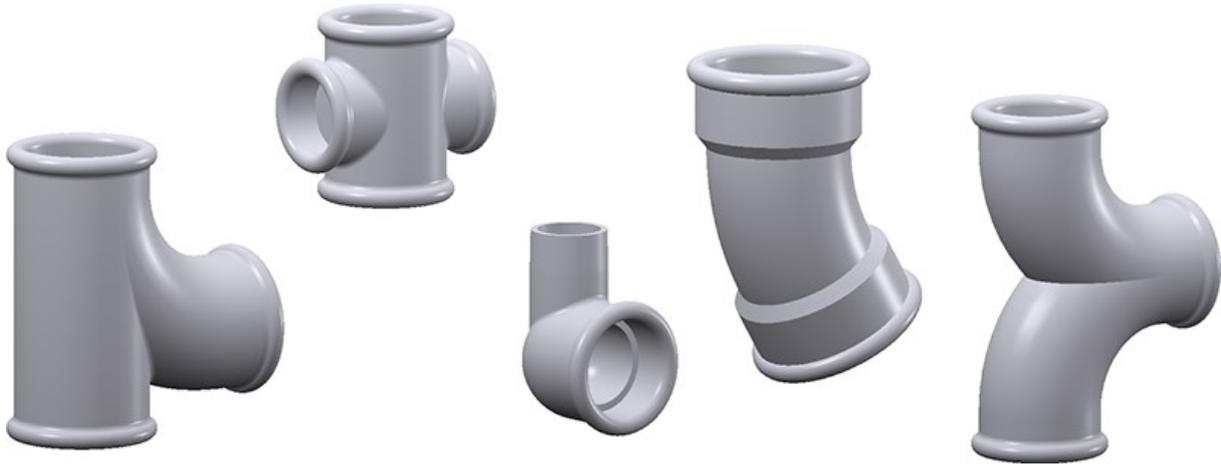
In this case the PAA Editor will take the values directly over into the list of attributes, so that you do not need to enter them again.

Please note however that the PAA Editor cannot evaluate any formulas!

## New standard parts

### Malleable cast iron fittings acc. to EN 10242

Further malleable cast iron fittings acc. to EN 10242 by the company have been added to the standard parts catalogue.



The table below contains all parts of this standard. The **New** column indicates whether a part is new:

Elbows	New	VAA file
+GF+ Elbow S 1	no	GF10242_S01.vaa
+GF+ Elbow S 1A	no	GF10242_S01A.vaa
+GF+ Elbow S 2	no	GF10242_S02.vaa
+GF+ Elbow S 2A	no	GF10242_S02A.vaa
+GF+ Elbow S 3	no	GF10242_S03.vaa
+GF+ Elbow S 40	no	GF10242_S40.vaa
+GF+ Elbow S 41	no	GF10242_S41.vaa
+GF+ Elbow S 50	yes	GF10242_S50.vaa
+GF+ Elbow S 51	yes	GF10242_S51.vaa
+GF+ Elbow S 53	yes	GF10242_S53.vaa
+GF+ Elbow S 54	yes	GF10242_S54.vaa
+GF+ Double elbow S 60	no	GF10242_S60.vaa
Crossovers	New	VAA file
+GF+ CrossoverS 85	yes	GF10242_S85.vaa
Crossover tees	New	VAA file
+GF+ Crossover tee S87	yes	GF10242_S87.vaa
Knee	New	VAA file
+GF+ Angle S 90	no	GF10242_S90.vaa

+GF+ Angle S 90 R	yes	GF10242_S90R.vaa
+GF+ Angle S 92	no	GF10242_S92.vaa
+GF+ Angle S 92 R	yes	GF10242_S92R.vaa
+GF+ Angle S 94	no	GF10242_S94.vaa
+GF+ Union elbow S95, flat seat	yes	GF10242_S95.vaa
+GF+ Union elbow S 96	yes	GF10242_S96.vaa
+GF+ Union elbow S 97	yes	GF10242_S97.vaa
+GF+ Union elbow S 98	yes	GF10242_S98.vaa
+GF+ Union elbow S100	yes	GF10242_S100.vaa
+GF+ Union elbow S101	yes	GF10242_S101.vaa
+GF+ Union elbow S102	yes	GF10242_S102.vaa
+GF+ Union elbow S103	yes	GF10242_S103.vaa
+GF+ Angle 45° S120	yes	GF10242_S120.vaa
+GF+ Angle 45° S121	yes	GF10242_S121.vaa
<b>T-pieces</b>	<b>New</b>	<b>VAA file</b>
+GF+ T-piece S 130	no	GF10242_S130.vaa
+GF+ Pitcher tee S130R	yes	GF10242_S130R.vaa
+GF+ Pitcher tee S131	yes	GF10242_S131.vaa
+GF+ Pitcher tee, reduced S131R	yes	GF10242_S131R.vaa
+GF+ Twin elbow S132	yes	GF10242_S132.vaa
+GF+ T-piece S133	yes	GF10242_S133.vaa
+GF+ T-piece S134	yes	GF10242_S134.vaa
+GF+ T-piece, reduced S134R	yes	GF10242_S134R.vaa
+GF+ T-piece, equal, S 135	yes	GF10242_S135.vaa
+GF+ Pitcher tee for distribution battery, reduced S137	yes	GF10242_S137.vaa
<b>Branches</b>	<b>New</b>	<b>VAAfile</b>
+GF+ T-piece 45° S 165	no	GF10242_S165.vaa
<b>Kreuze</b>	<b>New</b>	<b>VAA file</b>
+GF+ Cross S180	yes	GF10242_S180.vaa
+GF+ Cross, reduced S180R	yes	GF10242_S180R.vaa
<b>Y-pipes</b>	<b>New</b>	<b>VAA file</b>
+GF+ Y-piece S 220	yes	GF10242_S220.vaa
<b>Reducers</b>	<b>New</b>	<b>VAA file</b>
+GF+ Socket, reduced S 240	no	GF10242_S240.vaa

+GF+ Reduced nipple S 241	yes	GF10242_S241.vaa
+GF+ Double nipple, red. S 245	no	GF10242_S245.vaa
+GF+ Socket, reduced S 246 EN 10242	no	GF10242_S246.vaa
+GF+ Reducer, eccentric S260	yes	GF10242_S260.vaa
<b>Sockets</b>	<b>New</b>	<b>VAA file</b>
+GF+ Socket S 270	no	GF10242_S270.vaa
+GF+ Socket S 271	yes	GF10242_S271.vaa
<b>Nipples</b>	<b>New</b>	<b>VAA file</b>
+GF+ Double nipple S 280	no	GF10242_S280.vaa
+GF+ Double nipple S 281	yes	GF10242_S281.vaa
<b>Caps</b>	<b>New</b>	<b>VAA file</b>
+GF+ Plug, beaded S 290	yes	GF10242_S290.vaa
+GF+ Plug, plain S 291	yes	GF10242_S291.vaa
+GF+ Plug, with hole S 294	yes	GF10242_S294.vaa
+GF+ Cap S 300	yes	GF10242_S300.vaa

- Variant GF10242\_S92R.vaa replaces Variant GF10242\_S92R1.vaa.
- Variant GF10242\_S130R.vaa replaces Variant GF10242\_S130R23.vaa.
- The file GF10242.lst for part data synchronization lists all above parts - old and new.

## Service Pack 1 2018 (V 2301)

### Supplied database

#### List of variants

New in the HiCAD PlantParts directory is the file **starter.lst**. This file contains a list of all VAA and PAA files which are contained in an unmodified, supplied database.

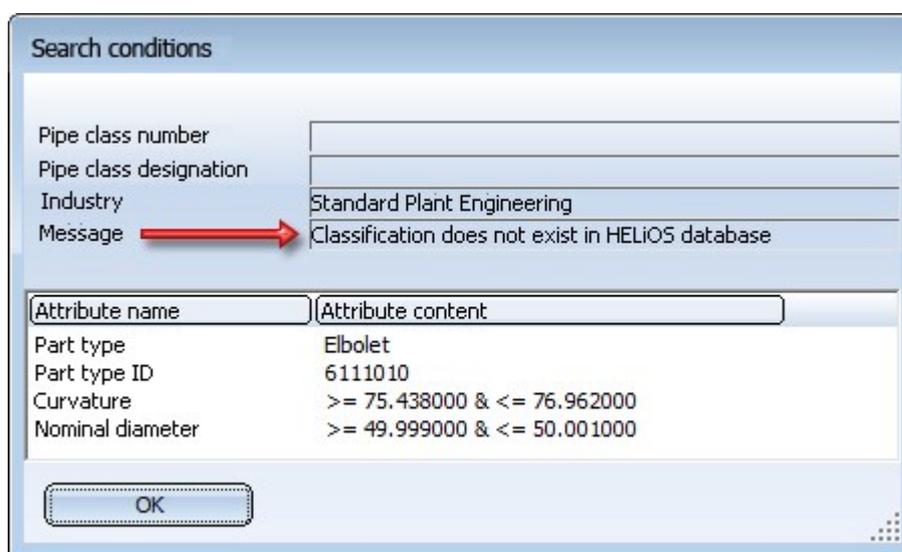
This facilitates an updating of the associated article and document masters and allows an clearing of the database with the help of PartDataAutoSync.exe or DBPlantDataImport.exe.

### Part search

#### Messages in the Search conditions dialogue window

If the **Show search criteria if part search unsuccessful** checkbox on the **Part search** tab of the Plant Engineering Settings dialogue window has been activated, the **Search conditions** dialogue window with the utilized search criteria will be displayed after an unsuccessful or disrupted part search.

If parts are managed via HELiOS an additional row shows error messages of the HELiOS database that may have occurred.



These messages can be as follows:

- You have not been logged into the HELiOS database
- Classification does not exist in HELiOS database
- The link AllgType-Variante does not exist in the HELiOS database
- The link Part-Part geometry does not exist in the HELiOS database
- No document for article found

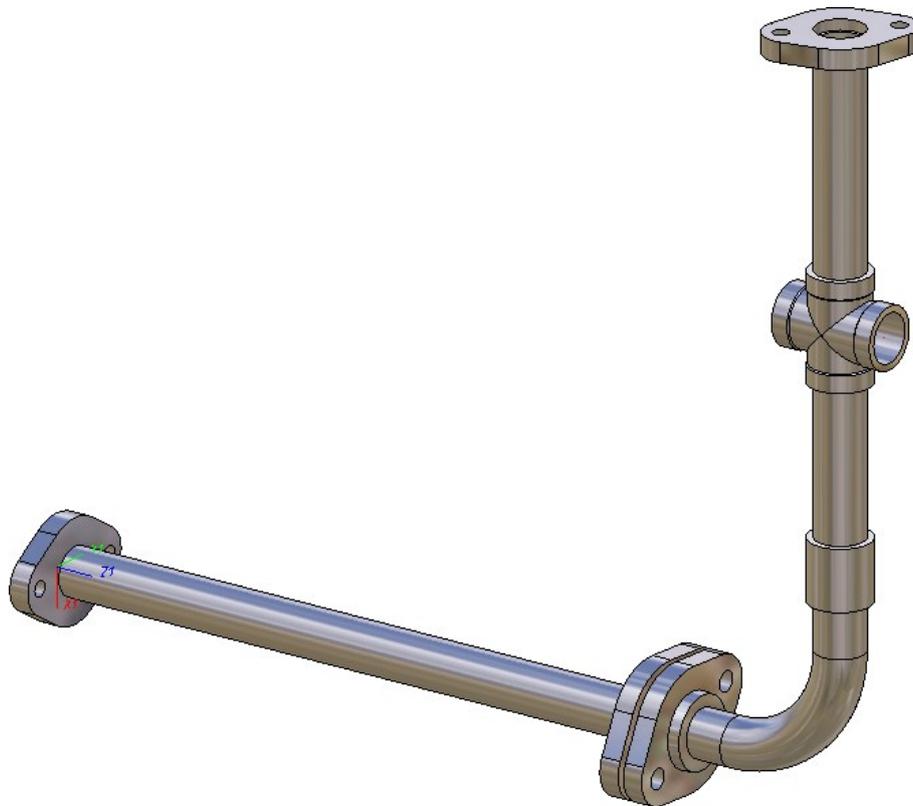
## New parts

### Threaded flanges, pipes and fittings

Variants according to

- DIN EN10255,
- DIN EN 10241,
- DIN EN1092-1/13 and
- DIN 2561 und DIN 2558

have been added to the Plant Engineering Standard Parts. All those parts have threads.



<b>Straight pipes acc. to DIN EN10255, light, medium weight and heavy</b>	<b>Variant</b>
	EN10255-H.VAA EN10255-M.VAA EN10255-L.VAA EN10255-L1.VAA EN10255-L2.VAA
<b>Threaded fittings</b>	<b>Variant</b>
Hexagon cap	EN10241_CAP.VAA
Cross	EN10241_CROSS.VAA
Elbow 90°	EN10241_ELBOW90.VAA
Angle 45°	EN10241_KNEE45.VAA
Angle 90°	EN10241_KNEE90.VAA
Angle 90° with inner and outer thread	EN10241_KNEE90_MF.VAA
Pipe nipple	EN10241_NIPPLE.VAA
Screwed connection	EN10241_NIPPLE_HEXAGON.VAA
Double pipe nipple	EN10241_NIPPLE_LONG.VAA
Weld-on nipple	EN10241_NIPPLE_WELD.VAA
Plug	EN10241_PLUG.VAA
Reducer	EN10241_RED.VAA
Hexagon reducer double nipple	EN10241_RED_NIPPLE.VAA
Reducer socket	EN10241_RED_SOCKET.VAA
Socket	EN10241_SOCKET.VAA
Hexagon double nipple	EN10241_SOCKET_HEXAGON.VAA
Half socket	EN10241_SOCKET_SHORT.VAA
T-piece	EN10241_TEE.VAA
<b>Threaded flanges, oval</b>	<b>Variant</b>
	N2561.VAA N2558.VAA
<b>Threaded flanges, round</b>	<b>Variant</b>
	EN1092-1-13-B-PN10.VAA EN1092-1-13-B-PN100.VAA EN1092-1-13-B-PN16.VAA EN1092-1-13-B-PN25.VAA EN1092-1-13-B-PN40.VAA EN1092-1-13-B-PN6.VAA EN1092-1-13-B-PN63.VAA

For part synchronizations you can find those parts in the file **threaded.lst**.

## Part Tools

### Delete parts

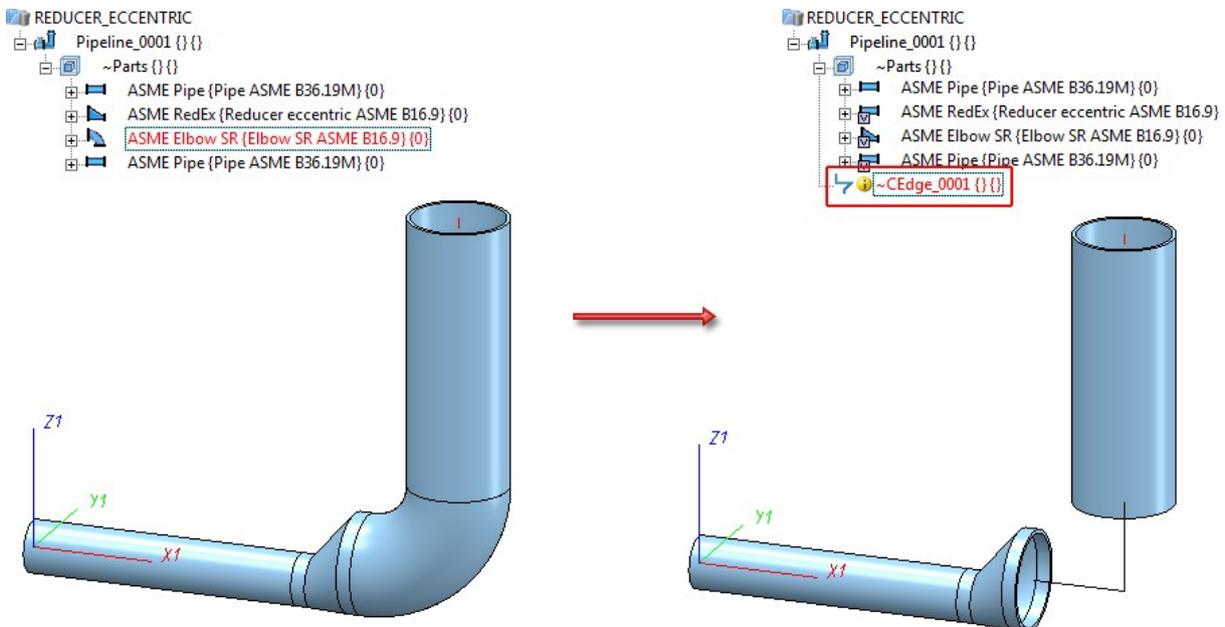
Sometimes it is easier to place parts directly instead of drawing a guideline that exactly matches the part to be placed, and then place the part onto it. When you deleted parts that have been inserted in this (easier) way in older versions, the information about the pipeline route was lost due to the missing guideline.

Therefore, the functions

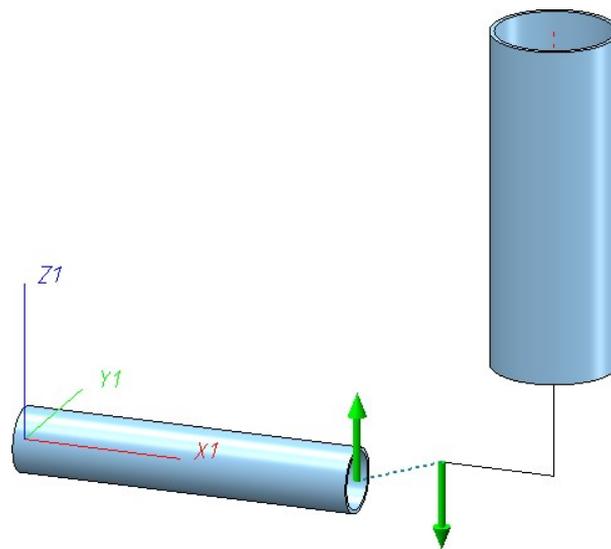
Delete part  and

Delete several parts 

have been enhanced. The functions now create "replacement guidelines" for parts that have been placed without guidelines, as shown in the example with the elbow below:

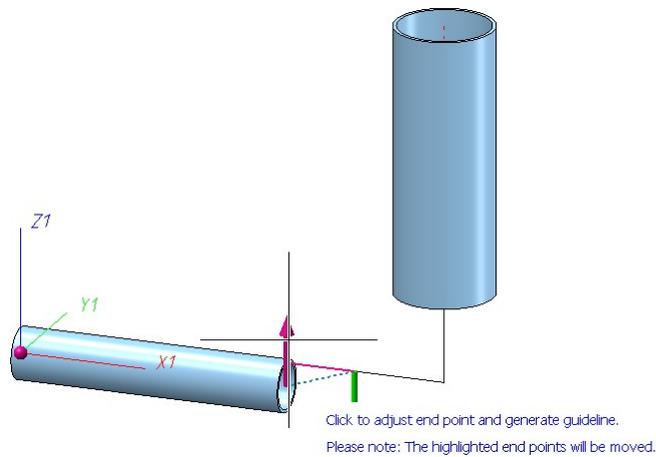


Please note that parts whose guidelines do not match their connection direction will be handled in a special way. In the example shown below this is the asymmetrical reducer. If you delete such a part, you have the option to adjust connecting points. In the model drawing the adjustment will be indicated by means of arrow symbols. The arrows show in which direction a connecting point will be moved for this purpose.



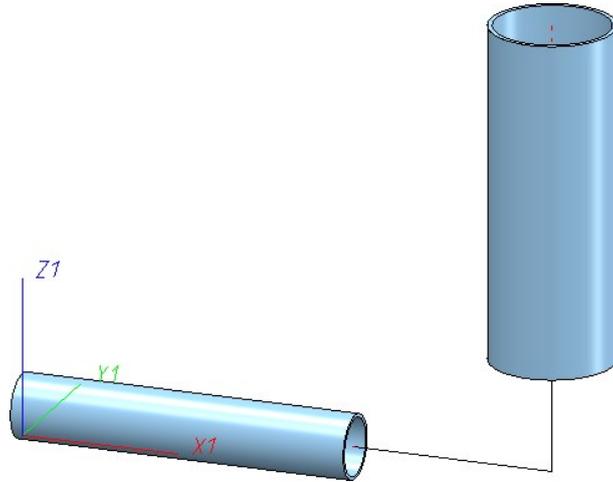
The arrows are selectable. As soon as you move the cursor over an arrow you receive an information about the expected result of a mouse click.

For instance, if you move the cursor over the left arrow in the drawing shown below, a line highlighted in a different colour (Special colour Marking 1) appears, showing you the route of the guideline. Also, an info text will be displayed, telling you that end points will be moved. In the example below, this text refers to the left end point of the horizontal pipe, which is also highlighted in the marking colour.

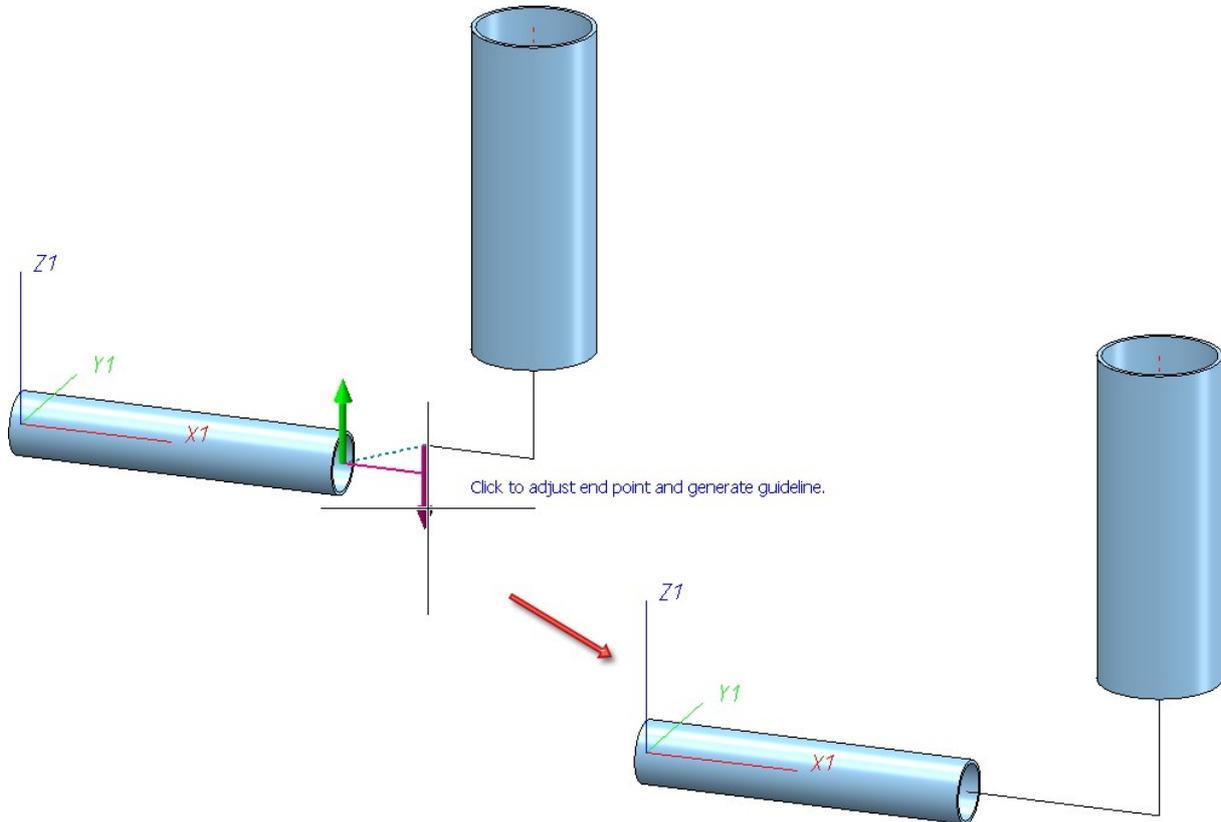


If this info text appears, you should check whether you actually agree with the displacement of the marked points. Please note that such points can also lie outside the border of visible areas.

When you now click on the left arrow, the result will look as follows:



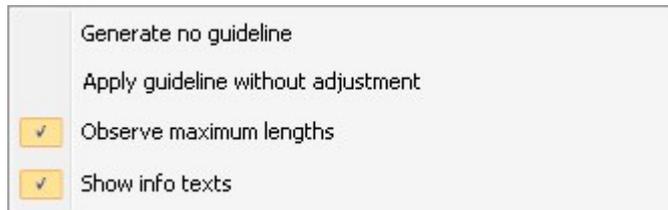
If you select the right arrow instead of the left arrow, the result will look as follows:



This time, no end point was moved; therefore, no info text appeared. instead, the vertical guideline section was lengthened a little bit.

In the background, a route change will be used for the adjustment of the end points. This means that HiCAD will always attempt to move the connecting point by length changes first. If this attempt fails, pipeline end points will also be changed and the info text appears.

For parts that need to be handled in a special way - such as the asymmetrical reducer in the examples below - you can use additional functions in a context menu that you can open with a right-click:

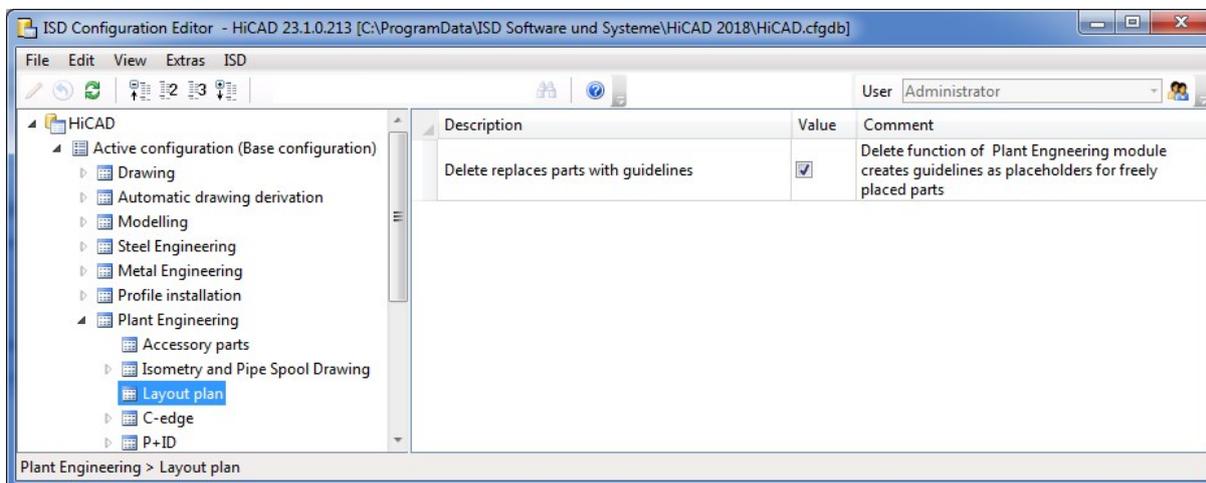


Function	
Generate no guideline	No guideline will be generated, the pipeline remains disconnected. (this can be preset in the Configuration Editor at <b>Plant Engineering &gt; Layout plan</b> with the parameter <b>Delete replaces parts with guidelines</b> .)
Apply guideline without adjustment	A guideline will be created along the dashed blue line. This will usually cause inflexions in the route.
Observe maximum lengths	A route change option that determines whether the maximum lengths of the pipes are to be observed when adjusting the connecting points by means of length changes.
Show info texts	Determines whether info texts are to be shown when you move the cursor over an arrow.

When deleting several parts, one also distinguishes between parts that have been placed on a guideline and parts that have been placed directly, i.e. without the help of a guideline. If the part selection contains parts that require a special handling with regard to the Delete part functions, these parts will be excluded from deletion. In this case a corresponding info text will be displayed. For instance, this is the case if the direction of an automatically created guideline does not match the connection direction.

## Deactivation of guideline creation

The guideline creation can also be switched off if desired. To do this, open the Configuration Editor, go to **Plant Engineering > Layout plan** and deactivate the **Delete replaces parts with guidelines** checkbox. Please note however that if you delete parts that were not inserted on a guideline, the information on the pipeline routing will be lost due to the missing guideline.



## Bolted flange connections

For bolted flange connections in Plant Engineering you have the option to reuse connections that have already been saved during insertion. For this to happen, the **Use saved bolted flange connection** checkbox on the **Flange connection, bolted** tab of the **Plant Engineering Settings** dialogue window must have been activated. This is the ISD default setting.

In this mode HiCAD 2018 SP1 will proceed as follows:

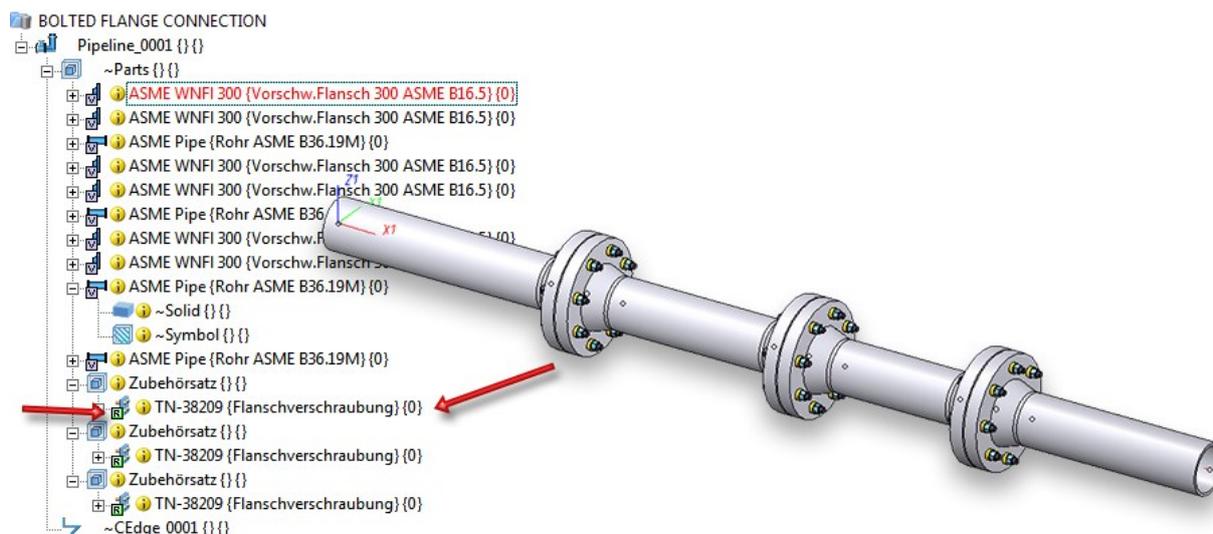
Once used, a bolted flange connection will be saved **as referenced KRA file**, namely, to the HiCAD subdirectory **RefTeile1**. In the HELiOS database, a document master and an article master will be automatically created for this KRA file, which will be classified as a Bolted flange connection. In the article attributes, the following data will be saved:

- Nominal diameter of flange,
- Number of bolts,
- Bolt hole pattern diameter,
- Diameter of bolt holes,
- Clamping length (without washers),
- Level of detail (i.e. compressed, without details or with all details), and
- Standard designation defined as search pattern in the settings.

Bolted flange connections matching these attributes will be regarded as "identical parts" with regard to referencing and updating.

When inserting a new bolted flange connection, HiCAD will use these attributes to check whether a matching article and document master exists in HELiOS. If this is the case, the matching bolted flange connection (KRA) will be loaded from the **RefTeile1** directory and inserted as a referenced element. If this is not the case, a new bolted flange connection will be generated and saved as a referenced element.

Due to the referencing, subsequent changes will now affect all identical bolted flange connections (=identical parts).



Please note that this changed mode also affects processings applied to bolted flange connections.

### Modifications to only one bolted connection

If you want to carry out modifications that are to apply to only one bolted connection, you can break up the referencing beforehand. To do this, use the context menu (right-click) of the referenced bolted connection in the ICN. When you are asked whether you want to delete the article master links, it is recommended to answer with Yes, since the stored article master will probably no longer match the part.

### Updating of existing flange connections from older versions

When you load a model drawing that has been created with a HiCAD Version before 2018 SP1, the following adjustments will be carried out automatically - irrespective of whether you have already worked with saved bolted flange connections or not:

- The origin of the local coordinate system will be placed in front of the first flange face (seen from the bolt heads). The origin was formerly located between the flange faces.
- Formerly, bolted flange connections had 3 named points. These will be removed.

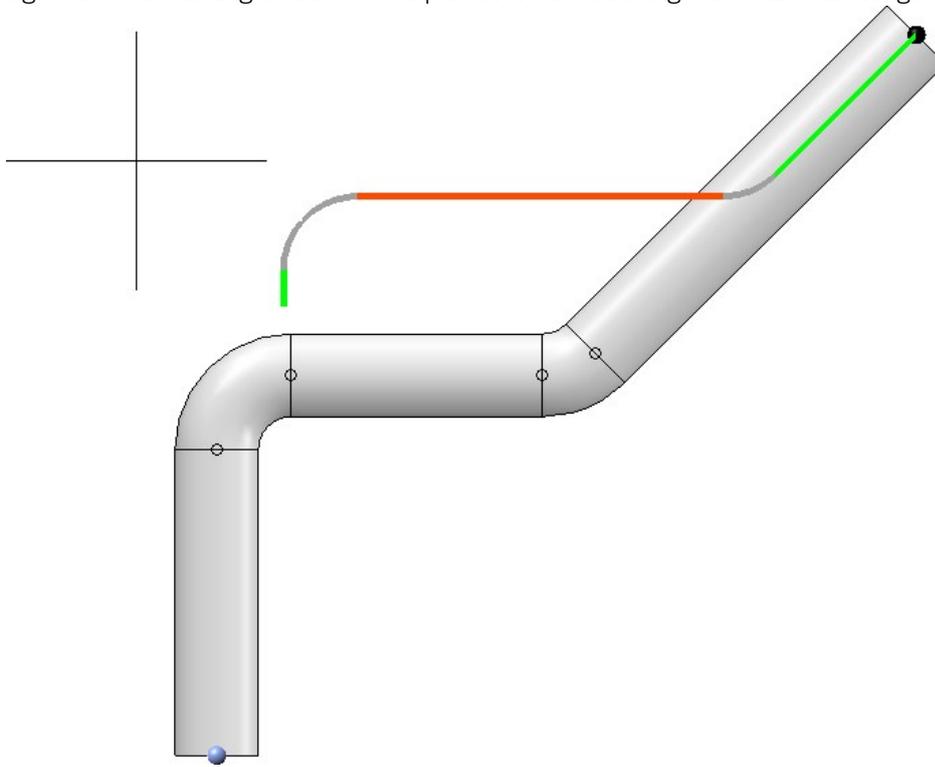
If bolted flange connections referring to an article master exist in the model drawing, it is assumed that these bolted connections were inserted while the **Use saved bolted flange connections** option was activated. In this case, the following additional actions will be performed:

- The KRA file for the bolted connection in the PlantParts directory will be deleted.
- The bolted connection will be saved as a referenced element. In the process, a new KRA file will be created in the RefTeile1 directory. If a matching KRA file already existed in the KRA directory, the bolted connection will only be marked as a reference to it and updated to the state of the existing KRA file.
- The document master linked to the bolted connection will be adjusted, so that HEL\_DATEINAME now points to the KRA file in the RefTeile1 directory.

## Pipeline Tools

### Change route - Highlighting of pipelines when maximum length is reached

If the **Observe maximum lengths** mode has been activated when you use the Change route function, pipes will be highlighted in the marking colour 5 in the preview when reaching their maximum length.

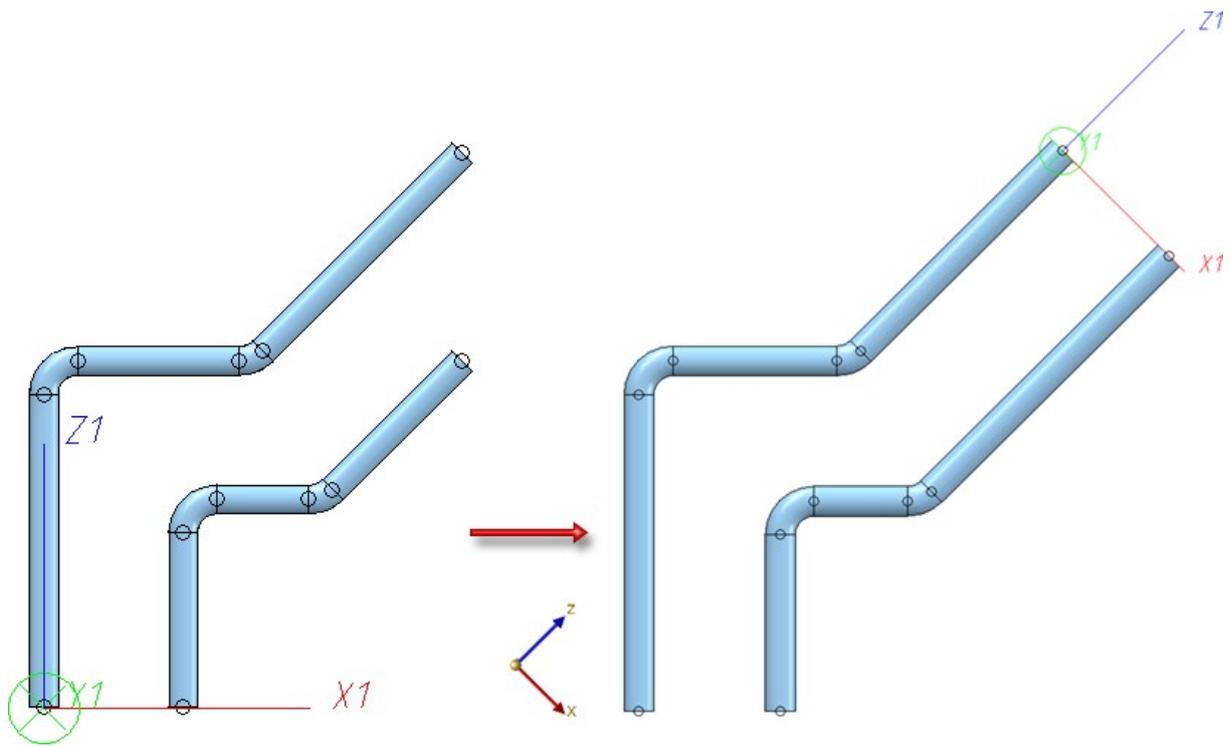


### Change route - Distance calculation according to L1 norm

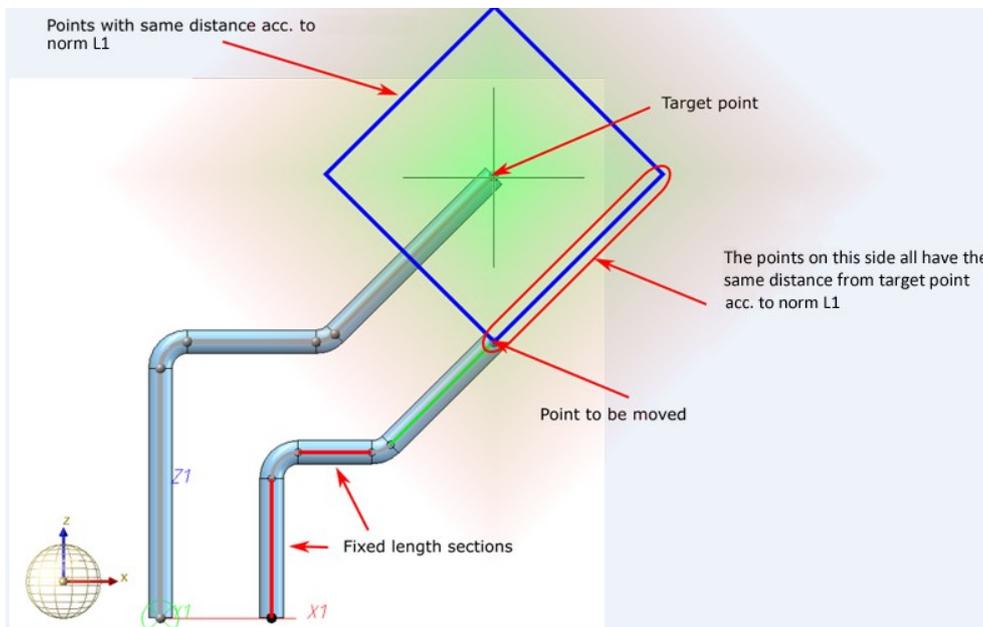
When changing pipeline routes, the minimum distance calculation takes place in accordance with the L1 norm. The effect of this is that, starting from the target point, the directions running parallel to the coordinate axes will be preferred if the target point itself cannot be reached. As of HiCAD 2018 SP1, the local coordinate system will be considered for the calculation.

The effect of the coordinate system on distance calculation can be explained by means of the following example:

Let us assume that you want to perform the change shown below.

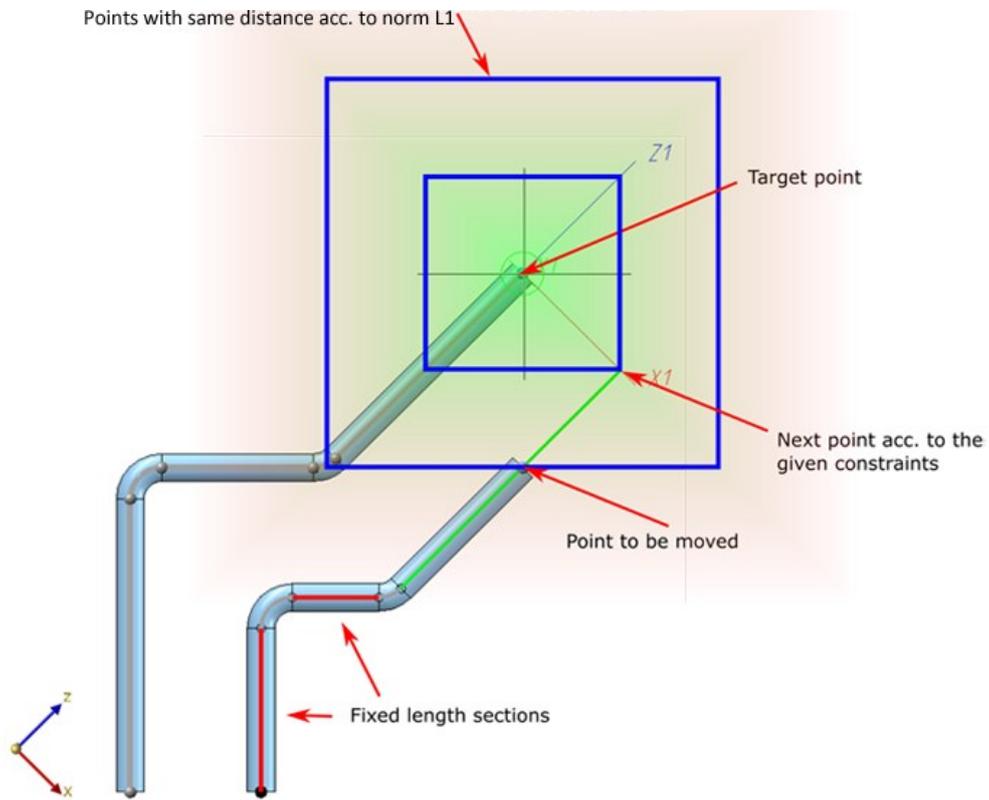


To do this, fix the horizontal and the vertical sections and move the graphic cursor to the target point. If you try to change the route in the World CS in this way, you will realize that this will not work. The reason for this is the above mentioned distance calculation.



But if you switch to a different coordinate system, e.g. with the function **Plant Engineering > Settings > Settings >**

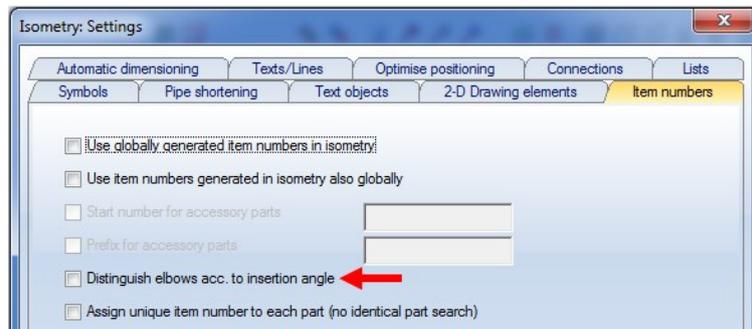
**> Z points outwards** , the change will be successful.



## Isometry / Pipe Spool Drawing

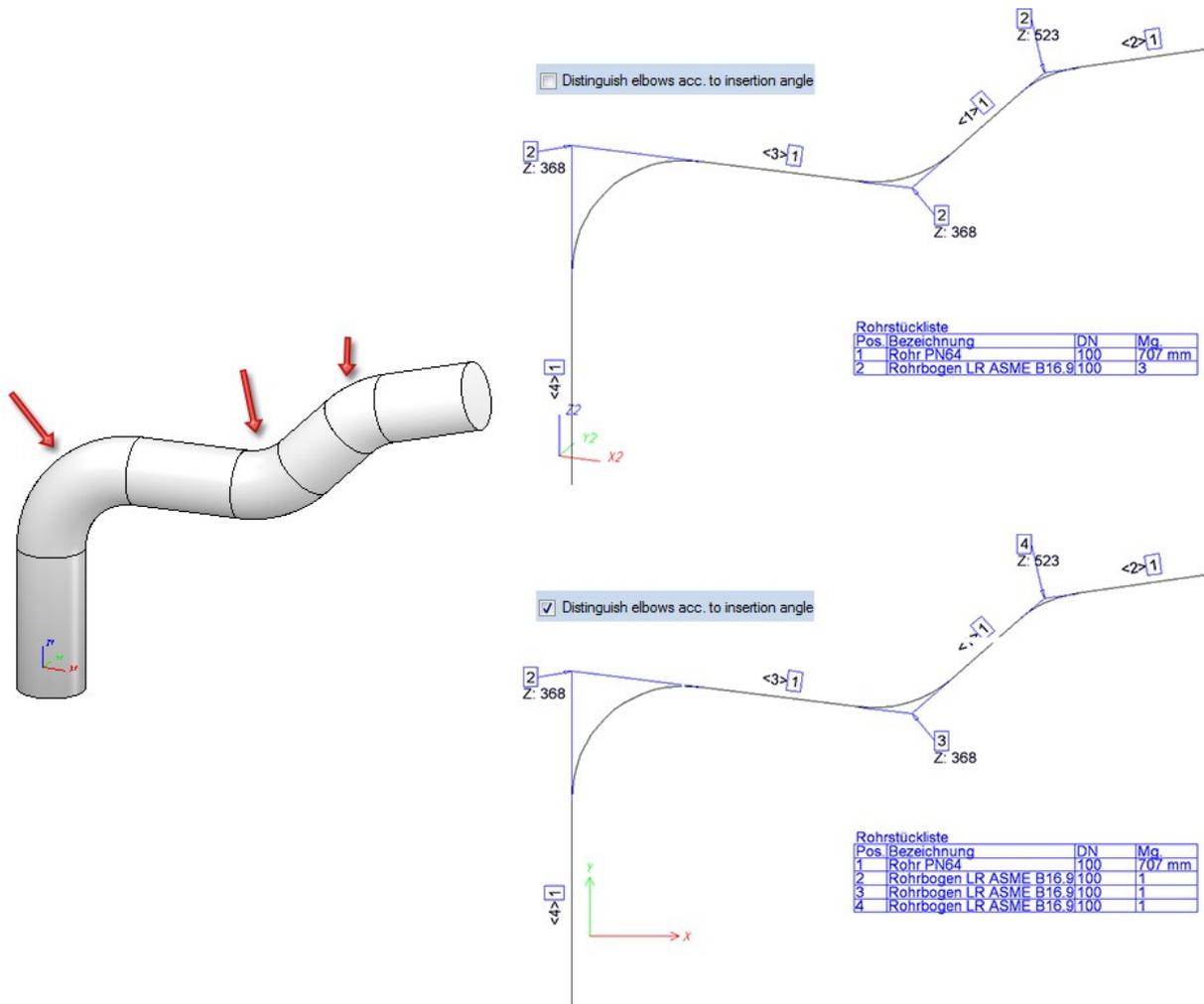
### Settings - Item numbers

Formerly, elbows in Plant Engineering BOMs were not distinguished by their insertion angles. This can now be changed in the settings for isometries and pipe spool drawings. For this purpose, the new **Distinguish elbows acc. to insertion angle** checkbox is available on the **Item numbers** tab of the settings dialogues.



### Example:

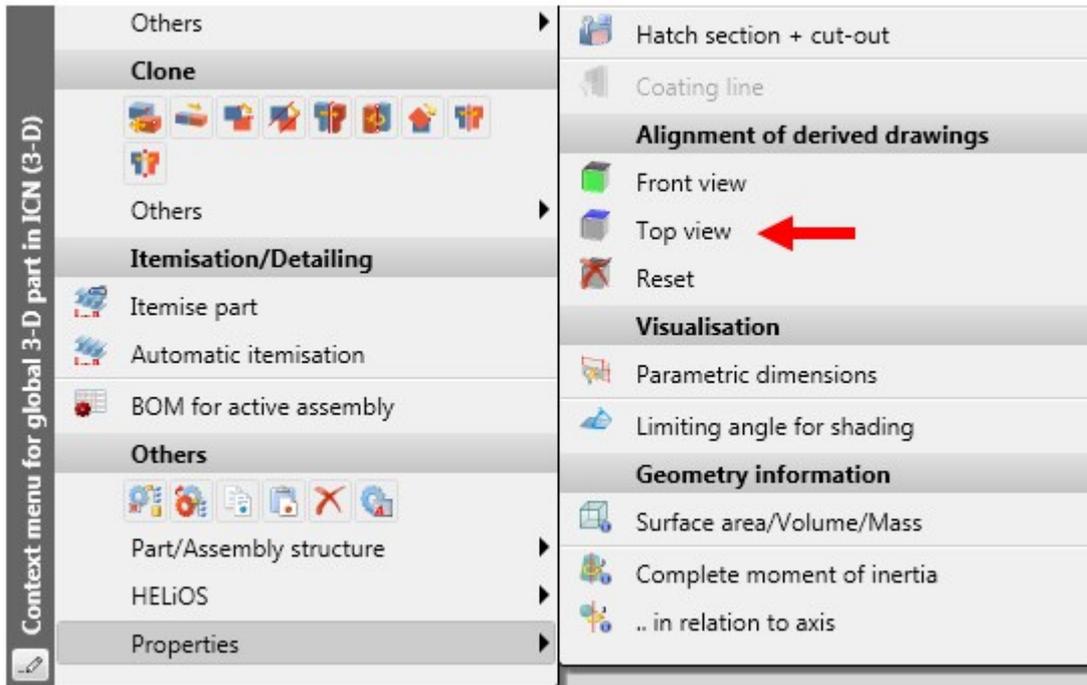
The pipeline shown below contains 3 elbows with different insertion angles.



### Taking into account of the functions beneath "Alignment of derived drawings"

The functions beneath **Alignment of derived drawings** in the context menu of assemblies allow you to determine whether the active view is to be interpreted as  **Front view** or  **Top view** of derived drawings. This means that a View CS will be defined for the front view or top view of the corresponding assembly.

These functions can also be applied to pipelines. In the ICN, right-click on the pipeline, open the **Properties** sub-menu and choose the desired function.



Isometry and pipe spool drawing will then consider the view coordinate system that was defined in this way:

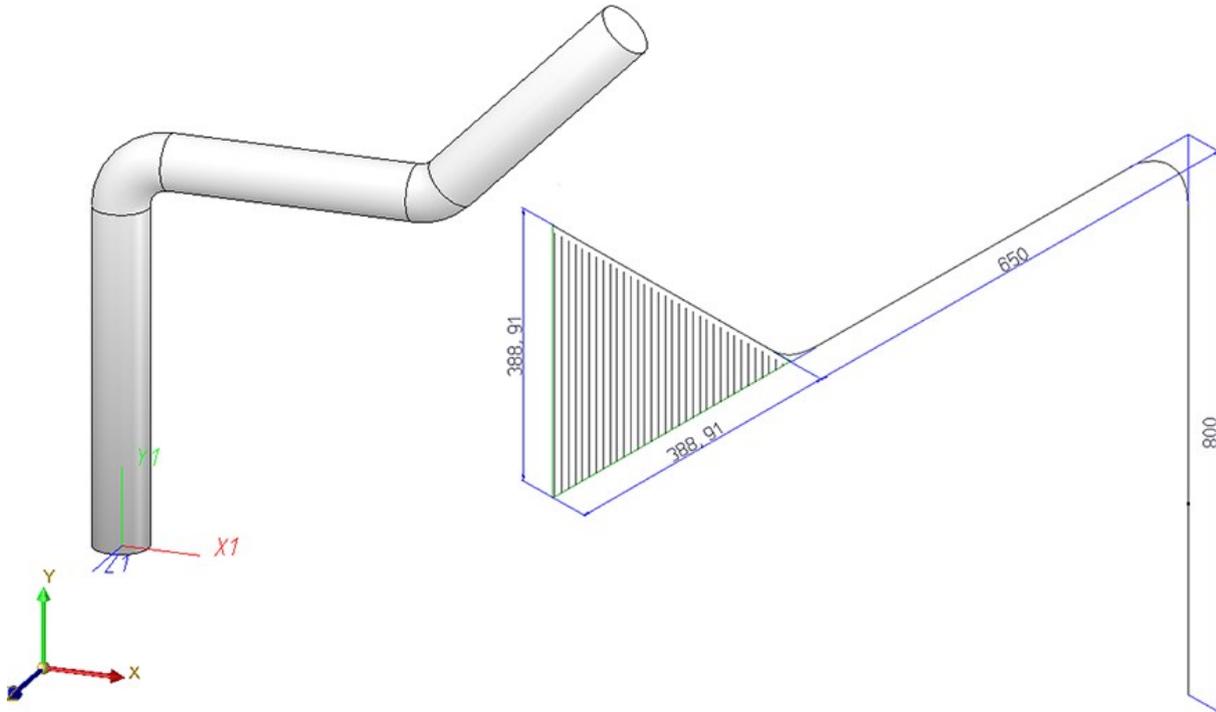
- In the pipe spool drawing, all views will be created with regard to the defined view coordinate system.
- This also applies to the isometry - with the exception of the **Retain perspective** projection. It serves the purpose of creating an isometry in the view coordinate system of the currently active view; therefore the selected alignment has no effect here.
- The **Set reference CS for isometry**  function that determines in which coordinate system the coordinates in dimension tags are to be calculated remains unaffected by this.

To **Reset**  the selected alignment, choose the same -named function in the context menu of the pipeline.

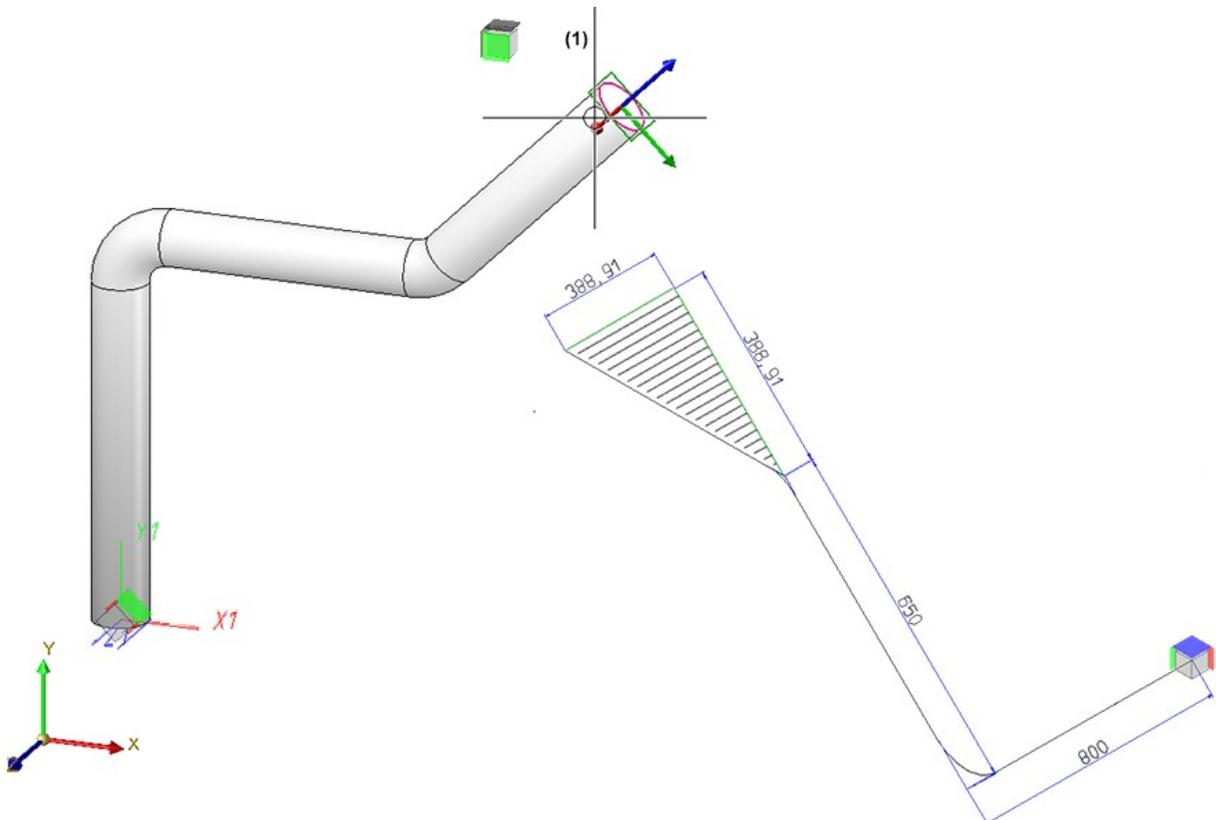
The alignment chosen for a pipeline can be marked accordingly in the model drawing.

**Example:**

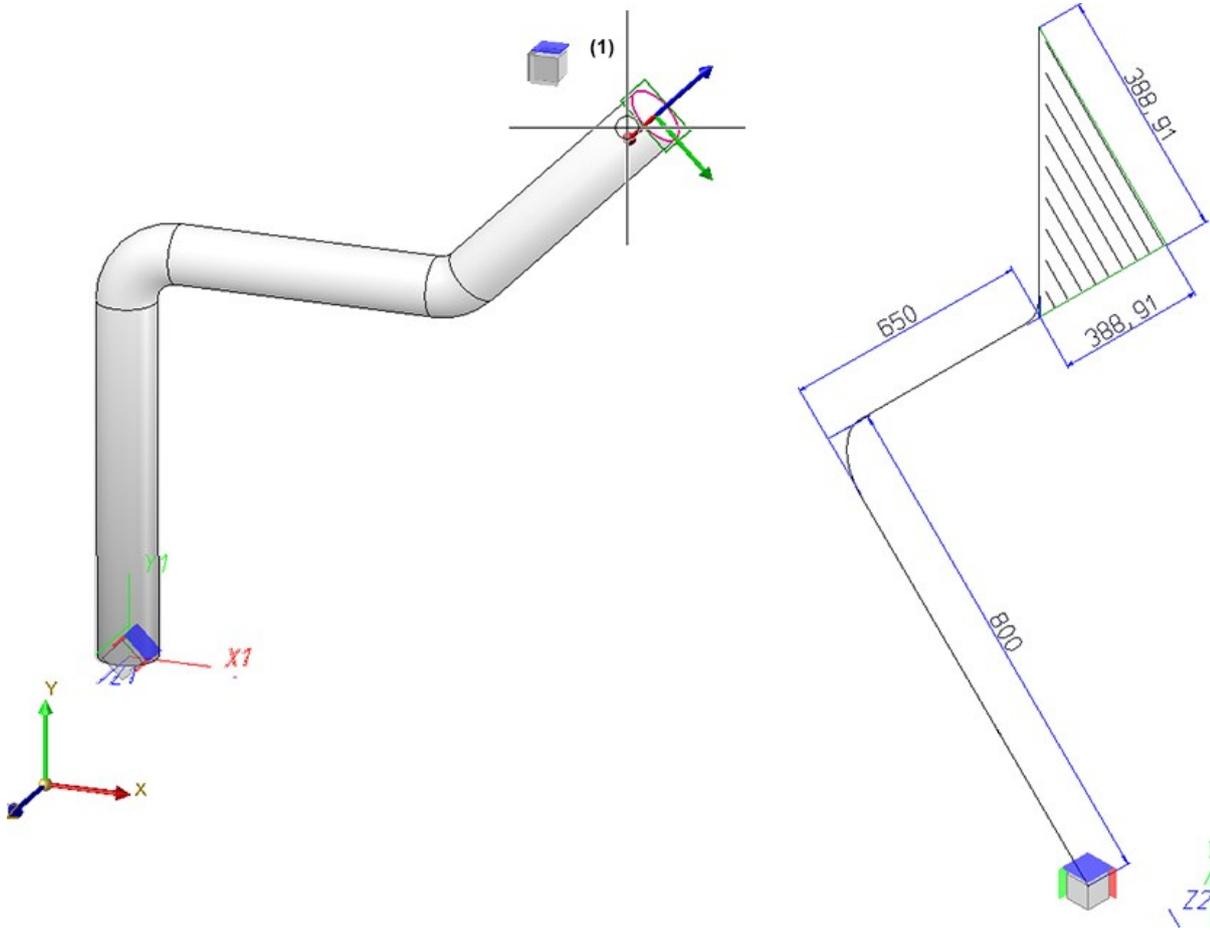
The image below shows a pipeline for which no view alignment has been set. For the isometry, the Isometric projection 1 has been chosen.



If you now open the context menu with a right-click, choose **Properties > Alignment of derived drawings > Front view**  , and select the marked surface (1) in the image below, this will affect the isometry:



In the next image, **Properties > Alignment of derived drawings > Top view**  has been chosen, and then the marked surface (1):

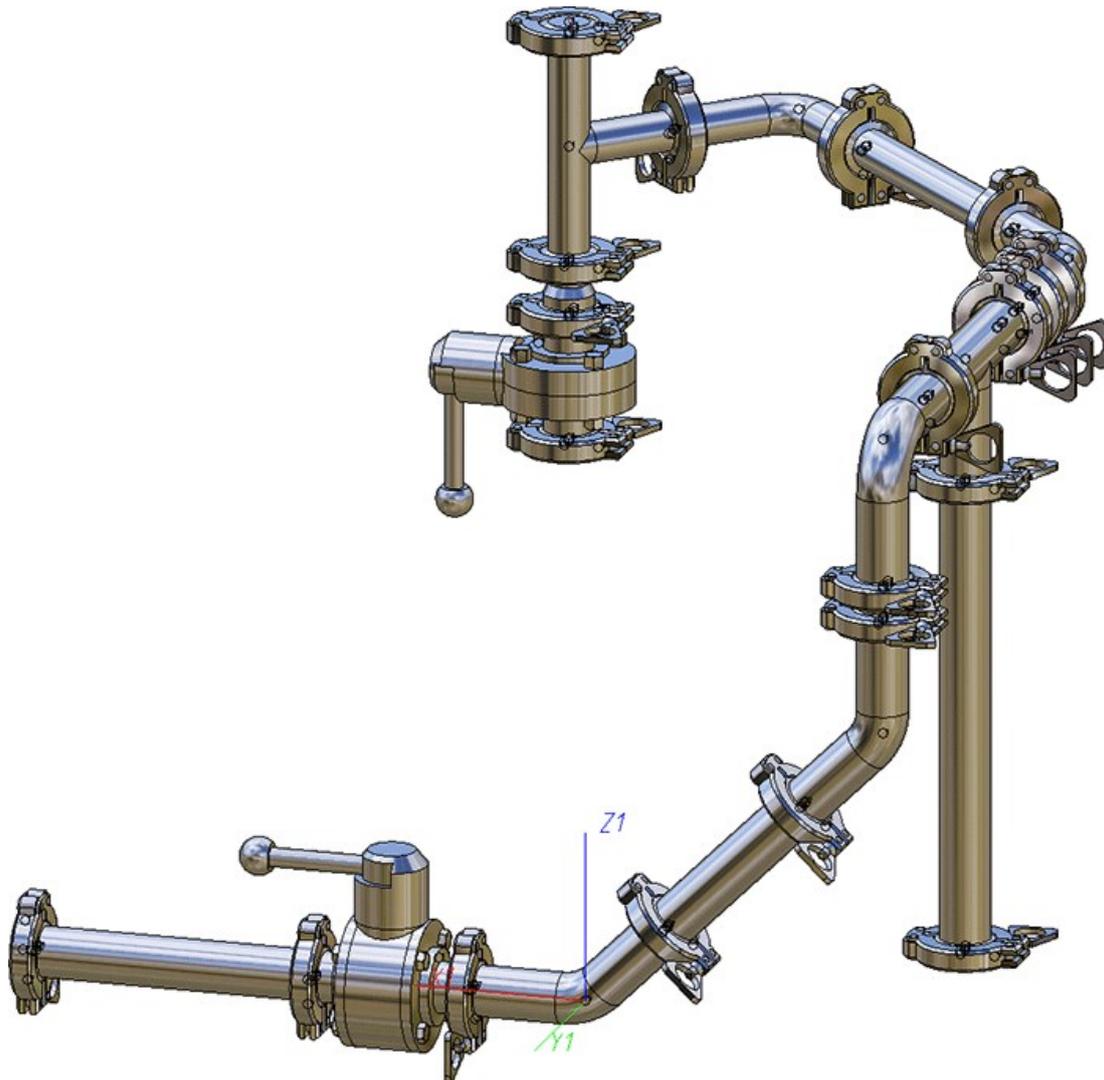


## Major Release 2018 (V 2300)

### New parts

#### Parts acc. to DIN 32676 (with Tri-Clamp connection)

The Standard Parts inventory for Plant Engineering has been expanded by parts according to DIN 32676.



Clamped intermediate pieces	Variant
DIN 32676-A	N32676-B_STRAIGHT.VAA
DIN 32676-B	N32676-B_STRAIGHT.VAA
DIN 32676-C	N32676-C_STRAIGHT.VAA

The Clamped intermediate pieces behave like straight pipes.

Fittings acc. to DIN 32676-A	Variant
Clamped connection	N32676-A_CLAMPJOINT.VAA

<b>Fittings acc. to DIN 32676-A</b>	<b>Variant</b>
Clamped cross	N32676-A_CROSS.VAA
Clamped elbow	N32676-A_ELBOW.VAA
Clamped reducer	N32676-A_REDUCER.VAA
Clamped reducer, excentric	N32676-A_REDUCER_EX.VAA
Clamped T-piece	N32676-A_TEE.VAA
<b>Fittings acc. to DIN 32676-B</b>	<b>Variant</b>
Clamped connection	N32676-B_CLAMPJOINT.VAA
Clamped cross	N32676-B_CROSS.VAA
Clamped elbow	N32676-B_ELBOW.VAA
Clamped reducer	N32676-B_REDUCER.VAA
Clamped reducer, excentric	N32676-B_REDUCER_EX.VAA
Clamped T-piece	N32676-B_TEE.VAA
<b>Fittings acc. to DIN 32676-C</b>	<b>Variant</b>
Clamped connection	N32676-C_CLAMPJOINT.VAA
Clamped cross	N32676-C_CROSS.VAA
Clamped elbow	N32676-C_ELBOW.VAA
Clamped reducer	N32676-C_REDUCER.VAA
Clamped reducer, excentric	N32676-C_REDUCER_EX.VAA
Clamped T-piece	N32676-C_TEE.VAA

<b>Valves acc. to DIN 32676-A</b>	<b>Variant</b>
Clamped ball valve, hand operated	N32676-A_BALLVALVE.VAA
Clamped butterfly valve, hand operated	N32676-A_BUTTERFLYVALVE.VAA
<b>Valves acc. to DIN 32676-B</b>	<b>Variant</b>
Clamped ball valve, hand operated	N32676-B_BALLVALVE.VAA
Clamped butterfly valve, hand operated	N32676-B_BUTTERFLYVALVE.VAA
<b>Valves acc. to DIN 32676-C</b>	<b>Variant</b>
Clamped ball valve, hand operated	N32676-C_BALLVALVE.VAA
Clamped butterfly valve, hand operated	N32676-C_BUTTERFLYVALVE.VAA

<b>Fasteners acc. to DIN 32676-A</b>	<b>Variant</b>
Clamp	N32676-A_CLAMP.VAA
O-ring	N32676-A_ORING.VAA
<b>Fasteners acc. to DIN 32676-B</b>	<b>Variant</b>
Clamp	N32676-B_CLAMP.VAA
O-ring	N32676-B_ORING.VAA
<b>Fasteners acc. to DIN 32676-C</b>	<b>Variant</b>
Clamp	N32676-C_CLAMP.VAA
O-ring	N32676-C_ORING.VAA

<b>Clamped sockets acc. to DIN 32676-A</b>	<b>Variant</b>
Clamped dummy socket	N32676-A_STUB_END.VAA
Clamped welded fusion spigot, long	N32676-A_STUB_LONG.VAA
Clamped welded fusion spigot, short	N32676-A_STUB_SHORT.VAA
Clamped female connector	N32676-A_STUB_THREADED_FEMALE.VAA
Clamped male connector	N32676-A_STUB_THREADED_MALE.VAA
<b>Clamped sockets acc. to DIN 32676-B</b>	<b>Variant</b>
Clamped dummy socket	N32676-B_STUB_END.VAA
Clamped welded fusion spigot, long	N32676-B_STUB_LONG.VAA
Clamped welded fusion spigot, short	N32676-B_STUB_SHORT.VAA
Clamped female connector	N32676-B_STUB_THREADED_FEMALE.VAA
Clamped male connector	N32676-B_STUB_THREADED_MALE.VAA
<b>Clamped sockets acc. to DIN 32676-C</b>	<b>Variant</b>
Clamped dummy socket	N32676-C_STUB_END.VAA
Clamped welded fusion spigot, long	N32676-C_STUB_LONG.VAA
Clamped welded fusion spigot, short	N32676-C_STUB_SHORT.VAA
Clamped female connector	N32676-C_STUB_THREADED_FEMALE.VAA
Clamped male connector	N32676-C_STUB_THREADED_MALE.VAA

To facilitate part synchronization, the Standard Parts have been centralized in the following archive file lists (lst):

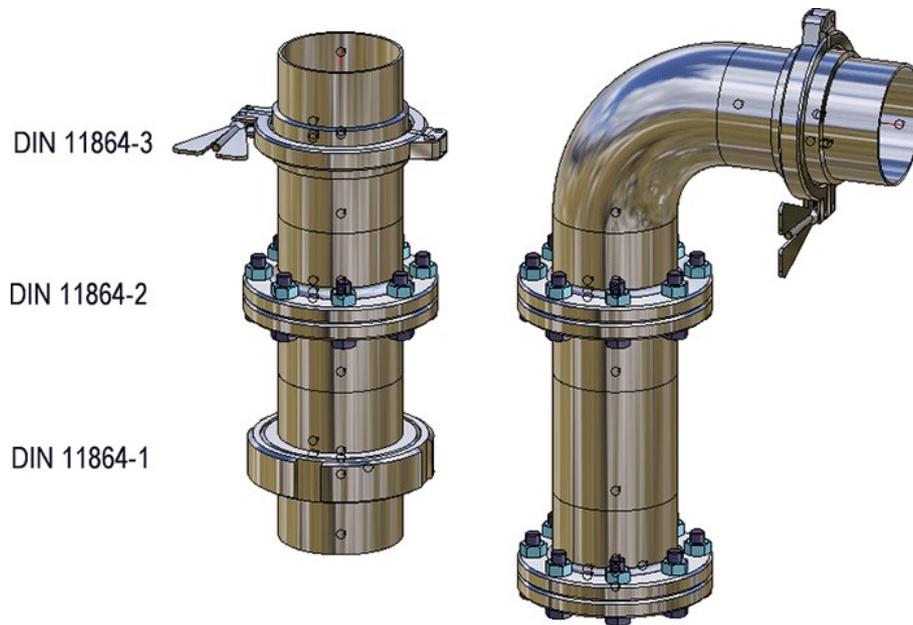
- DIN32676-A.lst
- DIN32676-B.lst
- DIN32676-C.lst
- DIN32676.lst, which combines the three standards above.

The lst files additionally contain straight pipes acc. to DIN 11866 which are combinable with the Clamped welded fusion spigots:

- N11866-A.vaa
- N11866-B.vaa
- N11866-C.vaa

### Pharmaceutical pipes acc. to DIN 11864 und DIN 11865

New in the Standard Part inventory for Plant Engineering are parts acc. to DIN 11864 and DIN 11865.



Bolted connections acc. to DIN 11864-1	Variant
Dummy socket with notch	N11864-1_BBS.VAA
Threaded dummy socket	N11864-1_GBS.VAA
Socket with notch	N11864-1_BS.VAA
Threaded socket	N11864-1_GS.VAA
Pipe union	N11864-1_RV.VAA

Along with these parts you can find the Grooved nut N11851\_NUT.vaa and the Sealing ring N11851\_OR.vaa in the Standard Parts inventory.

Flange connections acc. to DIN 11864-2	Variant
Flange with notch	N11864-2_BF.VAA
Flange with groove	N11864-2_NF.VAA
Flange connection	N11864-2_FV.VAA
Blank flange with groove	N11864-2_BNF.VAA
Blank flange with notch	N11864-2_BBF.VAA

Along with these parts you can find the Sealing ring N11864\_OR.vaa in the Standard Parts inventory.

Clamped connections acc. to DIN 11864-3	Variant
Clamped socket with notch	N11864-3_BK.VAA
Clamped socket with groove	N11864-3_NK.VAA
Clamp	N11864-3_KL.VAA

Clamped connections acc.to DIN 11864-3	Variant
Clamped connection	N11864-3_KV.VAA
O-ring	N11864-3_OR.VAA
Clamped dummy socket with groove	N11864-3_BNK.VAA
Clamped dummy socket with notch	N11864-3_BBK.VAA

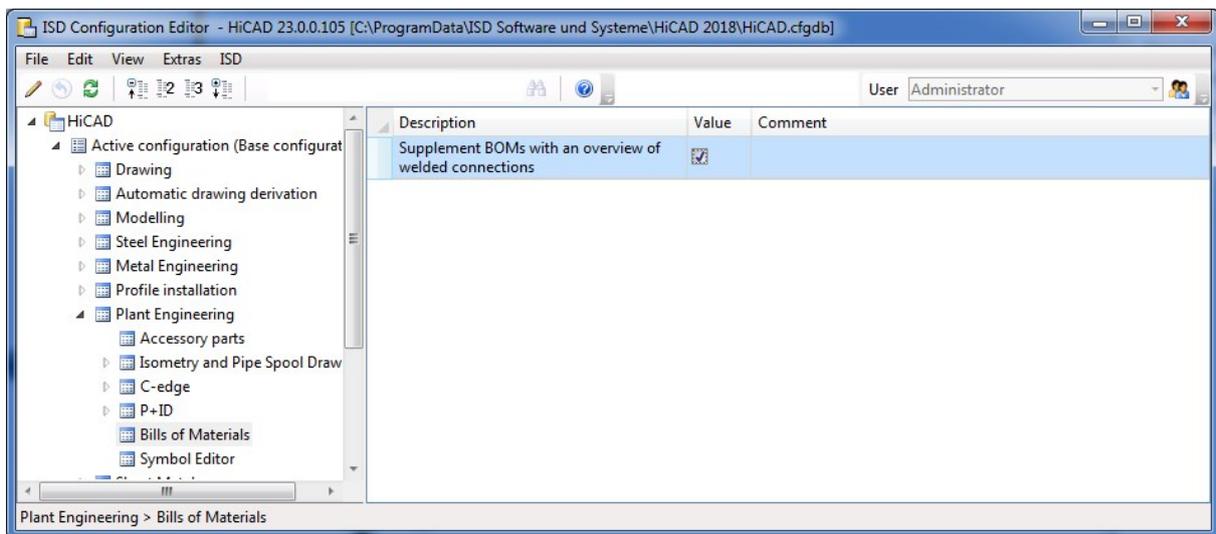
Fittings for orbital welding acc. to DIN 11865	Variant
Reducer	N11865-RK.VAA
Reducer, excentric	N11865-RE.VAA
Elbow 45°	N11865-BS-45.VAA
Elbow 90°	N11865-BS-90.VAA
Elbow 90°, long	N11865-BL-90.VAA
Rohrbogen 180°	N11865-BS-180.VAA
T-piece	N11865-TS.VAA
Reduced T-piece	N11865-TS-RED.VAA
T-piece, short	N11865-TK.VAA
Reduced T-piece, short	N11865-TK-RED.VAA
Cross	N11865-XS.VAA
Cross, short	N11865-XK.VAA

The parts according to DIN 11864 and 11865 can be found, combined for part synchronization, in the file **pharmaline.lst**. The above mentioned, matching Sealing rings and Grooved nuts are also contained in this file. Furthermore, the straight pipes that are combinable with these parts according to DIN 11866 are contained here.

## Evaluation

### Overview of welded connections in BOMs

You have now the option to display an overview of welded connections in BOMs. For this to happen, go to **Plant Engineering > Bills of Materials** in the Configuration Editor and activate the checkbox of the parameter **Supplement BOMs with an overview of welded connections**.



[Click here to view an Example.](#)

The display of this overview is only possible in BOMs that have been created with the BOM functions of the **Evaluation** function group; it is not possible for BOMs in isometries and pipe spool drawings.

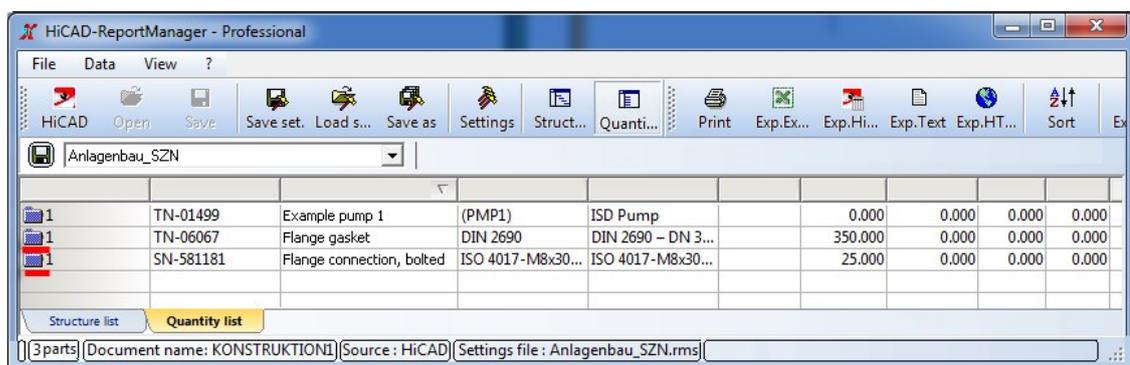
## Accessory sets

### Accessory sets for other Plant Engineering articles

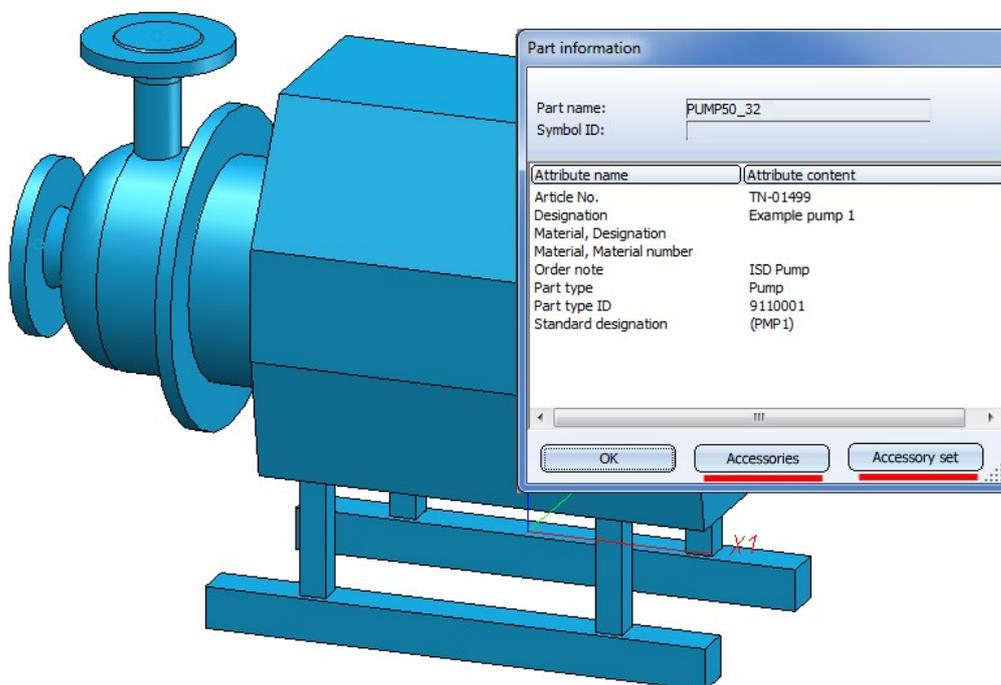
During part insertion via HELIOS part search HiCAD Plant Engineering supports the taking over of linked accessory sets. That is, if an accessory set is linked to a part/article, the corresponding entries will be read out of the accessory product structure and stored in the inserted part. This allows the accessories to appear in the HiCAD BOMs.

This mechanism was previously only supported for particular parts. From HiCAD 2018 onwards this will also be possible for components, i.e. for parts that do not belong to a pipeline. Furthermore, such an assignment is now also possible for pipe clamps and caps. The corresponding article masks have been adjusted accordingly.

In the image below a flange gasket and a flange connection that appear in the BOM have been assigned to a pump:



	Article No.	Description	Material	Material number	Quantity	Quantity	Quantity	Quantity
1	TN-01499	Example pump 1 (PMP1)	ISD Pump		0.000	0.000	0.000	0.000
1	TN-06067	Flange gasket	DIN 2690	DIN 2690 – DN 3...	350.000	0.000	0.000	0.000
1	SN-581181	Flange connection, bolted	ISO 4017-M8x30...	ISO 4017-M8x30...	25.000	0.000	0.000	0.000



## Expanded HELIOS article masks

The HELIOS article masks for components and parts of the type

- Vessel,
- Pump,
- Other components,
- Cap, round and
- Pipe clamp

have been expanded by the attribute **ZUBEHOERSATZ** (Accessory set).

*Article mask for Vessels*

Find article

Search conditions

Article Classification Vessel

General

Design:

Dimensions & Content

Diameter (1):  mm

Cyl. length  mm

Useable volume:  m<sup>3</sup>/h

Insertion properties

Preferred type:

Accessory set:

P-ID symbols

Article mask for Pumps

Find article

- Search conditions

Article Classification Pump

General pump parameters

Design:

NPSH value:  m

Delivery volume:  m<sup>3</sup>/h

Delivery height:  mm

Nominal performance:  kW

Pump nozzle

DN Pump inlet side (1):  mm

DN Pump outlet side (2):  mm

Pump diameters

Length (3):  mm

Width (4):  mm

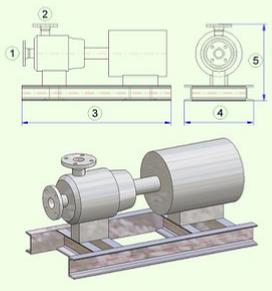
Height (5):  mm

Insertion properties

Preferred type:

**Accessory set:**

P-ID symbols:



Article mask for Other components

Find article

- Search conditions

Article Classification Other component

General

Design:

Dimensions

Length (3):  mm

Width (4):  mm

Height (5):  mm

Insertion properties

Preferred type:

**Accessory set:**

P-ID symbols:



Article mask for Pipe clamps

Find article

- Search conditions

Article Classification Pipe clamp

Connections

Nominal diameter:  mm  inch

Outer diameter (1):

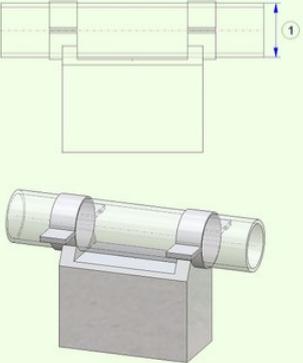
Fitting

Preferred type:

Type ID 1:

Type ID 2:

Accessory set:



Article mask for Cap, round

Find article

- Search conditions

Article Classification Cap, round

Connections

Nominal diameter:  mm  inch

Outer diameter (1):

Connection type:

Pipe part properties

Wall thickness (2):

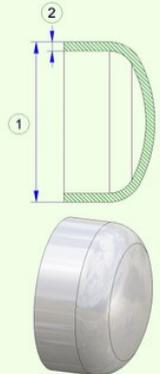
Schedule:

Pressure:

Fitting

Preferred type:

Accessory set:



## Change pipeline route

The **Change route**  function has been revised and expanded.

## Create connecting guidelines

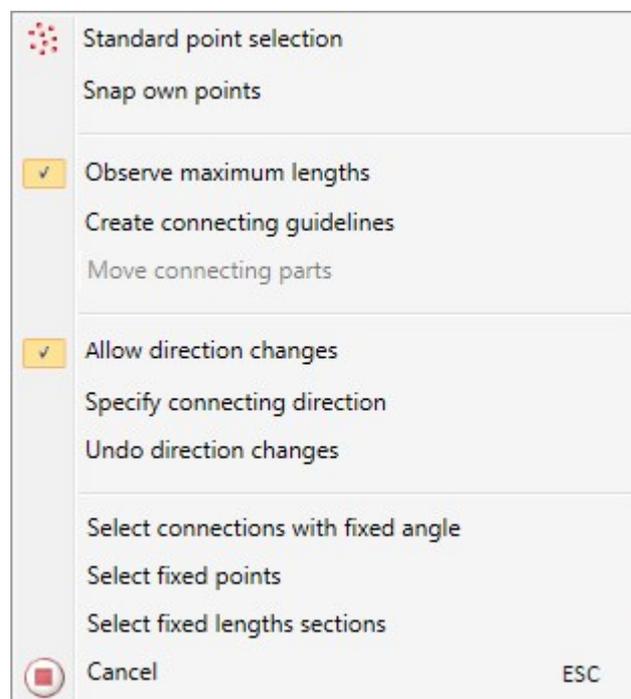
The **Create connecting guidelines** function can now be activated via the context menus for pipelines, guidelines or parts. Right-click them to open the context menu.

## Changed context menu

The context menu that can be activated with a right-click after selecting the point to be moved has been changed:

- The creation of connecting guidelines must now be activated in the context menu first (it is now activated by default).
- The menu items that switch options on or off are now marked with a checkmark  symbol. This concerns the following menu items:
  - Snap own points
  - Observe maximum lengths
  - Create connecting guidelines
  - Move connecting parts.

For example, instead of the two menu items **Observe maximum lengths** and **Ignore maximum lengths** there is now only the menu item **Observe maximum lengths** - either with (activated) or without checkmark (deactivated).



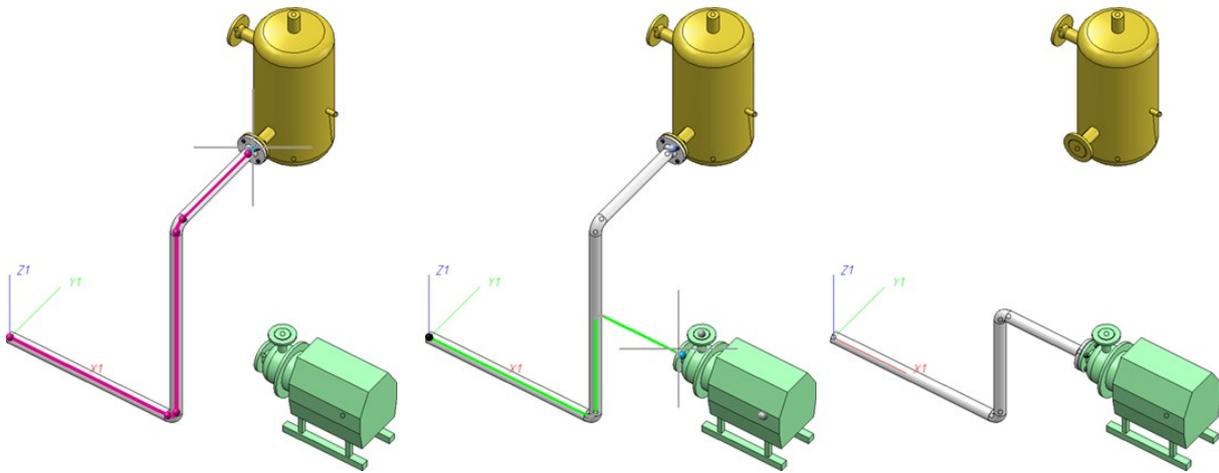
### Change angles

The function now also allows a changing of angles. The connections of the pipeline parts are regarded as possible rotation axes in the process.

Please note the following:

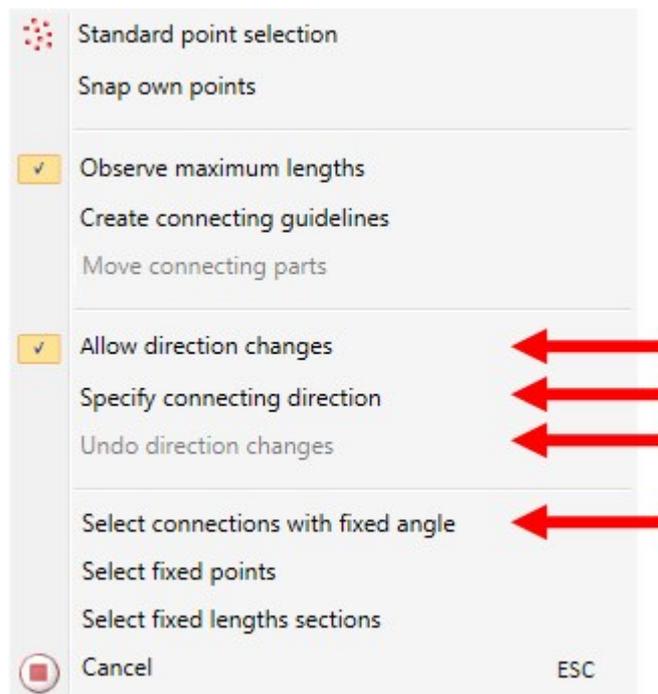
- If you modify a pipeline with the **Change route** function and choose a free connecting point as the point to be moved, angle changes will be possible.
- If you have selected a point within the pipeline, no angle changes will be possible.

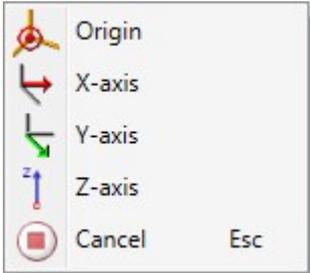
As soon as you choose a target point, its connecting direction will be evaluated and the preview will be adjusted accordingly. A click on the target point ends the route change and the new route will be applied.



The direction at the target point will even be applied if the target point cannot be reached and will remain when you move the cursor away from the target point again. This allows you to apply a direction from the drawing without having to choose a fixed target point.

Due to the new option to change angles, the context menu (right mouse button) has been expanded. This menu can be activated while HiCAD requests the selection of a target point.

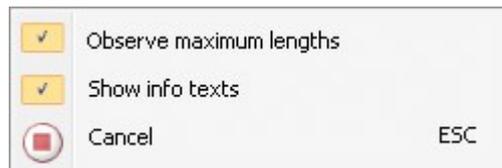


Option	
<b>Do not allow direction changes</b>	<p>By default, the connecting direction of Plant Engineering connecting points are used in such a way that the orientation of the pipeline can be changed. If you choose the <b>Do not allow direction changes</b> in the context menu, this behaviour will be deactivated.</p> <p>If the <b>Do not allow direction changes</b> option is active, you will find the <b>Allow direction changes</b> option in the context menu that restores the default behaviour.</p>
<b>Specify connecting direction</b>	<p>The connecting direction cannot just be changed at Plant Engineering connecting points. The <b>Specify connecting direction</b> allows you to pick an arbitrary direction in the drawing. You use the function of a context menu (right-click) for direction specification:</p>  <p>If you specify a connecting direction, this will be indicated by an arrow pointing in the connecting direction. This will also deactivate the applying of the connecting direction from Plant Engineering connecting points, i.e. the specified connecting direction will always be preferred.</p>
<b>Undo direction changes</b>	<p>If you select this option, all directions will be reset to the state of the original pipelines, i.e. all specified direction changes will be undone.</p>
<b>Select connections with fixed angle</b>	<p>Use this option to select connections for which no direction changes should be allowed. The procedure is the same as the one for selection of fixed points. A single click on a connection marks the connection, another click removes the marking.</p>

## Close gaps within pipelines

### Changed context menu

The context menu for the **Close gaps within a pipeline (PE)** function has been revised. The menu items that switch options on or off are now marked with a checkmark symbol. For example, instead of the two menu items **Observe maximum lengths** and **Ignore maximum lengths** there is now only the menu item **Observe maximum lengths** - either with (activated) or without checkmark (deactivated).



## Variant Editor

### Automatic deriving of variants with varying sub-types

You can use the Variant Editor for an automatic deriving of variants. With its help it is possible to create new .VAA variants whose attributes have been modified via a template in the form of a CSV file. In previous versions the automatic deriving was confined to an adjustment of attributes which are constant throughout all sub-types of a variant.

As of HiCAD 2018 it is also possible to modify sub-types of automatically derived variants. Function call and procedure are largely identical to that for automatic variant deriving with constant sub-types. However, additional attributes are available for varying sub-types:

- **VAREDIT\_CLUSTERING\_KEY**  
A list of columns from the CSV file is assigned to this attribute, which are to be checked for equality.
- **VAREDIT\_SUBTYPE\_KEY**  
A number of HELIOS attributes which determine the columns to be compared can be assigned to this attribute. In contrast to VAREDIT\_CLUSTERING\_KEY, column names of the CSV file are not used here. .

# P+ID

## Service Pack 1 2018 (V 2301)

### P+ID dialogue types

In the list of dialogue types, the types 510 - 599 are now available as optional item attributes. They do not have a name, as the name will be defined by the user.

### Start P+ID project with new sheets

From now on new P+ID projects will be started with a new sheet. The sheet will be based on the template defined in the Settings. In addition, you can specify via the Configuration Editor whether the master data dialogue for a sheet is to be shown when it is created.

### Take over changed pipeline diameter in P+ID to 3-D layout plan

You now have the option to change the nominal diameter of a pipeline in the P+ID, even if its pipeline parts have been subordinated to it by means of a part assignment. When changing the nominal diameter, existing part assignments of subordinated pipeline parts will be adjusted accordingly; this will be indicated by a corresponding message. If more than one matching part will be found, one of these parts needs to be selected.

When switching to the 3-D layout plan with the **Assigned 3-D layout plan**  function, the assigned pipeline will be exchanged after confirming the security prompt.

### Assign part data

If you want to assign part data to a symbol, select the **Edit symbol**  function and activate the part search. If pipe class or nominal diameter have been preset, article search will start without any further inputs.

## Major Release 2018 (V 2300)

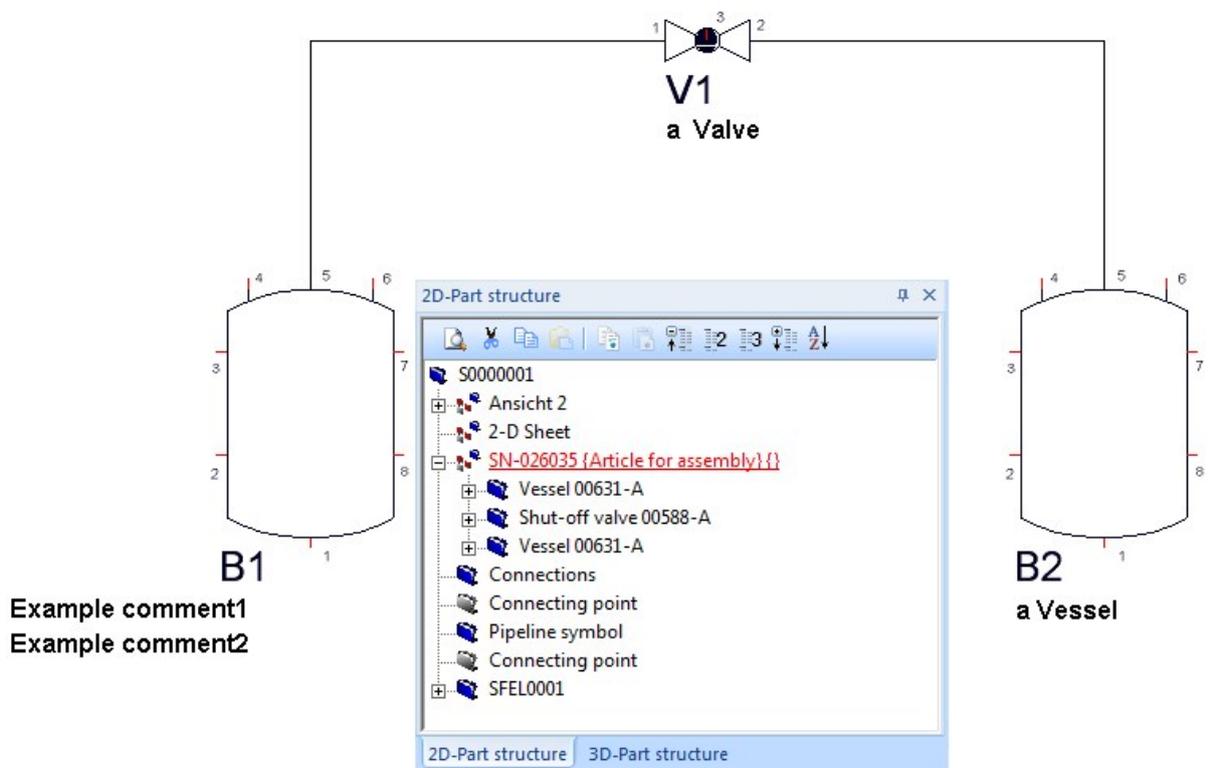
### Apply product structure changes in HELiOS

Changes to article product structures in HELiOS can be taken over by the P+ID.

The steps described below enable you, in the following example, to synchronize comments and update part IDs via

the **Update item attributes (P+ID)**  function.

#### Pipeline-20



### Step 1: Preparation

If you want any particular P+ID attributes occur multiple times as item attributes in the P+ID symbol mask, these attributes must be clearly identified in the **attrcadhelios.dat** according to their position **p** within the P+ID mask, namely, with the syntax:

```
"hcad";"P+ID attribute key(p);"Designation of item attribute";"TYP";
```

For instance, the effect of the lines

```
"hcad";"HICAD_PID_KOMMENTAR(1);"PID_KOMMENTAR1";"STRING";
```

```
"hcad";"HICAD_PID_KOMMENTAR(2);"PID_KOMMENTAR2";"STRING";
```

in **attrcadhelios.dat** will be that the P+ID comment attribute can be declared twice as item attribute.

To insert the Part ID, add the following line:

```
"hacad";"HICAD_PID_BAUTEILKENNUNG";"PID_BAUTEILKENNUNG";"STRING";
```

Then, call the **HELIOS for Plant Engineering** function in the **DBPlantDataImport.exe** program.

To be able to modify the P+ID attributes in the attrcadhelios.dat (sys directory) in HELIOS, they need to be entered into the input masks for item attributes with the help of the **Mask Editor** maskedt2.exe (see HELIOS).

First, open the **Position** mask.

After opening the **Position** mask, right-click on the mask to open a context menu. Choose **New data field** and enter the new attributes (Comment(1), Comment(2) and Database Part ID. More information on the Mask Editor can be found in the HELIOS Online Help

## Step 2: P+ID



First, draw a P+ID. Use the **Edit symbol** function to assign comments and part data to the symbols.

Dialogue type	Dialogue text
Comment (1)	Example comment 1 ←
Comment (2)	Example comment 2 ←
Plant code	
Plant unit	
Plant/Plant unit	
Aggregate	BB000
Plant/Plant unit/Aggregate	
Location	
Function	
Code	B1

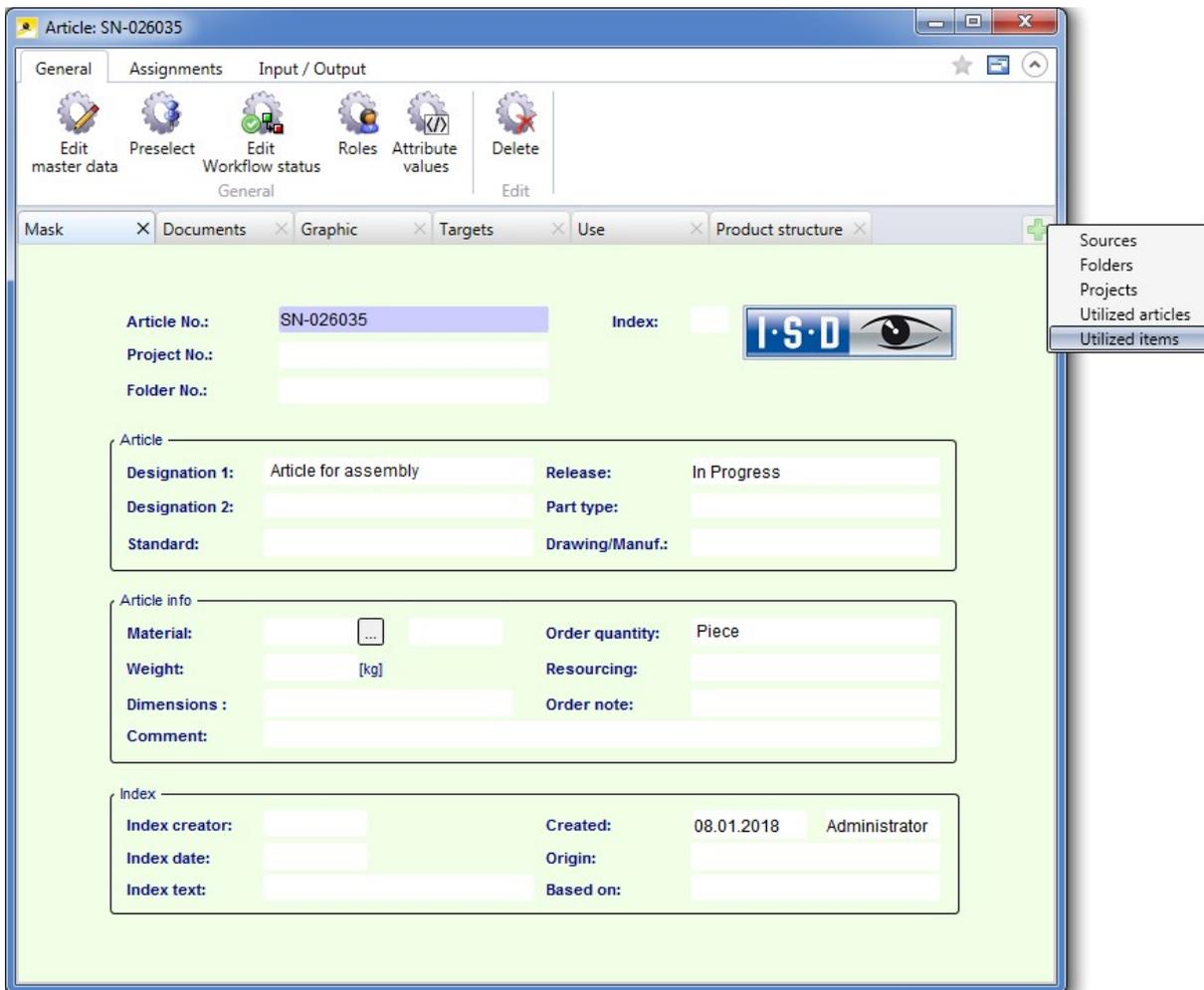
Buttons at the bottom: **Part data** (circled in red), **Delete part assignment**, **Transfer part assignment**, **OK**, **Cancel**.



Create a **Main assembly** and move the symbols whose assigned articles (=part data) are to belong to their product structure, beneath the main assembly. Assign an article master to the main assembly. Now you can transfer the product structure for this article (right-click article and choose **HELiOS > Transfer structure, one level**).

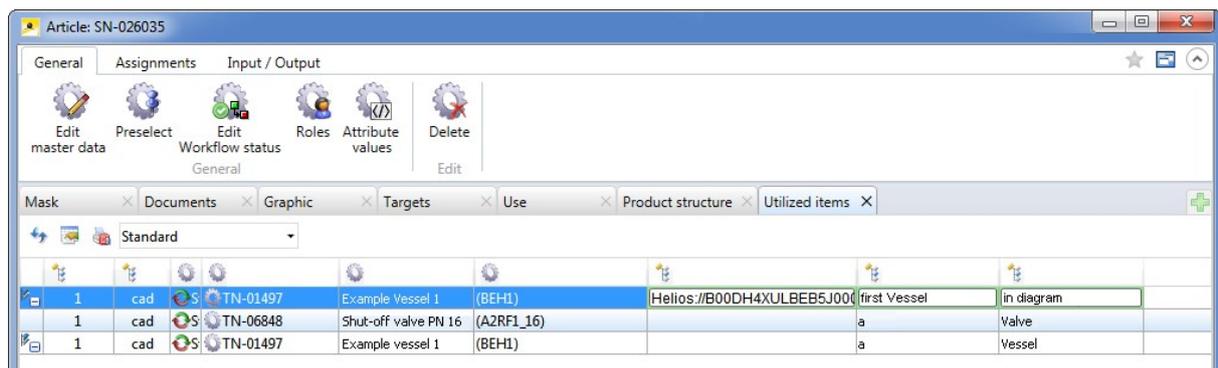
### Step 3: Change comment in HELiOS and synchronize with P+ID

Configure the item mask as described in [Step 1](#). In HELiOS, call the article master of the assembly. Click on the "Plus" icon  and activate the **Utilized items** tab in the mask.



After this, you need to configure the mask. In the Utilized items tab, click on the table header and choose **Configure**. Activate the table columns **Database Part ID** (PID\_BAUTEILKENNUNG), **Comment 1** and **Comment 2** (PID\_KOMMENTAR1, PID\_KOMMENTAR2).

In the **Utilized items** table you can now change the comment fields.



By adding the P+ID attribute key **HICAD\_PID\_BAUTEILKENNUNG (HICAD\_PID\_PART\_ID)** (HELIOS URL) a P+ID base symbol can be replaced with a regular symbol in the product structure, namely, by replacing the existing HELIOS URL with the one of the regular symbol.

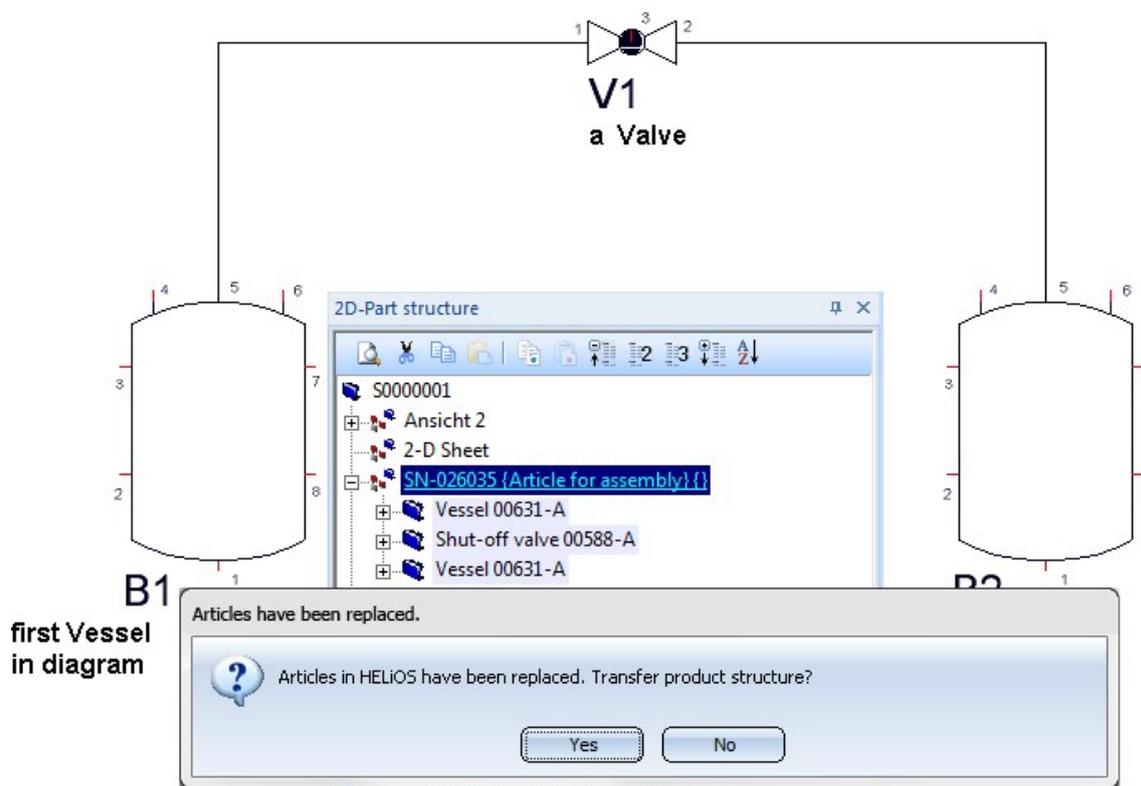
You obtain the HELIOS URL of a symbol by searching for the corresponding article and choosing the **Copy HELIOS URL** function (right-click) in the article master.

Transfer the edits to the article master of the assembly with the **Apply item edits** function (right-click article master of main assembly).

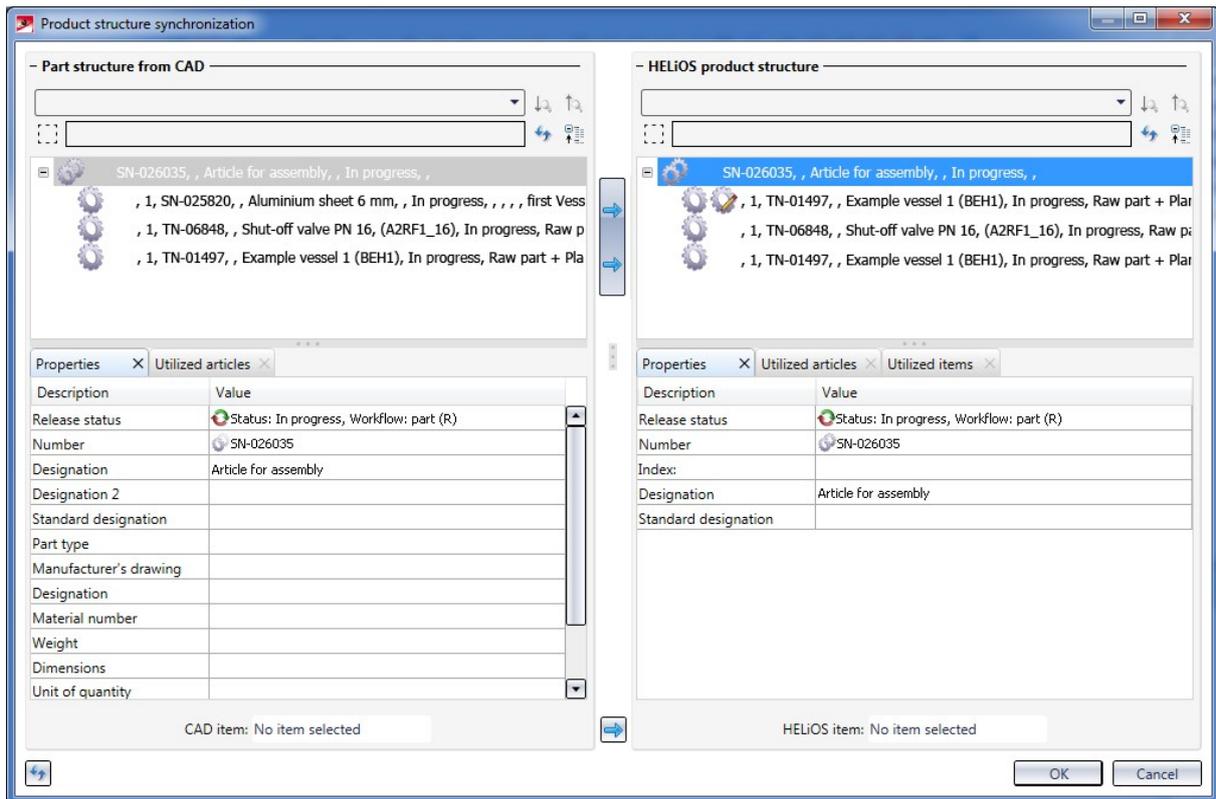
After applying the item changes in HELIOS, you can also update them in the P+ID with the **Update item attributes** function (in the ICN, right-click the assembly and choose **HELIOS**).

If text attributes have been assigned to some item attributes, changes in these text points will be instantly visible.

### Pipeline-20



To allow HELIOS a possible replacing of P+ID base symbols with regular symbols, the product structure must be transferred once more. To do this, click **Yes** in the message window.



Transfer the part structure to the HELIOS product structure and close the mask with **OK**.

## Renamed item attributes

The following changes have been made to the IDs for item attributes in order to make the designations more descriptive:

- HICAD\_PID\_SACHNUMMER was replaced with HICAD\_PID\_BAUTEILKENNUNG und
- HICAD\_PID\_BAUTEILATTRIBUT was replaced with HICAD\_PID\_DOKUMENTKENNUNG

Further attributes can be found in the list of dialogue types.

## Settings from TXTPAR.DAT moved to Configuration Editor

In previous versions the settings for texts and annotations could be set in the TXTPAR.DAT file. As of HiCAD 2018 this file is no longer available. Also, the corresponding setting options at **Settings > Basic settings > 2-D** have been removed.

Instead, these settings are now available in the Configuration Editor, namely, at

- **Drawing > Annotations > Text > Font** and
- **System settings > Annotations > Text**.

### Text

Use the **Default parameters**  function (2-D Dimensioning + Text > Edit (Text) Settings  > ...) to load the settings from the Configuration Editor at **Drawing > Annotations > Text > Font**.

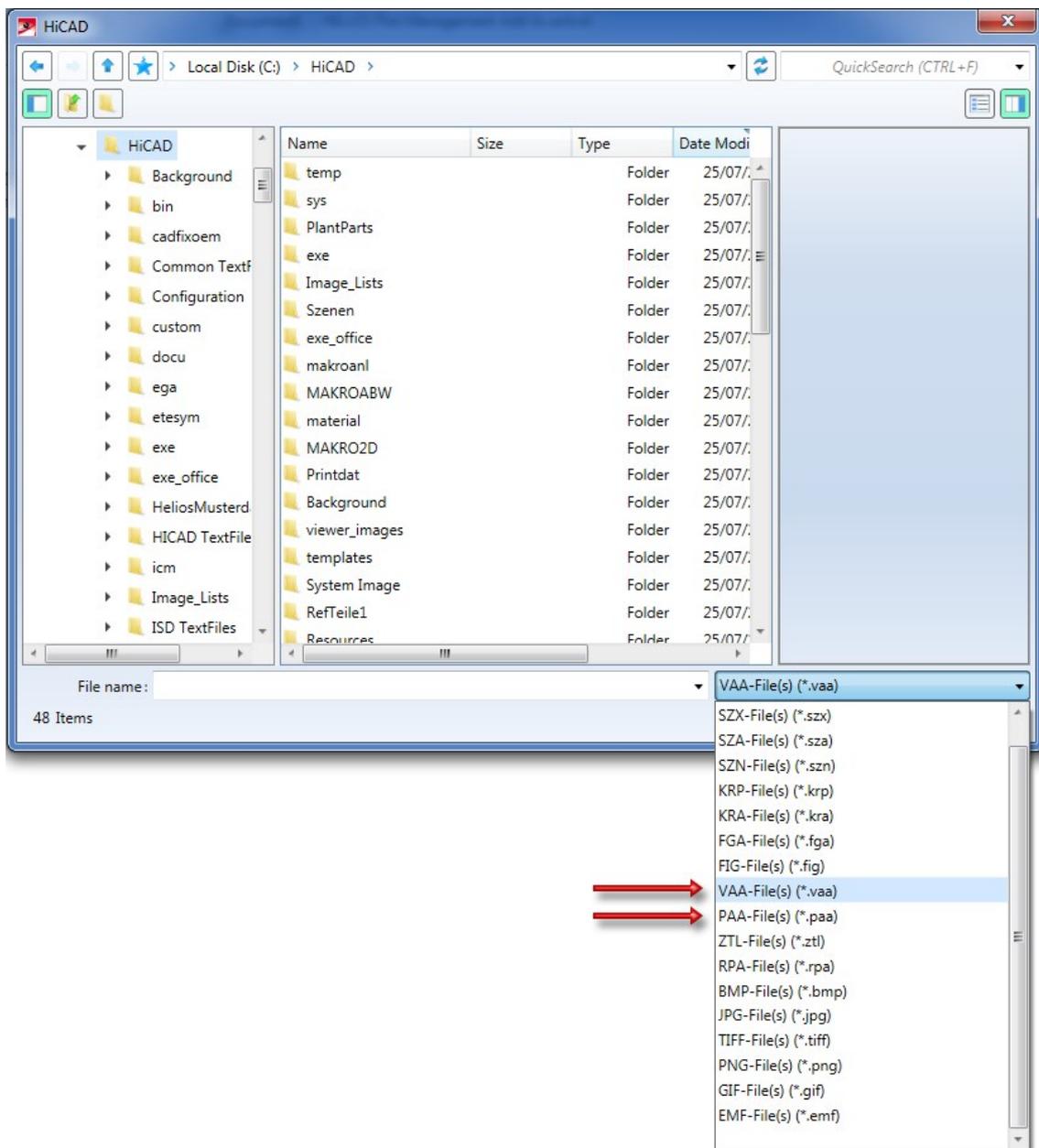
The values of the drop-down lists for font size, text width coefficient and line spacing in the Text Editor and Annotation Editor can be arbitrarily expanded in the Configuration Editor at **System settings > Annotations > Text > Preferred text sizes**. After restarting HiCAD the new default settings will be used.

# Viewer

## Major Release 2018 (V 2300)

### Display of VAA and PAA files

Like KRA files, the contents of VAA files and PAA files can now be visualized in the Viewer. Only the geometry contained therein will be displayed (General type) and not the individual sub-types. For VAA files the display is limited to the preview images in the EMF format, and for PAA files to the contained KRA file. If no EMF file exists, or if the KRA file contains no viewing data no preview can be provided.

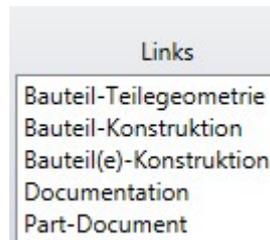


# HELiOS Desktop

## Service Pack 2 2018 (V 2302)

### Combined search: Filter options for Documents or Articles without Links

The behaviour of the **Combined search** function has been modified: If you deactivate all link classes for the search, i.e. nothing has been selected in the field beneath **Links**, all objects matching the entered search criteria that are not linked to any other object will be shown in the corresponding result list.



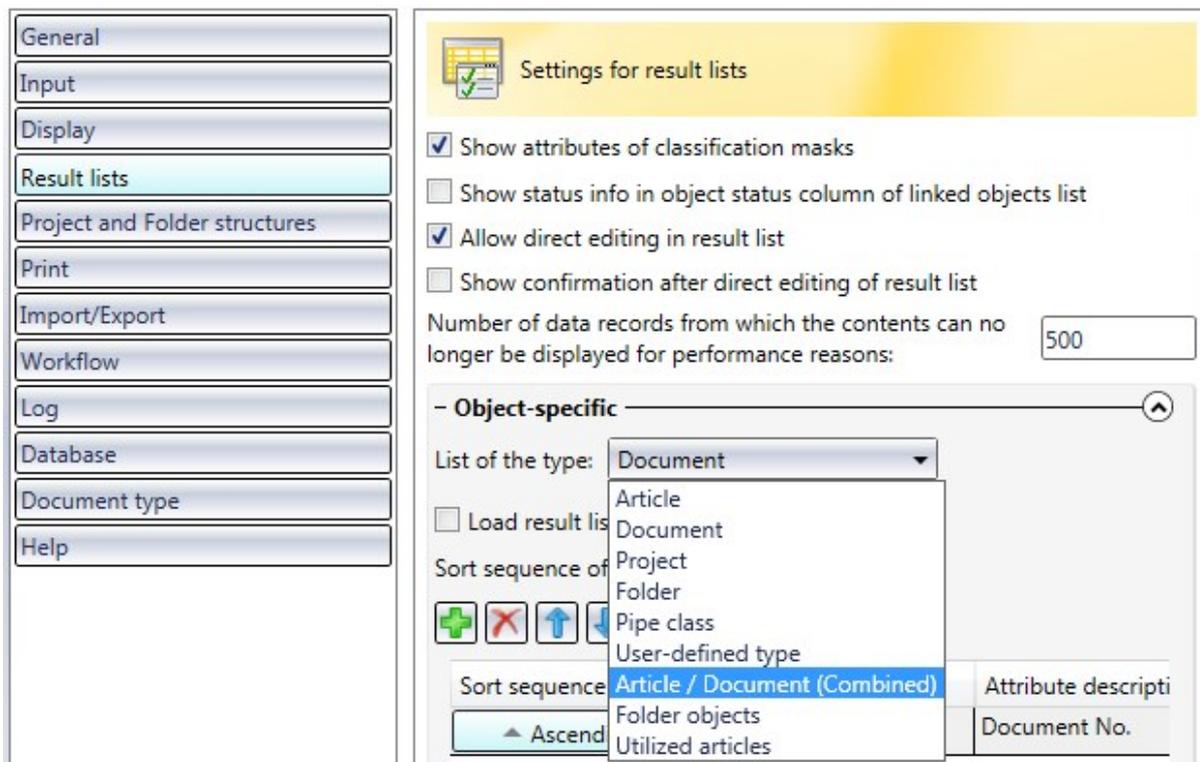
In this way you can, for instance, list documents of already deleted articles in conjunction with other, attribute-related search criteria, e.g. in order to link, move or delete them.

Furthermore, you can search for articles without assigned drawing in order to determine for large assemblies whether drawings have been created for all related parts, or whether parts have been "overlooked" by mistake.

## Result list settings for Combined search

Beneath **Object-specific** on the **Result lists** tab of the **HELiOS Options** dialogue window you can find a new pull-down menu item called **Document and Article**.

This setting is for the **Combined search** function. It allows you, for instance, to display the result lists of a combined search sorted by document numbers in ascending order by default.



## Enhanced model structure representation

The **Model structure** representation in Document detail masks has been enhanced.

Icons in the tree structure indicate the model type (part, assembly...) and dependent parts are also marked as such.

Also, it is indicated for each Inventor-related item whether it is a "regular" Part / Assembly, a parameter reference or a third party reference.

Multiple occurrences of items are indicated in the **Qty.** column.

Model	Qty.	Article No.	Designation	BOM-relevance
DN-002178	1			
DN-002180	1	SN-027430	Bottom	Normal
<i>fx</i> DN-002182	1	SN-027432	Bottom - Master	-
DN-002181	1	SN-027431	Bottom - Back	Normal
DN-002182	1	SN-027432	Bottom - Master	-
DN-002183	1	SN-027433	Bottom - Front	Normal
DN-002182	1	SN-027432	Bottom - Master	-
DN-002184	1	SN-027434	Lower Mount	Normal
DN-002185	1	SN-027435	Spring	Normal
DN-002186	1	SN-027436	Bottom - Anvil	Normal
DN-002187	2	SN-027437	Upper	Normal
DN-002188	1	SN-027438	Upper - Back	Normal
DN-002189	1	SN-027439	Upper - Master	-
DN-002190	1	SN-027440	Upper - Front	Normal
DN-002189	1	SN-027439	Upper - Master	-
DN-002191	1	SN-027441	Channel - Spring Clip	Normal
DN-002192	1	SN-027442	Channel	Normal
DN-002193	1	SN-027443	Channel - Base	Normal
DN-002194	1	SN-027444	Channel - Pusher	Normal
DN-002195	1	SN-027445	Channel - Cover	Normal
DN-002196	1	SN-027446	Channel - Hinge Pin	Normal
DN-002197	1	SN-027447	Staples	Normal
DN-002198	1	SN-027448	Channel - Bumper	Normal

Number of data records: 23



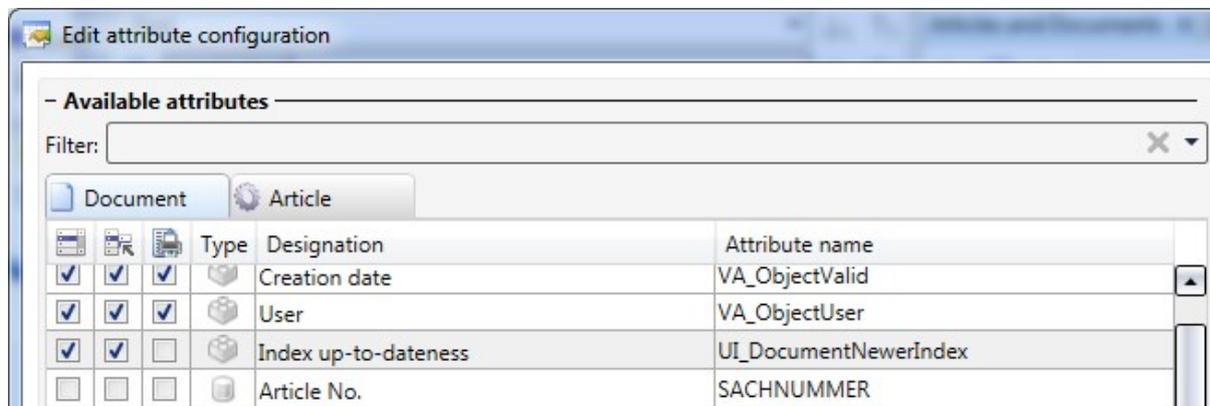
**Please note:**

To ensure a correct result list representation of the model structure, the Article context of the top part must be unambiguous!

The Article context of the sub-parts is determined by the saving in the CAD system.

## Result lists: Marking of new indices

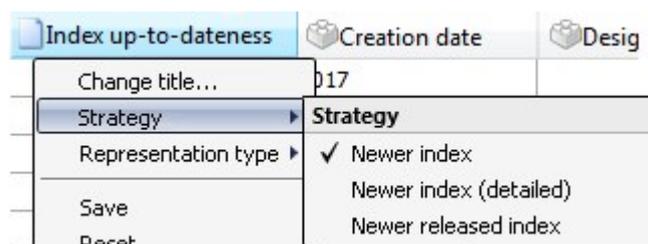
In the result list configuration you can find the new **Index up-to-dateness** attribute ("UI\_DocumentNewerIndex" for documents, "UI\_ArticleNewerIndex" for articles):



If an Article or a Document has a newer index, older indices will be indicated in the **Index up-to-dateness** column with a star symbol:

Number	Index	Index up-to-dateness
DN-000242	e	
DN-000242	d	★
DN-000242	c	★

Right-clicking the title of the **Index up-to-dateness** opens a context menu. The menu item **Strategy** allows you to choose between three different additional display options:



- **Newer index:** The ★ symbol indicates that a newer index exists for the listed object, i.e. that the index of the listed object is not up to date.
- **Newer index (detailed):** If the object is marked with the ★ symbol, a higher (newer) index exists, but no newer released version of the object. The 🌟 symbol indicates that higher and released indices exist for the listed object, but that the highest index has not been released yet. If a result list entry is marked with the 🌟 symbol, this indicates that a version with a higher index with Workflow status "Released" exists.
- **Newer released index:** If a result list entry in the **Index up-to-dateness** column is marked with the 🌟 symbol, this indicates that a higher index with Workflow status "Released" exists.

A simple click on the star in the result list opens the **Version Management** for the corresponding object, a double-click will additionally open the Document / Article detail mask.

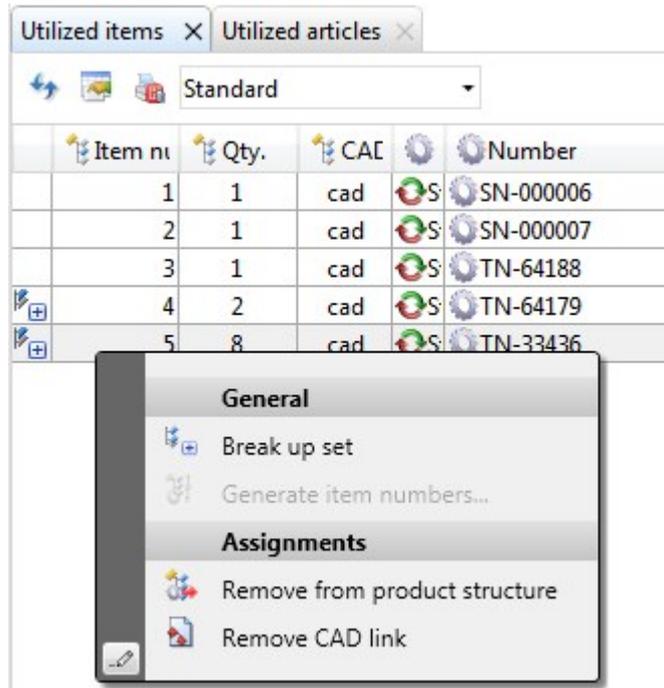


Whether only current or all indices are to be displayed in the result lists can be specified in the **HELiOS Options at Result lists > Show objects**.

### Items combined into sets: Enhanced context menu

For the editing of items that have been combined into sets you can find an extended context menu that does not only consider the combined set of items, but also offers the option to edit the individual items.

For instance, you can delete items from the product structure, or remove the CAD link if desired.



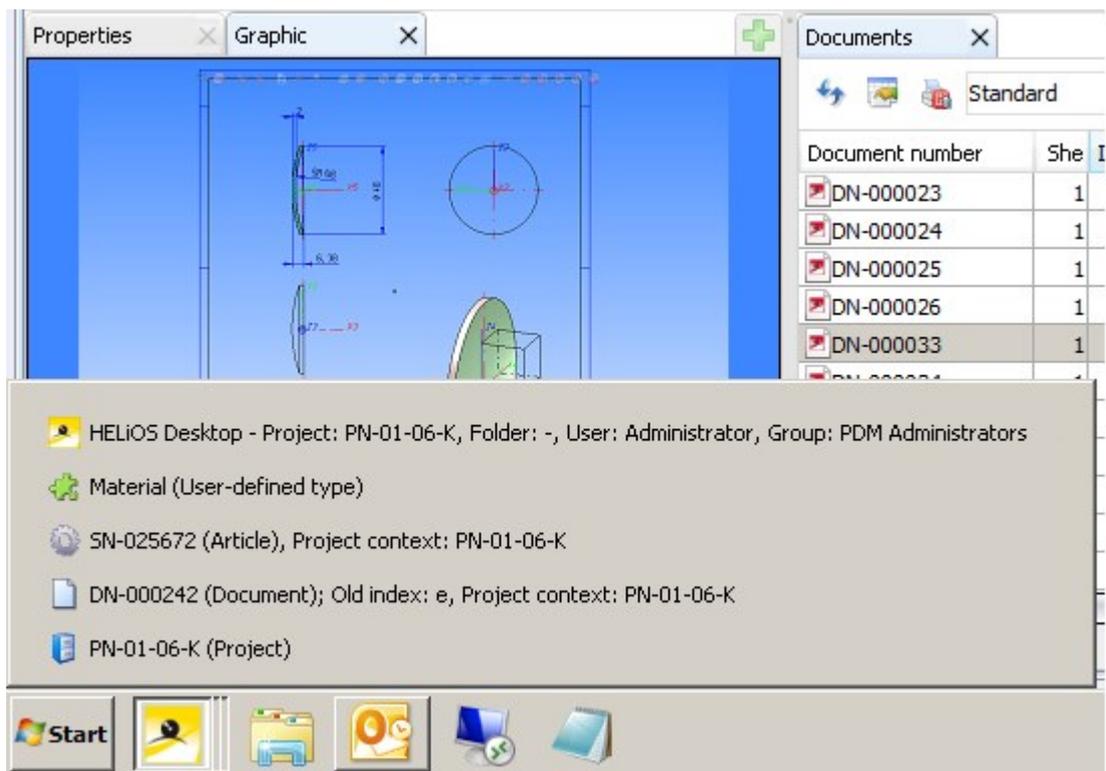
### Detail dialogues in Windows task bar

Detail dialogue windows of HELiOS are shown as minimized windows in the Windows task bar.

Each of the different object types (Article, Document, Project, Folder, User-defined type) is represented by a corresponding icon. In the text line next to the icon the name of the object is shown (e.g. the document number).

If an object from the result list of a Folder Explorer or Project Explorer has been opened, the Folder context or Project context, respectively, will also be shown, accordingly.

Also, older indices of the corresponding object will also be indicated.



## Service Pack 1 2018 (V 2301)

### Performance

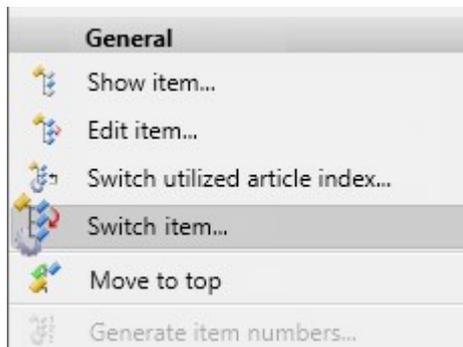
A performance increase of 20 - 30 % could be achieved for the loading and build up of mask elements.

### Product structure: Switch item

The context menu for items now contains the new  **Switch item** function.

Use this function to switch an article of an item for another item.

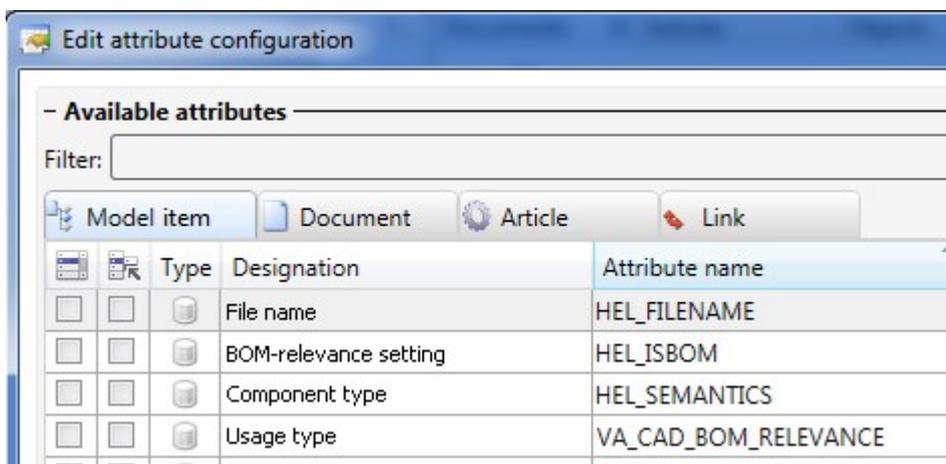
When you call the function the Find article dialogue window will be displayed. Here, choose the required article (a search for a concrete index is also possible here). Confirm to exchange the article.



Please note that the function considers the settings for the **Database** option **Allow updating of released product structures**.

### Attribute configuration for Model items

The **Edit attribute configuration** dialogue window for result list configuration on the **Model structure** context of Documents now offers the additional **Model item** tab with model item attributes.



## New date selection

The dialogue for data selection, e.g. for the finding or the input of objects, the selection of a particular index date in the result list display etc. comes with a new Look & Feel.

This also includes a menu with extended selection options for a date-based search for objects (e.g. documents, articles, folders).

The dialog box is titled "Index" and contains the following elements:

- Index creator:** [Text input field]
- Index date:** [Text input field] [Calendar icon]
- Index text:** [Text input field]
- File name:** [Text input field]
- Origin:** [Text input field]
- Date Selection Options (Left):**
  - Today
  - Yesterday
  - This week
  - Last week
  - Last 7 days
  - This month
  - Last month
  - This year
  - Last year
- Search Criteria (Center):**
  - Exact
  - From
  - To
  - Between
- Calendar Views (Center):** Two calendar views for May 2018. The first calendar has the 15th selected. The second calendar has the 16th selected.
- Actions (Bottom):**
  - Replace
  - Add
  - 
  -

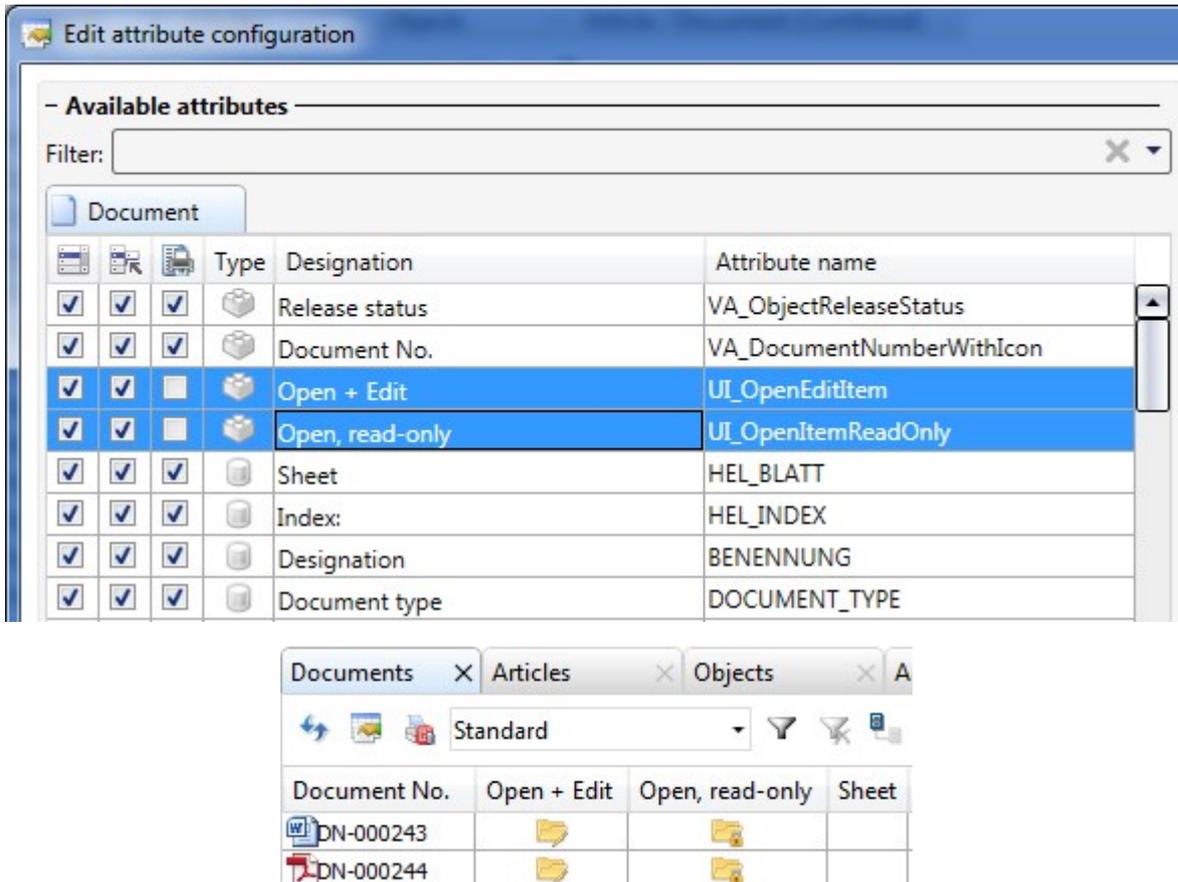
You can also search for objects here that have been created at a particular data and time, or select multiple specific dates.

Furthermore, you can limit your object search to specific creation, index or checking dates or periods.

### Result list button for opening of documents

As a faster alternative to a function selection via context menu (right-click and choose function), you have now the option to assign buttons to document result lists, allowing you to open documents in the linked application with one mouse click, either for editing or in read- only mode.

For this purpose the two attributes **UI\_OpenEditItem** and **UI\_OpenItemReadOnly** have been created, which just need to be activated in the result list configuration.



AS of Service Pack 1 these buttons are available in every result list with document context, e.g. in the **Document Class Explorer**, in the **HELIOS Workspace**, in the **Targets** tab of article master masks etc.

## Major Release 2018 (V 2300)

### Editing of combined items

The attributes of combined items on the Utilized items tab can now be edited in one step if the values of the item attributes are identical.

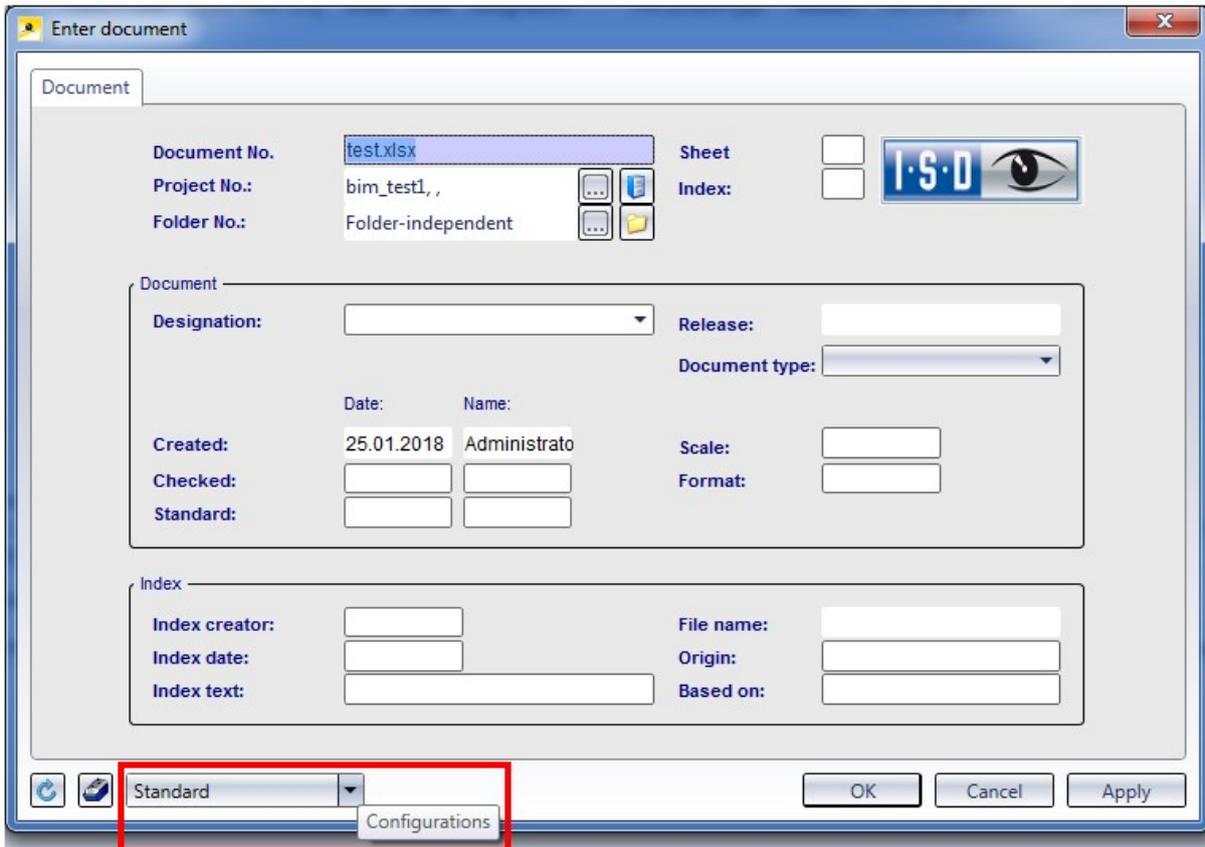
If this is not the case, the concerned attribute value cannot be edited on the basis of the combined items. For the corresponding combined item the entry "*Different values*" will be shown in the **Comment** column. In such cases the items must be edited individually.

Utilized items		Utilized articles					
Item nr	Qty.	CAD link	Number	Designation	Comment	Standard designation	
1	1	cad	SN-000010	Guide sleeve			
2	1	cad	SN-000012	Lock ring			
3	1	cad	SN-000013	Protecting cap			
4	2	cad	SN-000011	Spring guide sleeve	Pos 4		
5	2	cad	SN-000014	Surfacing pin	<i>Different values</i>		
6	2	cad	TN-51648	Hexagon socket set screw	Pos 6	DIN 913-M 10x12-45H	
8	1	cad	TN-39174	Hexagon head screw		ISO 4017-M 5x10-10.9	
9	1	cad	TN-55506	External tooth lock washer		DIN 6797-A 5.3-FSt	

## New attribute mapping configuration for document input via Drag & Drop

As for the serial document input in Version 2017 SP2, the attribute mapping is now also controlled via new configuration files for document input via Drag & Drop, i.e. you have the option to individually expand the configuration options for attribute mapping.

When you enter a new document via Drag & Drop, you will now find an additional pull-down menu for attribute mapping configuration at the bottom left of the displayed **Enter document** dialogue window:



If different configuration files have been stored in the system, these will be offered in this pull-down menu. Otherwise, only the **Standard** configuration will be available for selection.

In this way you can assign recurring initialization patterns with only a few mouse clicks, instead of having to set them manually every time.



**Please note:**

This new procedure replaces the previous configuration file `hel_fileattributes.ini`. This means that utilized mapping configurations may be renewed since the data from `hel_fileattributes.ini` will not be used any more.

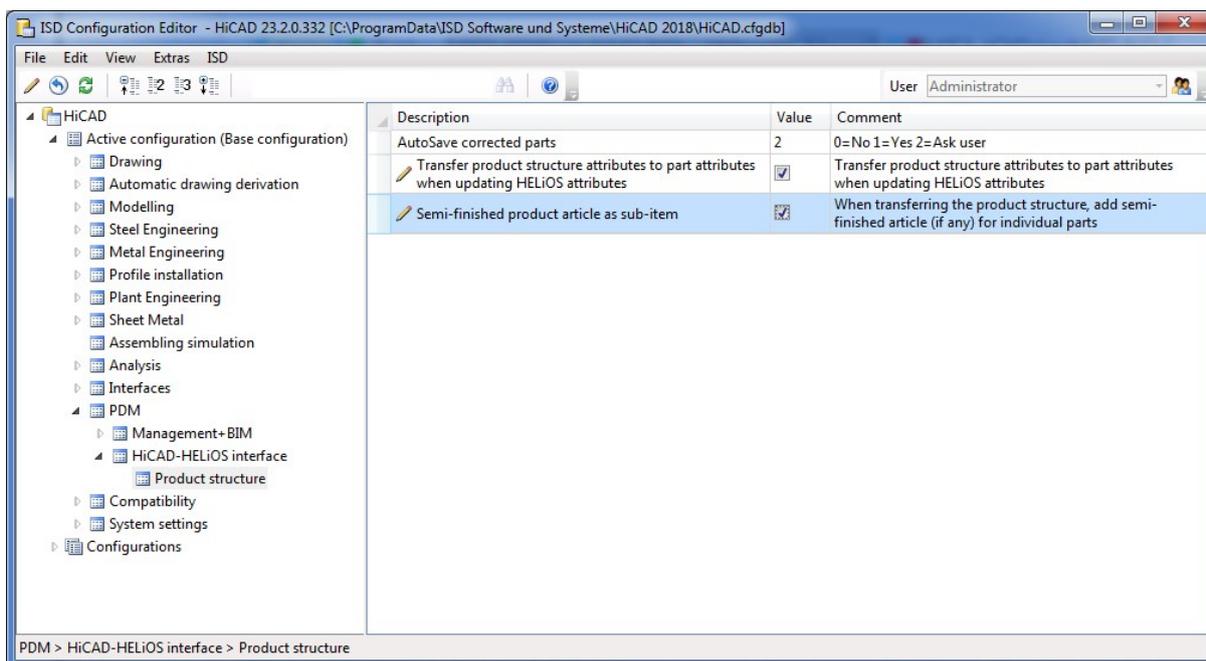
# HELiOS in HiCAD

## Service Pack 2 2018 (V 2302)

### Semi-finished Product Article as Sub-item

Semi-finished products can be automatically transferred as sub-items of a part to HELiOS.

This can be set in the Configuration Editor at **Active configuration > Base configuration > PDM > HiCAD-HELiOS interface > Product structure**. Here, activate the **Semi-finished product article as sub-item** checkbox.



If a HiCAD model drawing (.SZA document) or an individual geometry (.KRA document) is saved in HELiOS while the option has been activated, it will be checked whether a semi-finished product needs to be saved as a structure sub-item of the corresponding part (=HELiOS article).

The relevant semi-finished product article masters will then be added as sub-item, or updated accordingly in the product structure of the part.

## Service Pack 1 2018 (V 2301)

### Performance

A performance increase of 10 - 20% could be achieved for product structure transfers from HiCAD to HELiOS, which may involve very large amounts of data.

# HELIOS Spooler

Service Pack 2 2018 (V 2302)

## DSTV-NC-Export

When converting files via the HELIOS Spooler you can now also choose the DSTV-NC format. The format-specific export options that were last specified and saved in HiCAD will be used in the process.

## Service Pack 1 2018 (V 2301)

### New module structure

The module structure of the **HELIOS Print and Plot Management** has changed. The following three license modules are available:

- HiCAD Spooler
- HELIOS Spooler
- HELIOS Spooler Client

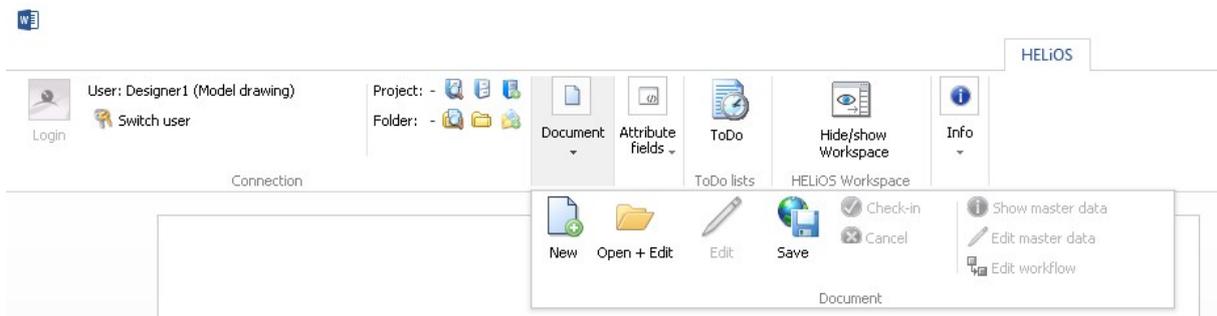
# HELiOS MS Office Interface

## Service Pack 2 2018 (V 2302)

### Project and folder data input

The HELiOS Ribbon tab for Office applications has been further enhanced with functions for the input of new projects and folders.

This allows you to create new projects or folders directly in your Office application (e.g. in Word) and assign the Office documents to them, which spares you the additional step via the HELiOS Desktop.



## Major Release 2018 (V 2300)

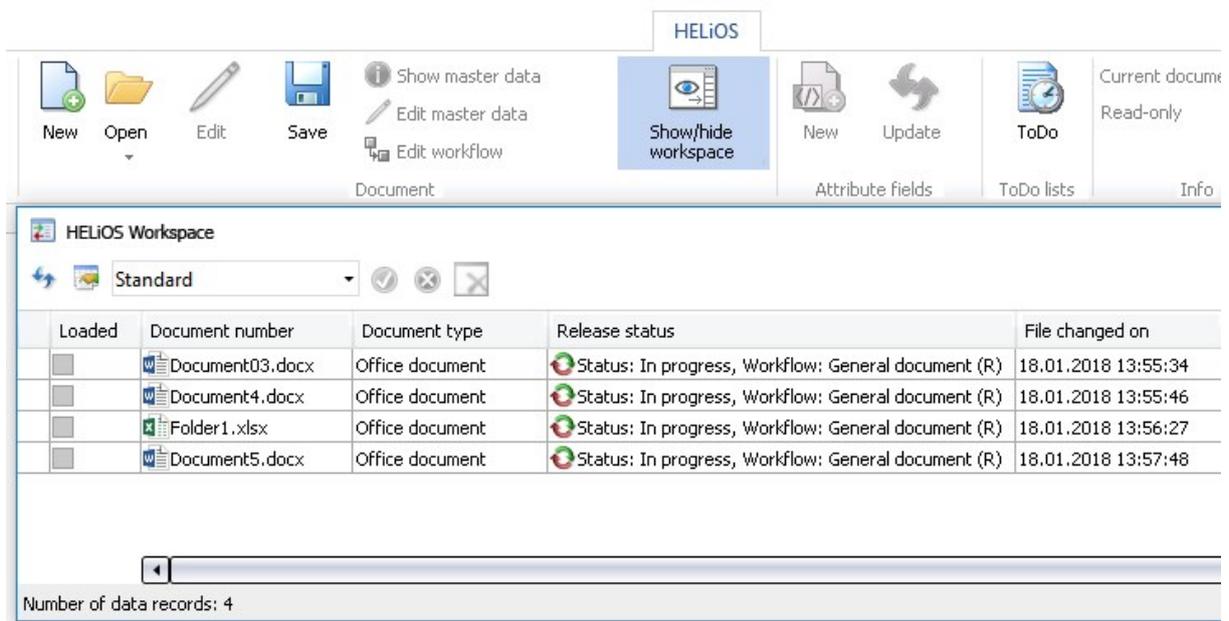
### New HELiOS Workspace

All file-based MS Office interfaces (Excel, Word, PowerPoint) now share a common **HELiOS Workspace**, as is already known from the HELiOS -MultiCAD interfaces.

The Workspace provides a clear overview of the locally checked-out files at any time, as well as a direct check-in or cancelling of the corresponding file edits.

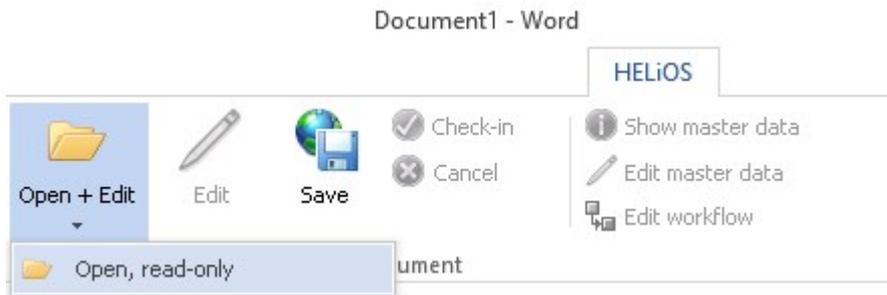
Furthermore, you have access to all HELiOS functions for the listed documents in the Workspace via a context menu (right-click document).

Depending on your preferred working method, you can decide whether you want HELiOS to generate unique file names, or if you want to assign file names manually.



### Open documents for editing or in read-only mode

In the HELiOS Ribbon tab of your Office application you can choose between two opening modes for documents: If you select **Open+Edit**, the Office document will be loaded from the HELiOS database, and can then be edited; if you select **Open, read-only**, the loaded Office document can only be viewed, but not edited.



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